

DesignCAD 3-D
Version 4.0
Reference Manual

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System Requirements

Required for DesignCAD 3-D:

1-2 megabytes of Expanded or Extended Memory (EMS or XMS)
510K DOS memory
Hard Drive

Recommended for DesignCAD 3-D:

Math Coprocessor (or 486 system)
Mouse or Digitizer
Printer or Plotter

DesignCAD 3-D will operate with 470K to 480K DOS memory if you use EMS (Expanded Memory) instead of XMS (Extended Memory). Using XMS (Extended Memory) requires about 40K more DOS memory. If you have a lot of memory resident drivers (drivers for network, CD-ROM, mouse, scanner, etc.), you might find it necessary to use EMS instead of XMS for DesignCAD 3-D.

DesignCAD 3-D requires 512K Expanded or Extended Memory plus enough memory for the graphics screen. If you use a 512K VGA, then DesignCAD 3-D will run with a total of 1 megabyte of Expanded or Extended memory. If you use a 1 Megabyte VGA at 256 color 1024x768 resolution, you will need a minimum of 1.5 megabytes of Expanded or Extended memory.

DesignCAD 3-D works with almost any printer or plotter. You can produce very impressive shaded images on an ordinary dot-matrix printer. Laser printers and color printers are also fully supported. When you run the program **SETUP3D**, you will be able to select your printer and plotter from the list supported by DesignCAD 3-D.

Introduction

Thank you for purchasing DesignCAD.

DesignCAD is a powerful 3-D computer aided design system. While its features and capabilities compare to CAD systems costing thousands of dollars, DesignCAD 3-D is extremely easy-to-use, and virtually anybody can make drawings of professional quality.

As a member of the DesignCAD family you are entitled to free technical support. **Be sure to mail, fax, or phone your registration information, because technical support is provided only to registered users.** Also, by being a registered user, you will receive update notices and other information.

DesignCAD is continually updating and improving, and we would like to hear your ideas! We make it a point to be responsive to the requests of our users. Send us a note and let us know what you think.

There are three DesignCAD 3-D manuals:

1. *DesignCAD 3-D Reference Manual* (this manual)
2. *DesignCAD 3-D Installation and Tutorial Manual*
3. *BasicCAD Reference Manual*

The *DesignCAD 3-D Installation and Tutorial Manual* contains detailed information about installing DesignCAD 3-D, as well as several lessons on how to use DesignCAD 3-D. Unless you are already familiar with DesignCAD, you should use that manual to get started.

The *BasicCAD Reference* manual is the manual for the BasicCAD programming language. BasicCAD is a language you can use to write programs that run within DesignCAD. You can write your own DesignCAD 3-D commands! You will need to use this manual only if you write BasicCAD programs.

About DesignCAD 3-D

DesignCAD 3-D is a true 3-Dimensional CAD system with advanced features such as hidden line removal, shading, and solid object modeling. DesignCAD 3-D allows up to 4 simultaneous views on the screen, at any angle or perspective.

DesignCAD 3-D also provides complex extrusions - linear, scalar, and circular. Advanced surface creation capabilities such as connects and patches are also included. Extensive 3-D text capabilities and auto dimensioning are provided.

DesignCAD 3-D provides the capability to read drawings from most other CAD systems (DesignCAD 2-D, AutoCAD's DXF, and IGES file formats). You can also send a view of a 3-D drawing to other CAD systems.

DesignCAD 3-D supports more than 200 dot matrix printers (at very high resolution), more than 80 plotters, as well as most digitizers and graphics adapters available for "PC Compatible" systems.

DesignCAD 3-D supports materials definition and libraries for realistic shading. With the keyframe animation capability of the DesignCAD Slide Show command and the outstanding print and plotter output, you can produce realistic images on your screen or on paper.

DesignCAD 3-D commands can be selected from the menu or entered as fast, single keystroke commands. We have found DesignCAD 3-D to be easier to learn and easier to use than any 3-Dimensional CAD system for the PC, at any price!

DesignCAD 3-D and DesignCAD 2-D

DesignCAD 3-D is a 3-dimensional CAD system. It is a completely separate product from DesignCAD 2-D, and provides 3-Dimensional drawing and solid object modeling capabilities. Because it is 3-Dimensional, DesignCAD 3-D may be more difficult to use in making 2-Dimensional drawings than the DesignCAD 2-D.

DesignCAD 2-D is our 2-dimensional CAD system. DesignCAD 2-D is fully compatible with DesignCAD 3-D. It is faster and easier to use for 2-Dimensional drawing and design than 3-D CAD systems.

If you make a lot of 2-dimensional drawings (plans, etc.), you might find it beneficial to get DesignCAD 2-D in addition to DesignCAD 3-D. This makes it possible to make a 2-dimensional plan drawing in DesignCAD 2-D, retrieve it in DesignCAD 3-D, and convert it to a 3-D model. You can also take a 3-dimensional drawing from DesignCAD 3-D and create 2-dimensional views (front, top, side, isometric, etc.) for DesignCAD 2-D.

If you have both DesignCAD 2-D and DesignCAD 3-D, you can place both on the same directory of your hard drive. This makes it easy to exchange drawings between the packages.

Quick Start

To install DesignCAD 3-D, place the Installation disk in drive A: and enter **A:INSTALL** (or **B:INSTALL** for drive B:). Then follow the instructions on the screen.

The installation program will install DesignCAD 3-D onto the hard drive. If you have a math coprocessor, be sure to install the math coprocessor version of the program disks. If you have a 486 CPU, then you should use the math coprocessor version of the program.

During the installation, you will be asked for the program serial number. This number is found on the Installation disk. You will also be asked for your registration information. After you have entered this information, it will be written to your registration disk.

To register your DesignCAD 3-D, put the registration disk in the postage-paid envelope and mail it - it's already addressed, and it doesn't need any stamps.

You can also print your registration information and register by fax at (918) 825-6359. Be sure to include the serial number on your installation disk.

Be sure to mail, fax, or phone your registration information, because technical support is provided only to registered users. Also, as a registered user you will receive update notices and other information.

The DesignCAD 3-D Setup program **SETUP3D** is run automatically when installation is complete. In the Setup program, select the printer, plotter, display, and input device you use.

After you run the Setup program (automatically executed after **INSTALL**), you are ready to run DesignCAD 3-D. When you exit the Setup program you will be left in the DesignCAD 3-D subdirectory. To run DesignCAD 3-D, enter **DC3**.

For more information on DesignCAD installation and configuration, see the "Getting Started" section of the *DesignCAD 3-D Installation & Tutorial* manual.

NOTE: The files on the DesignCAD disks have been compressed, so merely copying the files to your hard drive will not work. You must run the installation program.

DesignCAD is available on both 5.25" and 3.5" disks. If you have the wrong disk size you can send the package to us for a free exchange:

American Small Business Computers, Inc.
One American Way
Pryor, OK 74361

To copy the DesignCAD disks from 3.5" disks to 5.25" disks (or vice versa), copy the files from one disk to the other and make sure the volume names on the new disks are the same as the old. For more information on converting from one size disk to another, see the *DesignCAD 3-D Installation & Tutorial* manual.

Using DesignCAD

To begin using DesignCAD 3-D, there are a few things you need to know, such as setting points, executing commands, and drawing lines. The following sections present a quick overview of these topics. For more detailed information, you can read about the specific commands in the "command reference" section of this manual. You should also go through the lessons in the *Installation and Tutorial* manual.

Working in 3 Dimensions

3-dimensional drawing means that your drawing will actually be created in 3-D space - that is, each point has a position on the horizontal (X) axis, vertical (Y) axis, and depth (Z) axis. The X, Y, and Z positions of a point are called its coordinates.

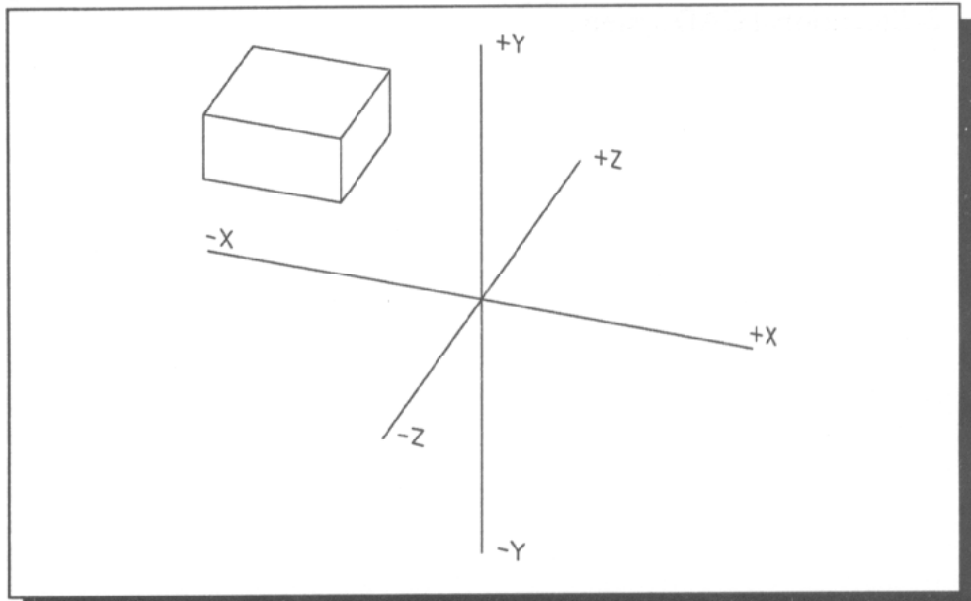


Figure 1 - The X, Y, and Z axis.

Two dimensional graphics systems use only the horizontal and vertical (X and Y) axes, while 3-Dimensional drawing includes the depth or Z axis. In two dimensions, you can draw only a single view of an object. When you draw on paper, for example, you are drawing in two dimensions.

3-Dimensional drawing allows you to draw the entire 3-D object and display it at any view. This provides consistency in all views and allows any view of the object to be produced easily - without drawing the object again.

However, all this utility does not come without cost. Drawing the entire 3-Dimensional object is sometimes more difficult than drawing

a single view of the object in two dimensions. When you draw an object in 3-Dimensions, you have to draw all faces of the object - not just the visible part.

Suppose, for example, that you want to draw a cube. Drawing a 2-Dimensional representation of the cube requires 4 or 9 lines, depending on the angle of the cube. Drawing the cube in three dimensions requires 12 lines. However, if you want to draw three views of a cube, it is much faster to use a 3-Dimensional CAD system and draw it only once, as opposed to drawing it three times on a 2-dimensional CAD system.

The 3-D Coordinate System

Each point in the 3-Dimensional coordinate system has an X and a Y coordinate, like the 2-Dimensional system, but it also has a Z coordinate.

The coordinates of a point are the X, Y, Z coordinates, such as 10, 25, 15. The Z coordinate gives the depth of the point. The larger the Z coordinate, the farther away the point is.

The 3-Dimensional coordinate system is displayed on the screen at a certain viewpoint. A 2-Dimensional view of the 3-Dimensional world is displayed on the screen. This viewpoint can be altered using the View command.

The distance of the image from the "viewer" can be altered using the View command to give perspective to the drawing. As the view distance gets smaller, the perspective becomes more pronounced.

The view angle and distance of the main view or any of the three smaller views can be set, and the three smaller views can even be disabled. The three views on the left of the screen are (from the top down) front, top, and side views unless they are changed with the View command.

Controlling the Cursor

Moving the cursor is the fundamental means of specifying locations in DesignCAD 3-D. The cursor can be moved with the cursor keys (see Figure 2), with a mouse or digitizer, and with several DesignCAD 3-D commands.

In addition to the normal cursor movements of Left, Right, Up, and Down, you can move the cursor In and Out. Moving the cursor "In" means to move it in toward the screen (conceptually) - toward the rear of the computer. Moving the cursor out moves it out from the screen toward you.

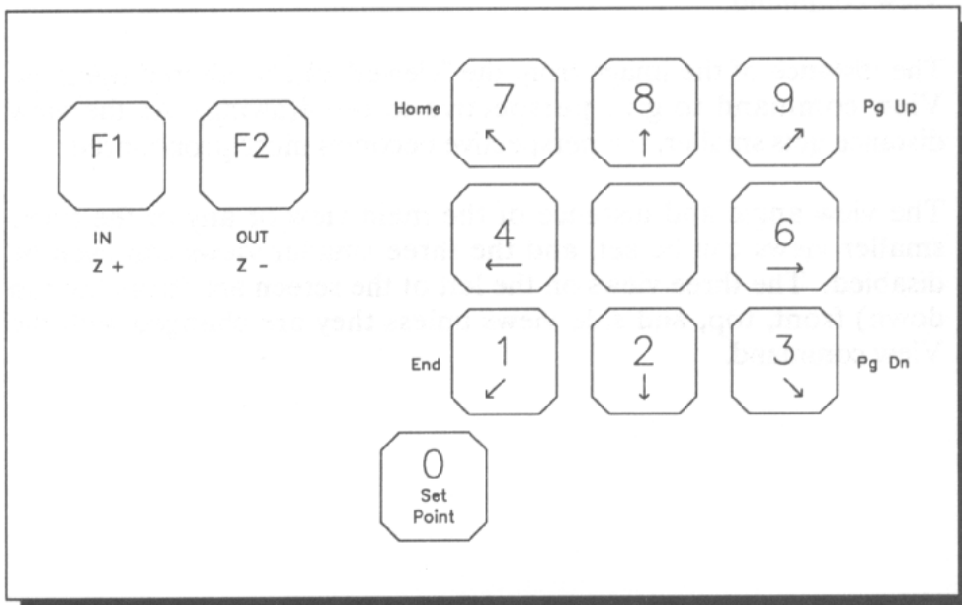


Figure 2 - The Cursor Control Keys.

Moving the cursor "In" is normally the positive direction of the Z axis - "Out" is in the negative Z direction. (You can reverse this by specifying "Left-Hand Coordinates" in the system parameters.)

The cursor control keys can be used to move the cursor in several directions. The arrow keys are used to move the cursor left, right, up, or down. The **Home**, **PgUp**, **End**, and **PgDn** keys move the cursor

diagonally in the upper left, upper right, lower left, and lower right directions. (These are the corner keys of the numeric keypad.)

The **F1** and **F2** keys are used to move the cursor in and out on the **Z** axis.

The distance the cursor moves with each keypress is the "cursor step size." This value can be changed (under the **CURSOR** menu). There is a large cursor step size and a small cursor step size. The Shift key can be used with the cursor control keys to move short cursor steps, which moves the distance set by the small cursor step size.

If you have a mouse or digitizer, you can use it to move the cursor. The cursor normally moves on the X-Y plane (horizontally and vertically) with a mouse or digitizer.

You can hold the **Shift** key down to force the mouse to move horizontally (along the X axis). **Ctrl** is used to move the mouse vertically (along the Y axis), and **Ctrl** and **Shift** can be pressed together to move the mouse in and out along the Z axis.

Setting Points

DesignCAD 3-D drawing commands use points as a reference. For example, using the Line command, you can draw a line through a set of points.

These points can be set using the **Ins** key, or **0** on the numeric keypad. You can also set a point using the left button on a mouse, or the first digitizer button.

The Insert key and Mouse button cause a point to be set at the current cursor position. Points can be set in a variety of other ways. For other methods of setting points, see the descriptions of the following commands:

Point XYZ
Point Relative
Point Polar
Gravity Point
Line Snap
Plane Snap
Midpoint
Line Plane
Center of Gravity

The Point Set command sets a point at the current cursor location. It is the simplest of the point setting commands. To use the Point Set command, press the first button on the mouse or digitizer, or press the **Insert** key.

The Gravity Point command is used to set a point exactly upon the nearest point. With the keyboard, this saves keystrokes by moving the cursor immediately to its destination (when another point is the destination) without having to "zero in" on the point. With a digitizer or a mouse, it allows you to find the spot without worrying about a steady hand.

When you zoom, move, and rotate a drawing, the points and lines may end up lying a little bit off the center of the dots on the screen.

The cursor, however, remains exactly on top of the dots or pixels on the screen. A point set on top of another using the Point Set command (**0** key) may in fact be a fraction of a dot off, although it will appear to be correct. When the drawing is zoomed in, the difference can be seen. The use of the gravity point prevents this error from occurring.

For example, draw a diagonal line on the screen. Zoom by a factor of .3. (Press **Z** and enter .3 for the zoom factor.) Then, using the **0** key and the Line command, make a triangle out of the diagonal line. Then zoom in on one of the end points of the original line by a factor of 10. (Set a point on one of the end points and then press **Z** for zoom.) You will see that the lines of the triangle do not actually meet. While this was not apparent on the screen, it would be seen on printer or plotter output.

The Point XYZ command (":" key or **Point XYZ**) can be used to set a point at an exact location on the screen. The coordinates are entered in the current units of measurement. The units of measurement, or Drawing Units, are set using the Units command.

The Point Relative command ("" key or **Point Relativ**) is similar to the Point XYZ command, but the X, Y, and Z values are relative instead of absolute. This means that the X, Y, and Z values are added to the X, Y, and Z value of the current cursor position.

The Point Polar command (";" key or **Point Polar**) can be used to set a point a certain distance and angle from the last point that was set. This can be done on the X-Y plane (the normal drawing plane), the X-Z plane (the "flat" drawing plane), or the Y-Z plane.

The Line Snap command sets a point at the nearest location on the nearest line. This command is similar to the Gravity Point command, but it goes to the nearest line instead of the nearest point.

The Plane Snap command sets a point at the nearest location of the nearest plane. This command is similar to the Line Snap command, but it goes to the nearest plane instead of the nearest line.

The Midpoint command sets a point at the midpoint of the nearest line. If the line has more than one segment, it sets a point at the midpoint of the nearest line segment.

The Line Plane command sets a point at the intersection of a line and a plane. The Intersect command is used to set a point at the intersection of two lines, circles, or arcs. Any combination of these can be used. This command requires two points to be set, one on each of the intersecting entities. Those two points will be erased, and a new point will be set on the intersection. If points are set before the Intersect command is used, the points will not be lost - the point of intersection will be added to these points.

The Center of Gravity command is a special purpose command used to set a point at the center of mass of a Solid.

Points can generally be set either before or after a DesignCAD 3-D command is issued. For example, you can press the **C** key (for Curve command) and then set 4 points for the curve, followed by **Enter**. Or, you can set the four points and then press the **C** key. If you set the points first, you do not have to press **Enter** after setting points. If you press the **C** key first, then you will be given instructions on setting the points. Use the method that is best suited for your needs.

Using DesignCAD 3-D Commands

Each of the DesignCAD 3-D commands can be executed using the menu or the command line. The quick reference card and the Help command can help you get used to the new command language. After you are familiar with the DesignCAD 3-D command language, you will find it very fast, efficient, and productive.

To select a command from the command menu, move the cursor up to the menu with the mouse or press **Enter**. Use the mouse or the arrow keys to select the menu you need, then press the left mouse button or the **Enter** key. When the menu "pulls down," select the command you want and press the mouse button (or **Enter**) again.

To use the command menu with the keyboard, press enter and use the arrow keys to select the menu you need, then press **Enter**. A "shortcut" method of selecting a menu is to press the highlighted letter of the desired menu with the **Alt** key. For example, you can press **Alt-F** to bring up the FILES menu.

When the menu "pulls down," you can select the command you want using the arrow keys (or mouse), or you can press the highlighted letter of the desired command. For example, under the FILES menu, you can press **S** to save the drawing.

When a command is highlighted on the command menu, you can press the **F1** key or the right mouse button for help. DesignCAD 3-D has detailed information on each command that can be displayed on the screen at the touch of a button.

DesignCAD 3-D commands can also be entered from the command line. To do this, enter the command mode by pressing the Space Bar. The command line (**Command:**) will appear at the bottom of the screen. Enter the name of the desired command and press **Enter**. That command will then be executed.

Many of the menu commands have short forms which can be used when entering the commands from the command line. For example, **C3** can be used instead of **Circle-3** to draw a 3-point circle.

Sometimes when you select a DesignCAD 3-D command you will be asked to set a number of points and then press **Enter**. If you use a mouse, you can use the middle mouse button instead of enter. If your mouse has only two buttons, you can use the "Enter Box", the small box with a diagonal line through it in the upper left corner of the screen. You can move the cursor there, press the left mouse button, and it will be the same as pressing **Enter**.

Many of the DesignCAD 3-D commands also have corresponding keystroke commands - commands that are executed by pressing only one key on the keyboard. These commands are the most commonly used DesignCAD 3-D commands. To use a DesignCAD 3-D keystroke command, press key for that command. For example, press **V** to draw a line through a set of points. You do not have to press **space** before you press the key for a keystroke command.

Some of the DesignCAD keystroke commands require the **Ctrl** key to be pressed with another key. For example, **^C** means to hold the **Ctrl** key down while you press the **C** key.

When the phrase "enter **ARC**" or "enter *command*" is used in this manual, it means to enter the command either from the menu, the command line, or to press the single key for the keystroke command.

After a DesignCAD command has been executed, you can press the **F3** key followed by **Enter** to repeat the command. This makes it easy to perform the same command a number of times.

Many DesignCAD 3-D commands require one or more parameters to be input from the keyboard. For example, when you retrieve a drawing from disk you will be asked for the drawing name. When you use the Point XYZ command, you will be asked for the X and Y coordinates.

In commands such as these, you can optionally enter the input parameters from the command line. For example, you can press **Space** and enter "**retrieve bank**" if "bank" is the name of the drawing you want to read. You can also enter "**pointxyz 20 20 20**" on the command line to set a point at location 20, 20, 20, or "**layer 7**" to change to layer number 7. This flexibility is one more way DesignCAD 3-D is made easy for you to use.

When you are asked to enter a number in DesignCAD 3-D, you can enter a formula or expression in brackets ("[" and "]"). For example, the following commands could be entered from the command line:

```
POINTXYZ [2/3, 22/7, 0]
POINTXYZ [SQRT(2)], 7, 0
POINTXYZ [SIN(45)], [COS(45)], [SIN(30)]
```

The expression can contain mathematical functions as well as the following operators: +, -, *, /, and ^. (^ is the operator to raise a number to a power. 2^3 is equal to 8.) The valid mathematical functions are the same as those listed in the *BasicCAD Reference Manual*.

You can also use this method to perform on-screen calculations. To perform a calculation, enter the formula, enclosed in brackets, at the command line. A box will pop up in the middle of the screen with the answer. So if you need to know the Logarithm of 18, you can enter:

```
[LOG(18)]
```

at the command line. The answer will then be displayed on the screen.

Using a Mouse with DesignCAD 3-D

A mouse can be used with DesignCAD 3-D to position the cursor, set points, draw lines, and enter commands from the command menu.

A mouse usually has 2 or 3 buttons. If you have a two button mouse, you can simulate the third button by pressing the two buttons at the same time. The buttons on the mouse are used for the following:

| | |
|----------------|---------------|
| Left Button: | Point Set |
| Right Button: | Gravity Point |
| Middle Button: | Enter |

Several DesignCAD commands ask you to set a number of points, and then press **Enter**. Instead of going to the keyboard to press **Enter**, you can press the middle mouse button.

The mouse can also be used to enter commands from the command menu. To do this, move the cursor to the top of the screen with the mouse, or press the middle mouse button. Move to the desired menu entry by moving the mouse. Select the command by pressing the left button on the mouse.

For help on any DesignCAD 3-D command, press the right mouse button instead of the left button when the command is highlighted.

There is a small box in the upper left corner of the DesignCAD 3-D screen. This is the "Enter Box." Any time you are asked to set a number of points and then press **Enter**, you can use the mouse to set a point in the "Enter Box" instead of pressing the **Enter** key. If you have a 3-button mouse, it is easier to press the middle button instead of moving to the "Enter Box."

When you are asked a "Yes or No" question, you can press the left mouse button for "Yes" and the right mouse button for "No."

Any time a file is being read from disk, you can press the left mouse button when you are asked for the file name and a directory listing will be displayed. You can then select the file with the mouse.

In several DesignCAD 3-D commands such as the **Layer** and **Parameter** commands, you can enter data onto several lines of the screen. In these commands, you can move the mouse (instead of pressing arrow keys) to move the cursor from one field to the next. You can also use the left mouse button for **Enter** and the right mouse button for **Esc**.

The **Snap Grid** is a very important tool to ensure precise cursor movement with a mouse. The snap grid may be enabled to force the cursor to be aligned on a certain grid spacing. This makes it easier to move the cursor to a precise location with the mouse. Use the **Snap Grid Size** command (under the **CURSOR** menu) to set the snap grid size, and the **Snap Grid** command (**^G** key) to enable or disable the snap grid. The word "SNAP" appears on the status line (the line at the bottom of the screen) when the snap grid is enabled.

You can move a mouse precisely horizontally, vertically, or in the **Z** direction using the **Shift** and **Ctrl** keys. To move the mouse horizontally, hold the **Shift** key down and move the mouse. To move vertically, hold the **Ctrl** key down. To move in or out parallel to the **Z** axis, hold down both the **Shift** and **Ctrl** keys.

The mouse normally moves on the **X-Y** plane. If the view has been changed so the drawing is being viewed from the side, the mouse will move on the **Y-Z** plane. If the drawing is viewed from above, the mouse will move on the **X-Z** plane. In essence, the mouse moves the cursor on the plane closest to the plane of your screen.

Using a Digitizer with DesignCAD 3-D

DesignCAD 3-D supports many different digitizers (also called graphics tablets). If you do not have a digitizer, you may skip this section.

To set up DesignCAD 3-D to use your digitizer, run the program **SETUP3D**. Select option 3 and enter the number of your digitizer. You will then be asked for the communications information.

Using the Digitizer Menu

With your DesignCAD 3-D package, there is an L-shaped card with several DesignCAD 3-D commands on it. This is the digitizer menu. The digitizer menu can be affixed to your digitizer to allow you to enter DesignCAD 3-D commands directly from the digitizer. If the digitizer menu is too large for your digitizer, cut it off with a pair of scissors. Most of the more important commands on the menu are found near the center.

To use the digitizer menu, first attach it to your digitizer in the lower left corner. Then, in DesignCAD 3-D, use the Digitizer Menu Load command (**DMenu Load** from the FILES menu) and enter **DC3MENU** when asked for the digitizer menu name.

You will be asked to set a point on the digitizer menu handle. To do this, set a point with the digitizer at the lower left corner of the digitizer menu. If you need to move the digitizer menu, use the Digitizer Menu Move command (**DMenu Move**).

DesignCAD 3-D can be set up to automatically load a digitizer menu when the program is run. To do this, first load the digitizer menu and then execute the Configuration Save command (**Config Save** or **csa**).

The digitizer menu can be used only in "digitizer mode," as described in the next section.

Using the Digitizer

The digitizer can be used to move the cursor, set points, and enter DesignCAD 3-D commands. To move the cursor, merely move the stylus or cursor around the digitizer. The cursor will move about the screen in a corresponding manner.

Your stylus or digitizer cursor may have from 1 to 16 buttons. Up to 15 buttons can be used to enter DesignCAD 3-D commands. Each button corresponds to a different command:

| |
|---------------------------|
| Button 1. Point Set |
| Button 2. Gravity Point |
| Button 3. Line |
| Button 4. Curve |
| Button 6. Fillet |
| Button 7. Chamfer |
| Button 8. Line |
| Button 11. Point Move |
| Button 12. Section Delete |
| Button 13. Erase |
| Button 14. Escape |
| Button 15. Unerase |

Some digitizers with 8 or more buttons may have some of these buttons transposed. Try out your digitizer to verify which buttons correspond to which commands.

Your digitizer may have either a stylus (a pen-like device) or a cursor (also called a mouse). A stylus can be depressed onto the digitizer in order to set a point. If the stylus has a button or barrel switch, this may be used to set a point with gravity.

The Snap Grid is a very important tool to ensure precise cursor movement with a digitizer. The snap grid may be enabled to force the cursor to be aligned on a certain grid spacing. This makes it easier to move the cursor to a precise location with the digitizer. Use the Drawing Parameter command (**Q** key) to set the snap grid size, and the Snap Grid command (**^G** key) to enable or disable the snap grid. The word "SNAP" appears on the status line (the line at the bottom of the screen) when the snap grid is enabled.

You can move the cursor precisely horizontally, vertically, or in the Z direction using the **Shift** and **Ctrl** keys with the digitizer. To move horizontally, hold the **Shift** key down and move the digitizer. To move vertically, hold the **Ctrl** key down. To move in or out parallel to the Z axis, hold down both the **Shift** and **Ctrl** keys.

Digitizer Mode and Mouse Mode

There are two modes in which the digitizer can operate: "digitizer mode" and "mouse mode." The digitizer normally works in "digitizer mode." This means that the digitizer movement remains consistent with the drawing when the drawing size changes. In mouse mode, the digitizer movement remains consistent with the screen when the drawing size changes.

Suppose, for example, that you have a drawing on the screen and you Zoom in by a factor of 2 to double the drawing size. In digitizer mode, the cursor will move twice as fast on the screen as it did before, but one inch movement on the digitizer will cover the same area on the drawing.

In mouse mode, after you zoom by a factor of 2, the cursor will move at the same rate on the screen, but one inch on the digitizer will cover only half the distance on the drawing.

"Digitizer mode" is better to use if you are using a paper drawing on your digitizer as a reference. "Mouse mode" can be useful if you need to zoom your drawing to work on fine detail, because the cursor movement will not be exaggerated.

In digitizer mode, the area of the screen accessed by the digitizer may be varied using the Zoom command (**Z** key). When DesignCAD 3-D is initialized, the area on the screen accessed by the digitizer is the largest area in the shape of the digitizer that can be displayed on the physical screen.

In digitizer mode, you can use the digitizer menu. When the digitizer is in mouse mode, you can use only the command menu on the screen.

The **Zoom** command (in digitizer mode) will not change the relationship of the drawing in the computer and the digitizer - it only changes the size or location of the drawing on the screen.

If you need to change the location of the drawing in the computer with respect to the digitizer, you can use the **Static Zoom** command (**^Z** key) or the **Static Zoom-4** command to zoom instead of using the **Zoom** command. The **^Z** command works like the normal **Zoom** command except that the digitizer remains constant with the screen instead of with the drawing in the computer.

Erasing Parts of Your Drawing

There are several ways to erase parts of a drawing:

Undo command.
Erase command.
Section Delete command.
Section Delete-Cutoff.
Block Delete command.
Slice command

The Undo command (**Esc** key or **Undo**) can be used to erase the last line or entity drawn. It can be repeated to erase more than one command. If an entity is erased in error, the Unerase command ("**!**" or **Oops**) can be used to reverse an Undo (or the other erase commands). Like the Undo command, the Unerase command can be repeated. If anything has been drawn since the last erasure, the Unerase command cannot be used.

The Erase command (**E** or **Erase**) is used to selectively erase lines or entities in the drawing. With the Erase command, you set a point on the line to be erased. The point can be on an endpoint of the line or it can be on or near the line itself. If the point lies directly on an endpoint, the command will function faster in a large drawing. The Gravity Point command is very useful for this. More than one entity can be erased at a time with the Erase command by setting more than one point with the command.

The Section Delete command (**D** key) allows an entire section of the drawing to be erased at one time. A section is made up of all the lines or entities completely contained within the rectangle defined by two points set in opposite corners (see the Section Delete command). Any command that lies completely within the section is erased.

Section Delete-Cutoff command will actually break lines and surfaces that cross the section boundaries and delete the portions of those entities inside the section. This command is particularly useful in making an opening in a line or a surface.

The Block Delete command erases a previously defined Block (see the section in this manual on Blocks).

The Slice command is used to "slice" the drawing and erase everything on one side of a plane. This is also known as making a cross section.

When a line that intersects another line is erased on the drawing, a small portion of the intersected line is erased from the screen. This portion is not erased from the drawing itself - it is only erased from the display. The drawing can be regenerated to repair these line breaks using the Zoom command (**Z** key) with a Zoom factor of 1.

By the same token, it is NOT possible to erase a line by drawing over the line and pressing the **Esc** key. While this does erase the line from the screen, it is not removed from the actual drawing. It will reappear when the drawing is regenerated with a Zoom, Rotate, or Save and Retrieve.

Using Zoom Commands

The Zoom function of DesignCAD 3-D is a very powerful feature. You can make your drawing larger than the screen or smaller than a single dot on the screen, without losing any part of the drawing. In addition, you can use the Zoom to "pan" or move your drawing across the screen.

DesignCAD 3-D supports the following Zoom commands:

| | |
|---------------|---|
| Zoom | Zoom in or out |
| Zoom Window | Zoom into a window |
| Zoom View | Zoom a specific view |
| Zoom Static | Zooms & doesn't change zoom factor |
| Zoom Previous | Zooms to the previous size and location |

To use the Zoom command to make your drawing larger or smaller, press the **Z** key and enter the zoom factor. The zoom factor is the relative size you want the drawing to be. For example, to make a drawing twice as large, enter 2 for the zoom factor. To double the size again, zoom with a factor of two. To reduce the drawing to half its original size, zoom with a factor of .125 (1/8). The absolute zoom factor is displayed on the status line.

If no points are set with the Zoom command, the center of the screen will be used as the point to zoom into (or out of). To zoom into a particular location on the drawing, set a point at that location. To zoom into a location and move it to another location on the screen, set a second point at the destination location.

If you want to move the drawing one direction or another without changing its size, set a point at the "from" location and a point at the "to" location. Then zoom with a factor of one. The entire drawing will be moved or "panned" to the new location.

To restore the drawing to its original size and location, a null zoom factor can be used with the Zoom command. (Press **Enter** when asked for the zoom factor).

To make the drawing as large as possible on the screen, use a zoom factor of 0.

A digitizer will normally "follow" the Zoom command. As you Zoom in, the digitizer will move the cursor the same distance on the drawing (and farther on the screen) for a given distance on the tablet. If you use the DigiMouse command to operate the digitizer in mouse mode, then the digitizer will remain consistent with the screen and will not "follow" the Zoom command.

The Zoom Window command is similar to the Zoom command, but instead of entering a zoom factor, the area of the drawing to be displayed on the screen is enclosed in a box or window.

The Zoom Previous command returns the drawing to the previous zoom factor and location. If you want to zoom into an area temporarily, you can use the Zoom previous command to return to the previous drawing position and size.

The Static Zoom command (^Z key) can be used to prevent the digitizer from "following" the Zoom, and also to keep the absolute zoom factor (displayed on the status line) from changing. This can be used to alter the default drawing size - a drawing is Zoomed back to a zoom factor of 1 when it is saved.

The Zoom command changes the size of the drawing in all four views. The Zoom View commands (Zoom View 1, Zoom View 2, Zoom View 3, and Zoom View 4) change the drawing size in a single view. The large view is view 1, the upper left view is view 2, the left center view is view 3, and the lower left view is view 4.

Using Text

DesignCAD 3-D is capable of displaying text several different ways. The simplest way is to use a single point with the Text command. The text will then be drawn at the default size and angle beginning at that point. The default text size and angle can be modified using the Text Size and Text Angle commands, or under the Drawing Parameter command.

To center the text, set two points in the same location. The text will be centered about that point at the default size and angle.

To right justify text, set three points in the same location. The text will be right justified to that point.

To position text between two points, set those two points with the Text command. The text will be drawn from the first point to the second point. The size of the text depends on the distance between the points. The angle of the text depends on the angle of the two points.

A third point can be used with the Text command to specify the plane on which the text is drawn.

There are several character fonts that come with DesignCAD 3-D. The character font for the text can be selected using the Font command, or by selecting the font under the System Parameter command. See the Font command in this manual for a list of DesignCAD 3-D character fonts.

Storing and Retrieving

It is easy to save and retrieve DesignCAD 3-D drawings. To save a drawing, use the Store command (**F10** keystroke command).

To retrieve a drawing, use the Retrieve command (**F9** keystroke command). To select a drawing from a directory listing, press **Enter** when asked for the drawing name.

Each drawing saved to disk has three reference points called "handles." The first handle is normally at the lower left corner of the drawing and the second handle defaults to the lower right corner of the drawing. The third handle defaults to a point above the second handle. It is used to specify the plane on which the drawing is retrieved. To specify handles other than the lower left and lower right corners of the drawing, set three points for the handle locations when the drawing is saved (see the Save and Retrieve commands in the "Command Reference" section of this manual).

Drawings may be retrieved from disk either with or without points. If the Retrieve command (**F9** or **Retrieve**) is used without points, the drawing will be retrieved at its original size and location.

If points are set when a drawing is retrieved, the first handle is placed at the first point and the second handle is placed at the second point. This is very useful in positioning symbols.

More than one drawing can be retrieved and merged into a single drawing. This is done by retrieving the first drawing, and then retrieving the second drawing on top of it. The Drawing Units of the second drawing are matched to those of the drawing already on the screen.

Editing Your Drawing

One of the primary advantages of CAD systems is the ability to edit and manipulate drawings. DesignCAD 3-D provides complete editing facilities for this. There are many editing commands in DesignCAD 3-D. Most will be mentioned in this section, but you will need to look up the commands in the "Command Reference" section of this manual for more complete information.

Section Delete-Cutoff command (^D key) can be used to "break" a line or erase a section of it.

If a portion of a line or command needs to be modified, the Point Move (* key) command can be used. This command allows you to actually move a point in the drawing. Every line or command using the point is "stretched" to accommodate the new location of the point. A point can also be deleted using the Point Move command. This is useful in erasing one end of a line.

The Stretch command is another very useful editing command. The stretch command actually stretches the part of the drawing inside a section and moves it to a new location. Every entity lying partly inside and partly outside the section is stretched to the new location, and every entity entirely contained in the section is moved to the new location.

The Fillet and Chamfer commands are used to round or cut off a corner at a specified radius. Fillet Edge and Fillet Corner commands can be used to round off the edge or corner of a solid object or a set of planes.

The Join command moves several endpoints that are near one another to the same location.

The Hammer commands are used to modify an existing surface by making a "dent" or pulling the surface in one direction. The Drill and Surface Subtract commands are used to make "holes" of specific shapes in surfaces.

The Block commands also provide extensive editing capabilities, as described in the following section.

Using Blocks

A Block, in DesignCAD, is a portion of the drawing that is recognized as a single unit. The Block commands operate on the current Block - that part of the drawing that was defined to be a Block. There are several Block commands in DesignCAD 3-D:

| | |
|------------------------------|--|
| Block Add | Add an entity to a Block. |
| Block All | Make a Block of the entire drawing. |
| Block Circular Repeat | Duplicate a Block in a circle. |
| Block Color | Change the color of a Block. |
| Block Define | Define a Block. |
| Block Define, Cut | Define a Block, with cutoff. |
| Block Delete | Delete the Block. |
| Block Extrude | Extrude a Block Linearly. |
| Block Extrude Varying | Extrude a Block at varying scales. |
| Block Handle | Set or move a Block's handles. |
| Block Insert | Copy a Block one time. |
| Block Layer | Change the layer of a Block. |
| Block Load | Retrieve a drawing from a disk as a Block. |
| Block Mirror | Make a mirror image of a Block. |
| Block Move | Move a Block from one place to another. |
| Block Repeat | Duplicate a Block a number of times. |
| Block Reset | "Undefine" a Block. |
| Block Rotate | Rotate a Block. |
| Block Save | Save a Block to disk. |

| | |
|---------------------|---|
| Block Scale | Change the relative scale of a Block along the X, Y, or Z axis. |
| Block Slice | Create a "cross section" of a Block. |
| Block Solid | Define a Block to be a Solid. |
| Block Spiral | Extrude a Block in a spiral. |
| Block Sweep | Extrude a Block circularly. |
| Block Zoom | Zoom a Block, making it larger or smaller. |

A Block is a portion of the drawing that is manipulated as a single unit. Once a block has been defined, it can be manipulated by any of the DesignCAD 3-D Block commands. One, two, or three handles can be defined with a Block. Handles are points used by the Block commands as reference points. For example, with the Block Insert command, you can give the location where the Block handles are to be located, and the Block will be scaled and rotated accordingly.

The Block Define command defines a Block to be all the entities inside a window or box. Single drawing entities can be added to the Block using the Block Add command.

The Block Extrude, Block Extrude Varying, Block Sweep, and Block Spiral commands are very important in 3-dimensional drawing. These commands let you take a 2-dimensional shape and convert it to a 3-dimensional surface or object.

The Block Extrude command extrudes a 2-dimensional shape linearly. You can extrude along a set of points, and you can specify a scale to reduce or expand the object along the extrusion.

The Block Extrude Varying command is similar to the Extrude command, but you can specify varying scale factors along the extrusion points. This is useful in making shafts, for example.

With the Block Sweep command, you can draw the profile of a circular solid object and extrude it in a circle or arc to create the solid itself.

The Block Spiral command is similar to the Block Sweep command, except you can specify an offset to extrude in a spiral instead of a circle.

The Block Insert command can be used to copy and optionally rotate and scale a Block. The Block can be positioned so that its two reference points or "handles" fit between two points with the plane of the third handle aligned with a third point.

The Block Repeat command duplicates a Block a number of times. The copies of the Block can be spaced any distance and any direction.

The Block Circular Repeat command duplicates a Block in a circle or arc. It can be used to put the teeth on a gear, for example. In addition, you can specify an offset with the Block Circular Repeat command to copy the Block in a spiral.

The Block Reset and Block Handle command can be used to define a Block without including entities in a window. This makes it possible to define a Block to be a single line or drawing entity with the Block Add command.

The Block Load command can be used to read a drawing from disk and define it to be a Block.

The Block Layer command is useful when you want to change the layer of several lines or entities. The Block Color command can be used to change the color of several lines at a time. Block Solid command is used to define the current Block to be a Solid.

The Block commands are among the most powerful DesignCAD 3-D commands. You should read about each of the Block commands in the Command Reference section of this manual.

Printing and Plotting

To print a drawing in DesignCAD 3-D, use the Print command (**F5** key). The print command in DesignCAD is actually a separate program. Within the Print command, you can print one drawing or several drawings at once.

To plot a drawing in DesignCAD 3-D, use the Plot command (**F6** key). The Plot command is part of the main DesignCAD 3-D program, so you do not need to save the drawing before plotting it, but you must load the drawing onto the screen before you plot it.

Drawings can be printed or plotted in wireframe, quick-shaded, smooth shaded, or with hidden line removal.

For more information on the Print and Plot commands, see the "Command Reference" section of this manual. To configure DesignCAD 3-D for your printer and plotter, run the program **SETUP3D**.

Geometric Construction

DesignCAD 3-D has many commands that can be used to construct virtually any object. Some of these commands are discussed in this section, but they are only "outlined." For more complete information on these commands, see the "Command Reference" section of this manual.

All DesignCAD 3-D entities can be drawn in true 3-dimensional space. For example, a circle does not have to be drawn on the X-Y plane. It can be drawn at any orientation. A curve is a true 3-Dimensional spline curve. It doesn't even have to lie on a single plane. Any object can be drawn at any orientation with DesignCAD 3-D.

There are four ways to draw a circle with DesignCAD 3-D. You can specify the center point and radius (Circle command), the diameter (Circle-2 command), or three points on the outside of the circle (Circle-3 command). You can also draw a circle tangent to two existing lines (Circle Tangent command).

There are three DesignCAD 3-D arc commands. The Arc command draws an arc based on the radius and the angle of the arc. The Arc-2 command draws the arc between two points with a specific radius. The Arc-3 command is a 3-point arc command similar to the Circle-3 command. It draws an arc from the first point through the second to the third point.

The Perpendicular command draws a line from a given point perpendicular to another line. The Parallel command draws a line parallel to another line in the drawing. The Surf-Surf command draws a line along the intersection of two surfaces.

The Center of Gravity command is used to determine the center of gravity, or center of mass, of an object.

The midpoint command sets a point at the midpoint of a line. An angle can be bisected by using the midpoint command on an arc across the angle. The Intersect command can be used to set a point at the intersection of two lines, circles, or arcs.

Any regular polygon (a polygon with all sides the same length) can be generated using the Polygon command - you simply set two points for the first side and enter the total number of sides for the polygon.

To convert lines into surfaces or solids, you can use the Block extrusion commands: Extrude, Extrude Varying, Sweep, and Spiral.

There are DesignCAD 3-D commands to draw boxes, spheres, hemispheres, cones, and cylinders. After solid objects have been drawn, you can subtract one object from another (Subtract command), get the boolean intersection of an object (Solid Intersect), and determine whether one Solid interferes with another (Interference command).

Dimensioning with DesignCAD 3-D

DesignCAD 3-D has three dimension commands to provide extensive dimensioning capabilities. Among the dimension commands are:

| | |
|-----------------|---------------------------------------|
| Dimension | Dimension horizontally or vertically. |
| Dimension Angle | Dimension an angle. |
| Dimension Line | Dimension a line at any angle. |

With DesignCAD 3-D, you can dimension anywhere in 3-D space, not just on the X-Y plane.

See the "Command Reference" section of this manual for details on these different dimension commands.

Using Solids

DesignCAD 3-D supports the definition and use of Solids for some very powerful operations and calculations, including intersections, subtractions, interference checking, volume and surface area calculations, and more.

Solids

A Solid is a collection of DesignCAD 3-D entities (planes, surfaces, etc.) that make up a solid object. This is similar to a Block, but once a Solid is defined it remains a Solid. Also, a Solid must be a closed 3-dimensional object. This means that not all Blocks can be legitimate Solids.

Some simple Solid objects are cubes, cones, cylinders, and spheres. These objects are automatically defined to be Solids when they are drawn. Some invalid Solid objects are a cube with one face missing, a cone with no bottom, and a single plane.

One way to think of a valid Solid is that all the surfaces must completely enclose a space. You can imagine that the Solid, if hollow, would have to be able to hold compressed air.

A Solid can be defined using the Solid Define command, or by drawing a Solid with the Box, Cone, Cylinder, Sphere, Hemisphere, Sweep, or Extrude commands. (The Sweep and Extrude commands do not always draw valid Solids. For example, a 37 degree sweep of a line would not make a Solid.)

The Solid Define command allows you to define a Solid as all entities within a 3-D box, defined by 2 points in opposite corners.

Up to 200 independent Solids can be used in a drawing at any one time.

Solid Object Calculations

Once a Solid is defined, you can perform area, volume, and center of gravity calculations on the Solid. The Volume command is used to find both the volume and the surface area of a Solid. The Center of Gravity command is used to snap to the center of gravity (center of mass) of a Solid.

Interference checking between two Solids can be done using the Interference command.

Solid Operations

Several Solid operations can be done with DesignCAD 3-D. You can get the union of two Solids with the Solid Add command. One Solid can be removed from another with the Solid Subtract command. A Solid can be removed from a non-Solid surface with the Drill command. The Solid Intersect command removes all of two Solids except the intersection.

The DesignCAD 3-D tutorial has several examples of how these Solid operations can be used in practical applications.

Using Macros

The Macro feature of DesignCAD 3-D allows you to perform a task one time, record the keystrokes for that task, and repeat it at any time.

A Macro can be created, for example, to break a line and insert an object at the break in the line. This can be repeated at different locations by moving the cursor and executing the Macro.

The single keystroke macros (MACRO1 through MACRO12, for keys **^F1** through **^F12**) make it possible to define your own keystroke commands.

When you define a Macro, the keystrokes used in that macro are recorded to disk. When you move the cursor one step, it moves a certain distance. When the Macro is later executed, the settings for cursor step size and snap grid should remain the same as they were when the Macro was created for consistent operation.

The Move XYZ, Move Polar, and Move Relative commands can be used to move the cursor without regard to the cursor step size and snap grid settings. This provides a more positive method for cursor movement.

The Macro execution begins at the last point set. If no points have been set, the macro begins at the current cursor position. Mouse and digitizer cursor movements cannot be used within a Macro.

A macro is a very simple way to "program" DesignCAD 3-D. For more complex and comprehensive programming, you should use the BasicCAD programming language.

BasicCAD Overview

BasicCAD is a powerful programming language that allows you to run your own programs within DesignCAD. With BasicCAD, you can write a program and execute it as you would a DesignCAD command.

BasicCAD is similar to standard Quick Basic, but it provides the capability to execute DesignCAD commands as well as Basic statements. This gives BasicCAD the flexibility of Basic together with the power of DesignCAD.

To create a BasicCAD program, edit an ASCII file using a text editor and write the BasicCAD program. (The file extension should be .BSC.) To run the program, use the DesignCAD **RUN** statement:

RUN program

where "*program*" is the name of your program file.

Try it. Edit a file and enter the following program:

| | |
|----------------------|---------------------------|
| GETXYZ X Y | ' get the cursor position |
| >POINTXYZ [X, Y] | ' Set a point |
| >POINTXYZ [X+5, Y+5] | ' Set a point |
| >BOX | ' Draw a box |

As in standard Basic, BasicCAD uses the apostrophe (') for comments. The first statement above assigns the current cursor coordinates to variables X and Y.

The second statement sets a point at the current cursor position. The third statement sets a point 5 units up and 5 units to the right of the current cursor position. In BasicCAD, DesignCAD commands are preceded by ">". (POINTXYZ and BOX are DesignCAD commands.) In the DesignCAD commands, expressions enclosed in brackets ("[" and "J") are evaluated before the command is executed.

The fourth line uses the DesignCAD Box command to draw a box using the two points already set for opposite corners.

After you have entered the program, save it as "TEST.BSC". Be sure to save it in ASCII mode if you are using a word processor to edit. BSC is the file extension used for BasicCAD programs. (BSX is used for encrypted BasicCAD programs.)

Now run DesignCAD and enter **Run Test** from the command line. A box will be drawn with its lower left corner at the cursor position.

A BasicCAD program test utility called **PTEST** is included with DesignCAD. **PTEST** lets you run your BasicCAD program without running DesignCAD. It is very useful in debugging BasicCAD programs.

For more information on programming in BasicCAD, see your *BasicCAD Reference Manual*.

DesignCAD 3-D Command Reference

DesignCAD 3-D Commands

| | |
|------------------------------|---|
| Arc | Draw arc with center, radius, angle. |
| Arc-2 | Draw arc using beginning, end, radius. |
| Arc-3 | Draw arc using 3 points. |
| Arrow | Draw an Arrow. |
| Attribute | Enter an Attribute. |
| Block Add | Add an entity to a Block. |
| Block All | Make the entire drawing a Block. |
| Block Circular Repeat | Duplicate a Block in a circular pattern. |
| Block Color | Change the color of a Block. |
| Block Define | Define a Block. |
| Block Define, Cut | Define a Block with cutoff. |
| Block Delete | Delete a Block. |
| Block Extrude | Extrude a Block Linearly. |
| Block Extrude Varying | Extrude a Block at varying scales. |
| Block Handle | Set or move the Block's Handle. |
| Block Insert | Copy a Block one time. |
| Block Layer | Change the layer of a Block. |
| Block Load | Retrieve a drawing from disk as a Block. |
| Block Mirror | Make a mirror image of a Block. |
| Block Move | Move a Block from one place to another. |
| Block Repeat | Duplicate a Block a number of times. |
| Block Reset | "Undefine" a Block. |
| Block Rotate | Rotate a Block. |
| Block Save | Save a Block to disk. |
| Block Scale | Scale or stretch a Block along an axis. |
| Block Slice | Get the cross section of a Block. |
| Block Solid | Define the entities in a Block to be a Solid. |
| Block Spiral | Extrude a Block in a spiral. |
| Block Sweep | Extrude a Block circularly. |
| Block Zoom | Zoom a Block. |
| Box | Draw a box. |
| Center of Gravity | Find the center of gravity of a Solid. |
| Chamfer | Chamfer the intersection of two lines. |
| Change Color | Change the color of an existing entity. |
| Change Entity | Change the characteristics of an entity. |
| Circle | Draw a circle using center and radius. |
| Circle-2 | Draw a circle using the diameter. |
| Circle-3 | Draw a 3-point circle. |
| Circle Tangent | Draw a circle tangent to two lines. |

| | |
|------------------------------|--|
| Clear Screen | Clear the Screen. |
| Color | Change drawing color. |
| Color Edit | Create and edit a palette of colors. |
| Color Read | Load a color palette. |
| Color Reset | Return to the default color palette. |
| Color Save | Save the color palette. |
| Combine Line | Combine connected lines into a single line. |
| Cone | Draw a cone. |
| Configuration Save | Save the current system configuration. |
| Connect | Connect two or more lines with a surface. |
| Connect Curve | Connect two or more lines with a curved surface. |
| Connect Smooth | Connect two or more lines with a smooth surface. |
| Cursor Step Size | Set the cursor step size. |
| Cursor Type | Change cursor type. |
| Curve | Draw a curve. |
| Cut Plane | Cut a plane between two points. |
| Cylinder | Draw a cylinder. |
| DigiMouse | Set the digitizer to mouse mode or digitizer mode. |
| Digitizer Menu Delete | Delete the digitizer menu. |
| Digitizer Menu Load | Load a digitizer menu. |
| Digitizer Menu Move | Move the digitizer menu. |
| Dimension | Dimension horizontally or vertically. |
| Dimension Angle | Dimension an angle. |
| Dimension Line | Dimension at any angle. |
| Directory | Display drawings on disk. |
| Disc | Draw a circle and make it a surface. |
| DOS | Shell to DOS temporarily. |
| Drawing Parameter | Enter drawing parameters. |
| Drill | Drill or subtract a Solid from a surface. |
| End | Stop the program. |
| Erase | Erase an entity. |
| Fillet | Make a fillet at the intersection of two lines. |
| File Convert | Run the file conversion utility DCFILES. |
| Fillet Corner | Round a 3-D corner with a surface. |
| Fillet Edge | Round 3-D edge with a surface. |
| Font | Change character fonts. |
| Get Entity | Set all points that define an entity. |
| Gravity Move | Move the cursor to the nearest point. |
| Gravity Point | Move the cursor to the nearest point in the drawing and set a point there. |
| Grid | Turn display grid on and off. |
| Grid Center | Set the center of the display grid. |
| Hammer-1 | Distort a surface with a rounded "hammer" |
| Hammer-2 | Distort a surface with a sharp "hammer" |
| Help | Display DesignCAD 3-D help. |
| Hemisphere | Draw half a sphere. |

| | |
|------------------------------|--|
| Hide | Perform hidden line removal on view 1. |
| Hide View 2 | Perform hidden line removal on view 2. |
| Hide View 3 | Perform hidden line removal on view 3. |
| Hide View 4 | Perform hidden line removal on view 4. |
| Home | Move the cursor to the center of the view 1. |
| ID | Display information about an entity. |
| Interference | Perform interference checking on two Solids. |
| Intersect | Set a point at the intersection of 2 lines. |
| Join | Connect the ends of two or more lines. |
| Layer | Change the active layer, set visibility of layers, name layers. |
| Lights | Set light source information. |
| Line | Draw a line. |
| Line Distance | Measure the distance between two lines in 3-D space. |
| Line Plane | Set a point at the intersection of a line and a plane. |
| Line Snap | Snap to the nearest line. |
| Macro Define | Define a Macro. |
| Macro Execute | Execute a Macro. |
| Make Plane | Convert a line into a plane. |
| Material | Create and edit a material. |
| Memory | Display the amount of memory used. |
| Midpoint | Snap to the midpoint of a line. |
| Move Polar | Move the cursor using polar coordinates. |
| Move Relative | Move the cursor using relative coordinates. |
| Move XYZ | Move the cursor using absolute coordinates. |
| Origin | Move the origin of the drawing. |
| Orthogonal Line | Draw a line vertically or horizontally. |
| Pan | Pan the drawing across the screen. |
| Parallel | Draw a parallel line. |
| Patch | Fit a surface over 4 connected lines. |
| Perpendicular | Draw a perpendicular. |
| Perpendicular Plane | Create a plane perpendicular to a line. |
| Plane | Draw a plane surface. |
| Plane Snap | Snap to the nearest plane. |
| Plane Subtract | Subtract one plane from another. |
| Plot | Output the drawing to the plotter. |
| Point Move | Move a point in an existing entity. |
| Point Polar | Set a point using polar coordinates. |
| Point Relative | Set a point using relative X,Y,Z coordinates. |
| Point Relative Cursor | Set a point using X,Y,Z coordinates relative to the current cursor position. |
| Point Set | Set a point at the current cursor position. |
| Point XYZ | Set a point using X,Y,Z coordinates. |
| Polygon | Draw a regular polygon. |
| Print | Output the drawing to the printer. |
| Quick Shade | Shade view 1 with quick shading. |

| | |
|----------------------------|---|
| Quick Shade 2 | Shade view 2 with quick shading. |
| Quick Shade 3 | Shade view 3 with quick shading. |
| Quick Shade 4 | Shade view 4 with quick shading. |
| Retrieve | Read a drawing or symbol from disk. |
| Rotate | Rotate the drawing. |
| Rubber Band Box | Enable the rubber band box. |
| Run | Run a BasicCAD program. |
| Save | Save the drawing. |
| Save 2-D Drawing | Save a 2-D view of the drawing. |
| Screen Color | Set background and text colors for the screen. |
| Screen Read | Read a screen image from disk. |
| Screen Save | Save a screen image to disk. |
| Section Delete | Delete a section of the drawing. |
| Section Delete, Cut | Delete a section with cutoff. |
| Section Stretch | Stretch a section of the drawing. |
| Set Path | Set the drives and directories for files. |
| Set View | Enable or disable the views 2, 3, and 4. |
| Shade | shade View 1. |
| Shade View 2 | Shade view 2 |
| Shade View 3 | Shade view 3. |
| Shade View 4 | Shade view 4. |
| Show Information | Display version and release date of DesignCAD 3-D. |
| Slice | Erase parts of the drawing in front of a plane - get a cross section. |
| Slide Show | Make a slide show of a drawing. |
| Snap Grid | Turn the Snap Grid on or off. |
| Snap Size | Set the snap grid size. |
| Solid Add | Add two Solids together (get the union). |
| Solid Block | Define a Solid to be a Block. |
| Solid Define | Define an object to be a Solid. |
| Solid Free | "Undefine" or release a Solid. |
| Solid Intersect | Get the intersection of two Solids. |
| Solid Move | Move a Solid from one location to another. |
| Solid Subtract | Subtract one Solid from another. |
| Sphere | Draw a Sphere. |
| Status Line Format | Set the status line format. |
| Surface Area | Find the area of a surface. |
| Surf-Surf | Get the intersection of two surfaces. |
| System Parameter | Set the System Parameters. |
| Text | Draw Text. |
| Text Size | Set the default text size. |
| Undo | Undo the last line or entity drawn. |
| Unerase | Redraw the last entity erased. |
| Units | Specify drawing units of measurement. |
| Un-Working Plane | Undo the working plane rotation. |
| Vector Arc | Draw an arc made of vectors. |

| | |
|----------------------|--|
| Vector Circle | Draw a circle made of vectors. |
| Vector Curve | Draw a curve made of vectors. |
| Vector Text | Draw text made of vectors. |
| View | Set view 1 angle and distance. |
| View 2 | Set view 2 angle and distance. |
| View 3 | Set view 3 angle and distance. |
| View 4 | Set view 4 angle and distance. |
| View Read | Read a set of view information from disk. |
| View Save | Save the current view information to disk. |
| Volume | Find the volume and surface area of a Solid. |
| Wall | Draw a wall. |
| Working Plane | Rotate the drawing to a working plane. |
| Zoom | Zoom in or out. |
| Zoom Previous | Zoom to the previous zoom level. |
| Zoom Static | Static Zoom. |
| Zoom View 1 | Zoom view 1 only. |
| Zoom View 2 | Zoom view 2 only. |
| Zoom View 3 | Zoom view 3 only. |
| Zoom View 4 | Zoom view 4 only. |
| Zoom Window | Zoom into a window defined by two points. |

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Arc Command

| | |
|--------------------|-----------------|
| Menu: | ARC/CIRC |
| Menu Command: | Arc |
| Short Form: | A |
| Keystroke Command: | A |

Point 1: Arc center.

Point 2: Arc radius and beginning.

Point 3: Plane definition (optional).

The Arc command is used to draw an arc (a portion of a circle) between two points. To use the command, set 2 or 3 points:

1. The center of the arc.
2. The beginning point that lies on the arc.
3. The third point is used to define the plane on which the arc lies (optional).

You will be asked to enter the angle of the arc. An arc will then be drawn from the first point counter-clockwise for a distance of that angle. (The angle is relative to the second point.)

The third point can be used, together with the first two points, to define the plane on which the arc lies.

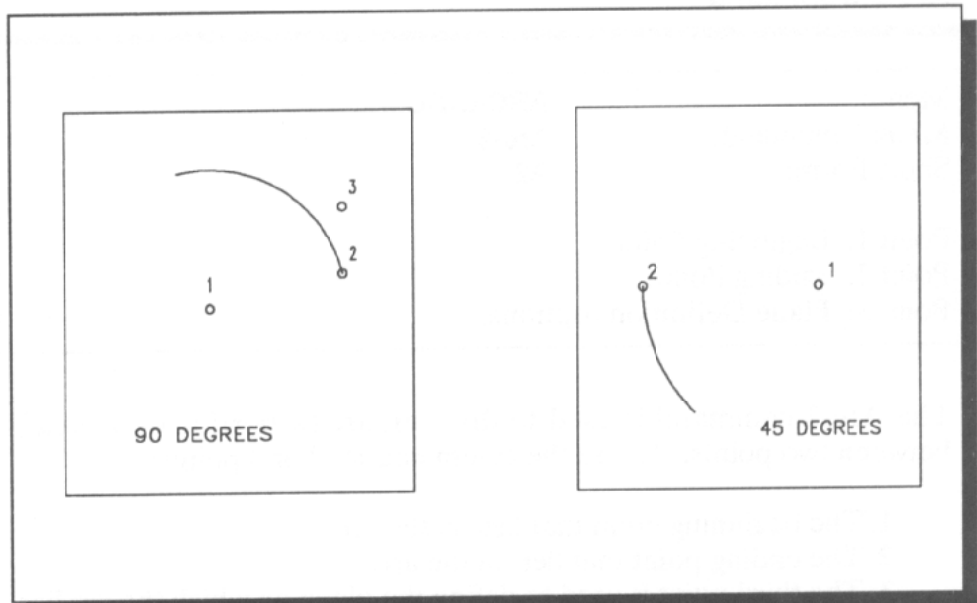


Figure 3 - Set a point at the center, a point at the beginning, and enter the angle. A third point can be used to specify the plane on which the arc lies.

Example: Draw a 90 degree arc on the X-Z plane. First set a point at the center of the arc. Next set a point for the beginning of the arc, in our example, directly to the right of the first point. Set the third point "in" on the Z axis some distance. The precise distance is not important, because this point only determines the plane on which the arc will lie. When asked, enter 90, and an arc will be drawn from the first point counter-clockwise 90°, back and to the left.

Related Topics: Vector Arc, Arc-2, Arc-3, Circle.

Arc-2 Command

| | |
|---------------|-----------------|
| Menu: | ARC/CIRC |
| Menu Command: | Arc-2 |
| Short Form: | A2 |

Point 1: Beginning Point.

Point 2: Ending Point.

Point 3: Plane Definition (optional).

The Arc-2 command is used to draw an arc (a portion of a circle) between two points. To use the command, set 2 or 3 points:

1. The beginning point that lies on the arc.
2. The ending point that lies on the arc.
3. The third point is used to define the plane on which the arc lies (optional).

You will be asked for the length of the radius of the arc. An arc of that radius will then be drawn from the first point counter-clockwise to the second.

The third point can be used, together with the first two points, to define the plane on which the arc lies.

Example: Draw an arc between two points with a radius of 47. Set a point at the arc's beginning point. This is the "left" point of the arc, if the arc is "bowed" upward. Set a second point at the arc's ending point. You can set a third point if you need to define the plane for the arc. When asked for the radius, enter **47**. An arc will be drawn from the first point clockwise to the second.

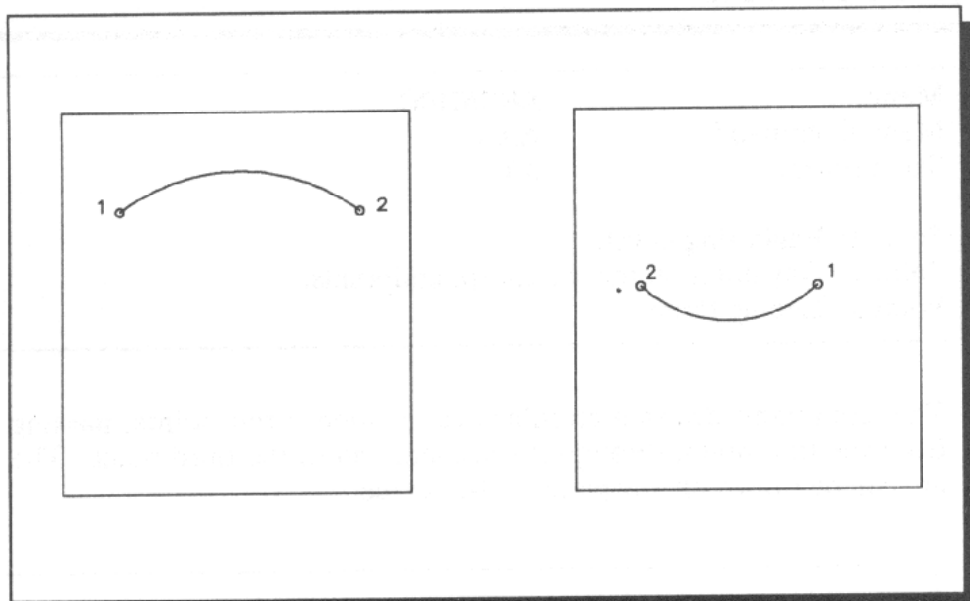


Figure 4 - Set 2 points for the arc beginning and end, and enter the radius. A third point can be used to specify the plane on which the arc lies.

Tip: With DesignCAD 3-D, you can enter a formula in brackets instead of a number. To enter a radius of $5/3$, enter **[5/3]** when asked for the radius.

Related Topics: Arc, Arc-3, Circle-2.

Arc-3 Command

Menu: **ARC/CIRC**
Menu Command: **Arc-3**
Short Form: **A3**

Point 1: Beginning point.

Point 2: Any point on the arc except endpoints.

Point 3: Ending Point.

This command draws a circular arc through three points, passing from the first point, through the second, and to the third point. The arc can be drawn on any plane in 3-D space.

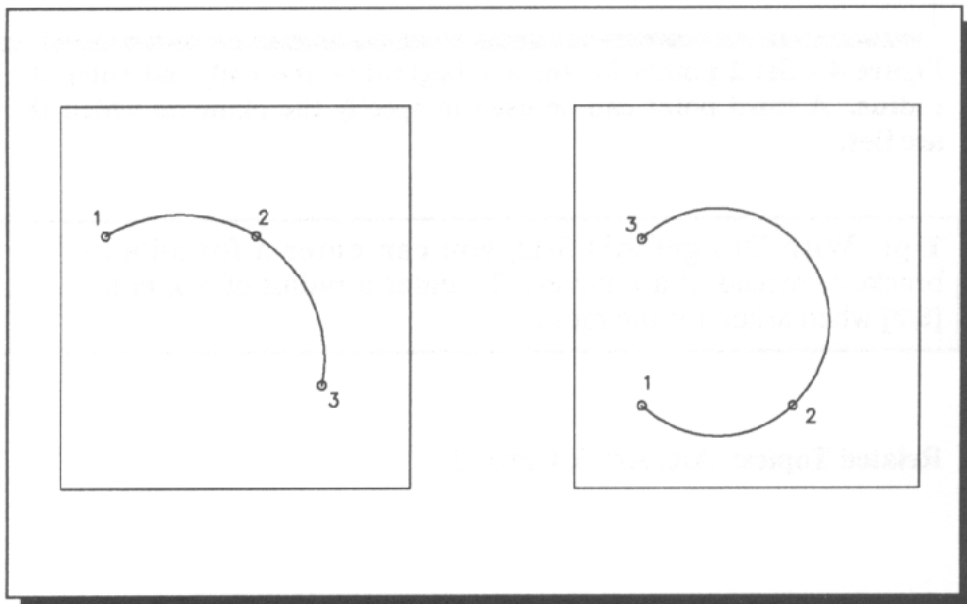


Figure 5 - Set 3 points through which the arc is to be drawn.

Example: Draw an arc through three points. Set a point at one of the endpoints for the arc. Next set a point for the arc to pass through. This point can be anywhere on the arc, except the endpoints. Then set a point on the other endpoint of the arc. A circular arc will be drawn from the first point through the second point to the third point.

Tip: The Arc-3 command is particularly useful when tracing a drawing on a digitizer.

Related Topics: Arc, Arc-2, Circle-3.



Arrow Command

| | |
|--------------------|--------------|
| Menu: | LINES |
| Menu Command: | Arrow |
| Short Form: | > |
| Keystroke Command: | > |

Point 1-n: Beginning points.

Point n+1: Location for the arrowhead.

This command draws an arrow from one point to another. To use the command, set two points. An arrow will be drawn from the first point to the second.

The size of the arrowhead is determined by the default text size. To change the arrowhead size, use the Text Size command.

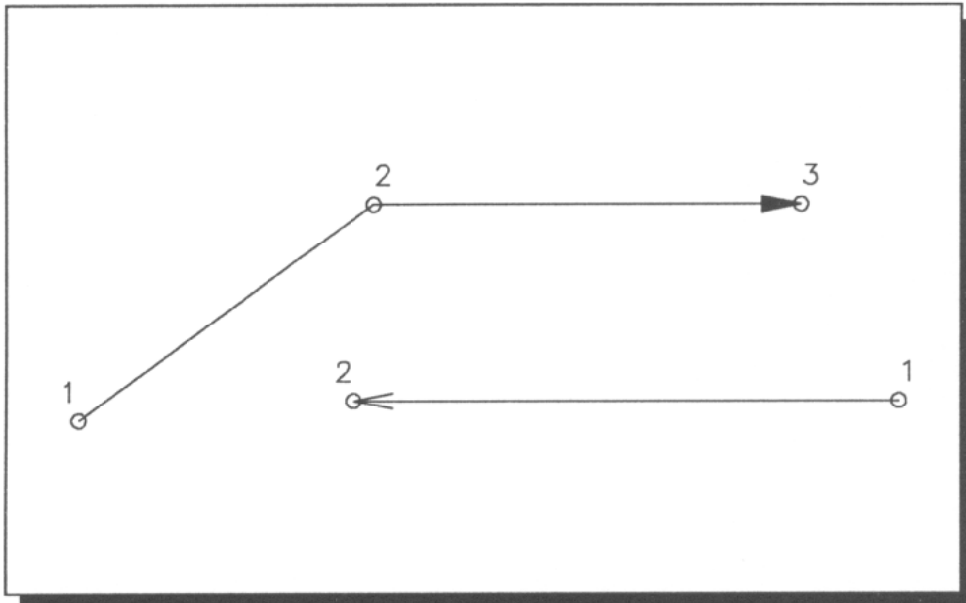


Figure 6 - An arrow is drawn through the points with an arrowhead at the last point.

If more than two points are used with this command, a line will be drawn through all the points and an arrow will be drawn at the last point.

Example: Draw an arrow. Set one point for the arrow beginning and a second point for the location of the tip of the arrowhead. An arrow will be drawn from the first point to the second, with the arrowhead tip on the second point.

Related Topics: Text Size, Line, Dimension.

Attribute Command

| | |
|--------------------|------------------|
| Menu: | NOTES |
| Menu Command: | Attribute |
| Short Form: | Attr |
| Keystroke Command: | \$ |

Point 1: Location for attribute.

The Attribute command is used to assign information to an object in the drawing. This information can later be extracted from the drawing file and used by other programs for applications such as parts lists and bills of materials.

To use the command, set a point where the Attribute is to be located and select **Attribute** from the NOTES menu or press the **\$** key. You will be asked for the Attribute. Enter a line of text that will be associated with that location in the drawing.

Attributes will not be displayed on the screen unless the Display Attribute flag is enabled under the System Parameter command. Even though you cannot see an Attribute, it can be moved and copied along with the rest of the drawing.

The Materials List program that comes with DesignCAD 3-D can be used to count and list the Attributes in a drawing.

Example: Make a drawing of a house, keeping track of the number and types of windows. Before you begin drawing the house, make a drawing of each type of window. In each window drawing, add an attribute that contains the window's name, size, and part number. Then make the house drawing, retrieving a window from disk whenever a window is needed. After the drawing is finished, use the Materials List program to print a report of the quantity of each window.

Related Topics: Materials List program.

Block Add Command

| | |
|---------------|------------------|
| Menu: | BLOCKS |
| Menu Command: | Block Add |

Point 1-n: Entities to be added to the Block.

The Block Add command adds one or more lines or entities to the current Block. This command can be used with the Block Reset command to define a Block by selecting certain entities in the drawing. It can be very useful if you need to select entities for a Block that are not located together in the drawing.

Example: Define a Block to have two lines, one on each end of the screen. We cannot use the Block Define command to make a Block out of both lines (unless there is nothing between the two lines). So, first set a point on the first line and press the **B** key. This defines the first line to be a Block, with a handle at the point that was set. Next, use the Block Add command and set a point on the second line. That line will be added to the Block, and the Block will contain both lines. The Block Add command does not move the Block Handle.

Related Topics: Block Define.

Block All Command

| | |
|---------------|------------------|
| Menu: | BLOCKS |
| Menu Command: | Block All |
| Short Form: | BA |

Point 1-3: Block Handles (optional).

The Block All command makes a Block of all visible entities (all entities in visible layers) in the drawing. To use this command, select **Block All** from the BLOCKS menu. You may optionally set 1, 2 or 3 points for the Block Handles.

Example: Change everything in the drawing to be Color 4. Use the Block All command to define the entire drawing to be a Block. In this case, since we are just changing the color, the location of the Block handles are not important. Next, use the Block Color command to change the color of the Block (the entire drawing) to color 4.

Related Topics: Block Define, Layer.

Block Circular Repeat Command

| | |
|---------------|---------------------|
| Menu: | BLOCKS |
| Menu Command: | Block Circle |
| Short Form: | BC |

Point 1: Location for the first Block in the circle.

Point 2: Center of rotation.

Point 3: Axis of revolution (optional).

The Block Circular Repeat command can be used to duplicate a Block several times around a circle, arc, or spiral. Two or three points are used with the command. The first point is the location for handle 1 of the Block.

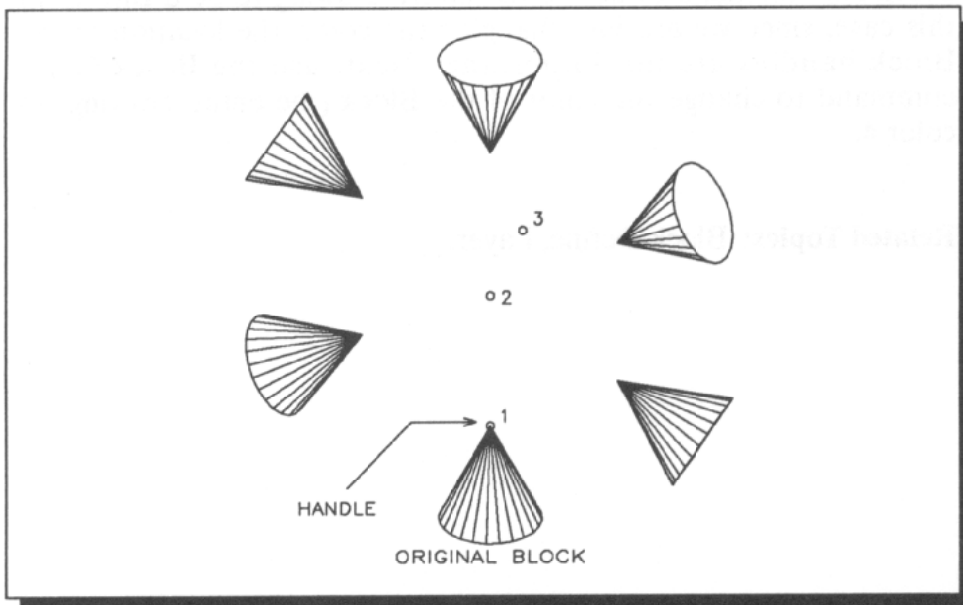


Figure 7 - Set a point for the location of the first copy and two points for the axis of revolution.

The second point is the center of the circle about which the Block is to be duplicated. The third point defines, together with the second,

the axis about which the Block is to be duplicated. If the third point is not used, the Block is duplicated about the second point parallel to the Z axis.

If the first copy lies directly on the original Block, the Block will not be duplicated onto itself. Otherwise, the first copy of the Block will be at the first point.

When asked for the number of copies, enter the number of copies of the Block you would like to have.

When asked for the number of degrees, enter 180 to copy the Block in a semicircle, 360 for a circle, etc. An angle greater than 360 degrees can be entered to duplicate the Block several revolutions. This is useful when copying a Block in a Spiral.

When asked for the offset, enter 0 to duplicate the Block in a circle, or the spiral offset to duplicate the Block in a spiral. The spiral offset is the distance along the axis of revolution (points 2 & 3) between the first and the last copy of the Block.

Example: Draw the stairs for a spiral staircase. First, draw a single step and define it to be a Block. Set the Block handle on an inside corner of the step. Next, use the Block Circular Repeat command to copy the step. Set the first point at the location for the bottom step. Set the second point at the center of the staircase, and the third point immediately above the second. When asked, enter 20 copies in 720 degrees (2 complete circles), with an offset of 10. (10 feet with 20 copies would make 6" steps.)

Tip: The location of the first Block handle is important. It is used as the primary reference point by most Block commands. When you define a Block, choose the Block handle judiciously.

Related Topics: Block Define, Block Sweep.

Block Color Command

| | |
|---------------|--------------------|
| Menu: | BLOCKS |
| Menu Command: | Block Color |
| Short Form: | BCO |

Points: None.

The Block Color command is used to change the color of all the lines or entities in a Block. After entering the new color for the Block, each entity in the Block is redrawn in the new color.

Example: Change everything in the drawing to be Color 4. Use the Block All command to define the entire drawing to be a Block. In this case, since we are just changing the color, the location of the Block handles are not important. Next use the Block Color command to change the color of the Block (the entire drawing) to color 4.

Related Topics: Color, Drawing Parameters.

Block Define Command

| | |
|--------------------|-------------------------------|
| Menu: | BLOCKS |
| Menu Command: | Block define |
| Menu Command: | Block defineC (cutoff) |
| Short Form: | B |
| Keystroke Command: | B |
| Keystroke Command: | ^B (cutoff) |

Point 1: Corner of 3-D Block area (or single entity for Block).

Point 2: Opposite corner of 3-D Block area.

Points 3-5: Block handles (optional).

The Block Define command is used to define a Block. A Block is a subset of the drawing that can be manipulated with the Block commands. There are many DesignCAD 3-D Block commands. Each of these commands requires a Block to be set before it can be used.

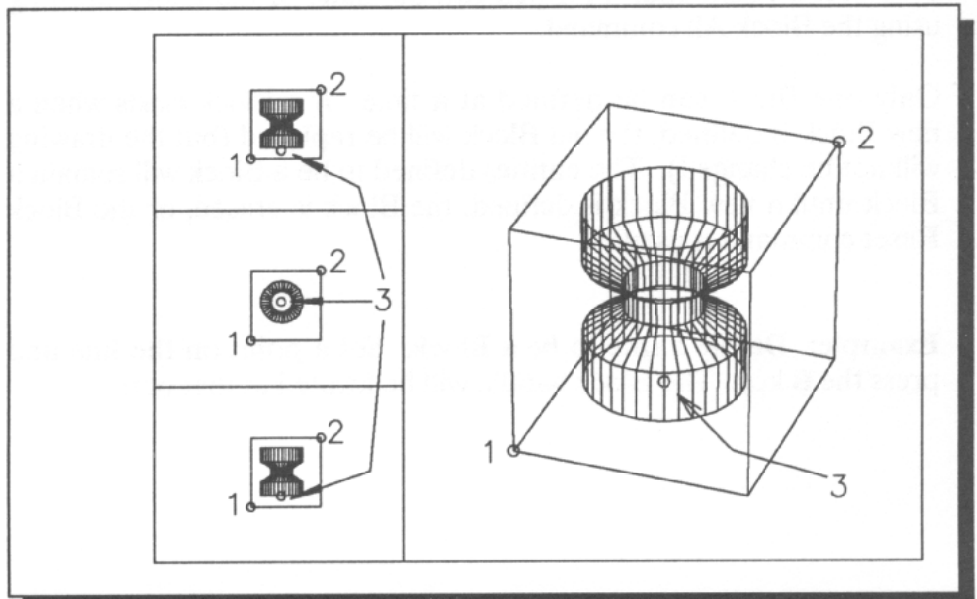


Figure 8 - Set points in opposite corners of a 3-dimensional section that contains the Block.

To use the Block Define command, define the Block by setting two points in opposite corners of a 3-Dimensional box. Any drawing entity (e.g., line, curve, surfaces) within the section will become part of the Block. A third, fourth, and fifth point may be used to define the location of the Block Handles.

The Block Define command selects the drawing entities located entirely within the section or box. If an entity lies partly within and partly outside the section, it will not be included in the Block.

The Block Define-Cutoff (Block DefineC or ^B) is the same as the Block Define command, except that entities partly inside and partly outside the section will be "broken" or cut off. Those parts inside the section will become part of the Block.

If a single point is used with this command, then the nearest entity to that point will become the Block, and that point will be the position of the primary Block handle. This is the easiest way to make a Block of a single line or entity.

Additional drawing entities can be added to the Block using the Block Add command. A Block can be made of the entire drawing using the Block All command.

Only one Block can be defined at a time. If a Block exists when a new Block is defined, the old Block will be replaced (but the drawing will not be changed). The entities defined to be a Block will remain a Block until a new Block is defined, the Block is erased, or the Block Reset command is used.

Example: Define a line to be a Block. Set a point on the line and press the B key. The Block handle will be located at that point.

Example: Define two boxes to be a Block. Set a point in front, below, and to the left of both boxes. Set a second point above, to the right, and behind both boxes. Set a third point at a convenient place for the primary Block handle.

Related Topics: Block All, Block Add, Block Reset.

Block Delete Command

| | |
|---------------|---------------------|
| Menu: | BLOCKS |
| Menu Command: | Block Delete |
| Short Form: | BD |

Points: None.

The Block Delete command erases or deletes from the drawing each line or entity in the Block. If this command is executed in error, the Unerase command can be used to "unerase" the Block one entity at a time.

Example: Make a copy of part of a drawing and then erase the original. To do this, define the part of the drawing to be copied as a Block. Then copy the Block using the Block Insert command. If the copy is correct, use the Block Delete command to erase the original Block. This is one way to do a Block Move and verify the result before erasing the original Block.

Related Topics: Erase, Unerase.

Block Extrude Command

| | |
|--------------------|----------------------|
| Menu: | BLOCKS |
| Menu Command: | Block Extrude |
| Short Form: | X |
| Keystroke Command: | X |

Point 1: Where the extrusion starts.

Point 2: Extrusion direction and distance.

Point 3-n: Change in direction, distance and/or size (optional).

The Block Extrude is a very powerful feature of DesignCAD 3-D. It allows you to take a line or plane entity and extrude it along a set of points. The extrusion can be scaled larger or smaller, and it need not follow a straight line.

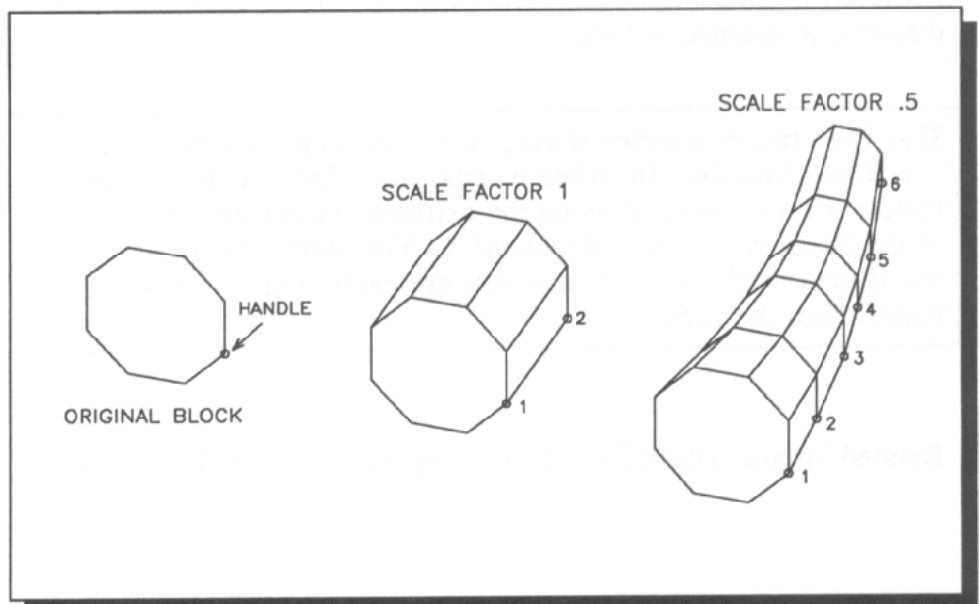


Figure 9 - The Block Extrude command.

To use the command, set two or more points to define the direction and distance of the extrusion. The first point is the point where the first occurrence of the Block is to be located. The Block handle is

located at the first point. The Block will be extruded from the first point through the other points, with the first Block handle, or primary handle, located at the points.

Next, you will be asked for the scale factor. The scale factor is the relative size of the Block at the last point of the extrusion. A scale factor of 1 causes the Block to remain the same size. A scale factor of .5 reduces the end of the extrusion by 1/2, and a scale factor of 0 extrudes the Block down to a point.

The Block is extruded the direction and distance from the first point to the second, from the second point to the third, etc.

Example: Draw a hexagonal rod 3 feet long. First draw a six-sided polygon on the X-Y plane using the Polygon command. Then define the Polygon to be a Block using the Block Define command - set a point on the polygon and press **B**. Set one point on the polygon and a second point 3 feet "in" along the Z axis from the first point, and press **X** for Block Extrude. The polygon will be extruded in the Z direction a distance of 3 feet.

Tip: If an object is scaled during an extrusion, it is scaled about the Block handle. In other words, the Block at the Block handle will be extruded along the extrusion points and the rest of the Block will be scaled around it. You should be aware of the Block handle's location if you are extruding with a scale factor other than one.

Related Topics: Block Extrude Varying, Block Sweep, Block Spiral.

Block Extrude Varying Command

| | |
|--------------------|---------------------|
| Menu: | BLOCKS |
| Menu Command: | Block ExtVar |
| Short Form: | BXV |
| Keystroke Command: | ^X |

Point 1: Where the extrusion is to start.

Point 2: Extrusion direction, distance and size.

Point 3-n: Changes in direction, distance and/or size (optional).

The Block Extrude Varying command is like the Block Extrude command, except that the extrusion scale can be varied at each point along the extrusion.

To use the command, set two or more points to define the direction and distance of the extrusion. Next, you will be asked for the scale factor for each point except the first. (The scale factor at the first point is always 1.)

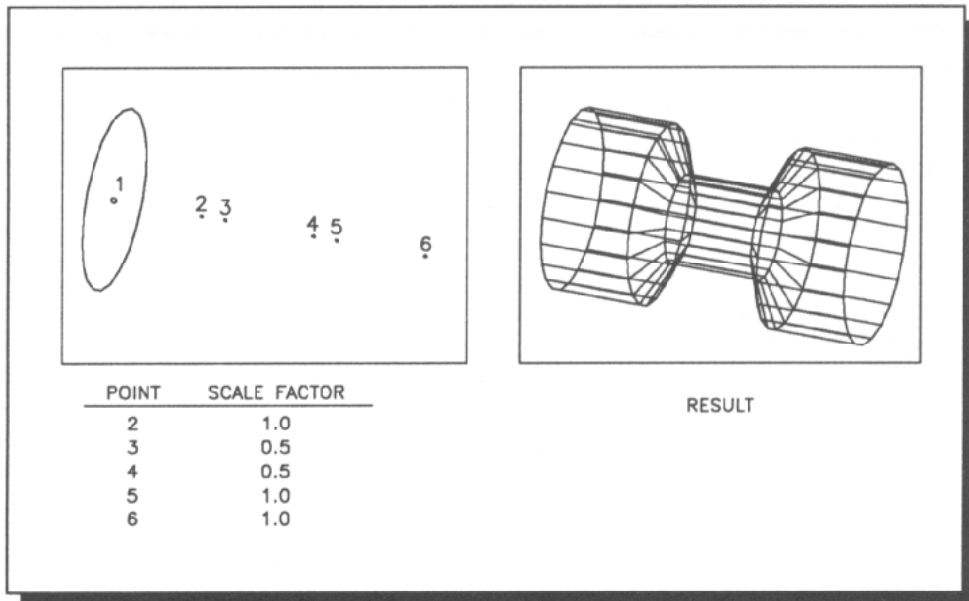


Figure 10 - With the Block Extrude Varying command, you can specify a scale factor for each extrusion point.

For example, in Figure 10 extrusion scales of 1, .5, .5, 1, and 1 were entered to make a shaft. Notice that you can set more than one point in the same location to change the diameter or scale at that location.

The scale factor is the size of the Block at that point, relative to the Block's original size. A scale factor of 1 causes the Block to remain the same size. A scale factor of .5 reduces the size by 1/2, and a scale factor of 2 doubles the size.

The Block is extruded the direction and distance from the first point to the second, from the second point to the third, etc. At each point, the Block is duplicated. The copies of the Block are connected with surfaces.

Example: Draw a round shaft with varying thicknesses. Use the Disc command to draw a round surface on the X-Z plane - set points at (0, 0, 0), (0, 0, 5), and (0, 1, 5). Then set a point at (0,0,0) and press the **B** key to make a Block out of the circular plane. The Block handle will automatically be at location 0,0,0. Now, set several points

directly to the right of the first point. Press ^X or select Extrude Varying from the menu. Enter different scale factors for the different extrusion points.

Tip: You can set two points in the same place and use different scale factors to "jump" to a different diameter or size in the extruded object.

Related Topics: Block Extrude, Block Sweep, Block Spiral.

Block Handle Command

| | |
|---------------|---------------------|
| Menu: | BLOCKS |
| Menu Command: | Block Handle |
| Short Form: | BH |

Point 1: Block Handle 1.
Point 2: Block Handle 2 (optional).
Point 3: Block Handle 3 (optional).

A Block can have one, two, or three reference points or handles. These handles are used with several of the Block commands as reference points for manipulating the Block.

The first Block handle, or primary handle, is used to determine the location of the Block for DesignCAD 3-D commands that move or copy the Block. The Block Insert, Block Move, Block Drag, Block Repeat, and Block Extrude commands are among the DesignCAD 3-D commands that use the primary Block handle to determine the location of the Block.

The Block Zoom, Block Scale, Block Mirror, and Block Rotate commands use the primary Block handle for the "origin" of the Block manipulation. For example, the Block Rotate command rotates the Block about the primary Block handle.

The second Block handle is used to determine the size and angle at which the Block is to be positioned in commands such as the Block Insert, Block Move, and Block Drag.

Commands that use the second Block handle also use the third Block handle to determine the plane at which the Block is positioned, that is, the rotation about the line between the first two handles. When the third Block handle is used, the plane defined by the three Block handles is used to determine the position of the Block.

To use this command to set the location for the Block handles, select **Block Handle** from the BLOCKS menu and set one, two, or three points where you want the handles to be located.

If no handles are specified for a Block, the front lower left, lower right, and front upper right corners of the Block are used. The handles can also be defined with the Block Define command.

Example: Place a box so it's center of gravity is on the top point of a cone. First, define the box to be a Block using the Block Define command. Next, use the Center of Gravity command to set a point at the box's center of gravity. Then select the Block Handle command to make that point the Block's primary handle. Use the Block Move command to move the box to the cone's top by setting a single point at the top of the cone. The box will then be moved so its center of gravity is on the top point of the cone.

Related Topics: Block Define, Block All.

Block Insert Command

Menu: **BLOCKS**
Menu Command: **Block Insert**
Short Form: **BI**

Point 1: Location for Block handle 1.
Point 2: Location for Block handle 2 (optional).
Point 3: Location for Block handle 3 (optional).

The Block Insert command makes a copy of the Block. One, two, or three points may be used to specify the location where the Block is to be copied. The primary handle of the Block is positioned at the first point.

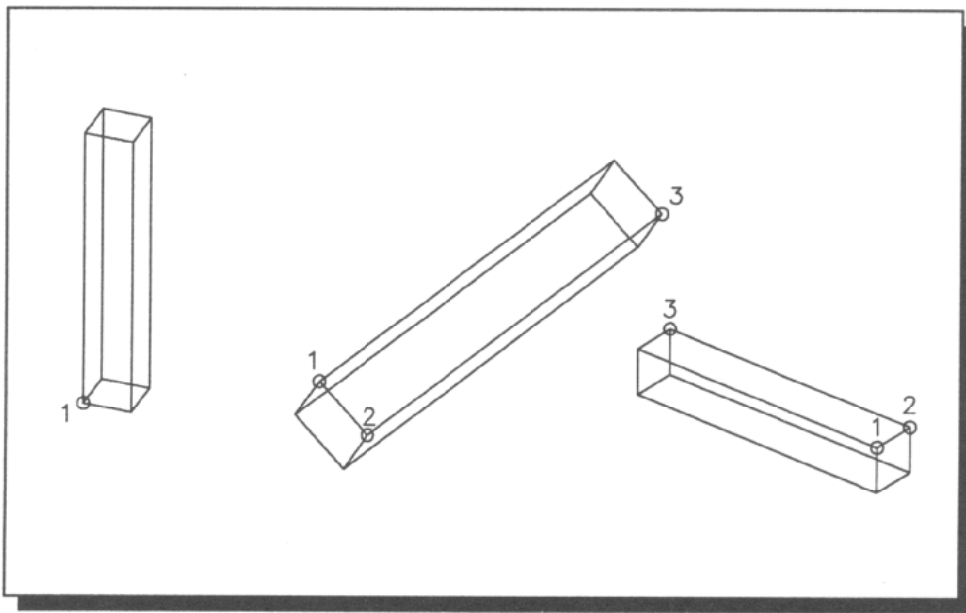


Figure 11 - One, two, or three points can be used to orient the Block.

If a second point is used, the Block's size and angle is adjusted so that the secondary handle is located at the second point. If only one point is used, then the Block will be located at that point at the Block's original size and angle.

If a third point is used, the Block will be positioned at an angle so that the three Block handles lie on the same plane as the three points.

If no handles have been specified for the Block, the front lower left, front lower right, and front upper right corners of the Block are used for handles.

This command is identical to the Block Move command, except that the Block is copied instead of moved.

Example: Make a copy of a Block rotated 90° about the X axis. First define an object to be a Block (a box, or anything). Set the Block handles to be located at the lower left front corner, lower right front corner, and the upper right front corners of the Block. This can be done in the Block Define command or the Block Handle command. Now use the Block Insert command and set 3 points: one for the location of the lower left front corner of the copy, one directly to the right of the first point, and one "in" on the Z axis. (Press **F1**, then **Insert** after setting the second point.) The precise location of the third point is not important, since it is used only to define the plane. The original Block handles were all on the X-Y plane. The three points set with the Block Insert command were on the X-Z plane, so the Block was rotated over on the X axis when it was copied.

Tip: If you need to copy a Block several times, you can press **F3** and then **Enter** to execute the last command. The Block Repeat command and Block Circular Repeat commands can be used to copy a Block a number of times in a line or in a circle.

Related Topics: Block Move, Block Repeat, Block Circular Repeat, Block Mirror.

Block Layer Command

| | |
|---------------|--------------------|
| Menu: | BLOCKS |
| Menu Command: | Block Layer |
| Short Form: | BL |

Points: None.

The Block Layer command is used to change the layer of all the lines or entities in a Block. After the new layer number is entered, each entity in the Block is assigned to the new layer.

This command makes it easy to reconfigure your drawing into different layers.

Tip: If you are making a drawing in layers and mistakenly draw something in the wrong layer, this command makes it easy to move it back to the correct layer.

Related Topics: Layer.

Block Load Command

| | |
|---------------|-------------------|
| Menu: | BLOCKS |
| Menu Command: | Block Load |
| Short Form: | BLO |

Point 1: First drawing handle (optional).
Point 2: Second drawing handle (optional).
Point 3: Third drawing handle (optional).

The Block Load command retrieves a DesignCAD 3-D drawing and defines it to be a Block. The Block handles are the handles of the drawing. The Block Load command is the same as the Retrieve command, except that the drawing retrieved becomes a Block.

This command can be used to retrieve a symbol and have that symbol automatically become a Block so that it can be moved, copied or manipulated with the Block commands.

Example: Retrieve a symbol called BRACKET and place it in two locations in the drawing. Use the Block Load command to load the drawing from disk, and place it by setting a point for the primary Block handle. The symbol will be scaled automatically to the drawing on the screen unless a second point is set to define the size of the symbol. Use the Block Insert command to make a copy of the Block at the second location.

Tip: Use the Block Load command to load a symbol (or merge drawings), because if it is loaded into the wrong place or if the wrong symbol is loaded, you can easily move it with the Block Move command or erase it with the Block Delete command.

Related Topics: Retrieve, Block Save.

Block Mirror Command

Menu: **BLOCKS**
Menu Command: **Block Mirror**
Short Form: **BMI**

Point 1: Location for the "mirrored" Block.

The Block Mirror command is used to make a mirror image out of a Block. One point is used to specify the location of the "mirrored" copy of the Block. The first Block handle will be located at that point.

You will be asked "Mirror normal to the X, Y, or Z axis?" If the Block is to be mirrored along the X axis, then enter X. Likewise, enter Y or Z for the Y or Z axis.

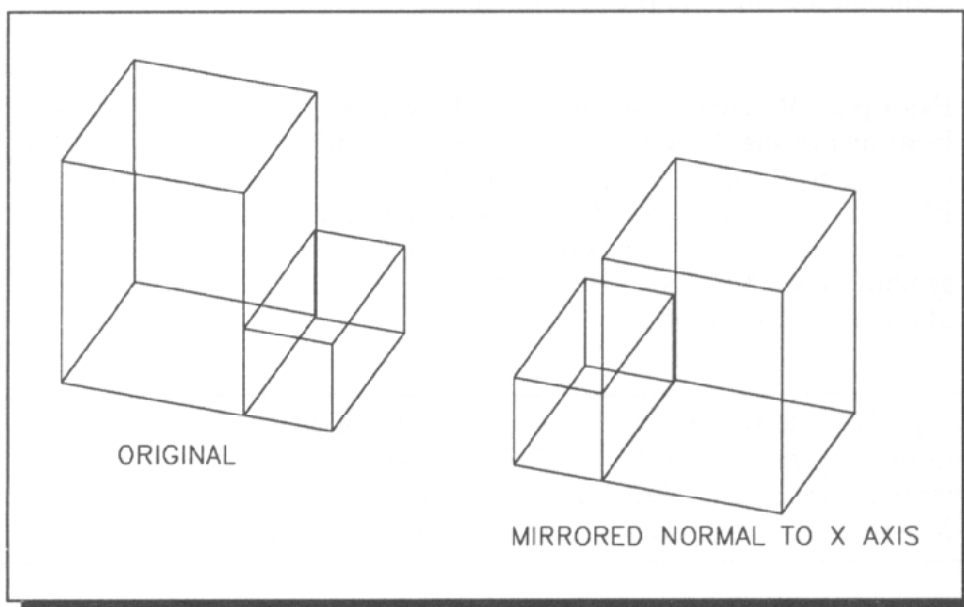


Figure 12 - This object was mirrored normal to the X axis.

To understand the mirror operation normal to the X (Y or Z) axis, assume that the X (Y or Z) axis passes directly through the mirror, perpendicular (normal) to the mirror.

This command makes a copy of the Block. It does not affect the original Block.

Example: Make a "horizontal" mirror image of an object. First define the object to be a Block. Then use the Block Mirror command to make a "mirrored" copy of the Block. Set a point for the location of the mirror image (the location for the Block handle). When asked, specify normal to the X axis for a horizontally mirror image. A copy of the original Block will then be made at the new location.

Tip: To convert an object to a mirror image of itself, use the Block Mirror command and then erase the original Block with the Block Delete command. You may want to regenerate the screen by Zooming with a factor of 1 after the Block Delete command.

Related Topics: Block Insert.

Block Move Command

| | |
|--------------------|-------------------|
| Menu: | BLOCKS |
| Menu Command: | Block Move |
| Short Form: | BM |
| Keystroke Command: | M |

Point 1: Location for first Block handle.

Point 2: Location for the second Block handle (optional).

Point 3: Location for the third Block handle (optional).

The Block Move command moves a Block from one location to another. One, two, or three points may be used to specify the location where the Block is to be moved. The primary handle of the Block is positioned at the first point.

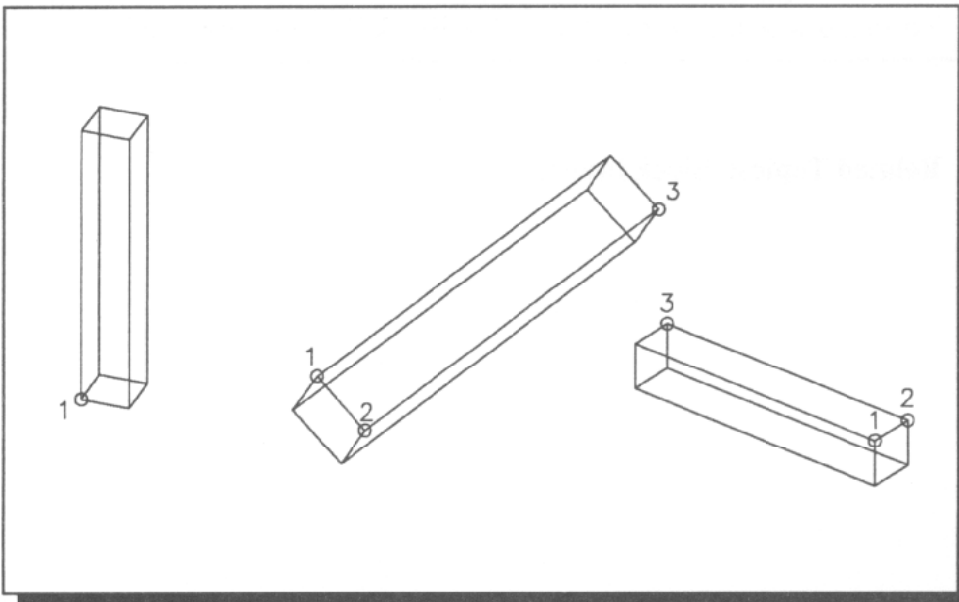


Figure 13 - One, two, or three points can be used to orient the object.

If a second point is used, the Block's size and angle is adjusted so that the secondary handle is located at the second point. If only one point

is used, then the Block will be located at that point at the Block's original size and angle.

If a third point is used, the Block will be positioned at an angle so that the three Block handles lie on the same plane as the three points.

If no handles have been specified for the Block, the lower left and lower right corners of the Block are used for handles.

This command is identical to the Block Insert command except that the Block is moved instead of copied.

Example: Place a box so its lower left front corner is on the top point of a cone. First, define the box to be a Block using the Block Define command. Set the first Block handle to be the lower left front corner of the box. Use the Block Move command to move the box to the cone's top by setting a single point at the top of the cone. The box will then be moved so its corner is on the top point of the cone.

Related Topics: Block Insert.

Block Repeat Command

| | |
|---------------|---------------------|
| Menu: | BLOCKS |
| Menu Command: | Block Repeat |
| Short Form: | BR |

Point 1: Location for the first copy.

Point 2: Location for the second copy.

The Block Repeat command makes it possible to duplicate a Block a number of times in a straight line.

To use this command, select **Block Repeat** from the **BLOCKS** menu. Set one point at the location for the first copy, and another point at the location for the second copy of the Block. Then enter the number of times the Block is to be duplicated.

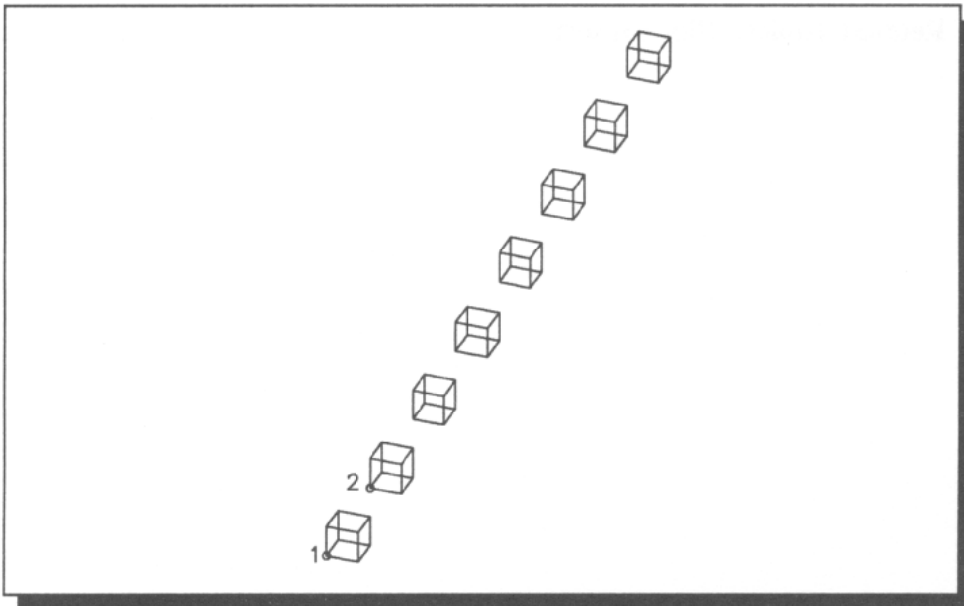


Figure 14 - Set a point at the locations for the first and second copies, and enter the number of copies to be made.

The Block will then be duplicated. The first copy is located with its primary handle on the first point, the second copy is located with its primary handle at the second point, and each subsequent copy is offset the same distance and direction as the first two.

If the first copy lies directly on the original Block, the Block will not be duplicated onto itself. Otherwise, the first copy of the Block will be at the first point.

Example: Copy a single step to make a stairway 20 feet long and 10 feet high. First draw a step and define it to be a Block. It should be about 6" tall. Next, use the Block Repeat command and set one point at the Block handle of the step, and a second point 1 foot to the right and 6" above the first. The second point is the location of the second step. This assumes your stairway will go to the right. Otherwise, set the second point in the appropriate direction. When asked, enter **20** copies. Then 20 steps (including the original one) will be copied, each being 1 foot away horizontally and 6" above the last.

Related Topics: Block Circular Repeat, Block Insert.

Block Reset Command

| | |
|---------------|--------------------|
| Menu: | BLOCKS |
| Menu Command: | Block Reset |
| Short Form: | BRE |

Points: None.

The **Block Reset** command resets a Block definition, or "undefines" the Block. This can be used when a new Block is to be defined using the **Block Add** command.

The **Block Define** command automatically resets any existing Block definition. The **Block Reset** command does not have to be used before the **Block Define** command.

Related Topics: **Block Define**, **Block Add**.

Block Rotate Command

| | |
|---------------|---------------------|
| Menu: | BLOCKS |
| Menu Command: | Block Rotate |
| Short Form: | BRO |

Point 1: Center of rotation (optional).

The Block Rotate command rotates a Block. You can rotate the Block about the X, Y, or Z axis any number of degrees.

One point may be used to specify the location about which the Block is to be rotated. If no point is set, then the Block is rotated about the primary Block handle.

The Block is rotated counter-clockwise. A negative angle may be entered to rotate the Block clockwise.

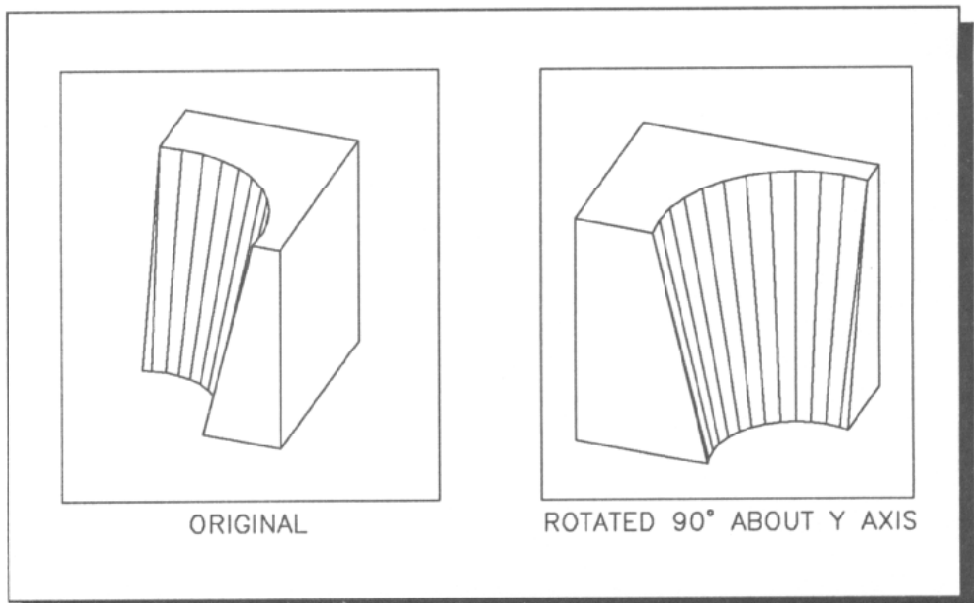


Figure 15 - The Block Rotate command.

This command does not make a new copy of the Block. It merely rotates the original Block.

Example: Rotate a box 45° about the Y axis. Define the box to be a Block. Then use the Block Rotate command to rotate the Block. When asked to set a point, set a point about which the Block is to be rotated or press Enter to rotate about the Block handle. Then enter the **Y** axis and **45** degrees when asked. The Block will then be rotated.

Tip: Use this command to rotate a portion of the drawing. Use the Rotate command to rotate the entire drawing.

Related Topics: Block Move, Rotate.

Block Save Command

| | |
|---------------|-------------------|
| Menu: | BLOCKS |
| Menu Command: | Block Save |
| Short Form: | BS |

Point 1: Drawing handle 1 (optional).

Point 2: Drawing handle 2 (optional).

Point 3: Drawing handle 3 (optional).

The Block Save command saves a Block to disk as a DesignCAD 3-D drawing file. The reference points or handles of the drawing saved to disk may be specified by entering one, two, or three points before the command is entered.

If no points are used with this command, the handles of the Block are used for the drawing handles.

Example: Make a symbol out of part of the drawing for later use. Define the appropriate part of the drawing to be a Block. Then use the Block Save command to save the drawing. (A symbol is just a DesignCAD 3-D drawing.)

Related Topics: Save, Block Load, Retrieve.

Block Scale Command

| | |
|---------------|--------------------|
| Menu: | BLOCKS |
| Menu Command: | Block Scale |
| Short Form: | BSC |

Point 1: Point about which the Block is to be scaled (optional).

The Block Scale command is used to scale a Block along the X, Y, or Z axis. In other words, you can "stretch" the Block to make it taller, shorter, longer, wider, etc.

To use the command, select **Block Scale** from the BLOCKS menu. You will be asked for the X, Y, and Z scale factors. The X scale factor determines the relative Block size along the X axis (left to right). The Y scale factor determines the relative height of the Block, and the Z scale factor is for the Block size along the Z axis (front to back).

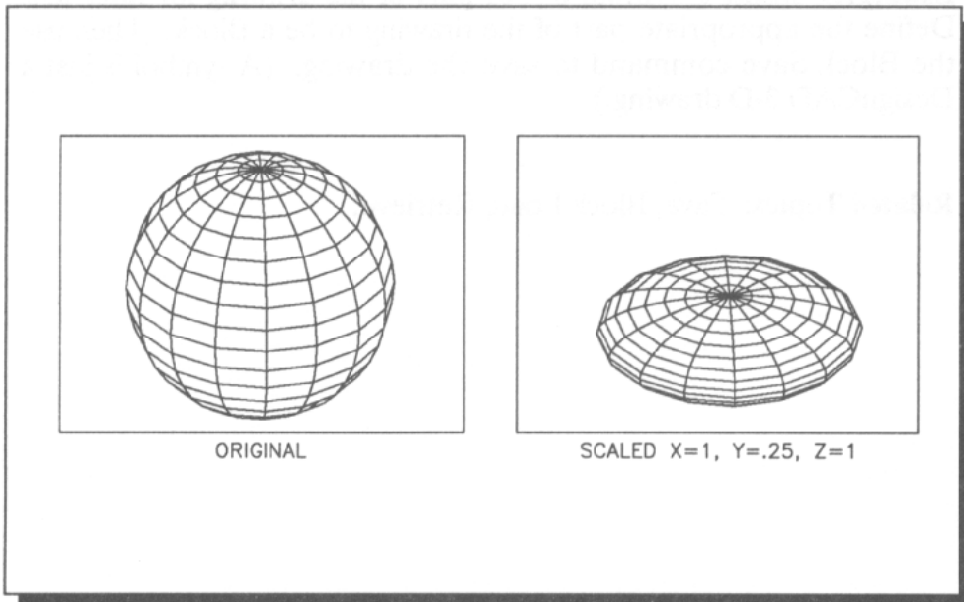


Figure 16 - The Block Scale command scales a Block on the X, Y, and Z axis.

The Block Scale command can be used to easily construct objects that would otherwise be difficult or impossible to draw. For example, you can easily use the Block Scale command to construct an ellipse from a circle.

Example: Make a "flattened" sphere. Draw a sphere and define it to be a Block. Set a point in the center of the sphere and enter **Block Scale** or **BSC**. Enter an X scale factor of 1 (no change), a Y scale factor of .5 (half as tall), and a Z scale factor of 1 (no change). The sphere will be drawn with the same width and depth, but will be "flattened" to be only half as tall as it was.

Tip: Make sure you know where the first Block handle is located because the scaling will take place about that point.

Related Topics: Block Zoom.

Block Slice Command

| | |
|---------------|--------------------|
| Menu: | BLOCKS |
| Menu Command: | Block Slice |
| Short Form: | BSL |

Points 1,2,3: Cutting plane.
Point 4: Side of the plane to be erased.

The Block Slice command is used to erase parts of a Block in front of or behind a plane. The plane can be any plane defined by three points.

The fourth point can be set to determine which side of the plane is to be erased. All parts of the Block that lie on that side of the cutting plane will be erased. All lines and surfaces that cross the plane will be broken at the plane. If the fourth point is not used, then all entities and parts of entities in front of the cutting plane will be erased.

Example: Cut a sphere in half diagonally. Define the sphere to be a Block. Then select **Block Slice** from the menu and set three points to define the "cutting plane." To define the diagonal cutting plane, set one point at the center of the sphere, one point up and to the right of the first, and a third point directly behind the second (in the positive Z direction). Set a fourth point to the right of the imaginary cutting plane to specify which part to erase. The right half of the sphere will be removed.

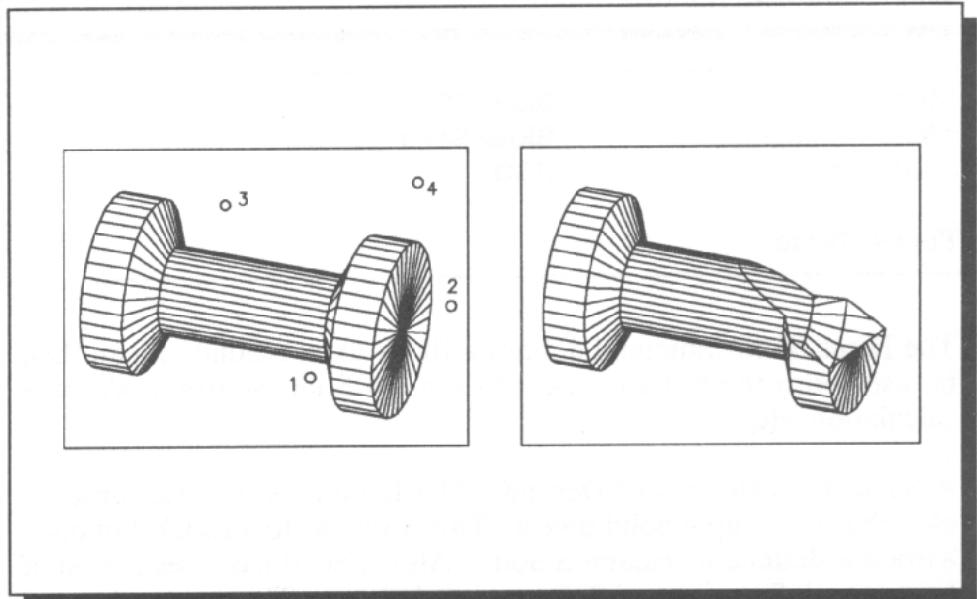


Figure 17 - Set 3 points to define the cutting plane, and a fourth point to indicate which side of the plane is to be erased.

Tip: When setting points for the cutting plane, look at the views at the left of the screen to make sure the points are in the correct locations.

Tip: The Block Slice command erases only parts of the Block. The Slice command can be used to "slice" the entire drawing.

Related Topics: Slice, Solid Subtract.

Block Solid Command

| | |
|---------------|--------------------|
| Menu: | BLOCKS |
| Menu Command: | Block Solid |
| Short Form: | BSO |

Points: None.

The Block Solid command defines a Block to be a Solid. It can then be used with the Solid operations such as Solid Subtract, Volume calculation, etc.

A Solid is a collection of DesignCAD 3-D entities (planes, surfaces, etc.) that make up a Solid object. This is similar to a Block, but once a Solid is defined it remains a Solid. Also, a Solid has a restriction of having to define a closed 3-dimensional object. This means that not all Blocks can be legitimate Solids.

When you use this command, be sure that the object defined as a Block is actually a Solid.

All the planes and surfaces making up a Solid must form a closed solid object. Figure 18 shows some valid and invalid Solids. If you define an invalid Solid, the Solid operations may not work correctly.

When one of the DesignCAD 3-D solid objects is drawn (Box, Cylinder, Cone, Sphere, and Hemisphere), it is automatically defined to be a Solid. Also, when a solid object results from a Block Extrude, Block Sweep, or Block Spiral command, it will automatically be defined as a Solid.

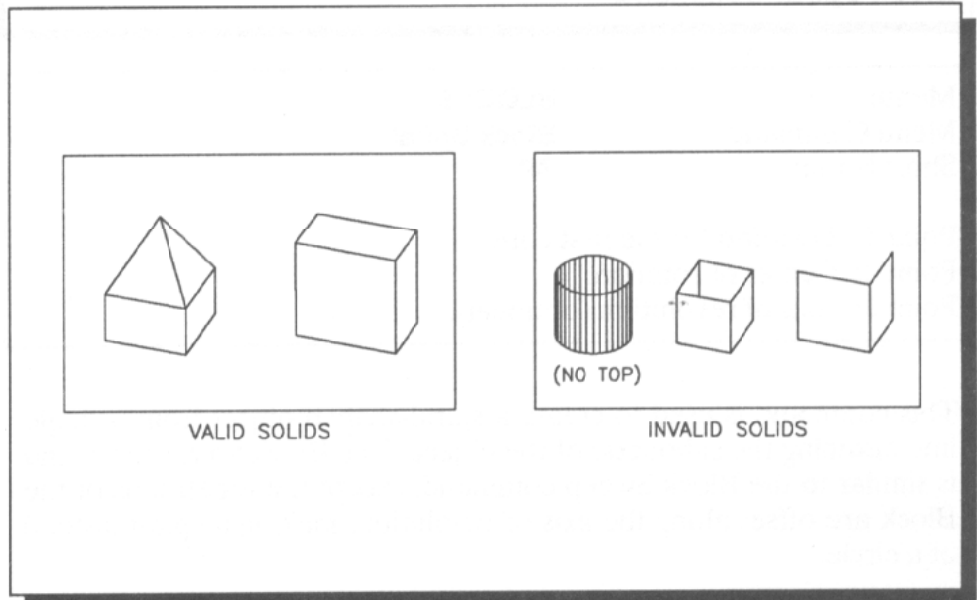


Figure 18 - A Solid is a closed figure made up of planes and surfaces.

Tip: Use the ID command if you need to determine whether a
an object is defined as a Solid.

Related Topics: Solid Define.

Block Spiral Command

| | |
|---------------|---------------------|
| Menu: | BLOCKS |
| Menu Command: | Block Spiral |
| Short Form: | SP |

Point 1: Location for the first copy.

Point 2: Center of rotation.

Point 3: Axis of revolution (optional).

This command is used to create a spiral-shaped object from a single line outlining the silhouette of the object. The Block Spiral command is similar to the Block Sweep command, except the repetitions of the Block are offset along the axis of revolution, making a spiral instead of a circle.

Three points are used with the command. The first point is the location for handle 1 of the Block.

The second point is the center of the circle about which the Block is to be duplicated. The third point, together with the second, defines the axis about which the Block is to be replicated. If the third point is not used, the Block is duplicated about the second point parallel to the Z axis.

Next, the number of times the Block is to be copied and the number of degrees in which these copies are to be placed are entered. (Press **Enter** for a full circle.) More than 360 degrees can be entered to make more than one complete revolution.

The total offset is then entered to specify the total offset along the axis of revolution. This is the distance along the axis of revolution from the first occurrence of the Block to the last occurrence.

The Block will then be copied in a spiral, with each copy connected by surfaces.

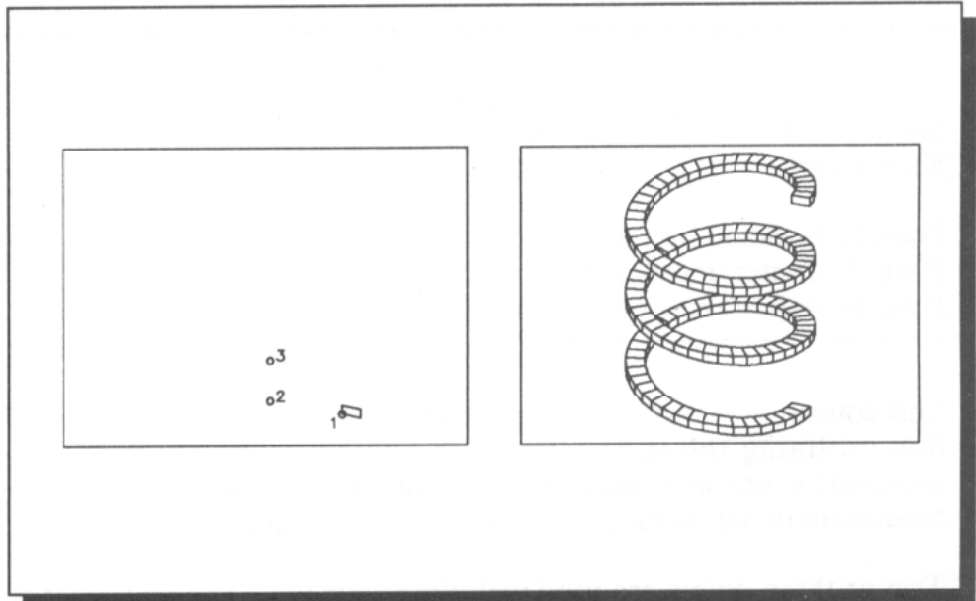


Figure 19 - The Block Spiral command extrudes a Block in a spiral. Set a point for the first copy and two points for the axis of revolution. Then enter the number of degrees and total offset. In this example, the Block was extruded 1080 degrees with an offset of 25.

Example: Construct a spring that is ten units long and has 6 revolutions. Draw a circle the size of the spring wire. Then make a Block out of the circle - set a point at the circle center and press **B**. Next set three points: one at the circle center, one to the side of the circle, and a third point directly above the second. Select **Block Spiral** from the **BLOCKS** menu. When asked for the angle, enter 2160 (6 times 360 for 6 revolutions). When asked for the offset, enter 10. A spring will then be drawn with 6 revolutions and a distance of 10 between the center of the beginning and ending of the "wire."

Related Topics: Block Sweep.

Block Sweep Command

| | |
|---------------|--------------------|
| Menu: | BLOCKS |
| Menu Command: | Block Sweep |
| Short Form: | SW |

Point 1: Location for the first copy.

Point 2: Center of rotation.

Point 3: Axis of revolution (optional).

This command is used to create a circular solid object from a single line outlining the silhouette of the object. The Block Sweep command is similar to the Block Circular Array command, except the repetitions of the Block are "tied together" with surfaces.

Two or three points are used with the command. The first point is the location for handle 1 of the Block.

The second point is the center of the circle about which the Block is to be duplicated. The third point, together with the second, defines the axis about which the Block is to be duplicated. If the third point is not used, the Block is duplicated about the second point parallel to the Z axis.

Next, the number of times the Block is to be copied is entered, and the number of degrees in which these copies are to be placed. (Press **Enter** for a full circle.)

The Block will then be copied in a circular manner, with each copy connected with plane surfaces.

The Block Spiral command is similar to the Block Sweep command, except the Block Spiral allows the Block to be offset along the axis of revolution as it is extruded.

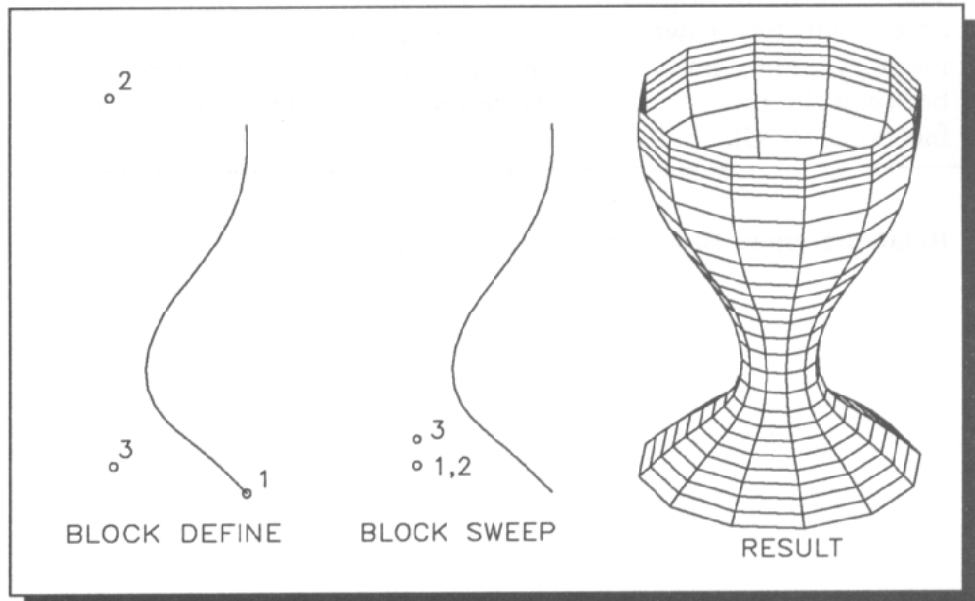


Figure 20 - The Block Sweep command extrudes a Block in a circle. The first point is the location for the first copy. The second and third points define the axis of revolution.

Example: Draw a washer (the type that goes with bolts and nuts). First, use the Box command to draw a short 2-dimensional box, about 3 times wider than tall. To do this, set two points, one in the lower left corner and one in the upper right corner. Make a Block out of the box. Then use the Block Sweep command to make a ring out of the box - set a point on the Block handle, a point to the left of the box, and a point above the second point. Enter **20** copies and **360** degrees, when asked. The square will be rotated (or revolved, in this case) about the second and third points, forming a ring.

Tip: The more copies or facets you specify, the slower the object will be drawn and the more space it will occupy in memory. The Shade command can "smooth out" the borders between the facets. So, it may be more efficient to use fewer facets (20 to 30).

Related Topics: Block Circular Repeat, Block Spiral.

Block Zoom Command

| | |
|---------------|-------------------|
| Menu: | BLOCKS |
| Menu Command: | Block Zoom |
| Short Form: | BZ |

Point 1: Zoom Center (optional).

The Block Zoom command zooms a Block, making it larger or smaller.

If you set a point with this command, the Block will be zoomed "into" that point. In other words, that point on the Block will remain in the same place after the zoom operation. If no points are used, the Block will be zoomed into the primary Block handle.

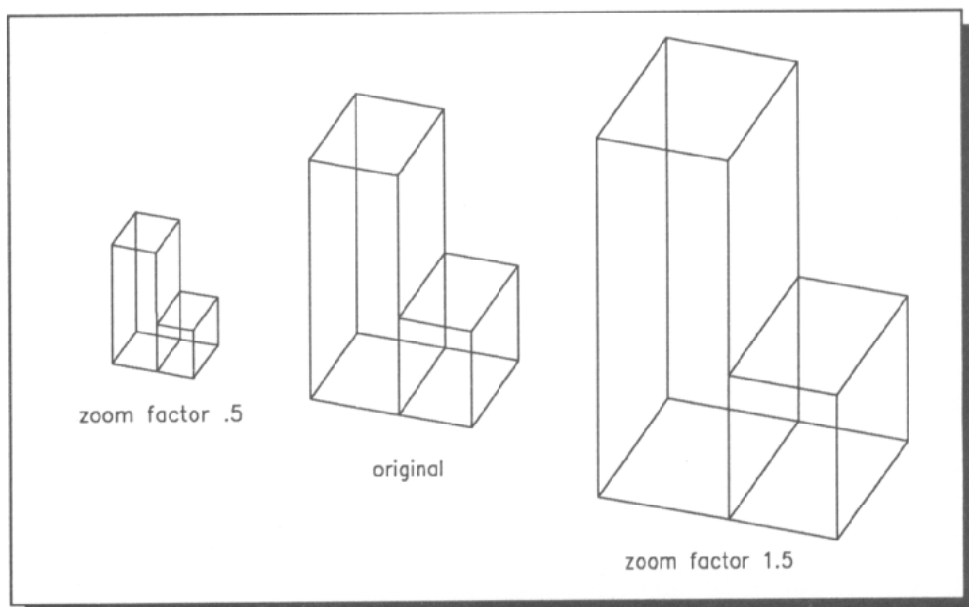


Figure 21 - The Block Zoom command makes a Block larger or smaller.

The Block zoom factor is similar to the zoom factor of the Zoom command. For example, enter 3 for 3 times larger, .5 for half as large, etc.

This command does not make a new copy of the Block. It merely scales the original Block.

Example: Make an object in the drawing two times larger. Define the object to be a Block. Place the Block handle at the point on the object that should remain in the same place after the zoom. Then execute the Block Zoom command (press **Enter** when asked for points) and enter a Zoom factor of 2.

Related Topics: Block Scale.

Box Command

| | |
|--------------------|---------------|
| Menu: | SOLIDS |
| Menu Command: | Box |
| Short Form: |]] |
| Keystroke Command: |]] |

Point 1: One corner of the box.

Point 2: Opposite corner of the box.

This command draws a box defined by two points in opposite corners of the box. To use the command, set two points: the first point in one corner of the box and the second point in the opposite corner. A box will be drawn with its size and location determined by the two points set.

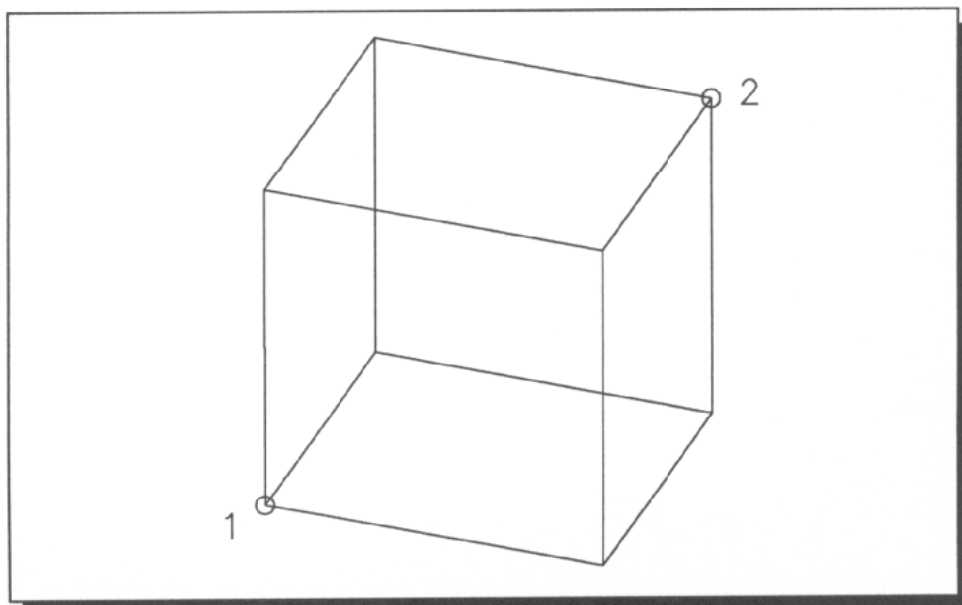


Figure 22 - Set two points in opposite corners of the Box.

If the two points have different X, Y, and Z values or positions, then the box will be a 3-Dimensional box made of grid (surface) entities. If the two points lie on the same X-Y, X-Z, or Z-Y plane, then a rectangular plane will be drawn.

Example: Draw a 6x3x2 box. Press the **J** key and set a point at the location for the box's front lower left corner. Then press the apostrophe key (' , the Point Relative command) and enter **6, 3, 2** when asked for the relative X, Y, Z values. A box will be drawn between the two points.

Related Topics: Plane.

Center of Gravity Command

| | |
|---------------|-----------------------|
| Menu: | SOLIDS |
| Menu Command: | Center Gravity |
| Short Form: | CG |

Point 1: Set a point on the Solid.

This command is used to find the center of gravity of a Solid. To use the command, select the command from the menu and set a point on the Solid. The cursor will move to the center of gravity (center of mass) of the Solid and a point will be set there.

Tip: The X, Y, Z coordinates of the center of gravity can be displayed by selecting status line format 3. This displays the location of the cursor, so when the cursor is moved to the center of gravity that location will be displayed.

Related Topics: Midpoint.

Chamfer Command

| | |
|--------------------|----------------|
| Menu: | EDIT |
| Menu Command: | Chamfer |
| Short Form: | CHF |
| Keystroke Command: | ^F |

Point 1: First line making up the corner to be chamfered.

Point 2: Second line making up the corner to be chamfered.

The Chamfer command "cuts off" a corner of two lines or a plane.

To use the command, select **Chamfer** from the menu and set a point on each of the two lines making up the corner to be chamfered.

You will then be asked for the depth of the chamfer. Enter the depth or "radius" of the chamfer. The corner between the lines will be "cut off" with the chamfer and the two lines will be trimmed to meet the chamfer.

The two lines selected do not have to actually meet, and the lines can cross one another. The lines will be trimmed to the endpoints of the chamfer. The two lines do have to lie in the same plane, and must not be parallel. In other words, it must be possible to draw an arc tangent to both lines in three dimensions.

The Fillet command is similar to the Chamfer command, except it "rounds off" the corner with an Arc instead of a straight line.

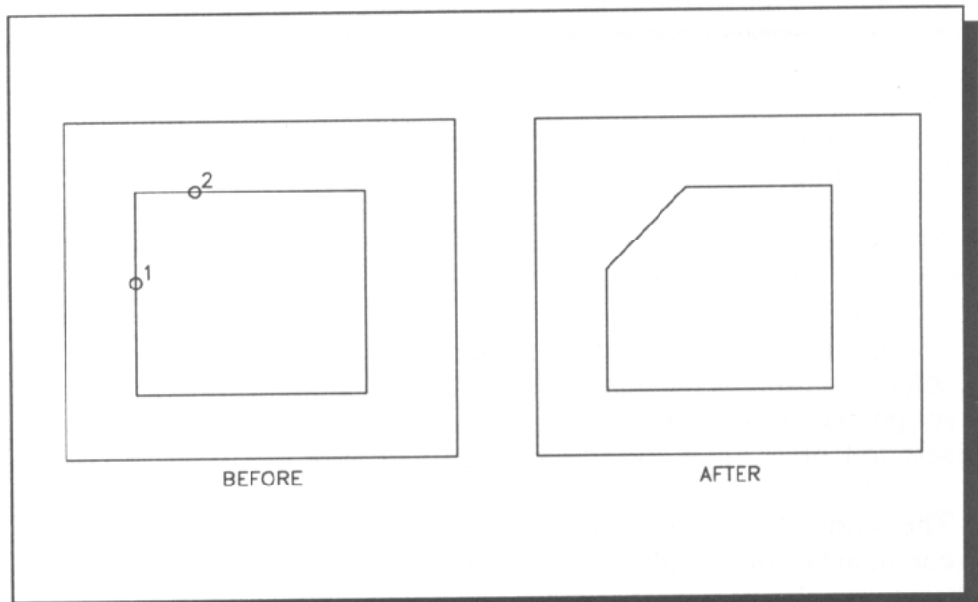


Figure 23 - The Chamfer command is used to "cut off" a corner. Set a point on each of the two lines that form the corner.

Example: Chamfer the upper right corner of a 2-dimensional rectangle. Draw a 2-dimensional rectangle with the box command. Use the Chamfer command and set a point on the upper edge and a point on the right edge of the rectangle. Enter a chamfer depth when asked. The corner will then be "cut off" at a 45° angle.

Tip: Use the Slice, Block Slice, or Solid Subtract command to chamfer a 3-dimensional object.

Related Topics: Fillet, Trim.

Change Color Command

| | |
|---------------|---------------------|
| Menu: | EDIT |
| Menu Command: | Change Color |
| Short Form: | CHC |

Points 1-n: Points on the entities to be changed.

The Change Color command changes selected entities to a specified color. To use this command, select **Change Color** or press **^H**, then set points on the entities to be changed. After all entities have been selected, press **Enter** and select the desired color.

The color of an object can also be changed using the Block Color command or the Change Entity command.

Example: Change the color of a line from green to red. Use the Change Color command, set a point on the entity, and then select red.

Related Topics: Change Entity, Block Color.

Change Entity Command

| | |
|---------------|---------------|
| Menu: | EDIT |
| Menu Command: | Change |
| Short Form: | CH |

Point 1: Set a point on the entity to be changed.

The Change Entity command can be used to change the color or layer of a line or entity.

When you use this command, set a point on the line to be modified. The color and layer of the line will be displayed on the screen:

Either of these can be modified by moving the cursor to the field and entering a new value. After the new values have been entered, press **F1**. The line will be redrawn with the new values. You can press **Esc** to return and leave the line in its original form.

Example: Change the color of a line from green to red. Use the Change Entity command, set a point on the line, and then change the color to red (color 2) when the color and layer of the line are displayed on the screen.

Related Topics: Block Color, Block Layer, Change Color.

Circle Command

| | |
|--------------------|-------------------------------|
| Menu: | ARC/CIRC |
| Menu Command: | Circle |
| Short Form: | Cir |
| Keystroke Command: | O (Letter O, not zero) |

Point 1: Center of the circle.

Point 2: Any point on the circle.

Point 3: Plane definition (optional).

The Circle command is one of four commands used to draw a circle. This command draws a circle based on a point in the center and a point on the outside of the circle.

With this command, a circle is drawn through the second point with its center at the first point. If the third point is used, it defines the plane on which the circle lies.

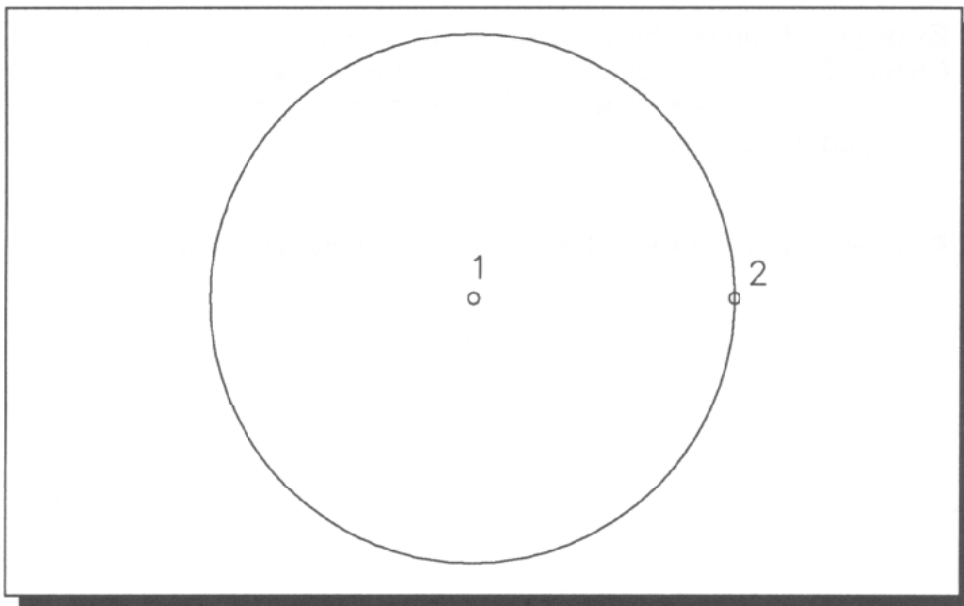


Figure 24 - Set a point on the center and a point on the outside of the circle. A third point can be used to specify the plane for the circle.

Example: Draw a circle on the X-Z plane. Set a point for the center of the circle. Set a second point to the right of the first. The distance between these two points will be the radius of the circle. Set a third point behind or in the positive Z direction from the second. This causes the circle to lie "flat" on the X-Z plane instead of "standing up" on the X-Y plane.

Tip: The Disc command can be used to draw a circle as a plane instead of a line.

Related Topics: Circle-2, Circle-3, Circle Tangent, Disc, Vector Circle.

Circle-2 Command

| | |
|---------------|-----------------|
| Menu: | ARC\CIRC |
| Menu Command: | Circle-2 |
| Short Form: | C2 |

Point 1: Endpoint of a diameter of the circle.

Point 2: Other endpoint of the diameter.

Point 3: Plane definition (optional).

The Circle-2 command is one of four commands used to draw a circle. This command draws a circle based on two points set on the diameter of a circle.

When you use this command, a circle is drawn between the two points set. The third point, if used, defines the plane on which the circle lies.

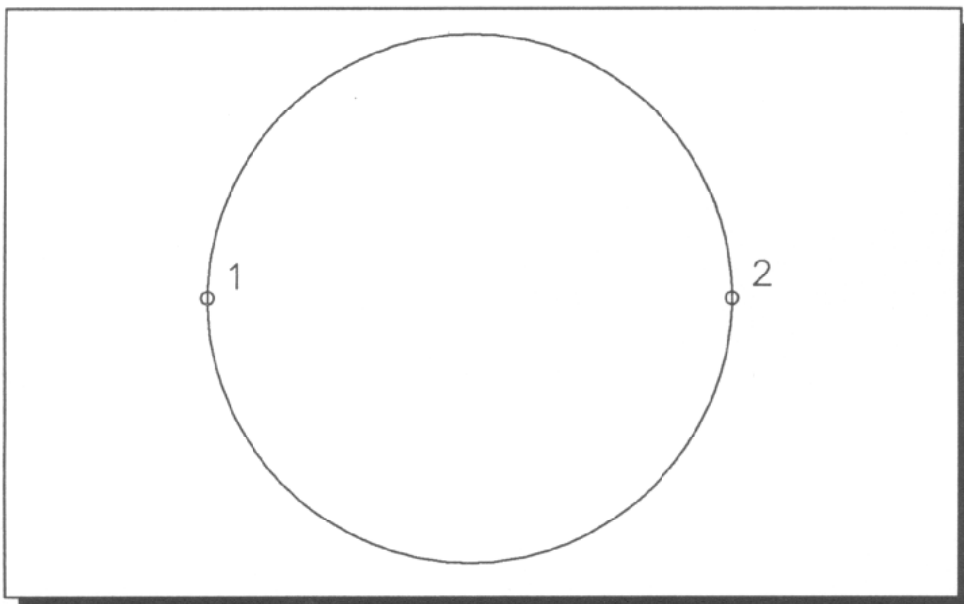


Figure 25 - Set two points on the diameter of the circle. A third point can be used to specify the plane on which the circle lies.

Example: Draw a circle between two points. Use the Circle-2 command and set the two points between which the circle is to be drawn.

Related Topics: Circle, Circle-3, Circle Tangent.

Circle-3 Command

| | |
|---------------|-----------------|
| Menu: | ARC/CIRC |
| Menu Command: | Circle-3 |
| Short Form: | C3 |

Point 1: Any point on the circle.

Point 2: Any other point on the circle.

Point 3: Any other point on the circle.

The Circle-3 command is one of four commands used to draw a circle. This command draws a circle through three points.

To use the command, select **Circle-3** from the ARC/CIRC menu, then set the three points through which the circle is to be drawn. There is only one circle that can be drawn through any three points.

If the three points lie on a straight line, then a circle cannot be drawn through the points.

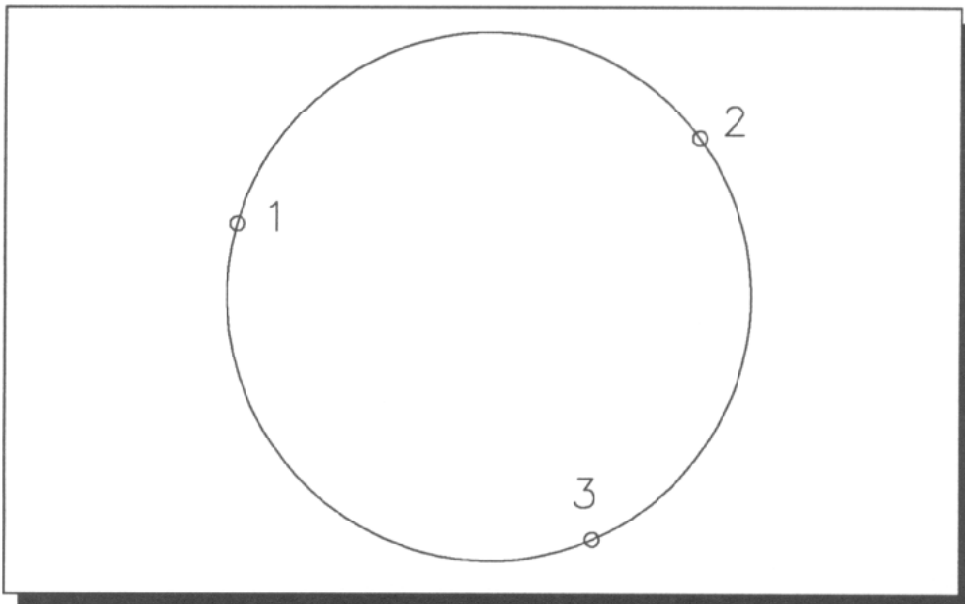


Figure 26 - Set any three points on the circle.

Example: Draw a circle touching three points in the drawing. Use the Circle-3 command and set points on the three points in the drawing. A circle will be drawn through the three points.

Tip: It is easy to copy a circle from a drawing on a digitizer using the Circle-3 command because you don't have to guess where the center of the circle is.

Related Topics: Circle, Circle-2, Circle Tangent, Arc-3.

Circle Tangent Command

| | |
|---------------|----------------------|
| Menu: | ARC/CIRC |
| Menu Command: | Circle Tangnt |
| Short Form: | CT |

Point 1: Set a point on one of the tangent lines.

Point 2: Set a point on the second tangent line.

The Circle Tangent command is one of four commands used to draw a circle. This command draws a circle of a specific radius tangent to two lines.

To use the command, select **Circle Tangent** from the menu and set two points on the lines to which the circle is to be tangent. You will then be asked for the Radius of the circle.

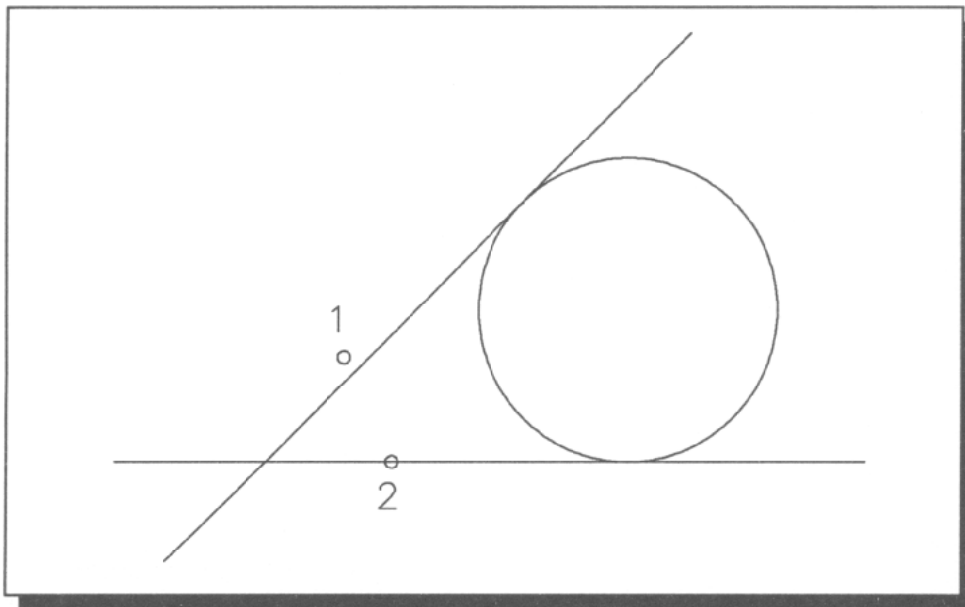


Figure 27 - Set a point on each tangent line.

A circle will be drawn with that radius, tangent to the two lines. If the radius is too large or too small, the circle may not actually touch the two lines, but it will be tangent to the lines if the lines are extended to the circle. The two lines selected with the points must not be parallel.

Example: Draw a circle tangent to two lines in the drawing. Use the Circle Tangent command and set a point on each of the two lines. A circle tangent to the two lines will then be drawn.

Tip: The Fillet command can be used to draw an arc tangent to two lines.

Related Topics: Circle, Circle-2, Circle-3, Fillet.

Clear Screen Command

| | |
|--------------------|---------------------|
| Menu: | DISPLAY |
| Menu Command: | Clear Screen |
| Short Form: | CLS |
| Keystroke Command: | Y |

Points: None.

The Clear Screen command erases everything on the screen. To use the command, press the **Y** key or use the **Clear Screen** menu command. You will be asked if you really want to erase everything. If so, press **Y**. The screen will then be cleared.

Before clearing the screen, be sure to save your drawing if you want to keep it. Everything on the screen is lost when it is cleared.

When the screen is cleared, the relative zoom and position of the drawing is reset.

Color Command

| | |
|--------------------|---------------|
| Menu: | COLORS |
| Menu Command: | Color |
| Short Form: | H |
| Keystroke Command: | H |

Points: None.

The Color command is used to specify the color to be used in subsequent lines or entities in the drawing. The frame of the drawing area is displayed in the current color.

With most displays, DesignCAD 3-D supports 15 basic colors. More colors are often used in shading, but when you make the drawing you can generally select from only 15 colors.

To use the Color command, press **H** or select **Color** from the **COLORS** menu. You will be asked for the color number (1-15). You can press **?** to select the color from the "color bar."

Your computer may not be capable of displaying the entire range of 15 colors, but it is still possible to use the entire range of colors in your drawing. This makes it possible to output your drawing to a plotter that has more colors, or to transfer the drawing to a computer with more colors.

This command only changes the color of lines drawn after the command is executed. The Change Color command can be used to change the color of an existing line.

You can determine the color of a line in the drawing with the ID command.

If your display is capable of 256 colors or more, you can use the Color Edit command or the Materials command to change the actual color associated with color numbers 1-15.

Example: Change the drawing color to red. Press the **H** key, and then press **?** to display the available colors. Select red. The drawing frame will change to red and subsequent lines will be drawn in red.

Related Topics: Color Edit, Materials.

Color Edit Command

| | |
|---------------|-------------------|
| Menu: | COLORS |
| Menu Command: | Color Edit |
| Short Form: | Cedit |

Points: None.

Note: The Color Edit command works only with displays that have 256 colors or more.

The Color Edit command is used to set the colors for DesignCAD 3-D. Both the wireframe colors (the colors of lines in the drawing) and the shading colors can be set.

Colors in DesignCAD 3-D

There are 15 basic colors in Design 3-D. Each of these colors has a set of shading colors associated with it. The shading colors are those colors displayed with the Shade commands.

When you draw something in DesignCAD 3-D, you draw it in the current drawing color, one of the 15 basic colors. When you shade something in DesignCAD 3-D, the shading colors used are based on the basic color of the objects in the drawing.

Color Selection Box

In the Color Edit command, the basic colors can be selected from the large group of colors on the upper left of the screen using the mouse or the keyboard. To select a color, move the cursor to a color and press the left mouse button twice (or the **Enter** key). If you do not have a mouse, use the **Tab** key to move from one box on the screen to the next.

Red, Green, Blue, Intensity

The Red, Green, Blue, and Intensity values can be modified by selecting those boxes on the upper right of the screen with the mouse or **Tab** key. Press **+** or **-** (or click on them with the mouse), or enter the number to change one of the values.

The Red, Green, and Blue values change the hue of the color by affecting the amount of Red, Green, and Blue in the color. The Intensity value changes the overall brightness of the color. These values can range from 0 to 100.

Contrast

The Contrast box, below the Intensity box, is the difference between the brightest shading color and the darkest. It determines the amount of contrast when an object of this color is shaded. At the lower right of the large color selection area are the shading colors. The bright colors are on top, gradually changing to dark colors at the bottom. This box changes as the contrast value changes.

Wireframe Color

The Wireframe box contains the color that will be used on the screen for wireframe drawing (lines that are not shaded). This can be any color in the box of shading colors. To make the wireframe color brighter, press **+** (or click on the **+** with the mouse). To make the wireframe color darker, use **-**.

Color Number

Below the Wireframe box is the Color Number box. This is the basic color number, 1 to 15, for the color you are setting up. You can change color values for any or all of the basic colors by changing this value. When you change from color 1 to color 4, for example, the changes you made to color 1 will disappear from the screen, but they are not lost. When you change back to color 1 those color values will return.

The Color Globe

The color globe shows the current wireframe color. You can shade the globe to see how shaded objects will look by pressing the **Space** bar.

Current Color

You can press the **C** key to load the values for the current color. These will replace any changes you have made for that basic color number.

When you are finished

After you have finished modifying the color values, press F2 to exit and use the new colors. To save the new colors as DesignCAD 3-D defaults, press F3. To ignore the changes you have made and return to the original color configuration, press **Esc**.

Loading and Saving Colors

You can save your color configuration to disk by pressing F4 in the Color Edit command. This saves the color information for all 15 basic colors. You can load a color configuration from disk using the F5 key.

Using the Mouse

With the Color Edit command, you can use the mouse to select any of the boxes, including the Function key boxes. You can change a value by clicking on the **+** or **-** to the right of the value.

Help!

At any time in the Color Edit command, you can press F1 for help.

Tip: If you have trouble with the Color command, you can use the Color Reset command to restore the original DesignCAD 3-D colors.

Tip: With the Material command, you can edit the colors of materials as well as additional shading properties, such as reflection, texture, and background light.

Related Topics: Color Read, Color Save, Material, Color Reset.

Color Read Command

| | |
|---------------|-------------------|
| Menu: | COLORS |
| Menu Command: | Color Read |
| Short Form: | Cread |

Points: None.

The Color Read command is used to read a set of colors from disk. Those colors will then be used in DesignCAD.

A color data file can be created using the Color Edit command to modify the colors, and the saving the color information with the Color Save command. The extension for DesignCAD 3-D color data files is **.PAL**.

This command does not change the color number of the entities in the drawing. It changes the colors associated with the color numbers. A line in color 4 will remain color 4, but after the Color Read command color 4 may change, for example, from blue to red.

With the Color Read command, you can press **Enter** when asked for the color data file name to select the file from a directory listing.

A color data file contains color information for all 15 basic colors. All colors will be affected when you read a color data file, including the shading colors. The Material command can be used to change the color information for a single color, as well as the shading parameters.

Example: Load the colors from the file called **MYCOLORS**. Use the Color Read command, and enter **mycolors** when asked for the color data file name. The drawing colors will change to those from the file **MYCOLOR.PAL**.

Tip: You can use the Color Reset command to restore the original DesignCAD 3-D colors.

Related Topics: Color Edit, Color Save, Material, Color Reset.

Color Reset Command

| | |
|---------------|--------------------|
| Menu: | COLORS |
| Menu Command: | Color Reset |
| Short Form: | Creset |

Points: None.

The Color Reset command restores the colors to the default DesignCAD 3-D colors. If a color configuration has been saved as defaults for DesignCAD in the Color Edit command, that configuration will be used for the new colors.

Example: After changing the colors in the Color Edit command, restore the colors to the original setting. Use the Color Restore command. The colors will be reset to the default values.

Tip: To remove the "custom default" colors set in the Color Edit command, erase or rename the file AUTOEXEC.PAL.

Related Topics: Color Save, Color Read, Color Edit, Material.

Color Save Command

| | |
|---------------|-------------------|
| Menu: | COLORS |
| Menu Command: | Color Save |
| Short Form: | Csave |
| Points: None. | |

The Color Save command is used to save the current color information to disk. Those colors can then be used with the Color Read command.

The color information can be established using the Color Edit command. A color data file contains color information for all 15 basic colors.

The extension for DesignCAD 3-D color data files is **.PAL**.

All colors will be saved when you save a color data file, including the shading colors. The Material command can be used to save the color information for a single color as well as the shading parameters.

Example: Save the color information from a drawing called PART7. First, load the drawing onto the screen. Then use the Color Save command to save the color information. You can then read that color file any time with the Color Read command to load the same color configuration that is used on the drawing PART7.

Related Topics: Color Edit, Color Read, Color Reset, Material.

Combine Line Command

| | |
|---------------|----------------|
| Menu: | EDIT |
| Menu Command: | Combine |
| Short Form: | CM |

Points 1-n: Set a point on each line to be combined.

This command is used to combine two or more lines into a single line. To use the command, set a point on each line to be combined. Each line that is combined must be connected to the next line - that is, each line selected must share an endpoint with another selected line.

This command can be used to combine lines that will be extruded or swept, making the product a single surface or grid instead of having one grid for each line.

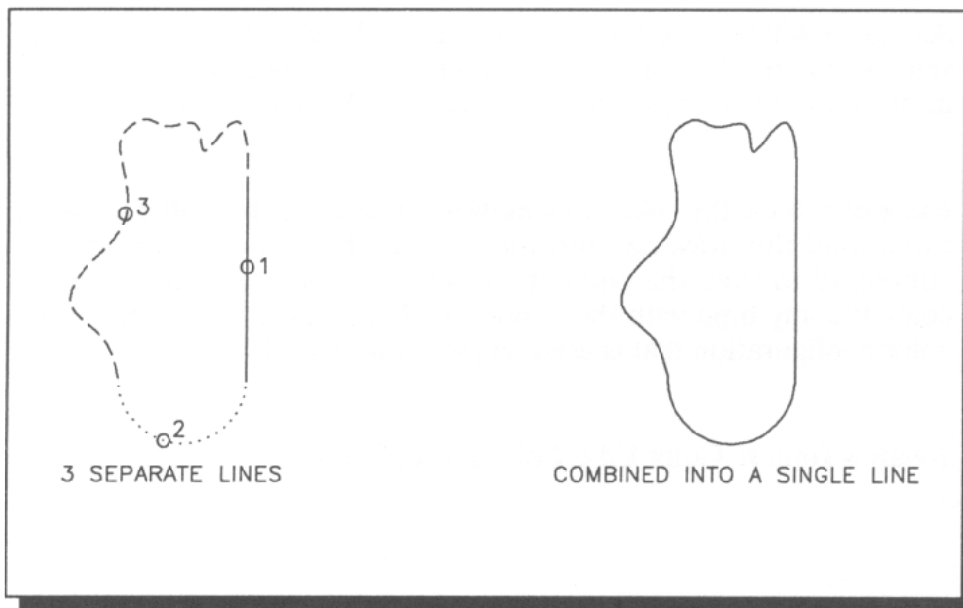


Figure 28 - The Combine Line command combines two or more lines into a single line entity. Lines, curves, and arcs can be combined.

The lines used with this command can be Line, Curve, or Arc entities. The maximum total number of points in the resulting line is 200.

Tip: The Make Plane command can be used to combine lines into a surface instead of a single line.

Related Topics: Make Plane.

Cone Command

Menu: **SOLIDS**
Menu Command: **Cone**

Point 1: Center of cone base.
Point 2: Outside of cone base.
Point 3: Distance for cone point.

This command is used to draw a cone. Three points are used with the command. The first two points are the center and radius of the circle on one end of the cone.

The third point defines the direction in which the cone lies and the length of the cone. The circle is extruded toward the third point until the new end of the cone is even with the third point.

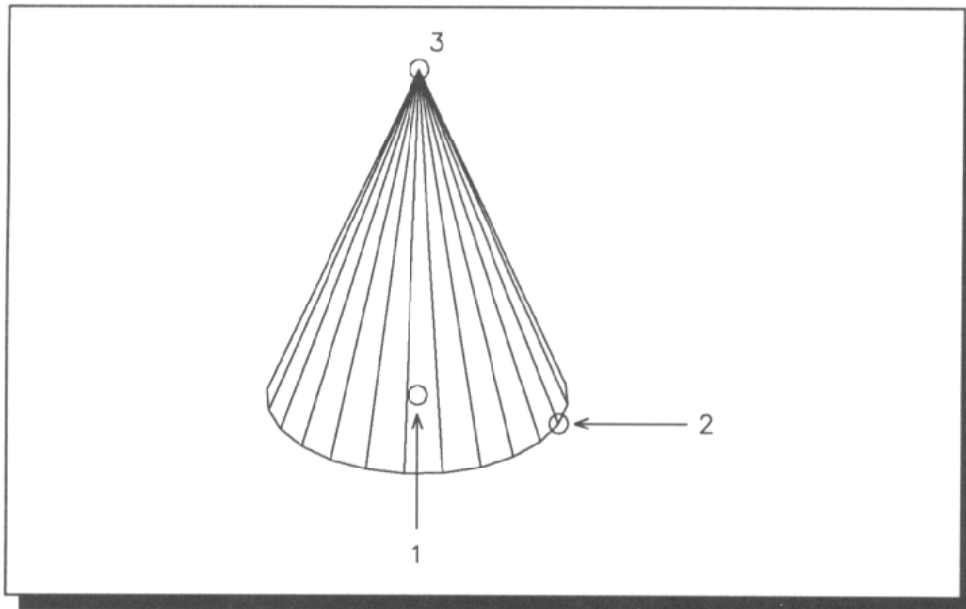


Figure 29 - The first points is set at the center of the base. The second point is the edge of the base. The third point determines the direction and distance of the pointed end of the cone.

Configuration Save Command

| | |
|---------------|--------------------|
| Menu: | PARAMETER |
| Menu Command: | Config Save |
| Short Form: | CSA |

Points: None.

This command is used to save the system configuration to disk. When a configuration is saved, it becomes the default configuration for DesignCAD. That configuration will be loaded whenever DesignCAD is run.

The configuration information includes the Parameters and View information. The following items are saved by the Configuration Save command:

- Attribute Display (On or Off)
- Character Font
- Command Menu
- Cursor Step Sizes
- Default Text Size and Angle
- Digitizer menu location
- Digitizer or mouse mode
- Digitizer plane
- Dimension Format
- Dimension Precision
- Drawing Units per Inch on Output
- Drawing Unit Size
- Dynamic Dimensioning (On or Off)
- Light settings
- Mouse Sensitivity
- Point Type
- Save System Parameters to Drawing (Yes or No)

Save as a Binary File (Yes or No)
Screen colors
Section Commands Limited to Current Layer (Yes or No)
Snap Grid
Sound
Status Line Format
2-D Snap (Yes or No)
View Configuration
Wall thickness for the Wall command

Tip: To remove the default parameters and return to the "factory" DesignCAD 3-D settings, erase the file DCAD4.SYS.

Related Topics: System Parameters, Drawing Parameters.

Connect Command

| | |
|---------------|-----------------|
| Menu: | SURFACES |
| Menu Command: | Connect |
| Short Form: | CN |

Point 1-n: Lines to be connected with a surface.

This command is used to "stretch" a surface between two or more lines. To use the command, select **Connect** from the **SURFACES** menu and set a point on each line to be connected. The lines can be Line, Curve, Circle, or Arc entities.

You will then be asked for the number of planes and the number of intermediate breaks. The number of planes is the number of individual planes to be constructed **ALONG** the original lines. The number of intermediate breaks is the number of breaks in the surface **BETWEEN** the first and last line. The larger you make these values, the smoother the surface will be and the larger the drawing will be.

If this command is used to connect a series of closed polygons, circles, or curves, the points set determine where the grid lines will be located. For a simple, non-twisted surface, each point should be set at the same relative position in the polygons. For example, you could set all points in the lower right corner of the polygons.

The **Connect** command makes a ruled surface between each pair of lines. This surface is a straight or linear fit between the original lines. The **Connect Curve** or **Connect Smooth** commands can be used to produce more complex surfaces between the lines.

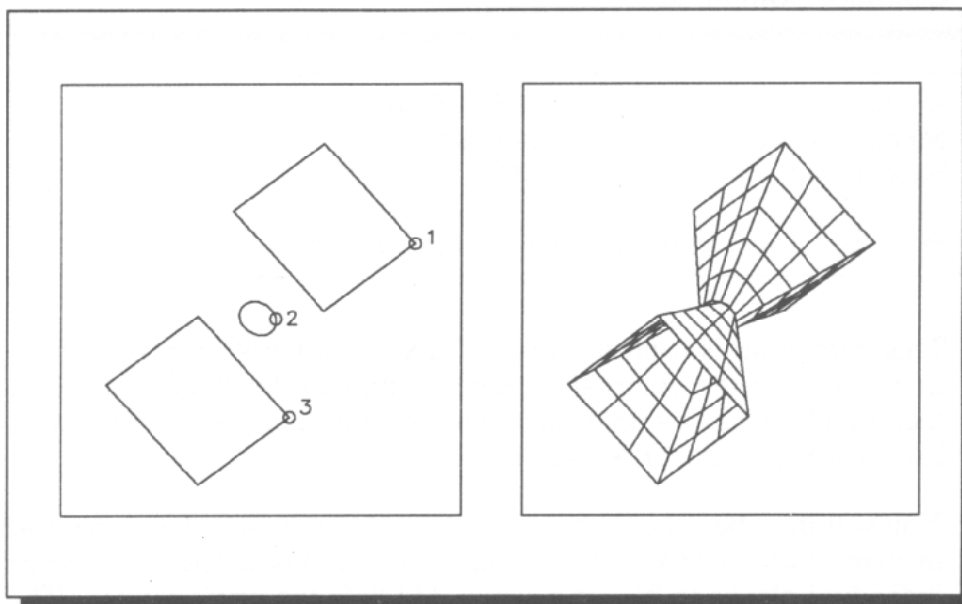


Figure 30 - Set a point on each line to be connected. 12 surfaces and 4 intermediate breaks were used in this example.

Example: Use the **Connect** command to make an object with a square bottom and a circular top. Use the **Box** command to draw a box on the X-Z plane. Use the **Disc** command to draw a circular plane directly above the box on the X-Z plane. Use the **Connect** command to connect the two planes with a surface. Set a point on the box and the disc, and enter **36** when asked for the number of planes. A surface with 36 facets will be drawn to connect the two shapes.

Related Topics: **Connect Curve**, **Connect Smooth**.

Connect Curve Command

| | |
|---------------|----------------------|
| Menu: | SURFACES |
| Menu Command: | Connect Curve |
| Short Form: | CC |

| | |
|----------------------|--|
| Point 1: | First line to be connected with a surface. |
| Intermediate Points: | Set on lines to be connected, or set to define the surface path. |
| Point n: | Last line to be connected with a surface. |

This command is used to "stretch" a surface between two or more lines along a curve. To use the command, select Connect Curve from the SURFACES menu and set a point on each line to be connected. The lines can be Line, Curve, Circle, or Arc entities. You may also set intermediate points between the original lines to force the surface to follow a specific curve.

You will then be asked for the number of planes and the number of intermediate breaks. The number of surfaces is the number of individual planes to be used ALONG the original lines. The number of intermediate breaks is the number of breaks in the surface BETWEEN the original lines. The larger you make these values, the smoother the surface will be and the larger the drawing will be.

The surface drawn is a simple ruled surface "bent" to follow the curve defined by the points set.

If this command is used to connect a series of closed polygons, curves, and circles, the points set determine where the grid lines will be located. For a simple, non-twisted surface, each point should be set at the same relative position in the polygons.

The Connect Curve command is similar to the Connect command, except the surfaces created by Connect Command do not follow a curve. The Connect Smooth command can be used to produce a more complex surface that fits smoothly over a set of lines.

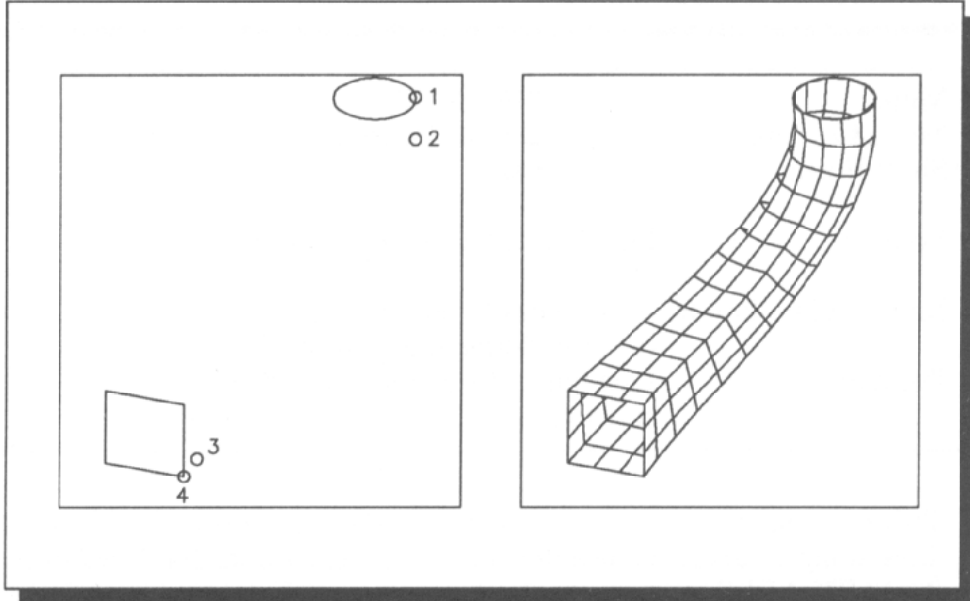


Figure 31 - Set points on the lines to be connected, with intermediate points to define the curvature. In this example, points 1 and 4 were set on the lines. Points 2 and 3 were used to specify the curvature.

Example: Connect two curves with the Connect Curve command. Draw a curve on the X-Y plane. Move back in the positive Z direction and draw a another curve on the X-Y plane directly behind the first. Use the Connect Curve command to connect these lines with a surface. Set a point on the right endpoint of the back curve. Set a second point between the curves, higher than both curves. Set a third point at the right endpoint of the first (front) curve. Enter **20** planes and **12** intermediate breaks when asked. A surface will be drawn that connects the two curves and passes through the middle point.

Related Topics: Connect, Connect Smooth.

Connect Smooth Command

| | |
|---------------|----------------------|
| Menu: | SURFACES |
| Menu Command: | Connct Smooth |
| Short Form: | CS |

Point 1-n: Lines and changes in surface direction.

This command is used to fit a surface between two or more lines. To use the command, select **Connect Smooth** from the **SURFACES** menu. Set a point on each line to be connected, and optionally one or more points between the lines to be connected. The lines can be Line, Curve, Circle, or Arc entities.

You will be asked for the number of planes and the number of intermediate breaks. The number of planes is the number of individual planes to be used **ALONG** the original lines. The number of intermediate breaks is the number of breaks in the surface **BETWEEN** the original lines. The larger you make these values, the smoother the surface will be and the larger the drawing will be. A smooth surface will be fit over the lines.

This command is different from the **Connect** and **Connect Curve** command. It actually does a 3-Dimensional curve fit to get the smoothest possible surface over the lines set.

Example: Connect 3 rectangles with a smooth surface. Draw three 2-dimensional boxes on the X-Z plane: one on bottom, one in the middle, and one on top. Make the middle box smaller than the top and bottom boxes. Use the **Connect Smooth** command to connect the boxes with a smooth surface. Set a point on the front right corner of each box, and enter **4** planes and **10** intermediate breaks when asked. A smooth surface connecting the boxes will be drawn.

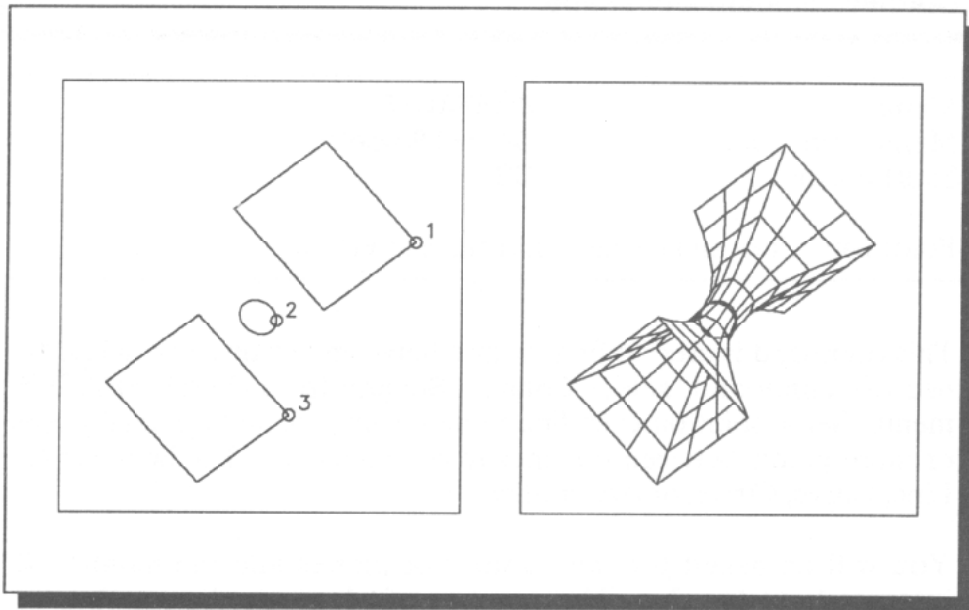


Figure 32 - Set a point on each line to be connected. In this example, 12 planes and 4 intermediate breaks were used.

Tip: This command is very useful in making a solid representation of an object from a series of cross sections.

Related Topics: Connect, Connect Curve.

Cursor Step Size Command

| | |
|---------------|--------------------|
| Menu: | CURSOR |
| Menu Command: | Cursor Step |
| Short Form: | Cstep |

Points: None.

This command is used to specify the distance that the cursor moves when an Arrow key, **F1**, or **F2** is pressed. There are two cursor step sizes: a large one and a small one.

The large cursor step size is used when an arrow key, **F1**, or **F2** is pressed. The small cursor step size is used when an arrow key, **F1**, or **F2** is pressed simultaneously with the **Shift** key. If your keyboard has a numeric keypad, the number keys will move the cursor the large cursor step size when the **Num Lock** is off or the small cursor step size when the **Num Lock** is on.

When you are asked for the new cursor step size, the present value is displayed in brackets. Press **Enter** to keep the same cursor step size, or enter the new value. You will then be asked for the small cursor step size.

These values can also be entered under the Drawing Parameters. To save the cursor step size as a defaults, use the Configuration Save command.

When the drawing is zoomed, the cursor step size normally remains constant with respect to the screen. This means that when you zoom to make the drawing 3 times larger, the cursor step size will be 3 times smaller on the drawing.

Under the System Parameters, you can change the cursor step size to be constant with respect to the drawing instead of the screen. This causes the cursor movement to be greater on the screen when you increase the drawing size.

The cursor step size is automatically saved with the drawing.

Example: Set the large cursor step size to 1 foot and the small cursor step size to 1 inch (1/12 of a foot). Assume that the drawing units are one foot. Select the Cursor Step Size command from the CURSOR menu. Enter 1 when asked for the large cursor step size. Enter [1/12] when asked for the small cursor step size.

Related Topics: Cursor Type, Drawing Parameters.

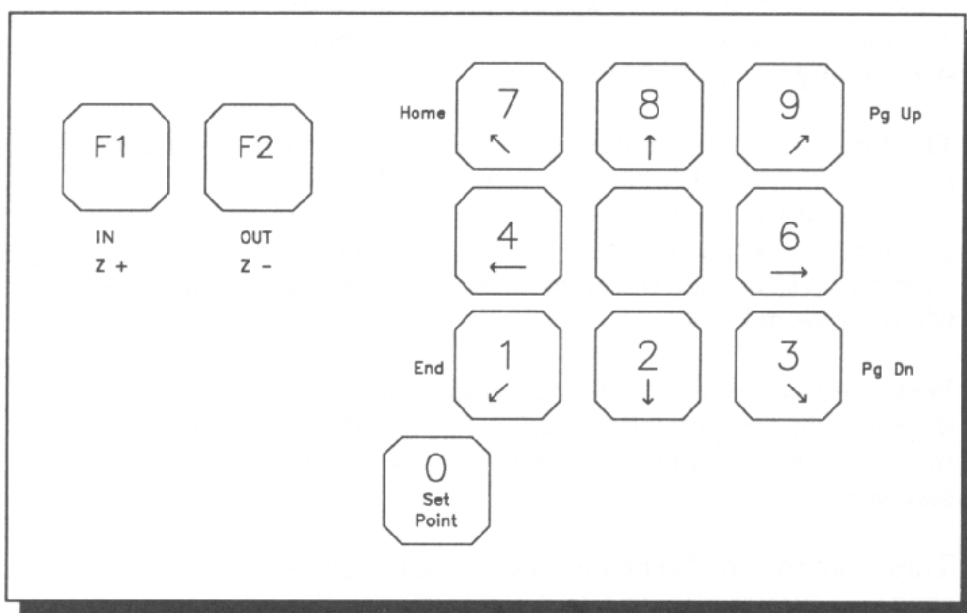


Figure 33 - The Cursor Control keys.

Cursor Type Command

| | |
|--------------------|--------------------|
| Menu: | CURSOR |
| Menu Command: | Cursor Type |
| Short Form: | + |
| Keystroke Command: | + |

Points: None.

The Cursor command allows an alternate cursor type to be used. There are three cursor types: a 3-D cursor, a small "x", and a small "+". To change cursors, select **Cursor Type** from the **CURSOR** menu or press the **+** key.

The cursor type can also be selected in the Drawing Parameter command. To save the cursor type as a program default, use the Configuration Save command.

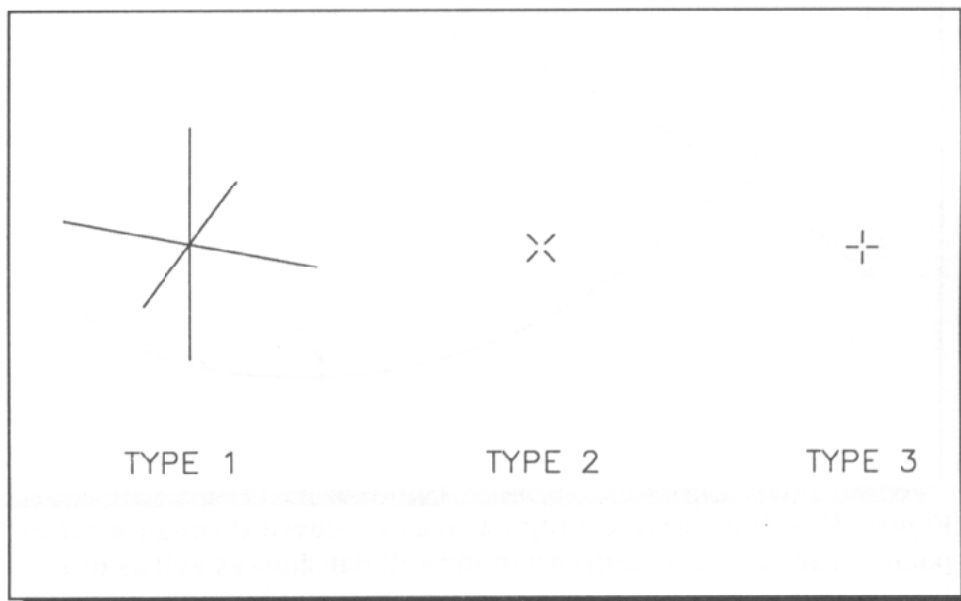


Figure 34 - DesignCAD Cursor Types.

Curve Command

| | |
|--------------------|--------------|
| Menu: | LINES |
| Menu Command: | Curve |
| Short Form: | C |
| Keystroke Command: | C |

Points 1-n: Points through which the curve is drawn.

The Curve command draws a curve through three or more points. To use the Curve command, set the points through which a curve is to be drawn. A curve will be drawn through each point, beginning with the first and continuing to the last point. The points for the curve do not have to lie on the same plane.

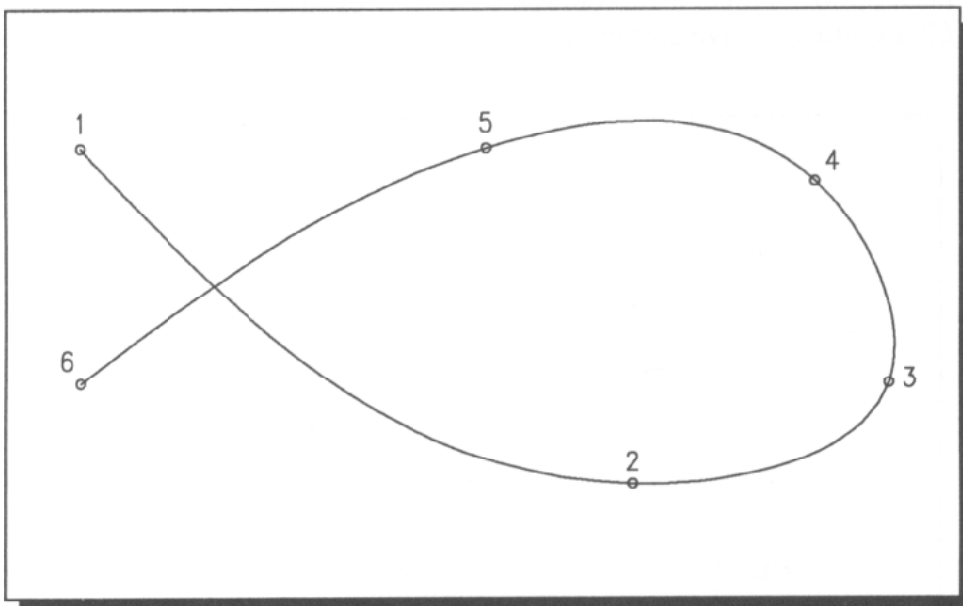


Figure 35 - The Curve command draws a curve through a set of points. The curve can be drawn in three dimensions as well as two.

A "corner" can be made in the curve by setting two points at the same location. This causes, in effect, two separate curves to be drawn on each side of that location. If the last point of the curve is in the same location as the first point, then the curve will be "smoothed" at that location.

The mathematical formula used to determine the curve is a cubic spline.

Example: Use the **Curve** command to draw a curve through four points. Set any four points. They do not have to lie on the same plane. Then press the **C** key. The four points will be connected with a smooth curve.

Related Topics: Vector Curve, Line.

Cut Plane Command

| | |
|---------------|------------------|
| Menu: | SURFACES |
| Menu Command: | Cut Plane |
| Short Form: | CPL |

Point 1: Line along which the plane is to be cut.

Point 2: Plane to be cut.

The Cut Plane command is used to "cut" a plane entity along a line. To cut a plane with this command, draw a line on the plane where the plane is to be cut. This "cutting line" can be a Line, Curve, Circle or Arc.

Two points are used with the Cut Plane command: a point on the line and a point on the plane. The plane will be cut along the line.

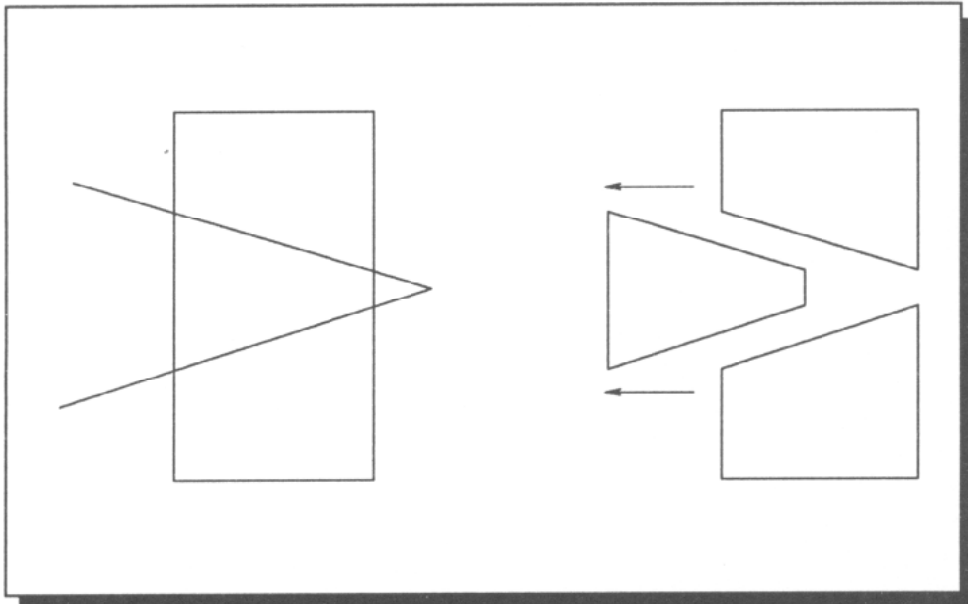


Figure 36 - The Cut Plane command "cuts" a plane along a line. Set one point on the line and one point on the plane to be cut.

The "cutting line" must divide the plane into two or more pieces. For example, it must go across the plane or it must form a closed polygon within the plane. The cutting line must also lie on the plane, not in front of or behind it.

This command works only on simple plane entities - it does not work with a grid entity (a complex surface). A Plane entity is a single plane created by the Plane or Polygon command. A Grid entity is a complex surface created by an extrusion or a sweep. Use the ID command to determine the surface type if you are not sure.

Example: Cut the corner off of a rectangular plane. Draw a 2-dimensional box on the X-Y plane using the Box command. Then draw a line across the corner of the box, extending past both edges. Use the Cut Plane command to divide the 2-dimensional box along the line - set a point on the line and a point on the plane. The plane will be divided along the line. Use the Erase command to erase the "divided" corner - set a point on the corner and press the **E** key.

Related Topics: Block Slice.

Cylinder Command

| | |
|---------------|-----------------|
| Menu: | SOLIDS |
| Menu Command: | Cylinder |
| Short Form: | Cyl |

Point 1: Center of cylinder end.

Point 2: Outside or radius of cylinder end.

Point 3: Distance to the other end of the cylinder.

This command is used to draw a cylinder. Three points are used with the command. The first two points are the center and radius of the circle on one end of the cylinder.

The third point defines the direction in which the cylinder lies and the length of the cylinder. The circle is extruded toward the third point until the new end of the cylinder is even with the third point.

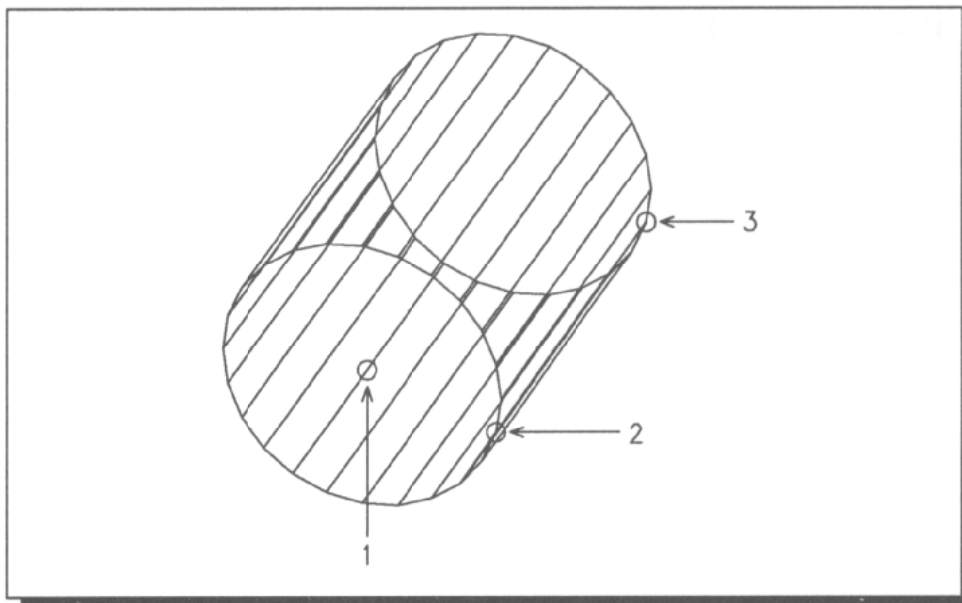


Figure 37 - To draw a cylinder, set one point at the center of one end of the cylinder and a second point at the edge of that end. The third point determines the length and direction of the cylinder.

Example: Draw a bar or cylinder 30mm long and 4mm in diameter. Use the Cylinder command. Set one point where the left end of the cylinder is to be located. Set the second point 2 draw units (mm) above the first point. (Since 4 is the diameter, the radius is 2.) Set the third point 30mm directly to the right of the first. When asked for the number of facets, press **Enter**. A cylinder 30mm long and 4mm in diameter will be drawn.

Related Topics: Cone, Box, Sphere, Hemisphere.

DigiMouse Command

| | |
|---------------|------------------|
| Menu: | CURSOR |
| Menu Command: | DigiMouse |
| Points: | None. |

The DigiMouse command is used to change the operation of the digitizer from digitizer mode to mouse mode and back again.

There are two modes in which the digitizer can operate: "digitizer mode" and "mouse mode." The digitizer normally works in "digitizer mode." This means that the digitizer movement remains consistent with the drawing when the drawing size changes. In mouse mode, the digitizer movement remains consistent with the screen when the drawing size changes.

In digitizer mode, the area of the screen accessed by the digitizer may be varied using the Zoom command (**Z** key). When DesignCAD 3-D is initialized, the area on the screen accessed by the digitizer is the largest area in the shape of the digitizer that can be displayed on the physical screen.

In digitizer mode, you can use the digitizer menu. When the digitizer is in mouse mode, you can use only the command menu on the screen.

To use the command, select DigiMouse from the CURSOR menu. When asked, enter **D** for digitizer mode or **M** for mouse mode.

Tip: "Digitizer mode" is better to use if you are using a paper drawing on your digitizer as a reference. "Mouse mode" can be useful if you need to zoom your drawing to work on fine detail because the cursor movement will not be exaggerated.

Related Topics: "Using a Digitizer with DesignCAD 3-D."

Digitizer Menu Delete Command

| | |
|---------------|---------------------|
| Menu: | FILES |
| Menu Command: | DMenu Delete |
| Points: | None. |

This command disables the digitizer menu. To use the command, select **DMenu Delete** from the **FILES** menu . The Digitizer menu will then no longer function. You can enable the digitizer menu again by loading it from disk with the Digitizer Menu Load command.

This command can be used to disable the digitizer menu so more drawing space will be available on the digitizer.

If the Configuration Save command (**Config Save**) is executed while the digitizer menu is disabled, the digitizer menu will no longer be loaded automatically when DesignCAD is run. To disable the automatic loading of the digitizer menu, use the Digitizer Menu Delete command followed by the Configuration Save command.

This command does not erase the digitizer menu from the disk. It merely disables it in memory.

Related Topics: Digitizer Menu Load, Digitizer Menu Move.

Digitizer Menu Load Command

| | |
|---------------|-------------------|
| Menu: | FILES |
| Menu Command: | Dmenu Load |
| Short Form: | DML |

Point 1: Set position for menu handle.

This command is used to load a digitizer menu from disk. To use the command, select **DMenuLoad**. You will then be asked to set a point for the menu handle. Then enter the menu name. If you are not sure of the file name, press **Enter** to select the name from a directory listing. The digitizer menu will then be loaded and ready to use.

This command loads only the "data" portion of the digitizer menu. The "paper" portion of the menu should already be attached to the digitizer.

You will be asked to set a point on the digitizer menu handle. The digitizer menu handle is designated by a mark on the digitizer menu. It is normally located at the lower left corner of the digitizer menu.

DesignCAD 3-D can be set up to automatically load a digitizer menu when the program is run. To do this, first load the digitizer menu and then execute the Configuration Save command (**Config Save** or **csa**).

Tip: Be sure to place the digitizer menu card level on the digitizer. If you attach it at an angle, some of the commands may not be aligned very well.

Tip: The digitizer menu cannot be used when the digitizer has been placed in "mouse" mode by the DigiMouse command.

Related Topics: Digitizer Menu Delete, Digitizer Menu Move, DigiMouse.

Digitizer Menu Move Command

| | |
|---------------|-------------------|
| Menu: | FILES |
| Menu Command: | Dmenu Move |
| Short Form: | DMM |

Point 1: New location for the digitizer menu handle.

This command is used to move the digitizer menu to a new location on the digitizer.

To move the digitizer menu, first attach the digitizer menu to the digitizer at the desired location.

Next, select **DMenu Move** from the **FILES** menu, and set a point at the digitizer menu handle. Use the digitizer to set the point. The digitizer menu handle is designated by a mark on the digitizer menu. It is normally located at the lower left corner of the digitizer menu. If your digitizer menu does not have the handle marked, set a point at the lower left corner of the digitizer menu.

After the point is set, the digitizer menu can be used at its new location. To make this change permanent, use the Configuration Save command.

Tip: This command moves the "data" portion of the menu only. The paper template should be moved before this command is issued so that the menu can be placed precisely.

Related Topics: Digitizer Menu Load, Digitizer Menu Delete.

Dimension Command

| | |
|--------------------|------------------|
| Menu: | NOTES |
| Menu Command: | Dimension |
| Short Form: | Dim |
| Keystroke Command: | @ |

Point 1: First point to be measured.
Point 2: Second point to be measured.
Point 3: Location for dimension information.

The Dimension command is used to add dimensioning or distance information to a drawing. To use the command, set the first point at the location in the drawing to be measured from, and the second point at the location to be measured to. The third point is the location where the dimensioning information is to be displayed.

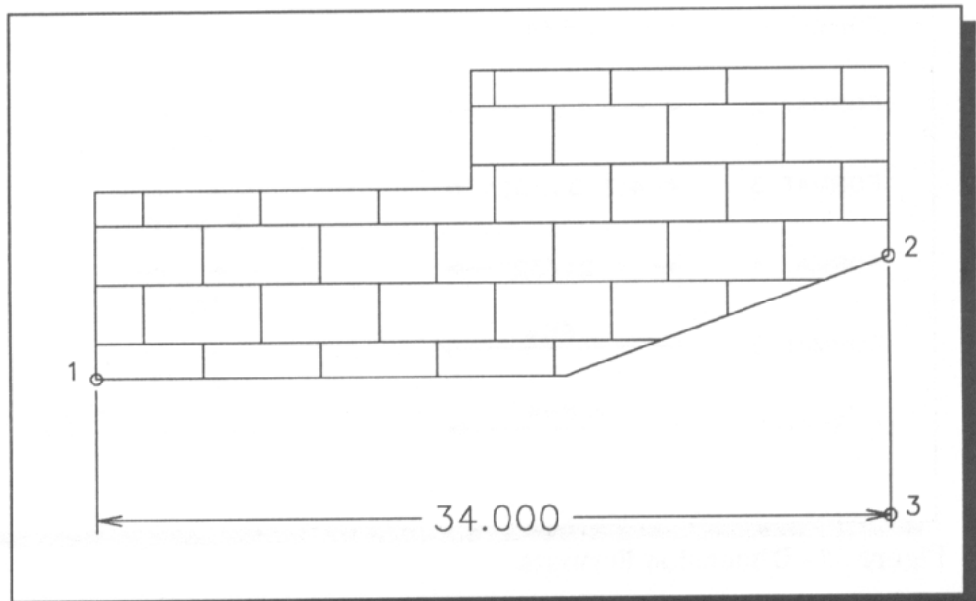


Figure 38 - Set two points for the distance to be measured. Set a third point where the text is to be located.

The Dimension command measures the distance along the X (horizontal), Y (vertical), or Z (depth) axis only. The Dimension Line command can be used to measure the actual distance between two points at any angle.

Dimensioning information is always drawn from the first point to the second. If the information is to be drawn upside down, for example, set the point on the right first, then the point on the left.

Drawing Units as specified by the Units command are used for units of measurement. The size of the numbers and arrows is determined by the default text size.

Any of several dimension formats may be used in DesignCAD 3-D. The dimension formats are selected using the System Parameter command. These formats are shown in Figure 39.

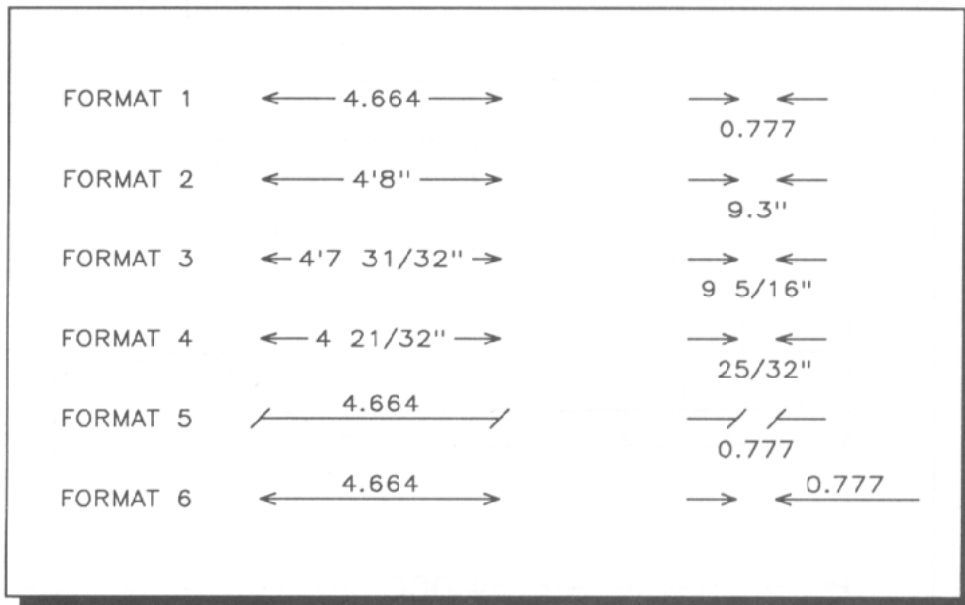


Figure 39 - Dimension Formats.

Dimension information can be maintained dynamically if specified in the Parameter command. This means that the dimensioning information will be automatically changed to correspond to the current units of measurement. If dynamic dimensioning is disabled,

the dimensions will be drawn as lines and text and can be edited as such.

Example: Measure the horizontal distance between two locations in the drawing. Use the Dimension command. Set a point at the measurement location on the left. Set a second point at the measurement location on the right. Set a third point beneath and between the first two points. The horizontal dimension will be added to the drawing.

Related Topics: Dimension Angle, Dimension Line.

Dimension Angle Command

| | |
|---------------|------------------|
| Menu: | NOTES |
| Menu Command: | Dim Angle |
| Short Form: | DA |

Point 1: Center point of the angle to be measured.

Point 2: Beginning point of the angle.

Point 3: Ending Point of the angle.

Point 4: Position for dimension arc (optional).

This command is used to draw angular dimensioning information automatically. Three or four points are used with this command. The first point is the center of the angle to be measured. The second and third points represent the beginning and ending of the angle to be measured.

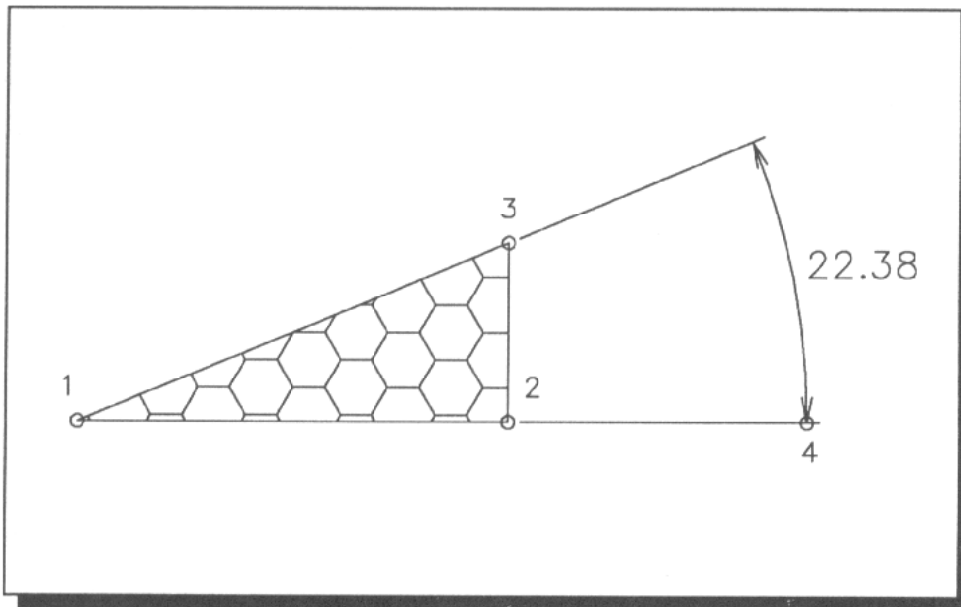


Figure 40 - Set one point at the angle center and two points for the angle "legs." Set the third point where the dimension arc is to be located.

The fourth point may be used to specify the location for the arrows and text. If the fourth point is not used, then the text and arrows will be drawn at the second point.

The dimension format parameter in the System Parameters does not affect the angular dimensions. It affects only the linear dimensions created by the Dimension and Dimension Line commands.

Example: Dimension the corner of a pentagon. Use the Polygon command to draw a pentagon on the X-Y plane. Use the Dimension Angle command to dimension the lower left corner. Set a point at the lower left corner of the pentagon. Set a second point at the lower right corner. Set the third point at the upper left corner. Set the fourth point at the location for the dimension information. The angle of the pentagon will then be displayed in the drawing.

Tip: The second and third points of the Dimension Angle command can be located anywhere on the "leg" of the angle. Use the Gravity Point or Line Snap command to place these points precisely.

Related Topics: Dimension, Dimension Line.

Dimension Line Command

| | |
|---------------|-----------------|
| Menu: | NOTES |
| Menu Command: | Dim Line |
| Short Form: | DL |

Point 1: First point to be measured.
Point 2: Second point to be measured.
Point 3: Location for dimension information (optional).

This command is used to draw dimensioning information automatically. Three points are used with this command. The first two points are the points to be measured. The third point is the location for the dimensioning information.

The dimension information will be drawn at the third point. The distance measured will be the distance between the first two points, at the angle between those points.

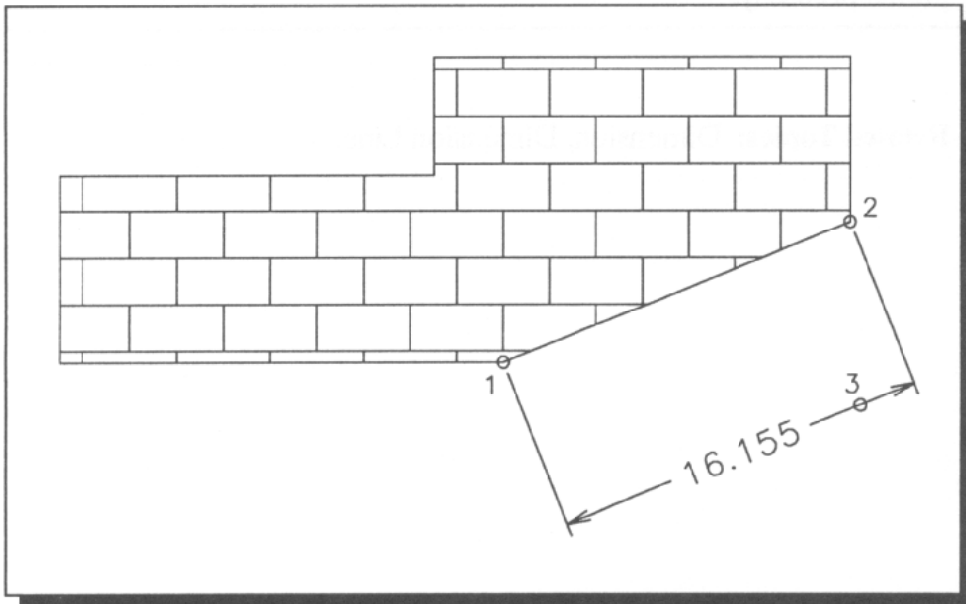


Figure 41 - Set two points for the distance to be measured and a third point where the text is to be located.

Dimension information can be maintained dynamically if specified in the System Parameter command. This means that the dimensioning information will be automatically changed to correspond to the current units of measurement. If dynamic dimensioning is disabled, the dimensions will be drawn as lines and text, and can be edited as such.

Any of several dimension formats may be used in DesignCAD 3-D. The dimension formats are selected using the System Parameter command. These formats are shown in Figure 39.

Example: Dimension the distance from the top front left corner to the top right rear corner of a 3-dimensional box. Draw the box using the Box command. Use the Dimension Line command to dimension the box. Set a point at the top front left corner and a second point at the top right rear corner. Set a third point above the second, where the dimension information is to be placed. The distance between the points will be displayed in the drawing.

Tip: Use the Text Size command to set the size of the text in dimensions.

Related Topics: Dimension, Dimension Angle.

Directory Command

| | |
|--------------------|------------------|
| Menu: | FILES |
| Menu Command: | Directory |
| Short Form: | Dir |
| Keystroke Command: | F7 |

Points: None.

The Directory command is used to display a list of DesignCAD drawing or other files on disk.

The file listing is shown at the left of the screen. This initially contains a listing of DesignCAD drawings. To the right of the file listing is a list of drives and directories. You can select from this list to change drives or directories.

You can use the **Tab** key or the mouse to move to the File Name box at the upper left, and enter a file specification for the directory listing. For example, you could enter **C:\G*.*** to for a listing of all files on the root directory of drive C that begin with the letter G.

In the directory command, you can delete a file by moving the cursor to that file and pressing **Alt-D** (hold the **Alt** key down and press **D**).

Related Topics: DOS command

Disc Command

Menu: **ARC/CIRC**
Menu Command: **Disc**

Point 1: Center of the disc or circle.
Point 2: Any point on the outside of the disc.
Point 3: Plane definition (optional).

The Disc command draws a round plane based on a point in the center and a point on the outside of the plane. It is identical to the Circle command, except with the Disc command the circle is a plane and not a line.

When you use the Disc command, a circular plane will be drawn through the second point with its center at the first point. If the third point is used, it defines the conceptual plane on which the circle lies.

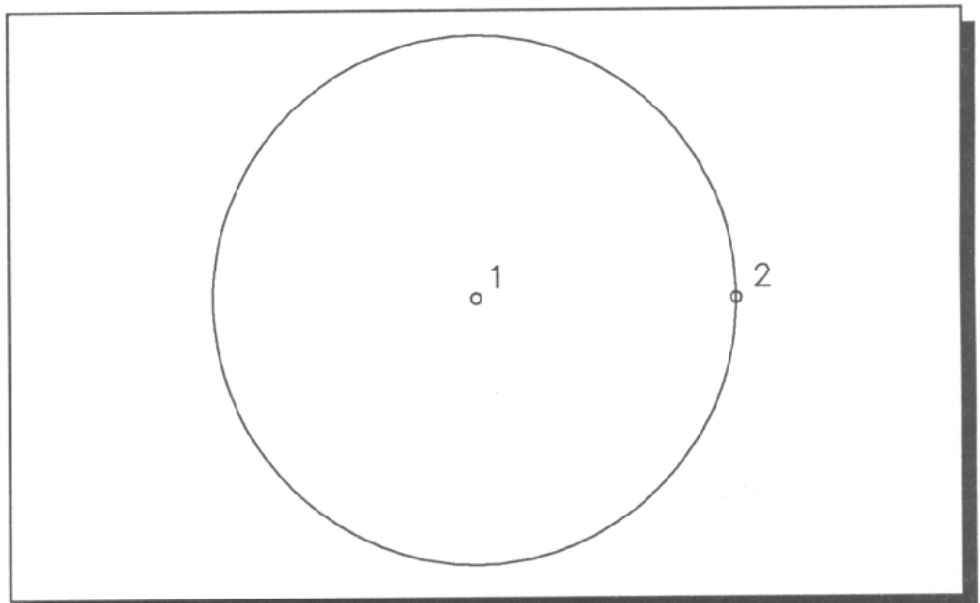


Figure 42 - Set a point in the center and a point on the outside of the circle. A third point can be used to define the plane for the circle.

Example: Draw a round surface on the X-Z plane. Use the Disc command. Set a point for the center of the circle. Set a second point to the right of the first. The distance between these two points will be the radius of the circle. Set a third point behind, or in the positive Z direction from the second. This causes the circle to lie "flat" on the X-Z plane instead of "standing up" on the X-Y plane.

Related Topics: Circle.

DOS Command

| | |
|---------------|--------------|
| Menu: | FILES |
| Menu Command: | DOS |

Points: None.

The DOS command is used to temporarily exit DesignCAD 3-D to DOS. After performing any DOS functions, you can enter **exit** from the DOS prompt to return to DesignCAD 3-D.

When you use this command to exit or "shell" to DOS, DesignCAD 3-D remains in memory. Because of this, you will have limited memory available under DOS. Some programs may not run under the DOS shell because of this memory limitation.

Before you exit to DOS, you will be asked if you want to save your drawing (if it has been modified since it was last saved). It is a good idea to save it, because under DOS it is possible to allocate memory, erase files, etc., so that you cannot return to DesignCAD.

Example: Exit to DOS, copy a file, and then return to DesignCAD. Select **DOS** from the FILES menu. (You may be asked if you want to save your drawing.) Under DOS, copy the file using the DOS Copy command. The enter **Exit** from the DOS prompt. You will then return to DesignCAD with your drawing on the screen.

Drawing Parameter Command

| | |
|--------------------|---------------------|
| Menu: | PARAMETER |
| Menu Command: | Drawing Parm |
| Short Form: | DP |
| Keystroke Command: | Q |

Points: None.

The Drawing Parameter command is used to specify DesignCAD 3-D drawing parameters, such as Cursor Step Size, Default Text Size and Angle, and Snap Grid size.

When you use the Drawing Parameter command, a list of the parameters will be displayed on the screen. To change one of the parameters, move the cursor to that location and enter the new value. You can also use the right and left mouse buttons or the space bar to change the value of the parameters.

The drawing parameters are described below.

Color

The Color parameter defines the color in which subsequent lines are to be drawn. The drawing screen frame is displayed in the current color.

Large Cursor Step Size**Small Cursor Step Size**

The Cursor Step Size determines the distance the cursor travels when one of the numeric keypad keys, **F1**, or **F2** is pressed. A cursor step size of 1 causes the cursor to be moved a distance of 1 drawing unit when a cursor key is pressed. The initial cursor step size is 1.

The small cursor step size is the distance moved when a cursor movement key (arrows, **F1**, **F2**) is pressed with the **Shift** key. The large cursor step size is the distance moved when a cursor movement key is pressed without the **Shift** or **Num Lock** key.

When the drawing is zoomed, the cursor step size normally remains constant with respect to the screen. This means that when you zoom to make the drawing 3 times larger, the cursor step size will be 3 times smaller on the drawing.

Under the System Parameters, you can change the cursor step size to be constant with respect to the drawing instead of the screen. This causes the cursor movement to be greater on the screen when you increase the drawing size.

The Cursor Step sizes are measured in Drawing Units. For more information, see the Cursor Step command.

Default Text Size

The default text size determines the height of the text when only one point (or two points in the same location) is used with the Text command (T key). A default text size of 1 causes text to be drawn 1 drawing unit high, for example. The default text size also determines the size of dimensioning labels and arrowheads.

Default Text Angle

The default text angle determines the angle at which text is to be displayed when only one point (or two points in the same location) is used with the Text command (T key). An angle of 0 is normal horizontal text, 90 is vertical beginning at the bottom, 180 is upside down, etc.

Wall Thickness

The wall thickness parameter determines the thickness of the walls produced by the Wall command. See the Wall command for more information.

Display Grid Size

This option sets the distance between lines of the display grid. A display grid size of 1, for example, will space the lines of the display grid 1 drawing unit apart. See the Grid command for more information.

Display Grid Extent (Grid Lines) from the center

This option determines the overall size of the display grid. It specifies how many lines in each direction from the center are to be displayed with the display grid. See the Grid command for more information.

Display Grid Plane (1 = XY, 2 = XZ, 3 = YZ)

With this option, you can select the X-Y, X-Z, or Y-Z plane for the display grid.

Status Line Format

One of three status line formats can be selected. The status line formats display the following information:

1.
 - a. Number of Points Set.
 - b. Current Layer.
 - c. Current Zoom Factor.
 - d. Current Color.
 - e. SNAP, if the Snap Grid is on.
 - f. MACRO, if a Macro is being made.
2.
 - a. Number of Points Set.
 - b. Distance from the cursor to the last point.
 - c. Delta X, Horizontal distance from the cursor to the last point.
 - d. Delta Y, or vertical distance from the cursor to the last point.
 - e. Delta Z, or vertical distance from the cursor to the last point.
3.
 - a. Number of Points Set.
 - b. X coordinate of the cursor.
 - c. Y coordinate of the cursor.
 - d. Z coordinate of the cursor.

The values in formats 2 and 3 are updated as the cursor is moved about the screen. The distance and X, Y, Z coordinates are measured in Drawing Units specified by the Units command (**U** key).

Point Type

The Point Type selects the type of point to be displayed on the screen. Point type 1 is a single dot, and point type 2 is a "+".

Cursor Type

The Cursor Type selects the type of cursor to be displayed on the screen. Cursor 1 is the large 3-D cursor, cursor 2 is a small "+", and cursor 3 is a small "x".

Use 2-D Point Snap?

The Gravity Point and Gravity Move commands can operate in two modes: 3-D snap and 2-D snap. The 3-D snap is the normal 3-dimensional Gravity function - the cursor moves to the nearest point in 3-D space. This may not be the nearest point to the cursor on the 2-dimensional screen, however.

With the 2-D snap, the cursor moves to the nearest point on the screen 2-dimensionally - it moves the shortest possible distance on the screen. The actual distance moved in 3-D space may be considerable, however, because the cursor position in 3-D space may be very different from the nearest point on the 2-D screen.

You can press **F1** for help, to display information on the drawing parameters.

The Drawing Parameter command can be terminated three ways:

- F2** - Use the drawing parameters but do not save them to disk.
- F3** - Save the parameters as program defaults. This causes the drawing parameters to be used every time the program begins. It is equivalent to the Configuration Save command.
- Esc** - Return and don't change anything.

Tip: To remove the default parameters and return to the "factory" DesignCAD 3-D settings, erase the file DCAD4.SYS.

Related Topics: System Parameters.

Drill Command

Menu: **SOLIDS**
Menu Command: **Drill**

Point 1: Set a point on the Solid to be removed from the Block.

This command is used to drill or remove a hole of any shape from a Block. In essence, it subtracts a Solid from a Block. This is similar to the Subtract command, which subtracts one Solid from another Solid. However, the Drill command makes it possible to drill holes in single surfaces and other non-Solid objects (see the section in this manual on "Using Solids" for more information on Solids).

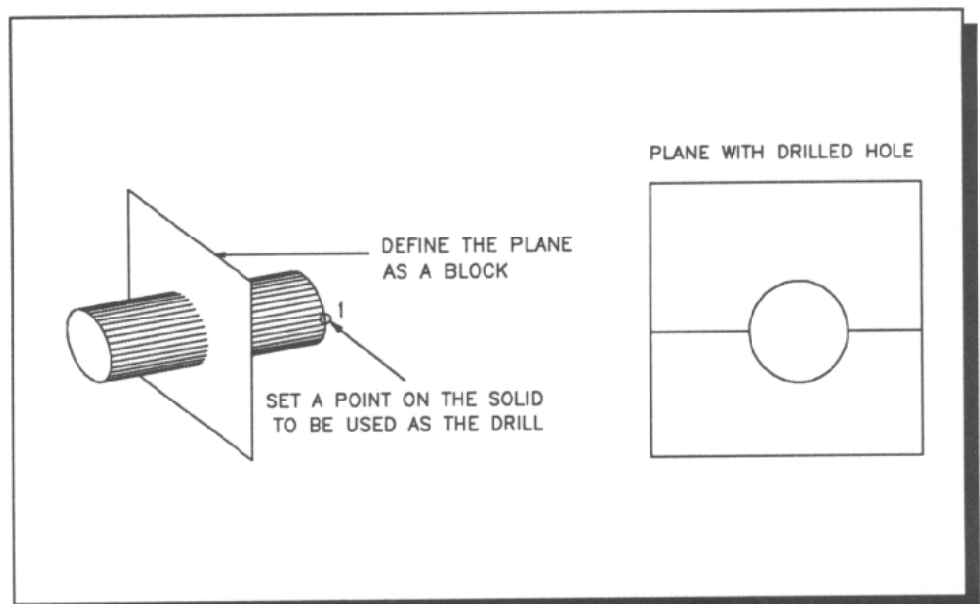


Figure 43 - The Drill command.

To use the Drill command, first make a Block of the object in which you want to drill a hole (see the Block Define command). Then make a solid object in the shape you want removed from the Block. (For example, to drill a round hole use a cylinder for the Solid.)

Next, set a point on the Solid you want to be removed from the Block. All surfaces in the Block will be cut and the parts that lie within the Solid will be removed.

Example: Draw a square plane with a hole in it. First, draw a 2-dimensional box for the square plane. Then draw a cylinder that passes through the square plane. Define the plane to be a Block. Use the Drill command to subtract the cylinder from the Block (the plane, in this case) - set a point on the cylinder when asked to set a point on the cutting Solid. A hole will then be cut in the plane.

Related Topics: Cut Plane, Solid Subtract.



Erase Command

| | |
|--------------------|--------------|
| Menu: | EDIT |
| Menu Command: | Erase |
| Short Form: | E |
| Keystroke Command: | E |

Points 1-n: Set a point on each line or entity to be erased.

The Erase command can be used to erase individual lines or entities in the drawing.

To erase an entity with this command, set points on the entities to be erased, or on one of the points used to create the entities (such as the center of an arc).

The Unerase command (! key) can be used to replace items accidentally deleted with the Erase command.

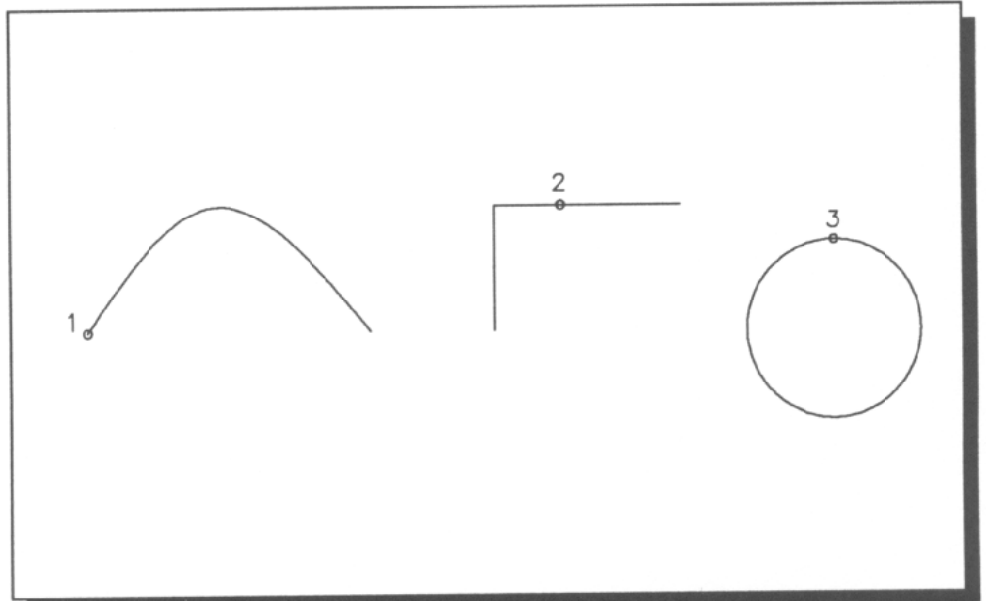


Figure 44 - The Erase command erases items in the drawing. Set a point on each item to be erased.

Example: Draw three lines and erase the center one. Draw the three lines. Then set a point on the center line and press the **E** key. That line will be erased.

Tip: In a large drawing, the Erase command will be faster if you use the Gravity Point command to set the points.

Related Topics: Undo, Section Delete, Unerase.

Exit Command

| | |
|--------------------|--------------|
| Menu: | FILES |
| Menu Command: | Exit |
| Keystroke Command: | F8 |

Points: None.

This command terminates the DesignCAD 3-D session and returns to the operating system. If your drawing has been changed since it was last saved, you will be asked if you want to save the drawing before the program terminates.

File Convert Command

| | |
|---------------|---------------------|
| Menu: | FILES |
| Menu Command: | File Convert |
| Points: None. | |

This command runs the file conversion utility DCFILES. It can be used to convert to and from other graphics file formats. For more information on file conversion, see the section in this manual "DesignCAD Utilities."

When you execute the File Convert command, you will be asked if you want to proceed with the File Conversion and if you need to save your drawing.

The file conversion utility is actually a separate program from DesignCAD 3-D, so the current drawing must be saved before running the utility or it will be lost.

Fillet Command

| | |
|--------------------|---------------|
| Menu: | EDIT |
| Menu Command: | Fillet |
| Short Form: | F |
| Keystroke Command: | F |

Point 1: First line making up the corner to be rounded off.

Point 2: Second line making up the corner to be rounded off.

The Fillet command "rounds off" a corner with a smooth arc. To use the command, enter **FILLET** and set a point on each of the two lines making up the corner to be filleted.

You will then be asked for the fillet radius. Enter the radius of the arc in the fillet. An arc will be drawn at the intersection of the two lines tangent to both lines, making a smooth corner.

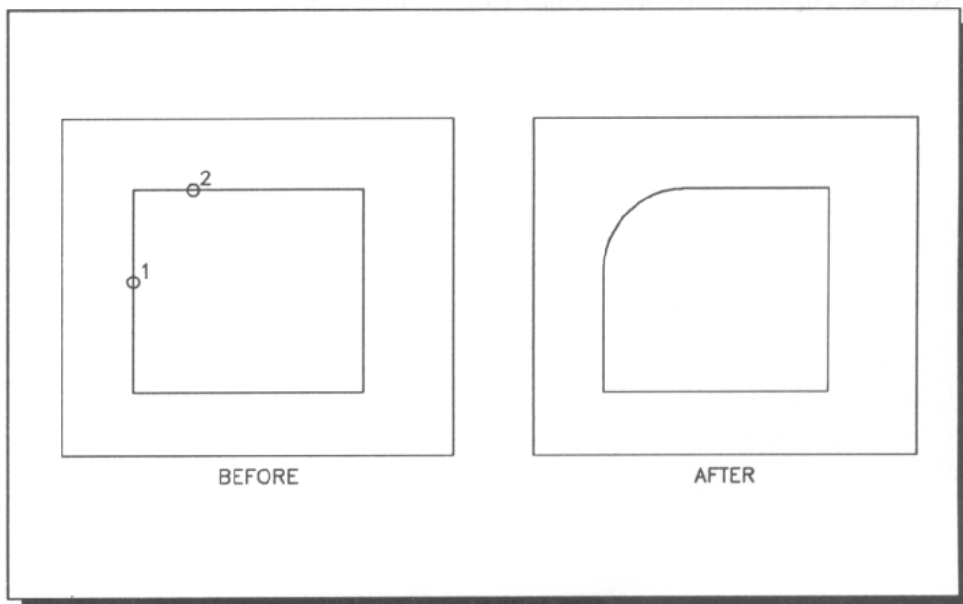


Figure 45 - The Fillet command is used to "round off" a corner. Set a point on each of the two lines making up the corner.

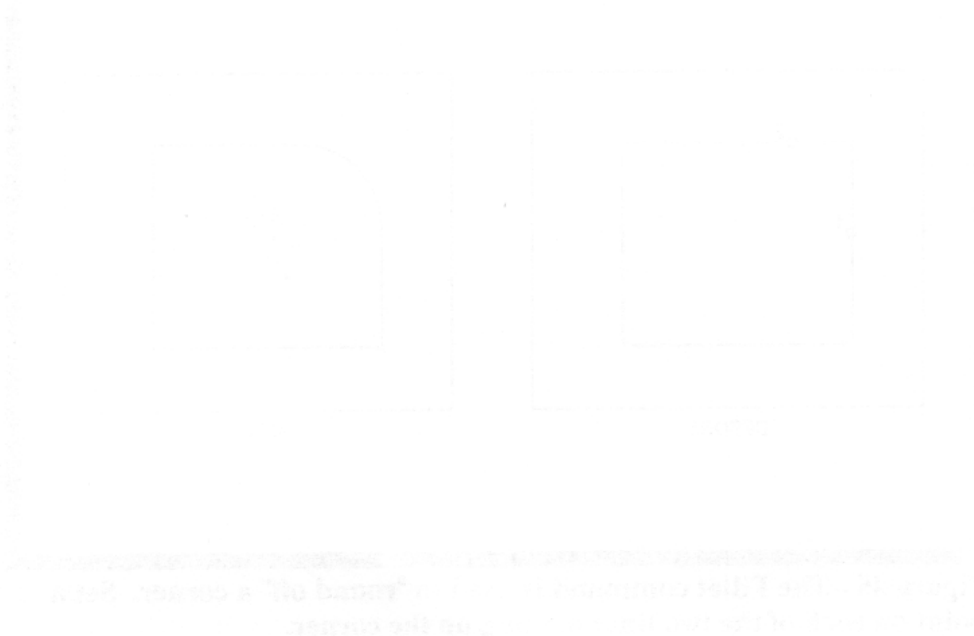
The two lines selected do not have to actually meet, and the lines can cross one another. The lines will be trimmed to the endpoints of the arc. The two lines do have to lie in the same plane and must not be parallel. In other words, it must be possible to draw an arc tangent to both lines in three dimensions.

The Chamfer command is similar to the Fillet command, but it "rounds off" the corner with a straight line instead of an arc.

The Fillet command works on both lines and planes. The Fillet Edge and the Fillet Corner commands can be used to fillet Solid objects.

Example: Fillet the corner of a 2-dimensional box. Draw a 2-dimensional box with the Box command. Use the Fillet command and set 1 point on the top edge and 1 point on the right edge of the box. Enter **3** when asked for the fillet radius. The box's upper right corner will be rounded off by an arc with a radius of 3.

Related Topics: Chamfer, Fillet Edge, Fillet Corner.



Fillet Corner Command

| | |
|---------------|----------------------|
| Menu: | EDIT |
| Menu Command: | Fillet Corner |
| Short Form: | FC |

Point 1: Set a point on the corner to be filleted.

The Fillet Corner command is used to fillet or round off the corner of the Solid object, such as a box. This command performs a 2-dimensional fillet on each of the 3 surfaces meeting at the corner and places a smooth surface over the rounded corner.

With this command, you set a point on the corner to be filleted and enter the fillet radius. You will then be asked for the number of facets for the fillet. This determines the smoothness of the fillet. The more facets you use, the smoother the fillet. However, more facets occupy more memory and take more time to draw.

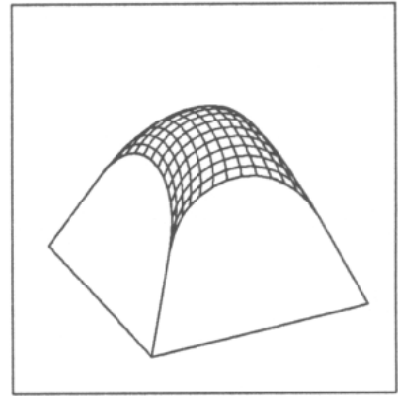
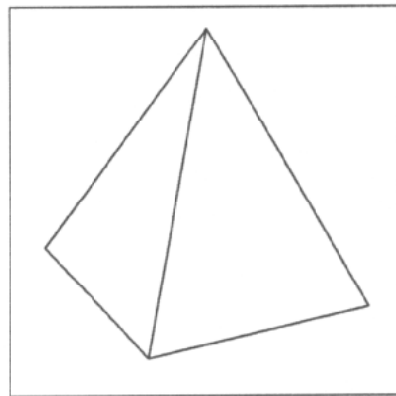


Figure 46 - The Fillet Corner command makes a rounded corner on a 3-dimensional object.

Example: Fillet the corner of a Box. Use the Fillet Corner command and set a point on a corner of the box. Enter **3** when asked for the fillet radius. Enter **10** when asked for the number of facets. The corner of the box will then be rounded off.

Related Topics: Fillet, Fillet Edge.



Fillet Edge Command

| | |
|---------------|--------------------|
| Menu: | EDIT |
| Menu Command: | Fillet Edge |
| Short Form: | FE |

Point 1: Point on the edge line.

Point 2-5: Points on the lines connected to the edge.

The Fillet Edge command is used to fillet an edge of a Solid object, such as a box. This command replaces the sharp edge with a rounded edge.

With this command, you set a point on the edge to be filleted. You will be asked for the fillet radius on each end of the fillet and for the number of facets on the rounded surface. The number of facets determines the smoothness of the edge. The more facets you use, the smoother the edge. However, more planes occupy more memory and take more time to draw.

You can use a different fillet radius at each end of the edge to make a smooth transition from one size fillet to the other.

Example: Fillet the edge of a Box. Use the Fillet Edge command and set a point on the front left vertical edge of the box. Enter **3** when asked for the fillet radius for each end. Enter **10** when asked for the number of facets. The edge of the box will then be rounded off.

Related Topics: Fillet, Fillet Corner.

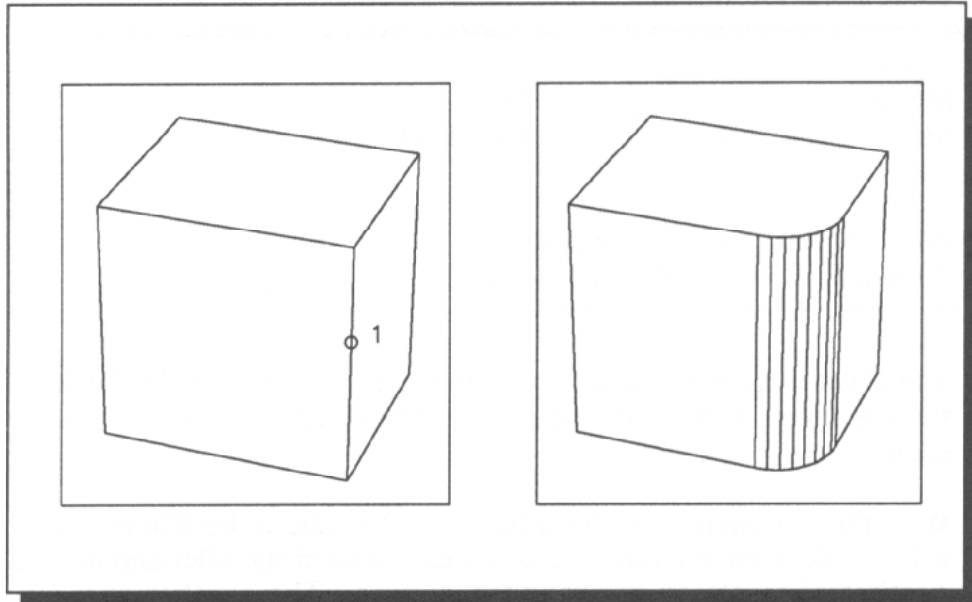


Figure 47 - The Fillet Edge command "rounds off" the edge of a 3-dimensional object

Font Command

Menu: **PARAMETER**
Menu Command: **Font**

Points: None.

The Font command is used to specify the character font to be used for text in the drawing. After a new font is selected, all the text entities on the screen are erased and re-written in the new character font. The DesignCAD 3-D fonts are shown in Figure 48.

When asked for the font number, you can press **F1** to display the available fonts. You can optionally enter the font number on the command line with the Font command. For example, to select font number 6 you can enter **font 6** on the command line.

| | |
|---------------------------|------------------------------------|
| 1. Simplex 1 | ABCDEFGFGabc defg |
| 2. Simplex Curved | ABCDEFGFGabc defg |
| 3. Simplex 2 | ABCDEFGFGabc defg |
| 4. Complex Roman | ABCDEFGFGabc defg |
| 5. <i>Complex Italic</i> | <i>ABCDEFGFGabc defg</i> |
| 6. Duplex Roman | ABCDEFGFGabc defg |
| 7. Triplex Roman | ABCDEFGFGabc defg |
| 8. <i>Triplex Italic</i> | <i>ABCDEFGFGabc defg</i> |
| 9. <i>Simplex Script</i> | <i>ABCDEFGFGabc defg</i> |
| 10. <i>Complex Script</i> | <i>ABCDEFGFGabc defg</i> |
| 11. Σιμολεχ Ηρεεκ | ABΓΔΕΖΗαβγδεζη |
| 12. Γξμολεχ Ηρεεκ | ABΓΔΕΖΗαβγδεζη |
| 13. Вомпллѣ Вшснллив | АВВГДЕЖабвгдеж |
| 14. Gothic English | A B C D E F G a b c d e f g |
| 15. Gothic German | A B C D E F G a b c d e f g |
| 16. Gothic Italian | A B C D E F G a b c d e f g |

Figure 48 - DesignCAD 3-D Fonts.

The Vector Text command (^T key) can be used instead of the Text command to save the text as line entities. This prevents text from being redrawn in the new character font, and allows more than one character font to be used in the drawing. However, text drawn using the Vector Text command takes more memory than does text drawn with the Text command.

Character font number 0 may be used to "hide" the text. When font 0 is used, each line of text in the drawing is displayed as a rectangle, although the text is still part of the drawing. Since the text is not redrawn when the Zoom and Rotate functions are performed, the Zoom and Rotate commands are faster when character font 0 is used.

Related Topics: Text, Vector Text, Drawing Parameters.

Get Entity Command

| | |
|---------------|-------------------|
| Menu: | POINTS |
| Menu Command: | Get Entity |

Point 1: Set a point on the entity to select.

The Get Entity command is used primarily in BasicCAD programs to access entities (lines, curves, circles, etc.) in the drawing.

The Get Entity command sets the points of the entity closest to the cursor. For example, you could place the cursor near a line and select **Get Entity**. The line will remain, but all points making up the line will be set. These points can then be used with another DesignCAD command or with a BasicCAD statement such as **POINTVAL**.

In addition, the layer and color of the line can be accessed within BasicCAD using the **SYS** function. For more information, see the *BasicCAD Reference Manual*.

Gravity Move Command

| | |
|--------------------|---------------------|
| Menu: | CURSOR |
| Menu Command: | Gravity Move |
| Short Form: | Gmove |
| Keystroke Command: | , (comma) |

Points: None.

This command is used to move the cursor to the nearest point in the drawing (or the nearest point that has been set). To use the command, select **Gravity Move** from the Points menu or press the comma key (","). The cursor will move to the nearest point in the drawing.

There are two snap modes that can be used with the Gravity commands: 3-D Snap and 2-D Snap. The 2-D/3-D point snap can be set in the Drawing Parameter command. The 3-D snap is the normal, 3-dimensional Gravity function - the cursor moves to the nearest point in 3-D space. This may not be the nearest point to the cursor on the 2-dimensional screen, however.

With the 2-D snap, the cursor moves to the nearest point on the screen 2-dimensionally. With the 2-D snap, the cursor moves the shortest possible distance on the screen. The actual distance moved in 3-D space may be considerable, however, because the cursor position in 3-D space may be very different from the nearest point on the 2-D screen.

If you use a mouse to select the Gravity Move command from the menu, you will be asked to set a point from which the cursor movement is to be calculated.

If the current drawing has been zoomed in and out, or if it has been retrieved at various sizes, the points of the drawing may not lie exactly on the dots or pixels on the screen. This command allows you to access these points to extend and intersect lines with accuracy.

Related Topics: Gravity Point, Drawing Parameter.

Gravity Point Command

| | |
|--------------------|-----------------------|
| Menu: | POINTS |
| Menu Command: | Gravity Point |
| Short Form: | GV |
| Keystroke Command: | . (period key) |

Points: None.

This command is used to move the cursor to the nearest point in the drawing (or the nearest point that has been set) and set a point there. To use the command, select **Gravity Point** from the Points menu or press the period key ("."). The cursor will move to the nearest point in the drawing and a point will be set there.

There are two snap modes that can be used with the Gravity commands: 3-D Snap and 2-D Snap. The 2-D/3-D point snap can be set in the Drawing Parameter command. The 3-D snap is the normal, 3-dimensional Gravity function - the cursor moves to the nearest point in 3-D space. This may not be the nearest point to the cursor on the 2-dimensional screen, however.

With the 2-D snap, the cursor moves to the nearest point on the screen 2-dimensionally. With the 2-D snap, the cursor moves the shortest possible distance on the screen. The actual distance moved in 3-D space may be considerable, however, because the cursor position in 3-D space may be very different from the nearest point on the 2-D screen.

If you use a mouse to select the Gravity Point command from the menu, you will be asked to set a point from which the cursor movement is to be calculated.

If the current drawing has been zoomed in and out, or if it has been retrieved at various sizes, the points of the drawing may not lie exactly on the dots or pixels on the screen. This command allows you to access these points to extend and intersect lines with accuracy.

Related Topics: Gravity Move, Intersect, Line Snap.

Grid Command

| | |
|--------------------|----------------|
| Menu: | DISPLAY |
| Menu Command: | Grid |
| Short Form: | G |
| Keystroke Command: | ^ G |

Points: None.

The Grid command enables or disables the Display Grid. To turn the display grid on or off, select **Grid** from the **DISPLAY** menu or press **^ G**.

The "display grid" is a grid of lines displayed on the screen. The display grid does not become part of the drawing. It cannot be printed or plotted.

The display grid can be oriented on the X-Y, X-Z, or Y-Z plane. Its size, orientation, and spacing can be set in the Drawing Parameters command.

Related Topics: Grid Center, Drawing Parameters.

Grid Center Command

| | |
|---------------|--------------------|
| Menu: | DISPLAY |
| Menu Command: | Grid Center |
| Short Form: | Gcenter |

Point 1: Center of the display grid.

The Grid Center command sets the location for the center of the display grid.

The display grid can be oriented on the X-Y, X-Z, or Y-Z plane. Its size, orientation, and spacing can be set in the Drawing Parameters command.

Example: Set the center of the display grid to the bottom of a box. Use the Grid Center command and set a point at the bottom of the box. The display grid will be located there. Use **^G** to turn the display grid on or off.

Related Topics: Grid, Drawing Parameters.

Hammer-1 Command

| | |
|---------------|-----------------|
| Menu: | EDIT |
| Menu Command: | Hammer-1 |
| Short Form: | H1 |

Point 1: Point on the grid entity you want to move.

Point 2: New location for point 1.

The Hammer-1 command is used to "dent" or "stretch" a surface. A point on the surface is moved to a new location, and the surface is "stretched" to the new location. The Hammer-1 command makes a rounded "bump", while the Hammer-2 command makes a sharp "point."

Two points are used with this command. The first point is the point on the surface that will be moved. The second point is the new location for the first point. The first point must be set on a grid entity.

After setting the points, you will be asked to enter the radius. The radius represents the size of the area on the surface that will be "stretched" by the Hammer command. For example, if you enter **3** for the radius, a bump with a radius of 3 will be stretched up (or down) on the surface.

A grid selected with the Hammer command should have grid lines running both horizontally and vertically. While the sides of a cylinder are made of a grid entity, it cannot be used successfully with the Hammer command because there are no intermediate breaks.

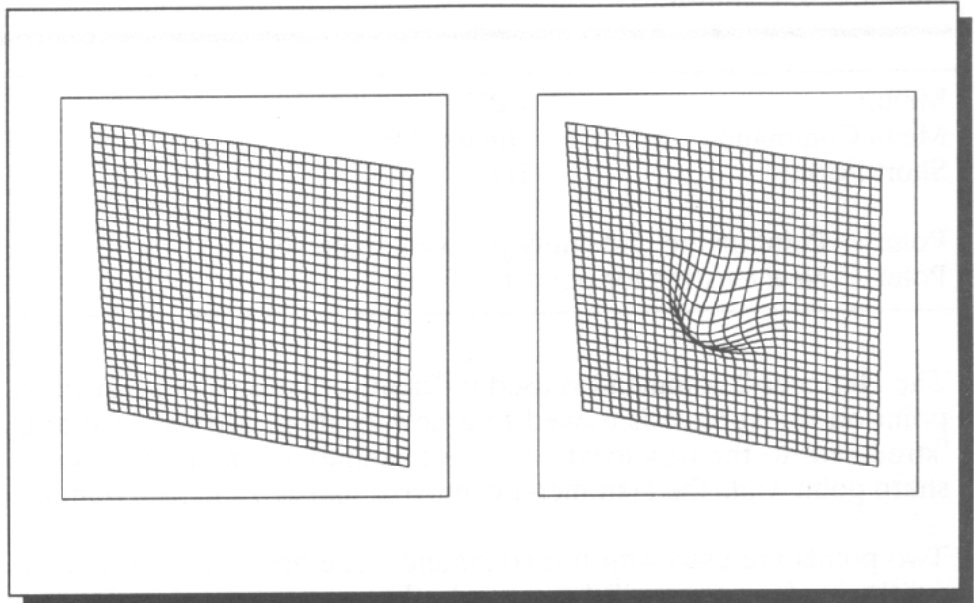


Figure 49 - The Hammer-1 command is used to "bend" or stretch a surface with rounded bump.

Example: Use the **Hammer-1** command to make a bump on a surface. Draw two horizontal lines, one above the other. Set a point on each line and use the **Connect Curve** command to fit a surface between the lines. Enter **30** planes and **30** intermediate breaks when asked. Use the **Hammer-1** command, and set 1 point near the center of the surface and 1 point in the negative Z direction from the first point (use the **F2** key). Enter **3** when asked for the hammer radius. A rounded bump with a radius of 3 will be made, moving the surface from the first point to the second point.

Tip: Use a relatively fine grid when creating a surface that will be used with the Hammer commands.

Related Topics: Hammer-2, Point Move.

Hammer-2 Command

| | |
|---------------|-----------------|
| Menu: | EDIT |
| Menu Command: | Hammer-2 |
| Short Form: | H2 |

Point 1: Point on the grid entity you want to move.

Point 2: New location for point 1.

The Hammer-2 command is used to "dent" or "stretch" a surface. A point on the surface is moved to a new location, and the surface is "stretched" to the new location. The Hammer-2 command makes a sharp point while the Hammer-1 command makes a rounded bump.

Two points are used with this command. The first point is the point on the surface that will be moved. The second point is the new location for the first point. The first point must be set on a grid entity.

After setting the points, you will be asked to enter the radius. The radius represents the size of the area on the surface that will be "stretched" by the Hammer command. For example, if you enter **3** for the radius, a bump with a radius of 3 will be stretched up (or down) on the surface.

A grid selected with the Hammer command should have grid lines running both horizontally and vertically. While the sides of a cylinder are made of a grid entity, it cannot be used successfully with the Hammer command because there are no intermediate breaks.

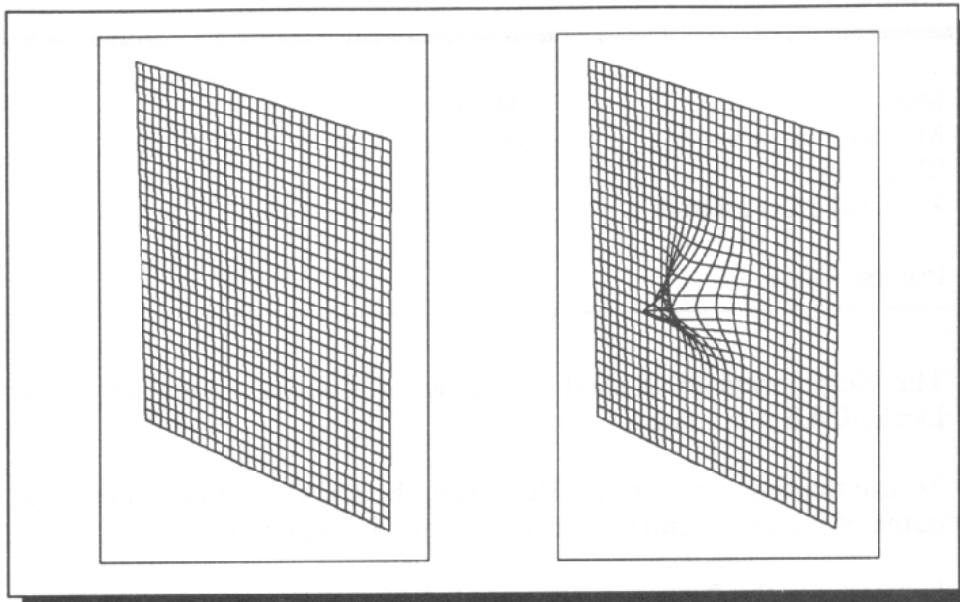


Figure 50 - The Hammer-2 command is used to "dent" or stretch a surface to a point.

Example: Use the **Hammer-2** command to make a point on a surface. Draw two horizontal lines, one above the other. Set a point on each line and use the **Connect Curve** command to fit a surface between the lines. Enter **30** planes and **30** intermediate breaks when asked. Use the **Hammer-2** command, and set 1 point near the center of the surface and 1 point in the negative Z direction from the first (use the **F2** key). Enter **3** when asked for the hammer radius. A rounded bump with a radius of 3 will be made, moving the surface from the first point to the second point.

Tip: Use a relatively fine grid when creating a surface that will be used with the Hammer commands.

Related Topics: Hammer-1, Point Move.

Help Command

| | |
|--------------------|-------------|
| Menu: | INFO |
| Menu Command: | Help |
| Short Form: | ? |
| Keystroke Command: | ? |

Points: None.

The Help command is used to display information about one of the DesignCAD commands.

To use the command, select **Help** from the INFO menu and enter the name of the command on which you need information.

If you press the **?** key, you can get help for any keystroke command by pressing the key for the keystroke command.

You can also display help for any DesignCAD 3-D command by pressing the right mouse button or **F1** when the command is highlighted on the menu.

Example: Display help for the Arc-3 command. Move the cursor to the ARC/CIRC menu and press the left mouse button (or press **Alt-A**). This brings down the ARC/CIRC menu. Then move the cursor to the Arc-3 command and press the right mouse button or **F1**. Help for the Arc-3 command will be shown on the screen.

Hemisphere Command

| | |
|---------------|-------------------|
| Menu: | SOLIDS |
| Menu Command: | Hemisphere |
| Short Form: | Hemi |

Point 1: Center of hemisphere.

Point 2: Outer edge or radius of hemisphere base.

Point 3: Plane definition of hemisphere top (optional).

This command is used to draw a hemisphere (half a sphere). Two or three points are used with the command. The first two points are the center and the radius of the hemisphere. The third point, if used, determines the direction in which the "hump" of the hemisphere lies.

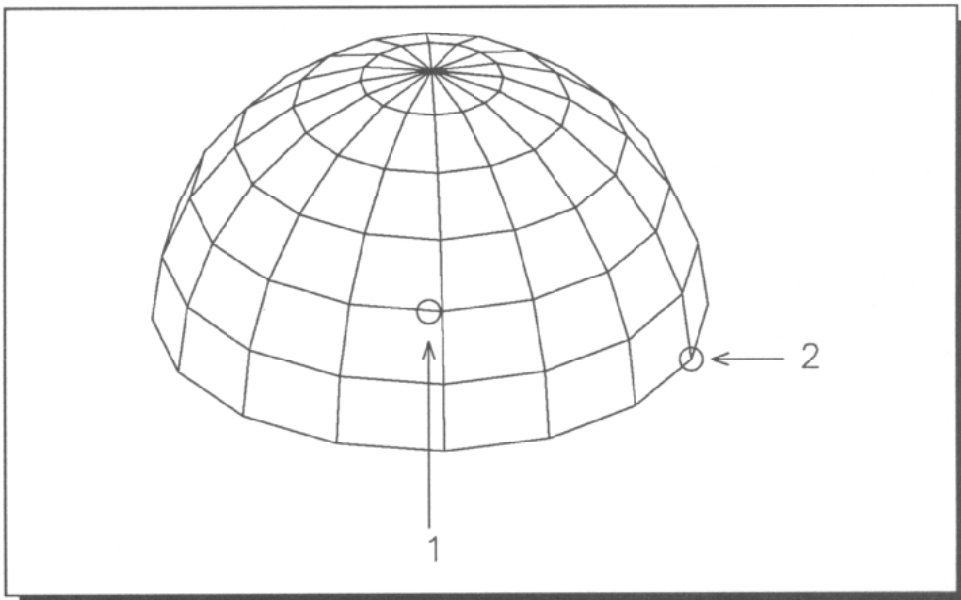


Figure 51 - With the Hemisphere command, set one point at the center of the base and a second point at the edge of the base. A third point can be used for the direction of the dome.

After setting the points, you can enter the number of facets for the hemisphere. This determines the smoothness of the hemisphere. The more facets you use, the smoother the surface. However, more facets occupy more memory and take more time to draw. You can press **Enter** to use the default of 16 facets.

Example: Draw a hemisphere with its base on the X-Z plane. Use the Hemisphere command. Set 1 point for the center and 1 point to the right of the first. Set the third point above the second point. The hemisphere will then be drawn.

Related Topics: Sphere, Cone.

Hide Commands

Menu: **DISPLAY**
Menu Command: **Hide, Hide2, Hide3, Hide4**
Points: None.

The Hide commands are used to perform hidden line removal on the drawing. This causes all lines behind surfaces to be removed from the screen. After the hidden line removal is performed, a more realistic image results.

Use **Hide** to perform hidden line removal on View 1 (the large view), and **Hide 2**, **Hide 3** and **Hide 4** to perform hidden line removal on views 2, 3, and 4.

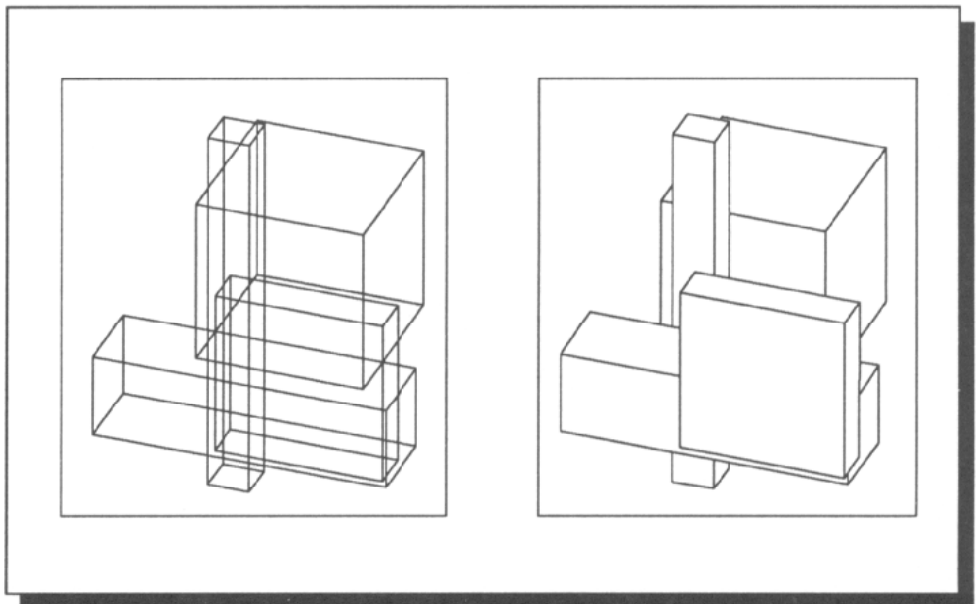


Figure 52 - The Hide command performs hidden line removal on a drawing.

This command does not need to be executed in order to save a 2-Dimensional image of the drawing with hidden lines removed or to plot the drawing with hidden lines removed.

The Hide command changes the screen display only. The drawing itself is not affected by the hide command.

Tip: You can abort the hidden line removal process by pressing **Esc**.

Tip: Hidden line removal can take a minute or more on larger drawings. It is a good idea to save your drawing beforehand.

Related Topics: View, View2, View3, View4, Set View.

Home Command

| | |
|--------------------|---------------|
| Menu: | POINTS |
| Menu Command: | Home |
| Keystroke Command: | ^H |
| Points: None. | |

The Home command moves the cursor to the center of the screen. This can be useful if the cursor is off the screen somewhere in the infinite world of 3-D space and you are not sure where. Press **^H** and the cursor will return to the center of the large view.

ID Command

| | |
|--------------------|--------------|
| Menu: | NOTES |
| Menu Command: | ID |
| Keystroke Command: | I |

Point 1: Set a point on the entity for information.

The ID command displays the entity type, color, layer, and number of points of a line or drawing entity.

If the entity is a line or surface, then the length or perimeter of the line or surface is displayed. If the line or surface is planar, that is, if all the points lie on a single plane, then the area will also be displayed.

The ID command also displays the word "SOLID" followed by the Solid number if the entity is part of a Solid. The word "BLOCK" will be displayed if the entity is part of a Block.

To use the command, set a point on the line or entity from which you want information. The information about that entity will be displayed on the screen.

Example: Determine the entity type of a Box. Draw a 3-dimensional box using the Box command. Then use the ID command and set a point on the box. The information on the entity will be displayed on the screen. Notice that the box is made up of grid entities, and that the box is a Solid.

Tip: You can use the ID command to quickly display the length of a line.

Interference Command

| | |
|---------------|---------------------|
| Menu: | SOLIDS |
| Menu Command: | Interference |
| Short Form: | IF |

Point 1: Set a point on one Solid.

Point 2: Set a point on the other Solid.

The Interference command performs interference checking on any two Solids in the drawing - it determines whether any parts of the two Solids occupy the same area.

Two points are used with this command, one on each Solid. If the Solids interfere with one another (if some parts of the Solids occupy the same space) then the message "Interference" will be displayed on the screen. Otherwise, the message "No Interference" will be displayed.

Example: Check to see if a box and a cone are touching one another. Draw the box and the cone. Using the Interference command, set one point on the box and a second point on the cone. After some calculation, either "Interference" or "No Interference" will be displayed on the screen.

Tip: You can use the Solid Intersect command to remove all of the two Solids except the "interfering" parts. Be sure to save the drawing first, if you want to keep the original Solids.

Related Topics: ID, Line Distance, Volume, Memory, Surface.

Intersect Command

| | |
|---------------|------------------|
| Menu: | POINTS |
| Menu Command: | Intersect |
| Short Form: | Int |

Point 1: Set a point on the first intersecting line.

Point 2: Set a point on the second intersecting line.

The Intersect command is used to set a point at the intersection of two lines. To use the command, select **Intersect** from the **POINTS** menu (or enter **int** on the command line) and set a point on or near each of the two intersecting lines. The cursor will then move to the intersection and a point will be set there.

This command temporarily erases any previously set points and replaces them after the command is complete. For example, you might have three points set and want to set a fourth point at an intersection. After "INTERSECT" is entered, the original 3 points are erased. You then set two points to designate the intersecting lines. The two points just set are erased, the original 3 points are brought back, and the fourth point is set at the intersection of the lines.

This command does not allow the points to be set before the command is entered.

The lines selected with the intersect command do not have to lie on the X-Y, X-Z, or Y-Z plane. However, they must intersect somewhere in 3-D space. It is not sufficient if the lines appear to intersect when they never actually meet.

Example: Set a point at the intersection of two lines. Draw two intersecting lines on the X-Y plane. Select **Intersect** from the **POINTS** menu, and set a point on each line. The two points you set will be replaced by a single point on the intersection of the lines.

Join Command

Menu: **EDIT**
Menu Command: **Join**

Point 1: Corner of area of points to be joined.
Point 2: Opposite area of points to be joined.

The Join command is used to join the endpoints of a number of lines.

Suppose, for example, that three lines are supposed to meet at a single point, and that they are not precisely on that point due to a drawing mistake or imprecise input data. The Join command will move the endpoint of each of the lines to the "average" point among them.

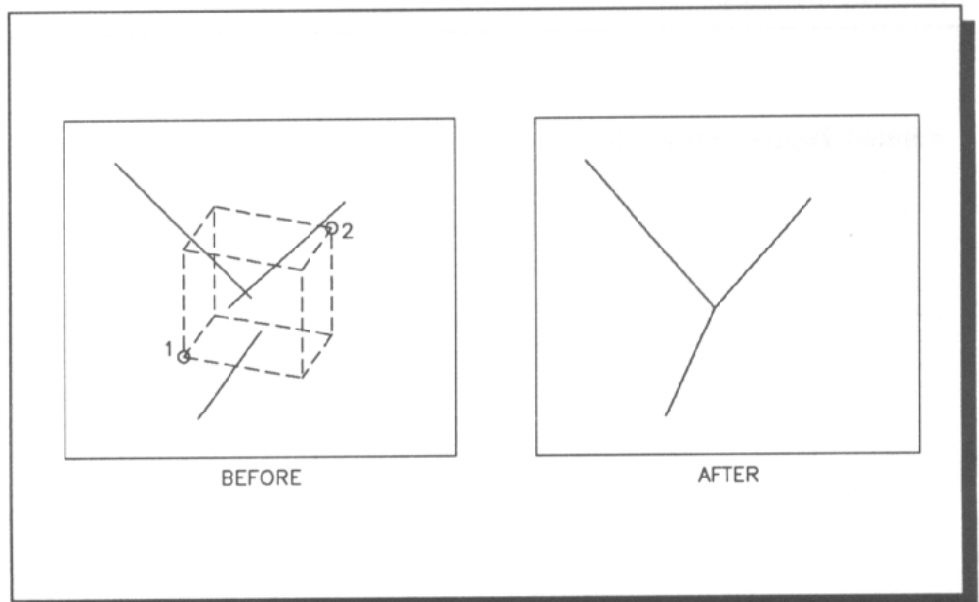


Figure 53 - The Join command moves all the endpoints in the section to a common location.

With the Join command, two points are used to designate an area that contains the endpoints of the lines to be joined. The points are set in opposite corners of the conceptual 3-dimensional box that encloses the endpoints.

Example: Join two lines whose endpoints are 1 cursor step apart. Draw two lines so that an endpoint of the first line is one cursor step away from the endpoint of the second line. Use the Join command and set two points that enclose both endpoints in a conceptual box. The endpoints are then moved to a common point between their former locations.

Tip: This command works in 3 dimensions as well as two.

Tip: The Combine Lines command requires that the lines to be combined be connected. The Join command can be used to connect the lines.

Related Topics: Point Move.

Layer Command

| | |
|--------------------|------------------|
| Menu: | PARAMETER |
| Menu Command: | Layer |
| Short Form: | L |
| Keystroke Command: | L |

Points: None.

A DesignCAD 3-D drawing can have more than sixty layers. A layer is a subdivision of the drawing. You can assign any entity or group of entities to any layer. You can display certain layers, modify certain layers, and print or plot certain layers.

When you create a drawing, the entities are added to the Current Layer. The Current Layer is layer 1 unless otherwise specified.

Each layer can be set to active or inactive status. An active layer is displayed on the screen, while an inactive layer is "invisible."

To bring up the Layer menu, select **Layer** from the **PARAMETER** menu, or press the **L** key. Under the Layer menu you can set the current and active layers, and can give a name to any layer. After entering the information, press **F2** to continue with the drawing.

You can also set the current layer on the command line. For example, entering **Layer 4** or **L 4** on the command line sets the current layer to layer 4.

There are several DesignCAD 3-D commands that modify or erase existing lines or entities. These commands can operate on the current layer only, or an option can be selected to allow lines or entities in all visible layers to be affected.

This option is set using the System Parameter command. You should be aware of this option's setting because it influences the effect of a number of DesignCAD 3-D commands. See the System Parameter command.

In the Layer menu, you can move up and down the left side of the screen using the mouse or the arrow keys. You can turn a layer on or off by entering 0 or 1 for that layer. At the top of the screen, you can enter the number for the current layer. To the right of the layer enable/disable fields, you can enter the layer names. Layers that contain drawing entities are marked with asterisks (*).

There are several other functions available in the Layer menu:

F1 - Help

This displays help for the Layer command.

F2 - Exit and use the new information

This function activates the changes made on the Layer menu.

F3 - Move entities from one layer to another

This "moves" one layer to another. It actually changes the layer on all the entities in one layer to another layer.

F4 - Delete the entities in a layer

This erases all the entities in a layer.

F5 - Save a layer to disk as a separate drawing

This lets you save a single layer in the drawing as a separate drawing.

F6 - Set the color for an entire layer

With this function you can assign a color to every entity in a layer. This function does not affect items drawn after the Layer command.

F7 - Separate the colors of the drawing into layers

This function places all entities of color 1 into layer 1, color 2 into layer 2, etc. This function can completely reorganize the layers of a drawing. Use it with care.

F8 - Define an entire layer to be a Block

Using this function, you can effectively use all the Block commands on a layer.

F9 - Set Material

This function invokes the Material command from within the Layer command. This can be useful because in the Layer command you can see all the layer names and which layers contain entities. For more information, see the Material command.

Esc - Cancel - Restore original layer information

This function restores the layer names, the current layer, and the layer enable/disable information. However, it does **not** undo the functions listed above. Functions **F3** through **F9** are executed immediately and cannot be undone. Use them with caution.

In the Layer menu, you can move from section to section with the mouse or with the **Tab** key. You can change numeric values by clicking on a field with the mouse, or by pressing the **Space** bar.

Tip: When a drawing is retrieved, entities in layer 0 are retrieved into the Current Layer, even if the Current Layer is not 0. Other layers are retrieved into their respective layers. This allows symbols to be saved in layer 0 and retrieved into the Current Layer.

Related Topics: Material command, System Parameters.

Lights Command

| | |
|---------------|---------------|
| Menu: | SHADE |
| Menu Command: | Lights |

Points: None.

This command allows you to control the 8 light sources for realistic lighting in shading. You can enable or disable eight light sources. You can also control the placement and intensity of each light source.

When you use this command, the light source screen will be displayed. From this screen you can enter the horizontal angle, vertical angle, and intensity for each light source. You can also disable or enable any light source.

The light source angles are always relative to the viewpoint. For example, a horizontal light source angle of 30 degrees will always be 30 degrees to the right of the "viewer," regardless of the view angle.

If more than one light source is enabled, the total amount of light is not increased. It is split up among the enabled light sources.

You can press **F1** at the light source screen for help.

The mouse can be used to move from field to field on the Lights screen, and you can press the left and right mouse buttons to change the light source settings.

After you have the desired light source settings, press **F2** to use the settings, or **F3** to save the settings as DesignCAD 3-D defaults. Pressing **F3** executes the Configuration save command and saves all the system parameters.

To cancel the Lights and restore the original light source settings, press **Esc**.

Tip: You can select **F1**, **F2**, **F3**, or Esc with the mouse as well as by pressing the keys. This is also true for other DesignCAD 3-D commands.

Related Topics: Configuration Save.

Line Command

| | |
|--------------------|--------------|
| Menu: | LINES |
| Menu Command: | Line |
| Short Form: | V |
| Keystroke Command: | V |

Points 1-n: Points for line.

The Line command is used to draw straight lines between points. To use this command, select **Line** from the LINES menu or press the **V** key. Then set the points through which the line is to be drawn and press **Enter**. As you set the points, a "rubber band" line will follow the cursor about the screen to show you where the line will be drawn.

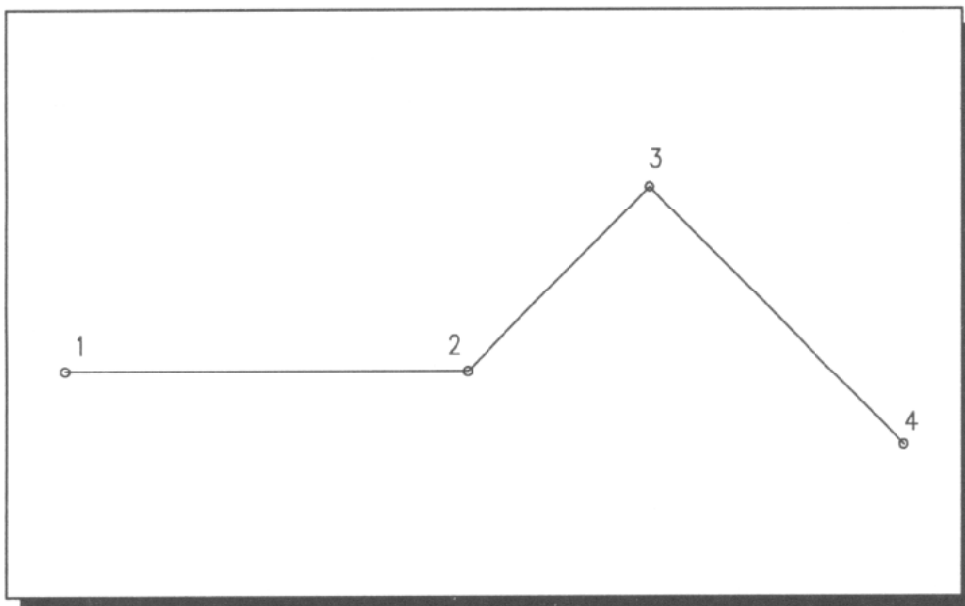


Figure 54 - The Line command draws a line through a set of points.

If you use a mouse, you can set a point in the "Enter Box" at the upper left corner of the screen instead of pressing **Enter** on the keyboard. If your mouse has three buttons, you can press the middle button for **Enter**.

After the points have been set, a line will be drawn through the points beginning at the first point and continuing to the last point.

Example: Draw a horizontal line. Set a point at the current cursor location. Set a second point to the right of the first point. Press the **V** key. A line connecting the points will be drawn.

Tip: You can set the points either before or after issuing the Line command. If you set points first, you do not have to press **Enter** when you are finished setting points. If you set the points after issuing the command, a line of instructions for setting points is displayed. This is also true for other DesignCAD 3-D commands.

Related Topics: Orthogonal Line.

Line Distance Command

| | |
|---------------|----------------------|
| Menu: | INFO |
| Menu Command: | Line Distance |

Point 1: Point on one line.
Point 2: Point on a second line.

The Line Distance command calculates the distance between two lines, at their nearest points. The distance is displayed on the screen.

Two points are used with this command: a point on each line to be measured. This command will determine point on each line nearest the other line, and will calculate the distance between those points. This can be done even if the lines do not lie on the same plane.

Example: Calculate the distance between two lines in 3-D space.

Use the Point XYZ command and the Line command to draw two lines. Draw the first line between points (0,0,0) and (10,10,10). Draw the second line between points (10,0,0) and (0,10,20). Use the Line Distance command to calculate the distance between the line, setting a point on each line. The distance will be displayed on the screen.

Related Topics: Dimension Line.

Line Plane Command

| | |
|---------------|-------------------|
| Menu: | POINTS |
| Menu Command: | Line Plane |
| Short Form: | LP |

Point 1: Set a point on the intersecting line.

Point 2: Set a point on the intersecting plane.

The Line Plane command is used to set a point at the intersection of a line and a plane. To use the command, select **Line Plane** from the POINTS menu. Then set a point on the line and a second point on the plane. The cursor will then move to the intersection of the line and the plane and a point will be set there.

This command temporarily erases any previously set points and replaces them after the command is complete. For example, you might have three points set and want to set a fourth point at an intersection. After **Line Plane** is entered, the original 3 points are erased. You then set two points to designate the intersection. The two points just set are erased, the original 3 points are brought back, and the fourth point is set at the intersection of the line and the plane.

This command does not allow the points to be set before the command is entered.

Example: Find the intersection of a line and a plane. Draw a 2-dimensional box on the X-Z plane. Draw a line passing through the box. Select **Line Plane** from the POINTS menu. Then set a point on the line and a second point on the plane. The cursor will move to the intersection of the two objects, and those two points will be replaced by a point at the intersection.

Related Topics: Intersect, Plane Snap.

Line Snap Command

| | |
|--------------------|------------------|
| Menu: | POINTS |
| Menu Command: | Line Snap |
| Short Form: | LS |
| Keystroke Command: | K |

Points: None.

The Line Snap command moves the cursor to the nearest line or entity and sets a point there. To use the command, select **Line Snap** from the POINTS menu or press **K**. The cursor will move to the nearest location on the nearest line and a point will be set there.

When the Line Snap command is selected from the menu with a mouse, you will be asked to set a point near the line to be snapped to. The point used to select the line will be removed, and other points set will remain. Essentially, the point set for this command will be moved to the nearest point on the line.

There are two snap modes that can be used with the Line Snap command (and other Gravity commands): 3-D Snap and 2-D Snap. The 2-D/3-D point snap can be set in the Drawing Parameter command. The 3-D snap is the normal, 3-dimensional Gravity function - the cursor moves to the nearest line in 3-D space. This may not be the nearest line to the cursor on the 2-dimensional screen, however.

With the 2-D snap, the cursor moves to the nearest line on the screen 2-dimensionally. With the 2-D snap, the cursor moves the shortest possible distance on the screen. The actual distance moved in 3-D space may be considerable, however, because the cursor position in 3-D space may be very different from the nearest line on the 2-D screen.

Tip: You can use the Line Snap keystroke command (**K** key) to set a point on a line during another command.

Related Topics: Plane Snap, Gravity Point, Drawing Parameters.

Macro Define Command

| | |
|---------------|----------------|
| Menu: | FILES |
| Menu Command: | Mac Def |
| Short Form: | MD |

Points: None.

A Macro is a set of keystrokes that can be saved and later used as a single command. This is a very powerful DesignCAD feature.

The Macro Define command is used to create a Macro. To create a Macro, select **Macro Define** from the FILES menu. You will then be asked for the Macro name. A Macro name can be 1 to 8 characters and may contain letters and numbers only.

After you enter the Macro name, you will return to the drawing screen. The word **MACRO** will appear on the status line at the bottom of the screen while the Macro is being created. Every keystroke entered will become part of the Macro.

The macro consists only of keystrokes. Any movement of the cursor from a mouse or digitizer will not be saved with the macro. You should enter commands at the command line and not from the pull-down menus.

After you have finished building the Macro, press the **&** key to save it. Pressing **&** is the only way to finish making a macro. It can be pressed at any time, even within a command or a menu.

There are 12 special names that can be used with Macros so that the macros can be executed by pressing a single key. Macros **MACRO1** through **MACRO12** can be executed by pressing **^F1** through **^F12** (**^F1** means to hold the **Ctrl** key down and press **F1**.) For example, if you name a macro **MACRO3**, you could use it by pressing **^F3**.

A macro can be created called **AUTOEXEC** that is run at the beginning of every DesignCAD 3-D session. This macro can be used to perform tasks necessary for your specific application.

The macro execution begins at the last point set. If no points have been set, then the macro execution begins at the current cursor position.

You can prompt the user of a macro for points by pressing **^Q** when the points are to be entered. You will then be asked for the prompt, or message. After you enter the prompt, set the points as instructed and continue with the macro. When the macro is run, the prompt or message you entered will be displayed when the points are to be entered.

If a macro uses the cursor keys to move the cursor, then the distance moved when the macro is executed depends on the cursor step size at the time the macro is executed. A more positive way to accomplish cursor movement in a macro is to use the Point XYZ, Point Relative, Move XYZ, and Move Relative commands.

The file extension for macro files is .D3M. For more information on Macros, refer to the section "Using Macros" in this manual.

When you create a macro, you essentially record a set of keystrokes. For more complex and flexible programming, you should use the BasicCAD programming language. For more information on BasicCAD, see the *BasicCAD Reference Manual*.

Example: Make a Macro named BOXMAC to draw a box 1 cursor step wide and 1 cursor step tall. Select the command **Macro Define** and enter the name of the Macro when requested: **BOXMAC**. Then, on the drawing screen, use the arrow keys to draw the box. Press the **"&"** key to save the Macro when the box is finished. The Macro will then be saved to disk. To execute the Macro and draw a box at the current cursor position, use the Macro Execute command.

Tip: An easy way to execute a macro is to enter *&macname* on the command line, where *macname* is the name of the macro.

Related Topics: Macro Execute.

Macro Execute Command

| | |
|---------------|-------------------|
| Menu: | FILES |
| Menu Command: | Macro Exec |
| Short Form: | ME |

Point 1: Relative point for macro execution (optional).

The Macro Exec command is used to execute a Macro. To use the command, select **Macro Execute** from the Files menu or **me** on the command line. Enter the Macro name when requested. That Macro will be performed at the current cursor position.

It is also possible to execute a Macro by entering:

&macname

where *macname* is the name of the Macro. Macros named MACRO1 through MACRO12 can be executed by pressing **^F1** through **^F12**. (**^F1** means to hold the **Ctrl** key down and press **F1**.)

Related Topics: Macro Define.

Make Plane Command

| | |
|---------------|-------------------|
| Menu: | SURFACES |
| Menu Command: | Make Plane |
| Short Form: | MP |

Points 1-n: Set points on the lines to be converted to a plane.

This command is used to make a plane out of one or more connected lines. To use the command, set a point on the line(s) to be converted to a plane. Each line that is combined must be connected - that is, each line selected must share an endpoint with another line that has been selected.

The Make Plane command is useful in making a surface out of lines so the surface can be shaded and so the object containing the surface can be converted to a Solid.

The lines used can be Line, Circle, Curve, or Arc entities. The maximum total number of points in the resulting plane is 200.

Example: Convert a triangle to a plane. Draw three lines in a triangle. Use the Make Plane command, setting a point on each line. The three lines will be converted to a plane.

Related Topics: Plane, Combine Line.

Material Command

| | |
|---------------|-----------------|
| Menu: | SHADE |
| Menu Command: | Material |
| Short Form: | Mat |

Point 1: Set a point on a sample object (optional).

The Material command is used to create and manage material properties for use with shaded objects. For example, you can assign the material "marble" to an object in the drawing, and it will look like marble when it is shaded.

With the Material command, you can create and edit materials, save materials to disk, and read materials from disk. DesignCAD 3-D comes with more than 20 materials that can be used in your drawings.

With DesignCAD 3-D, a different material can be assigned to each layer. For example, you can use Gold for layer 1, Walnut for layer 2, Brass for layer 3, etc. Then you can draw all the Gold items in the drawing on layer 1, all the Walnut on layer 2, and so forth.

There are two parts to every material: the colors and the shading parameters. These two combined make an object look real when it is shaded.

ASSIGNING A MATERIAL TO A LAYER

The most common function of the Material command is loading a material from disk and assigning it to a layer in the drawing. To do this, select **Material** from the SHADE menu.

In the Material command, Press **F3** to load a material from disk. You will be able to select the material from a directory listing. When the material is loaded, the colors and shading parameters will be shown on the screen.

Press **F2** to assign that material to a layer in the drawing. You will be asked which layer and color to assign the material properties to. The layer will have the color information associated with it. Whenever

you change to that layer, the material's color will become the current color. The color for the material must be assigned to color 1-15. This may cause a change in the palette of your display. Entities already drawn in the color will appear in the new color.

Suppose, for example, that you have part of your drawing in color 4, which is normally Blue. Then you load the material Gold and assign it to layer 3 and color 4. The blue lines in your drawing will become gold. Similarly, the objects in the drawing already in layer 3 will take on the shading properties of the Gold material.

Ideally, you will have objects in your drawing (either before or after loading the material) that are in layer 3 and color 4. These objects will look like gold when they are shaded.

CREATING AND EDITING MATERIALS

The Material screen is similar to the Color Edit screen with the addition of the shading parameters. To create or edit a material, first adjust the color parameters. The color parameters are located at the upper right of the screen. You can select these items with the mouse or the **Tab** key. Changes to the color parameters are reflected immediately in the "test globe." You can press the **Space** bar to shade the test globe.

You can "bring up" a material that is already being used in the drawing by setting a point on that object before issuing the Material Command. The material properties will then be initialized to those of that object.

You can press **C** at any time to return to the original material properties and "undo" the changes made. This can be very useful if the colors or shading properties get too far off.

Color Selection Box

The basic color for the material can be selected from the large group of colors on the upper left of the screen using the mouse or the keyboard. To select a color, move the cursor to a color and press the left mouse button twice (or the **Enter** key). If you do not have a mouse, use the **Tab** key to move from one box on the screen to the next.

Red, Green, Blue, Intensity

The Red, Green, Blue, and Intensity values can be modified by selecting those boxes on the upper right with the mouse or **Tab** key. Press **+** or **-** (or click on them with the mouse), or enter the number to change one of the values.

The Red, Green, and Blue values change the hue of the color, as you might guess, by affecting the amount of Red, Green, and Blue in the color. The Intensity value changes the overall brightness of the color. These values can range from 0 to 100.

Contrast

The Contrast box (below the Intensity box) is the difference between the brightest shading color and the darkest. It determines the amount of contrast when an object of this color is shaded. At the lower right of the large color selection area are the shading colors. The bright colors are on top, gradually changing to dark colors at the bottom. This box changes as the Contrast value changes.

Wireframe Color

The Wireframe box contains the color that will be used on the screen for wireframe drawing - lines that are not shaded. This can be any color in the box of shading colors. To make the wireframe color brighter, press **+** (or click on the **+** with the mouse). To make the wireframe color darker, use **-**.

The Color Globe

The color globe shows the current wireframe color. You can shade the globe to see how shaded objects will look by pressing the **Space** bar.

THE SHADING PARAMETERS

The shading parameters are located below the color parameters, at the right of the screen. You can select these items using the mouse or the **Tab** key. To view the effects of the shading parameters, press the **Space** bar to shade the test globe.

Background Light

This determines the brightness of "dark" or shadowy areas of a shaded object. If this value is too high, the shaded object will have little or no contrast.

Diffuse Reflection

This is the amount of normal or diffuse reflection from an object. Diffuse reflection is the type of reflection you get from flat latex wall paint. It is not shiny.

Specular Reflection

This is the amount of specular reflection. Specular reflection is the type of reflection you get from a shiny object. When the specular reflection value is high, you can see the reflection of the light source as a spot of light in the test globe.

Specular Contrast

The specular contrast is essentially how shiny the object is. The specular reflection is the amount of light reflected. The specular contrast is the directness of the reflection. With a specular contrast of zero, specular reflection is the same as diffuse reflection.

Texture Type

There are six textures with DesignCAD 3-D:

1. None, or smooth.
2. Marble
3. Grain
4. Wood
5. Cement
6. Patch

These simulate the textures of real objects. For example, an object with Wood texture actually has wood grain, like real wood.

Texture Index

The texture index defines the amount and the size of the texturing. The precise effect depends on the texture and on the size of the object in the drawing. Use the test globe to see the effect of the texture index.

WHEN YOU ARE FINISHED

After you have finished creating or editing a material, you can do the following:

1. Press **F2** to assign the new material to a layer and color in the drawing.
2. Press **F4** to save the material to disk as a material (.MAT) file.
3. Press **Esc** to cancel the changes and return to the DesignCAD screen.

You can also select these functions with the mouse or **Tab** key.

Materials in your Drawing

The material properties assigned to layers and colors of your drawing will be automatically saved with the drawing. Next time the drawing is loaded, those material properties will be used.

Using the Mouse

With the Material command, you can use the mouse to select any of the boxes, including the Function key boxes. You can change a value by clicking on the + or - to the right of the value.

Help!

At any time in the Materials command, you can press **F1** for help.

Tip: The Material command uses a minimum of 256 colors. On displays with fewer colors than 256, the materials can be used, but only the shading properties of the materials will affect the shaded output.

Related Topics: Color Edit, Layer Command, Shade.

Memory Command

| | |
|---------------|---------------|
| Menu: | INFO |
| Menu Command: | Memory |
| Short Form: | Mem |

Points: None.

This command is used to display the amount of memory used by the drawing. To determine the amount of memory used, select **Memory**. The amount of memory used by the drawing (a percentage) will then be displayed on the screen.

Midpoint Command

| | |
|---------------|-----------------|
| Menu: | POINTS |
| Menu Command: | Midpoint |
| Short Form: | Mid |

Point 1: Set a point near the line.

The Midpoint command moves the cursor to the center of the nearest line and sets a point there. To use the command, select **Midpoint**. The cursor will then move to the midpoint of the nearest line or arc and a point will be set there.

If you use a mouse to select this command from the menu, you will be asked to set a point near the line. Otherwise, the cursor will move to the midpoint of the nearest line.

There are two snap modes that can be used with the Midpoint command (and other Gravity commands): 3-D Snap and 2-D Snap. The 2-D/3-D point snap can be set in the Drawing Parameter command. The 3-D snap is the normal, 3-dimensional Gravity function - the cursor moves to the nearest line in 3-D space. This may not be the nearest line to the cursor on the 2-dimensional screen, however.

With the 2-D snap, the cursor moves to the nearest line on the screen 2-dimensionally. With the 2-D snap, the cursor moves the shortest possible distance on the screen. The actual distance moved in 3-D space may be considerable, however, because the cursor position in 3-D space may be very different from the nearest line on the 2-D screen.

Related Topics: Line Snap, Drawing Parameters

Move Polar Command

| | |
|---------------|-------------------|
| Menu: | CURSOR |
| Menu Command: | Move Polar |
| Points: | None. |

This command is used to move the cursor by specifying the distance and angle (polar coordinates) of the new location from the last point set. To use the command, select **Move Polar** from the **CURSOR** menu.

You will be asked which plane you want to use. If the angle you are going to enter lies on the X-Y (horizontal-vertical) plane, enter 1. Enter 2 for the Z-Y plane, or 3 for the X-Z plane.

You will then be asked for the distance and angle from the last point set to the new location. Enter the distance in the current Drawing Units and the angle in degrees. The cursor will be moved to the new location; at that distance and angle from the last point set. If no points have been set, the distances will be calculated from the current cursor position.

The distance can be entered using feet and inches by separating the feet and inches with the apostrophe (') key. The angle can be entered using degrees, minutes, and second by separating the degrees and minutes with letter D and separating minutes and second with the letter M.

In addition, the distance can be entered as a mathematical expression enclosed in brackets. For example, the following could be entered for the distance in this command:

[2/3]
[SQRT(2)]
[SIN(45)]

The expression can contain mathematical functions as well as the following operators: +, -, *, /, and ^. (^ is the operator to raise a

number to a power. 2^3 is equal to 8.) The valid mathematical functions are the same as those listed in the *BasicCAD Reference Manual*.

This command is like the Point Polar command except a point is not set at the new cursor location.

Tip: The Move XYZ, Move Polar, and Move Relative commands can be used in Macros and BasicCAD programs to move the cursor to a specific location.

Related Topics: Move Relative, Move XYZ, Point Polar.

Move Relative Command

| | |
|---------------|----------------------|
| Menu: | CURSOR |
| Menu Command: | Move Relative |
| Short Form: | MoveRel |

Points: None.

The **Move Relative** command is used to move the cursor relative to the last point set. The new position for the cursor is given in horizontal (X), vertical (Y), and depth (Z) distances from the last point set. If no points have been set, the distances will be calculated from the current cursor position.

To use this command, select **Move Relative** from the **CURSOR** menu. You will then be asked for the relative X, Y, and Z offsets. Enter the three values separated by a space or a comma. The relative X offset is the horizontal distance from the cursor to the new location. The relative Y offset is the vertical distance from the cursor to the new location. Use negative numbers to move left on the X axis, down on the Y axis, or toward the front on the Z axis.

After these values are entered, the cursor will move to the new location.

You can omit an X, Y, or Z value if you want the current value to remain the same. For example, to move to 5 along the X axis and 10 along the Y axis, enter "**5, 10**" or "**5 10**". The Z position will remain the same. To move along the Z axis a distance of 45, you can enter "**„45**". The X and Y positions will remain the same in this case.

When you are asked to enter the offsets, you can enter a formula or expression in brackets. For example, the following could be entered for the X,Y,Z offsets in this command:

[2/3, 22/7, 6]
[SQRT(2)], 7, 0
[SIN(45)], [COS(45)], [SIN(30)]

The brackets can enclose each expression, or a single pair of brackets can enclose all three values.

The expression can contain mathematical functions as well as the following operators: +, -, *, /, and ^ . (^ is the operator to raise a number to a power. 2^3 is equal to 8.) The valid mathematical functions are the same as those listed in the *BasicCAD Reference Manual*.

Tip: The Move XYZ, Move Polar, and Move Relative commands can be used in Macros and BasicCAD programs to move the cursor to a specific location.

Related Topics: Move Polar, Move XYZ, Point Relative.

Move XYZ Command

| | |
|---------------|-----------------|
| Menu: | CURSOR |
| Menu Command: | Move XYZ |

Points: None.

This command is used to move the cursor by specifying the X, Y, and Z coordinates of the new location. To use the command, select **Move XYZ** from the CURSOR menu. You will be asked for the X, Y, and Z coordinates of the new location. Enter the three values separated by a comma or space. The cursor will then be moved to that location.

The X coordinate is the horizontal location of new cursor location. The Y coordinate is the vertical location. The Z coordinate is the depth of new cursor location. The coordinates are in Drawing Units. The X-Y-Z position of the cursor can be displayed by selecting format 3 with the Status Line Format command or the Drawing Parameter command.

You can omit an X, Y, or Z value if you want the current value to remain the same. For example, to move to X,Y location 24,40 you can enter **24, 40** or **24 40**. The Z position will remain the same. To move along the Z axis to location 45, you can enter **„45**. The X and Y positions will remain the same.

When you are asked to enter the coordinates, you can enter a formula or expression in brackets. For example, the following could be entered for the X, Y, Z coordinates in this command:

[2/3, 22/7, 6]
[SQRT(2)], 7, 0
[SIN(45)], [COS(45)], [SIN(30)]

The brackets can enclose each expression, or a single pair of brackets can enclose all three values.

The expression can contain mathematical functions as well as the following operators: +, -, *, /, and ^ . (^ is the operator to raise a

number to a power. 2^3 is equal to 8.) The valid mathematical functions are the same as those listed in the *BasicCAD Reference Manual*.

Tip: The Move XYZ, Move Polar, and Move Relative commands can be used in Macros and BasicCAD programs to move the cursor to a specific location.

Related Topics: Point XYZ, Move Relative, Move Polar.

Origin Command

| | |
|---------------|---------------|
| Menu: | CURSOR |
| Menu Command: | Origin |

Point 1: New location for the origin.

This command is used to move the origin to another location. The origin is the location of coordinate 0,0,0. It lies in the center of the screen before any zooms, pans, rotates, etc. are performed.

To use the command, set a point at the location for the new origin and select **Origin** from the **CURSOR** menu. Until it is changed again, the origin will be at that point.

Zoom, Rotate, and Pan may change the location of the origin on the screen, but it will remain at the same point on the drawing unless it is changed with the origin command.

Example: Move the origin .3" to the left. Select the Origin command from the **CURSOR** menu. Use Point XYZ command (":" key) to set a point at location (-.3, 0, 0) when asked. The origin of the drawing will be moved, and the X coordinate of every point in the drawing will increase by 3.

Tip: It can be helpful to set the origin at a significant point in the drawing and use status line format 3 to display the current cursor location.

Related Topics: Point XYZ, Home.

Ortho Line Command

| | |
|--------------------|-------------------|
| Menu: | LINES |
| Menu Command: | Ortho Line |
| Short Form: | OL |
| Keystroke Command: | ^V |

Points 1-n: Points for the line.

The Ortho Line command is used to draw a line exactly horizontally, vertically, or front-to-back (parallel to the X, Y, or Z axis). The command is similar to the Line command - press **^V** and set two or more points. A line will be drawn through the points, but at each point the line will be adjusted so that it is parallel to the X, Y, or Z axis, whichever is closer.

This command is very useful when vertical and horizontal lines are being drawn with a mouse or digitizer.

Example: Draw a horizontal line. Using the Ortho Line command, set a point and move the cursor to the right. Even if the cursor varies a little from horizontal, the "rubber band line" will be displayed horizontally. Set another point for the right endpoint of the line and press **Enter**. The rubber band line will become a line in the drawing.

Tip: You can move the cursor horizontally, vertically, or along the Z axis with the mouse or digitizer by holding down the **Shift**, **Ctrl**, or **Shift+Ctrl** keys.

Related Topics: Line.

Pan Command

Menu: **DISPLAY**
Menu Command: **Pan**

Points: None.

The Pan command is used to move or pan the drawing across the screen. To use the command, enter **Pan** from the DISPLAY menu. You will be asked for the distance, plane, and direction of the pan.

Enter the distance you want the drawing to be moved, in Drawing Units. Then enter plane (X-Y, Z-Y, or X-Z) along which the drawing is to be moved and the angle of movement on that plane. The drawing will be moved accordingly.

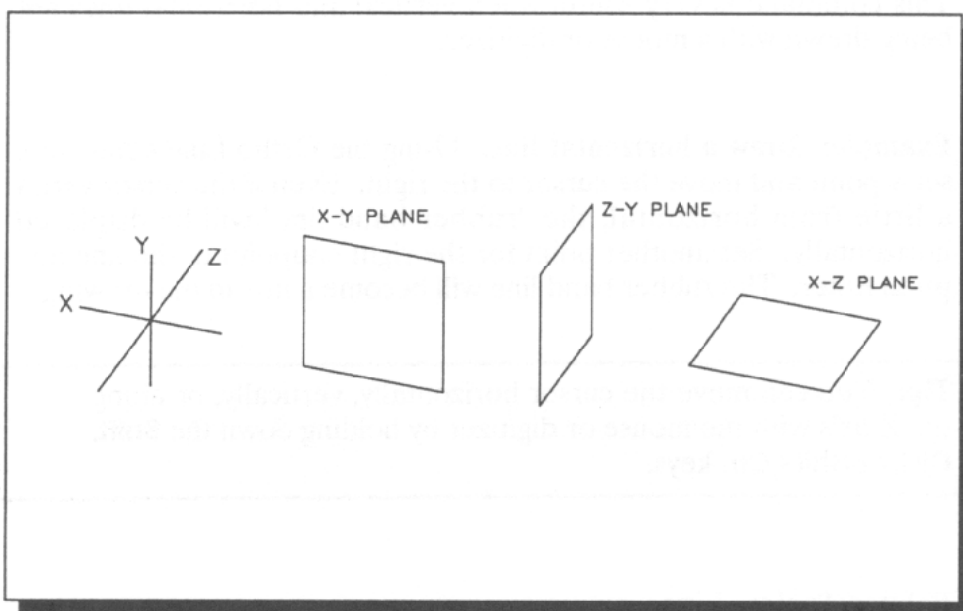


Figure 55 - Planes for the Pan command.

Tip: You can use two points ("from" and "to") with the Zoom command to perform a pan. To leave the drawing the same size, use a zoom factor of 1.

Related Topics: Rotate, View, Zoom.

PARALLEL

| | |
|---------------|-----------------|
| Menu: | LINES |
| Menu Command: | Parallel |
| Short Form: | Para |
| Keystroke: | = |

Point 1: Set point on original line.

Point 2-n: Locations for parallel lines.

The Parallel command enables you to draw a line or lines parallel to any other line, curve, circle, arc, ellipse, or elliptical arc.

To use the command, select **Parallel** from the LINES menu (or press the = key). Set a point on the line to which a parallel is to be drawn and a point at the location for each parallel line. A line parallel to the first line will then be drawn at the second point and each succeeding point.

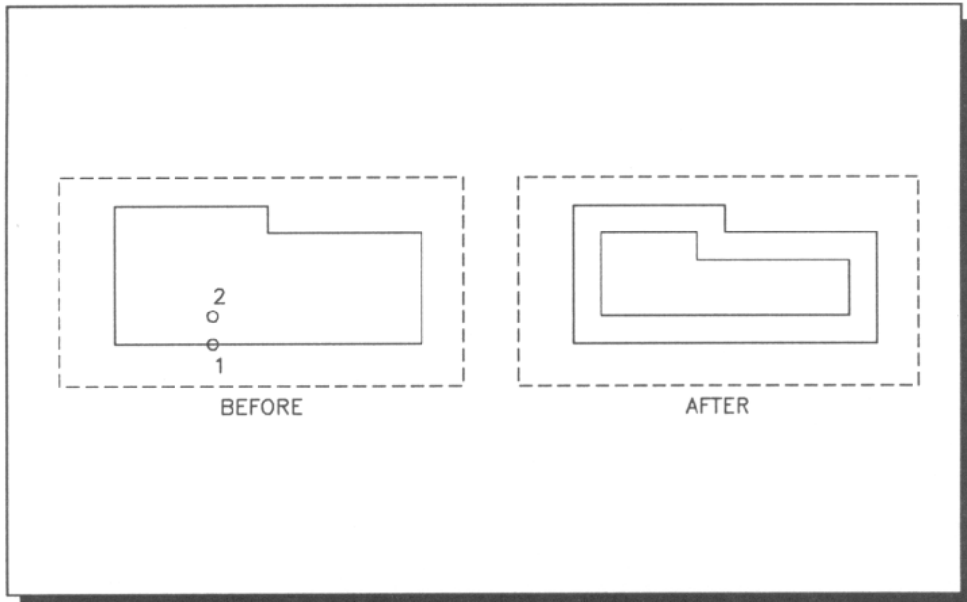


Figure 56 - The Parallel command is used to make a parallel line to an existing line in the drawing.

Example: Draw a line parallel to another line. Draw a diagonal line. Set a point on the line and another point a short distance away from the line. Then press the = key. A line parallel to the original line will be drawn.

Patch Command

| | |
|---------------|-----------------|
| Menu: | SURFACES |
| Menu Command: | Patch |
| Short Form: | PA |

Points 1-4 or 1-3: Set a point on each of the sides to patch.

The Patch command is used to fit a surface over a "frame" of three or four connected lines.

The patch command requires 3 or 4 points, one on each side of the patch "frame." The lines must be connected, but they need not lie in the same plane. The lines can be Line, Curve, Circle, or Arc entities.

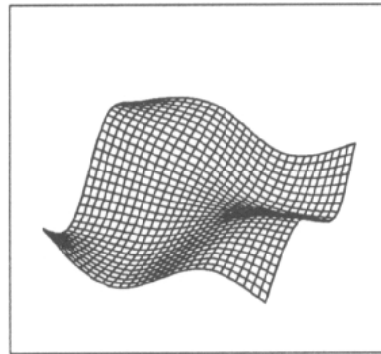
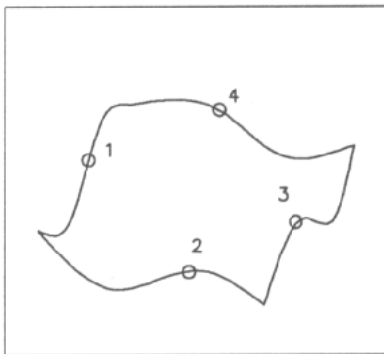


Figure 57 - Set points on the lines to make up the edges of the patch.

After setting points, you will be asked for the "Number of Planes Across and Down." "Across" and "Down" are relative terms. The line selected by the first point is assumed to be the top of the patch. Enter the number of planes or facets used to make up the patch. The more planes you select, the more precise and smooth the patch will be, but more planes also use more memory.

A smooth surface will be "stretched" between the 3 or 4 lines.

This command makes it easy to construct a surface over an area that would be very difficult to describe otherwise.

Related Topics: Connect.

Perpendicular Command

| | |
|---------------|---------------------|
| Menu: | LINES |
| Menu Command: | Perpendiculr |
| Short Form: | Perp |

Point 1: Point from which the perpendicular line is drawn.

Point 2: Point on an existing line.

The Perpendicular command is used to draw a line perpendicular to another existing line.

To use the command, select **Perpendiculr** from the **LINES** menu and set two points. The first point is the point where the perpendicular line is to start. Set the second point on the existing line to which the perpendicular is to be drawn.

A new line will be drawn from the first point to the line designated by the second point, and will be perpendicular to that line.

Example: Draw a line perpendicular to another line. Draw a diagonal line. Use the Perpendicular command and set a point above and to right of the center of the diagonal line. Set a second point on (or near) the original line. A perpendicular line will be drawn from the first point to the original line.

Related Topics: Parallel.

Perpendicular Plane Command

| | |
|---------------|----------------------|
| Menu: | SURFACES |
| Menu Command: | Perpend Plane |
| Short Form: | PPlane |

Point 1: Line to which a perpendicular plane will be drawn.

Point 2: Location on the line for the plane.

This command can be used to draw a plane perpendicular to a line in the drawing.

To use the command, select **Perpend Plane** from the **SURFACES** menu. Set one point on the existing line and a second point where the plane is to be located on that line. You will be asked to enter the width of the plane.

A square plane will be drawn through the second point, perpendicular to the original line. The plane will be centered about the original line.

The line selected can be a line, curve, circle, or arc.

Example: Draw a plane perpendicular to a line on the midpoint of the line. Draw a diagonal line. Use the midpoint command to set two points at the midpoint of the line. Then select **Perpend Plane** from the **SURFACES** menu. Enter 5 when asked for the plane width. A square plane 5 drawing units wide will be drawn through the midpoint of the line, perpendicular to the line.

Related Topics: Perpendicular.

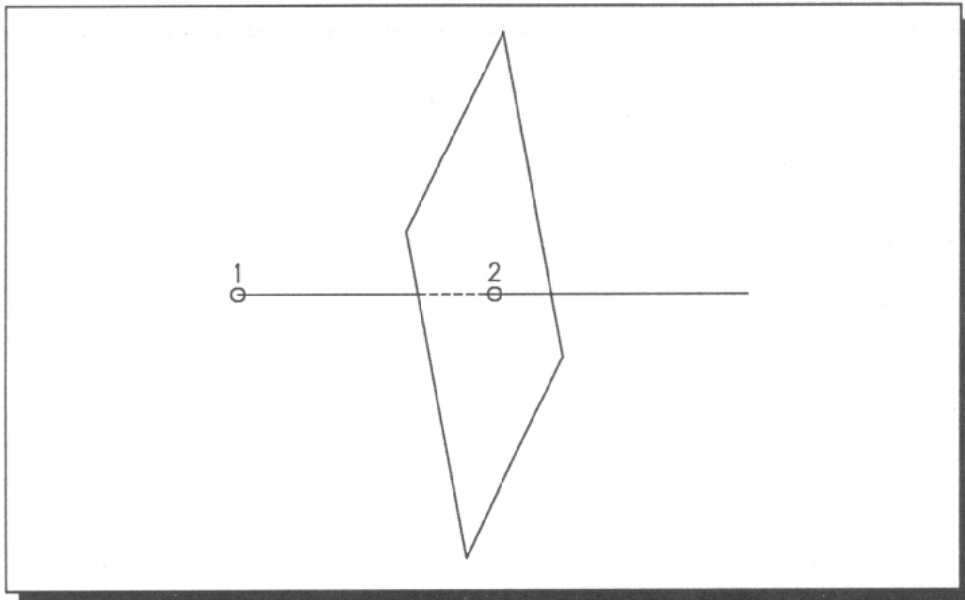


Figure 58 - With the Perpendicular Plane command, the first point selects the line and the second point determines the location for the plane.

Plane Command

| | |
|--------------------|-----------------|
| Menu: | SURFACES |
| Menu Command: | Plane |
| Short Form: | P |
| Keystroke Command: | P |

Points 3-n: Points around the edge of the plane.

The Plane command is used to draw a plane or surface in the drawing. A Plane entity is similar to a line, except that it must lie on a single plane. A plane will obscure or hide lines it is in front of after hidden line removal or shading, while a line will not.

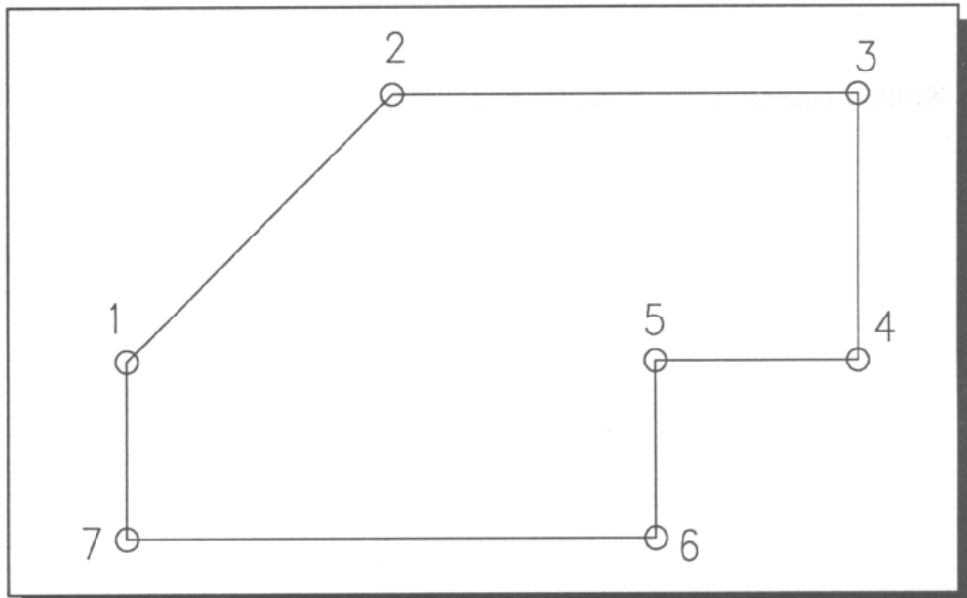


Figure 59 - To create a plane, set points around the edge of the plane.

To draw a Plane, select **Plane** from the SURFACES menu (or press **P**) and set 3 or more points for the plane. A line will be drawn through the points, and the last point will be connected to the first. All the points must lie on a single plane. This does not mean that all points must lie on the X-Y plane, however. It means that the points must represent a flat surface.

The Surface commands (Patch, Connect, Connect Curve, and Connect Smooth) and the Extrude commands (Extrude, Extrude Varying, Sweep, and Spiral) can be used to construct non-flat surfaces.

Example: Draw a triangular plane. Set three points in a triangle and press **P**. A plane will be drawn with a corner at each of the three points. Enter **shade** from the command line. A plane looks like a line entity until shading or hidden line removal is performed.

Related Topics: Line, Polygon, Box.

Plane Snap Command

| | |
|---------------|-------------------|
| Menu: | POINTS |
| Menu Command: | Plane Snap |
| Short Form: | PS |
| Points: | None. |

The Plane Snap command moves the cursor to the nearest location on the nearest surface and sets a point there.

When the Plane Snap command is entered from the keyboard, the plane nearest to the cursor will be selected.

When the Plane Snap command is selected with a mouse, you will be asked to set a point near the plane to be snapped to. The point used set to select the plane will be removed, and other points set will remain. Essentially, the point set for this command will be moved to the nearest location on the nearest plane.

Related Topics: Line Snap, Gravity Point.

Plane Subtract Command

Menu: **SURFACES**
Menu Command: **Plane Sub**
Short Form: **Psub**

Point 1: Plane to subtract.
Point 2: Plane to be subtracted from.

The Plane Subtract command subtracts one plane from another. It can be used to make a hole or opening of a specific shape in a plane. To use this command, set a point on the plane to be subtracted and a second point on the plane from which the first plane is to be subtracted.

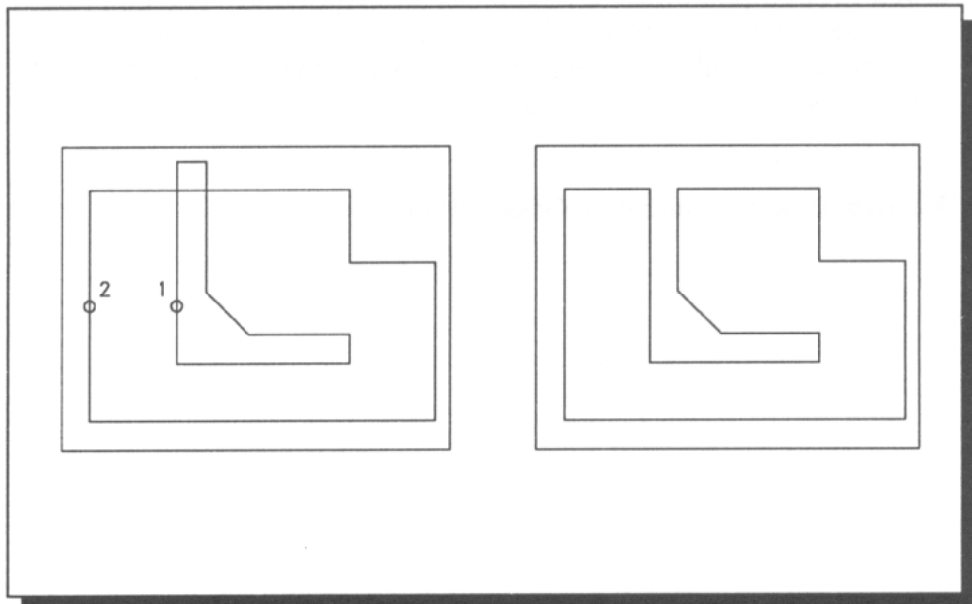


Figure 60 - The Plane Subtract command is used to subtract one plane from another.

The first plane will then be subtracted from the second. The two planes need not lie on top of one another. If the first plane is not parallel with the second, then the second plane will be projected onto the first before it is subtracted.

Example: Cut a round hole in a square box. Use the Box command to draw a 2-dimensional box. Use the Disc command to draw a round plane inside the square plane. Use the Plane Subtract command to subtract the round plane from the box, setting 1 point on the round plane and a second point on the box. A hole will be cut in the box. Shade the drawing to verify this.

Related Topics: Cut Plane, Drill.

Plot Command

| | |
|--------------------|--------------|
| Menu: | FILES |
| Menu Command: | Plot |
| Keystroke Command: | F6 |

Points: None.

This command outputs the current drawing to the plotter. To use the command, select **Plot** from the **FILES** menu or press the **F6** key. You will be asked for the following information:

Drawing Length**Drawing Height**

You can specify the precise drawing length and height at which the drawing is to be plotted by entering those values here.

Drawing Units per Inch

To output your drawing to scale, enter a number other than zero. For example, if you want 1/4" to be equal to one foot, enter 4 Drawing Units per Inch. If the view being plotted has very much perspective (a view distance of 50 or less), the scale of the drawing will not be very accurate.

Paper Length (?-Select)**Paper Height (?-Select)**

This is the size of the plotter paper. Press **?** to select the paper size from a list of standard paper sizes.

Center Plot on Paper

Your drawing can be centered on the paper or it can be plotted in the lower left corner.

Hidden Line Removal?**Shading?**

The drawing can be output to the plotter with hidden line removal or shading. It takes longer to plot a complex drawing with hidden line removal or shading enabled, but the output can be more effective.

Plot Which View?

While View 1 is the view that is normally plotted, you can plot any of the 4 views. This makes it easy to plot a front, top, or side view of the drawing.

Plot to Disk?

You can send the plot file to disk instead of the plotter. This makes it convenient to produce plots if the plotter is on another computer. It can also be used to transfer drawings to other graphics software.

Special Plotter Commands

If you want a particular command output to the plotter, you may enter it here. For example, if you are plotting on transparencies with the HP 7550 plotter, you may want to use the slower pen velocity. If so, enter that plotter command: **VS 5**. (Commands for your particular plotter may be found in your plotter manual.)

There are some options at the bottom of the Plot screen. To select these options, press the function key (**F1** through **F5**) or click on the option with the mouse.

F1 - Help

F1 can be used to display help.

F2 - Continue with the Plot

Press F2 when you are ready to plot the drawing.

F3 - Save the Plot parameters as Defaults

This function saves all the plot parameters as the DesignCAD 3-D default plot settings. The initial settings of the Plot command will be set to those in effect at the time this function is selected.

F4 - Color Pen Assignment

You can assign any of DesignCAD's 15 basic colors to any of the plotter pens. For example, if the blue plotter pen is pen number 7 and color 4 is blue in the DesignCAD drawing, you can assign color 4 to pen number 7.

F5 - Set Paper Margin

Plotters generally cannot plot all the way to the edge of the paper. The margin is the area around the outside of the paper that cannot be addressed by the plotter. You can adjust the margin on your plotter if necessary. If the margin is too small, some lines will be "cut off" around the edge of the paper. If the margin is too large, the full size of the paper will not be used.

To specify the type of plotter you use, run the program SETUP3D. This program needs to be run only once.

Tip: If you have trouble getting your plotter to plot, check Appendix G in this manual titled "Hardware Interfacing."

Tip: If you do not have a plotter, you can use the configuration program SETUP3D to select a plotter and output only to disk. You can then output HPGL or other plot files to other software.

Related Topics: Print command.

Point XYZ Command

| | |
|--------------------|----------------------|
| Menu: | POINTS |
| Menu Command: | Point XYZ |
| Short Form: | PXYZ |
| Keystroke Command: | : (colon key) |

Points: None.

This command is used to set a point by specifying the X, Y, and Z coordinates of that point. To use the command, select **Point XYZ** from the POINTS menu or press the ":" key. You will be asked for the X, Y, and Z coordinates of the point. Enter the three values separated by a comma or space. A point will then be set at that location.

The X coordinate is the horizontal location of the point. The Y coordinate is the vertical location of the point. The Z coordinate is the depth of the point. The coordinates are in Drawing Units. The X-Y-Z position of the cursor can be displayed by selecting format 3 with the Status Line Format command or the Drawing Parameter command.

You can omit an X, Y, or Z value if you want the current value to remain the same. For example, to move to X,Y location 24,40 you can enter **24, 40** or **24 40**. The Z position will remain the same. To move along the Z axis to location 45, you can enter **„45**. The X and Y positions will remain the same.

When you are asked to enter the coordinates, you can enter a formula or expression in brackets ("[" and "]"). For example, the following could be entered for the X,Y,Z coordinates in this command:

[2/3, 22/7, 6]
[SQRT(2)], 7, 0
[SIN(45)], [COS(45)], [SIN(30)]

The brackets can enclose each expression, or a single pair of brackets can enclose all three values.

The expression can contain mathematical functions as well as the following operators: +, -, *, /, and ^. (^ is the operator to raise a number to a power. 2^3 is equal to 8.) The valid mathematical functions are the same as those listed in the *BasicCAD Reference Manual*.

Example: Set a point at location 2,4,8. Press the : key and enter **2,4,8** when asked for the X,Y,Z coordinates. A point will be set where X=2, Y=4, and Z=8.

Related Topics: Move XYZ, Point Relative, Point Polar.

Point Move Command

| | |
|--------------------|-------------------|
| Menu: | EDIT |
| Menu Command: | Point Move |
| Short Form: | PM |
| Keystroke Command: | * |

Point 1: Set a point on the point to be moved.

Point 2: Set a new location for that point.

Point 3: Set a point on the entity to be affected (optional).

The Point Move command is used to move a point in the drawing. To use the command, set a point directly upon another point in the drawing. The Gravity Point command (. key) can be used for this. Then set another point at the location to which the first point is to be moved. Every entity in the drawing containing the first point will be redrawn using the second point instead.

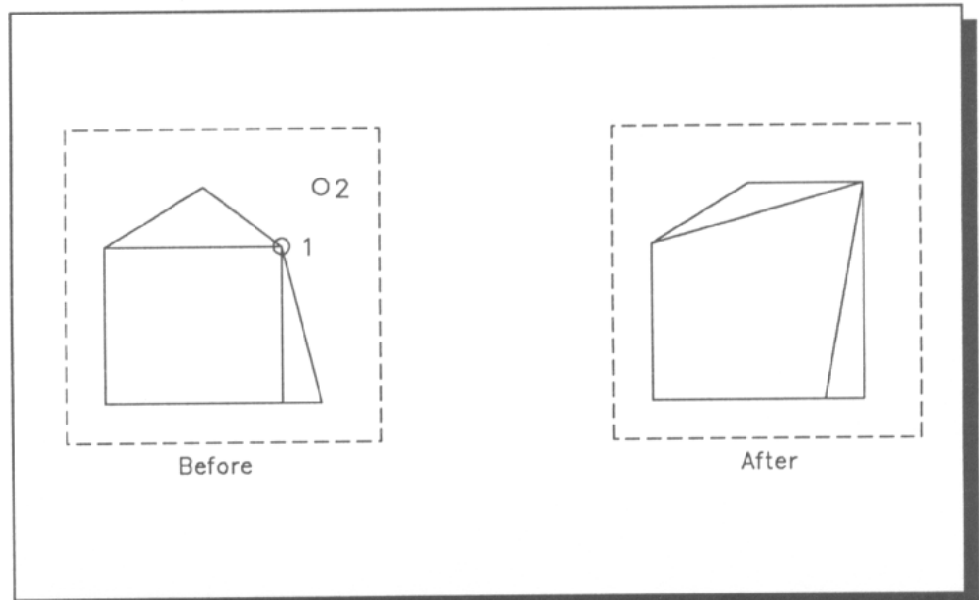


Figure 61 - The Point Move command moves a point in the drawing. Set the first point on the point to be moved, and the second point at the new location for the first point.

If several entities share the same point, you can set a third point on the entity to be changed. This leaves the other entities unchanged and moves only the point belonging to the selected entity.

You can remove a point from an entity using this command by setting two points on the point to be removed. For example, to convert a box into a triangle, set two points on a corner of the box and press *.

Example: Stretch the corner of a 2-dimensional box. Use the Box command to draw a 2-dimensional box. Use the Gravity Point command (. key) to set a point on the upper right corner of the box. Move the cursor up and to the right and set a second point. Then press the * key. The upper right corner of the box will be stretched up to the second point.

Related Topics: Extend, Hammer-1, Hammer-2.

Point Polar Command

| | |
|--------------------|----------------------|
| Menu: | POINTS |
| Menu Command: | Point Polar |
| Short Form: | PP |
| Keystroke Command: | ; (semicolon) |

Points: None.

This command is used to set a point by specifying the distance and angle (polar coordinates) of the new point from the last point set. To use the command, select **Point Polar** from the menu or press the ";" key.

You will be asked which plane you want to use. If the angle you are going to enter lies on the X-Y (horizontal-vertical) plane, enter 1. Enter 2 for the Z-Y plane, and 3 for the X-Z plane.

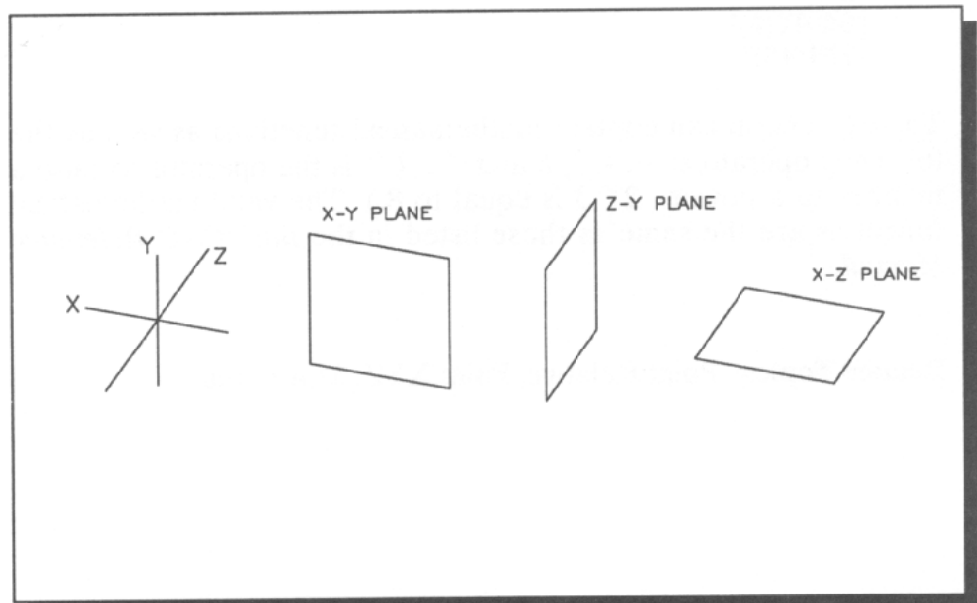


Figure 62 - The X-Y, X-Z, and Y-Z planes.

You will then be asked for the distance and angle from the last point set to the new point. Enter the distance in the current Drawing Units and the angle in degrees. A point will then be set at that distance and angle from the last point set. If no points have been set, the distances will be calculated from the current cursor position.

If this command is selected from the menu with a mouse, you will be asked to set a temporary point from which the distances will be calculated.

The distance can be entered using feet and inches by separating the feet and inches with the apostrophe key ('). The angle can be entered using degrees, minutes, and second by separating the degrees and minutes with letter D, and separating minutes and second with the letter M.

In addition, the distance can be entered as a mathematical expression enclosed in brackets. For example, the following could be entered for the distance in this command:

[2/3]
[SQRT(2)]
[SIN(45)]

The expression can contain mathematical functions as well as the following operators: +, -, *, /, and ^. (^ is the operator to raise a number to a power. 2^3 is equal to 8.) The valid mathematical functions are the same as those listed in the *BasicCAD Reference Manual*.

Related Topics: Point Relative, Point XYZ, Move Polar.

Point Relative Command

| | |
|--------------------|----------------------|
| Menu: | POINTS |
| Menu Command: | Point Relativ |
| Short Form: | PR |
| Keystroke Command: | ' (apostrophe) |

Points: None.

The Point Relative command is used to set a point relative to the last point set. The position of the new point is given in horizontal (X), vertical (Y), and depth (Z) distances from the last point set. If no points have been set, the distances will be calculated from the current cursor position.

If this command is selected from the menu with a mouse, you will be asked to set a temporary point from which the distances will be calculated.

To use this command, select **Point Relativ** from the menu or press the apostrophe key ('). You will be asked for the relative X, Y, and Z offsets. Enter the three values separated by a space or a comma. The relative X offset is the horizontal distance from the cursor to the new point location. The relative Y offset is the vertical distance from the cursor to the new point location. Use negative numbers to move left on the X axis, down on the Y axis, or toward the front on the Z axis.

After these values are entered, the cursor will move to the new point location and a point will be set there.

You can omit an X, Y, or Z value if you want use 0 for the value. For example, to move to 5 along the X axis and 10 along the Y axis, enter **5, 10** or **5 10**. The Z position will remain the same. To move along the Z axis a distance of 45, you can enter **„45**. The X and Y positions will remain the same in this case.

When you are asked to enter the offsets, you can enter a formula or expression in brackets ("[" and "]"). For example, the following could be entered for the X,Y,Z offsets in this command:

```
[2/3, 22/7, 6]  
[SQRT(2)], 7, 0  
[SIN(45)], [COS(45)], [SIN(30)]
```

The brackets can enclose each expression, or a single pair of brackets can enclose all three values.

The expression can contain mathematical functions as well as the following operators: +, -, *, /, and ^. (^ is the operator to raise a number to a power. 2^3 is equal to 8.) The valid mathematical functions are the same as those listed in the *BasicCAD Reference Manual*.

Related Topics: Point Polar, Point XYZ, Move Relative.

Point Relative Cursor Command

| | |
|---------------|------------|
| Menu Command: | None |
| Short Form: | PRC |

Points: None.

This command is identical to the Point Relative command except that the distance is calculated from the current cursor position instead of from the last point set. See the Point Relative command on the previous page.

This command can be executed only by entering **prc** on the command line.

Related Topics: Point Relative.

Polygon Command

| | |
|---------------|-----------------|
| Menu: | SURFACES |
| Menu Command: | Polygon |
| Short Form: | Poly |

Point 1: One endpoint of a side of the polygon.
Point 2: Other endpoint of a side of the polygon.
Point 3: Plane that the Polygon will lie in (optional).

The Polygon command draws a regular polygon, that is, a convex shape in which each side is the same length. To use the command, select **Polygon** from the SURFACES menu. Then set two points to designate one side of the polygon. A third point can be used to determine the plane on which the polygon lies.

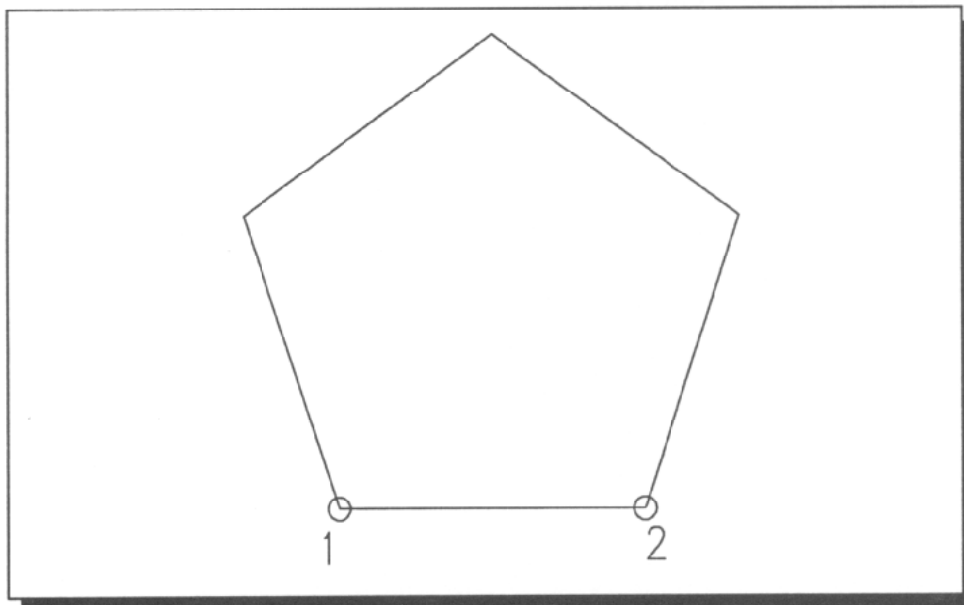


Figure 63 - Set two points for the first side of the polygon, and enter the number of sides. A third point can be used to specify the plane on which the polygon lies.

You will then be asked for the number of sides for the polygon. Enter **3** to draw a triangle, **4** for a square, **5** for a pentagon, etc.

The polygons drawn with this command are Plane entities. This means that a polygon can be shaded or can be extruded into a Solid.

Example: Draw an octagon (a stop sign shape). Select **Polygon** from the SURFACES menu. Set two points, the second one directly to the right of the first. When asked for the number of sides, enter **8**. An octagon will then be drawn using the two points as one of its sides.

Related Topics: Disc, Plane, Box.

Print Command

| | |
|--------------------|--------------|
| Menu: | FILES |
| Menu Command: | Print |
| Keystroke Command: | F5 |

Points: None.

The DesignCAD 3-D Print command is capable of producing very high resolution drawings on ordinary dot matrix printers, laser printers, and ink jet printers. The drawing can be printed at any size or view. A drawing can even be paneled - printed large enough to be broken up and printed in page-size sections.

To use the Print command, select **Print** from the FILES menu or press the **F5** key. You will be asked if you want to proceed with the print command and if you need to save your drawing.

The Print command is actually a separate program from DesignCAD 3-D, so the current drawing must be saved before entering the Print command or it will be lost. After entering the Print command, any drawing on disk may be printed.

The Print program can print the drawing from any view of the drawing, but the default view settings are those of View 1.

When the Print command begins, you will be given the following options:

1. Print a DesignCAD 3-D Drawing.
2. Print Several DesignCAD 3-D Drawings.
3. Display DesignCAD 3-D Drawings on Disk.
4. Specify Layers to be Printed.
5. Terminate the Print Command and Return.
6. Stop the Program.
- (7. Print the drawing in memory: *filename*)

Option 7 is displayed only after you have retrieved a drawing.

Option 1 is used to actually print a drawing. Option 2 can be used to print more than one drawing at a time.

When you select option 1, you are asked for the drawing name. Press **Enter** to select from a list of drawing files.

After the drawing is retrieved, you will be asked for the information below. When option 2 is selected, the information below is entered before the drawings are retrieved, and each drawing is printed using the same set of parameters.

Density

This is the printer resolution: normal, high, or very high. (If your printer does not support multiple resolution or densities, you will not be asked for this.) Higher resolution output looks better, but takes longer to print.

Requested Drawing Units per Inch

This makes it easy to make scale drawings on your printer. For example, if one Drawing Unit represents a foot in your drawing, and you want your output to be scaled at 1 inch = 4 feet, then specify 4 drawing units per inch. Items 3 and 4 will be automatically scaled according to your drawing (provided that the requested scale will fit on your printer).

If you do not want your drawing scaled automatically, enter **0** for the Drawing Units per Inch.

The size of your drawing in drawing units is displayed at the bottom of the screen (unless you are printing multiple drawings). If you have entered a number too small for "Drawing Units per Inch on Output," the drawing will be paneled, or broken up into page-size sections for printing.

If the view being printed has very much perspective (a view distance of 50 or less), the scale of the drawing will not be very accurate.

Printing Area Width

This is the width of the printing area on your printer. It corresponds to the output drawing height, unless the drawing is rotated 90 degrees (see below). The maximum paper width is limited by the printer's capabilities. This is usually 8" for a printer with a 10" carriage or a little over 13" for a printer with a 15" carriage. If you specify a larger drawing size, the drawing will be paneled, or broken up into page-size sections for printing.

Printing Area Length

This is the length of the printing area on your printer. It corresponds to the output drawing width, unless the drawing is rotated 90 degrees (see below). The Printing Area Length can be greater than one page.

Paper Height

Paper Length

The paper size is the size of the paper on the printer. The paper height is actually the width of the printer. It is called the height because a drawing is normally printed in landscape mode, and the paper height will correspond to the printing area height. And, as you might guess, the paper length is actually the height of the paper in the printer.

Rotate Drawing 90 Degrees?

The drawing normally comes out sideways on the page. If you want the drawing to come out with the drawing width the same direction as the printer width, enter **Y**.

Center Drawing?

This option can be used to center the drawing on paper. If this option is enabled, the drawing will be centered based on the paper size specified above.

Character Font

This is the character font to be used in the printer output. You can display the names of the character fonts on your screen by pressing "?" at this field.

Shading Type

Your drawing can be output in wireframe mode with No shading, with **Q**uick shading, or with **S**mooth shading. Enter **N** for no shading, **Q** for quick shading, or **S** for smooth shading.

Shading Pattern

The shading pattern is the size of the pattern used in shading. It can be 1 for fine or 2 for medium density.

Print with Hidden Lines Removed?

This option allows you to print your drawing with hidden lines removed. Printing a complex drawing with hidden line removal takes longer than printing it without hidden line removal.

Number of Copies to be Printed

More than one copy can be printed at a time by specifying the number.

Print to Disk

Your drawing can be output to a disk file instead of the printer. The output file can then be transferred to another computer or to other graphics software. If you print a single file to disk, you will be asked for the file name. If you print multiple files to disk, the drawing name will be used with the extension P01. (If more than one panel is printed, the file extension will be Pnn, where nn is the panel number.)

Print Attributes

Attributes are normally left "invisible" and not printed. You can enable this option for the attributes in your drawing to be printed. See the Attribute command for more information.

The light source and view information can be modified by pressing **F3**. When all the information is correct, press **F2** to begin printing. The Printing process can be terminated at any time by pressing the **Esc** key.

You can press **F1** for help at any time in the Print command.

After a drawing has been printed, it can be printed again by selecting option 7 from the main menu. This saves the time of reading the drawing from disk again. This selection is available only after a drawing has been read from disk.

Paneling

If you need a printout of your drawing larger than the printer, you can do it with the Print command. Enter a Printing Area Width and Length greater than the paper size. A message will be displayed at the bottom of the screen that tells how many panels will be printed.

You can elect to have a frame or corner marks on each panel to aid in the assembly of the pages. You can also select panel numbering to have the panel numbers placed on each panel.

Tip: To produce a Postscript file of a drawing, use the program SETUP3D to select a Postscript printer. Then print to disk. While you can produce a Postscript file with the file conversion program DCFILES, the Print command allows you to produce the Postscript file at any view and with hidden line removal.

Related Topics: Plot.

Quick Shade Command

| | |
|---------------|------------------------------|
| Menu: | SHADE |
| Menu Command: | Quick Shade |
| Menu Command: | Quick Shade 2 |
| Menu Command: | Quick Shade 3 |
| Menu Command: | Quick Shade 4 |
| Short Form: | QSH, QSH2, QSH3, QSH4 |

Point 1: Corner of area to shade (optional).

Point 2: Opposite corner of area to shade (optional).

The Quick Shade commands are used to shade your drawing - that is, to add color to the surfaces in the drawing, resulting in a more realistic image.

Quick Shading is the name for DesignCAD's fast shading. The Shade command produces more realistic and smoother shading, but it takes longer to shade the drawing.

To use this command, select **Quick Shade** from the SHADE menu. The drawing will then be shaded on the screen. DesignCAD 3-D can even shade your drawing on a display with limited or no color capability.

View 1, the large view, is shaded with the Quick Shade command. Views 2, 3, and 4 can be shaded with the Quick Shade 2, Quick Shade 3, and Quick Shade 4 commands.

To shade only a portion of the drawing, set two points in opposite corners of the section on the screen to be shaded. If two points are set with the Quick Shade command, the screen will not be cleared before shading and only the section of the screen between the points will be shaded. The rectangle or section of the screen that is shaded is defined to be left-right, up-down with respect to the screen, not the drawing.

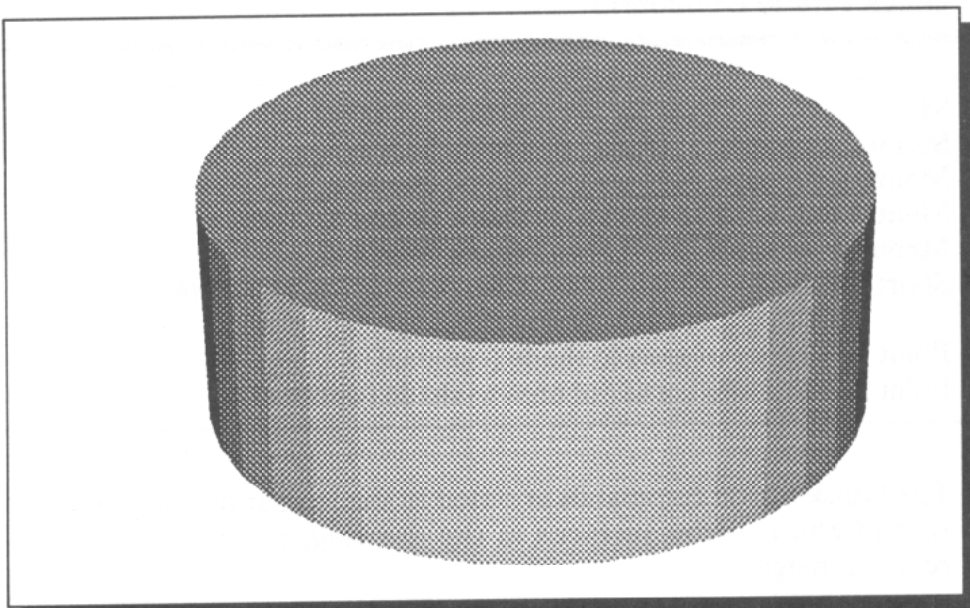


Figure 64 - A Cylinder "Quick Shaded."

One to eight light sources can be enabled and set at any angle for shading (see the Lights command).

The Print and Plot commands can produce shaded drawings on paper. It is not necessary to shade the drawing on the screen before producing a shaded drawing on a printer or plotter.

The Quick Shade command is not affected by the material properties of the Material command. Use the Shade command for realistic shading with materials.

Example: Shade a drawing of a cone. Draw a cone using the Cone command. Then select **Quick Shade** from the SHADE menu to shade the cone.

Related Topics: Shade, Hide.

Retrieve Command

| | |
|--------------------|-----------------|
| Menu: | FILES |
| Menu Command: | Retrieve |
| Short Form: | Load |
| Keystroke Command: | F9 |

Point 1: Location for drawing handle 1 (optional).

Point 2: Location for drawing handle 2 (optional).

Point 3: Plane for drawing handle 3 (optional).

This command is used to retrieve a drawing from disk and display it on the screen. It allows you to specify the location, size, and angle for the drawing.

Up to three handles can be specified when the drawing is saved. If no handle is specified, the front lower left, lower right, and upper right corners of the drawing are used for handles.

With the Retrieve command, you can use up to 3 points. The first point specifies the location of the drawing being retrieved. Handle 1 is positioned at the first point.

The second point specifies the size and angle of the drawing. The second handle of the drawing will be positioned at the second point. If only one point is used, the drawing will be retrieved at the size and angle at which it was saved.

The third point specifies a second angle at which the drawing is to be positioned. If three points are set when the drawing is retrieved, the three handles of the drawing will lie on the same plane as the three points.

If no points are used, the drawing will be retrieved at the same size and location from which it was saved. The relative size, location, and drawing units, which may have been modified by the Zoom or other commands, are taken into consideration when the drawing is retrieved.

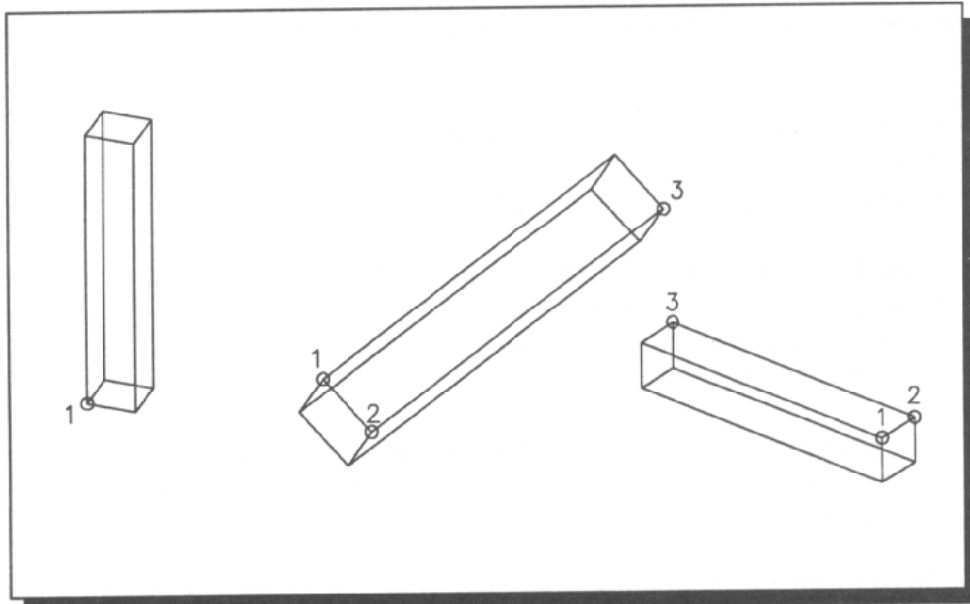


Figure 65 - 1, 2, or 3 points can be used with the Retrieve command to position the drawing. The first two handles of the drawing are located at the first two points, and the third handle is located on the plane defined by the third point.

If 0 or 1 points are used to retrieve a drawing, the drawing will be retrieved at its original size. However, if there is a drawing already on the screen, the size of the drawing being retrieved will be adjusted to match the Drawing Units of the drawing already on the screen.

If the drawing is on a drive or subdirectory other than the one DesignCAD 3-D is on, you may precede the drawing name by the drive specification or path name:

B:*filename*

If you press **Enter** or press the first mouse button when asked for the drawing name, a directory of drawing files will be displayed on the screen and you can select a file using the arrow keys or a mouse.

The drawing name can be entered from the command line along with the Retrieve command:

load *filename* or **retrieve** *filename*

DesignCAD 3-D ordinarily retrieves 3-D drawing files with a file type of .DW3. It is also possible to retrieve 2-D drawings from DesignCAD 2-D by specifying a file extension of .DC2, or to retrieve DesignCAD 3-D drawings in the older ASCII format by specifying a file of .DC3.

If there is already a drawing on the screen, it can be merged with another by merely retrieving the second drawing. This is the way symbols can be used with DesignCAD 3-D.

You can automatically load a drawing when DesignCAD 3-D starts by entering the drawing name on the command line:

DC3 *filename*

Tip: You can use the utility DCDW to convert any drawings in .DC3 format to the newer .DW3 format.

Related Topics: Block Load, Save.

Rotate Command

| | |
|--------------------|----------------|
| Menu: | DISPLAY |
| Menu Command: | Rotate |
| Short Form: | R |
| Keystroke Command: | R |

Point 1: Center of rotation (optional).
Point 2: New location for point 1 (optional).

The Rotate command allows you to rotate the entire drawing. To use the command, press the **R** key or select **Rotate** from the **DISPLAY** menu. You will be asked for the Rotation Axis and Rotation Angle. The Drawing can be rotated about the X, Y, or Z axis.

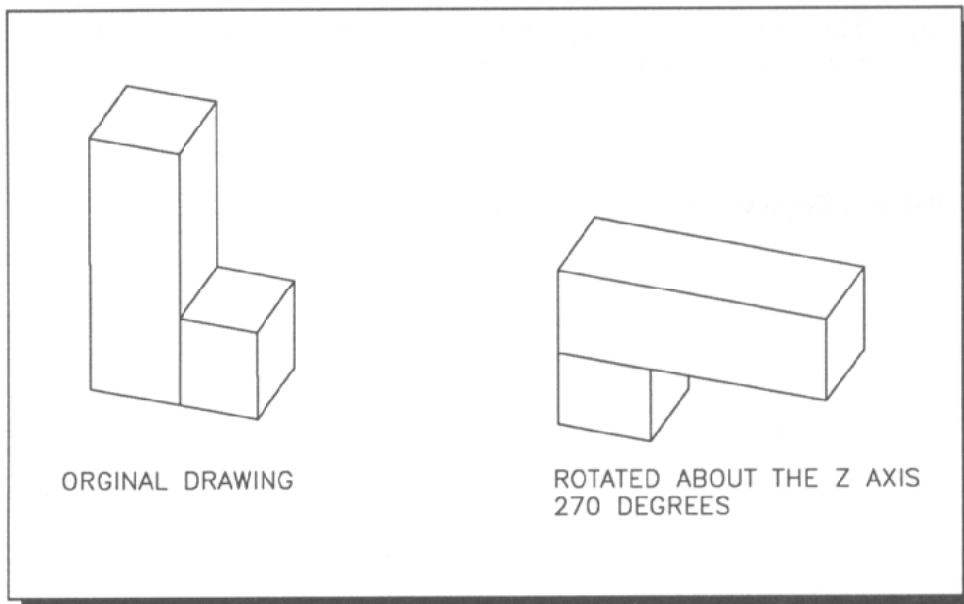


Figure 66 - The Rotate command.

Rotation about the X axis causes the drawing to be rotated toward the rear. Rotation about the Y axis causes the drawing to rotate counterclockwise looking at the drawing from above. Rotation about

the **Z** axis causes the drawing to rotate counterclockwise looking at the drawing from the front.

The rotation angle tells how far you want your drawing to be rotated. (0 leaves it the same, 45 moves it counter-clockwise 45 degrees, -45 moves it clockwise 45 degrees, etc.)

You may set 0, 1, or 2 points with this command. The first point is the point about which the drawing is rotated. If you do not set any points, the drawing will be rotated about the center of the screen.

The second point, if specified, is the location to which the first point is to be moved. If only 1 point is specified, that point will remain in the same location.

Example: Rotate a box 45° about the Y axis. Use the Box command to draw a 3-dimensional box. Set a point near the center of the base of the box and press **R**. Choose the **Y** axis and enter **45** degrees when asked. The box will be rotated about the vertical axis.

Related Topics: View, Block Rotate.

Rubber Band Box Command

| | |
|--------------------|-------------------|
| Menu: | CURSOR |
| Menu Command: | Rubber Box |
| Short Form: | RB |
| Keystroke Command: | [|

Points: None.

This command displays a 3-D box using the cursor and the last point that was set to define the opposite corners of the box. The box moves around with the cursor until the next point is set or until the next drawing command is performed.

To activate the Rubber Band Box, press the "[" key, or select **Rubber Box** from the **CURSOR** menu. The Rubber Band Box stays on until it is turned off by pressing the "[" key again (or by selecting **Rubber Box** again from the menu).

Related Topics: Box.

Run Command

| | |
|---------------|--------------|
| Menu: | FILES |
| Menu Command: | Run |

Points: None.

This command runs a BasicCAD program. To use the command, enter **Run** or **Run program**, where *program* is the name of the BasicCAD program to be run. If *program* is not specified, you will be asked for the program name. To select the program from a directory listing, press **Enter** or press the first mouse button when asked for the program name.

BasicCAD is a complete programming language that can be used to develop programs to be run under DesignCAD. For more information on BasicCAD, see the *BasicCAD Reference Manual*.

A BasicCAD program can also be run by entering the program name from the command line (without **run**), as long as the program name is not the name of a DesignCAD 3-D command.

Related Topics: Macro Define.

Save Command

| | |
|--------------------|--------------|
| Menu: | FILES |
| Menu Command: | Save |
| Keystroke Command: | F10 |

Point 1: Drawing handle 1 (optional).
Point 2: Drawing handle 2 (optional).
Point 3: Drawing handle 3 (optional).

The Save command is used to save a drawing to disk. To use this command, select **Save** from the **FILES** menu or press the **F10** key. You will then be asked for the output drawing name. Enter the name under which you want to store your drawing. The name must be a valid DOS file name from 1 to 8 characters long, using letters or numbers. The drawing will be stored to disk under that name.

You can also enter the drawing name on the command line with the Save command:

SAVE *filename*

One, two, or three points may be used with the Save command to specify the origins or handles of the drawing. The handles of the drawing are the reference points by which the drawing can be retrieved (see the Retrieve command). If no point is set, the handles of the drawing will be the front lower left, lower right, and upper right corners of the drawing.

To store the drawing on a drive or subdirectory other than the current drive and directory, you may precede the drawing name by the drive specification or path name:

B:*filename*

When you are asked for the output file name, the current file name (if any) will be shown in brackets. Press **Enter** to use this file name.

The file name under which a drawing is stored is *filename.DW3*, where *filename* is the name of the drawing. This command will store the entire drawing, even though parts of it may not lie on the screen.

To delete a drawing from the disk, use the Directory command (**F7** key). A drawing may also be deleted using the MSDOS Erase command:

ERASE *filename* .DW3

where *filename* is the name of the drawing to be deleted. This command can be executed only after you exit DesignCAD 3-D.

Tip: Earlier versions of DesignCAD 3-D use a different drawing file format with the extension .DC3 . You can save a drawing in the .DC3 format by entering the file extension .DC3 with the file name.

Tip: You can use the utility DCDW to convert any drawings in .DC3 format to the newer .DW3 format.

Related Topics: Block Save, Retrieve, Save 2D.

Save 2-D Drawing Command

| | |
|--------------------|----------------|
| Menu: | FILES |
| Menu Command: | Save 2D |
| Keystroke Command: | F4 |

Points 1-3: Set drawing handles (optional).

This command saves a 2-Dimensional copy of the current view of the drawing. This 2-Dimensional copy can be used with other 2-Dimensional CAD systems such as DesignCAD 2-D.

To use the command, select **Save-2D** from the FILES menu or press the **F4** key. Enter the file name when asked. As with the Save command, the drawing name can be preceded by the drive or path name.

You can also enter the drawing name on the command line with the Save-2D Drawing command:

SAVE-2D *filename*

After the file name is entered you will be given the option to save the drawing with hidden lines removed. This is typically slower but results in a more realistic 2-Dimensional image.

One or two points may be set with this command to specify the handles of the 2-D drawing.

The file name under which a drawing is stored is *filename.DW2*, where *filename* is the name of the drawing. This command will store the entire drawing, even though parts of it may not lie on the screen.

2-Dimensional File Formats

DesignCAD 2-D versions 5 and earlier read .DC2 files. Version 6 and later support the .DW2 file format. You can use the DCFILES file conversion program (**File Convert** from the FILES menu) to convert from one format to another. You can also use the file

conversion program to convert to WPG (Word Perfect format), DXF, and other common file formats.

Tip: To convert a hidden-line image of the drawing to another file format, save the hidden-line image with the Save 2-D command and then use the file conversion program DCFILES.

Related Topics: Save.

Screen Color Command

| | |
|---------------|---------------------|
| Menu: | COLORS |
| Menu Command: | Screen Color |
| Short Form: | Scolor |

Points: None.

This command can be used to change the colors on the DesignCAD 3-D screen, including the background color, menu colors, and dialogue box colors.

To use the command, select **Screen Color** from the **COLORS** menu. A menu will be displayed, and you will be able to set the following colors:

Background Color

To set the background color, press **+** or **-**. This will page through the 16 basic colors. If you have a display capable of 256 colors, you can press the **B** key to page through the 16 shades available for the currently displayed basic color. For example, you can press **+** until you get to Blue. Then you can press **B** until you get to the desired shade of Blue. Then press **Enter** to select the color.

Top Menu Bar Color

The top menu bar is the command menu at the top of the screen. You can press 1 to 7 to change the following colors on the top menu bar:

- 1 - Normal text background color
- 2 - Normal text foreground color
- 3 - Highlighted text background color
- 4 - Highlighted text foreground color
- 5 - Normal text emphasized character
- 6 - Highlighted text emphasized character

Drop Down Menu Color

The drop down menu is the command menu that "drops down" from the menu at the top of the screen. You can press 1 to 6 to change the following colors on the drop down menu bar:

- 1 - Normal text background color
- 2 - Normal text foreground color
- 3 - Highlighted text background color
- 4 - Highlighted text foreground color
- 5 - Shadow color
- 6 - Normal text emphasized character
- 7 - Highlighted text emphasized character

Command Line Text Color

The command line is the text line at the bottom of the screen where you enter commands. You can press 1 to change the background color or 2 to change the foreground color.

Progress Bar Color

The progress bar is the line at the bottom of the screen that displays the progress of commands such as Hidden Line Removal and Solid Subtract. You can press 1 to change the foreground color or 2 to change the background color.

Input Dialogue Box Color

The input dialogue box is the box that appears in the center of the screen when you enter information. For example, the Text command an input dialogue box. You can press 1 to change the background color or 2 to change the foreground color.

Warning Message Color

The warning message color is the color used with warning messages that are displayed in the center of the screen. You can press 1 to change the background color or 2 to change the foreground color.

Help Message Color

The help message color is the color used with help messages that are displayed in the center of the screen. You can press 1 to change the background color or 2 to change the foreground color.

When you press a key to change the color, the color changes instantly on the screen so you can see the results. You can press a key to

change the color 16 times, and the color will start over at the original color.

After the colors for an item are set correctly, press **Enter** to return to the Screen Color menu. To cancel the new colors are restore the original colors, press **Esc**.

From the Screen color menu, you can press any of the following keys:

- F1** - Display Help
- F2** - Exit and use the new colors
- F3** - Exit, use the new colors, and save them as defaults. This is equivalent to the Configuration Save command.
- Esc** - Cancel the Screen Color command and restore the original colors.

Example: Change the background color of the drawing screen to green. Select **Screen Color** from the COLORS menu. Select the Background Color (option 1) from the Screen Color menu. Then press the **+** key until the background color is green. Then, if you have a 256 color display, press **B** until the desired shade of green is reached. Then press **Enter** to select the color. You will return to the Screen Color menu. If you really want to use the green background, you can press **F1**. Otherwise, press **Esc** to cancel the command.

Tip: You can get rid of all the custom colors and return to the "factory settings" by erasing the file DCAD4.SYS. This also erases the other system parameters.

Related Topics: Configuration Save.

Screen Read Command

| | |
|---------------|------------------------------|
| Menu: | FILES or SHADE |
| Menu Command: | Screen Read |
| Short Form: | SR |

Point 1: Screen image location, upper left corner (optional).

The Screen Read command reads a "Screen Image" and displays it on the screen. To use the command, select **Screen Read** from the **FILES** menu and enter the file name when asked. The screen image will then be displayed on your screen.

You can optionally enter the file name along with the Screen Read command on the command line:

sr *filename*

where *filename* is the name of the screen image.

If you press **Enter** when asked for the file name, a directory of screen image files will be displayed on the screen and you can select a file using the arrow keys or a mouse.

One point can be used with this command to specify the location for the upper left corner of the screen image. This should be done only when reading a partial screen image, because a full screen image should always be retrieved at the upper left corner of the screen. A partial screen image can be created with the Screen Save command by setting points to define the area to be saved. See the Screen Save command for more information.

After a screen image is read, the DesignCAD cursor will be hidden. Press any key to enable the cursor and continue with the program.

This command reads a screen image onto your screen. This is not the same as reading a drawing. You cannot perform zooms, rotates, or other manipulations on a screen image. It is displayed only. If you retrieve a screen image over a drawing, the drawing will be

temporarily hidden but can be regenerated using the Zoom command.

The Screen Read command reads screen images generated with the Screen Save or Slide Show command. The screen image being read must be compatible with the graphics adapter being used. For example, you cannot read a 256-color screen image onto a monochrome screen with this command.

Tip: You can add a parameter to the Screen Read command so you don't have to press a key after the screen image is loaded. For example, enter **SR filename Y** instead of **SR filename**. This is useful in BasicCAD programs when you don't want the program to pause after a screen image is loaded.

Example: Save the screen image of a shaded cone and read it back again. Draw a cone using the Cone command. Shade it using the Quick Shade command. Then save the screen image using the Screen Save command. Clear the screen. Use the Screen Read command to load the screen image back onto the screen. The image on the screen is NOT the DesignCAD 3-D drawing of the cone. It is only a screen image of the shaded cone. To verify this, zoom with a zoom factor of 1. The screen will be erased because there is not drawing in memory.

Related Topics: Screen Save, DCSLIDE, Slide Show.

Screen Save Command

| | |
|---------------|------------------------------|
| Menu: | FILES or SHADE |
| Menu Command: | Screen Save |
| Short Form: | SS |

Point 1: Corner of area of screen to save (optional).

Point 2: Opposite corner of area to save (optional).

The Screen Save command saves a "Screen Image" (a copy of the screen) to disk. To use the command, select **Screen Save** from the FILES menu or **ss** from the command line. Enter the file name when asked. If no points are set, the entire screen will be saved.

You can optionally set points in opposite corners of a window or rectangle, and only the portion of the screen inside that window will be saved. Before the screen is saved you will be given the option to erase the frame and menu.

Screen images saved with this command can be read onto the screen using the Screen Read command. Several screen images can be displayed in sequence using the DesignCAD DCSLIDE utility (see the section in this manual on DCSLIDE).

The Screen Save command can be used to save screens with a drawing shaded or with hidden lines removed for later viewing. The Slide Show command can be used to create a series of these screen images at several different viewpoints.

The screen image files can also be used with other software. The Screen Save command saves screen images in the PCX file format, which is compatible with many Paint and Desktop Publishing packages.

When a screen image is saved, only the colors on the screen are saved. The drawing itself is not saved. When you read a screen image, you cannot zoom, rotate, or modify it as you can a DesignCAD 3-D drawing.

The file name under which a screen image is stored is *filename.PCX*, where *filename* is the name of the drawing.

Example: Save the screen image of a shaded cone, and read it back again. Draw a cone using the Cone command. Shade it using the Quick Shade command. Then save the screen image using the Screen Save command. Clear the screen. Use the Screen Read command to load the screen image back onto the screen. The image on the screen is NOT the DesignCAD 3-D drawing of the cone. It is only a screen image of the shaded cone. To verify this, zoom with a zoom factor of 1. The screen will be erased because there is not drawing in memory.

Related Topics: Screen Read, DCSLIDE, Slide Show.

Section Delete Command

| | |
|--------------------|--------------------|
| Menu: | EDIT |
| Menu Command: | Section Del |
| Short Form: | D |
| Keystroke Command: | D |

Point 1: Corner of section.

Point 2: Opposite corner of section.

The Section Delete command is used to delete or erase a section of the drawing. A section is a 3-Dimensional box defined by two points in opposite corners.

To use this command, define the section by setting two points in opposite corners of a 3-Dimensional box.

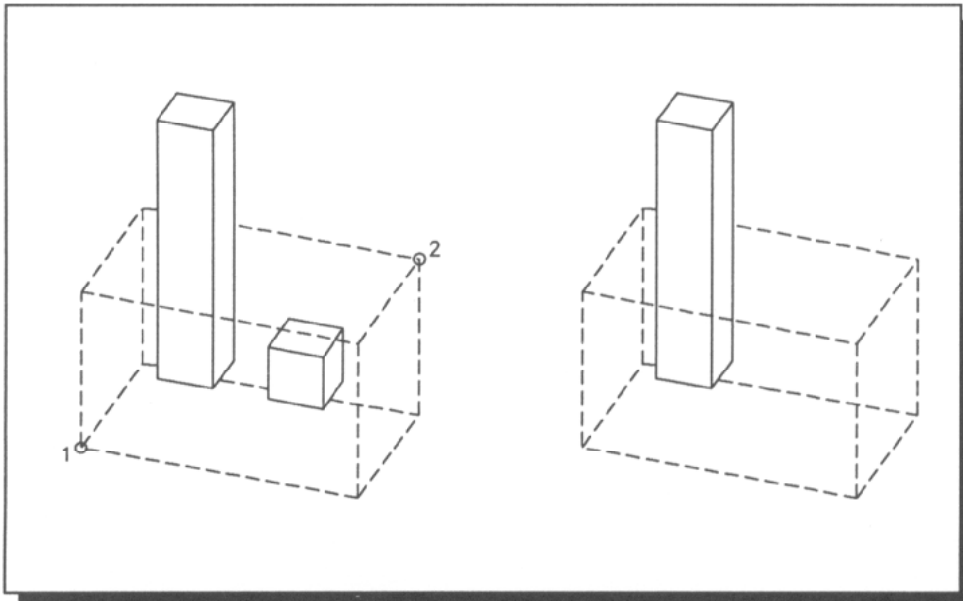


Figure 67 - Set two points in opposite corners of a 3-dimensional box. Any entity entirely in the box or "section" will be erased.

Any line or drawing entity (e.g., line, curve, circle, arc, text) completely within the section defined will be deleted from the drawing.

This command deletes the lines or entities located entirely within the section or box. If a command lies partly within and partly outside the box, it will not be deleted.

To "cut off" or break lines at the border of the section, the **^D** keystroke command or the **Section DelC** menu command can be used. The Unerase command (! key) can be used to replace items accidentally deleted with the Section Delete command.

Tip: If you press the **D** key before setting points, it will automatically enable the rubber band box, making it easier to see the section to be erased.

Related Topics: Section Delete Cutoff, Block Delete.

Section Delete Cutoff Command

| | |
|--------------------|---------------------|
| Menu: | EDIT |
| Menu Command: | Section DelC |
| Keystroke Command: | ^D |

Point 1: Corner of area.
Point 2: Opposite corner of area.

The Section Delete command is used to delete or erase a section of the drawing. A section is a 3-Dimensional box defined by two points in opposite corners. This command is the same as the Section Delete command, except that any line that is partly inside the section will be "cut off" and the part that lies inside the section will be erased.

To use this command, define the section by setting two points in opposite corners of a 3-Dimensional box.

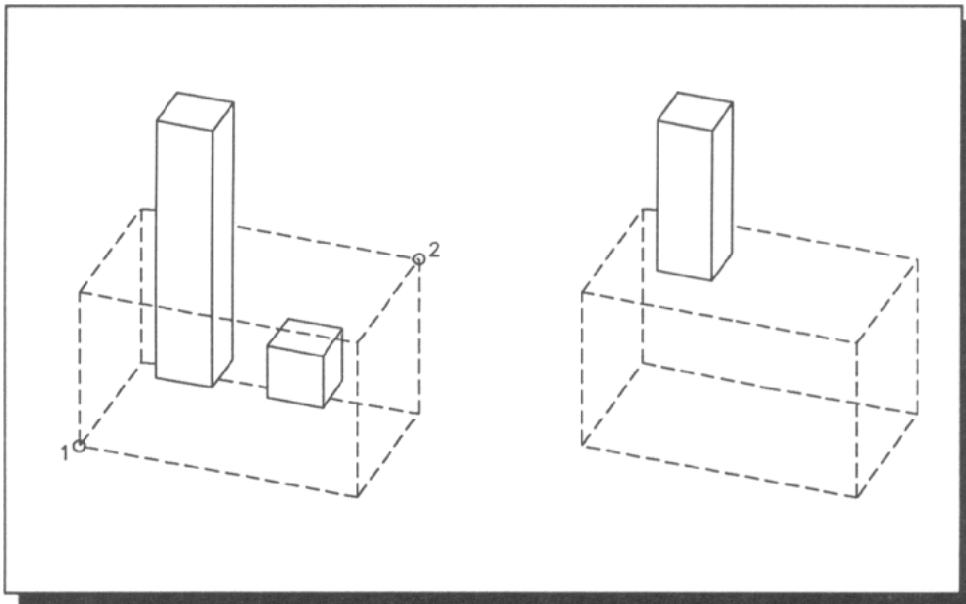


Figure 68 - Set two points in opposite corners of a 3-dimensional box. Entities partly inside the box or "section" will be cut off.

Any line or drawing entity (e.g., line, curve, circle, arc, text) within the section defined will be deleted from the drawing. An entity partly inside the section will be cut off, and only the part inside the section will be erased.

The Unerase command (! key) can be used to replace items accidentally erased with the Section Delete command.

Tip: If you press **^D** before setting points, it will automatically enable the rubber band box, making it easier to see the section to be erased.

Related Topics: Section Delete, Block Delete, Solid Subtract, Slice.

Section Stretch Command

| | |
|--------------------|----------------|
| Menu: | EDIT |
| Menu Command: | Stretch |
| Short Form: | S |
| Keystroke Command: | S |

Point 1: Corner of section.

Point 2: Opposite corner of section.

Point 3: Point to which the section is to be stretched.

The Section Stretch command is used to stretch a section of the drawing from one location to another. A section is a 3-Dimensional box defined by two points in opposite corners.

To use this command, define the section by setting two points in opposite corners of a 3-dimensional box. Set a third point at the position to which you want the lower left corner of the section to be moved.

After the points have been set, press the **S** key. Any drawing entity (e.g., line, curve, circle, arc, text) within the section will be moved to the new location. If a line or entity lies partly within and partly outside the section, it will be stretched from the new location to the old.

Example: Stretch the top of a box to make it taller. Use the Box command to draw a 3-dimensional box. Then press the **S** key. Set two points enclosing the top of box in an imaginary rectangle. (A rubber band box will be displayed to help you see this.) Look at the side views to make sure you enclose the box in 3 dimensions. Set a third point directly over the front left corner of the box. The Box will be stretched upward. The distance the box is stretched is the distance between the bottom of the section and the third point.

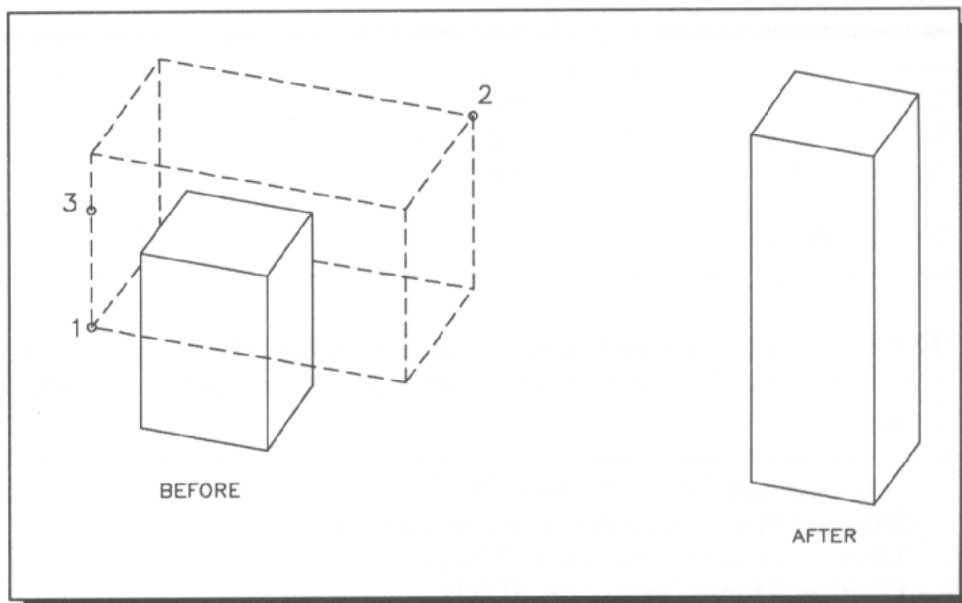


Figure 69 - Set two points to define the section, and a third point for the destination.

Tip: If you use the front lower left corner of the section for the second point, it is easier to see how far and in what direction the object will be stretched. It will be stretched from the second point to the third point.

Tip: If you press the **S** key before setting points, it will automatically enable the rubber band box, making it easier to see the section to be stretched.

Related Topics: Block Scale, Point Move.

Set Path Command

| | |
|---------------|-----------------|
| Menu: | Files |
| Menu Command: | Set Path |
| Short Form: | Path |

Points: None.

The Set Path command is used to set the DOS path for the various files used with DesignCAD 3-D. Paths can be set for the following files:

- Screen Image files (extension .PCX)
- Drawing files (extensions .DW3 and .DC3)
- View information (extension .VW3)
- Digitizer Menus (extension .SDM)
- BasicCAD Programs (extension .BSC)
- Material Data (extension .MAT)
- Color Data (extension .PAL)
- Slide Show (extensions .SV3 and .SHW)

The DOS path can include the drive as well as the directory in which the files are to be located.

After you have set the paths, you can press **F2** to use the new paths, **F3** to save the new paths as defaults, or **Esc** to cancel the command and ignore the new paths. **F1** can be used to display a help screen.

The path information can also be saved permanently by changing the path information, pressing **F3**, and later executing the Configuration Save command.

Example: Set the default path for DesignCAD 3-D drawings to the root directory of drive D:. Select **Set Path** from the FILES menu. Move the cursor to "Drawing Files" and enter **D:**. Press **F3** to save the new path permanently.

Tip: To reset all the DesignCAD system parameters to their original values, you can erase the file DCAD4.SYS.

Related Topics: Save, Retrieve, Directory.

Set View Command

| | |
|---------------|-----------------|
| Menu: | VIEW |
| Menu Command: | Set View |
| Short Form: | SV |

Points: None.

The Set View command is used to enable or disable views 2, 3, and 4. These views are the small views on the left of the DesignCAD 3-D screen.

When the three small views are disabled, the large drawing screen occupies their space. This can be useful when saving screen images of the drawing.

After the views have been configured, press **F2** to use the new configuration, **F3** to save the view configuration as DesignCAD 3-D defaults, or **Esc** to cancel the view command and ignore changes.

You can press **F1** to display help in the Set View command.

Tip: When a DesignCAD 3-D drawing is regenerated on the screen, it is normally drawn in all four views. You can turn off views 2, 3, and 4 and the drawing will be redrawn 4 times faster.

Related Topics: Screen Save, View.

Shade Command

| | |
|---------------|---|
| Menu: | SHADE |
| Menu Command: | Shade, Shade 2, Shade 3, Shade 4 |
| Short Form: | SH, SH2, SH3, SH4 |

Point 1: Corner of area to shade (optional).

Point 2: Opposite corner of area to shade (optional).

The Shade commands are used to shade your drawing - that is, to add color to the surfaces in the drawing, resulting in a more realistic image.

There are two types of shading in DesignCAD 3-D: Shade and Quick Shade. The Shade command produces high-quality images and smooth shading, material properties, and reflection properties. The Quick Shade command does not, but it is faster than the Shade command.

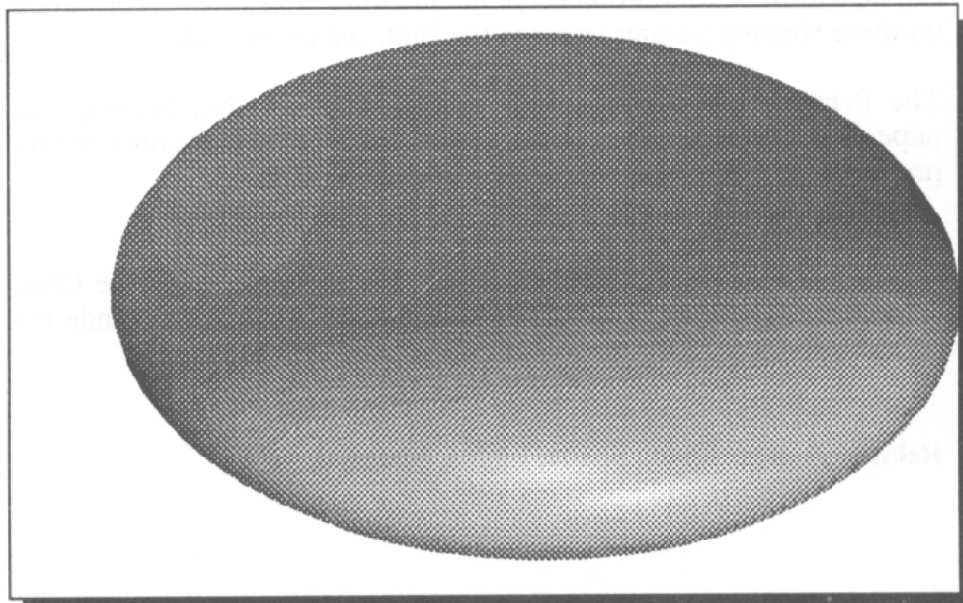


Figure 70 - The Shade command performs smooth shading.

To use this command, select **Shade** from the SHADE menu. The drawing will then be shaded on the screen. DesignCAD 3-D can shade your drawing even on a display with limited or no color capability.

View 1 (the large view) is shaded with the Shade command. Views 2, 3, and 4 can be shaded with the Shade 2, Shade 3, and Shade 4 commands.

To shade only a portion of the drawing, set two points in opposite corners of the section on the screen to be shaded. If two points are set with the Shade command, the screen will not be cleared before shading and only the section of the screen between the points will be shaded. The rectangle or section of the screen that is shaded is defined to be left-right, up-down with respect to the screen, not the drawing. This is a fast way to "check" small parts of the drawing.

One to eight light sources can be enabled and set at any angle and intensity for shading (see the Lights command).

Shading parameters such as colors, texture, and reflection properties can be set for different objects in the drawing. For more information on these shading parameters, see the Material command.

The Print and Plot commands can produce shaded drawings on paper. It is not necessary to shade the drawing on the screen before producing a shaded drawing on a printer or plotter.

Example: Shade a drawing of a cone. Draw a cone using the Cone command. Then select **Shade** from the SHADE menu to shade the cone.

Related Topics: Quick Shade, Lights, Material.

Show Info Command

| | |
|---------------|------------------|
| Menu: | FILES |
| Menu Command: | Show Info |
| Points: None. | |

The Show Info command displays the version number, release date, and serial number of your DesignCAD 3-D. If you need to call DesignCAD 3-D technical support, you will be asked for this information. You get this information using the Show Info command or by running the utility CHECKDC.

Example: Get the serial number of your DesignCAD 3-D. Select **Show Info** from the FILES menu. The serial number will then be displayed on the screen.

Related Topics: CHECKDC program.

Slice Command

Menu: **EDIT**
Menu Command: **Slice**
Short Form: **SL**

Points 1,2,3: Define the "cutting plane."
Point 4: Which side of the plane to erase (optional).

The Slice command is used to erase parts of the drawing in front of or behind a plane. The plane can be any plane defined by three points.

To use the command, select **Slice** from the EDIT menu. Set three points to define the "cutting plane." Then set a point either in front of or behind the "cutting plane." All parts of the drawing that lie on that side of the cutting plane will be erased.

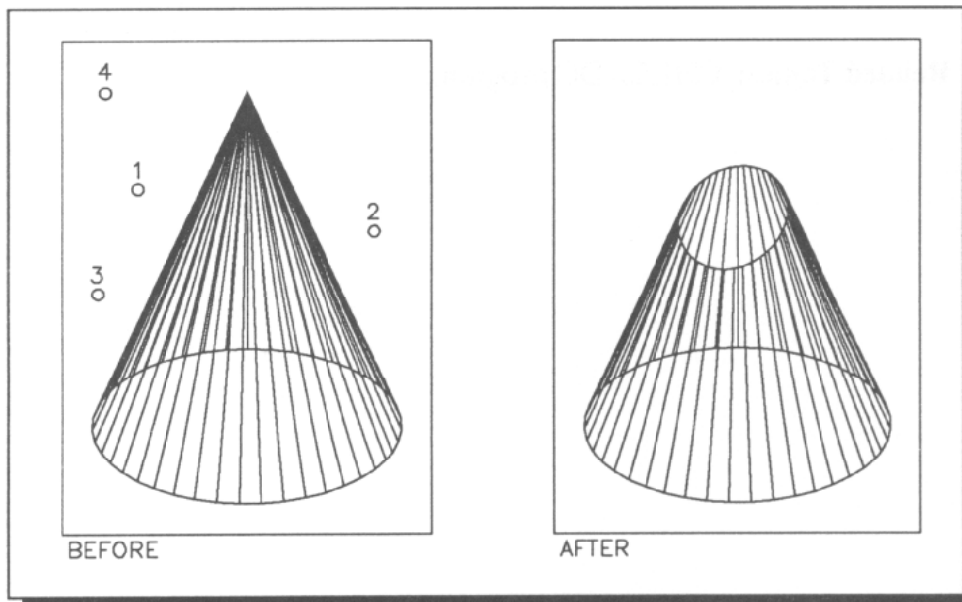


Figure 71 - The Slice command.

All lines and surfaces that cross the plane will be broken at the plane. If the fourth point is not used, then all entities and parts of entities in front of the cutting plane will be erased.

Example: Cut a sphere in half diagonally. Draw a sphere. Select **Slice** from the menu and set three points to define the "cutting plane." To define the diagonal cutting plane, set one point at the center of the sphere, one point up and to the right of the first, and a third point directly behind the second (in the positive Z direction). Set a fourth point to the right of the imaginary cutting plane to specify which part to erase. The right half of the sphere will be removed.

Tip: This command is very useful in making a cross section of a Solid object.

Tip: The Slice command erases parts of the entire drawing. The Block Slice command can be used to "slice" only a Block.

Related Topics: Section Delete Cutoff, Block Slice.

Slide Show Command

| | |
|---------------|-------------------|
| Menu: | FILES |
| Menu Command: | Slide Show |
| Short Form: | Slides |

Points: Special (optional).

The Slide Show command is used to save a series of screen images and a slide show file. The screen images can then be "played back" as an animation sequence using the DCFILES program. The screen images saved with the Slide Show command are different views of a single drawing. The drawing can be processed in wireframe mode, with hidden line removal, shaded, or quick shaded.

The screen images are generated using a "keyframe animation" technique. Several "keyframes" or views are specified, and a number of screen images are generated between each pair of keyframes. The view for each screen image is generated in a way that there is a smooth transition between keyframes.

Suppose, for example, that the viewpoints of two keyframes are 90° apart, and that 9 screen images or slides are to be generated between these keyframes. The first slide will have the same viewpoint as the first keyframe. The viewpoint of the second slide will shift 10° toward the second keyframe. The viewpoint of the third slide will shift another 10° toward the second keyframe, and so on, until the 10th slide has the same view as the second keyframe.

Several items of information can be modified between each keyframe. The view angle, the view distance, the rotation of the object, and the zoom factor of the object can all be modified between keyframes. These items are interpolated between keyframes for each slide generated.

In short, the way you generate a slide show is to do the following:

1. Set the viewing information for 2 or more keyframes.

2. Enter the number of slides to be generated between each pair of keyframes.

The slides for the animation sequence will be generated automatically, and they can be "played back" using the DCSLIDE program.

Of course, there are many options available with the Slide Show command. Those will be discussed now.

When you execute the Slide Show command, a screen with the following options will be displayed:

Save Slides to .PCX files

When you make a slide show, you can choose to save the slides (PCX files), or you can just "preview" the slide show.

Slide View file name (.SV3)

The Slide View file is a file that contains the view information for the keyframes in the slide show. You can save a View File and use it with other drawings. You can use a previously saved View File by entering its name here. To select the Slide View file from a directory listing, leave the name blank and press **F3** (show slides).

Slide Show file name (.SHW)

The slide show file name is a listing of the slides generated by the Slide Show command. This file is used by the DCSLIDE program to "play" the animation. The screen image files are saved under the name *name1.PCX*, *name2.PCX*, *name3.PCX*, etc., where *name* is the slide show file name.

Title file name (.PCX)

A title file is a PCX file that can be merged with every screen image in the slide show. For example, you can make a small PCX file containing a company logo to be placed on each slide in the slide show. The title file will be placed at its original location on the screen - at the location where it was when it was saved. Since the title file will be placed on top of the other images, you should place it carefully so it will not obscure the slide show.

If you do not want a title file, leave this space blank.

Erase the frame?

Enter **Y** here to erase the drawing frame, the command line, and the status line, so the only thing in the slide show is the drawing itself.

Slide show Mode (1-4)

There are four slide show modes. These modes designate the form of the drawing for the screen images:

1. Shading
2. Quick Shading
3. Wireframe
4. Hidden Line Removal

To save shaded screen images, select mode 1. For wireframe images, select mode 3.

Tip: If you are previewing a slide show, use the wireframe mode. The slide show will be processed much faster than it would with the other modes. When you are ready to save the slides, you can change to the final output mode such as shading.

You can press **F1** at any time for help.

When these options are set, you can select one of the following:

F2 - Make Keyframes

Select this option to create the keyframes for the slide show. A keyframe is a view configuration for the drawing - the View Angle, View distance, Rotation, and Zoom. Screen images are created between each pair of views to make a smooth transition between views.

Use this option if you not using an existing Slide View file.

F3 - Generate Slides

This option can be selected if you are using an existing Slide View file. It skips the step of making keyframes.

Esc - Cancel

Press **Esc** to cancel the Slide Show command and return to the DesignCAD drawing screen.

MAKING KEYFRAMES

When you select the "Make Keyframes" option, the drawing will be displayed on the screen and you will be asked to set a point for the center of zoom and a point for the center of rotation. Press **Enter** to use the origin (marked with a +) for these points. Otherwise, set a point about which the drawing will be zoomed in the slide show generation. Set a second point for the center of rotation. If only one point is set, it will be used for the center of zoom and rotation.

After setting the center of zoom and rotation, you can begin saving keyframes. You can change the view angle, view distance, zoom, and rotation of the drawing between keyframes. Press **F2** to save a keyframe. Each time you save a keyframe (except the first), you will be asked how many slides are to be generated before that keyframe. Press **F3** after all the keyframes have been saved.

To change the view angle, hold down the left mouse button and move the mouse, or press the arrow keys. To change the view distance, use the + and - keys. Press **F4** to enter numeric values for the view. This is similar to the View command.

Press < and > to zoom the drawing. You do not need to hold down the **Shift** key with the > and < keys.

Press **F6** to rotate the drawing. You can rotate about only one axis in each keyframe.

The Frame Center is the point at which the "camera" is aimed. You can change this point by pressing **F5**. This causes the drawing to be "shifted" across the screen.

You do not have to memorize these functions. They are displayed on the screen, and you can press **F1** for further help.

GENERATING SLIDES

After making the keyframes (or reading them from disk) the slides are generated. The drawing will be displayed as it was in the first keyframe. The drawing will then be shaded, quick shaded, have hidden lines removed, and the screen image or slide will be saved to disk, depending on the options of the main menu.

The slide generation can be cancelled by pressing **Esc**.

The slide generation step may take some time. For example, if it takes 15 seconds to shade a drawing, and if you have 20 keyframes with 12 slides between each keyframe, it will take more than one hour to generate a shaded slide show.

After slides have been saved to disk, you can view the slide show using the program DCSLIDE. See the "DesignCAD Utilities" section of this manual for more information.

Snap Grid Command

| | |
|--------------------|------------------|
| Menu: | CURSOR |
| Menu Command: | Snap Grid |
| Short Form: | Snap |
| Keystroke Command: | G |

Points: None.

The Snap Grid command enables or disables the Snap Grid. The Snap Grid is an imaginary grid placed over your drawing. When the snap grid is enabled, the cursor "jumps" from point to point on the snap grid when it is moved by the mouse, digitizer, or cursor keys. Each point set with the **Insert** key or the first mouse button will be located exactly on one of the grid points.

Using the Snap Grid makes it easier to move the cursor to precise locations with the mouse or digitizer.

To enable the Snap Grid, press **G** or select **Snap Grid** from the **CURSOR** menu. To disable the Snap Grid, press **G** or select **Snap Grid** again.

The distance between the points on the Snap Grid is the snap grid size. This can be set using the Snap Grid Size command under the **CURSOR** menu.

The Snap Grid does not affect points set with the "snap" commands such as Gravity Point, Midpoint, or Intersect. It also does not affect points set with the Point Absolute, Point Polar, or Point Relative commands.

When the Snap Grid is enabled, the word **SNAP** is displayed on the Status Line (when the status line format is 1).

Related Topics: Snap Size.

Snap Grid Size Command

| | |
|---------------|------------------|
| Menu: | CURSOR |
| Menu Command: | Snap Size |

Points: None.

The Snap Grid Size command is used to set the Snap Grid size. To use this command, select **Snap Size** from the CURSOR menu and enter the snap grid size.

The snap grid size is saved permanently when the Configuration Save command is executed.

Related Topics: Snap Grid, Drawing Parameters.

Solid Add Command

| | |
|---------------|------------------|
| Menu: | SOLIDS |
| Menu Command: | Solid Add |
| Short Form: | Add |

Points 1 & 2: Set on the two Solids.

The Solid Add command is used to "add" one Solid to another. It makes a single Solid out of two Solids and eliminates the unnecessary surfaces after the addition of the two Solids.

To use the command, set a point on each of the Solids and select **Solid Add** from the **SOLIDS** menu. The Solids will then be combined into a single Solid, and the surfaces "internal" to the resulting Solid will be removed.

For more information on Solids, see the section in this manual "Using Solids."

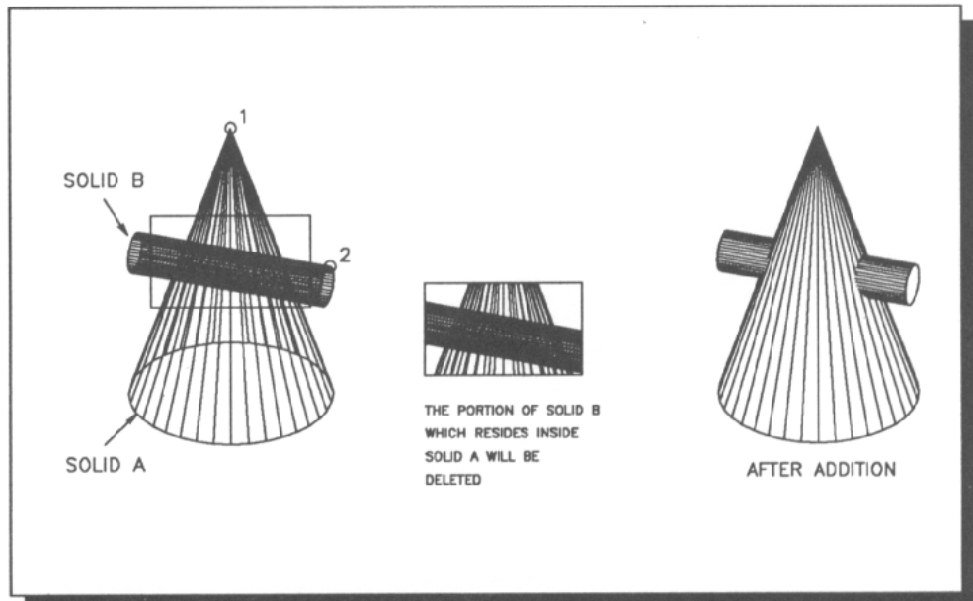


Figure 72 - The Solid Add command.

Example: Combine two 3-dimensional boxes into a single Solid. Use the Box command to draw two 3-dimensional boxes with a corner of each box inside the other box. Use the Solid Add command to combine the two boxes, setting a point on each box. The two boxes will be combined into a single Solid. The surfaces on the interior of the combined Solid will be removed.

Related Topics: Solid Subtract, Slice, Drill, Section Delete Cutoff.

Solid Block Command

| | |
|---------------|--------------------|
| Menu: | SOLIDS |
| Menu Command: | Solid Block |
| Short Form: | SB |

Point 1: Set a point on the Solid.
Points 2 and 3: Block Handles 2 and 3 (optional).

The Solid Block command makes a Block out of a Solid. (The Solid remains a Solid - it can be both a Solid and a Block.) The first point selects the Solid to become a Block. Block Handle 1 is located at this point. The second and third points can be used to specify the locations for Block handles 2 and 3.

This command enables you to use the Block operations on a Solid, such as Block Repeat, Block Insert, Block Scale, and Block Delete. Any Block command can be used on a Solid except the extrusion commands Extrude, Extrude Varying, Sweep, and Spiral. These commands cannot extrude grid entities (complex surfaces) which are commonly found in Solids.

Solid Define Command

| | |
|---------------|---------------|
| Menu: | SOLIDS |
| Menu Command: | Solid |
| Short Form: | SO |

Point 1: Corner of window for entity selection.

Point 2: Opposite corner of window.

The Solid Define command is used to define a Solid. A Solid is a set of planes and surfaces that make up a closed Solid object. A Solid must be defined before performing Solid operations such as Solid Subtract, Solid Add, Solid Intersect, and Interference Checking.

Two points in opposite corners of a 3-Dimensional box are used with the Solid command. All entities (planes, surfaces, etc.) within the section become part of the Solid.

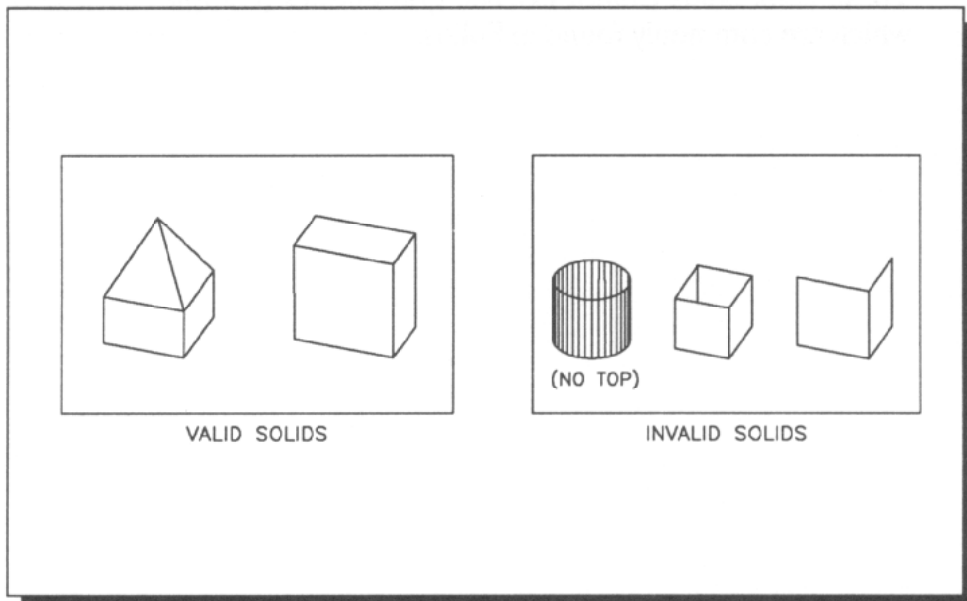


Figure 73 - A Solid is a closed figure made up of planes and surfaces.

This command includes the planes and surfaces located entirely within the section or box. If an entity lies partly within and partly outside the section, it will not be included in the Solid.

All the planes and surfaces making up a Solid must form a closed Solid object. Figure 73 shows some valid and invalid Solids. If you define an invalid Solid, the Solid operations may not work correctly.

When one of the DesignCAD 3-D Solid objects is drawn (Box, Cylinder, Cone, Sphere, and Hemisphere), it is automatically defined to be a Solid. More than one Solid object can be defined to be a single Solid, in which case it will be treated as a single object.

For more information on Solids, see the section in this manual "Using Solids."

Tip: Use the ID command if you need to determine whether a an object has been defined as a Solid.

Related Topics: Block Solid.

Solid Free Command

| | |
|---------------|-------------------|
| Menu: | SOLIDS |
| Menu Command: | Solid Free |
| Short Form: | SF |

Point 1: Set a point on the Solid to be freed.

The Solid Free command can be used to "undefine" a Solid. A Solid is a set of planes and surfaces that make up a closed solid object. A Solid must be defined before performing Solid operations such as Solid Subtract, Solid Add, Solid Intersect, and Interference Checking.

The Solid Free command converts a Solid object in the drawing into a non-Solid. This command does not affect the entities in a Solid. It only removes their status as a Solid.

There can be a maximum of 200 solids in a drawing. If you reach this limit, you can use the Solid Free command to free some of the solids to make room for more.

Related Topics: Solid Define.

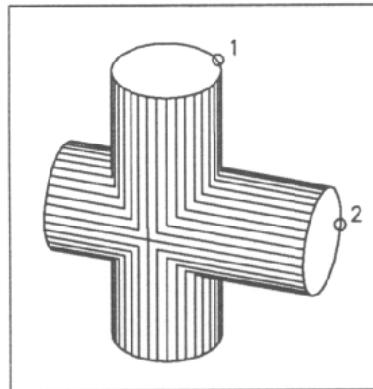
Solid Intersect Command

| | |
|---------------|------------------|
| Menu: | SOLIDS |
| Menu Command: | Solid Int |
| Short Form: | Sint |

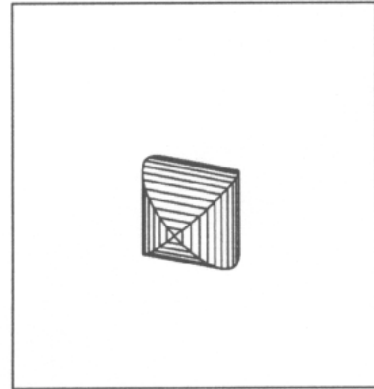
Points 1 & 2: Set on the two Solids.

The Solid Intersect command erases all of two Solids except the intersection of the two Solids - the area that the both Solids occupy.

To use the command, set a point on each Solid and select **Solid Int** from the **SOLIDS** menu. The two Solids will be converted to a single Solid at the intersection of the two original Solids.



BEFORE



AFTER

Figure 74 - The Solid Intersect command erases all but the intersection of two solids.

This command is useful in constructing objects that would otherwise be very difficult to describe. It can also be used to display the area of interference between two Solids.

For more information on Solids, see the section in this manual, "Using Solids."

Example: Produce the intersection of a Cone and a Box. Draw a 3-dimensional box. Then draw a cone through the box. Use the Solid Intersect command on the two objects, setting one point on the box and one point on the cone. The resulting Solid is the intersection of the box and the cone.

Related Topics: Interference, Solid Add, Solid Subtract.

Solid Move Command

| | |
|---------------|-------------------|
| Menu: | SOLIDS |
| Menu Command: | Solid Move |
| Short Form: | SM |

Point 1: Point on the Solid to be moved.

Point 2: New location for Solid (relative to point 1).

The Solid Move command moves a Solid from one location to another. Two points are used to specify the direction and distance the Solid is to be moved. The Solid is moved from the first point to the second. Even if the first point does not lie directly on the Solid, the Solid is moved the direction and distance from the first point to the second.

The entity closest to point 1 must be part of the Solid that is to be moved.

The Solid Move command is used to position Solids in the drawing. It is particularly useful in moving Solids into position for commands such as Solid Subtract, Solid Add, and Solid Intersect.

For more information on Solids, see the section in this manual on Using Solids.

Example: Move a box using the Solid Move command. Draw a 3-dimensional box using the Box command. Use the Solid Move command, and set a point on the box. Set a second point above the first. The box will be moved from the first point to the second.

Tip: The Solid Move command moves a Solid - it does not make another copy of it. To duplicate a Solid, use the Solid Block command to make a Block out of the Solid and then use the Block Insert, Block Repeat, or Block Circular Repeat command.

Related Topics: Block Move.

Solid Subtract Command

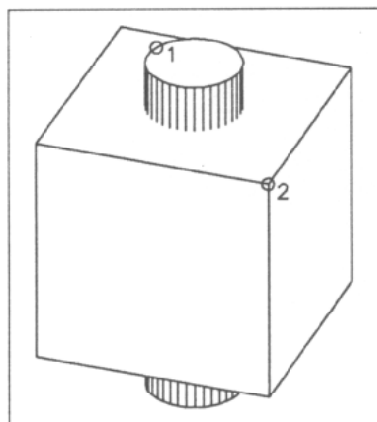
| | |
|---------------|-----------------|
| Menu: | SOLIDS |
| Menu Command: | Subtract |
| Short Form: | Sub |

Point 1: Point on the Solid to be subtracted.

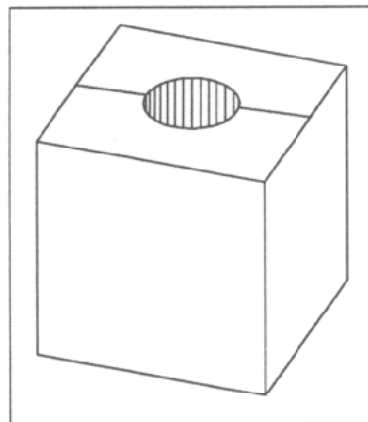
Point 2: Point on the Solid to be subtracted from.

The Solid Subtract command is used to "subtract" one Solid from another. For example, to drill a hole in a Solid object, you can subtract a cylinder from it.

Two points are used with the Subtract command: One point on the Solid to be subtracted and a second point on the Solid to remain. The first Solid will be removed from the second, leaving all the second Solid except that part that coincided with the first Solid.



BEFORE



AFTER

Figure 75 - The Solid Subtract command subtracts the first solid from the second.

For more information on Solids, see the section in this manual "Using Solids."

Related Topics: Section Delete-Cutoff, Slice, Block Slice.

Sphere Command

| | |
|---------------|---------------|
| Menu: | SOLIDS |
| Menu Command: | Sphere |
| Short Form: | Ball |

Point 1: Center of the sphere.

Point 2: Outside of the sphere.

This command is used to draw a sphere. Two points are used with the command: one for the center and one for the radius of the sphere.

After setting the points, you can enter the number of facets for the hemisphere. This determines the smoothness of the hemisphere - the more facets, the smoother the surface. However, more planes occupy more memory and take more time to draw. You can press **Enter** to use the default of 16 facets.

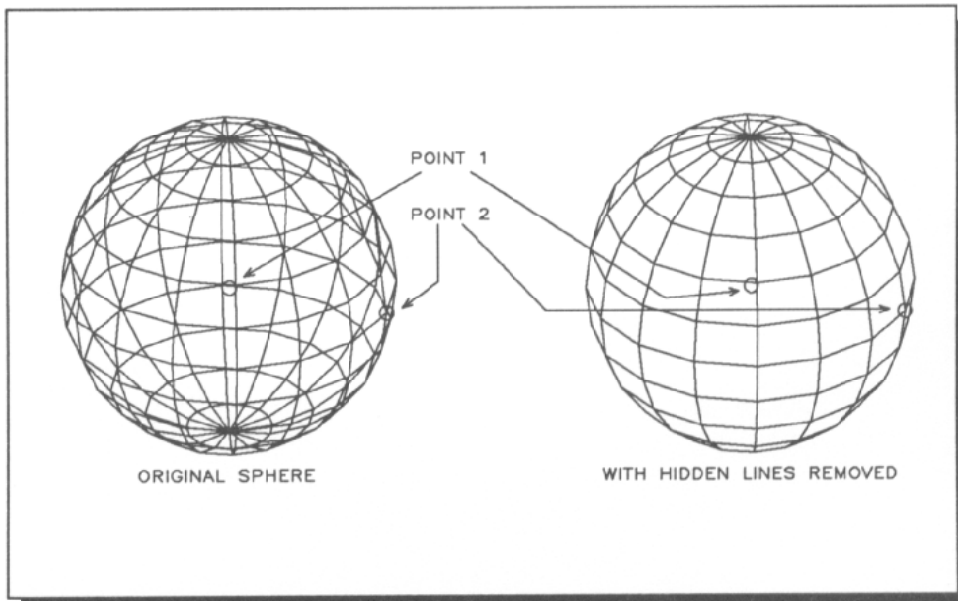


Figure 76 - To draw a sphere, set a point in the center and a point on the surface.

Example: Draw a sphere with a diameter of 16. Use the Sphere command. Set 1 point for the center and 1 point 8 units to the right of the first. When asked, press **Enter** to use 16 facets. A hemisphere with a radius of 8 (and a diameter of 16) will be drawn.

Related Topics: Hemisphere, Circle, Disc.

Status Line Format Command

| | |
|--------------------|--------------------|
| Menu: | PARAMETER |
| Menu Command: | Status Line |
| Short Form: | ST |
| Keystroke Command: | J |

Points: None.

This command is used to specify the format of the DesignCAD 3-D status line. The status line is the line of information at the bottom of the screen.

One of three status line formats can be selected. The status line formats display the following information:

1.
 - a. Number of Points Set.
 - b. Current Layer.
 - c. Current Zoom Factor.
 - d. Current Color.
 - e. SNAP, if the Snap Grid is on.
 - f. MACRO, if a Macro is being made.
2.
 - a. Number of Points Set.
 - b. Distance from the cursor to the last point.
 - c. Delta X, Horizontal distance from the cursor to the last point.
 - d. Delta Y, or vertical distance from the cursor to the last point.
 - e. Delta Z, or vertical distance from the cursor to the last point.
3.
 - a. Number of Points Set.
 - b. X coordinate of the cursor.
 - c. Y coordinate of the cursor.
 - d. Z coordinate of the cursor.

The values in formats 2 and 3 are updated as the cursor is moved about the screen. The distance and X, Y, Z coordinates are measured in Drawing Units specified by the Units command.

Related Topics: Drawing Parameters.

Surface Area Command

| | |
|---------------|---------------------|
| Menu: | INFO |
| Menu Command: | Surface Area |
| Short Form: | SA |

Point 1: Set a point on the surface.

This command is used to calculate the area of a surface. To use the command, set a point on the surface and select **Surface Area** from the INFO menu. The area of the surface will be displayed on the screen.

The Surface Area command determines the area of a single Plane or Grid entity. A plane entity is a single plane created by the Plane or Polygon command. A grid entity is a complex surface created by an extrusion or a sweep.

The Volume command is used to calculate the area (and volume) of an entire Solid, while the Surface Area command calculates the area of a single entity. For example, the Cone command constructs a Solid object made of a plane entity (the base) and a grid entity (sides). To get the surface area of the entire cone including the base, you could use the Volume command. To get the surface area of only the sides of the cone, you could set a point at the top of the cone and use the Surface Area command.

Example: Calculate the surface area of the top of a hemisphere. Use the Hemisphere command to draw a hemisphere oriented like the top half of a sphere. Use the Surface Area command to get the surface area of the curved part of the hemisphere, setting a point near the top of the hemisphere. The surface area will be displayed on the screen.

Related Topics: Volume.

Surf-Surf Command

Menu: **SURFACES**
Menu Command: **Surf-Surf**
Short Form: **SFSF**

Point 1: Set a point on an intersecting surface.
Point 2: Set a point on the other intersecting surface.

This command is used find the intersection of two surfaces. The surfaces can be simple planes, grids, or a combination of the two.

To use the command, set a point on each of the two intersecting surfaces and select **Surf-Surf** from the **SURFACES** menu. A line will be drawn along the intersection of the two surfaces.

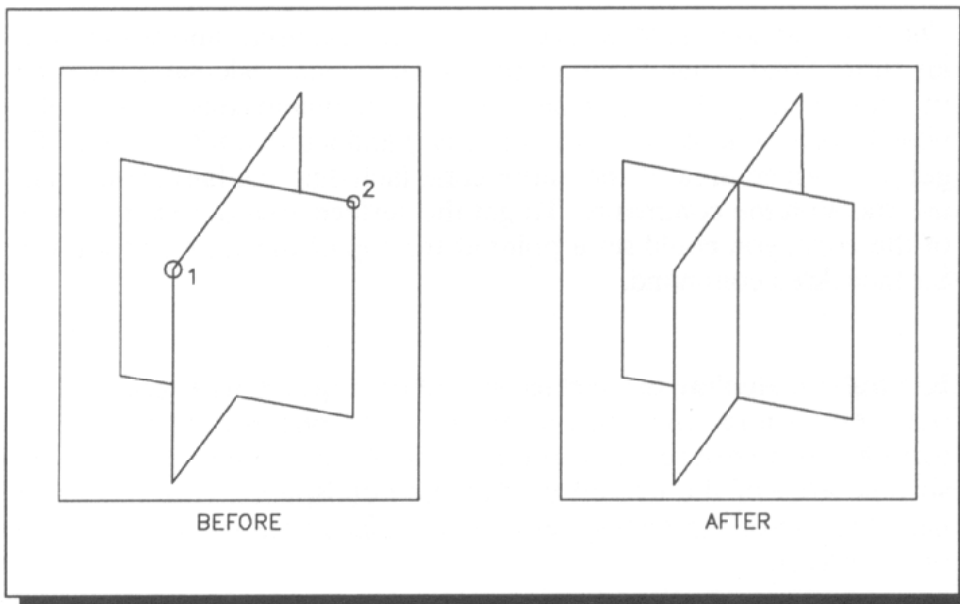


Figure 77 - The Surface-Surface command draws a line along the intersection of two planes or surfaces.

This command is useful in highlighting the intersection of two surfaces that would otherwise be clear only when the drawing is shaded.

Example: Draw a line at the intersection of a cone and a plane.
Draw a cone using the Cone command. Use the Box command to draw a horizontal 2-dimensional box through the cone. Then use the Surf-Surf command, setting a point on the cone and a point on the plane. A line will be drawn at the intersection of the two surfaces.

Related Topics: Drill.

System Parameter Command

| | |
|--------------------|--------------------|
| Menu: | PARAMETER |
| Menu Command: | System Parm |
| Keystroke Command: | ^Q |

Points: None.

The System Parameter command is used to specify DesignCAD 3-D system parameters, such as precision, sound, and coordinate system.

To use the command, press **^Q** or select **System Parm** from the **PARAMETER** menu. A list of the system parameters will be displayed. To change one of the parameters, move the cursor to that location and enter the new value. You can also change the value of a parameter using the **Space** bar or the left and right mouse buttons.

The following system parameters can be set with this command:

Character Font

You can select alternate character fonts or type styles with DesignCAD 3-D. After a new font is selected, all the text on the screen is erased and rewritten in the new character font.

You can display the names of the character fonts on your screen by pressing "?" at this field. For more information on fonts, see the Font command.

The Vector Text command (**^T** key) can be used instead of the Text command (**T** key) to prevent text from being affected by a change in character fonts. For example, to use text in both fonts 1 and 2 in a drawing, you can draw all the text to be displayed in character font 1 using the Vector Text command, change to font number 2, and then draw all the text for character font 2. The text drawn using the Vector Text command will not be changed to character font 2.

Character font number 0 can be used to "hide" the text. When font number 0 is used, each line of text in the drawing is displayed

as a rectangle although the text is still part of the drawing. Since the text is not redrawn when the Zoom and Rotate functions are performed, the Zoom and Rotate commands are faster when character font 0 is used.

Drawing Units per inch on Output

This specifies the number of Drawing Units to be output per inch on paper. This makes it convenient to produce scale drawings.

For example, to make a scale drawing at 1 inch equals 4 feet, set one drawing unit equal to one foot in the drawing using the Units command. Then set the units per inch on output equal to 4.

If you chose units of measurement other than inches during DesignCAD 3-D configuration, then this item will reflect those units. For example, "Units per mm on output" may be displayed instead of "Units per inch on output."

Precision (Fractional Digits) (-7..7)

The precision is the number of digits to the right of the decimal point for values on the status line and in the dimension annotation. This value does not affect the way the drawing is maintained or saved.

Sound? (0=Off, 1=Error Only, 2=On)

This determines whether a "click" will be made after each point is set and after a command has been performed.

Display Attributes?

This allows you to specify whether Attributes will be displayed on the screen. An Attribute is a line of text entered with the Attribute command (\$ key). For more information, see the Attribute command.

"Manipulate Current Layer Only?

This item determines whether the Section and Block commands are used with and affect the current layer only, or all visible layers.

Command Menu On?

The command menu can be disabled or enabled by entering N or Y here.

Dimension Format (1-6)

Any of the following dimension formats can be selected:

- 1 - Decimal, number on the line.
- 2 - Feet/Inches, decimal to .1 inch.
- 3 - Feet/Inches, fraction to 1/32"
- 4 - Inches, fraction to 1/64"
- 5 - Decimal, number above the line, slashes instead of arrows.
- 6 - Decimal, number above the line.

Dynamic Dimensioning Enabled?

If Y is entered here, the dimensions will be "dynamic." This means that the dimensioning information will be changed to correspond to the current units of measurement. If dynamic dimensioning is disabled, the dimensions will be drawn as lines and text and can be edited as such.

Save System Parameters with Drawing?

If Y is entered here, the current system parameters will be saved with the drawing. If N is entered, the system parameters will not be saved with the drawing.

If the system parameters are saved with the drawing, they will become the new system parameters whenever that drawing is retrieved unless there is already a drawing with system parameters on the screen.

Use Binary Drawing File Format?

DesignCAD 3-D normally saves drawings to binary files with a **.DW3** extension. Older versions of DesignCAD 3-D (versions 3 and earlier) used an ASCII drawing file format. These files have an extension of **.DC3**. This option can be used to produce files compatible with earlier versions of DesignCAD 3-D.

The binary file format is more compact and faster to work with than the older ASCII file format. You should use the binary file format unless you need to produce drawing files compatible with earlier versions of DesignCAD 3-D.

Cursor Step Consistent with (1=Screen, 2=Drawing)

If you set the cursor step size and then Zoom the drawing to a different size, one of two things will happen:

1. The cursor step size will change with respect to the drawing
2. The cursor step size will change on the screen.

This option determines which of these will happen when you zoom the drawing. With option 1, the cursor step size is always the same on the screen. Even after you zoom the drawing, the number of cursor steps it takes to move across the screen will be the same.

With option 2, the cursor step size will remain consistent with the drawing rather than the screen. For example, when the drawing is zoomed with a factor of 4, the cursor step size will be 4 times larger on the screen, but the cursor will move the same distance in Drawing Units.

Coordinate System (1=Left-hand, 2=Right-hand)

This option determines whether Z coordinates increase or decrease when the cursor is moved "away" from you. The normal DesignCAD coordinate system is the left-hand coordinate system, where values on the Z axis increase from front to back. With the right-hand coordinate system, Z values increase from back to front.

Mathematical or Geographical Angles

There are two types of angular measurement that can be used with DesignCAD on the status line: mathematical and geographical. With mathematical angles, the degrees progress counter-clockwise, with 0 degrees at the right (3 o'clock) and 90 degrees straight up (12 o'clock). With Geographical angles, the degrees progress clockwise, with 0 degrees straight up (12 o'clock) and 90 degrees to the right (3 o'clock).

Treat Solids as Single Entities? (Y or N)

Several DesignCAD commands such as Erase or Change Color ask you to select an entity by setting a point on it. If you set a point on a Solid object, you can select a single entity or the entire Solid, depending on the setting of this option.

For example, if you draw a box using the Box command, then use the Erase command and set a point on the box, one of two things happen:

1. If Solids are treated as single entities, the entire box will be erased because it is a Solid.
2. If Solids are not treated as single entities, only part of the box will be erased because the box is made up of three surfaces.

The commands affected by this option are Erase, Color Change, and Change Entity. The Block Define command is also affected by this option when only one point is set.

Mouse Sensitivity

The Mouse Sensitivity option lets you set the sensitivity of your mouse. The Mouse Sensitivity can be set from 1 to 10. The larger the number, the slower the cursor moves.

F1 can be pressed at any time for help with the system parameters.

The System Parameter command can be terminated three ways:

- F2** - Use the system parameters but do not save them to disk.
- F3** - Save the parameters as program defaults. This causes these system parameters to be used every time the program begins. It is equivalent to the Configuration Save command.
- Esc** - Return and don't change anything.

Tip: To remove the default parameters and return to the "factory" DesignCAD 3-D settings, erase the file DCAD4.SYS.

Related Topics: Drawing Parameters.

Text Command

| | |
|--------------------|--------------|
| Menu: | NOTES |
| Menu Command: | Text |
| Short Form: | T |
| Keystroke Command: | T |

Point 1: Lower left corner of text.

Point 2: Lower right corner of text (optional).

Point 3: Plane for text (optional).

The Text command is used to add text to a drawing.

With the Text command, you can set 1 to 3 points. If only one point is used, the size and angle of the text is determined by the default text size and the default text angle. These values can be set using the Drawing Parameter command and the Text Size command.

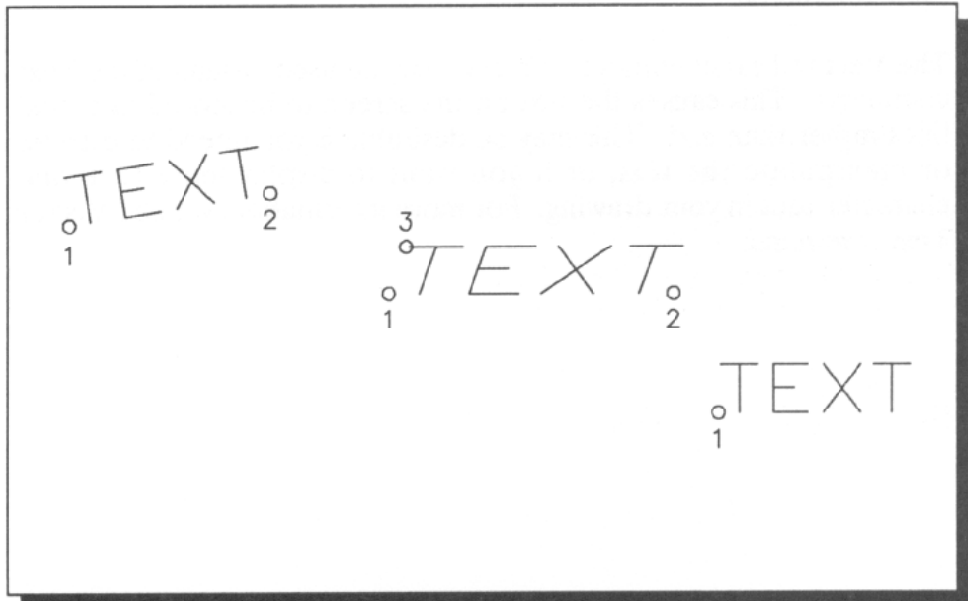


Figure 78 - The Text command uses up to 3 points: the lower left corner, the lower right corner, and a point for the plane.

The second point is used to specify the length and angle of the text. The lower right corner of the text will be positioned at the second point.

The third point, if used, defines the plane on which the text is to be drawn. For example, if the third point is above the first, the text will be upright. If the third point is "behind" the first, then the text will be positioned to be read from above. It will be "flat" on the X-Z plane.

Text can be centered about a point by setting two points in the same location and then pressing the T key. The text will be centered about the two points. The size and angle of centered text is determined by the default text size and angle. Figure 78 gives some examples of the Text command.

Text can be right-justified on a point by setting the first three points in the same location and then pressing **Enter**. The size and angle of right-justified text is determined by the default text size and angle.

The character font used with this command is specified using the Font command.

The Vector Text command (^T key) may be used instead of the Text command. This causes the text on the screen to be stored as actual lines rather than text. This may be desirable if you intend to extrude or manipulate the text, or if you want to display more than one character font in your drawing. For more information, see the Vector Text command.

Tip: You can enter ASCII codes to use characters not found on the keyboard. For example, to use the degree symbol (°) with the text command, hold the **Alt** key down and type **248** on the numeric keypad. Then release the **Alt** key. The character for ASCII code 248 (the degree symbol) will be "typed" onto the screen. You can also use ASCII code 241 to display the "±" symbol.

Example: Place your name into the DesignCAD drawing. Press **T** for text. Then set a point for the text beginning, a point for the text end, and a third point above the second. Enter your name when asked for **Text:** . Your name will be inserted into the drawing between the first two points.

Related Topics: Vector Text, Text Size, Drawing Parameter.

Text Size Command

| | |
|---------------|------------------|
| Menu: | PARAMETER |
| Menu Command: | Text Size |
| Short Form: | TS |

Points: None.

This command is used to specify the default text size. The text size is the height of the text in Drawing Units. To use the command, enter **Text Size**. The current value will be displayed between brackets ("[]"). Enter the new value, or press **Enter** to keep the present value. The default text size can also be set in the Drawing Parameter command.

Example: Set the default text size to 3 Drawing Units tall. Select **Text Size** from the **PARAMETER** menu. When asked for the default text height, enter **3**. Text drawn with one point (or centered text) will be drawn 3 units tall.

Related Topics: Text, Drawing Parameters, Units.

Undo Command

| | |
|--------------------|----------------|
| Menu: | EDIT |
| Menu Command: | Undo |
| Keystroke Command: | Esc key |

Points: None.

The Undo command is used to erase or undo the last line or entity drawn. To use the command, press the **Esc** key. The last entity that was drawn will be erased from the screen. The Undo command can be repeated to erase more than one line or entity.

When you are setting points for a DesignCAD command, you can press **Esc** to erase the last point set. Most DesignCAD 3-D commands can be cancelled by pressing **Esc**.

If you accidentally erase something with this command, it can be replaced using the Unerase command ("!" key).

The Undo command will not undo a Zoom, Rotate, Section, or Block command.

Example: Draw three lines, erase them, and restore them. Draw three lines on the screen. Press **Esc** three times to erase them. Then press **!** three times to unerase the three lines.

Related Topics: Unerase.

Unerase Command

| | |
|--------------------|----------------|
| Menu: | EDIT |
| Menu Command: | Unerase |
| Short Form: | Oops |
| Keystroke Command: | ! |

Points: None.

This command unerases the last item in the drawing that was erased - it places the erased item back into the drawing.

The Unerase command can be used only after one of the DesignCAD erase commands: Erase, Section Delete, Undo, or Block Delete. If anything has been drawn since the last erasure, it will not be possible to unerase anything.

You can use this command repeatedly to unerase more than one entity.

Example: Draw three lines, erase them, and restore them. Draw three lines on the screen. Press **Esc** three times to erase them. Then press **!** three times to unerase the three lines.

Tip: When an entity is erased and then unerased it will no longer be part of a Solid or a Block, even if it was before it was erased.

Related Topics: Undo, Erase.

Units Command

| | |
|--------------------|------------------|
| Menu: | PARAMETER |
| Menu Command: | Units |
| Short Form: | U |
| Keystroke Command: | U |

Point 1: Beginning point of a known distance.

Point 2: Ending point of a known distance.

The Units command is used to set the units of measurement for the drawing. These units of measurement are called Drawing Units. Drawing Units can be feet, miles, centimeters, inches - any unit of measurement that makes sense for the drawing.

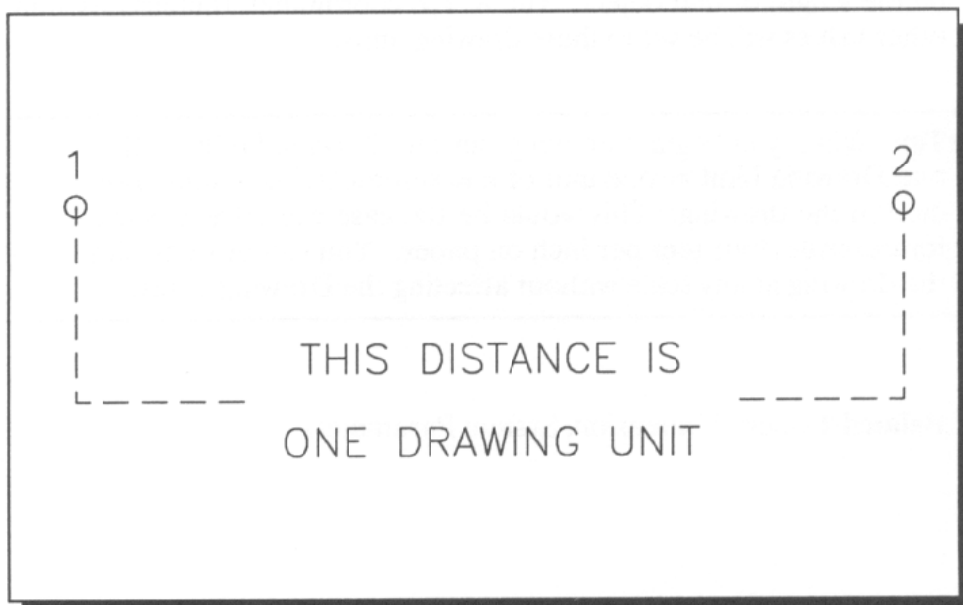


Figure 79 - To set the Drawing Units, set two points a certain distance apart in the drawing. Then enter the number of Drawing Units that distance represents. In this example, you would enter "1".

To use the Units command, set two points a certain distance apart. Then press the **U** key. You will then be asked the number of units that the distance between the points represents. Enter that distance.

The Drawing Units are very important. All DesignCAD 3-D coordinates, dimensioning, and calculations are based on Drawing Units. For example, when you calculate the area or volume of an object, you get the answer in square or cubic Drawing Units. The X,Y,Z coordinates entered or displayed are Drawing Units. When you enter a distance, you enter it in Drawing Units. The Drawing Units must conform to your drawing or these values will be irrelevant.

Example: Set the drawing units based on an 8-inch distance in the drawing. Suppose that you know an object in your drawing is supposed to be 8 inches long. Set points at each end of the object, press **U**, and answer "8" when asked for the number of units. The Drawing Units for the entire drawing will be scaled to inches, based on the length of that object. The X,Y,Z coordinates, dimensions, and other values will be set to these drawing units.

Tip: When you begin a drawing, set the Drawing Units so that one Drawing Unit = one unit of measurement (inch, mm, foot, etc.) in the drawing. This would be the case even if you want, for example, four feet per inch on paper. You can print or plot the drawing at any scale without affecting the Drawing Units.

Related Topics: Dimension, System Parameters.

Un-Working Plane Command

| | |
|---------------|---------------------|
| Menu: | DISPLAY |
| Menu Command: | UnWork Plane |
| Short Form: | UNWP |

Points: None.

The Un-Working Plane command is used to return the drawing to its original orientation after a **Working Plane** command. To use the command, select **UnWork Plane** from the **DISPLAY** menu. The drawing will then be rotated back to its original position. See "Working Plane command" for more information.

Example: Draw a 3 foot line up one of the walls of a pyramid. First, draw the pyramid. To do this, draw a 2-dimensional box on the X-Z plane. Use the Block Extrude command to extrude the box upward with a scale factor of zero. This causes the top of the extruded object to come to a point, forming the pyramid. Use the Working Plane command to rotate one of the pyramid walls onto the X-Y plane. Use the three corners of the pyramid wall for points with the Working Plane command. The pyramid will be rotated so its wall is on the X-Y plane. Then use the Plane Snap command to snap to the wall. Use the Line command with the Point Relative command to draw a line 3 feet long (enter **0,3,0** when asked for the X,Y,Z offsets of the second point). Finally, use the Unworking Plane command to rotate the pyramid back to its original position.

Related Topics: Working Plane.

Vector Commands

| Menu Command | Keystroke |
|----------------------|-----------|
| Vector Arc | ^A |
| Vector Curve | ^C |
| Vector Circle | ^O |
| Vector Text | ^T |

These commands are used the same way as the corresponding drawing commands are. The only difference is that the entities produced by the Vector commands are broken down into short lines (a line entity with several points) and saved as such. This is useful if the drawing (or parts of the drawing) will be scaled or distorted because the curved entities cannot always be distorted.

For example, the Curve command produces a Curve entity made up of a few points. The Vector Curve command produces a Line entity made up of many points.

The Vector Text command can be used to place text of different fonts into the same drawing. You can use the Vector Text command to draw text in one font, change fonts, and then use the Text command to draw text in the second font. The text drawn with the Vector Text command will not change when a new font is loaded.

The items drawn with vector drawing commands use more memory and may be less precise than the corresponding DesignCAD 3-D commands. For example, a Circle entity uses 6 points, while a Line entity produced by the Vector Circle command can use almost 200 points. The vector drawing commands should be used only when necessary.

View Command

| | |
|---------------|-------------------------------------|
| Menu: | VIEW |
| Menu Command: | View, View 2, View 3, View 4 |
| Points: | None. |

The View commands are used to change the view angle and perspective of the drawing. Use "View" to change the view of the drawing on View 1, the main view. Use "View 2" to change the view at the upper left of the screen, "View 3" for the view at the center left, and "View 4" for the view at the lower left of the screen.

To use the View command, select **View** from the VIEW menu or enter **View** from the command line. Your drawing will be displayed on the screen at the current view.

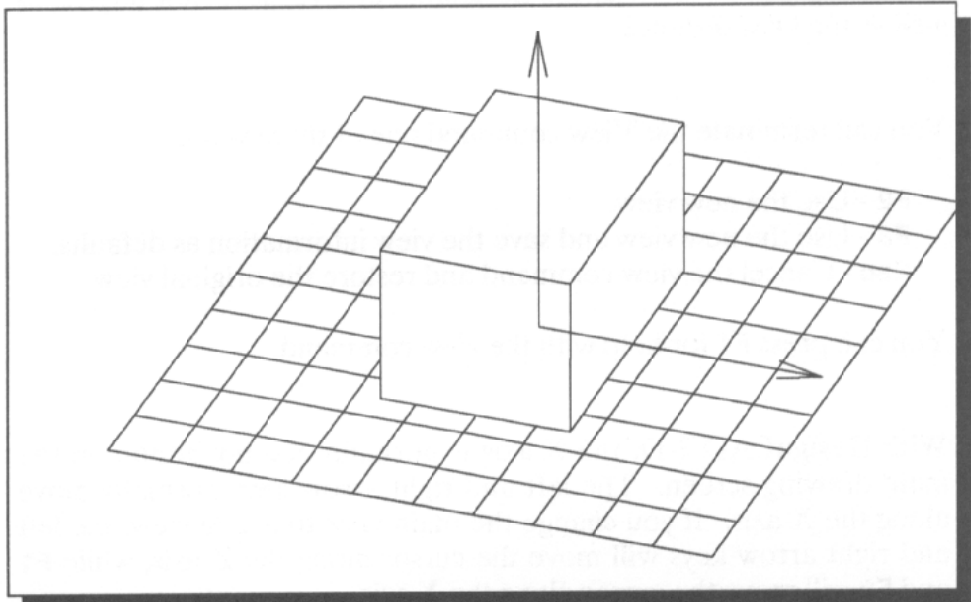


Figure 80 - The View command displays this box and arrows. You can change the view angle with the arrow keys or by pressing the left mouse button while moving the mouse.

Changing the View with the Mouse

To change the view with the mouse, hold down the left mouse button and move the mouse. To change only the horizontal view angle, hold down the **Shift** key together with the left mouse button. To change only the vertical view angle, hold down the **Ctrl** key together with the left mouse button. To change the view distance, or perspective, hold down **Shift**, **Ctrl**, and the left mouse button at the same time.

As the view is changed, the drawing will "change into" a box and two arrows, as shown in Figure 80. The box and arrows will change dynamically as you change the view. When you release the mouse button, your drawing will be shown on the screen.

Changing the View with the Keyboard

Use the arrow keys to change the view angle and the **+** and **-** keys to change the view distance or perspective. As with the mouse, you will see a box and arrows change dynamically as the view changes.

Entering Values from the Keyboard

You can press **F4** to enter the horizontal and vertical view angles, as well as the view distance.

You can terminate the View command one of three ways:

- F2** - Use the new view.
- F3** - Use the new view and save the view information as defaults.
- Esc** - Cancel the view command and restore the original view.

You can press **F1** for help with the view command.

With DesignCAD 3-D, the X axis is normally the horizontal on the main drawing screen. The left and right arrow keys normally move along the X axis. If you change the main view to a side view, the left and right arrow keys will move the cursor along the Z axis, while **F1** and **F2** will move the cursor along the X axis.

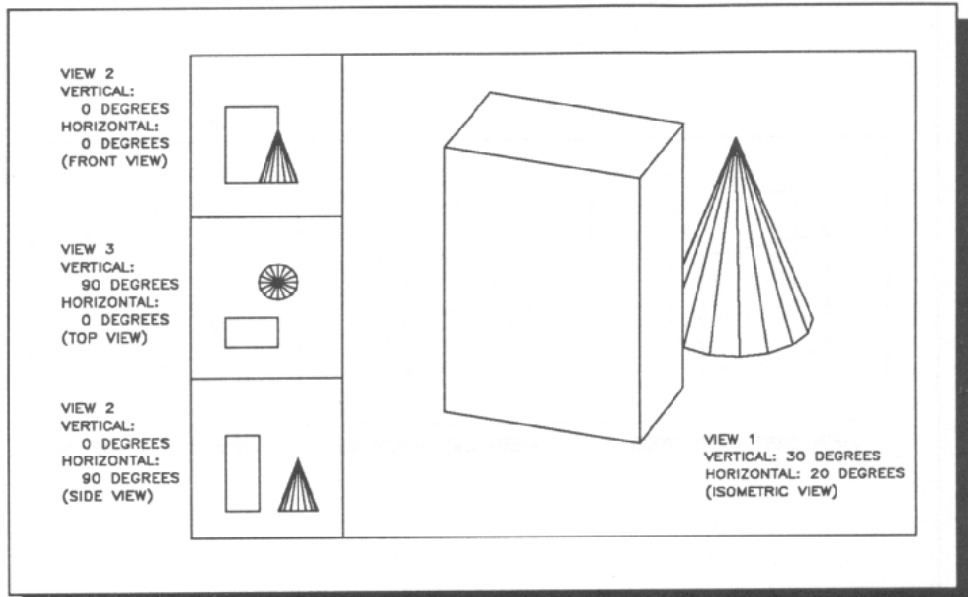


Figure 81 - View Angles.

If you change the main view to a top view, the up and down arrow keys will move the cursor along the Z axis, while **F1** and **F2** will move the cursor along the Y axis.

In other words, the cursor keys always move left, right, up and down on the main drawing screen, regardless of the view angle. The **F1** and **F2** keys move in and out on the main drawing screen, regardless of the view angle.

The mouse movement remains consistent with the screen also. If you move the mouse to the right it will move along the horizontal axis of the main drawing screen, whether it is the X axis or the Z axis. Similarly, if you move the mouse up, it will normally move along the Y axis. However, if the main view of the drawing is a top view then it will move along the Z axis.

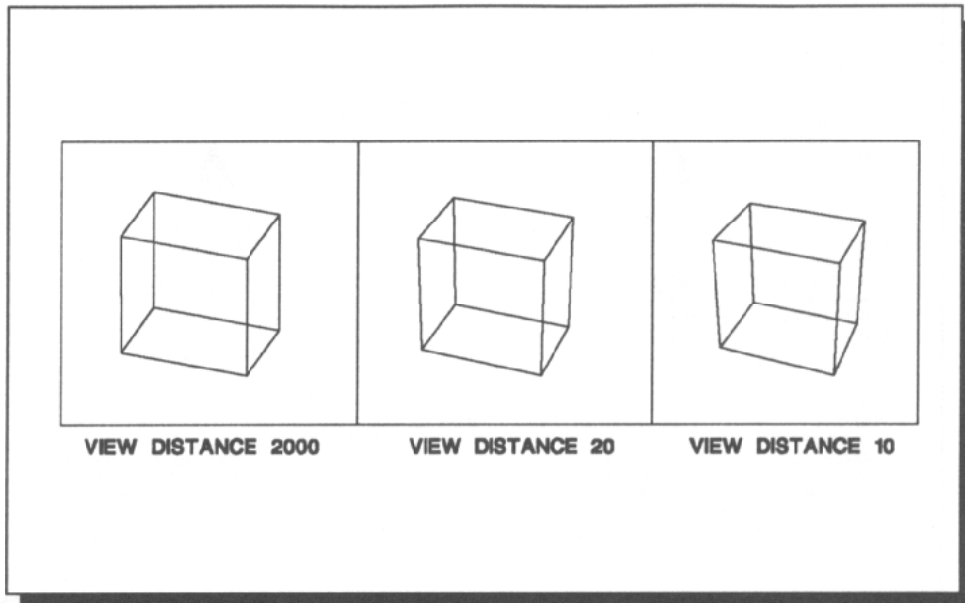


Figure 82 - View Distances.

Tip: You can use the View Save command to save a view to disk. You can read a view from disk with the View Read command. Using these commands, you can save several view configurations and use of any them with any drawing.

Tip: To get rid of the default view configuration and return to the "factory view settings," erase the file AUTOEXEC.VW3.

Related Topics: View Read, View Save.

View Read Command

| | |
|---------------|------------------|
| Menu: | VIEW |
| Menu Command: | View Read |
| Short Form: | VR |
| Points: | None. |

The View Read command is used to read view information from a disk file that was previously saved with the View Save command. The view angles, perspective, light sources, and enabled views are read with this command.

To use the View Read command, select **View Read** from the VIEW menu and enter the name of the view to be read.

You can optionally enter the file name along with the View Read command on the command line:

VR *filename*

where *filename* is the name of the view to be read.

If you press **Enter** when asked for the file name, a directory of views will be displayed on the screen and you can select a view using the arrow keys or a mouse.

This command can be very useful if there is more than one view used with a drawing or set of drawings. For example, you can save a perspective, top, front, and side view, and then select the view by name without having to enter the information in the View command.

Tip: There are two views that come with DesignCAD: **2D** and **3D**. The view **2D** is a front view with views 2, 3, and 4 disabled. It can be used for 2-dimensional drawing. The view **3D** is the normal DesignCAD 3-D view.

Related Topics: View, Lights, View Save, Set View.

View Save Command

| | |
|---------------|------------------|
| Menu: | VIEW |
| Menu Command: | View Save |
| Short Form: | VS |

Points: None.

The View Save command is used to save view information to a disk file so it can be used later with the View Read command. The view angles, perspective, light sources, and enabled views are saved with this command. The shading parameters and colors are not saved with the view.

To use the View Save command, select **View Save** from the **VIEW** menu and enter the name of the view to be saved.

You can optionally enter the file name along with the View Save command on the command line:

VS *filename*

where *filename* is the name of the view to be saved.

This command can be very useful if there is more than one view used with a drawing or set of drawings. For example, you can save a perspective, top, front, and side view, and select the view by name without having to enter the information in the View command.

The file saved is the view name with an extension of **.VW3**.

Tip: The view called AUTOEXEC contains the default view settings for DesignCAD 3-D. To change the default settings, you can save the view to the file AUTOEXEC.

Related Topics: View Read, View, Set View, Lights.

Volume Command

| | |
|---------------|---------------|
| Menu: | SOLIDS |
| Menu Command: | Volume |
| Short Form: | Vol |

Point 1: Set a point on the Solid.

This command is used to calculate the volume and surface area of a Solid. To use the command, set a point on the Solid and select the **Volume** from the SOLIDS menu. The volume and surface area of the Solid will be displayed on the screen.

The Volume command is used to calculate the volume and area of an entire Solid, while the Surface Area command calculates the area of a single entity. For example, the Cone command constructs a Solid object made of a plane entity (the base) and a grid entity (sides). To get the surface area of the entire cone including the base, you would use the Volume command. To get the surface area only of the sides of the cone, you could set a point at the top of the cone and use the Surface Area command.

Example: Find the volume of a cone with 16 facets and a cone with 36 facets. Use the Cone command to draw two cones: one with 16 facets and one with 36 facets. Make each cone with a radius of 4 and a height of 8. Then use the Volume command to find the volume of each cone. Notice the slight difference in the two volumes. The cone with more facets is a more accurate representation of a true circular cone.

Related Topics: Surface Area.

Wall Command

Menu: **SOLIDS**
Menu Command: **Wall**

Point 1: One corner of wall.
Point 2: Opposite corner of the wall.

The **Wall** command is used to draw a wall section. A wall section is a vertical box with a certain thickness.

To use the command, select **Wall** from the **SOLIDS** menu. Set one point in one corner of the wall and a second point in the opposite corner. After the first point is set, a "rubber band wall" will be displayed as the cursor moves about. This makes it easy to see how the wall section or box will be positioned. After the second point is set, the wall will be drawn between the two points.

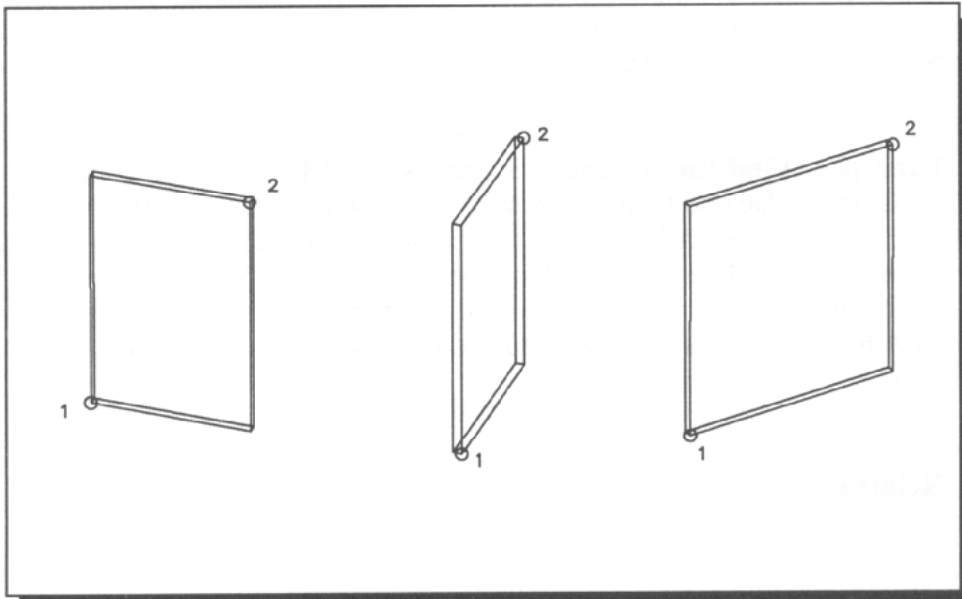


Figure 83 - Set two points in opposite corners of the wall.

The wall thickness is entered in the Drawing Parameter command.

This command is similar to the Box command, except with the Box command the front of the box always lies on the X-Y plane. With the Wall command, the vertical sides of the box are not restricted to the X-Y or Z-Y planes.

Related Topics: Box, Drawing Parameters.

Working Plane Command

| | |
|---------------|-------------------|
| Menu: | DISPLAY |
| Menu Command: | Work Plane |
| Short Form: | WP |

Points 1-3: Determine the plane.

Point 4: The side of the plane to be viewed (optional).

The **Working Plane** command is used to define a working plane and rotate the drawing so that plane lies on the X-Y plane. This allows cursor movement, polar coordinates, mirroring, and other operations to be done on any plane. The **Un-Working Plane** command is used to return the drawing to its original orientation.

To use this command, select **Work Plane** from the **DISPLAY** menu, and set three points to define the new working plane. You can optionally set a fourth point on the side of the plane from which it is to be viewed.

The drawing will be rotated so that the working plane lies on the X-Y plane. You can easily add to or edit items on that plane. This command causes a temporary rotation. When the drawing is saved, it is rotated back to its original position, even if the **Un-Working Plane** command has not been used.

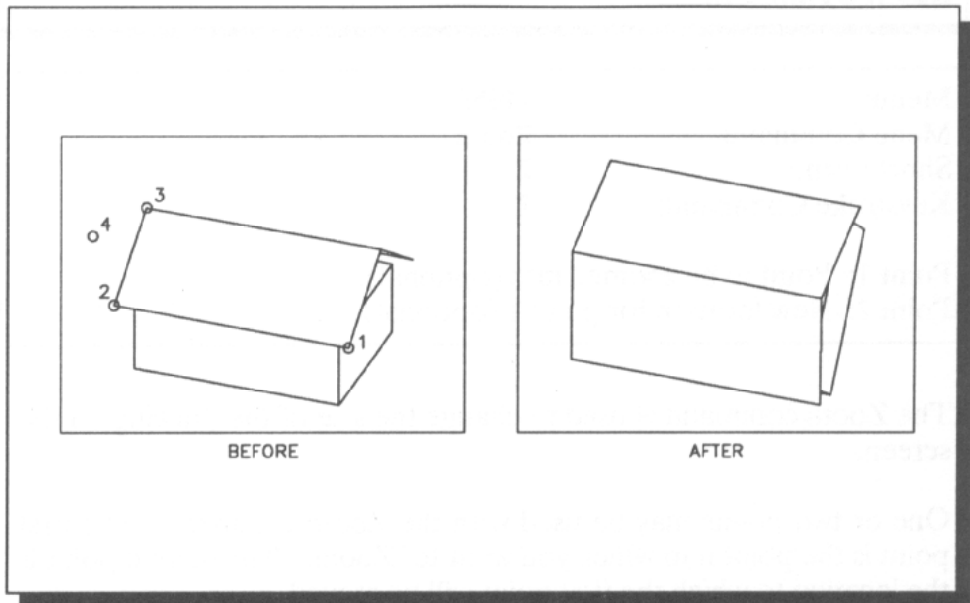


Figure 84 - The Working Plane command rotates the drawing so that the specified plane is moved to the X-Y plane. The first three points define the working plane and the fourth point determines which side of the plane faces the front.

Example: Draw a 3 foot line up one of the walls of a pyramid. First, draw the pyramid. To do this, draw a 2-dimensional box on the X-Z plane. Use the Block Extrude command to extrude the box upward with a scale factor of zero. This causes the top of the extruded object to come to a point, forming the pyramid. Use the Working Plane command to rotate one of the pyramid walls onto the X-Y plane. Use the three corners of the pyramid wall for points with the Working Plane command. The pyramid will be rotated so its wall is on the X-Y plane. Then use the Plane Snap command to snap to the wall. Use the Line command with the Point Relative command to draw a line 3 feet long (enter **0,3,0** when asked for the X,Y,Z offsets of the second point). Finally, use the Unworking Plane command to rotate the pyramid back to its original position.

Related Topics: Un-Working Plane, Rotate.

Zoom Command

| | |
|--------------------|-------------|
| Menu: | VIEW |
| Menu Command: | Zoom |
| Short Form: | Z |
| Keystroke Command: | Z |

Point 1: Point to be zoomed into (optional).

Point 2: New location for point 1 (optional).

The **Zoom** command is used to change the size of the drawing on the screen.

One or two points may be used with the **Zoom** command. The first point is the point into which you want to "Zoom." The second point is the location to which the first point will be moved.

If no points are specified, the center of the screen will be used for the zoom location. If only one point is specified, then that point in the drawing will be moved to the center of the screen.

After the points are set, the zoom factor is entered. The **Zoom Factor** is the relative size you want the drawing to become: 2 for twice as large, .5 for half as large, etc.

A zoom factor of 0 may be specified to scale the drawing to be as large as possible on the screen. When a zoom factor of 0 is used, the drawing will be expanded to the maximum size in each view independently.

To return your drawing to its original size and location, press **Enter** when asked for the zoom factor.

You can use the **Zoom** command with two points and a zoom factor of 1 to "pan" across your drawing. Place the first point at a location on your drawing, and the second point at the location to which you want the first point to be moved. For example, to move the drawing an entire screen from right to left, set the first point at the right edge of the screen and the second point at the left edge. Then press the **Z**

key and enter 1 for the zoom factor. The drawing will "pan" from right to left.

The Cursor Step Size and the Zoom

If you set the cursor step size and then Zoom the drawing to a different size, one of two things will happen:

1. The cursor step size will change with respect to the drawing
2. The cursor step size will change on the screen.

In the System Parameter command, there is an option: "Cursor Step Consistent with (1=Screen, 2=Drawing)"

This option determines which of these will happen when you zoom the drawing. With option 1, the cursor step size is always the same on the screen. Even after you zoom the drawing, the number of cursor steps it takes to move across the screen will be the same.

With option 2, the cursor step size will remain consistent with the drawing rather than the screen. For example, when the drawing is zoomed with a factor of 4, the cursor step size will be 4 times larger on the screen, but the cursor will move the same distance in Drawing Units.

The Digitizer and the Zoom

There are two modes in which the digitizer can operate: "digitizer mode" and "mouse mode." To change from one mode to the other, use the DigiMouse command.

With the digitizer mode, the digitizer movement remains consistent with the drawing when the drawing size changes. With mouse mode, the digitizer movement remains consistent with the screen when the drawing size changes.

Suppose, for example, that the drawing is zoomed by a factor of 4. In digitizer mode, the cursor will move 4 times faster on the digitizer but will cover the same area on the drawing. In mouse mode, the digitizer will cover the same area on the screen but will move 4 times slower with respect to the drawing. The digitizer mode must be used if you are tracing a drawing attached to the digitizer.

Related Topics: Zoom Static, Zoom View, Zoom Window.

Zoom Previous Command

| | |
|---------------|----------------------|
| Menu: | VIEW |
| Menu Command: | Zoom Previous |
| Short Form: | ZP |
| Points: None. | |

The **Zoom Previous** command zooms the drawing to its size and location before the last **Zoom** or **Zoom Window** command.

To use the command, select **Zoom Previous** from the **VIEW** menu. The drawing will be automatically zoomed back to its size and location before the last **Zoom** or **Zoom Window** command.

Example: Zoom the drawing by a factor of 2. Then zoom the drawing by a factor of 4. Finally, select **Zoom Previous** from the **VIEW** menu. The drawing will be zoomed back to a factor of 2, its "previous" zoom level.

Tip: This command is useful in returning the drawing to its size after a **Zoom Window** command, because with the **Zoom Window** command you do not necessarily know the zoom factor of the drawing.

Related Topics: **Zoom Window**, **Zoom**.

Zoom Static Command

| | |
|--------------------|--------------------|
| Menu: | VIEW |
| Menu Command: | Zoom Static |
| Short Form: | ZS |
| Keystroke Command: | ^Z |

Point 1: Point to be zoomed into (optional).

Point 2: New location for point 1 (optional).

The Zoom Static command is the same as the Zoom command except that the internal zoom factor and drawing position do not change, even though the drawing size and location does.

The command can be used to "permanently" change the size or position of a drawing. For example, if you have a drawing that is too small when read from disk, you can use the Static Zoom command with a zoom factor of 2, and then save the drawing. The drawing will then be twice its original size when read from disk.

For more information, see the Zoom command.

Related Topics: Zoom, Zoom View, Zoom Window.

Zoom View Command

| | |
|----------------|--|
| Menu: | VIEW |
| Menu Commands: | Zoom View 1 Zoom View 2 Zoom View 3 Zoom View 4 |
| Short Form: | Z1, Z2, Z3, Z4 |

Point 1: Point to be zoomed in to or out of (optional).
Point 2: New location for point 1 (optional).

The **Zoom View** commands are used to zoom a single view of the drawing. This is often much faster than using the **Zoom** command to zoom all 4 views at once. The **Zoom View** commands work exactly like the **Zoom** command, except only a single view is affected.

For example, to zoom the main view (view 1) by a factor of 2 (make it twice as large), select **Zoom View 1** from the **VIEW** menu and enter a zoom factor of 2 when asked. The three views on the left will not be redrawn.

As with the **Zoom** command, a zoom factor of 0 may be specified to scale the drawing to be as large as possible in the view. To return your drawing to its original size and location, press **Enter** when asked for the zoom factor.

One or two points may be used with the **Zoom View** commands. The first point is the point into which you want to "Zoom." The second point is the location to which the first point will be moved.

If no points are specified, the center of the screen will be used for the zoom location. If only one point is specified, then that point in the drawing will be moved to the center of the screen.

Related Topics: **Zoom**, **Zoom Static**, **Zoom Window**.

Zoom Window Command

| | |
|---------------|--------------------|
| Menu: | VIEW |
| Menu Command: | Zoom Window |
| Short Form: | ZW |

Point 1: Corner of area to zoom.

Point 2: Opposite corner of the area to zoom.

The Window Zoom command is a form of the Zoom command. To use the command, select **Zoom Window** from the VIEW menu. Next, set two points for the "window." The zoom window is a rectangle or window defined by two points in opposite corners.

The portion of the drawing inside the zoom window will be magnified so that it fills the screen. The location and magnification of the zoom depend on the size and location of the zoom window.

Tip: You can use the Zoom Previous command to return the drawing to its size before the Zoom Window command.

Tip: To return the drawing to its original size and location, you can use the Zoom command and press **Enter** when asked for the zoom factor.

Related Topics: Zoom, Zoom View, Zoom Static.

Customizing DesignCAD 3-D

DesignCAD 3-D can be customized to fit almost any application. Several means of customization have been incorporated into DesignCAD, allowing the menus to be changed, commands added, and programs run. It is even possible to redefine keys and mouse/digitizer buttons.

Materials, Views, and Colors

With DesignCAD 3-D, you can create materials, views, and color configurations for different applications.

Suppose, for example, that you are in the cabinet business. You can create materials for dozens of different types of wood, using the Materials command. That way, you can easily show what your cabinets look like with different wood and different stains.

You can save and retrieve commonly used views with the View Save and View Read commands. For example, you could save several views to be used with kitchen drawings. Then, when you make a kitchen drawing, you could load these views one at a time to show the different features of the kitchen.

You can also edit and save color configurations, using the Color Edit and Color Save commands. This allows you to configure DesignCAD 3-D to use the colors you need in your application.

In addition to changing the drawing colors, you can use the Screen Color command to change the colors of your drawing screen.

BasicCAD Programs

BasicCAD is a powerful programming language that allows you to run your own programs within DesignCAD. With BasicCAD you can write a program and execute it as you would a DesignCAD command.

BasicCAD is similar to standard Basic, but it provides the capability to execute DesignCAD commands as well as Basic statements. This

gives BasicCAD the flexibility of Basic together with the power of DesignCAD.

For more information on BasicCAD, see the *BasicCAD Reference Manual*.

Macros

DesignCAD has a complete macro facility, including keystroke macros and an AUTOEXEC Macro that is executed every time DesignCAD is run. Macros are fast and easy to make, but are not as powerful as BasicCAD programs.

The AUTOEXEC macro is very important because it can be used to perform tasks required by your application at the beginning of the DesignCAD session.

Digitizer Menus

A digitizer menu is similar to an icon menu, but it is on the digitizer instead of the screen. It can contain the same commands as icon menus and can also run macros and BasicCAD programs. You can use the DesignCAD 2-D Digitizer Menu Build command to create digitizer menus for use with DesignCAD 3-D.

Command Menu

The DesignCAD command menu at the top of the screen can be modified. To customize the command menu, see Appendix B.

Mouse/Digitizer Button Definition

The commands selected by the mouse or digitizer buttons can be changed using the Button command. You can assign any keystroke command to a mouse or digitizer button. You can even assign a button to run a keystroke macro.

Changing Command Names

You can change command names and keystroke commands by editing the file DCAD6.SYS. The three sections of this file **must** be kept in alphabetical order. For more information on customizing DesignCAD command names, see Appendix F.

DesignCAD 3-D Utilities

Several utility programs are included with DesignCAD 3-D. Here is a brief overview of each program. In the following pages there is a complete description of each program and how to use it.

DCFILES can translate between Hewlett Packard plotter (HPGL) files, ASCII text files, X,Y coordinate files, AutoCAD DXF format files, IGES (Initial Graphics Exchange Standard) format files, GEM files, Postscript format files, Word Perfect Graphics files, and DesignCAD 2D and 3D drawing files.

MATERIAL is a materials list program. It lists and counts the attributes in a drawing file.

CHECKDC is used to display information about your computer system, such as the amount of available memory, display type, etc.

DCSLIDE displays a slide show of DesignCAD 3-D screen images.

SLSETUP is used to configure the DCSLIDE program.

DCDW converts older .DC3 drawing files to .DW3 format.

Two other utilities are covered in the *BasicCAD Reference Manual*: PTEST3 is used to test BasicCAD programs, and ENCRYPT is used to encrypt BasicCAD programs.

You can run any of these programs by entering the program name at the DOS prompt.

DCFILES File Translation Utility

To use the DCFILES utility, select the File Convert command in the DesignCAD 3-D FILES menu, or enter DCFILES at the DOS prompt in your DesignCAD directory.

The following menu will appear:

Select the Input File Format

1. HPGL
2. TEXT
3. XY Coordinates
4. IGES 2D
5. IGES 3D
6. DXF 2D
7. DXF 3D
8. DesignCAD 2D
9. DesignCAD 3D
10. DesignCAD 2D ASCII
11. DesignCAD 3D ASCII

12. Return to DesignCAD
13. Exit the Program.

When any of options 1 through 11 are selected, a corresponding menu appears to select the Output File Format:

Select the Output File Format

1. DesignCAD 2D (DW2)
2. DesignCAD 3D (DW3)
3. DesignCAD 2D ASCII (DC2)
4. DesignCAD 3D ASCII (DC3)
5. DXF
6. Word Perfect Graphics (WPG)
7. IGES
8. GEM
9. Postscript
10. Renderman

11. Return to DesignCAD.
12. Exit the Program.

The DCFILES utility translates files from one of the formats listed in the first menu to one of the formats listed in the second menu.

After selecting the file formats, you will be asked if you want to translate a single file or multiple files. You can translate up to 50 files at a time.

If you translate a single file, you will be asked for the input and output file names. To select the file from a directory listing, press **Enter** when asked for the input file name.

If you choose to translate multiple files, you can select the files from the directory listing. The output file names will be the same as the input file names, with a different extension.

Tip: It is not necessary to use the DCFILES program to translate a DesignCAD 2-D drawing to DesignCAD 3-D and vice versa. You can use the Retrieve command to read a drawing into DesignCAD 3-D, and the Save 2D command to save a 2-D drawing from DesignCAD 3-D.

Tip: It is not necessary to use the DCFILES program to translate a drawing from the older ASCII DesignCAD file format (.DC2 and .DC3) to the newer Binary DesignCAD file format (DW2 and .DW3). You can use the DesignCAD 3-D Retrieve command to read .DC2, .DC3, .DW2, and .DW3 files. You can use the DesignCAD 3-D Save command to save .DC3 and .DW3 files. You can use the Save 2-D command to save .DC2 and .DW2 files.

Here is a description of the file formats supported by DCFILES:

HPGL

An HPGL file is a file created using Hewlett-Packard Graphics Language, or more simply, an HP Plotter file. You can create an HPGL file by selecting an HP plotter in DesignCAD and then plotting to a disk file. A large number of other software programs also allow you to create HPGL files, from large CAD/CAM systems to business graphics packages for personal computers - virtually any program that supports an HP plotter! You can then convert the HPGL file to DesignCAD format and read it into DesignCAD.

If your software does not allow you to plot to a disk file instead of directly to the plotter, there are software packages available that allow you to redirect plotter output from a serial or parallel port to a disk file.

ASCII TEXT

An ASCII text file is an ASCII file created using a word processor or other program. DCFILES can read an ASCII file and translate it into lines of text in a graphics file.

This makes it possible to merge text with your drawings.

XYZ Coordinates

This input option allows you to translate an ASCII file of X,Y or X,Y,Z coordinates into a different file format. An XYZ Coordinate file should have each set of coordinates on a separate line, and each number must be separated by one or more spaces or a comma (,). Each line must be terminated by a carriage return and optionally a line feed character.

At the beginning of the translation, you will be asked for the color and whether to connect the points with a straight line, a curve, or to place small squares or circles on the points. If you select small squares or circles, you will be asked for a Mark Size, which determines the size of the squares or dots in the output file.

If the lines in a file contain three coordinates per line (X, Y, and Z), the file can be translated into a 3D file. The Z coordinate may be ignored if you select an output file format that does not support three dimensions (such as DesignCAD 2D or Postscript).

A blank line separating a series of XY or XYZ coordinates will indicate the end of a line or curve and the beginning of another. For instance, the following string of XY coordinates would be translated into two separate lines or curves:

```
3,5  
5,3.6  
6,7.5  
10,11  
  
-5,3.7  
-3.6,7  
-2.5,-1  
0,0
```

IGES

IGES (Initial Graphics Exchange Standard) is a standard format supported by most larger CAD systems. This makes it possible to transfer DesignCAD drawings to other CAD systems and vice versa.

When you select IGES as the output file format, you will be asked if you want a carriage return at the end of each line. If the program in which you will be using the IGES file needs a carriage return at the end of each line, enter **Y**. When importing an IGES file, DCFILES will work with either type of file, with or without carriage returns.

The following IGES entities are accepted by the DCFILES utility:

| |
|-----------------------------|
| 100 - Arc |
| 104 - Conic Arc |
| 106 - Copious Data |
| 110 - Line |
| 112 - Parametric Spline |
| 116 - Point |
| 124 - Transformation Matrix |
| 212 - General Note |
| 214 - Arrow (Leader) |
| 308 - Subfigure Definition |
| 408 - Subfigure Insertion |
| 410 - View |

DXF

The DXF file format allows transfer of DesignCAD drawing files into AutoCAD and vice versa. In addition to allowing you to move files back and forth between DesignCAD and AutoCAD, DXF conversion makes DesignCAD compatible with many third-party software packages that utilize the DXF format.

With the IGES and DXF file formats, you can transfer DesignCAD drawings to and from most other CAD systems.

GEM

This option allows translation of any of the input file formats into the GEM format, which is supported by Ventura Publisher and other graphics and desktop publishing software.

Postscript

This option allows translation of any of the input file formats into Encapsulated Postscript format, which is supported by Postscript printers, as well as many popular desktop publishing programs.

Renderman

This option allows translation of any of the input file formats into the format used by the Renderman software by Pixar.

Word Perfect Graphics

The WPG file format can be used to transfer DesignCAD 3-D drawings to Word Perfect. You can also output DesignCAD 3-D drawings to Word Perfect by creating an HPGL file using the Plot command.

CHECKDC System Check Utility

The CHECKDC utility displays information about your computer configuration, such as your system's memory, operating system, type of CPU, etc. It also displays the contents of your AUTOEXEC.BAT and CONFIG.SYS files.

If you need to call DesignCAD technical support, please run this utility and have a printout of the results handy when you call.

To run CHECKDC, enter **CHECKDC** at the DOS prompt. The following information will appear on your screen:

```
DOS Memory: 640 K
Free DOS Memory: 503 K
Number of Parallel Ports: 1
Number of Serial Ports: 2
DOS Version: 5.0
Total EMS: 2960 K
Available EMS: 1576 K
Total XMS: 4096 K
Available XMS: 256 K
CPU: 80486
Math Coprocessor: Installed.
Mouse: Installed.
Display / Graphics Adapter: VGA, 640x480, 16 Color
DesignCAD Release Date: 1/ 3/1992
DesignCAD Serial Number: 1001 0373 8672
Configuration Date: 2/19/1992
```

Note: If you have a high resolution VGA graphics adapter, you will notice that CHECKDC reports the "VGA 640x480, 16 Color" anyway. This is normal, and does not indicate that you will not get the higher resolution in DesignCAD itself. CHECKDC reads certain key information from your graphics adapter to determine the minimal configuration. Once CHECKDC determines that the adapter is a VGA, it doesn't pursue it any further.

If you are having trouble finding the proper selection for your graphics adapter in SETUP3D, you can usually select the one displayed in CHECKDC, even if it is a lower resolution than your adapter is capable of. Run it that way until you find out the proper selection.

If you press a key, a new screen will display a listing of your CONFIG.SYS file. If you press another key, a new screen will display a listing of your AUTOEXEC.BAT file. This information may be important in troubleshooting many problems that concern not enough memory or memory conflicts.

If CHECKDC reports that you have less than 500K of Free DOS Memory, you may need to check your CONFIG.SYS and AUTOEXEC.BAT to see what memory resident programs are being loaded when you start up the system.

To print this information, press **Shift** and **Print Scrn** (Print Screen) when the information is displayed on the screen. This causes the current screen information to be output to the printer. Then do the same with the second and third screens.

DCSLIDE Slide Show Utility

The DCSLIDE utility is used to display a series of screen images, or a "slide show," on your screen. The screen images can be made using the DesignCAD 3-D Slide Show command or the Screen Save command.

The Slide Show command automatically creates a sequence of screen images and a slide show file. You can add pauses, waits, etc. to the slide show by editing the slide show file and inserting the DCSLIDE directives between screen image names.

Example: You have created a slide show called **TEST** and you want to add a 1 second pause between each slide. Use a word processor, a text editor, or the DOS editor EDIT to modify the file **TEST.SHW** and add the line "**=PAUSE 1**" after each slide name in the file. (The entire file will be a list of slide names when you begin.)

Another way to make a slide show is to edit a file and make a list of "slides" or screen image files to be used in the slide show.

For example, suppose you want make a slide show of 4 screen image files called TEST1, TEST2, TEST3, and TEST4. Make an ordinary text file called TEST.SHW with the following lines:

```
TEST1
TEST2
TEST3
TEST4
```

The slide show file is called TEST.SHW . .SHW is used for the file type of DCSLIDE slide show files. The file type of the screen image files is .PCX . This can be omitted in the slide show file.

Now run DCSLIDE. When asked for the slide show name, enter **TEST** or **TEST.SHW**. The screen image files listed in the file TEST.SHW will be displayed on the screen in order. They will all be repeated until the Escape key is pressed to terminate the slide show.

Within the slide show file you can put DCSLIDE directives, such as "**=PAUSE**" for a short wait before the next slide, or "**=WAIT**" to wait

until a key is pressed before continuing with the slide show. To do this, edit the slide show file (the file that ends with .SHW) with a text editor or word processor, and insert "=PAUSE" or "=WAIT."

The DCSLIDE directives include:

| | |
|--------------------|---|
| =CLEAR | Clear the screen |
| =END | Stop the slide show (prevents repeating) |
| =PAUSE <i>n</i> | Pause for <i>n</i> seconds. If <i>n</i> is omitted then a 1-second pause is assumed. |
| =WAIT | Wait until a key is pressed before continuing. (A message to "press any key" is displayed.) |
| =FLIP <i>n1 n2</i> | This directive sets the update page and the display page on a 256K EGA display. <i>n1</i> is the update page (0 or 1), and <i>n2</i> is the display page. This makes it possible to move to the next slide instantly on a 256K EGA. |

The following directives are for serial input/output, and can be used to communicate with video editing equipment:

| | |
|-------------------------|--|
| =SETCOM1 7 0 1 8 | Set the the serial communications parameters for serial port COM1. The first number is the baud rate (7 = 9600, 6 = 4800, 5 = 2400, etc.). The second number is the parity (0 = none, 1 = odd, 3 = even). The third number is the number of stop bits, and the fourth number is the word length. |
| =SENDCOM1 <i>string</i> | Output <i>string</i> to serial port COM1. |
| =WAITCOM1 <i>c</i> | Wait for character <i>c</i> to be input from serial port COM1. If <i>c</i> is omitted, wait for any character to be input from COM1. |
| =SETCOM2 | Same as SETCOM1, but for COM2. |
| =SENDCOM2 <i>string</i> | Same as SENDCOM1, but for COM2. |

=WAITCOM2 c Same as WAITCOM1, but for COM2.

The DCSLIDE program uses the DesignCAD 3-D configuration produced by the program SETUP3D. You can run DCSLIDE on a different directory or computer than the one DesignCAD is on, but the program SLSETUP needs to be run first to select the video driver for the DCSLIDE program.

The DCSLIDE and SLSETUP programs can be freely distributed by any registered DesignCAD 3-D user. You can create a slide show and distribute it with the DCSLIDE program to friends, neighbors, relatives, customers, and business associates, free of charge! This, of course, does NOT include the distribution of the other DesignCAD 3-D programs.

Materials List Utility

The DesignCAD Materials List Program provides the capability to extract all the Attributes in a drawing, count them, and output the list to the screen, a printer, or a disk file. To use the program, enter **MATERIAL** from the DOS prompt.

You will be asked to select one of the following:

1. List the Materials in a Drawing.
2. Display DesignCAD Drawings on Disk.
3. Stop the Program.

To create a materials list for a drawing, select option 1. You will be asked for the drawing name and where you want the output to be directed - screen, printer, or disk.

Screen and printer output is similar in format. The disk output contains the following on each line:

1. The number of occurrences of each attribute
2. a Comma
3. The attribute itself

After you have output the materials list, select option 4 to return to the main menu. You can then produce a materials list for another drawing or exit the program.

Attributes

The Materials List program extracts Attributes from a DesignCAD drawing. The Attributes are entered into the drawing using the DesignCAD Attribute command. An Attribute is actually a line of text that is associated with a particular position in the drawing.

An Attribute can be assigned to a particular object in the drawing, such as a window, door, or electrical component. You can give every "important" object in your drawing an attribute, using identical

attributes for identical objects. It is sometimes convenient to put an attribute with a symbol before it is used, so that when it is retrieved from disk or copied in the drawing the attribute will automatically be reproduced.

After the drawing has been completed, with all the attributes assigned, the Materials List program is used to produce a list of all the attributes in the drawing. Remember - the Materials List program only lists and counts attributes. The important part is the correct placement of attributes within your drawing.

Using BasicCAD with the Materials List Program

When you are creating attributes to be used with other software, the format of the data within each attribute may be fixed and inconvenient to enter via the Attribute command.

It is relatively easy to write a BasicCAD program that asks for several items of information and then create an attribute based on that information. Using the BasicCAD program can make it easier to enter the attributes and can reduce entry errors substantially.

Using Other Software with the Materials List Program

The Materials List program can list the Attributes in a drawing and output the list to a disk file. This makes it possible to use other software to read the disk file. For example, a database system can get part numbers from the attributes, look up current costs, and produce a cost estimate for the drawing.

The disk file is in the following format:

| |
|---|
| Columns 1-6: Quantity, right justified, padded with spaces. Column 7: Comma. Column 8+: Attribute string. |
|---|

The Attributes are listed in alphabetical order. The Attribute string is terminated with a line feed and carriage return. The Attribute can be no longer than 80 characters.

DCDW Drawing Conversion Utility

The DCDW utility is used to convert the older .DC3 drawing file format to the newer .DW3 format. DesignCAD 3-D versions 3 and earlier support the .DC3 drawing file format. DesignCAD 3-D version 4 supports the .DW3 format.

To convert .DC3 drawings to .DW3 drawings, use the program DCDW. Enter **DCDW** from the DOS prompt, and enter the file specification when asked. The file specification can be a file name or a file name with the "wildcard" symbols "*" or "?". You can also enter the file specification from the DOS command line.

For example, to convert all .DC3 drawing files in the current directory to .DW3 format, enter:

DCDW * or **DCDW *.DC3**

All the files with the .DC3 extension will then be copied to the newer .DW3 drawing format. The original files will not be erased. You should verify that the files were converted successfully before erasing the original files.

Appendix A: DesignCAD 3-D Drawing File Format

This section contains advanced information not normally needed by DesignCAD 3-D users. Unless you have a specific need for this information, you may skip this section.

Two drawing file formats are supported by Version 4 of DesignCAD 3-D: an ASCII file format (DC3 format) and a Binary file format (DW3 format). The DC3 file format is compatible with DesignCAD 3-D version 3 and earlier. The DW3 drawing file format is more compact and faster than the DC3 format, and should be used in the normal operation of DesignCAD 3-D.

The file format can be selected by specifying the file extension .DW3 or .DC3 in the save command, or by selecting the Binary or ASCII file format in the System Parameters command.

DW3 File Format

The DesignCAD DW3 file is a binary file containing header information and "entities." The header contains the following information:

The header is in the following format (sizes are in bytes):

| <u>size</u> | <u>description</u> |
|-------------|---|
| 2 | Drawing type: 1000 = 2-D, 1100 = 3-D |
| 2 | DesignCAD Version Number |
| 2 | Data type - 4 for single precision, 8 for double precision (Only single precision currently supported) |

The entities are in the following format:

| <u>size</u> | <u>description</u> |
|-------------|--|
| 2 | Entity Code |
| 2 | Number of bytes in this entity (not including the entity code and this field) |
| n | information for the entity |

The first entity in the drawing is the "Drawing Size" entity. It has the following format:

| | |
|---|---------------------------------|
| 2 | Drawing Size Entity Code - 1200 |
| 2 | Number of bytes following - 24 |
| 4 | X drawing minimum |
| 4 | Y drawing minimum |
| 4 | X drawing width |
| 4 | Y drawing height |
| 4 | Z drawing minimum |
| 4 | Z drawing depth |

Parameter Entities

Following the Drawing Size entity are the Parameter entities. The following parameter entities may appear in the drawing:

| entity | size | parameter(s) |
|--------|------|--------------|
|--------|------|--------------|

| | | |
|------|----------|---|
| 1300 | 2 | Number of digits of accuracy |
| 1301 | 2 | Display attribute flag |
| 1302 | 2 | Font type |
| 1303 | 2 | Snap grip on/off |
| 1304 | 2 | Beep type |
| 1305 | 2 | Current layer manipulation |
| 1306 | 2 | Dimension type |
| 1307 | 2 | Dynamic dimensioning on/off |
| 1308 | 4 | Cursor movement |
| 1309 | 4 | Drawing unit scale |
| 1310 | 4 | Snap grid size |
| 1311 | 4 | Text size |
| 1312 | 4 | Text angle |
| 1313 | 4 | Drawing unit per inch on output |
| 1314 | 4 | Drawing unit size |
| 1315 | 16 | View distance (4 views) |
| 1316 | 16 | View vertical angle (4 views) |
| 1317 | 16 | View horizontal angle (4 views) |
| 1318 | 32 | Light source horizontal angle (8 sources) |
| 1319 | 32 | Light source vertical angle (8 sources) |
| 1320 | 32 | Light source intensity (8 sources) |
| 1321 | 8 | Light sources on/off (8 sources) |
| 1322 | 252 | Background Light (63 layers) |
| 1323 | 252 | Diffuse reflection (63 layers) |
| 1324 | 252 | Specular exponent (63 layers) |
| 1325 | 252 | Specular reflection (63 layers) |
| 1326 | 252 | Texture index (63 layers) |
| 1327 | 63 | Texture type (63 layers, 1 per layer) |
| 1328 | 787 | Color mask and palette |
| 1340 | Variable | Layer names (63 ASCII strings) |

Entity 1328 contains the color mask and palette. The data section of this entity contains the following:

| <u>size</u> | <u>description</u> |
|-------------|--|
| 2 | Number of colors in the palette |
| 1 | Number of basic colors, normally 15 |
| 3*15 | R,G,B values for basic colors |
| 3*256 | R,G,B values for 256 colors used in drawing. |

Drawing Entities

The drawing entities are in the following format:

| <u>size</u> | <u>description</u> |
|-------------|--|
| 2 | Entity Number |
| 2 | Number of Bytes in entity attributes section |

Attributes Section

| | |
|---|---|
| 2 | Number of points |
| 6 | Color - R,G,B values |
| 2 | Solid Number (0 if entity is not a Solid) |
| 2 | Block Number (0 if entity is not a Block) |
| 2 | Line type (2-D only) |
| 2 | Line width (2-D only) |

Points Section

| | |
|-----|--|
| n*4 | X coordinate for all points in the entity |
| n*4 | Y coordinate for all points in the entity |
| n*4 | Z coordinate for all points in the entity (3-D only) |
| 2 | Number of bytes of text (0 for most entities) |
| n | text (for text entity, attributes, etc.) |

Points Used with Drawing Entities

Entity type 0: 1 or 2 points for the handles of the drawing.

Entity types 1, 4, 31, and 33, Line, Curve, Plane, and Grid Line - Up to 200 points through which lines are drawn.

Entity type 3 - Text

1. Lower left corner of text.
2. Lower right corner of text.
3. Defines the plane on which the text lies, with 1 & 2.

Entity type 15, Attribute - 1 point for location.

Entity type 16 - Circle, Circular Arc:

1. Center
2. Beginning Point
3. Ending Point
4. Minimum X,Y of circle or arc (Optional).
5. Maximum X,Y of circle or arc (Optional).
6. Plane on which the circle lies

If points 2 and 3 are the same, a circle is drawn.

Entity type 18 - Dimension, linear

1. First point for measurement, text.
2. Second point for measurement, text.
3. First point for leader line 1. (optional)
4. Second point for leader line 1. (optional)
5. First point for leader line 2. (optional)
6. Second point for leader line 2. (optional)

Entity type 19 - Dimension, angular

1. Center of angle.
2. First point for "arc."
3. Second point for "arc."
4. First point for leader line 1. (optional)
5. Second point for leader line 1. (optional)
6. First point for leader line 2. (optional)
7. Second point for leader line 2. (optional)

Entity type 21 - Layer

The number of points represents the layer number of subsequent drawing entities. No points are actually used.

Entity type 23 - Layer Names

One layer name for each "point" designated in the entity line.
No points are actually used.

Entity type 24 - Arrow

One 1 to n-2 points for the arrow (the arrowhead is at n-2).
The last two points determine the text height for the arrowhead.

Entity type 32 - Grid

No points are used - Number of Points entry on command line tells how many grid lines follow.

DC3 File Format

The DC3 drawing file is an ASCII file, with the data present in character format. Each "record" in the file is actually a line in a text file. There are four types of records, or lines, in the file:

1. Header line
2. Command line
3. Point line
4. String line

The Header line gives the X, Y, and Z range of the drawing. The Command line tells what type of drawing entity, how many points are to follow, and the color of the entity. The Point and String lines contain point coordinates and strings, respectively.

The Header Line

The Header Line contains 7 numbers:

1. The minimum X coordinate in the file.
2. The minimum Y coordinate in the file.
3. The horizontal length of the drawing.
4. The vertical height of the drawing.
5. Reserved.
6. The minimum Z coordinate in the file.
7. The depth of the drawing.

The coordinates and lengths are given in "dots" on the screen. These values are pertinent only within the drawing file. The drawing may change location, size, etc. when it is retrieved. The "Drawing Units," or user units of measurement for the drawing, are defined in the system parameter section of the drawing file.

The Entity Line

The entity line contains 5 numbers:

1. The type of entity:
 - 0 = Handles
 - 1 = Line
 - 2 = Oval
 - 3 = Text
 - 4 = Curve
 - 5 = Arc
 - 6 = Paint
 - 15 = Attribute
 - 16 = Circle, Circular Arc
 - 17 = Hatch
 - 18 = Dimension - Linear
 - 19 = Dimension - Angular
 - 20 = System Parameters - Old Format
 - 21 = New Layer
 - 22 = Text Arc
 - 23 = Layer Names
 - 24 = Arrow
 - 31 = Plane
 - 32 = Grid
 - 33 = Grid Line
 - 40 = Drawing Unit Size and Cursor Step Size
 - 41 = System Parameters
 - 72 = Diameter or Radius Dimension
2. The number of points to follow. (0-200)
 - (or the layer number, if entity = 21)
 - (or the number of grid lines to follow, if command = 32)
3. The line pattern scale. (1=normal)
 - (or the hatch pattern scale)
4. The line width.
5. The dimension format - used only in dimensions.
5. The line type (0-8)
6. The color (1-15).
7. Solid value - 0 if not part of a Solid.

Following the entity line are the point lines and the string lines. There is one point per line, and one string per line. Strings are found on Text, Text Arc, Attribute, and Hatch entities.

If the entity = 20, then the drawing parameters will follow, terminated by an asterisk (*). If the entity is 40 or 41, then the "number of points" represents the number of lines that follow on that entity. Each line may have more than one value, and the values do not represent points.

The Point Line

The point lines follow the command line. Each line contains three numbers: the X, Y, and Z coordinates of the point on the screen. These values will change when the drawing is retrieved according to the size, location, and angle of the Retrieve command.

There can be no more than 200 points per command. A line with more than 200 points may be used by splitting it into two or more commands.

The String Line

The string line contains a string of text. It follows the point lines. The string line is used only with the Text, Text Arc, Attribute, and Hatch entities. Be sure that the string is terminated with a carriage return character, because trailing blanks will be used in determining the string length and may cause undesirable results. The maximum string length is 80 characters.

System Parameters

Following the System Parameter command line are the system parameters. These consist of:

- Cursor Increment
- Reserved
- Reserved
- Snap Grid Size
- Number of digits accuracy
- Default Text Size
- Default Text Angle
- Drawing Units per Inch on Output
- Reserved
- Reserved
- Display Attributes Flag
- Drawing Unit Size
- Character Font
- Reserved
- Snap Grid On or Off
- Menu On or Off
- Reserved
- Reserved
- Sound (0, 1, or 2)
- Manipulate other than current layer flag
- Reserved
- Dimension format
- Dynamic dimensioning on or off
- View Distance - 4 values, 1 for each view
- View X Angle - 4 values, 1 for each view
- View Y Angle - 4 values, 1 for each view
- Horizontal Light Angles 1-4
- Horizontal Light Angles 5-8
- Vertical Light Angles 1-4
- Vertical Light Angles 5-8
- Light enable (1) / disable (0) - 8 values

An asterisk ("*") terminates the system parameters.

A very important system parameter is the Drawing Unit size. This value is usually used by programs that input DesignCAD 3-D drawing

files. To get the user coordinates (inches, centimeters, miles, etc.) divide the X,Y,Z points by this value.

Point Definitions

The points used with drawing entities in the DC3 file format are the same as those used in the DW3 format.

Appendix B: Customizing the Command Menu

The command menu is the list of commands that can be pulled down from the top of the screen. You can modify the supplied command menu to suit your particular needs, or you can create an entirely new one. You can modify the command menu with a text editor or a word processor that outputs in pure ASCII format. **Be sure to make a backup of the file before you modify it!**

To change the menu file, all you need to do is edit the file DC3MENU.MNU. The file will look something like this:

```

16 8
`
=0=====
`FILES
F
Show `Info
0
`Retrieve          F9
0
`Save             F10
0
Save `2D          F4
0
`Directory        F7
0
Set `Path
0
-----
-1
P`lot             F6
0
Pri`nt           F5
0
`File Convert
0
-----
-1
`BLOCKS
3
-----

```

```

Screen `Read
0
Screen `Save
0
Slide S`how
0
-----
.
.
.
=====

```

The format of the file is:

Total Number of Menus; Number of Menus across the top line
 Special Highlight Symbol
 Separator Line (=====)
 Menu Name with highlighted letter preceded by '
 Which letter activates the menu
 Entry Name 1
 Entry Value 1
 Entry Name 2
 Entry Value 2

```

.
.
.

```

The first of the two numbers at the top, the "16" in this example, tells how many menus there are going to be in the command menu, such as the FILES menu, the PARAMETER menu, the SHADE menu, etc. The second number, the "8" in this example, tells two things. It is the number of menus in one line and consequently how many lines will be in the command menu. Since 8 is half of 16 it will take two lines of menus to get all 16 on the screen.

Below this is a special symbol, the backwards apostrophe located above the **Tab** key on your keyboard. This symbol is used with the menu names to highlight one character in the menu name.

Next you will see a row of equal (=) signs and a number after the first equal sign. This row shows where one menu starts and another ends.

Each menu has to have this row of equal signs before and after it. The number after the first equal sign is just a programming note of which menu this is and carries no significance in the program.

Under the row of equal signs you will see the name of the menu, FILES in this example and the highlight symbol (' key). The symbol will always be located before the letter that is to be highlighted. In this example, the **F** key will be highlighted since the symbol is in front of the **F**. The title of the menu and the symbol together cannot be more than 10 characters. The highlight symbol is also used in the command names.

Next, under the name of the menu is a letter. This determines which keyboard character activates the menu. In this example the letter F is used, so pressing the **Alt** key plus the **F** key will activate this menu.

Next you will see the names of the commands and their single keystroke activators, if they have one. For example, **Retrieve F9** is used for the Retrieve command. It is important that the keystroke information be at least two spaces from the name of the command. If not, DesignCAD thinks that the keystroke is part of the command name and will be unable to find the command by the strange name. The command name can be no longer than 12 characters.

Note: You cannot change the command names and keystroke commands by editing the DC3MENU.MNU file. You must change the file DCAD6.SYS as well to change command names.

Under the command name is a zero (0). This tells DesignCAD that this is a command and to check another file, DCAD6.SYS, for how to operate this command.

The solid line (___) is just there for appearances. The -1 on the following line is used to mark this as a "non-command." This -1 means that the line is to be ignored by DesignCAD and you cannot select this line as a command.

In the example above you will see a name in all capital letters with an arrow beside it. This is the name of a sub-menu. The negative

number before the entry designates this as a sub-menu name. The submenu itself is enclosed between two dashed lines.

Following the submenu name is a number telling how many entries are in the submenu, followed by the submenu itself. The submenu itself is enclosed between dashed lines.

At the very end of the menu is the row of equal signs, signifying that either this menu is over and another is about to begin or this is the end of DC3MENU.MNU.

Adding a BasicCAD Command to the Command Menu

To add your BasicCAD program to the command menu, edit the file DC3MENU.MNU with a text editor or a word processor. Move to the location in the menu where you want to add the BasicCAD program. You can tell which menu to use by the menu title name. If you want to add the program to a sub-menu within this menu, then move to that sub-menu.

Now move to the spot on the menu or sub-menu where you want the file to be listed. This spot must be after a zero or -1, **not** between the zero and command name. Enter the name of the program and then on the line following, enter a zero (0). Save DC3MENU.MNU as ASCII text (DOS text on some word processors) and try it out.

When you use your program on the command menu, the file DCAD6.SYS will be searched for a command by that name. When it can't be found, the BasicCAD program with that name will be run.

Here is what the FILES menu would look like if you added the BasicCAD program "Hello" to the menu:

```

16 8
\
=0=====
`FILES
F
`HELLO
0
Show `Info
0
`Retrieve          F9
0
`Save              F10
0
Save `2D           F4
0
`Directory         F7
0
Set `Path
0

-----
-1
P`lot              F6
0
Pri`nt            F5
0
`File Convert
0

-----
-1
`BLOCKS
3

```



```

-----
Screen `Read
0
Screen `Save
0
Slide S`how
0
-----
.
.
.
=====

```

Appendix C: Windows and DesignCAD 3-D

DesignCAD 3-D is compatible with MicroSoft Windows. It runs under windows as a non-windows application.

Windows can operate under three modes: Real mode, Standard Mode, and 386 Enhanced mode. DesignCAD 3-D runs under 386 Enhanced mode.

Running DesignCAD 3-D

From the Program Manager, select the Non-Windows Application group by double clicking on its icon. Then select "File" from the pull down menu, and select "New." Check "Program Item" and then select "OK." For Description, enter "DesignCAD 3D." For Command Line, enter "C:\DC3\3D40.PIF" and then press OK. An icon will be created from which you can run DesignCAD 3-D.

Making your own PIF file

Use the PIF Editor provided with Windows. In this example, it is assumed that your DesignCAD 3D was installed in the default drive and directory of "C:\DC3." If you installed the program in a different drive or directory, substitute your drive and directory for every occurrence of "C:\DC3." The suggested PIF options are as follows:

| | |
|----------------------|------------------------|
| Program Filename: | \DC3\DC3.EXE |
| Window Title: | DESIGNCAD 3-D |
| Start Up Directory: | C:\DC3 |
| Memory Requirements: | 512 |
| KB Desired: | 640 |
| Display Usage: | (Check "Full Screen") |
| Execution: | (Check "Background") |
| Check | "Close Window on Exit" |

Advanced Options

Multitasking Options

| | |
|----------------------|--------------------|
| Background Priority: | 50 |
| Foreground Priority: | 100 |
| Check | "Detect Idle Time" |

Memory Options

| | |
|-----------------|---------------------------|
| Check | "High Memory Area" |
| Check | "Lock Application Memory" |
| Display Options | |
| Video Memory: | (Check "High Graphics") |
| Monitor Ports: | (Check "High Graphics") |
| Check | "Emulate Text Mode" |
| Check | "Retain Video Memory" |
| (Other Options) | |
| Check | "Allow Fast Pace" |

Memory Options for PIF Settings

This section will largely depend on your system. The memory options for a PIF can be set to extended memory management or expanded memory management.

XMS (Extended Memory) Settings

| | |
|--------------------------------------|--------------------------------|
| EMS KB Required: | 0 |
| EMS KB Limit: | 0 |
| XMS KB Required: | 2048 (or the amount available) |
| XMS Limit: | -1 |
| Check "Locked" beside the XMS limit. | |

EMS (Expanded Memory) Settings

| | |
|--------------------------------------|--------------------------------|
| EMS KB Required: | 2048 (or the amount available) |
| EMS Limit: | -1 |
| XMS KB Required: | 0 |
| XMS KB Limit: | 0 |
| Check "Locked" beside the EMS limit. | |

Save the finished PIF file to the DesignCAD 3D directory as "3D40.PIF". Close the PIF Editor. From the Program Manager, select the Non-Windows Application group by double clicking on its icon. Then select "File" from the pull down menu and the select "New." Check "Program Item" and select "OK." For Description, enter "DesignCAD 3D." For Command Line, enter "C:\DC3\3D40.PIF" and press OK. An icon will be created from which you can run DesignCAD

Appendix D: Files Used by DesignCAD 3-D

This section contains advanced information not normally needed by DesignCAD 3-D users. Unless you have a specific need for this information, you may skip this section.

The following files are used by DesignCAD 3-D:

| | |
|----------------------------------|--|
| DC3.EXE | DesignCAD 3-D Program |
| DC3PRINT.EXE | Print command |
| DCFILES.EXE | File Conversion program |
| MATERIAL.EXE | Materials List program |
| CHECKDC.EXE | System Check program |
| INSTALL.EXE | Installation program |
| SETUP3D.EXE | Configuration program |
| PTEST.EXE | BasicCAD Test program |
| ENCRYPT.EXE | BasicCAD Encryption program |
| | |
| DCAD n .VFN ($n = 1$ to 9999) | Character Fonts |
| DCAD1.SYS | Configuration File |
| DC3MENU.MNU | Menu File |
| DC3HELP.SYS | Help File |
| DC3SETUP.DAT | Configuration Data use by SETUP3D program |
| DCAD4.SYS | System Parameters |
| DCAD6.SYS | Command Names |
| DC3MENU.SDM | Digitizer Menu |
| DC3PMT.SYS | DesignCAD 3-D Messages |

DCAD1.SYS contains information on the DesignCAD 3-D hardware configuration. This file is covered in more detail in Appendix E.

DCAD4.SYS contains the default setup information. This file is created under the Drawing Parameter command or the System Parameter when the values are saved as program defaults. The following items are saved in this file:

Attribute Display (On or Off)
Character Font

Command Menu
Cursor Step Sizes
Default Text Size and Angle
Digitizer menu location
Digitizer or mouse mode
Digitizer plane
Dimension Format
Dimension Precision
Drawing Units per Inch on Output
Drawing Unit Size
Dynamic Dimensioning (On or Off)
Light settings
Mouse Sensitivity
Point Type
Save System Parameters to Drawing (Yes or No)
Save as a Binary File (Yes or No)
Screen colors
Section Commands Limited to Current Layer (Yes or No)
Snap Grid
Sound
Status Line Format
2-D Snap (Yes or No)
View Configuration
Wall thickness for the Wall command

DCAD6.SYS contains all the entity names, keystroke command mappings, and menu command mappings. This file can be edited to change the name of DesignCAD 3-D commands. However, the menu command section of the file must remain in alphabetical order. This file is described in more detail in Appendix F.

DC3MENU.SDM is the digitizer menu file. It is the same format as the digitizer menu file for DesignCAD 2-D. A custom digitizer menu named DC3MENU.SDM can be created with DesignCAD 2-D and used with DesignCAD 3-D.

Several types of files can be saved by DesignCAD:

| | |
|---------------------|------------------------------------|
| <i>filename.DW3</i> | DesignCAD Drawing (Save command) |
| <i>filename.DC3</i> | ASCII Drawing (Save command) |
| <i>filename.D3M</i> | Macro (Macro Save command) |
| <i>filename.PCX</i> | Screen Image (Screen Save command) |

| | |
|---------------------|--------------------------------------|
| <i>filename.VW3</i> | View (View Save command) |
| <i>filename.MAT</i> | Material (Material command) |
| <i>filename.PAL</i> | Color Palette (Color Save command) |
| <i>filename.SV3</i> | Slide Show View (Slide Show command) |
| <i>filename.SHW</i> | Slide Show File (Slide Show command) |

Appendix E: The DesignCAD Configuration File

This section contains advanced information not normally needed by DesignCAD 3-D users. Unless you have a specific need for this information, you may skip this section.

DesignCAD 3-D uses a file named DCAD1.SYS to store the following system information:

1. DesignCAD 3-D version number.
2. Display information.
3. English/Metric units.
4. Mouse and digitizer button commands.
5. Printer information.
6. Plotter information.
7. Digitizing pad information.

The file is a sequential ASCII file, logically divided into sections for the above data.

The first section after the version number is the display information. The following format is used for this section:

1. Display Name.
2. Display Type.
 - Maximum X coordinate on display.
 - Maximum Y coordinate on display.
 - Number of non-black colors.
 - Text mode BIOS value.
 - Graphics mode BIOS value.
 - Number of buffer pages required.
 - Buffer size, in bytes.
 - Number of bytes per graphics line.
 - Number of characters per line.
 - Graphics text X size.
 - Graphics text Y size.
 - Maximum number of command menu entries.
 - 0 for normal text, 1 for BIOS text.
3. Screen colors .
4. Aspect ratio.
5. Color map - 16 values.

6. External graphics driver name.

The color map is a set of sixteen values that correspond to colors 1 through 16 in DesignCAD 3-D. If you need to modify the color selections, you can edit the file and change these values.

The next two lines of the file contain the Units per Inch and the name of the units. For example, the inches and millimeters look like this:

| | | |
|--------|----|------|
| 1 | or | 25.4 |
| inches | | mm |

The next line has 5 numbers:

1. Display Grid color.
2. Cursor color.
3. Crosshair color.
4. Points color.
5. (not used)

Next in the file is the command section for digitizer buttons and mouse buttons:

```
0 126 125 86 67 79 70 161 86 78 34 42 68 69 27 33
0 126 125 67 86 27 161
```

The first line contains the keystroke command ASCII codes for the digitizer button commands. The second line contains the ASCII codes for the mouse buttons.

The next section of the file contains the printer information. The following format is used for the printer section:

1. Printer Name
2. Printer Type; 0 for black & white, 1 for color
3. A. Output Device used by the printer:
 - 0 - LPT1:
 - 1 - LPT2:
 - 2 - COM1:
 - 3 - COM2:
 - 4 - Disk File
- B. Baud Rate

- 0 - 110 4 - 1200
- 1 - 150 5 - 2400
- 2 - 300 6 - 4800
- 3 - 600 7 - 9600

C. Parity

- 0 - No Parity
- 1 - Odd Parity
- 3 - Even Parity

D. Number of Stop Bits

E. Word Length

Item B, C, D, and E must be present, although they won't be used if the printer device is the parallel port.

4. Number of Densities

For each density:

5. Row Type; Dots per Row; Count Type

6. Special codes 1-8

7. Code before each page

8. Code after each page

9. Code after page feed

10. Code before each section

11. Code after each section

12. Number of phases - 1

For each phase:

13. Code before each line

14. Code after each count

15. Code after each line

16. Horizontal dots per inch; Vertical dots per inch;
Vertical bytes per row.

17. Number of color planes (0 for Black & White)

For each color plane:

18. Code for color plane

19. Color Map

20. Maximum Printer Width; Maximum Printer Length;
Default Printer Width; Default Printer Length.

Following the printer information is the plotter information. The following format is used for the plotter section:

- 1. Plotter Name
- 2. 0 = Normal, 1 = Houston Instruments
- 3. A. Output Device used by the plotter:

- 0 - LPT1:
 - 1 - LPT2:
 - 2 - COM1:
 - 3 - COM2:
 - 4 - Disk File
 - B. Baud Rate
 - 0 - 110 4 - 1200
 - 1 - 150 5 - 2400
 - 2 - 300 6 - 4800
 - 3 - 600 7 - 9600
 - C. Parity
 - 0 - No Parity
 - 1 - Odd Parity
 - 3 - Even Parity
 - D. Number of Stop Bits
 - E. Word Length
- Item B, C, D, and E must be present, although they won't be used if the plotter device is the parallel port.
- 4. Separator character (between numbers).
 - 5. Terminator character (ends commands).
 - 6. Number of Pens (0 = Pause to Change).
 - 7. X-Min, X-Max, Y-Min, Y-Max.
 - 8. Move Absolute command.
 - 9. Draw Absolute command.
 - 10. Initialization command (Before each drawing).
 - 11. Change Pens command.
 - 12. Command After Plotting is Complete.
 - 13. Movement increments per inch.

Following the plotter information is the digitizer information. The following format is used for this section:

- 1. Tablet name
- 2. Tablet attribute byte:
 - (Values for ASCII Digitizers Only:)
 - Bit 0 - 0=No Echo, 1=Echo
 - Bit 1 - 0=Trigger Mode, 1=Stream
 - Bit 2 - 0=Button Last, 1=Button First
 - Bit 3 - 0=Ignore Space, 1=Space Separator
 - Bit 4 - 0=No Sign, 1=Sign Precedes Point
 - Bit 5 - 0=Normal, 1=Hi Pad format.

- Bit 6 - 0=Normal, 1=Summa Binary format.
- Bit 7 - 0=Normal, 1=Unique Format
- Tablet ID (Arbitrary Value)
- 3. Tablet width, in inches; tablet height, in inches.
- 4. X value at lower left corner; Y value at lower left corner;
Tablet resolution in lines per inch.
- 5. Length of trigger string
- 6. Trigger string ASCII decimal values. (65=A)
- 7. Length of initialization string
- 8. Initialization string ASCII decimal values.
- 9. A. Port used by the digitizer:
 - 0 - LPT1:
 - 1 - LPT2:
 - 2 - COM1:
 - 3 - COM2:
 - 4 - Disk File
- B. Baud Rate
 - 0 - 110 4 - 1200
 - 1 - 150 5 - 2400
 - 2 - 300 6 - 4800
 - 3 - 600 7 - 9600
- C. Parity
 - 0 - No Parity
 - 1 - Odd Parity
 - 3 - Even Parity
- D. Number of Stop Bits
- E. Word Length

Appendix F: Customizing Command Names

This section contains advanced information not normally needed by DesignCAD 3-D users. Unless you have a specific need for this information, you may skip this section.

The file DCAD6.SYS contains all the entity names, keystroke command mappings, and menu command mappings for DesignCAD 3-D. This file can be edited to change the name of DesignCAD 3-D commands. Before you modify the file DCAD6.SYS, be **sure** to make a copy of the original file.

The file DCAD6.SYS has three sections. Section 1 contains the DesignCAD 3-D entity numbers and the names associated with those entities. These names are used in messages in DesignCAD 3-D, such as in the ID command. Section 2 contains keystroke commands, and section 3 contains menu commands. Each section is separated by a row of "=".

Section 2 consists of two columns of numbers. The first number is the ASCII code for the keystroke, and the second number is the DesignCAD internal command code. For example, the following section of the file contains three keystroke commands:

```
65 10
66 40
67 6
```

65 is the ASCII code for the letter A, 66 is B, and 67 is C. 10 is the DesignCAD 3-D internal code for the Arc command, 40 is the internal code for the Block Define command, and 6 is the internal code for the Curve command. This means that when letter A is pressed on the keyboard, the Arc command is executed. When B is pressed, Block Define is executed, and C executes the Curve command. A comment can be added in this section by placing a semicolon to the right of the numbers, followed by the comment.

Lower case letters should not be used in the first section. Only upper case letters and special characters should be used.

Some keys (such as function keys) on the PC compatible keyboards return 0 for the ASCII code. For these keys, the ASCII code used in the file is the keyboard scan code plus 128.

The internal codes for the DesignCAD commands can be seen in Section 3 of the file.

Section 3 has the DesignCAD 3-D menu commands and the internal command codes. The menu commands **MUST** be in alphabetical order, and must not contain any lower case letters or spaces. Following each menu command is the internal command code and an optional parameter code. If the parameter code is 1, then one or more parameters can be entered on the command line. If the parameter code is 2, a line of text can be entered on the command line (for Text command, for example). Otherwise, the parameter code can be omitted. There may be a third number in this part of the file. This number is not currently used.

Here is a portion of Section 3 of the file:

| | | | |
|-------|----|---|---|
| ARC | 10 | 1 | 4 |
| ARC-2 | 11 | 1 | 4 |
| ARC-3 | 17 | 0 | 4 |

The ARC command uses internal code 10 and can have a parameter (how many degrees). ARC-2 uses internal code 11 and can have a parameter for the radius. ARC-3 does not have an input parameter, and uses internal code 17.

Appendix G: Hardware Interfacing

AlTech AC-30 Digitizer

Setup as AlTech AC30, AC40. Set the two banks of dip switches as follows:

| Left Bank | Right Bank |
|-----------------|-----------------|
| 1 2 3 4 5 6 7 8 | 1 2 3 4 5 6 7 8 |
| 0 1 1 1 1 1 0 0 | 0 0 0 0 1 0 0 0 |

AlTech AC-40 Digitizer

Setup as AlTech AC30, AC40. Set the four banks of dipswitches as follows:

| Upper Left | Upper Right |
|-----------------|-----------------|
| 1 2 3 4 5 6 7 8 | 1 2 3 4 5 6 7 8 |
| 0 1 1 1 0 0 1 1 | 1 0 1 0 1 1 0 0 |

| Lower Left | Lower Right |
|-----------------|-----------------|
| 1 2 3 4 5 6 7 8 | 1 2 3 4 5 6 7 8 |
| 1 0 1 1 1 0 0 0 | 1 1 1 1 1 0 1 0 |

Calcomp 2000 Digitizer

Dip Switches: 1=ON, 0=OFF, x=Doesn't Matter

Bank #1

| | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 |
|--|---|---|---|---|---|---|---|---|
| | 0 | 0 | 0 | 0 | 0 | 0 | 0 | 0 |

Bank #2

| | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|--|---|---|---|---|---|---|---|---|---|----|
| | 0 | 1 | 1 | 0 | 0 | 1 | 0 | 0 | 1 | x |

Bank #3

| | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|--|---|---|---|---|---|---|---|---|---|----|
| | 1 | 0 | 0 | 0 | 0 | 0 | 0 | 0 | 1 | 0 |

Calcomp 2100 Digitizer

Dip Switches: 0=Down (Open), 1=Up (Closed), x=Doesn't Matter

Bank #1

1 2 3 4 5 6 7 8

1 0 1 0 0 1 0 0

Bank #2

1 2 3 4 5 6 7 8 9 10

0 0 0 0 1 1 0 X X X

Two jumpers beside the switches should be horizontal to avoid having to use a null modem.

These settings should set the digitizer as follows:

8 Bit Word length
Parity Enabled, Even
Disable Fixed Parity
Disable Parity Check
9600 Baud

English Scaling
ASCII Data
<LF> After <CR>
Remote Mode

Calcomp 2500 Digitizer

| Bank | ASCII | Binary | 9100 |
|-------------|--------------|---------------|-------------|
| 1 | 00000001 | 00000001 | 00000001 |
| 2 | 11000100 | 11000100 | 11000100 |
| 3 | 00000000 | 00011100 | 01100100 |
| 4 | 00100001 | 00100001 | 00100001 |
| 5 | 01111100 | 01111100 | 01111100 |
| 6 | 00000000 | 00000000 | 00000000 |

Calcomp 9100 Digitizer

Dip Switches: 1=ON, 0=OFF, x=Doesn't Matter

Bank Area 1, SB1:

1 2 3 4 5 6 7 8
0 0 1 1 1 1 x x

Bank Area 2, SB1

1 2 3 4 5 6 7 8
1 0 0 1 1 1 1 0

Bank Area 2, SB2: Use Factory Settings

Bank Area 3, Sb2: Use Factory SETTINGS

These settings should set the digitizer as follows:

Run Mode
Format 1
1000 Lines per Inch
Even Parity
1 Stop Bit
7 Data Bits
9600 Baud

Calcomp 23180 Digitizer

Initial Settings: (0=OFF, 1=ON)

Mode

| | |
|--------|---|
| Point | 0 |
| Run | 1 |
| Track | 0 |
| Mouse | 0 |
| Prompt | 0 |

Parity

| | |
|-----|---|
| 7/8 | 0 |
| 1 | 0 |
| 2 | 0 |
| 3 | 1 |

Baud Rate

| | |
|---|---|
| 1 | 0 |
| 2 | 0 |
| 3 | 1 |

Format

| | |
|----|---|
| 1 | 0 |
| 2 | 0 |
| 3 | 1 |
| 4 | 1 |
| LF | 0 |

Data Rate

| | |
|---|---|
| 1 | 1 |
| 2 | 1 |
| 3 | 1 |

Resolution

| | |
|------|---|
| 1 | 0 |
| 2 | 0 |
| 3 | 1 |
| Freq | 1 |

GTCO Series 5 Digitizer

Dip Switches: 1=ON, 0=OFF, x=Doesn't Matter

Bank #1

1 2 3 4 5 6 7 8
0 0 1 1 0 0 0 1

Bank #2

1 2 3 4 5 6 7 8
1 1 1 1 1 0 0 0

Bank #3

1 2 3 4 5 6 7 8
0 0 1 1 x 0 1 0

These settings should set the digitizer as follows:

Baud Rate 9600
Parity Enable, even
2 Stop Bits
8 Bit Word

Include Pushbutton Code (ASCII)
Include Space (ASCII)
Include CR (ASCII)
Include LF (ASCII)
Hi Resolution
Disable Serial Port B
Disable Serial Port A
Enable Audible Alarm

GPIB instead of RS-232
Output Rate = 200
Continuous Mode (Stream)
16 Button Cursor
Metric Data
Packed Binary Format
Serial Handshaking? Leave Off

Uses Packed Binary High Resolution. Must have 8 bit word. There are two serial ports. One requires a null modem, and one doesn't.

Hewlett-Packard SketchPro

Dip Switches: 1 = On, 0 = Off

Bank #1

1 2 3 4 5 6 7 8
1 0 0 0 0 0 0 0

Bank #2

1 2 3 4 5 6 7 8
0 0 1 0 1 0 0 0

Bank #3

1 2 3 4 5 6 7 8
0 0 0 0 0 0 0 0

Bank #4

1 2 3 4 5 6 7 8
1 1 0 0 0 0 0 0

This will set up the digitizer for Hitachi emulation, Run Mode, Incremental, Maximum Rate, 9600 Baud, Even Parity, 7 Bit Word, and 1 Stop Bit.

For HP Vectra and AT-compatible computers, use the HP 45911-60009 Cable. For PC and PC-XT compatibles, use the HP 45911-60007 adapter (with the cable).

HP 45911-60009 Cable Setup

| Digitizer 25-pin Connector | Computer 9-pin Connector |
|----------------------------------|--------------------------------|
| 4----- | 1 |
| 2----- | 2 |
| 3----- | 3 |
| 5 & 6 (jumped)----- | 4 |
| 7----- | 5 |
| 20----- | 6 & 8 (jumped) |
| 8----- | 7 |

Hitachi Tiger Digitizer

Dip Swtches: 1 = On, 0 = Off

(ASCII)

Bank #1

1 2 3 4 5 6 7 8
1 0 0 0 0 0 0 0

Bank #2

1 2 3 4 5 6 7 8
0 0 1 0 1 0 0 0

Bank #3

1 2 3 4 5 6 7 8
0 0 0 0 0 0 0 0

Set up for Stream Mode, High Resolution (1000 DPI), Even Parity

(BINARY)

Bank #1

1 2 3 4 5 6 7 8
1 0 0 1 0 0 0 1

Bank #2

1 2 3 4 5 6 7 8
0 0 1 0 1 0 0 0

Bank #3

1 2 3 4 5 6 7 8
1 0 0 0 0 0 1 0

Setup for Stream Mode, Low Resolution (Use Kurta Series 2 Setup
for High Resolution)

Hitachi HDG-1111c Digitizer

Use Supplied Cable

Dip Switches:

Bank #1

1 2 3 4 5 6 7 8
1 1 0 0 0 0 0 0

Bank #2

1 2 3 4 5 6 7 8
1 0 1 0 1 0 1 0

Bank #3

1 2 3 4 5 6 7 8
0 0 0 0 0 0 0 0

These selections should set the digitizer as follows:

Unit Selection: .005"

Mode Selection: Run

Rate Selection: 1

Incremental Mode Normal

Floating Origin: Fixed Lower Left Corner

L/F Code Output: NO

Baud Rate: 9600

Parity: Even

1 Stop Bit, 7 Bit word

Ascii/Binary: Binary

Digitize/Test: Normal

Format: Hitachi

Not Prox Data: NO

Buzzer Enable

Active Area: OFF 11"x11"

Note: For Hitachi HDG-1111D, set ALL Dip Switches to OFF!

Houston Instruments Digitizers

HI-PAD Cable

| HI-PAD | IBM |
|--------|----------------|
| RJ14 | DB-25S |
| 2----- | 3 |
| 3----- | 2 |
| 4----- | 7 |
| | 4 jumper to 5 |
| | 6 jumper to 20 |

| HI-PAD | IBM |
|--------|---------------|
| RJ14 | DB-9S |
| 2----- | 2 |
| 3----- | 3 |
| 4----- | 5 |
| | 6 jumper to 4 |
| | 8 jumper to 7 |

TRUE-GRID Cable

| TRUE-GRID | IBM |
|-----------|-------------------------------------|
| | DB-9 |
| 2----- | 2 |
| 3----- | 3 |
| 4----- | 5 |
| 5----- | 8 |
| | 1, 4, 6, AND 9 jumpered together |

Kurta IS-One Digitizer

Bank A: 1-8 Off, except 4 On: 00010000
Bank B: 8 On, Others Off: 00000001
Bank C: All Off: 00000000

Kurta Series Three Digitizer

Bank of 8 - All Off: 00000000
Bank of 4 - 1 On, 2-3-4 Off: 1000

Kurta Series One Digitizer

Bank of 8 - 1-7 Off, 8 On: 00000001
Bank of 4 - 1 On, 2-3-4 Off: 0001

Pin 4 must be disconnected on the computer side of the cable.

Mitsubishi Grafnet 01 Digitizer

Dip Switch: (1=ON, 0=OFF)

| | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|----|
| 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
| 1 | 1 | 1 | 1 | 1 | 1 | 1 | 1 | 1 | 1 |

These settings should set the digitizer as follows:

Binary
Output Mode: STREAM
Millimeter
Baud Rate: 9600
Parity: EVEN
Operating Speed: 60 PPS

Use a straight-through cable.

Mitsubishi Grafnet 02 Digitizer

Dip Switches:

Bank 1

| | | | | | |
|---|---|---|---|---|---|
| 1 | 2 | 3 | 4 | 5 | 6 |
| 1 | 0 | 0 | 1 | 1 | 1 |

Bank 2

| | | | | | |
|---|---|---|---|---|---|
| 1 | 2 | 3 | 4 | 5 | 6 |
| 1 | 1 | 0 | 1 | 1 | 0 |

Setup as a Summagraphics Bit Pad I (Binary)

Settings: Baud Rate: 9600
Parity: Even
Stop Bit: 2
Word Length: 7

Numonics 2200 Digitizer

Dip Switches: 1=Up (Closed), 0=Down (Open)

Bank A

1 2 3 4 5 6 7 8

1 0 1 1 0 0* 0 1

Bank B

1 2 3 4 5 6 7 8

1 1 1 1 1 1 1 1

These settings should set up the digitizer as follows:

No Point Mode

Stream Mode

No Incremental Mode

No Switch Stream Mode

English Units

Absolute Mode (* = 1 for Calcomp 2000 Version)

Packed Binary

No <CR>

No <LF>

No Parity

Even Parity

1 Stop Bit

No Audible

No XON/XOFF

No Self Diagnostics

Scriptel Digitizer

Dip Switches: 1=ON, 0=OFF

Bank 1:

1 2 3 4 5 6 7 8 9

1 1 0 1 1 0 1 1 0

Bank 2:

1 2 3 4 5 6 7 8 9

1 0 0 0 0 0 0 0 0 (ASCII) or

1 2 3 4 5 6 7 8 9

1 1 0 0 0 0 0 0 0 (BINARY)

Use a straight-through cable.

Summagraphics Microgrid Digitizer

Dip Switches:

Bank #1

1 2 3 4 5 6 7 8
1 0 0 1 1 1 0 0

Bank #2

1 2 3 4 5 6 7 8
0 1 0 * 1 1 1 1

* 1 for Binary, 0 for ASCII (Can be set for either)

Bank #3

1 2 3 4 5 6 7 8
0 1 0 # # 1 1 1

Reserved (Do not change)

These settings should set up the digitizer as follows:

Baud Rate: 9600

Parity Enabled, Even

Stop Bits: 2

Echo Off

Proximity Transmission, only when in proximity

Resolution: 1000 LPI

Ascii counts report format: OFF

Report format: Binary (or ASCII, see above)

Report Collection Mode: Stream

Device Type: UIOF

ASCII Report Terminator: <CR> Tablet Identifier: 0

Report Rate: 80 RPS

ASCII is the factory setting. Factory setting for COM port is 9600, even parity, 7 bit word, and 2 stop bits.

There are two RS-232 Ports. Use the first one with a null modem adapter or cable, or the second one without a null modem adapter.

Summagraphics Bit Pad 2

Dip Switches: 1=ON, 0=OFF

Bank #1

1 2 3 4 5 6 7 8
0 0 1 0 0 1 1 0

Bank #2

1 2 3 4 5 6 7 8
0 1 0 0 0 0 1 0

Bank #3

1 2 3 4 5 6 7 8
0 0 0 1 0 1 1 0

These settings should set up the digitizer as follows:

Remote Enable - Off

Proximity Transmit

Stream Mode

Report Rate - 100

ASCII/Binary (Binary)

CR/LF Terminator

Increment Setting 0

Resolution Setting 200

Parity Enable Off

Even/Odd Even

Stop Bits 1

Cts Handshake Enable?

Cursor Output Code: B

Baud 9600

WIZ Digitizing Tablet

Setup the WIZ as a three button mouse for 2-D ver 4.1 and later, and 3-D ver 3.1 and later; use Logitech C7 driver for 2-D versions 4.0 or earlier.

Orchid ProDesigner Ver 2.2 VGA

Before running at 1024x768 or 800x600 VGA resolution you must run VMODE.COM (the utility program that comes with the card) and run in interlaced mode.

ATI VGA Wonder Graphics Card

You must run VSETUP (the utility program that comes with the card) to configure the card and the monitor before running DesignCAD.

IBM 8514A Graphics Card

You must have HDI (a memory-resident utility program that comes with the card) loaded on IBM computers. You must also have a full Megabyte of memory.

Calcomp 1043 Plotter

Set up the plotter as follows:

- PCI Format
- Baud Rate: 9600
- 8 Bit word length
- 1 Stop bit
- Parity: None
- Direct Control
- Sync Character 2
- Number sync 1
- End message 3
- Checksum disable

Calcomp 1044 Plotter

Set up as CalComp 1043 in DesignCAD. Set up the plotter as follows:

- Parity: None
- Stop Bit: 1
- Word Length: 8
- Clock: 0 PCI mode: 0
- Muting: OFF
- Check Sum: OFF
- ISO: OFF
- End of message Character: 3
- # sync Modes: 1
- Sync Character: 2
- Direct Control: YES
- XON/XOFF: OFF
- Duplex: 0

Dip Switches: 2 & 4 ON all others OFF

Hewlett Packard 7475 Plotter

Switch 1: 0 0 0 1 0 1 0 1 0

Use HP Cable 17255 or equivalent for a 25-pin connector on the computer or HP Cable 24542 for a 9-pin connector on the computer.

HP 24542 Cable

| Computer DB-9S | Plotter DB-25S |
|-------------------|-------------------|
| 1----- | 4 |
| 2----- | 2 |
| 3----- | 3 |
| 4----- | 5, 6 |
| 5----- | 7 |
| 6, 8----- | 20 |
| 7----- | 8 |

HP 17255 Cable

| Computer DB-25S | Plotter DB-25S |
|--------------------|-------------------|
| 1----- | 1 |
| 3----- | 2 |
| 2----- | 3 |
| 7----- | 7 |
| 20----- | 5, 6 |
| 5, 6----- | 20 |

XON/XOFF may be either enabled or disabled.

Use Stand Alone mode, not Eavesdrop.

Hewlett Packard Plotters

The following settings are necessary:

Full Duplex
 Hardware Handshaking
 DTR Normal
 XON/XOFF may be either enabled or disabled
 Use Stand Alone mode, not eavesdrop

Use HP Cable 17255 or equivalent for a 25-pin connector on the computer or HP Cable 24542 for a 9-pin connector on the computer.

HP 24542 Cable

| Computer DB-9S | Plotter DB-25S |
|-------------------|-------------------|
| 1----- | 4 |
| 2----- | 2 |
| 3----- | 3 |
| 4----- | 5, 6 |
| 5----- | 7 |
| 6, 8----- | 20 |
| 7----- | 8 |

HP 17255 Cable

| Computer DB-25S | Plotter DB-25S |
|--------------------|-------------------|
| 1----- | 1 |
| 3----- | 2 |
| 2----- | 3 |
| 7----- | 7 |
| 20----- | 5, 6 |
| 5, 6----- | 20 |

If your computer is communicating directly with an HP Plotter (and not plotting to disk), you can select "HP Auto Size" in the Setup program. This lets DesignCAD determine the actual size of the paper in the plotter. Otherwise, you should select the correct paper size in the setup menu.

Houston Instruments Plotter Cable Setup

25-pin Serial Connector

| Computer | Plotter |
|--|---------|
| DB-25S | DB-25S |
| 3----- | 2 |
| 2----- | 3 |
| 5----- | 4 |
| 4----- | 5 |
| 7----- | 7 (GND) |
| 6, 8, 20, and 22 jumpered together (both ends) | |
| 1, 9-19, 21, and 23-25 are not connected | |

| Computer | Plotter |
|--|-------------------------|
| DB-25S | RJ12 6-pin modular plug |
| (Not Connected)----- | 1 |
| 3----- | 2 |
| 2----- | 3 |
| 7----- | 4 |
| 5----- | 5 |
| (Not Connected)----- | 6 |
| (6, 8, 20, and 22 jumpered together | |
| 1, 4, 9-19, 21, and 23-25 not connected) | |

9-pin Serial Connector

| Computer | Plotter |
|--|---------|
| DB-9S | DB-25S |
| 2----- | 2 |
| 3----- | 3 |
| 5----- | 7 |
| 7----- | 5 |
| 8----- | 4 |
| (1, 4, 6, and 9 are jumpered together) | |
| (1, 9-19, 21, and 23-25 are not connected) | |

| Computer | Plotter |
|-------------------------------------|-------------------------|
| DB-9S | RJ12 6 pin modular plug |
| (Not Connected)----- | 1 |
| 2----- | 2 |
| 3----- | 3 |
| 5----- | 4 |
| 8----- | 5 |
| (Not Connected)----- | 6 |
| (1, 4, 6, and 9 jumpered together.) | |
| (7 is not connected.) | |

Toshiba 1100

The Toshiba 1100 should be configured as an IBM color graphics adapter with a monochrome display.

Panasonic KXP1180 Dot-Matrix Printer

In the default mode, this printer emulates an Epson FX-80. In order to emulate an IBM Proprinter then turn the printer off and set dipswitch #1 to ON. Turn the printer back on.

Hewlett-Packard LaserJet III Plot Mode

These settings let you switch your HP LaserJet III from Landscape mode to Portrait mode, in plotter emulation mode. You by no means have to switch modes, DesignCAD will work in either mode. However, you may discover that a drawing might fit better in the other mode. Here's how to do it.

First, you need to change the DCAD.SYS file. Using an ASCII text editor, change the following lines in the file:

The plotter section of the original file will look like this:

```
=====
HP Laserjet III, Plot Mode
0
 4 0 0 0 0
,
1
1 10160 0 8124
PU;PA
PD;PA
'027&a0h0v'027%BDF;SP!;RO90;PW0
SP
'027%0A'012
1016
=====
```

Make the following changes:

```
=====
HP Laserjet III, Plot Mode
0
 4 0 0 0 0
,
1
0 8636 0 11176
PU;PA
PD;PA
'027&a0h0V'027%1BDF;SP1;PW0
SP
'p27%oA'012
1016
=====
```

This allows a maximum paper size of 8.5x11 and a plot size of 8x10.

Appendix H: Glossary

axis - An axis is an imaginary straight line. The X, Y, and Z axes run horizontally (the X axis), vertically (the Y axis), and front-to-back (the Z axis).

axis of rotation - The axis of rotation of an object is the imaginary line about which the object rotates. For example, the axis of rotation of the Earth is the imaginary line from the North pole to the South pole.

axis of revolution - The axis of revolution of an object is the imaginary line about which the object revolves. For example, in the Block Circular Repeat command, the object is copied in a circle around the axis of revolution.

background light - This determines the brightness of "dark" or shadowy areas of a shaded object. If this value is too high, the shaded object will have little or no contrast.

Block - A Block is a group of items in the drawing. It can be moved, copied, and manipulated as a single unit. You can define a Block using the Block Define command.

Block handles - Block handles are reference points you can use when moving, copying, or editing a Block. For example, when you move a Block you specify a point for the first Block handle. There are three Block handles associated with a Block.

command line - The command line is the line at the bottom of the screen where you enter commands. To enter a command on the command line, press the **Space** bar.

command menu - The command menu is the menu at the top of the screen, together with the drop-down menus. To select a command from the command menu, move the cursor to the menu or press **Enter**.

diffuse reflection - This is the amount of normal or diffuse reflection from an object. Diffuse reflection is the type of reflection you get from flat latex wall paint. It is not shiny.

digitizer menu - A digitizer menu is a menu you can use to select commands on the digitizer. There are two parts to a digitizer menu: the paper menu which you attach to the digitizer, and the digitizer menu file which you load using the Digitizer Menu Load command.

drawing screen - The drawing screen is the large view on the DesignCAD screen.

Drawing Units - Drawing Units are the units of measurement for a DesignCAD drawing. They can represent inches, millimeters, feet, light years - anything.

Enter Box - The Enter Box is the small box in the upper left corner of the drawing screen with a diagonal line through it. Instead of pressing **Enter** on the keyboard, you can move the mouse to the Enter Box and press the left mouse button. If you have a 3-button mouse, you can use the middle mouse button instead of the Enter Box. If you do not use a mouse, the Enter Box will not be displayed.

entity - An entity is a single item in the drawing such as a line, curve, circle, or text.

facet - A curved surface is actually made up of small flat surfaces. These small flat surfaces are called the facets or a curved surface.

hidden line removal - Hidden line removal is the process of removing the lines of a drawing that lie behind surfaces in the drawing.

keyframe - A keyframe is a view of the drawing used in the Slide Show command. Several keyframes, or views, are defined, and a number of slides are generated between the keyframes.

macro - A macro is a set of DesignCAD 3-D commands that have been recorded to disk. A macro can be executed as a single command.

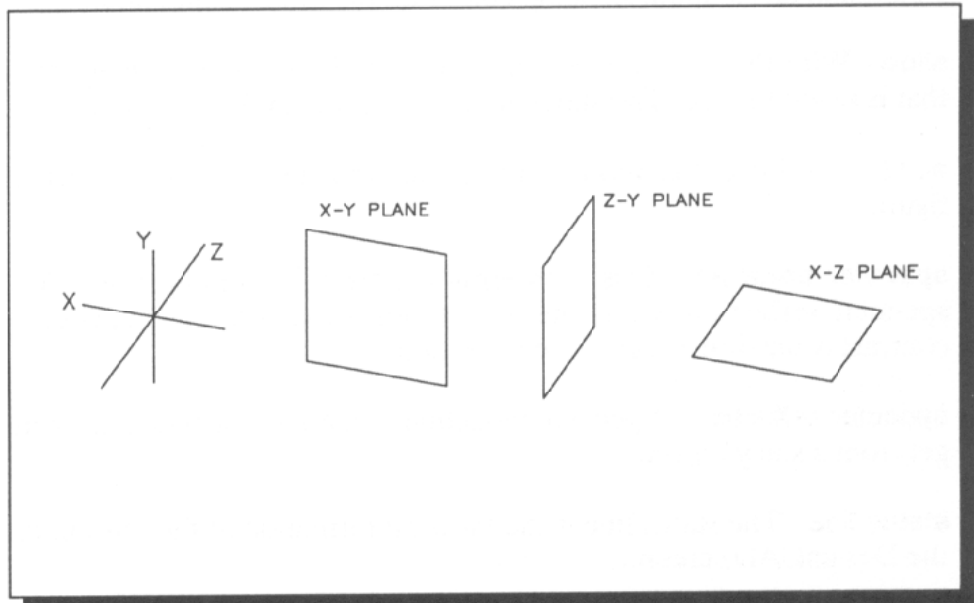


Figure 85 - The X-Y, X-Z, and Y-Z planes.

material - In DesignCAD 3-D, the term Material is used for a set of material properties: colors and shading information. These material properties are used to produce realistic images with the Shade command.

plane - A plane is a flat two-dimensional surface. It can lie at any angle or location, and can be any size or shape. There are three special planes. The X-Y plane is an imaginary, infinite plane that runs horizontally and vertically, along the X and Y axis, parallel to your computer screen. The X-Z plane runs parallel to your desk top, and the Y-Z plane runs parallel to the side of your computer. See Figure 85.

perspective - Perspective is the effect of the view distance on the drawing. The smaller the view distance, the greater the perspective. The perspective gives the idea of distance and size to the drawing.

shading - Shading the drawing "colors" all the visible surfaces in the drawing, making it look realistic. There are two types of shading with

DesignCAD 3-D: quick shading, done with the Quick Shade command, and smooth shading, done with the Shade command.

slide - With the Slide Show command, a slide is a screen image file that is saved to disk. The slides are saved in the .PCX file format.

solid - A solid is a group of planes and surfaces that form a closed figure.

specular contrast - This is essentially how shiny an object is. The specular reflection is the amount of light reflected. The specular contrast is the directness of the reflection.

specular reflection - Specular reflection is the type of reflection you get from a shiny object.

status line - The status line is the line of information at the bottom of the DesignCAD screen.

texture - Texture is one of the shading parameters used in the Material command. There are several textures available with DesignCAD 3-D, such as wood, marble, cement, etc.

view angle - The view angle is the angle from which the drawing is viewed. There is a view angle for the large DesignCAD 3-D view and a view angle for each of the smaller views.

view distance - The view distance is the distance from which the drawing is viewed. The smaller the view distance, the greater the perspective.

viewpoint - A viewpoint is the angle and distance from which the drawing is viewed. This can be set in the View command.

wireframe - Wireframe mode is the mode in which the drawing is normally displayed. In wireframe mode, the drawing is represented as lines on the screen, as if it were constructed with a wire frame.

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