

DesignCAD[®]

Version 16

REFERENCE MANUAL

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1 Introduction

About DesignCAD

DesignCAD is a comprehensive computer-aided design package that incorporates a full range of 2D and 3D drawing functions.

You can use DesignCAD to create drawings for any assignment, from simple to complex, and the finished drawing can be printed using any printer or plotter that the various 32-bit versions of Windows support.

DesignCAD can be customized to fit your particular application. You can create your own Custom Toolbox and even write your own DesignCAD commands using BasicCAD!

With its numerous high-end features, DesignCAD compares favorably with CAD systems costing thousands of dollars. Unlike other high-end systems though, DesignCAD is easy to learn and use. With a little practice, virtually anyone can make detailed drawings of professional quality using DesignCAD.

In addition to its 2D Mode, DesignCAD is a true three-dimensional CAD system. You can use it to construct realistic 3D models of your projects. You can show them in wireframe view, with hidden lines removed, or with full-color shading--from any viewing angle. You can also create animation files which step the viewer around your drawing in smooth increments. For example, you could start with an aerial view of a house, descend to ground level, and then walk all around it. You can even assign material properties to your creations, placing a brass doorknob on an oak door, or creating a lavatory of rose marble with chrome fittings.

If you have DesignCAD 3000, DesignCAD 2000, DesignCAD 97, or any DesignCAD 2D or DesignCAD 3D program, and DesignCAD 3D MAX, you can interchange drawings between the two applications. You can take a cross-section of a complex beam which you may have created in DesignCAD 2D, load the cross-section into DesignCAD 3D MAX and extrude it into a beam. Then you can save the extruded beam as a DesignCAD 3D MAX drawing, even with hidden lines removed! If you want to go a step further, you can extrude your floor plan into an elevation, add a roof, and save a perspective view back into DesignCAD 2D format.

DesignCAD imports and exports drawings in DWG, DXF, HPGL, and IGES formats, and also reads and writes Windows Metafiles. DesignCAD can also export WPG, RIB and WRL formats. Other imported formats include HPGL and XYZ. DesignCAD can pass drawing information to and from the Clipboard and export OLE 2.0 objects to applications that support them.

System Requirements

To use DesignCAD 3D MAX, you must have, at a minimum, the following hardware and software installed in your computer:

- Approximately 27 to 30 megabytes of hard drive space for a compact installation.
- Microsoft Windows 2000, or Windows XP.

- Although not required to run DesignCAD, the following hardware is recommended for better performance:

Pentium processor.

at least 16 megabytes of RAM.

SuperVGA graphics card capable of 256 or more colors.

SuperVGA monitor capable of at least 800x600 resolution.

Typographical Conventions

This manual uses a few special fonts, phrases, and symbols to refer to commands and instructions.

Mouse

When the word "click" is used alone, it means "left-click," or to press the left mouse button. When it is necessary to use the right mouse button, the manual states that explicitly. Phrases such as "click the right mouse button" and "right-click the mouse" assume that the **Enable Right Click Popup Menu** option in the **General Options** (see General Options) has been toggled off.

Keyboard

The keys on your keyboard may not be labeled exactly as they are in this manual. All key names are shown using bold, sans serif type. For example, the "Control" key is shown as Ctrl and the "Enter" key is shown as Enter.

Keys are sometimes used in combinations. For example, Ctrl+F means to hold down the Ctrl key while pressing the F key.

"Arrow keys" is the collective name for the up arrow, down arrow, left arrow, and right arrow keys.

To choose a command from a menu, you can use the mouse or press a key combination.

Technical Support

If you have a question about DesignCAD 3D MAX, please look in the Reference Manual or the on-line Help for the solution. Remember to check the Index and Table of Contents.

If you cannot find the answer to your question in the documentation, email the DesignCAD Technical Support Department: support@designcad.com.

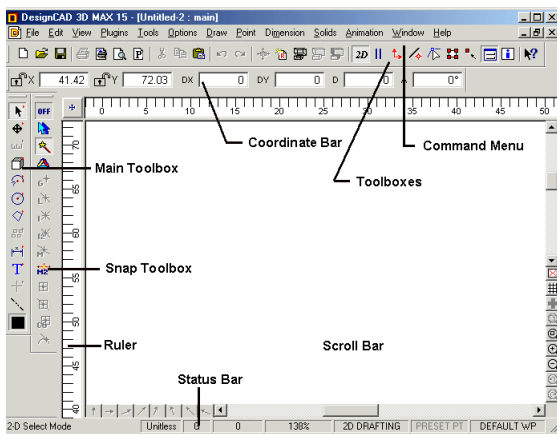
When emailing for support, please provide the following information:

- The serial number, version name of DesignCAD (e.g., DesignCAD 3D MAX, version 14), and release date.
- The release date is located in the About DesignCAD 3D MAX window. To open the About DesignCAD window, select the **About DesignCAD** command from the **Help** menu.
- The type of hardware you are using.

2 General Information

DesignCAD User Interface

The DesignCAD drawing screen is shown below. Descriptions of the components follow. For details on how to maximize, minimize, open, and close windows, refer to your Windows documentation.



Command Menu: Contains the drop-down menu selections. From the Command Menu you can pick every available DesignCAD command.

Coordinate Bar: Displays the cursor's location in 2D or 3D space depending on whether 2D Mode is active or not. If you are executing a drawing command, it displays the distance moved from the last point set as DX, DY, and DZ (DZ is not displayed in 2D Mode).

Main Toolbox: Contains graphic icons for the most frequently used drawing commands.

Scroll Bars: Let you pan across a drawing that is larger than the screen size.

Snap Toolbox: Contains commands to set points at specific locations in the drawing, such as midpoints and intersections.

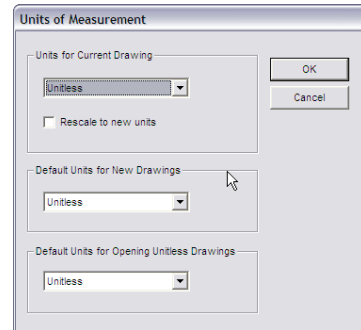
Status Bar: Displays quick help on using the current command. The Status Bar may be replaced with a progress bar for certain commands, such as the **Shading** command.

Toolboxes: Contains a number of icons to speed the selection of frequently used commands and functions.

Units of Measurement

Menu: Options / Units of Measurement

Lets you specify the base units of measurement for three sets of circumstances:



Rescale to New Units: This causes objects to be correctly sized when the units are changed. If this option is off an object 1cm is changed to 1mm or 1in when the unit type is changed. If the option is on the 1cm object would be converted to a 10mm or 2.54in object.

Units for Current Drawing: Sets (or changes) the default base units for the current drawing. Changing the current drawing's base units does

not affect the numeric value of the distance between two points - in other words, if you switch from inches to miles, the length of what was originally a 10-inch line is now 10 miles; no attempt is made to scale the drawing from the original base unit to the new one.

Default Units for New Drawings: The default base unit that will be used for any new drawings you create.

Default Units for Opening Unitless Drawings: Affects the way unitless symbols and blocks are imported into the drawing. When loading a unitless symbol or block, the imported object's base units are assumed to be the same as those specified by this setting.

If a unit-based symbol or block is loaded or merged into a unit-based drawings, the symbol's or block's base units are converted to those of the host unit, so that the symbol is imported at the correct "real-world" size. When loading a unit-based symbol or block into a unitless drawing, the symbol or block is also assumed to be unitless (for example, a distance of 10 base units in the symbol becomes a distance of 10 in the unitless drawing).

When entering a distance in an edit box, such as the DX, DY, or DZ fields in **Point Relative**, a plain decimal number is assumed to be in the current units of measurement. If a value is entered in feet-and-inches format, such as 3' 2", then that distance is converted to the current base unit of measurement. If the current base unit is inches, for example, an entry of 3' 2" becomes 38 inches; if the current base unit is meters, the entry is converted to 0.9652 meters. If the drawing is unitless, the base unit is implicitly treated as feet

anytime a feet-and-inches distance entry is specified (in other words, the example distance entry of 3' 2" is converted to 3.166666667 units.)

Dimension commands (see Dimensions) will normally dimension in the current base unit, so long as a decimal or fractional text format is chosen for the dimension command. However, if a Feet-and-Inches text format is chosen, then the displayed distance will be converted from the current base unit to feet and inches. If a Feet-and-Inches dimension text format is chosen in a unitless drawing, the distance is calculated as though the base unit were feet.

Drawing to Scale

When you draw to scale in DesignCAD, you are usually measuring the objects in a given base unit of measure. DesignCAD doesn't care whether your base unit is meters or miles or even leagues. What's important is that you use the same base unit throughout the drawing.

Let's say that you're drawing a house and the front wall is 32 feet long. To DesignCAD, it is 32 Drawing Units. You can draw a line by choosing the **Line** command, setting the first point (click the mouse or press Ins), then specifying that the next point is 32 Units away.

But what if the next item you measure in the same drawing is 10 centimeters tall? If you draw it at a height of 10 units, it will be much too large. Why? Because centimeters and feet are different units. In this example, you would need to convert the centimeters to feet, and then tell DesignCAD the size of the item in feet.

The key point is not to mix units. If feet are convenient, call out all distances in the drawing in feet. If centimeters are convenient, measure

everything in centimeters. As long as you're consistent, all is well. If you use feet, many of the commands allow you to enter distances in feet and inches: for example, to specify a line 9 feet 5 inches long, set the first point, then enter 9'5" in the DX field of the **Point Relative** command.

Printing to Scale

What about printing to scale? The **Print** window (see Print) shows the paper units selected and the scale of the printout. Scale here represents the number of paper units it takes to print one Drawing Unit.

Suppose we have drawn a 10-inch box, which we specified as 10 units when we drew it. Now we want to print it at 0.25 scale. The paper units default to inches, and our drawing is also in inches. Therefore it takes 0.25 inches on paper to represent one inch in the drawing. Our 10-inch box comes out 2.5 inches long on paper.

If our box had been specified at 10 feet instead of 10 inches, then it would take 1/4 inch of paper to represent 1 foot in the drawing. This is a real-world scale of 1/48, or 0.0208333. But to DesignCAD units are just units, so the scale is still shown as 0.25. It took 0.25 paper units (inches) to represent one Drawing Unit (feet). Printing scale is the length on paper that will represent one Drawing Unit.

We can show it as an equation:

True Scale = Scale/Ratio (where Ratio is the number of paper units in one Drawing Unit)

In the example above, our scale is .25. The paper unit is in inches, the Drawing Unit in feet. Since there are twelve inches in one foot, the Ratio equals twelve.

$$\text{True Scale} = \text{Scale}/\text{Ratio} = .25/12 = 1/48$$

What if your paper is too small to print at the scale you need? No problem. DesignCAD can print out your drawing in panels, all to scale, which you can then assemble into a composite drawing.

Here is a list of common Architectural and Engineering scales.

Architectural

Scale Name Actual Decimal Value

1/16"	= 1'.0625
3/32"	= 1'.09375
1/8"	= 1'.125
3/16"	= 1'.1875
1/4"	= 1'.25
3/8"	= 1'.375
1/2"	= 1'.5
3/4"	= 1'.75
1"	= 1'1.0
1.5"	= 1'1.5
3"	= 1'3.0

Engineering

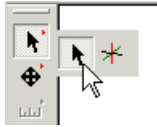
Scale Name Actual Decimal Value

1:1000	.001
1:500	.002
1:333	.003
1:100	.01
1:50	.02
1:20	.05
1:10	.1
1:5	.2
1:3	.33
1:2	.5
2:1	2
3:1	3
5:1	5

2-D Mode

Menu: Options / 2-D Mode

Toolbar Icon:



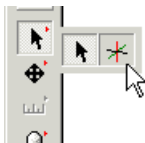
Shifts the viewer perspective to a front view and treats operations as if they were only occurring in 2D space. This makes it very easy to use DesignCAD to draw just as you would in a regular 2D drafting package.

If your drawing contains 3D objects, it will still be a 3D drawing; however the cursor will only move parallel to the X and Y axes, with the Z coordinate set to zero. This makes it very easy to add 2D information to your drawing.

NOTE: In 2D Mode, the **Trim** commands ignore the 3D aspects of any existing lines, and treat them as flat projections onto the XY plane. This means you can trim lines that would never meet in 3D space against each other's Front-View projections in 2D space. This is a powerful feature, but you must pay careful attention to what you are doing.

3-D Mode

Toolbox Icon:



Shifts the perspective into 3D mode, and all 3D commands become available.

Ruler

Menu: View / Ruler

Displays or hides vertical and horizontal rulers beside the drawing window, when in 2D Mode.

You can set ruler divisions with the **View Options** command. See View Options.

Colors

DesignCAD gives you 64 basic drawing colors. By default there are seven base colors in eight deepening shades each, and eight other lively colors. However, if the supplied colors don't meet your needs or fit your tastes, you can create your own.

You can open the Color Toolbox in the View options (see View Options) or by clicking the icon on the main toolbar:



The Color Toolbox has eight color buttons, an "A" button, a "pointing finger" button, and a "color palette" button.



The A (for "Apply") button applies the current drawing color to the items you have selected. The pointing finger is used to grab the color of the selected object and make that the current drawing color. The color palette lets you create a custom color based on the current drawing color. The other eight buttons are available drawing colors.

How do you get to all 64 colors? Click on one of the color buttons and hold down the button. This opens the color drawer, which shows eight colors hidden under one tool. These work like the tools in the Main Toolbox: click, hold down, slide the pointer over to the color you want, and release. The color you picked shows up at the bottom of the Main Toolbox, indicating that it is now the active drawing color. It also becomes the top color in its drawer.

If you want to edit a color, make that color the top one in its drawer. Then double-click on it to activate the **Custom Color** command (this command is also available through the Options menu - see Custom Colors).

Cursor (Crosshair)

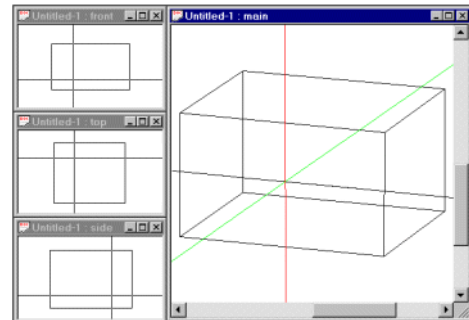
Menu: Options / Crosshair

Shortcut Key: F5

A toggle function that indicates the cursor's position with cross hairs in all views.

Using the Command

Using the **Crosshair** command helps you easily distinguish the cursor's position, in relation to objects, from several angles. This makes it easy to see the cursor's location in all three dimensions.



When you choose the command, the cross hairs become visible on the drawing screen. To remove the cross hairs, choose the **Crosshair** command.

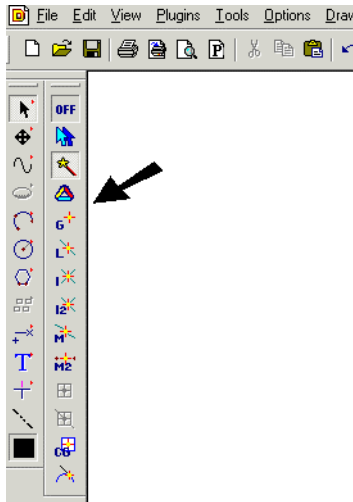
Grid and Snapping

The display grid is the mesh that is displayed while working, so that you can visualize the size and scale of your entities. The snap grid represents the increments to which the cursor will snap, when the snap grid is enabled. Snaps are points to which the cursor will jump, such as the midpoint of a line or the center of an arc.

Snap Toolbox

The Snap Toolbox (which is also dockable) contains a convenient set of tools for snapping to precise locations in your drawing. By default, it is displayed to the left of the drawing area. You can

click on these tools even if you are in the middle of a drawing command. The F10 key can be used to toggle the snap preview on and off.



Clicking the **Off** button turns off any selected snap modes. For explanations of the specific snaps, see Snap Points.

*NOTE: When Snap commands are selected from the Snap Toolbox, you have the option of making them set a point or just move to the position (**Move Cursor Only** option). However, if you select a Snap command from the **Point** menu, a point will always be set.*

Interruptible Snap

Menu: Options / Interruptible Snap Mode

Toolbox Icon:



When enabled, the behavior of the snap commands is changed so that one or more snap commands can be used to set the points for

another. If disabled, then choosing a snap command cancels the execution of an active snap command.

For example, you can start Midpoint2, then use Line Snap for the first point required for MidPoint2, and Intersect-2 for the second point). If **Interruptible Snap Mode** is disabled, choosing a snap command cancels the execution of an active snap command (e.g., if you start Midpoint2, then select and execute Line Snap, a line snap will be performed and the original Midpoint2 selection will be ignored).

When Interruptible Snap is off, any snap command cancels any active snap command. In the example above if you start Midpoint-2, then use Gravity snap, the Midpoint-2 snap will be cancelled, and one point will immediately be set at the Gravity snap location.

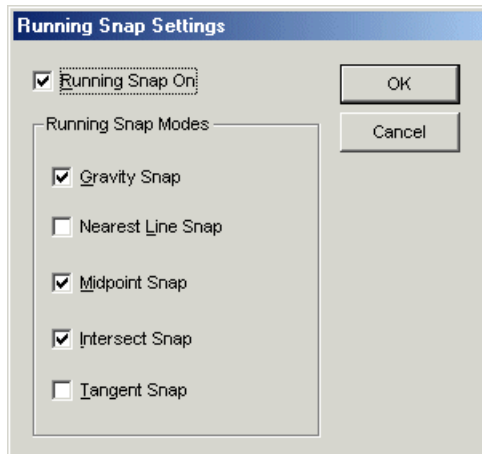
Running Snap

Menu: Options / Running Snap Settings

Toolbox Icon:



A toggle function that turns Running Snap Mode off or on, and specifies which snap locations are anticipated as the cursor moves.



NOTE: Some settings in the **Preferences** (see **Preferences**) will also affect the Running Snap preview calculations. If you plan to use Running Snaps a lot, you may find it more efficient to turn off the Preference options for snapping to entities in a block or symbol or lines in a hatch pattern. Running snaps may also cause difficulties in 3D mode, especially when 2D Selection Mode is active.

Grid Settings

Menu: Options / Grid Settings

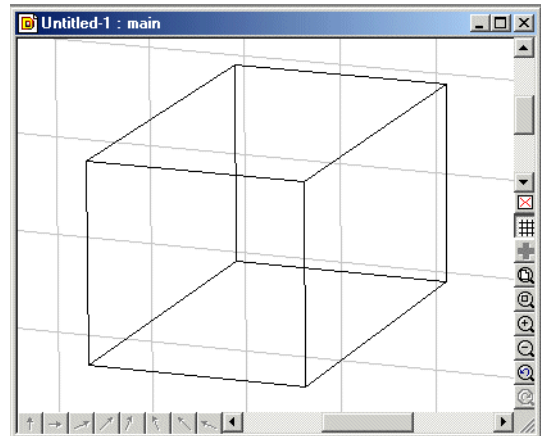
Shortcut Key: Ctrl+G

Opens the **Grid Options** folder, which contains options for both the Display Grid and the Snap Grid. The Grid sizes are measured in Drawing Units. See Grid Options.

Display Grid

Menu: Options / Display Grid

Shows a grid on the drawing screen which can be used as a reference for drawing new objects or comparing the size of existing objects.



To change grid properties, see Grid Options.

Set Grid Center

Menu: Options / Set Grid Center

Point 1: Set a point for the display grid center

Sets the location for the center of the display grid.

Using the Command

Choose the **Set Grid Center** command. Set a point anywhere on the drawing screen to specify the new center of the grid.

The display grid can be oriented on the XY, XZ, or YZ plane. To change grid properties, see Grid Options.

Snap Grid

Menu: Options / Snap Grid

Shortcut Key: G

Toggles the snap grid off and on. With **Snap Grid** enabled, any time you set a point with the mouse, the point is set at the nearest location on the grid.

Snap Preview

Menu: Options / Snap Preview

Shortcut Key: F10

Toggle the snap preview on and off. If a snap command is used in conjunction with a drawing command when the snap preview is enabled, a small box indicates the point to which the cursor will snap if the left mouse button is clicked at the current cursor location.

Cursor Movement Options

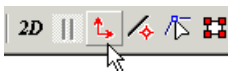
This section covers two ways you can constrain cursor movements - orthogonal mode and angle constraints.

Orthogonal Mode

Menu: Options / Orthogonal Mode

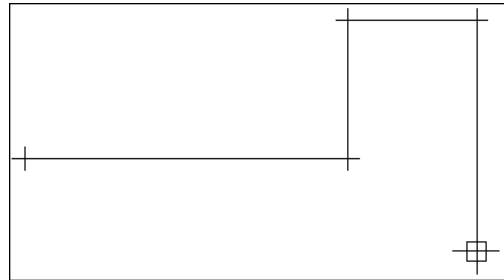
Shortcut Key: H

Toolbox Icon:



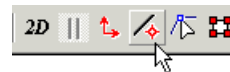
Constrains consecutive points in drawing commands so that the alignment of a pair of points is parallel to the X, Y, or Z axis. If you set points that are not aligned parallel to one of these axes in Orthogonal Mode, DesignCAD

automatically shifts the second point of the pair so that the points are aligned parallel to the nearest matching axis.



Angle Constraint Mode

Toolbox Icon:



Restricts point placement in drawing commands to a specified set of angles from the previous point. The set of angles used may be set using the **Angle Constraint Settings** Command.

Using the Command

Activate **Angle Constraint Mode**. Select a drawing command and set a beginning point. Set the number of points required or desired to draw the entity. Angle Constraint Mode will remain active until the command is selected again to turn it off.

Example: Draw a line with segments at 0, 30, 90, 150, 180, 210, 270, or 330 degrees.

Activate **Angle Constraint Mode**. Select the **Line** command and set a point on the screen. Move the cursor around slowly. Notice that the rubber-band line only moves in one of eight directions. Set a point for the body of the line. Slowly move the cursor around. Again, notice

that the rubber-band line only moves in one of eight directions. Set another point. Press Enter to complete the command and add the line to the drawing.

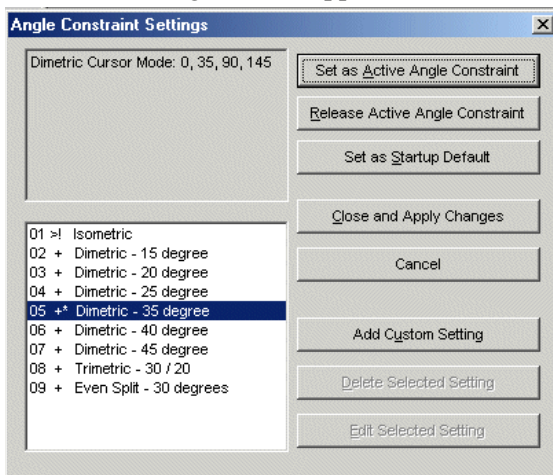
Angle Constraint Settings

Menu: Options / Angle Constraint Settings

Used to change the settings used in Angle Constraint Mode. Default settings may be used or custom settings may be added and used.

Using the Command

Select the **Angle Constraint Settings** command and the following window appears:



Set as Active Angle Constraint: Sets the selected set of angle constraints as the current one.

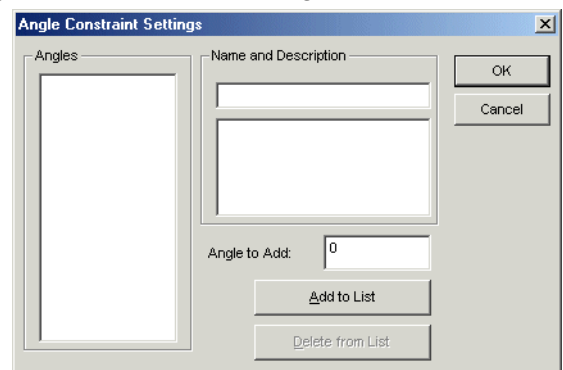
Release Active Angle Constraint: Disables all angle constraints.

Set as Startup Default: Makes the current set of angle constraints the default set. The next time you load the program, you won't have to load this set of angle constraints to use them. They will be used when Angle Constraint Mode is enabled.

Close and Apply Changes: Closes the window. DesignCAD is ready to use the selected set of angle constraints as the current one.

Cancel: Closes the window. DesignCAD is ready to continue using the set of angle constraints that was being used before the window was opened.

Add Custom Setting: Opens a window that lets you add sets of custom angle constraints.



- **Angles:** A list of the angles that will be used in the new set of angle constraints.
- **Name and Description:** The name and description that will be used for the new set of angle constraints.
- **Angle to Add:** Enter an angle in this box and then click on the **Add to List** button to add it to the angle constraint set.
- **Delete from List:** Removes the selected angle measurement from the list of angles in the **Angles** area.

Delete Selected Setting: Deletes the currently selected set of angle constraints. Only custom sets of angle constraints may be deleted.

Edit Selected Setting: Opens the currently selected set of angle constraints so that it may be edited. Only custom sets of angle constraints may be edited.

Calculator

Menu: Tools / Calculator

Allows you to perform calculations on the screen. Once the computation is made, you can copy the result and paste it into a drawing.

Using the Command

Choose the **Calculator** command. In the calculation box enter the values and operators for the calculation. Then press Enter or click on the **Compute** button. DesignCAD makes the calculation and displays it in the box. Click the **Close** button to end the command.

The expression entered in the calculation box can contain a formula or expression. The following are examples of valid expressions:

45*23

(6+23)*4

SQRT(9)

SIN(45)

The expression can contain mathematical functions and the following operators:

+ Addition

- Subtraction

* Multiplication

/ Division

^ Raises a number to a power ($2^3 = 8$)

Inserting a Calculation into a Drawing

Select the text in the calculation box if it is not already selected. Press Ctrl+C to copy the text to the Clipboard. Click the **Close** button to return to your drawing.

Next, choose the **Paste** command (Ctrl+V). As soon as you do, a rubber-band text box appears, showing how the text will be inserted. Set a point for the lower-left corner of the text. The text is inserted into the drawing.

You can also insert the results of a calculation into a drawing by using one of the text commands which allow you to change the font, style, alignment, size, and angle. First, copy the text from the calculation box to the clipboard. Then choose the **Text** command (see Text) and use paste to enter the text into the proper fields.

DesignCAD Workspace

The Workspace refers to a set of properties of the user interface. You can define multiple workspaces for different users, or for different work scenarios.

Setup DesignCAD Workspace

Setup DesignCAD Workspace Command

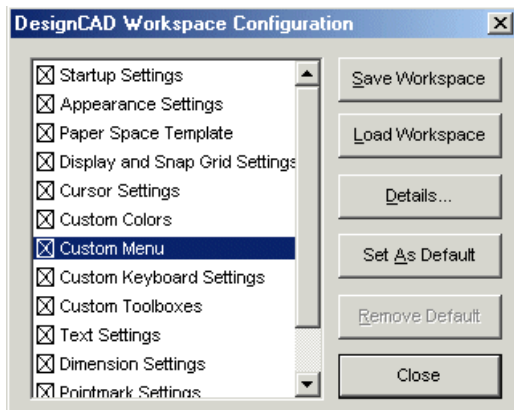
Menu: File / Workspace Configuration / Setup DesignCAD Workspace

The Setup DesignCAD Workspace command is used to configure various sets of DesignCAD settings so that they can be saved to disk and then retrieved in the future. This feature lets you

simply open a set of options instead of having to go through and set many options at the beginning of every drawing session.

Using the Command

Select this command and the following window appears:



To save a group of settings in the current Workspace template, place an X in the box to the left of the group name by clicking on it with the mouse. To remove a group of settings from the current Workspace template, click on the box to remove the X in the box to the left of the group name. If you want to change a group of settings, click on the group name to highlight it and then click on the **Details** button.

Save Workspace: Save the settings as a Workspace. The Save DesignCAD Workspace window opens. Enter a name for the file and where you want to save it. Then click **Save**.

Load Workspace: Base a new Workspace on an existing one.

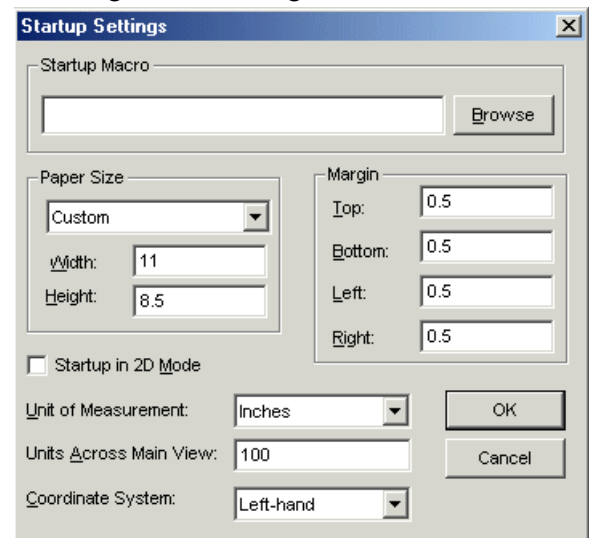
Details: To change a group of settings, click on the group name to highlight it and then click the **Details** button.

Set as Default: Sets the current workspace template so it will be opened automatically at the beginning of every drawing session.

Remove Default: Reverts to the original DesignCAD settings.

Startup Details

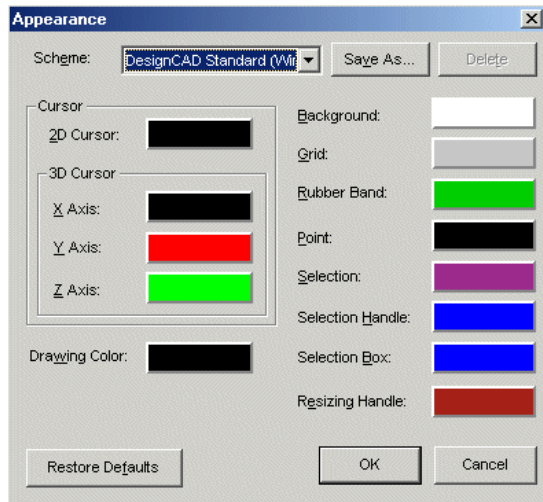
This group of settings controls the startup settings for DesignCAD drawing sessions.



Use this window to define the startup macro, paper size, margins, 2D/3D mode, units, and coordinate system.

Appearance Details

This group of settings controls the colors of various DesignCAD elements.



Click on the color sample to the right of the name of the element for which the color is to be changed and select the new color from the palette that appears.

Paper Space Template Details

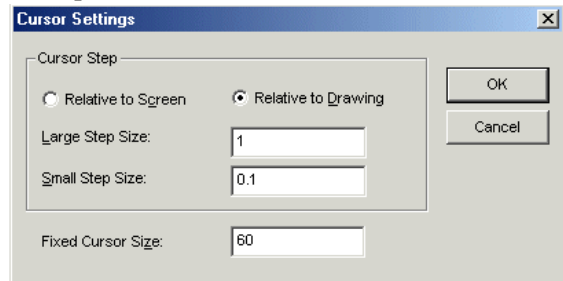
Select the Paper Space Template to be used with the current DesignCAD Workspace.

Display Grid / Snap Grid Details

This group of settings determines the settings for the Display and Snap Grids that will be used when the current DesignCAD Workspace is selected.

Cursor Details

The size of the cursor and the small and large step sizes that will be used in the current DesignCAD Workspace.



Custom Colors Details

This group of settings determines the custom colors that will be available in the drawings that use the current DesignCAD Workspace.

Custom Menu Details

Select the Customized Menu file to be used by clicking on the **Browse** button.

Custom Keyboard Details

Select the Customized Keyboard file to be by clicking on the **Browse** button.

Custom Toolboxes Details

Select the Customized Toolboxes file to be used by clicking on the **Browse** button.

Text Details

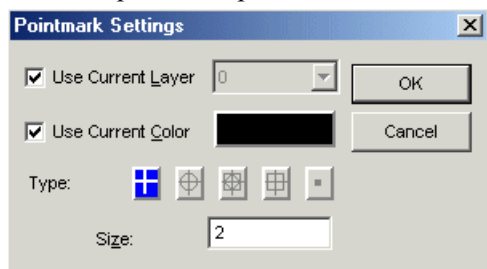
The default color and style of regular text and Attribute text for the different layers of drawings.

Dimension Details

The options for the **Dimensioning** commands, **Pullout** command, and **Balloon** commands.

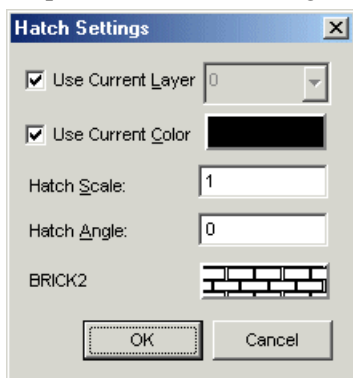
Pointmark Details

The default options for pointmarks.



Hatch Details

The default options for line hatching.



Load DesignCAD Workspace

Load DesignCAD Workspace Command

Menu: File / Workspace Configuration / Load DesignCAD Workspace

Used to load or change a DesignCAD Workspace.

Using the Command

Choose the command, and Select the workspace template you wish to use and click OK. A window appears to ask if you want to use the selected template as the default workspace template (when new drawings are started and in subsequent drawing sessions). space Command

Screen Configuration

Menu: File / Screen Configuration / Save

Saves the screen configuration of the current drawing. The window configuration and the view configuration for each window are saved with this command.

Using the Command

Choose the command, and the **Save View File** box appears. In the **File Name** box enter the name of the view to save. In the **Save In** box select the location where you want to store the view. Then click **Save**.

You can then load a saved configuration using this command:

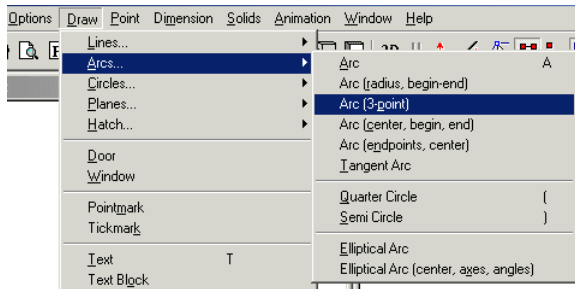
Menu: File / Screen Configuration / Load

Accessing Commands

There are several ways to access commands - command menu, keyboard shortcuts, toolbox icons, and the command line.

Command Menu

To access menu commands using the mouse, click on the menu title of interest. This action pulls down the list of commands available on that menu. You can now pick the command you want by clicking on it.

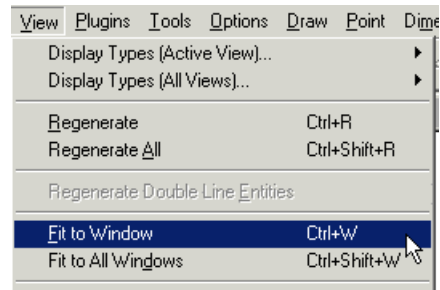


For the keyboard-conscious, menu commands are also available from the keyboard. You access a menu by pressing Alt+letter, where letter is the underlined letter in the menu title. To access the **F**ile menu, for example, press Alt+F. You then see a list of available commands, each of which also has an underlined letter. To choose a specific command when the menu is open, press the underlined letter in the command name. To use the **O**pen command in the **F**ile menu, for example, press Alt+F, then O.

Keyboard Shortcuts

For those who like to take shortcuts, many of the commands in DesignCAD have a shortcut key. It's often much faster to go directly to a command using a shortcut rather than using the Command Menu. For example, by pressing Ctrl+O you can bypass the menu and immediately use the **O**pen command.

Keyboard shortcuts are listed beside the menu commands. For example, **F**it to **W**indow has CTRL+W beside it on the **V**iew menu.



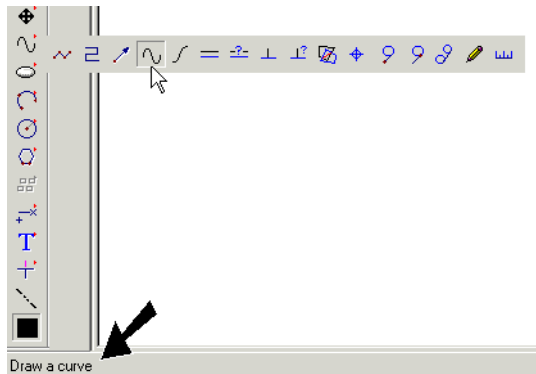
To customize keyboard shortcuts, see Keyboard Options.

Toolboxes

DesignCAD has a convenient set of icon toolboxes that provides push-button access to many frequently needed tasks. To use one of these tools, click on it with the mouse.

To see the commands stored in a drawer, click on the visible tool and continue to depress the mouse button. After a moment, the drawer slides open, revealing the tools it contains. Keeping the mouse button down, move the cursor onto the tool you want to use, and then release the mouse button.

As you move the cursor over a new tool, the Status Bar at the bottom of the screen gives a brief description of that tool.



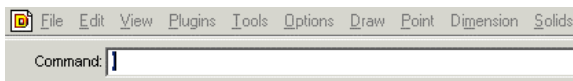
The most recently used command in a drawer is the one that shows in the toolbox. Like most of the toolboxes in DesignCAD, the Main Toolbox can be docked to an edge of the drawing screen or, if already docked, pulled loose to float anywhere on screen.

Command Line Entry

Menu: none

Shortcut Key: Spacebar

Lets you choose a command by typing the name of the command, or an alias, in the Command Line.



With this method you don't have to select commands from the menu or by other means. Also, the program keeps a list of recently entered commands. Once you have entered a command, you can easily choose it again by pressing the Spacebar and using the up or down arrow keys to scroll to the command.

Using the Command

Press the Spacebar to display the Command Line. Enter the name of the command or alias in the box and press Enter. If any options are associated with the command you have chosen, the window appears as usual. Press the Tab key to enter the window and navigate to the option you want. Set the options and then press Enter to return to the drawing.

NOTE: Many of the commands let you enter the parameters for the command on the same line in the Command box.

Editing the Command File

The program associates command names and aliases with command IDs in the MACROCMD.INI file. When you enter a custom command name, the program still identifies it with the real command name. This gives you a great deal of flexibility in customizing commands. You can even have multiple aliases for the same command.

Commented lines begin with a semicolon. They are not allowed on the same line as an alias. Please use caution when editing the MACROCMD.INI file. Make a backup of the file before you try to edit it. Edit only the Add Custom Command section, part of which is shown below. If you edit other sections, you may cause commands to stop working correctly.

```

;=====ADD CUSTOM COMMAND NAME
ENTRIES HERE=====

```

```
;
```

```
;These sample commands are laid out following
the menu's structure.
```

```
;  
; File Menu  
;  
Nu=ID_DCAD_FILE_NEW  
Ld=ID_DCAD_FILE_OPEN  
QF=ID_DCAD_FILE_CLOSE  
Sv=ID_SAVEFILE  
SvA=ID_SAVEFILE_AS  
SvS=ID_SELECT_SAVE
```

Creating an Alias

First, create a blank line for the new alias. Then enter the alias you want to use followed by the equals sign (without spaces) and the actual name of the DesignCAD command. It is not necessary to comment out the existing alias.

Suppose you want to add Clo as an alias for the **File Close** command. Go to the end of the following line:

```
QF=ID_DCAD_FILE_CLOSE
```

Press Enter to start a new line. Now enter the new alias as shown below:

```
clo=ID_DCAD_FILE_CLOSE
```

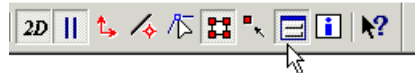
You may, of course, copy the DesignCAD command name and paste it in instead of typing it yourself. Save the file when you are sure the information is correct. The next time you start DesignCAD, it reads the file you have edited. Then you can press the Spacebar and enter “clo” in the Command Line to close a file.

Example: Draw a line from the origin (0,0) to (10,10).

Press the Spacebar to activate the Command Line. Start the **Line** command by entering “Line.” Then press the Enter key. Next, press the Spacebar again. In the Command Line enter pointxyz 0,0,0 and press Enter to set the first point. Press the Spacebar a third time. The previous entry (pointxyz 0,0,0) remains in the Command Line. Now change the 0,0,0 to 10,10,0 and press Enter. That sets the second point. Press Enter again to end the Line command. The line is drawn between those two points.

Command Window

Toolbar Icon:



A toggle which displays or hides the window for the current drawing command.

Using the Command

Select a drawing command. The **Command Window** button is pressed in, and if there are any options for that command the window appears. Click on the Command Window button to release it, and the window for the drawing command is hidden and your view of the drawing screen is unobscured. If you decide that you need to set the options for the drawing command, click the button again and the window will reappear.

Getting Help

There are several ways to get help with DesignCAD while you are working.

DesignCAD Help

Menu: Help / DesignCAD Help

Shortcut Key: F1

To get help quickly as you work, choose the **DesignCAD Help** command. When the **Help** window appears, you can display information on DesignCAD commands, or get a definition for a term you do not understand. You can also get help whenever you see an error message or if you need more information about an option in a window.

With Help, you can:

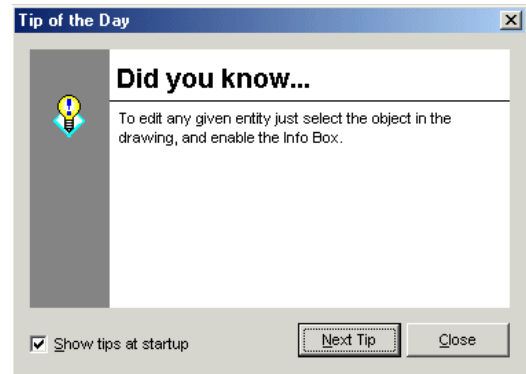
- Search for topics associated with a word or phrase.
- View instructions in Help while you perform a task in DesignCAD.
- Add your own notes to a Help topic.
- Mark topics you often use so you can jump to them quickly.
- Print a copy of a Help topic.

You can view Help side by side with a drawing so that both windows remain fully visible while you work. To display Help and your drawing at the same time, resize the windows so they do not overlap.

Tip of the Day

Menu: Help / Tip of the Day

Opens the **Tip of the Day** window, showing various tips for using DesignCAD.



To view another tip, click **Next**. Click **Close** to close the Tip of the Day and return to DesignCAD.

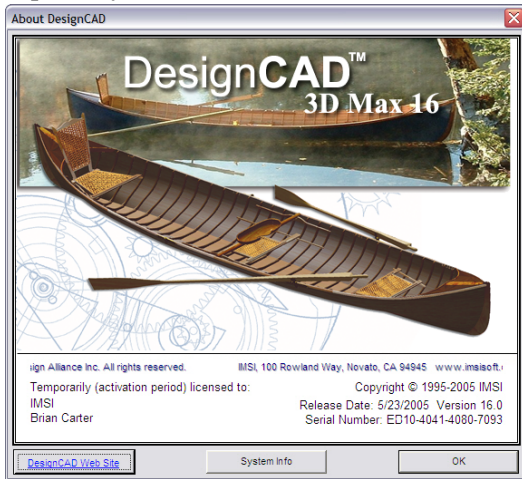
If the **Show Tips at Startup** option is checked, the Tip of the Day will appear when you start DesignCAD.

About DesignCAD

Menu: Help / About DesignCAD

Displays information about the program, including the DesignCAD release date, who the program is registered to, and the serial number. The command also offers a direct Internet links

the DesignCAD home page on the World Wide Web. You can also find information about your computer system.



DesignCAD Website: If you have an internet connection, you can go directly to the DesignCAD home page.

System Info: Basic information about your computer's operating system, memory, and free space on your hard drives.

File Options

This section covers what you can do with files: opening, closing, saving, importing, and exporting.

Opening, Closing, and Saving

Before starting to work in DesignCAD, you need to know how to create a new file, open an existing file, and save files.

New

Menu: File / New

Shortcut Key: Ctrl+N

Toolbox Icon:



Opens a new drawing document. It does not close any drawing that you already have open.

Using the Command

Choose the **New** command. The **New File Wizard** appears.



Open a Recently Used File: Browse for or select a DesignCAD file from a list of the most recently used files. Once you have selected a file, click OK to open it.



Create a New Drawing: Start a new drawing. A blank drawing appears on the screen, but it does not close the current drawing. You can switch from one drawing to the next using Ctrl+F6.



Paper Space Template: Start a new drawing using a Paper Space template. Select the template you wish to use and click OK.



Specific Settings: Start the wizard that guides you through a series of options for a new drawing.

- The first step lets you select format and precision settings for distance measurements.
- The next step lets you select format and precision settings for angular measurements.
- The third step deals with Snap Grid and Display Grid settings; it also allows you to set the width of the drawing screen in Units.
- The fourth step lets you set print options, these include: Print Units, Paper Margins, and Paper Orientation. Large and Small Cursor Step Sizes are set in the fifth step.

If the **Save as Default** option is enabled, all subsequent new drawings will use the settings you have selected. If this option is disabled, the settings will only be used in the drawing you are starting. Once you have finished setting all of the options, click **Finish**.

Show New File Wizard: Uncheck this if you do not want to see the New File Wizard again. If you decide to turn the New File Wizard back on later, select **Options / Options**, open the **View** tab, and check **Show New File Wizard**.

Open

Menu: File / Open

Shortcut Key: Ctrl+O

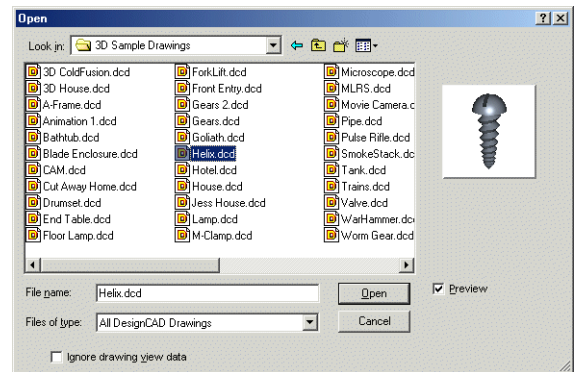
Toolbox Icon:



Opens a drawing file and loads it onto the screen as the current drawing.

Using the Command

Choose the **Open** command.



In the **Look In** box select the location of the file. In the **Files of Type** box select the type of file you want to open. Enter or select the name of the file you want to open. Click the **Open** button when you have entered the necessary information.

If you already have a drawing on the screen, it will not close that drawing but will open a second drawing. If you already have several documents open and try to start a new one, DesignCAD may warn you that it's not possible to another document.

In this case, close one or more of your drawings, or other open applications, and try **Open** again. The number of documents that you can have open depends on the amount of memory on your system, the number of other applications that are open, the complexity of each drawing, the number of view windows you have opened for each drawing, and other factors.

Ignore Drawing View Data: DesignCAD will ignore view angles and distances at which the drawing was previously saved. DesignCAD loads the drawing using the default views and viewing angles for the appropriate drawing mode (i.e., 2D Mode or 3D Mode).

Save

Menu: File / Save

Shortcut Key: Ctrl+S

Toolbox Icon:



Saves the active drawing to disk. If the drawing is new and has not yet been saved, you will be asked to name the file you want to save.

Using the Command

Choose the **Save** command. If this is your first time to save the drawing, the **Save As** box appears. Enter the name of the file you want to save and where you want to save it. Then click OK. If you have saved the drawing before, the **Save** command saves all changes to the drawing since the last time you chose the **Save** command.

The drawing handles are automatically placed at the lower left, lower right, and upper right front of the drawing. The **Set Drawing Handles** (see Set Drawing Handles) command can be used to specify the specific handle locations for the drawing.

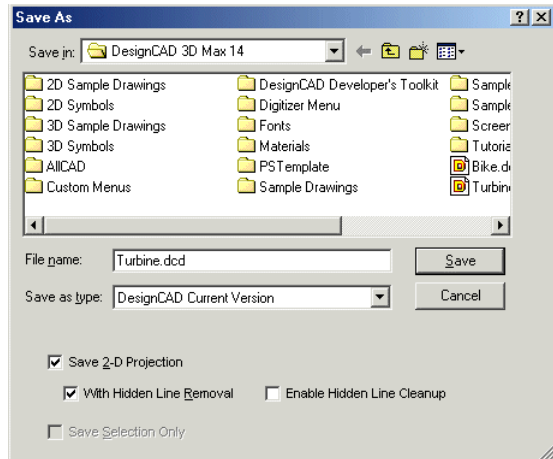
Save As

Menu: File / Save As

Saves the active drawing to disk.

Using the Command

Choose the **Save As** command. The **Save As** window appears.



In the **File Name** box enter or select the name of the file to save. In the **Save In** box enter the location where you want to store the drawing. If you want to save the drawing as a file type different than the one you're working in, select the type in the **Files of Type** box. Then click **Save**.

Save 2D Projection: Saves your 3D drawing as a 2D projection, so it can be opened as a 2D drawing.

With Hidden Line Removal: Only available if you have checked the **Save 2D Projection** option. This option will remove all hidden lines from the resulting 2D file.

Enable Hidden Line Cleanup: When saving with Hidden Line Removal, this option automatically cleans up the hidden line drawing. This removes many of the duplicate lines in the projected drawing.

Save Selection Only: Select the portion of the drawing to be saved as a separate file, then use **Save As** with this option checked.

Save a Copy

Menu: File / Save a Copy

Saves a copy of the active drawing under a new name, but doesn't change the name of the active drawing. This makes it easy to save incremental backups.

Using the Command

Choose the **Save a Copy** command. The **Save Copy As** window appears. For explanations of the options in this window, see **Save As**.

Save as Symbol

See Save As Symbol.

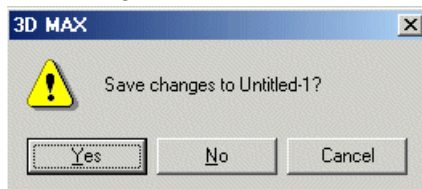
Close

Menu: File / Close

Closes an open drawing.

Using the Command

Choose the **Close** command. If the drawing has changed, the program gives you the opportunity to save the changes.



Clicking **Yes** saves the changes. Clicking **No** closes the drawing, dropping all modifications since it was last saved. Clicking **Cancel** or pressing Esc cancels the **Close** command and returns you to the drawing screen.

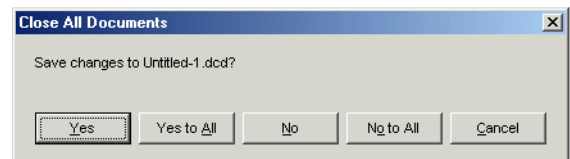
Close All

Menu: File / Close All

Closes all open drawings.

Using the Command

Choose the **Close All** command. If any of the drawings have changed, the program gives you the opportunity to save the changes.



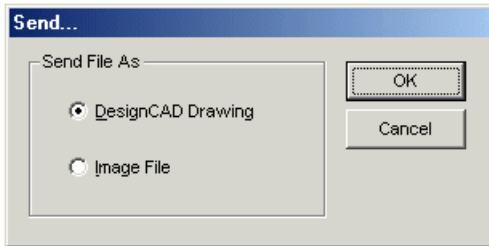
Clicking **Yes** saves the changes. Clicking **Yes to All** saves all of the changed drawings. Clicking **No** closes the drawing, dropping all modifications since it was last saved. Clicking **No to All** closes all of the drawings, dropping all modifications since they were last saved. Clicking **Cancel** or pressing Esc cancels the **Close All** command and returns you to the drawing screen.

Send Current Documents and Send All Open Documents

Menu: File / Send / All Open Documents

Menu: File / Send / Current Document

A Windows function that lets you send all the open drawings, or just the current document, through Microsoft Exchange for electronic mail and fax functions.



You can send the full DesignCAD drawing, if desired, or to send an image (jpg format) of the currently active view.

For more information, please refer to your Windows documentation.

Exit

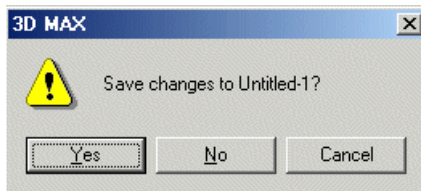
Menu: File / Exit

Shortcut Key: Alt+F4

Closes DesignCAD.

Using the Command

Choose the **Exit** command. If you have any unsaved drawings open, the program gives you the opportunity of saving each one before the program closes.



Choosing **Yes** activates the **Save As** command. Choosing **No** closes the program (or prompts you to save the next drawing if multiple drawings were open). Choosing **Cancel** returns you to the DesignCAD drawing screen with no change.

Importing and Exporting

In addition to DesignCAD's native format, you can work with files from different CAD applications.

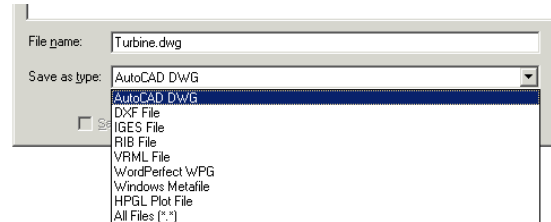
Export

Menu: File / Export

Lets you export files in several formats.

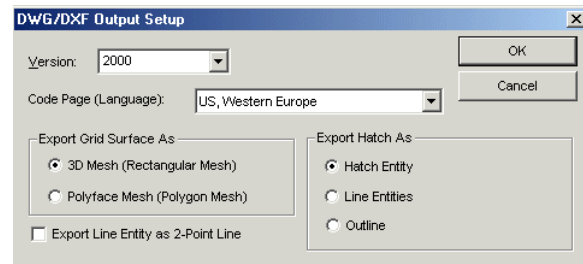
Using the Command

Select the **Export** command and select the type of file in the File of Type box.



DesignCAD exports to the following file formats:

DWG and DXF: AutoCAD formats.



Select the desired version from the **Version** box. Select the desired mesh type from the **Export Grid Surface As** area. Select the desired hatch type from the **Export Hatch As** area. The **Export Line Entity as 2-Point Line** option determines whether multi-segment lines will be exported as separate 2-point lines for each segment or if multi-segment lines will be exported as polylines.

HPGL: The Hewlett Packard graphics language. HPGL options can be set once you have selected HPGL in the SAVE AS TYPE box and clicked the SAVE button in the window.

IGES: Initial Graphics Exchange Specifications, a standard format that many CAD systems support.

WMF: Windows Metafile. Many Windows applications can read Windows Metafiles.

WPG: WordPerfect Graphic

RIB: RenderMan rendering package.

VRML: WRL format for Internet graphic files that can be read by any 3D web browser.

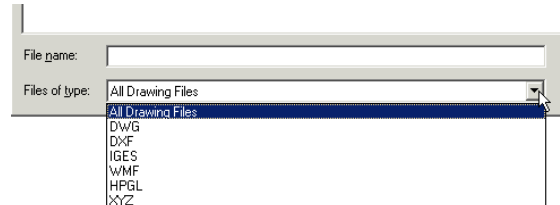
Import

Menu: File / Import

Lets you import files in several formats.

Using the Command

Choose the **Import** command. In the **Files of Type** box, find the type of file you want to import, locate the file in the browser, and click **Open**.



DesignCAD imports the following file formats:

DWG and DXF: AutoCAD formats. Blocks are converted to DesignCAD blocks, with the following limitations:

- DesignCAD explodes nested blocks from the original drawing. Only the outer block level is retained.
- Rotated attributes in scaled blocks are not "skewed" as they are in AutoCAD. Example: There is a 45-degree attribute definition in a block definition. Insert the block at X-scale=1.5, Y-scale=1.0. The Attribute gets distorted. When imported into DesignCAD, the attribute is not distorted; it remains at its original angle (relative to the block) and text size.

IGES: A standard format that many CAD systems support. DesignCAD supports the following IGES entities for input:

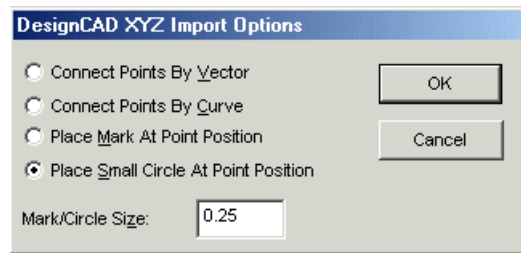
- 100 - Circular Arc
- 102 - Composite Curve
- 104 - Conic Arc
- 106 - Copious Data
- 108 - Plane
- 110 - Line
- 112 - Parametric Spline Curve
- 114 - Parametric Spline Surface
- 118 - Ruled Surface
- 120 - Surface Revolution
- 122 - Tabular Cylinder
- 124 - Transformation Matrix
- 212 - General Note
- 214 - Arrow
- 318 - Define Subfigure
- 408 - Insert Subfigure
- 412 - Rectangular Array

WMF: Many Windows applications can read Windows Metafiles, and the Import command allows you to bring those files into DesignCAD.

HPGL: The Hewlett Packard graphics language. You can configure other applications for an HP plotter, send the plotter output to disk, and import that file into DesignCAD.

XYZ (or XY): DesignCAD can import text files containing X,Y or X,Y,Z coordinates. The following conditions apply to the Import XYZ command:

- The coordinates for each point should be on one line. They must be separated by either a space, a comma, a semi-colon, or a tab.
- Individual line or curve entities must be separated by at least one empty line in the text file.
- Any line which starts with a semi-colon is treated as a comment.
- Comments may appear anywhere in the file.



You can connect the points with straight lines or smooth curves, or you can mark their position with a plus sign or small circle. If you choose to mark the points, you can set the size of the mark or circle in the edit box. The size is set in Drawing Units.

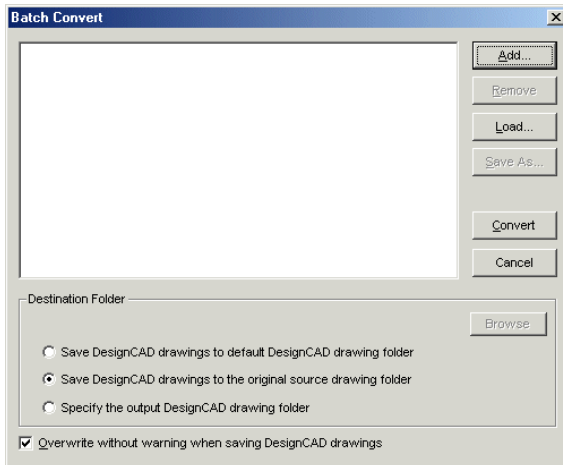
Batch Convert

Menu: File / Batch Convert

Makes it possible to convert several files of various types into DesignCAD format, all at one time.

Using the Command

Choose the **Batch Convert** command and the following window appears:



Add: Selects the files to be added to a batch.

Remove: Removes files from a batch.

Load: Loads a batch (.dbx) file that has been created and saved previously.

Save As: Saves a batch (.dbx) file under a specified filename.

Overwrite without warning when saving DesignCAD drawings: Overwrites an existing DesignCAD drawing with the same name as a drawing that is being converted, without first prompting the user.

Save DesignCAD drawings to default DesignCAD drawing folder: Places the converted DesignCAD drawings in the default DesignCAD drawing folder (this is the

DesignCAD installation location unless you've changed it in the **File Locations** folder in the **Options** window) when the Convert button is clicked. Otherwise the drawings are placed in the same directory as the files that are being converted.

Convert: Closes the window and begins converting the batch.

Program Options

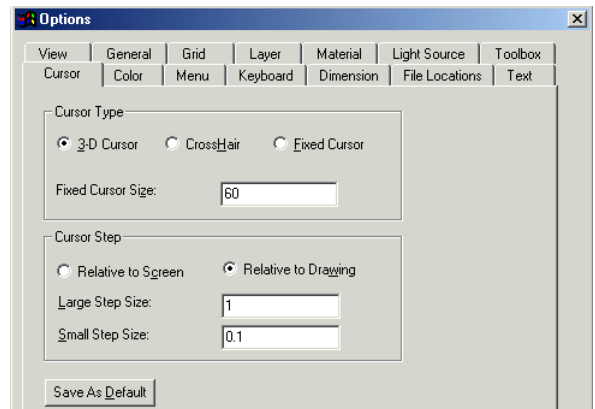
Menu: Options / Options

Keyboard Shortcut: Q

Cursor Options

Menu: Options / Cursor

Set the size of the cursor and the small and large step sizes for it.



Cursor Type

- **3D Cursor:** Causes the cursor to appear closer or further away according to its location on the Z axis.
- **Crosshair:** Turns the cursor into a pair of cross hairs that extend to the ends of the screen.

- **Fixed Cursor:** Causes the cursor to remain one size during the drawing session.
- **Fixed Cursor Size:** Sets the size of the cursor in Drawing Units.

Cursor Step:

Relative to Screen: The cursor will always move the same number of pixels across the screen, regardless of the zoom factor.

Relative to Drawing: Makes the cursor always move the same number of Drawing Units, regardless of the zoom factor.

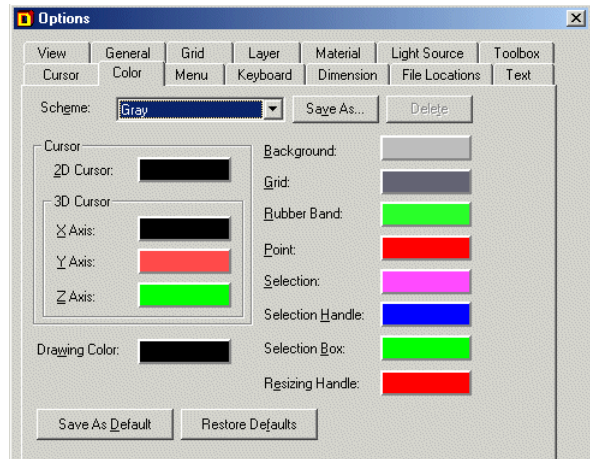
Large Step Size: Sets the number of Drawing Units that the cursor moves when you press the Arrow keys or Ctrl+Home or Ctrl+End.

Small Step Size: Sets the number of Drawing Units the cursor moves when you press Shift while using the Arrow keys or Ctrl+Home or Ctrl+End.

Save As Default: Saves the changes to the next session

Color Options

Set the color of the background, cursors, grid, points and various selection and rubber-band entities on the drawing screen.



Scheme: Select any of the provided color schemes. If you make changes to colors, you can save a new custom scheme by clicking **Save As** and assigning a name. Custom schemes can be deleted.

Click the **Color** box next to any screen entity to select a new color for it.

Cursor: Sets the color of the 2D cursor, and for the three axes of the 3D cursor.

Drawing Color: The default drawing color.

Background: The color used as the background of the DesignCAD drawing windows.

Grid: The color of the drawing grid. This can also be set in the **Grid** options, see Grid Options.

Rubber Band: The color of rubber-band entities.

Point: The color of point indicators used during drawing commands.

Selection: The color of entities when selected.

Selection Handle: The color of a selection handle when a point or drawing entity is selected.

Selection Box: The color of the selection box that appears when the cursor is used to drag an area around objects to be selected.

Resizing Handle: The color of resizing handles that appear around selections when the **Option /Use Resizing Handles** (see Use Resizing Handles) is enabled.

Restore Defaults: Return to the original color settings.

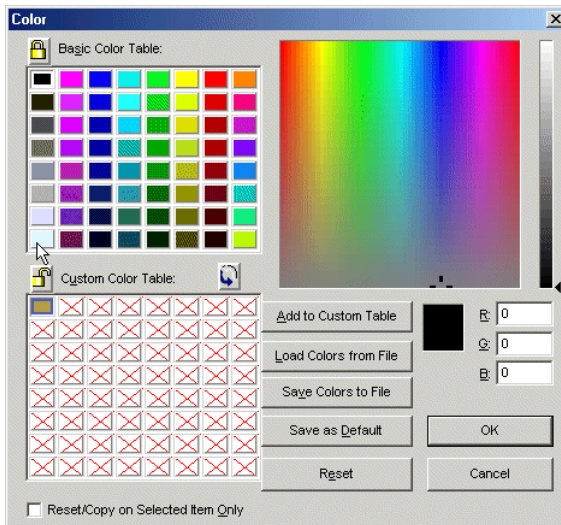
Custom Colors

Menu: Options / Custom Color

Used to edit the currently selected drawing color, the Basic Color Table, and the Custom Color Table.

Using the Command

Choose the **Custom Color** command.



Move the Color Selector to change the color in the Color box. Moving left or right will change the hue of the color and moving up or down will change the saturation of the color. The brightness of the color can be adjusted by moving the arrow beside the brightness bar. A custom color can also be set by entering new values in the Red, Green, and Blue boxes.

Add to Custom Table: Adds the color in the selected cell to the Custom Color Table.

Load Colors From File: Loads a custom color file.

Save Colors to File: Saves the current Basic Color Table and Custom Color Table to a file that can be loaded.

Save as Default: Saves the Basic Color Table and Custom Color Table in new drawings and subsequent DesignCAD drawing sessions.

Reset: Restores the original Basic Color Table and Custom Color Table.

Reset/Copy on Selected Item Only: Only the selected color cells will be reset or copied when a function is performed. When the selected color is the one you want to assign as the current drawing color, click OK or press Enter. That color appears in the Main Toolbox as the current color, and new objects will be drawn in that color.

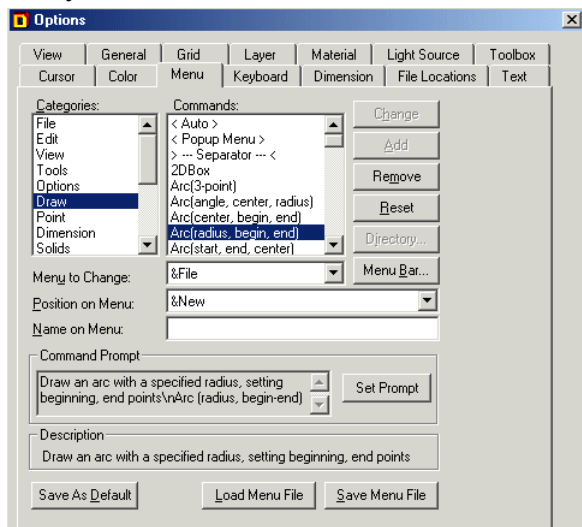
- Use the **Lock** icon to make either the Basic Color Table or the Custom Color table editable.
- To move colors in the Basic Color Table to the Custom Color table, make sure that the Basic Color Table is locked, the Custom Color Table is unlocked, and click the **Swap** button.

- To move colors in the Custom Color Table to the Basic Color table, make sure that the Basic Color Table is unlocked , the Custom Color Table is locked, and click the **Swap** button .
- To move colors in the Basic Color Table to the Custom Color table and colors in the Custom Color Table to the Basic Color Table, make sure that both tables are unlocked and click the **Swap** button .

NOTE: Any time the Basic Color Table is changed, the corresponding drawers in the Color Toolbox will also change.

Menu Options

Allows you to customize how commands appear on menus. You can also add commands you create to any menu.



For example, you can group often-used commands on a particular pull-down menu or create a command with the BasicCAD

programming language and add it to a menu. You can also record a macro and assign it to one of the pull-down menus.

To find the menu where the command you wish to add is located, click the appropriate menu in the **Categories** box. These correspond to the commands on DesignCAD's main menu. When a main menu is selected in **Categories**, commands on its corresponding pull down menu will appear in the **Commands** box.

When you're looking for an executable file, a macro, or a symbol in the lower three entries in the **Categories** box, the label on the **Commands** box changes to Files. The instructions below refer to the **Commands** box, but they also apply to the **Files** box. The box name changes depending on what category you're looking at, but the box's function remains the same.

Change: To change a command name, select the main menu in which the command you wish to change is located in the **Menu to Change** box. Select the command to change in **Position on Menu** box. Click in the **Name on Menu** box, enter a new name, then click **Change** and **OK**.

Add: When you add a command to a pull down menu, it's always inserted above the command that appears in the **Position on Menu** box. In the **Categories** box, select the main menu that contains the command you wish to add. Choose the command you wish to add to the menu from the list of possible commands in the **Commands** box. Select the main menu you want to add the command to in the **Menu to Change** box. Choose where you want to place the new command in the **Position on Menu** box. (You'll want to select the command that will be below your new

command.) Click in the **Name on Menu** box, enter a name for the command. Click **Add**, then **OK**.

*NOTE: If you want to set the prompt for the new command using the **Set Prompt** option, you must do so before clicking the **Add** button. To add an executable file, you must click the **Directory** button to find the file's location on the hard drive.*

Remove: To delete a command from a menu, scroll down and select the main menu in the **Menu to Change** box. Scroll down and select the command you want to remove in the **Position on Menu** box. Click the **Remove** button, then click **OK**.

Reset: Returns modified menus to their original DesignCAD defaults.

Menu Bar: You can add, remove or rename entire categories of commands from the main menu command bar by clicking the **Menu Bar** button, which brings up the **Menu Bar** window. For example, you could create a set of commands to handle certain symbols, and then add a command category named **Symbols** to the menu bar between the **Window** and **Help** headers.

To do this, enter the name of the new command category in the **Name on Menu Bar** box, select where you want to place the command in the **Position on Menu** box, then click the **Add** button and click **OK**. You may now add specific pull down menu commands to the **Symbols** category by using the **Add** command.

Any change made to a main menu category will affect all its pull down menus. For instance, when you remove a main menu command, all its corresponding pull down menu commands are removed as well.

Set Prompt: Lets you enter a prompt for the command you are currently adding to the menu. Type the prompt in the **Prompt** box then click **OK**. Now click **Add** to add the command to the menu.

*NOTE: The **Set Prompt** option will only work on the command that is currently being added to the menu. It will not work on commands that are already in the menu. To change the prompt for a command already in the menu, you must set the prompt while adding another instance of the command to the menu and then remove the old instance.*

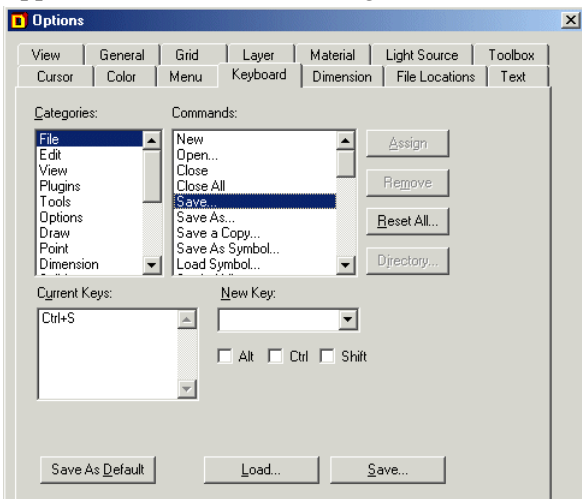
Save as Default: Saves the menu changes for the next time you run DesignCAD. Otherwise, your menu modifications will be in effect only for the current session and will be lost when you close DesignCAD.

Load Menu File: Load a previously saved menu file.

Save Menu File: Save the menu settings as an external file. This file may then be used for one or more DesignCAD Workspaces.

Keyboard Options

Lets you change a shortcut key or assign one to any menu command, executable file, or other application. Shortcut keys let you go directly to a command from the keyboard. You can also use them to start an executable file - such as a Visual Basic program or another DesignCAD application - from within DesignCAD.



To find the command you're making a shortcut for, click the appropriate menu command in the **Categories** box. These correspond to the commands on the Main Menu.

NOTE: If you assign shortcut keys to other applications in Windows, they may take precedence over the assignments in DesignCAD.

When a main menu command is selected in **Categories**, commands on its corresponding pull-down menu will appear in the **Commands** box. Click a command name. If it has a shortcut assigned, the keys will appear in the **Current Keys** box.

To change or create a shortcut, click in the **New Key** box, scroll through the keys list and click a key you want to assign. Check the ALT, CTRL or SHIFT box to select which function key you want to use with the other key. The new shortcut combination is now listed to the right of the **New Key** statement.

Click **Assign** to finish making the shortcut, and the new shortcut appears in the **Current Keys** box. Repeat the steps if you want to create other shortcuts. Click the **Save as Default** checkbox if you want DesignCAD to use these customized keyboard settings the next time you open the program. When the customization task is complete, click **OK**.

To load keyboard shortcut settings from an external file (*.dkf), click the **Load** button. Browse to the location of the shortcut settings file, select it, and click **Open**.

Click the **Save** button to save the keyboard shortcut settings as an external file (*.dkf). This file may then be used for one or more DesignCAD Workspaces.

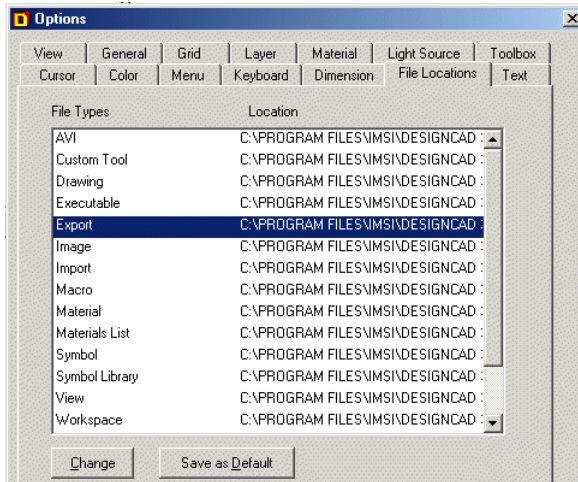
To remove a shortcut key you've made, select the command you want to change, click on the shortcut in the **Current Keys** box, then click **Remove**. You can remove all shortcuts and return to default settings by clicking **Reset All**.

Dimension Options

See Dimension Options.

File Location Options

Modify where DesignCAD stores the various files it uses and produces. This gives you complete control over file placement, no matter how many hard drives or drive partitions you may have on your computer.



Scroll through the list of file paths and highlight the file path you want to modify, then click the **Change** button. The **Path** box appears. Scroll through the list and select the drive that contains the folder or directory you want to use, and select the name of the folder you want to use.

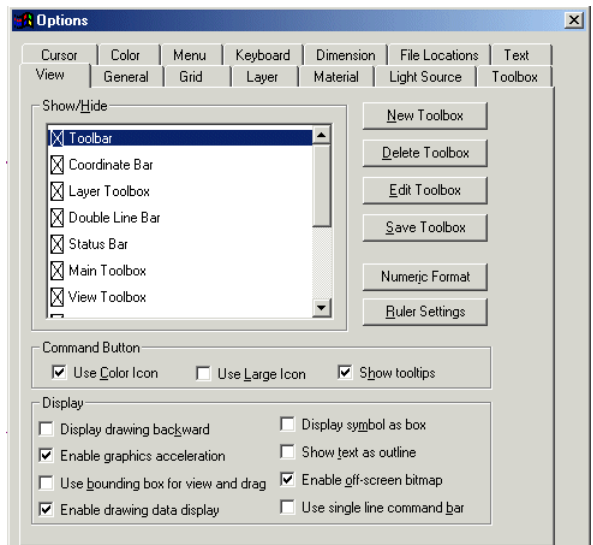
Text Options

See Text (and Attribute) Options.

View Options

Menu: View / Show/Hide

Sets which bars and toolboxes are displayed during the drawing session.



Show/Hide: To activate a bar or toolbox, click the checkbox beside its name.

New Toolbox: Creates a new Custom Toolbox. You may create as many as 40 Custom Toolboxes. Each one may contain command, macro, or BasicCAD icons.

Delete Toolbox: Deletes a Custom Toolbox.

Edit Toolbox: Opens the **Toolbox Options** (see Toolbox Options), with the current toolbox selected. Only Custom Toolboxes can be edited.

Save Toolbox: Saves the selected custom toolbox as an external file.

Numeric Format: Opens the **Numeric Format** window, in which you can control the format of coordinate and angular values displayed in the

Coordinate Bar. Coordinates can be displayed as decimal, fractional, engineering, or architectural. Angles can be displayed as degrees, grads, radians, or degrees, minutes, seconds.

Ruler Settings: Opens the **Ruler Settings** window in which you can set the ruler divisions. See Ruler.

Use Color Icon: Icons are displayed in color. If not checked, icons are displayed in grayscale.

Use Large Icon: Large icons are displayed. If not checked, small icons are displayed.

Show Tooltips: Displays a yellow text box beside the cursor when the cursor is placed over a tool in a toolbox.

Display Drawing Backward: DesignCAD regenerates objects in the drawing in the reverse order of that in which they were created.

Enable Graphics Acceleration: Sets this option on when using the **Shading** and **Hidden Line Removal** commands.

Use Bounding Box for View and Drag: Determines whether a Bounding Box or a bitmap is displayed when you set the view or drag an object by its selection handle.

Enable Drawing Data Display: DesignCAD displays various forms of information when drawing commands are used.

Display Symbol as Box: Represents symbols in drawings with boxes instead of the symbol. This feature greatly reduces redraw time of drawings containing several symbols.

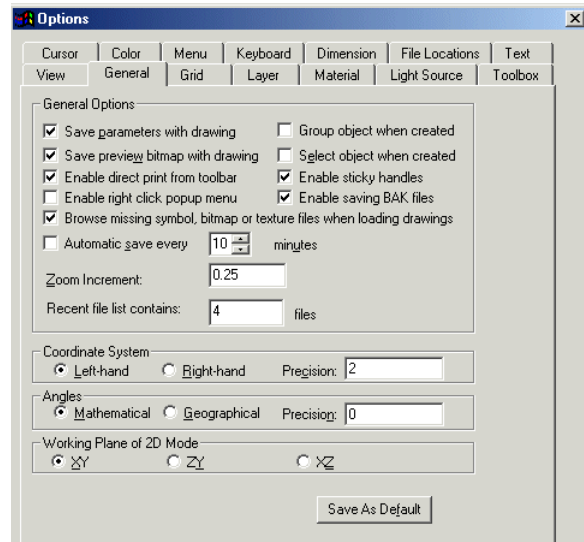
Show Text as Outline: Displays just the outline of filled text entities.

Enable Off-Screen Bitmap: DesignCAD saves a snapshot of the screen every time you perform certain functions. This option should normally be enabled because it makes the operation of DesignCAD faster.

Use Single Line Command Bar: Options for commands appear in the Command Line (the space directly below the Command Menu that is normally occupied by the Toolbar) instead of command windows.

General Options

Controls general as well as coordinate and angle options.



Save Parameters with Drawing: Saves your drawing with the current environment features, such as cursor step size, grid size and display, and other parameters.

Save Preview Bitmap with Drawing: A bitmap for the **Preview** area of the **Open** window is automatically saved with the drawing.

Enable Direct Print from Toolbar: (Also called **Fast Print**.) Affects the functionality of the **Print** icon in the Main Toolbar. The option toggles the icon to either bring up the **Print Command** window or automatically print the current view of the drawing.

Enable Right Click Popup Menu: Affects the right mouse button. When checked, the right mouse button brings up a right-click menu. All of the options in the right-click menu are available in other places in the program. When not checked, clicking the right mouse button executes the **Gravity** command, which moves the cursor to the nearest point in the drawing and sets a point there.

Browse Missing Symbol, Bitmap or Texture Files when Loading Drawings: Displays a browse dialog if a symbol, bitmap or texture file cannot be found, so that you will be able to locate it.

Group Object When Created: Objects that are the result of an array, circular array, etc. are automatically grouped when they are drawn.

Select Object When Created: An object is automatically selected when it is drawn.

Enable Sticky Handles: Determines whether or not DesignCAD remembers selection handles that were previously set for the item using the **Set Handles** command.

Enable Saving BAK Files: When checked, the previous save is renamed to "filename.bak." If you have saved a drawing since this option was enabled, and save changes to a drawing that you decide you don't want, you can just open the "filename.bak" file.

Automatic Save: Saves a drawing automatically at intervals defined in the **Minutes** box. An automatic save stores the open drawing with a name "filename(AutoSave).dcd." The original file ("filename.dcd") is not modified until the you perform either a **Save** or **Save As** command. These commands update the "filename.dcd." If the file is saved and the program terminates normally, "filename.dcd" is updated and "filename(AutoSave).dcd" is removed. If the program crashes, the "filename(AutoSave).dcd" file won't be removed, so you can open "filename(AutoSave).dcd" to recover the drawing as it was up to the last Automatic Save.

If you make a mistake and your drawing is "autosaved," you can close the program without saving the drawing file. The "filename(AutoSave).dcd" file is removed and the "filename.dcd" is unchanged from the last **Save** or **Save As** command.

Zoom Increment: The zoom factor used for the Zoom commands. The default value is .25 (or 25 percent).

Recent File List Contains: Determines how many recent files will be listed in the **File** menu.

Coordinate System: Left-hand makes the positive direction of the Z axis extend away from you. **Right-hand** makes the positive direction of the Z axis move toward you.

Precision: The number of digits that you want displayed after the decimal point in coordinate measurement.

Angles: Mathematical means 0 degrees is the positive X axis (three o'clock) and the degrees progress counterclockwise. **Geographical** means 0 degrees is on the positive Y axis (twelve o'clock) and the degrees progress clockwise.

Precision: The number of digits that you want displayed to the right of the decimal in angular measurement.

Working Plane of 2-D Mode: Click to select the workplane DesignCAD will display when in 2D Mode.

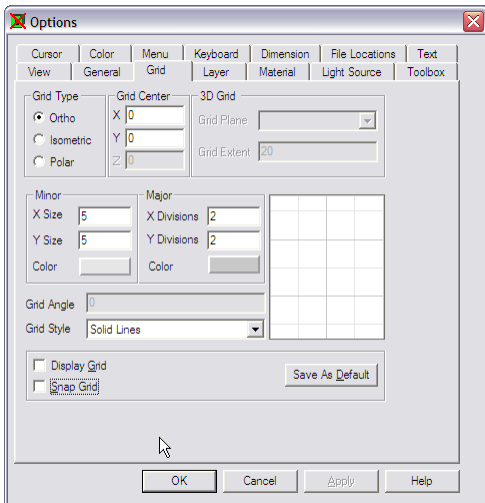
Save As Default: Saves the changes to the next session

Grid Options

Menu: Options / Grid Settings

Shortcut Key: Ctrl+G

Set Snap Grid and Display grid preferences.



Snap Grid: Snaps the cursor to the nearest point on an invisible grid each time you set a point in the drawing.

Display Grid: Causes the grid to be visible.

Grid Type: These grid types are based, by default, on a horizontal baseline.

- **Ortho:** Orthogonal grid.
- **Isometric:** Aligned along lines measured 30° and 150° from the horizontal axis. This type of grid is used in isometric drawing, which uses perspective drawing techniques to represent 3D objects.
- **Polar:** Points in the polar grid radiate from the origin, and are aligned by their radial distance from the center and their angular distance from the horizontal axis.

Minor: Sets the distance between all grid lines. Select X and Y size, or angular and radial values for a polar grid.

- **Color:** Set the color used for the minor grid lines.

Major: Controls for the intermittent (lighter by default) grid lines.

- **Divisions:** Establishes the number of minor grid lines by setting the number of divisions between each major grid point.
- **Color:** Set the color used for the major grid lines.

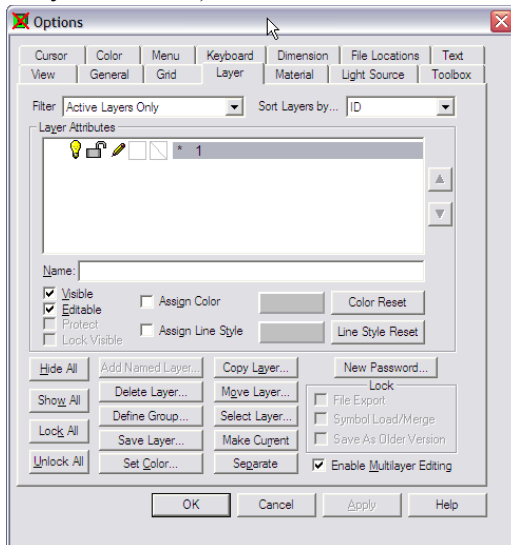
Grid Center: Enables you to use a "local" grid. By default, the grid center is located at the origin of the coordinate system, at (0,0) or (0,0,0). You can reset the center point of the grid

Grid Style: Select Dots, Crosses, or Solid Lines.

Grid Angle: Sets the projection angle when the Isometric Grid type is selected.

Layer Options

Enable or disable layers, name layers, select layers, and perform other layer functions. (See also Layer Toolbox.)

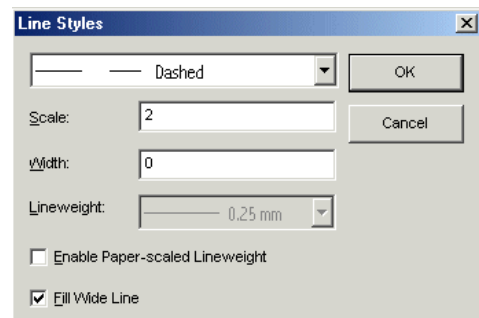


Layer Attributes

- **Name:** To assign names to the different layers in the drawing, highlight the layer number at the left and enter its new name in the **Name** field. To change the layer name using the keyboard, press the Tab key until the focus is on a layer number. Use the up and down arrows keys to select the layer number for which you want to change the name. Press the Spacebar. Enter the new layer name and press Enter. Press Esc to cancel.
- **Visible (light bulb icon):** You can hide layers by making them invisible. This is convenient when you need to reduce the "clutter" in a large drawing. To change the visibility of a layer using the keyboard, press

the Tab key until the focus is on a layer number. Use the up and down arrows on the keyboard to select the layer number for which you want to change the visibility status. Press Alt+V to change the visibility status of the layer.

- **Editable (lock icon):** Determines whether or not the objects in the highlighted layer may be edited. If a layer is not editable, it means that even though you can see the objects in the layer, you cannot modify them. To change the editability of a layer using the keyboard, press the Tab key until the focus is on a layer number. Use the up and down arrows on the keyboard to select the layer number for which you want to change the editability status. Press Alt+E to change the editability status of the layer.
- **Assign Color:** Sets the color of all entities to be drawn on that layer.
- **Assign Line Style:** Sets the line style of all entities to be drawn on that layer.



You can assign a line weight for printing on the selected layer by enabling the **Enable Paper-scaled Lineweight** box.

- **Color Reset:** Restores the assigned color for all layers to the DesignCAD default.

- **Line Style Reset:** Restores the assigned line style for all layers to the DesignCAD default.

Hide All: Hides all layers in a drawing except the current layer.

Show All: Makes all layers in a drawing visible.

Lock All: Makes all layers in drawing uneditable, except the current layer.

Unlock All: Makes it possible to edit all layers in a drawing.

Set Color: Assigns a color for all objects in a layer. All existing objects in the layer will be changed to the chosen color after the command is completed.

Delete Layer: Deletes all entities in the highlighted layer.

Select Layer: This selects all objects in the highlighted layer.

Separate: Sorts objects into layers by color. All objects of the same color will be put into the same layer, regardless of their previous layer.

Move Layer: Moves contents of one layer to another.

Copy Layer: Copies the contents of one layer to another.

Save Layer: Saves the contents of an individual layer as a separate file.

Make Current: Sets the selected layer as the active layer for the drawing.

Enable Multilayer Editing: Allows you to manipulate all visible objects in editable layers. If this option is disabled, you can only manipulate objects in the active layer; all layers except the current layer are uneditable.

Save As Default: Saves the changes to the next DesignCAD session.

New Password: Lets you set up a password for a drawing. Once a password has been established, you can lock the editability and visibility of specific layers of the drawing. You can also prevent the file from being exported as another file type, loaded as a symbol, or saved as an older DesignCAD file.

Click the **New Password** button. In the window, enter the new password and confirm it. Passwords can be up to 50 characters in length and are case sensitive. Click the OK button.

WARNING: Do not lose your password! There is no way of recovering a password-locked drawing without the password.

Once the password has been established, the **Protect Selected Layer(s)**, **Free Selected Layer(s)**, **Lock Show/Hide Status**, and **Unlock Show/Hide Status** buttons may be used to protect selected layers in the list on the left side of the window. To select multiple layers, click on a layer number; then press and hold the Ctrl key on the keyboard while clicking on additional layer numbers. (See Layer Options.)

File Export Lock: If enabled, DesignCAD will not export a copy of the drawing unless the password has been given in the Password Protection Manager.

Symbol Load/Merge Lock: If enabled, DesignCAD will not load a copy of the drawing into another drawing as a symbol. To disable the lock, open the original drawing, run the **Password Protection Manager** command, give the password, disable this option, click the OK button, and save the drawing.

Save as Older Version Lock: If enabled, DesignCAD will not save a copy of the drawing as an older DesignCAD drawing version unless the password has been given in the Password Protection Manager.

To disable password protection, click the **New Password** button. Leave the password text boxes blank and click the OK button. A window appears to inform you that all password protection settings will be removed and ask if you are sure that you want to remove password protection. Click the **YES** button.

Filter: This allows you to control which layers are shown: There are five options: *Active Layers Only*, *Named Layers Only*, *Active and Named Layers Only*, *All Layers*, and *Layer Groups Only*.

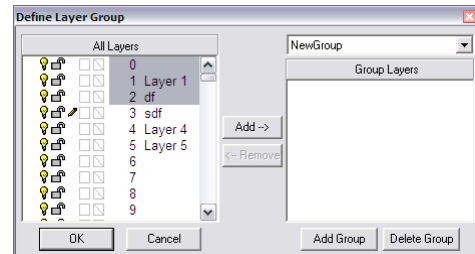
Add Named Layer: Will create an automatically named layer. this option is only available if the *Named Layers Only* option, or the *Active and Named Layers Only* option is selected as the Filter.

Sort Layers By...: This option allows you to set the order in which layers are shown, there are four options: *Name*, *ID*, *Status*, and *Content*.

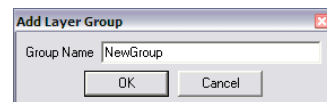
Name: Allows you to change the name of the selected layer if it is already named. you cannot use this field to name an unnamed layer.

To Name an unnamed layer, double click on it in the layer list. a text field will appear. Type in the layer name then press the 'Enter' key.

Define Group: This tool opens the Define Layer Group dialog which allows you to create and manage groups of layers.



- To create a group click the Add Group button. When prompted, type in a name for the group and click OK. Then Select the layers you wish to add to the group and the click ADD button.



- To delete a group, select the group in the drop-down list, click the Delete Group button, and click OK when prompted for verification.
- To remove layers from a group, select the group in the drop-down list, select the layers in the in the Group Layers list, and click the Remove button.

To set any changes you have made in the Define Layer Group dialog click the OK button. To cancel out any change you have made click the Cancel button.

Layer Toolbox

This toolbox contains options for manipulating all the current layers in the drawing. (For details on displaying toolboxes, see View Options.)



Layer List Options: Contains various ways to order the list of layers.

Same Layer As: Click this icon and select an entity on the layer you want to use.

Apply Layer to Selection: Places all selected objects on the active layer.

Enable / Disable Multi-Layer Editing: Determine whether entities on other layers can be edited.

Material Options

See Material Options.

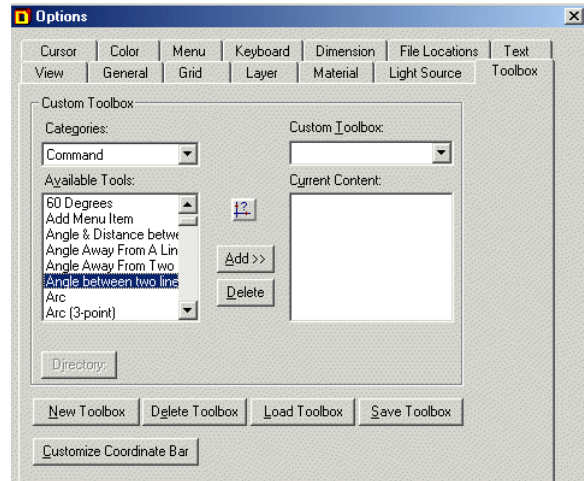
Light Source Options

See Light Source Options.

Toolbox Options

Menu: Tools / Customize / Toolbar

Configure toolbox settings.



Categories: You can choose to insert commands, macros, or BasicCAD programs into the Custom Toolbox.

Available Tools: Choose which tools you want to add to the Custom Toolbox from the list.

Current Content: This is a list of tools which are currently in the **Custom Toolbox**.

Icon: This button displays the icon for the highlighted tool.

Add: Inserts a selected tool into the **Custom Toolbox**.

NOTE: You can also insert a command icon into a **Custom Toolbox** by holding down **Ctrl** while clicking and dragging the icon from another tool box.

Delete: Removes a selected tool from the **Custom Toolbox**.

Directory: Allows you to choose the directory for macros and BasicCAD files.

New Toolbox: Prompts you for a Custom Toolbox name and creates a new toolbox to which you may add commands.

Delete Toolbox: Deletes the currently selected Custom Toolbox.

Load Toolbox: This button loads the .dct file for a Custom Toolbox.

Save Toolbox: Saves the toolbox settings as an external file. This file may then be used for one or more DesignCAD Workspaces.

Customize Coordinate Bar: Sets what values are displayed in the Coordinate bar.

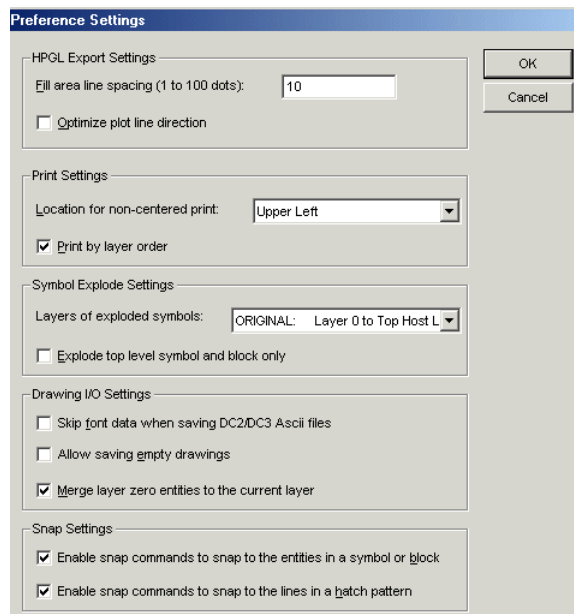


NOTE: The **Angle** field is only available for 2D Mode.

Preferences

Menu: Options / Preferences

Provides the advanced user some control over some of the more complex areas of DesignCAD.



HPGL Export Settings:

- **Fill Area Spacing (10 to 100 dots):** Specifies the distance between adjacent fill lines in plotter increments (usually around 1/1000 of an inch).
- **Optimize plot line direction:** Specifies whether DesignCAD will attempt to rearrange the point order in adjacent plotted lines to minimize pen movement.

Print Settings:

- **Location for non-centered print:** Choose which corner of the page a non-centered print will shift towards.

- **Print by Layer Order:** Specifies whether entities are printed in the order in which they were drawn or in the order of the layers.

Symbol Explode Settings: (see example at the end of this section)

- **Layers of exploded symbols:** Provides fine-level control over the layer behavior for exploded symbols and blocks (whether they were loaded with the **Load and Explode** option checked, or were exploded after being loaded into the drawing).

Original: Default behavior. A symbol's entities get exploded onto their original layers. Entities that were defined on layer 0 of the symbol (or on layer 0 of any nested symbols within the exploded symbol) move to the layer on which the outermost symbol was placed in the current drawing.

Flatten: All of the symbol's entities, including those in nested symbols, get exploded to the layer on which the symbol was placed in the current drawing.

Original2: Like Original behavior, except for entities that were on layer 0. Any entities that were on layer 0 of a nested symbol move to the layer on which the included symbol was loaded in it's host symbol or drawing.

Flatten2: All of the symbol's entities get exploded to the layer on which the symbol was placed, except for entities that were on layer 0. Any entities that were on layer 0 of a nested symbol move to the layer on which the included symbol was loaded in it's host symbol or drawing.

- **Explode top level symbol and block only:** Controls whether a symbol or block is exploded recursively or if only the outermost symbol or block is exploded. When this option is checked, only the selected block or symbol is exploded; any blocks or symbols that were entities within the exploded block or symbol remain intact. If the option is not checked, then exploding a block or symbol explodes All blocks and symbols contained within the outer block or symbol, so that no blocks or symbols remain.

Drawing I/O Settings:

- **Skip font data when saving ASCII DC2 or DC3 files:** If checked, removes the font name from the text strings in DC2/DC3 files. This provides better compatibility with the "classic" DOS versions of DesignCAD.
- **Allow saving empty drawings:** If checked, DesignCAD will allow the user to save empty drawings. This is useful for saving drawings with preset options such as grid size, default units of measurement, layer names, and other information.
- **Merge Layer 0 entities to the current layer:** Affects the way drawings are merged into the current drawing, and the way Paste handles entities on Layer 0. If this option is checked, then Pasting Layer 0 entities from the clipboard puts them on the currently active layer; otherwise they will be pasted onto Layer 0. When loading symbols with "Load as Original Entities" checked, this option also controls the placement of entities that were on Layer 0 in the file being merged.

Snap Settings:

- **Enable snap commands to snap to entities in a symbol or block:** Allows you to snap to entities inside a block or symbol entity. This can be useful sometimes, but it may cause a degradation in the performance of Running Snaps.
- **Enable snap commands to snap to the lines in a hatch pattern:** Allows you to snap to the internal points of a hatch entity. This option may cause a degradation in speed if Running Snaps are enabled.

Example: Symbol Explode

Symbol A contains a red line drawn in layer 0, and a black box on layer 1.

Symbol B contains Symbol A on layer 7, a blue circle on layer 9, and a green star on Layer 0.

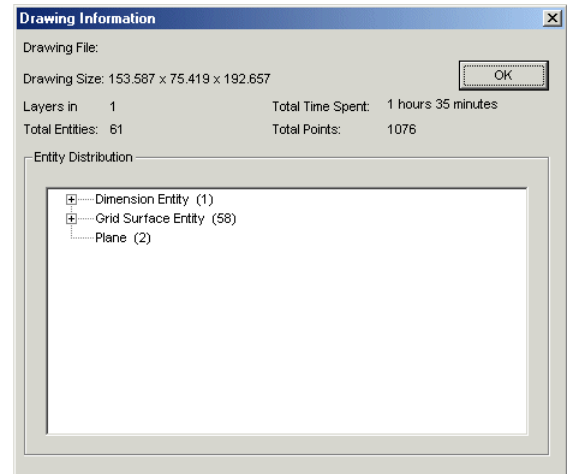
Now you include Symbol B on layer 12 of your drawing, and explode it.

- **Original:** The black box stays on layer 1, the blue circle stays on layer 9, and a red line and green star move to layer 12.
- **Flatten:** All four entities move to layer 12.
- **Original2:** The black box stays on layer 1, the red line moves to layer 7, the green star moves to layer 12, and the blue circle stays on layer 9.
- **Flatten2:** The red line moves to layer 7, the black box moves to layer 12, the green star moves to layer 12, and the blue circle moves to layer 12.

Drawing Info

Menu: Help / Drawing Info

Displays the number of entities and points in a drawing, along with various other information.



Setting the View

This section covers ways to control what appears on the drawing screen.

NOTE: For 3D view settings, see View Toolbox.

Fit to Window

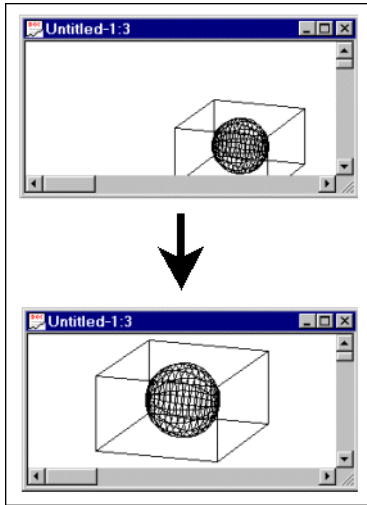
Menu: View / Fit to Window

Shortcut Key: Ctrl+W

Toolbox Icon:



Zooms the active view window so that the entire drawing is centered on the screen with all objects visible.

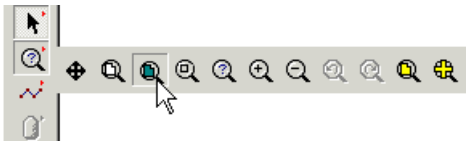


Fit to All Windows

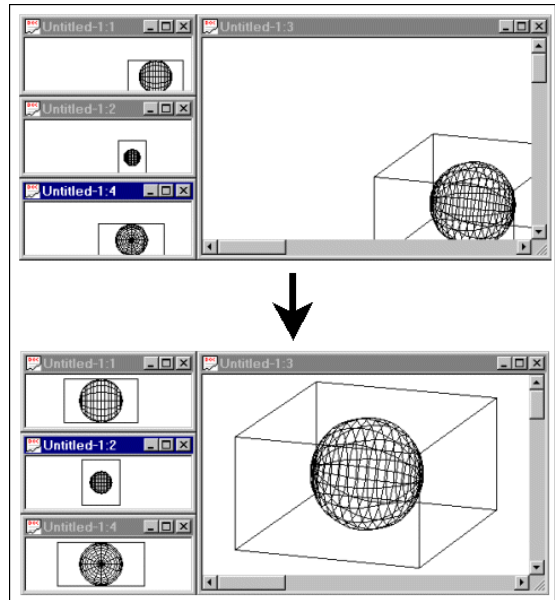
Menu: View / Fit to All Windows

Shortcut Key: Ctrl+Shift+W

Toolbox Icon:



Zooms all the view windows so that the entire drawing is centered on the screen with all drawn objects visible.



Pan

Menu: View / Pan

Shortcut Key: Ctrl+Shift+P

Toolbox Icon:



Point 1: Specific point to be moved

Point 2: New screen location for Point 1

Slides the drawing around on the screen.

Using the Command

Choose the **Pan** command. The cursor turns into a four-headed arrow. Drag the mouse across the screen until the drawing is in the position you want. Then release the mouse button. You can also press and hold Ctrl+Alt instead of holding down the mouse button.

Another alternative is that you can set a point for the "source" and "destination" of the drawing. The first point is the original location and the second point is the new location for that part of the drawing.

Zooming

This section covers the various ways you can increase or decrease the size of objects as they appear on the screen.

Zoom

Menu: View / Zoom

Toolbox Icon:

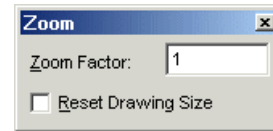


Point 1: Center of zoom

Makes the drawing appear larger or smaller on the screen. It does not affect the actual size or scale of the objects in the drawing unless you choose the **Reset Drawing Size** option.

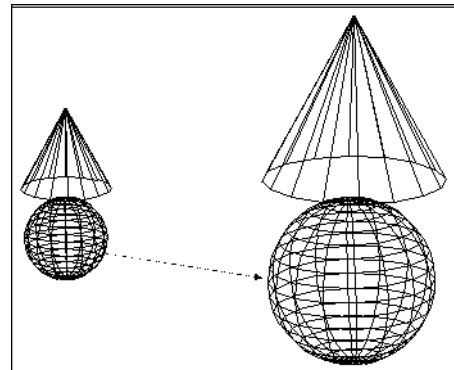
Using the Command

Choose the **Zoom** command. In the window set the **Zoom Factor**.



Reset Drawing Size: DesignCAD resizes the objects in the drawing according to the zoom factor. Otherwise, only the apparent size of the objects are changed.

The zoom factor is relative to the current size of the drawing. If you zoom with a factor of two, the drawing appears twice as large. If you zoom with a factor of 0.25, the drawing is displayed at one-fourth its current size.

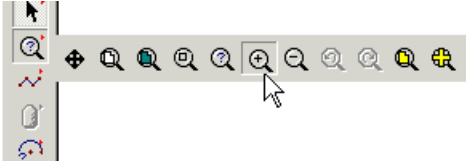


Zoom In

Menu: View / Zoom In

Shortcut Key: +

Toolbox Icon:



Point 1: Center of zoom

A quick way to zoom into your drawing (make entities appear larger).

Using the Command

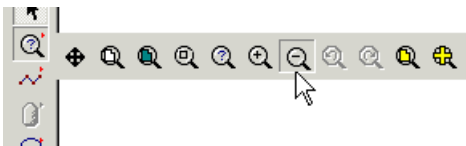
Select the **Zoom In** command. Position the cursor at the zoom center and click the left mouse button.

Zoom Out

Menu: View / Zoom Out

Shortcut Key: -

Toolbox Icon:



Point 1: Center of zoom

A quick way to zoom out, or reduce the size of the drawing as it appears on the screen.

Using the Command

Select the **Zoom Out** command and position the cursor at the zoom center. Then click the left mouse button.

Zoom Window

Menu: View / Zoom Window

Shortcut Key: Z

Toolbox Icon:



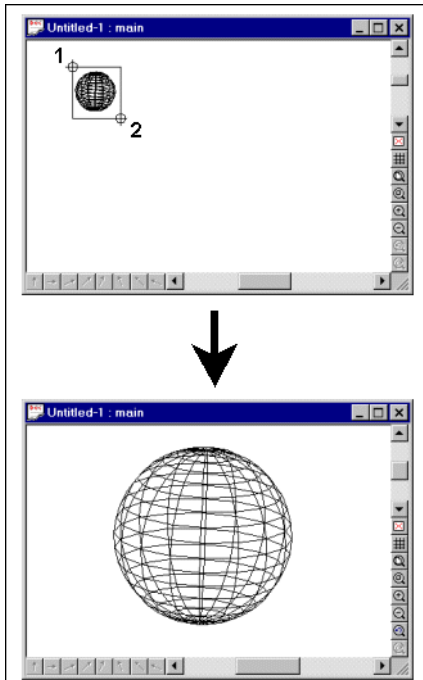
Point 1: One corner of the area to be magnified

Point 2: Opposite corner of the area to be magnified

Zooms in on your drawing, filling the screen with a specified area of a drawing.

Using the Command

Select the **Zoom Window** command. Drag a rectangle around the area you want to zoom into. That rectangle is then enlarged to fill the screen.



Zoom to Selection

Menu: View / Zoom to Selection

Fits the currently selected entity or entities in a drawing to all view windows. The drawing is displayed at as large a zoom factor as possible while maintaining the visibility of all of the selected entities. If only one entity is selected, it will be centered in all the view windows. Objects or portions of objects that are not selected may be visible, depending on their proximity to the selected entities.

Using the Command

Select one or more entities in the drawing. Choose the **Zoom to Selection** command. The drawing is zoomed so that the selected entity is displayed as large as possible while keeping the entire selected entity visible and centered in all the views.

Zoom Previous

Menu: View / Zoom Previous

Shortcut Key: Ctrl+M

Toolbox Icon:



Reverts back to the zoom setting you used last. If you select this command repeatedly, it goes backwards through the entire sequence of zoom operations since the last time you saved the drawing.

Zoom Redo

Menu: View / Zoom Redo

Shortcut Key: Ctrl+Shift+M

Toolbox Icon:



Reverts to the last zoom setting before the **Zoom Previous** command. For example, if you change your mind about a zoom setting after using **Zoom Previous**, you can select **Zoom Redo** to cancel that zoom action.

Original Size

Menu: View / Original Size

Toolbox Icon:



Restores a zoomed drawing back to its original size.

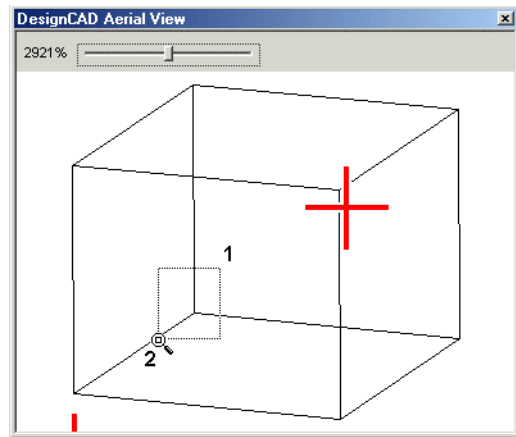
Aerial View

Menu: View / Aerial View

Makes it possible to see the part of the drawing you're working on relative to the entire drawing, even when you've zoomed in closely to a specific portion of the drawing.

Using the Command

Select the **Aerial View** command and the **Aerial View** window appears.



This command provides a bird's-eye view of the drawing, even when the view window is zoomed in on the drawing. The **Aerial View** window can be sized and placed anywhere on the screen.

To change the focus of the active view window, just click on the red crosshairs and drag them to the new focus location. To zoom in on a different area, set two points to define a zoom window.

Refreshing and Regenerating

If your screen gets cluttered with old entities or stray lines, or you have made changes that are not reflected, you may need to refresh or regenerate the screen.

Refresh

Toolbox Icon:



Refreshes the active drawing window. It cleans up images, such as cursor cross hairs, that have been left on the screen. Choose this icon when you do not want to redraw the screen but only refresh it.

NOTE: *When working with large, detailed drawings, using **Regenerate** takes more time than refreshing it. If you just want to clean up the effects of previous cursor locations or selection handles, use the **Refresh** command.*

Regenerate

Menu: View / Regenerate

Shortcut Key: Ctrl+R

Toolbox Icon:



Redraws the entire drawing in wireframe form. This erases any shaded or hidden-line areas of the screen. This command affects only the active view window.

Regenerate All

Menu: View / Regenerate All

Shortcut Key: Ctrl+Shift+R

Redraws the entire drawing in wireframe form in every open view window. It erases any shaded or hidden line areas of the screen.

Regenerate Double Line Entities

Menu: View / Regenerate Double Line Entities

Toolbox Icon:



Redraws all of the double line entities in the drawing. Sometimes deleting a line or entity that intersected a double entity will give the appearance that part of the double entity was erased. This command refreshes the double entities, removing any apparent gaps.

See Double Line Mode.

Window Options

In DesignCAD you can work with multiple windows. This can mean several panes of the same file or panes from different files. This section covers how you can create and display these windows.

New Window

Menu: Window / New Window

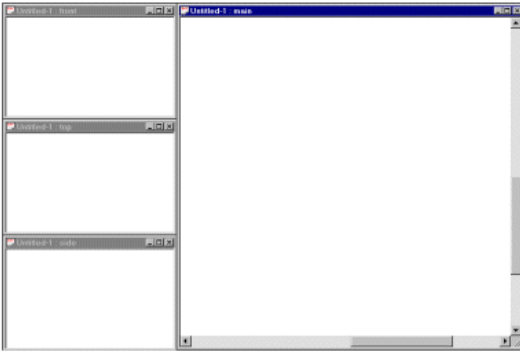
Opens a new view of a drawing. DesignCAD allows several different windows, or views, to be opened simultaneously. These can be zoomed or panned independently of each other.

DesignCAD Tile

Menu: Window / DesignCAD Tile

Arranges the active drawing windows according to the **DesignCAD Tile** Settings. The default setting places the Perspective view in the large

window on the right side of the screen, and the Front, Top, and Side views stacked vertically on the left. This is a convenient window arrangement for working in 3D space, when using 3D Selection Mode, and when working in 2D or 3D Mode with a large object which contains small items that need to be zoomed for detailed drawing.



DesignCAD Tile Settings

This command contains two sub-commands:

Restore DesignCAD Tile

Menu: Window / DesignCAD Tile Settings / Restore DesignCAD Tile

Changes the DesignCAD Tile setting back to its default configuration: the Perspective view in the large window on the right side of the screen, and the Front, Top, and Side views stacked vertically on the left.

Set As DesignCAD Tile

Menu: Window / DesignCAD Tile Settings / Set As DesignCAD Tile

Changes the DesignCAD Tile setting to the current view configuration. After rearranging the view windows, choose this command to set the configuration.

NOTE: The DesignCAD Tile settings for 2D Mode and 3D Mode are stored as different settings. Therefore, changes made to the DesignCAD Tile setting while in 2D Mode will not affect the setting for 3D Mode, and vice versa.

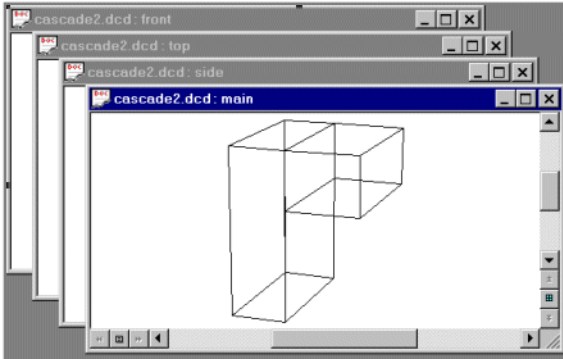
Example: Reverse the default DesignCAD Tile setting

To move a view window, click in the title bar for the view to be moved. While holding the mouse button down, drag the window to its new position. To place the window, release the mouse button. Move the Front, Top, and Side views from the left side of the screen to the right. Move the Perspective view from the right side of the screen to the left. Select the **Set as DesignCAD Tile** command. Until the DesignCAD Tile setting is changed again using the **Set as DesignCAD Tile** command or the **Restore DesignCAD Tile** command, this is the view configuration that will be used every time the DesignCAD Tile command is selected.

Cascade

Menu: Window / Cascade

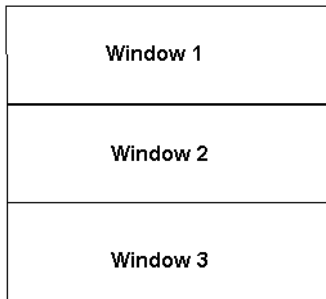
Arranges open drawing windows so that they overlap on the screen.



Tile Horizontal

Menu: Window / Tile Horizontal

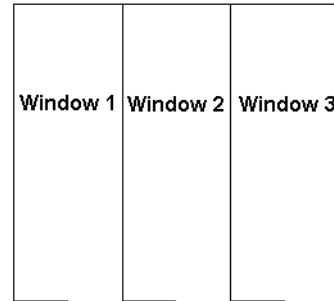
Arranges open windows by arranging them evenly and horizontally across the screen.



Tile Vertical

Menu: Window / Tile Vertical

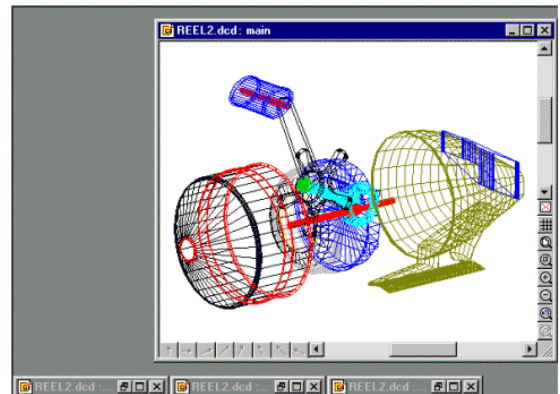
Arranges open windows by arranging them evenly and vertically across the screen.



Arrange Icons

Menu: Window / Arrange Icons

Organizes the view icons at the bottom of the window.



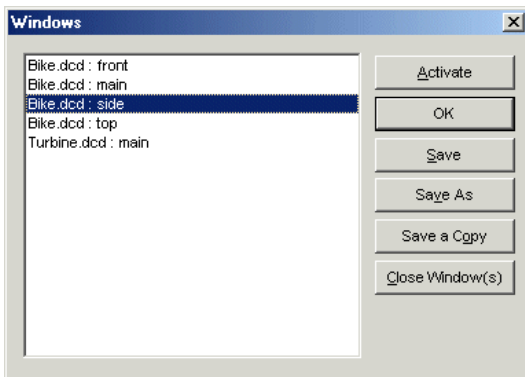
Windows

Menu: Window / Windows

Provides a convenient dialog that shows all open drawings and views, and makes it easy to switch between drawings. You may also close a drawing from this list, or save a copy of one of the open drawings.

Using the Command

Choose the **Windows** command and the following window appears:



Activate: Brings the drawing to the top and makes it active.

OK: Closes the window and returns you to the drawing area.

Save: Saves the drawing.

Save As: Saves the drawing under a new name.

Save a Copy: Saves a copy of the associated drawing under a new name, but you keep working on the current drawing. This makes it easy to save incremental backups.

Close Window(s): Closes the selected view.

Digitizer

This section covers commands you can use when tracing your drawing with a digitizer.

Digitizer Tracing Mode

Menu: Tools / Digitizer / Digitizer Tracing Mode

Lets you zoom the screen image of the drawing you are tracing while retaining the present scale on the digitizer pad.

Using the Command

Make sure there is a check mark next to the **Digitizer Tracing Mode** indicating that it's active. Use the **Zoom** commands to zoom the current screen image of your drawing. The Zoom commands will not change the relationship between the drawing information being entered and the digitizer pad. Zoom commands only change the size or location of the drawing on the screen.

Align Drawing

Menu: Tools / Digitizer / Align Drawing

Point 1: Point on the screen

Point 2: Corresponding point on the digitizer pad

Point 3: Point on the screen

Point 4: Corresponding point on the digitizer pad

Synchronizes the digitizer pad with the display quickly and easily. You should have already set up **Digitizer Tracing Mode** before you align the drawing.

Using the Command

Choose the **Align Drawing** command and set a point of reference on the drawing screen. Set a second point with regard to the digitizer pad that will correspond to the first point set on the drawing screen. Now set a third point as another point of reference on the drawing screen. Finally, set the fourth point with regard to the digitizer pad; this point will correspond to the third point which was set with respect to the drawing screen. This will tell DesignCAD the relationship that the area of the digitizer pad should have with the drawing screen.

NOTE: *DesignCAD will ignore this command if you do not have a digitizer or if it is not recognized by your computer.*

Create Digitizer Menu

Menu: Tools / Digitizer / Create Digitizer Menu

Creates a digitizer menu. The menu created with this command does not appear on the screen. The menu contains several selection boxes. These boxes are spaces where commands are selected.

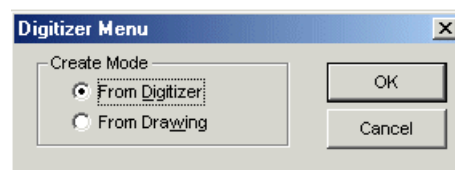
Using the Command

To create a digitizer menu, follow these steps:

1. Draw the menu template. This is the paper template that will be affixed to the digitizer. To start with, draw a border box the size of the menu. The menu can be any size, but the larger the menu template is, the more of the digitizer drawing area the menu will take up.
2. On the template, draw individual "command" boxes for each DesignCAD command you want on the menu. These

command boxes can be any size you want. Draw text or a picture to be associated with each command in the command box.

3. Print the finished template at a scale of 1. Cut it out and affix it to the digitizer, inside the active drawing area of the digitizer.
4. Choose the **Create Digitizer Menu** command. Choose the method of creation in the **Digitizer Menu** window, and click OK.



5. Now set a point in the lower left corner of the digitizer template. Set a point in the upper right corner of the digitizer template.
6. Next, set a point in the lower left corner of the command box. Set a point in the upper right corner of the command box.
7. Enter the command name in the **Command** box. The DesignCAD macro command names can be found in the DesignCAD Macro Command Name section of the MACROCMD.INI file. The command names appear to the right of the equal mark in the MACROCMD.INI file.
8. Set two more points for the next command box, or press the Enter button to end the command.

Load Digitizer Menu

Menu: Tools / Digitizer / Load Digitizer Menu

Loads the "data" portion of a digitizer menu. The "paper" portion of the menu should be attached to the digitizer before the digitizer is loaded. A maximum of 10 digitizer menus can be loaded at once.

Using the Command

First, attach the paper template onto the digitizer. Choose the **Load Digitizer Menu** command. The **Load Digitizer Menu** box appears. Choose the digitizer menu name in the **File Name** box, then click OK or press Enter.

Now set a point in the lower left corner of the digitizer menu. A rubber-band box shows how the digitizer menu will be loaded. Set a point in the upper right corner of the digitizer menu.

Save Digitizer Menu

Menu: Tools / Digitizer / Save Digitizer Menu

Point 1: Point inside the digitizer menu

Saves the current digitizer menu. This command saves the menu in .DGM file format.

Using the Command

After creating the digitizer menu, choose the **Save Digitizer Menu** command and set a point inside the menu area. The **Save Digitizer Menu** window appears. Enter the name of the new digitizer menu in the **File Name** box and click **Save**.

Close Digitizer Menu

Menu: Tools / Digitizer / Close Digitizer Menu

Point 1: Point inside the menu area

Closes the active digitizer menu.

Using the Command

Choose the **Close Digitizer Menu** command. Set a point inside the menu area. The digitizer menu is closed.

Add Menu Item

Menu: Tools / Digitizer / Add Menu Item

Point 1: Point inside the menu area

Adds a command to an existing digitizer menu.

Using the Command

Open the digitizer menu to be changed. Choose the **Add Menu Item** command and set a point in the area the new command is to occupy. The **Digitizer Menu Command** window. Enter the name of the new command in the **Command** box.

Click **Continue** to add another command or **Done** to close the window.

Remove Menu Item

Menu: Tools / Digitizer / Remove Menu Item

Point 1: Point inside the menu area

Removes a command from an existing digitizer menu.

Using the Command

Open the digitizer menu to be changed. Choose the **Remove Menu Item** and set a point in the area occupied by the command you want removed. The command is removed from the digitizer menu.

Macros and Executables

A macro is a saved set of drawing actions that can be used later as a single command.

Macro Record

Menu: Tools / Macro Record

Records a macro that can be used later as a single command.

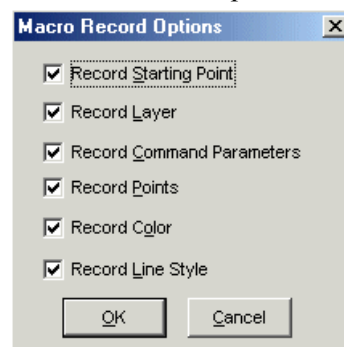
Using the Command

When you use the **Macro Record** command, you are first asked for the macro name. (The **Set Initial Reference Point** option is the same as the **Record Starting Point** option which is described below.) Select a name and a location for the macro in the **Record Macro** box and click **Save**. Then the **Macro Record** toolbox appears on the screen:



The icons represent **Record / Continue**, **Pause**, **Macro Options**, and **Stop**.

The **Macro Options** icon opens a window in which you can set different options for the macro.



Record Starting Point: Determines whether or not a starting point is used for the macro. If this option is enabled, a starting point is required at the beginning of the recording and when the macro is executed. The entities drawn during the recording and execution of the macro will have identical spatial relationships with their starting points.

If this option is disabled, no starting point is required during the recording or subsequent executions of the macro. The entities drawn during the execution of the macro will be located at the same drawing coordinates as when they were recorded.

Record Layer: Checking this box causes the macro to run in the same layer in which it was created.

Record Command Parameters: Checking this box saves option information in the macro. For example, if you create a sphere with this option enabled, the macro will retain information such as number of latitudinal and longitudinal facets, and whether you created it in Vertex, Pole, or

Midpoint format. If you leave this option disabled, then you will be asked for that information each time you run the macro.

Record Points: The points set in specific drawing commands will be retained in the macro.

Record Color: Creates all objects in the same colors in which they were recorded.

Record Line Style: Record the drawing's current line style with the macro.

After you enter the macro name, everything you draw on the screen becomes part of the macro. You can pause recording by clicking on the **Pause** button, and resume by clicking on the **Record** button.

When you are finished recording your macro, click the **Stop** icon or choose **Stop Recording** from the **Tools** menu. The macro can then be run using the **Macro Execute** command.

Example: Draw a box and a sphere of specific sizes in several drawings.

Select the **Macro Record** command. Make sure the **Set Initial Reference Point** option is enabled. Name the macro TEST.D3M and click **Save**. Set a starting point. Draw a box and a sphere on the screen. Click the **Stop** icon. The macro will be saved under the selected file name. Then, select the **Macro Execute** command and choose TEST.D3M from the window. Set a starting point for the macro and DesignCAD will carry out the recorded series of actions.

TIP: You can assign Toolbox buttons to the macros you create. See [Toolbox Options](#).

Stop Recording

Menu: Tools / Stop Recording

Ends the Macro Record command.

Using the Command

After you have finished recording a macro, choose **Stop Recording**. The macro will be saved under the file name entered in the **Macro Record** command.

Continue Recording

Menu: Tools / Continue Recording

Restarts the recording of a macro. After suspending the recording of a macro, use this command to resume recording.

Using the Command

After stopping or pausing a macro recording, choose the **Continue Recording** command. The macro resumes recording.

Pause Recording

Menu: Tools / Pause Recording

Suspends the recording of a macro.

Using the Command

While recording a macro, choose the **Pause Recording** command. The macro stops recording until you choose the **Continue Recording** command.

Record Options

Menu: Tools / Record Options

The **Record Options** command can be activated while the macro is being recorded. This command allows you to save and change options within the macro.

See Macro Record for details of the specific options you can change.

Macro Toolbox

Menu: Tools / Macro Toolbox

This command is a toggle command that either displays or hides the **Macro Toolbox**.



Macro Execute

Menu: Tools / Macro Execute

Shortcut Key: %

Runs a DesignCAD macro. A macro is a saved set of drawing actions that can be used later as a single command.

Using the Command

Choose the **Macro Execute** command, enter the macro name when asked for it, and click **Open**. Set a starting point for the macro if required. The macro begins its execution.

Example: Draw a box and a sphere of specific sizes in

Stop Macro

Menu: Tools / Stop Macro

Shortcut Key: &

Halts the execution of a DesignCAD macro. However, you cannot access the Command Menu or the shortcut key to stop the macro if a drawing or shading or similar command is in operation.

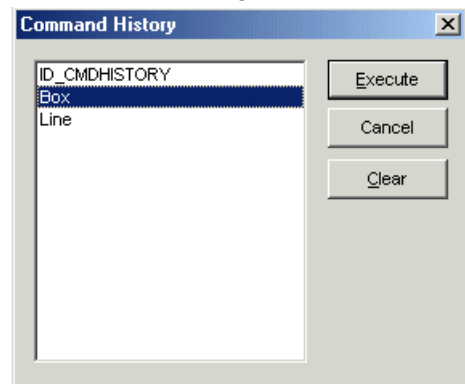
Command History

Menu: Tools / Command History

Lists the commands that have been used in the current drawing session, starting with the most recent. Commands in the list may be executed using the **Command History** window.

Using the Command

Select the **Command History** and the commands are listed in the following window:



Choose a command from the list by clicking on it with the mouse. Click **Execute** to start the command.

To return to the DesignCAD drawing screen without starting a command, click **Cancel**.

To clear the command history, click **Clear**.

Run Executable

Menu: Tools / Run Executable

Lets you run an executable from within DesignCAD.

Using the Command

Choose the **Run Executable** command. From the **Run Add-On** window, select or enter the name of the file and its location. When you have supplied the information, click the **Open** button. The executable initializes.

To stop an executable, select the program's **Exit** command.

Images

This section covers all the ways you can bring images into DesignCAD, and how you can trace images within DesignCAD.

Load Image File

Menu: File / Image / Load Image File

Point 1: Lower left corner for image placement

Point 2: Upper right corner for image placement

Adds a graphic image from another file to your drawing. If you only set a single point, the image will use that point as the lower left corner and be retrieved at full scale. If you set two points, the image will be scaled to fill a rectangle with those two points as opposite corners. The image is never rotated in 3D space; it always appears "flat" relative to the screen, regardless of your viewing angles.

Using the Command

Choose the **Load Image File** command. Choose the type of file to be added to the drawing from the drop-down list in the **Files of Type** box. In the **Look In** box enter the location of the image. In the **File Name** box enter the name of the file to load or select the file from the area below the **Look In** box.

This command is very useful in adding illustrations to your drawings. For example, it's possible to add a scanned photograph to be displayed along with your drawing.

Save Image File

Menu: File / Image / Save Image File

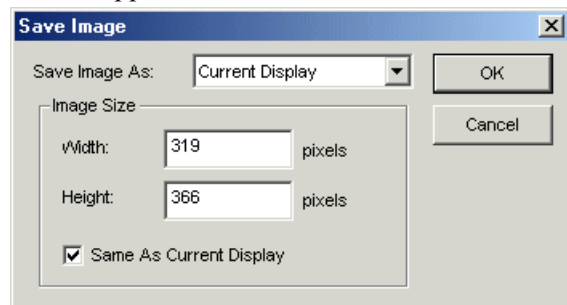
Point 1: First corner of rectangular region to save

Point 2: Opposite corner of rectangular region to save

Saves a screen image (or part of one) as a graphics file. The saved file can be used with other applications such as word processors and desktop publishing systems. This is a convenient way to transfer a shaded image to other applications.

Using the Command

Choose the **Save Image File** command, and an X-shaped cursor appears in the drawing window. To select a portion of the window to save in a common graphic format, set a point in two opposite corners of the region to be saved. To save the entire window as a graphics file, press Enter without setting any points. The **Save Image** window appears.



Save Image As: Lets you save the image in the current display mode, wireframe, quick shading, smooth shading, or hidden line removal.

Image Size: Set the width and height of the image in pixels.

Same as Current Display: Resets the values for the width and height of the image to be saved.

OK: Saves the changes you have made and opens the **Save Image File** window. Enter the name and location for the new graphic file and click OK to save it.

Cancel: Disregards the changes you have made and returns you to your drawing.

Scan Image

Menu: File / Image / Scan Image

Takes an image from a TWAIN compatible scanner that has been specified with the **Select Scanner Source** command. A TWAIN scanner allows you to run the scanner directly from within DesignCAD. The image can later be converted into a vector image with the **Auto Trace Bitmap** command. This allows scanned artwork to be used in a DesignCAD drawing.

Using the Command

Place the image in the scanner. Make sure that the scanner has been specified with the **Select Scanner Source** command. Select the **Scan Image** command.

*NOTE: Scanners vary in their capabilities and features, and this is reflected in the window that appears when the **Scan Image** command is used. Depending on the scanner's features, it may be possible to adjust the image's orientation, the direction of the scan, the resolution, etc. Refer to the scanner's software documentation or on-line Help for more information.*

Use the scanner's own software to scan the image. The image is scanned and imported into DesignCAD. Use the **Auto Trace Bitmap** command to trace the bitmap as a series of vectors.

If you are using a scanner that is not TWAIN compatible, you can still use scanned images in DesignCAD, but it takes more steps to process the image. Switch to the scanner software or driver to execute the scan, then return to DesignCAD. Once the image has been scanned as a bitmap, use the **Load Image File** command. After loading the image, use the **Auto Trace Bitmap** command to trace the bitmap as a series of vectors.

Select Scanner Source

Menu: File / Image / Select Scanner Source

Selects the scanner for use with the **Scan Image** Command.

Using the Command

Choose the **Select Scanner Source** command. Select the correct scanner source for use with the **Scan Image** command from the window, and click the **Select** button.

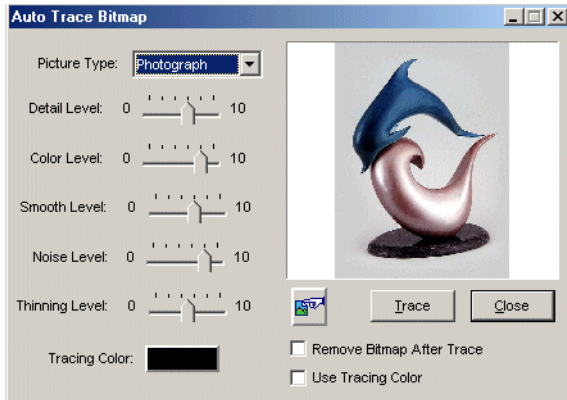
Auto Trace Bitmap

Menu: Tools / Auto Trace Bitmap

Creates vector outlines of specified colors in bitmaps. This allows the image to be manipulated in DesignCAD. With a color or black and white scanner, a drawing or other art can be scanned and saved as a bitmap, then each color can be traced to produce a DesignCAD drawing.

Using the Command

Bring a bitmap into DesignCAD with the **Load Image File** command. When the bitmap is displayed, select it and choose the **Auto Trace Bitmap** command. The following window appears:



The color to be traced is displayed on the **Tracing Color** button. To change color, click this button. The cursor turns into an eyedropper. Either in the thumbnail image at upper-right of the window or in the main drawing area, position the eyedropper on the item you want to trace. Click the mouse button. The sample is taken from the color at the eyedropper tip. The **Tracing Color** button will change to the color of the item you selected.



If there are multiple bitmaps in the drawing, you can click this button, then click on the bitmap you wish to trace.

The results of an Auto Trace are affected by the **Picture Type**, **Detail Level**, **Color Level**, **Smooth Level**, **Noise Level**, and **Thinning Level** options. Experiment with these options for optimal results. To change these values, click and hold the indicator using the mouse, then move the

slider closer left and right. The Auto Trace results will automatically update when one of these options is changed.

Once you have selected the bitmap to be traced and set all of the options to the desired levels, click the **Trace** button. DesignCAD will trace the bitmap. If you like the results, click the **Close** button to close the Auto Trace Bitmap window.

Once you are satisfied with the result, select the bitmap and delete it. The new vector image of the bitmap remains in the drawing.

The **Remove Bitmap after Trace** option can be checked so DesignCAD will automatically delete the bitmap from the drawing once it has been traced.

*NOTE: This command can only trace and convert one color at a time. If the bitmap image has entities in more than one color, you must execute the **Auto Trace Bitmap** command for each colored item you want traced.*

Printing and Paper Space

This section covers how to prepare your drawing for printing, and all the printing options.

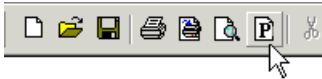
Paper Space

Paper Space is essentially the printing mode. In paper space you can define areas and views you want to print.

Paper Space Mode

Menu: File / Paper Space View Frame Setup / Paper Space Mode

Toolbox Icon:



Makes it possible to print multiple views of the same drawing in a single print job. Options such as perspective, shading, light source, focal point, view settings, and zoom can be set for each separate view created in the Paper Space.

Using the Command

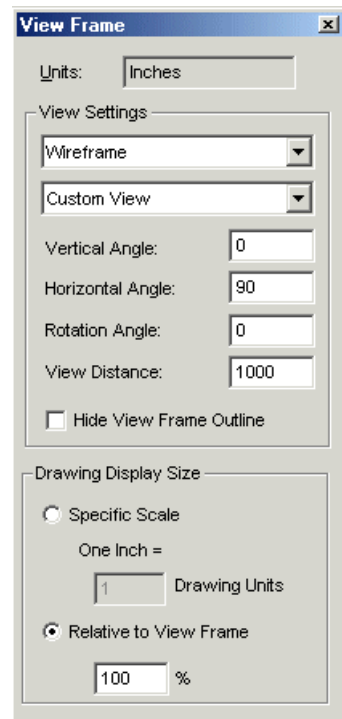
Choose the **Paper Space Mode** command, which puts you into Paper Space Mode with the most recently-define layout. However, if no layouts have been created yet, the command first displays the **Paperspace Configuration** window (see Paper Space Configuration) It also automatically creates a viewframe to fit the page, with the viewframe scale set to 100%. This view frame can be scaled and more customized view frames can be added using the **Create View Frame** command, available on the **Paper Space** window or the **View Frame** menu which appears once Paper Space Mode is enabled.

NOTE: To exit Paper Space mode, select **File / Paper Space** or click the **Paper Space** icon to deselect it.

There are several options available in the window which can be set for each view frame.



Create View Frame: Create a printable view window for which you set the placement, size, and view options.



Select the View Settings and Drawing Display Size for the View Frame. Now move the cursor to the location in the Paper Space at which you want to place the first corner of the View Frame. Press and hold the mouse button. Drag the mouse until the rubber-band preview is the size you want the View Frame. Release the mouse button. The rubber band preview is replaced with a View Frame border with your drawing inside.

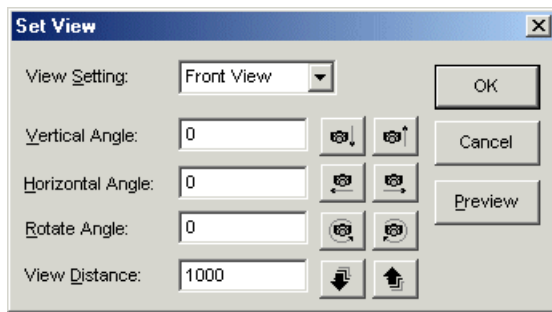
The View Frames can also be drawn using the **Point Polar** and **Point Relative** commands.



Pick an Entity as View Frame: Uses simple 2D entities, that have been drawn while in Paper Space Mode, as view frames. Just draw the entity, click on this option and then click on the entity you just drew. That entity is used as a view frame for the drawing.



Change View Setting: Provides several different view options for an individual View Frame can be set using this command.



- **View Setting:** Determines which view of the drawing is shown in the View Frame. The views listed are the same as those in the **Projection** list-box in the **View** toolbox.
- **Angles and View Distance:** Set the vertical or horizontal angle of the drawing, rotate the drawing angle, or set the view distance.
- **Preview:** Previews the view setting changes made to the current View Frame of the drawing.



Draw Outline Box: Displays or hides an outline border around the selected View Frame. This command works as a toggle, showing or hiding the border.



Fit to View Frame: Centers the drawing in the View Frame, with all objects visible.



Center on View Frame: To center the drawing in the View Frame without changing the zoom factor, use this command instead of **Fit to View Frame**.



Fit Drawing to Paper: Resizes the view frames so that all view frames will fit within the paper space at the largest possible size.



Set Focal Point: Drags the drawing in the View Frame by its center point.



Pan Drawing: Slides the drawing across the View Frame using a reference point and the new location for that point.



Light Source & Layer Settings: Sets these options for the current View Frame. See Layer Options and Light Source Options.



Section Zoom: This command uses a zoom window to zoom in on a specific area of the drawing.



Zoom In: Zooms in at increments preset by the program.



Zoom Out: Zooms out at increments preset by the program.



Change Zoom Factor: Changes the zoom factor of the currently selected view frame interactively. Click on this option and move the cursor into the Paper Space area. Press and hold the mouse button. Move the cursor up to increase the zoom factor or down to decrease the zoom factor. When the desired appearance is achieved, release the mouse button.



Save Current Layout as Template: Saves the current Paper Space Template, including all View Frames and the options set for each.



Info Box: When a view frame is selected, clicking this button opens the **Info Box** which contains information about the view frame such as view settings, the view frame's dimensions, and the drawing display size. Most of this information can be changed in the **Info Box**.



Preview: Shows a preview of how the printed Paper Space will look including shading and hidden line removal.

*NOTE: To see the page layout including margins, use the **Print Preview** command on the Toolbar.*

Display Method: Designates a display method for the current view frame. The changes to the Display Method will only be seen with the Preview command in the Paper Space window.

Paper Space Layout: Select a different Paper Space template in which to display the drawing.



Edit Layout: Opens the **Paper Space Template** window so you can change the paper size and margins for the current template.



Add a New Layout: Opens the **Paper Space Template** window so that a template can be added to the Paper Space Layout list box.



Delete Current Layout: Deletes the currently selected template in the Paper Space Layout list box.

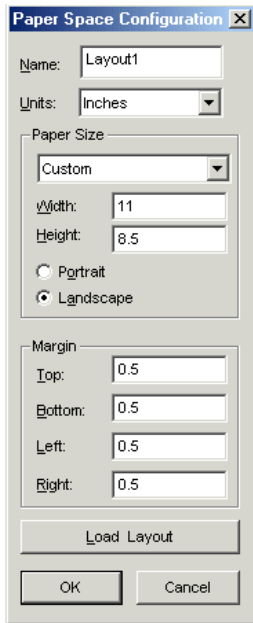
Paper Space Configuration

Menu: File / Paper Space View Frame Setup / Paper Space Configuration

Sets up the area to be used for the Paper Space Mode. This command should be used to ensure that the Paper Space that is about to be created will print correctly when the time comes.

Using the Command

Choose the **Paper Space Configuration** command and the following window appears:



Name: The name of the currently selected Paper Space Template.

Units: The unit of measurement used for specifying the margins for the Paper Space Configuration. Notice that the **Height** and **Width** values also automatically convert to the selected unit of measurement.

Paper Size: The paper size that your printer uses.

Width and **Height:** Measurements of the paper size selected in the units of measurement that are currently selected in the Units list box.

Portrait and **Landscape:** Vertical or horizontal paper orientation.

Margin: Set the margins from the top, bottom, left, and right.

Load Layout: Opens the **Paper Space Template** window which allows the selection of a previously saved Paper Space Template.

OK: Saves the changes you have made and opens Paper Space Mode.

Cancel: Disregards the changes you have made and returns you to your drawing.

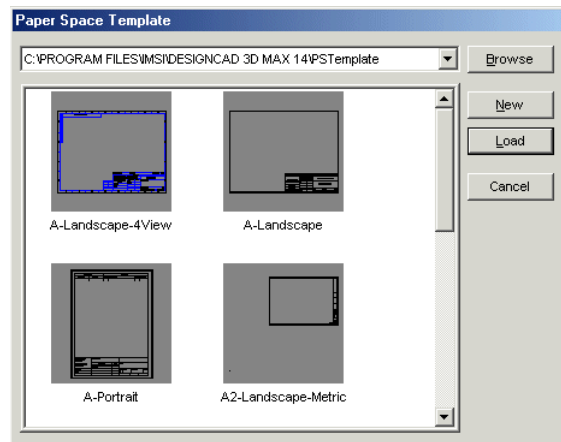
Load Paper Space Template

Menu: File / Paper Space View Frame Setup / Load Paper Space Template

Load or change a Paper Space Template.

Using the Command

Choose the **Load Paper Space Template** command and the following window appears:



Select the template you wish to use and click **Load**. To use a blank template, click **New**. Your drawing is arranged in the Paper Space according to the template you selected.

To change the template, follow the same steps. Your drawing is rearranged in the Paper Space according to the new template.

Automatic Rendering

Menu: File / Paper Space View Frame Setup / Automatic Rendering

A toggle that determines whether or not View Frames that have been set up to appear with hidden line removal or shading in Paper Space Mode will be rendered when the Paper Space Template is opened.

If **Automatic Rendering** is disabled, the drawing will initially display faster in Paper Space Mode because the view frames will open in wireframe, but the view frames that have hidden line removal or shading turned on, will still be previewed and printed that way.

Paper Space Layout

These commands are used to save your paper spaces as templates.

NOTE: *These commands are only available when DesignCAD is in Paper Space Mode.*

Current Layout Configuration

Menu: Layout / Current Layout Configuration (within Paper Space Mode)

Opens the **Configuration** (see Paper Space Configuration), which you can use to modify the current paper space.

Delete Current Layout

Menu: Layout / Delete Current Layout (within Paper Space Mode)

Deletes the current paper space layout.

Save Current Layout as Template

Menu: Layout / Save Current Layout as Template (within Paper Space Mode)

Saves the current Paper Space Template, including all the View Frames and the options set for each.

Using the Command

While in Paper Space Mode, choose the **Save Current Layout as Template** command. The command can also be selected by clicking on the **Save Current Layout as Template** icon in the **Paper Space Mode** window. Specify the name of the template and the location where you want to save it and click OK. The Paper Space Template is saved. Saving the template saves every View Frame's location and size, along with the specific options Templates you create and save can be loaded using the Load Paper Space Template command.

Print

Menu: File / Print

Shortcut Key: Ctrl+P

Toolbox Icon: **Fast Print** for the entire drawing



Fast Print for the current view

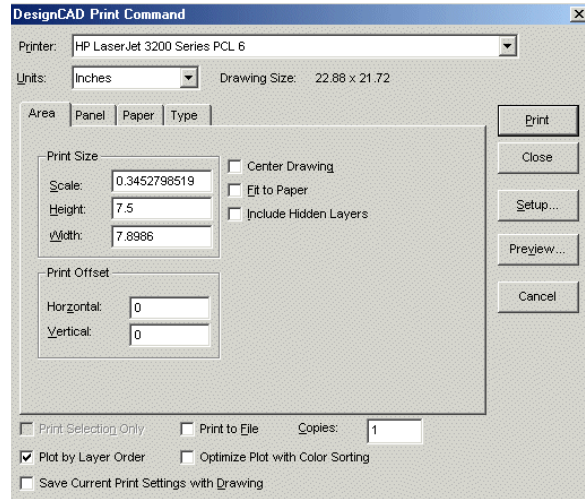


NOTE: *By default, clicking the **Print** icon automatically sends the current view to the currently selected printer and prints it at the largest size possible on a single sheet of paper. You can change this by selecting **Options / Options**, opening the **General** tab, and unchecking **Enable Direct Print from Toolbar**.*

Outputs your drawing to a printer or plotter. The drawing is printed at the view and perspective of the current view.

Using the Command

Choose the **Print** command, and the **Print** window appears:



DesignCAD offers many options, but you don't have to use them all. If you want to print your drawing on a single sheet of paper and the maximum size, just select **Fit to Paper** and click **Print**. The other options can be used to print to scale, rotate the drawing 90 degrees, set the margins, and so forth.

Printer: Select the printer for the pending print job from the list of available printers by clicking on the arrow to the right of the **Printer** field. If you would like to see the properties for the selected printer, click the Setup button.

NOTE: *If you do not see the printer you would like to use for the print job in the list of available printers, make sure the printer is installed correctly in Windows. If you have problems, consult your Windows documentation*

and your printer manual. Make sure you have not only the correct driver installed, but also that it is the most recent driver from the manufacturer.

Units: The unit of measurement for the paper on the printer. Don't confuse this item with Drawing Units. Drawing Units are the units of measurement of the drawing itself, not the printer.

Drawing Size: The size of the box that would just fit around the current view if the drawing were projected onto a flat surface. The size is given in DesignCAD Drawing Units.

Area: The size of your printed drawing.

Scale: A ratio that represents the number of paper units that will be used to print one Drawing Unit. Suppose, for example, that you are printing the front view of a 2x2 box, and you have chosen "inches" for the paper units. If you set Scale to 1.0, then one inch of paper will be used for each Drawing Unit and the box will be printed out at two inches by two inches on the paper. If you set Scale to 2.0, then two inches of paper will be used to print each Drawing Unit, and the box will be printed at four inches by four inches. With a Scale of 0.25, the box will be printed at 0.5 inches by 0.5 inches. See Printing to Scale.

Height and Width: The height and width of the printed drawing on the paper if printed at the current scale. If you are printing multiple panels, this number is the total height or width of the drawing across all panels.

Center Drawing: Centers the drawing on the paper.

Fit to Paper: Outputs the drawing on a single page in the largest possible size. If this option is selected, the **Scale**, **Height**, and **Width** cannot be changed.

Include Hidden Layers: Takes the entire drawing (including hidden layers and unselected objects) into consideration when determining the scale and center position of the printed drawing. This is an easy way to print transparencies.

Horizontal: How far the drawing will be printed from the left edge of the leftmost piece of paper for the print job. Positive values move the position of the printed image to the right.

Vertical: How far the drawing will be printed from the top edge of the topmost piece of paper for the print job. Positive values move the position of the printed image down.

Print Selection Only: Only the currently selected object or objects will be printed. If no objects are selected, this box is not available.

Print by Layer Order: Sends the drawing to a plotter one layer at a time.

*NOTE: The status of the **Print by Layer Order** option is not saved in the drawing file, thus you will need to reselect it when you get ready to plot the drawing in subsequent drawing sessions.*

Save Current Print Settings with Drawing: Saves the options set in the **Print** window so that the drawing can be printed again without resetting the options. This option also makes it possible to print the drawing with the same options using direct print from the Toolbar.

Print to File: Sends the print information to a file rather than to the actual printer. This is convenient if you need to print or plot your drawing at another location. It can also be used to transfer the drawing image to other applications.

Optimize Plot with Color Sorting: Sends the drawing to a plotter one color at a time.

*NOTE: The status of the **Optimize Plot with Color Sorting** option is not saved in the drawing file, thus you will need to reselect it when you get ready to plot the drawing in subsequent drawing sessions.*

Copies: The number of copies to be printed.

Print: Starts the printing process.

Close: Saves the changes you have made to the options in the **Print** window closes the **Print** window, and returns you to your drawing.

Setup: Displays the page setup options for the currently selected printer. See Page Setup.

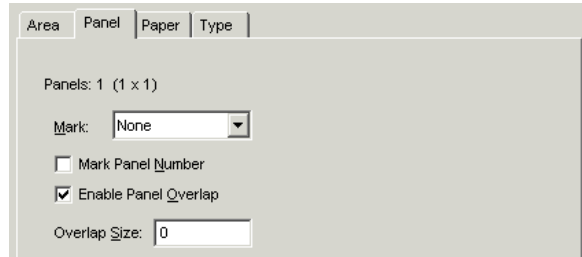
Preview: Displays the **Print Preview** screen, which shows exactly how your drawing will look on paper. See Print Preview.

Cancel: Disregards any changes you have made to the **Print** window and closes the Print window.

Panel Options

This group of options gives you information about how multiple panels will be handled. If a drawing is scaled too large to fit on a single page, DesignCAD will print the portions of the drawing on separate sheets of paper which can then be assembled. For example if a drawing is printed in four panels, then each panel would contain

one-quarter of the drawing. This makes it possible to print drawings at a large scale with standard paper sizes.



Panels: 1 (1x1): The total number of panels to be printed, and how many across by how many down.

Mark: How to mark the boundaries of the individual panels.

- **None:** No panel marking is done.
- **Corner Mark:** A registration mark is printed at each of the corners of the panels. This makes it easy to align the panels after printing.
- **Outline Box:** Boxes are drawn around each panel to indicate margin locations. Each panel can be trimmed to the edge of these boxes so that when the panels are assembled edge-to-edge, the alignment of the panels will be correct. This way no paper overlap will interfere with the image.

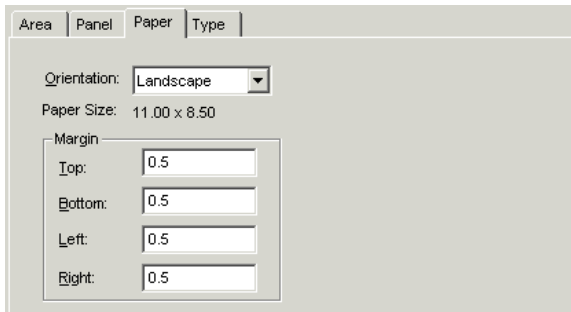
Mark Panel Number: Each panel will be numbered. This is useful in arranging the panels correctly when assembling the finished printout of the drawing.

Enable Panel Overlap: Eases the assembly of paneled drawings. This option creates a specified image overlap between the panels. This eliminates the need for precise panel trimming before assembly.

Overlap Size: The amount of overlap DesignCAD prints on a panel when the Panel Overlap option is used. The Overlap Size is applied to each side of every panel that overlaps another panel.

Paper Options

This group of options lets you choose your paper orientation and margins.



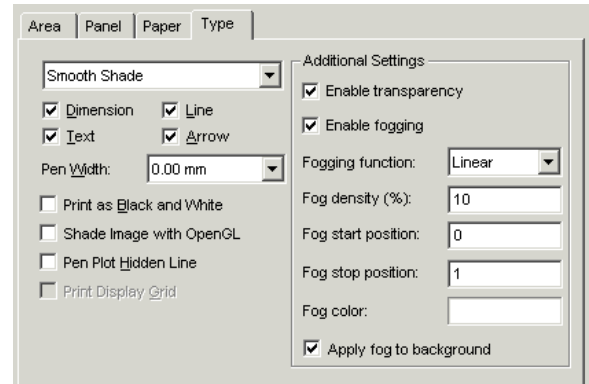
Orientation: Portrait is a vertical orientation, Landscape is horizontal. This can also be set via the **Setup** button on the **Print** window.

Paper Size: The currently selected paper size.

Margin: The size of the page margins, measured from the top, bottom, left, and right.

Type Options

These options let you print your drawing in several forms:



For the differences between wireframe, quick shade, smooth shade, and hide, see Object Display.

Dimension: Prints dimensions in the drawing.

Text: Prints text regardless of whether it is on the near side, inside, or far side of a shaded solid. This option also shows text in views in which the hidden lines have been removed.

Line: Prints lines regardless of whether they are on the near side, inside, or far side of a shaded solid. This option also shows all lines in views that have the hidden lines removed.

Arrow: This option prints arrows regardless of whether they are on the near side, inside, or far side of a shaded solid. This option also shows all arrows in views that have the hidden lines removed.

Pen Width: This option can be used to increase the print width of the lines in a drawing. With some high resolution printers, the zero width lines

are difficult to see or do not print as complete lines. This option alleviates some problems with printing zero width lines.

Print as Black and White: Some colors print very lightly on black and white printers. When you are not using a color printer to print your color drawings, check this option to print all drawing entities as if they were black.

Shade Image with OpenGL: Uses OpenGL shading to print the drawing.

Pen Plot Hidden Line: Send the print/plot information necessary for a hidden line removal to a pen plotter.

Print Display Grid: If you have used a display grid in your drawing and the drawing is in 2D Mode, this option tells DesignCAD whether or not to print the Display Grid with the drawing.

Enable Transparency: Show transparencies when a drawing contains objects that have been assigned materials that have transparent qualities.

Enable Fogging: Add a fog effect to the printed drawing.

Fogging Function: **Linear** produces proportionally more fog the "deeper" into the drawing you go. **Exp** produces a less dense fog than **Linear**. **Exp2** also produces a less dense fog than **Linear**, following a different function than **Exp**.

Fog Density (%): The maximum fog density.

Fog Start Position: The point in the drawing where the fog is at its minimum. This value is a scale factor based on the depth of the drawing from the front to the back at the current view settings. A value of 0 starts the fog at the very

front of the drawing. A value of -1 starts the fog one drawing-depth in front of the drawing's front. A value of 0.5 starts the fog in the middle of the drawing.

NOTE: This position and scaling will vary depending on the size of the drawing, the viewing angle, and distance settings.

Fog Stop Position: The point in the drawing where the fog reaches its maximum thickness. Like the Fog Start Position, this value is a scale factor based on the depth of the current view of the drawing.

Fog Color: The desired base color for the fog. To select the color, click the color box and select a color from the palette.

Apply Fog to Background: The entire drawing background becomes the same color as the fog.

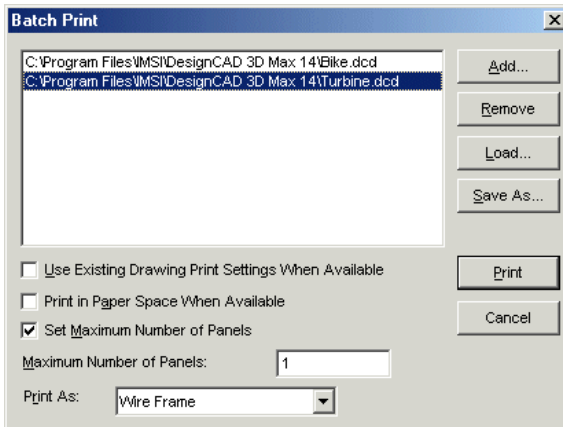
Batch Print

Menu: File / Batch Print

Prints several specified files at one time.

Using the Command

Choose the **Batch Print** command and the **Batch Print** window appears:



Add: Select the files to be added to a batch.

Remove: Remove files from a batch.

Load: This option is used to load a batch (.dbp) file that has been created and saved previously.

Save As: Save a batch (.dbp) file under a specified filename.

Use Existing Drawing Print Settings When Available: Use existing print settings, if the current print settings have been saved with the drawing by enabling the **Save Current Print Settings** option in the **Print** window.

Print in Paper Space When Available: Prints the drawing in Paper Space Mode, if a Paper Space has been configured for the drawing. See Paper Space.

Set Maximum Number of Panels: Makes it possible to constrain all of the files for the batch print to a specific number of panels.

Print As: Specifies whether the batch will be printed as Wireframe, Shading, Hide, or Hide (Pen Plotter).

Print: Sends the batch to the specified printer.

Close: Closes the window and returns to DesignCAD without printing or retaining any changes.

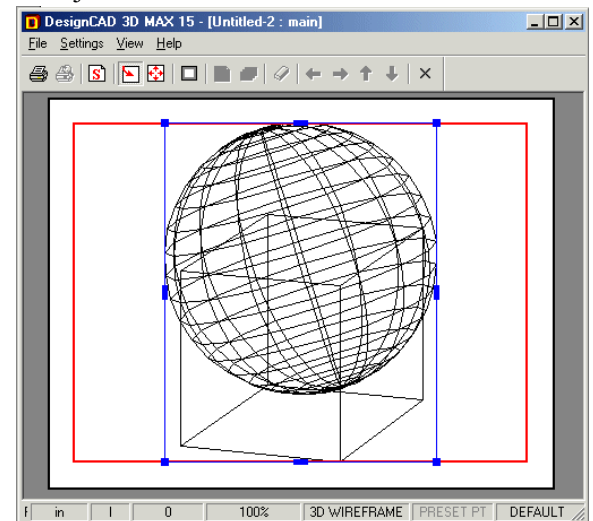
Print Preview

Menu: File / Print Preview

Toolbox Icon:



Shows exactly how your drawing is going to look on paper. There are also several options that can be adjusted from the **Print Preview** window.



Print: Prints the drawing.

Change Printer Setting: Brings up the **Print Command** window, making it possible to change the settings for the print job. Pressing Enter will print the drawing. To get back to the **Print Preview** screen, click the **Close** button.

Center Drawing: If the drawing fits on a single piece of paper, this command will center the drawing on the paper. If it is a multi-paneled drawing, this command centers the drawing so that there is the same amount of white space above and below the drawing and the same amount of white space on each side of the drawing.

Fit to Paper: Outputs the drawing on a single piece of paper in the largest scale possible.

Print Region: Lets you define a region of the drawing to be printed. Move the mouse to one corner of the region. Hold down the left button and drag a rectangle around the region. Release the mouse button at the opposite corner.

Show Single Panel: Shows only the currently selected panel, enlarged to fit the screen.

Show All Panels: Shows all panels of the drawing on the screen.

Mark/Unmark Skipped Panel: Works as a toggle to select or deselect individual panels and determine whether or not they will be printed. Panels that will not be printed are grayed out.

Left Page or Right Page: In a multi-paneled view, selects the panel to the left or right of the currently selected panel. In single-paneled view, changes the view so that the panel being viewed is the one to the left or right of the panel currently being viewed.

Up Page or Down Page: In a multi-paneled view, selects the panel directly above or below the currently selected panel. In a single-paneled view, changes the view so that the panel being viewed is the one directly above or below the panel currently being viewed.

Close Print Preview: Returns to the **Print** window.

When the entire drawing is shown, either on a single sheet or in panels, there are blue scaling and moving nodes around the drawing.

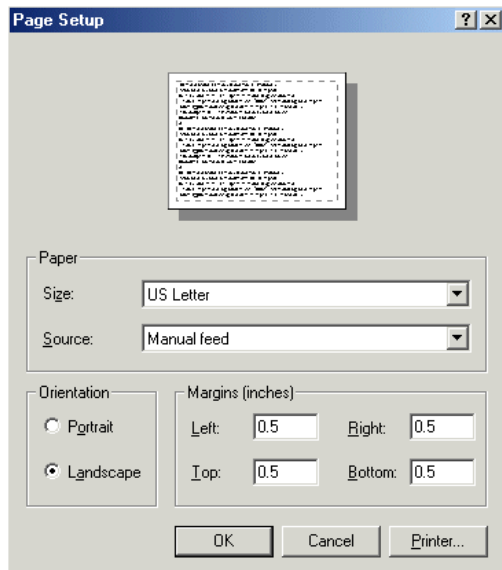
The small blue box at each corner is a scaling node and can be used to scale the drawing. To scale the drawing, click on one of the scaling nodes and hold down the mouse button. Move the mouse until the drawing is scaled to the desired size, then release the mouse button.

The small blue box on the top, bottom and each side of the drawing is a moving node. To move the drawing, simply click and hold on one of the moving nodes or one of the blue lines around the drawing. Use the mouse to move the drawing to the desired location, then release the mouse button. If you move the drawing past the edge of the pages shown, DesignCAD automatically adds pages as needed for the print job.

Page Setup

Menu: File / Page Setup

Used to change some of the DesignCAD printer options. This window can also be accessed by clicking the **Setup** button in the **Print** window.



Size: Select the paper size for your drawing from a list of standard sizes.

Source: Select the paper source from a list of standard printer paper sources.

Orientation: Portrait is a vertical orientation, Landscape is horizontal.

Margins: The size of the page margins, measured from the top, bottom, left, and right.

Printer: Brings up the window for selecting the printer for the pending print job.

OK: Saves the changes you have made and returns you to your drawing.

Cancel: Disregards the changes you have made and returns you to your drawing.

3 2D Drawing Tools

Points

Most 2D and 3D commands require points. You can snap to existing points or create new points. This section covers the various point options in DesignCAD.

Setting Points

Points form the basis of all drawing commands, determining the location of a line or curve, the diameter of a circle, or the radius of an arc.

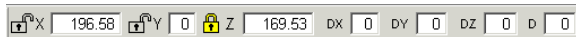
You can select point commands from the Command Menu or the Snap Toolbox, or you can use shortcut keys. The Snap Toolbox works the same way as the Main Toolbox does. To choose a point command, click on the icon for the command.

Setting a point is easy. Press the Ins key or click the left mouse button. That's all there is to it. The right mouse button activates the **Gravity** command (if the **Enable Right Click Popup Menu** option in the **General Options** - see **General Options** - is disabled).

See also Snap Points.

Coordinates

The location of the cursor is displayed in the Coordinate Toolbox. (For details on displaying toolboxes, see View Options.)



NOTE: In 3D Mode, the Coordinate Toolbox also has fields for Z coordinates

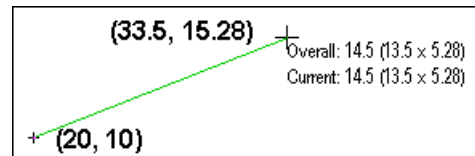
You can use the Coordinate Toolbox to see exactly where the cursor is, or to enter exact values.

Example: Creating a line

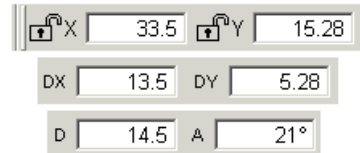
Choose the Line command. For the first point, place your cursor inside the X field of the Coordinate Toolbox and enter 20. Press Tab to jump to the Y field and enter 10.



Press Tab again or press Enter to place the first point of the line at (x=20, Y=10). Move your cursor to see the rubber-band line (do not click to place the point). Its total length and “delta” coordinates are indicated - the distance in X and Y measured from the first point (not the absolute coordinate).



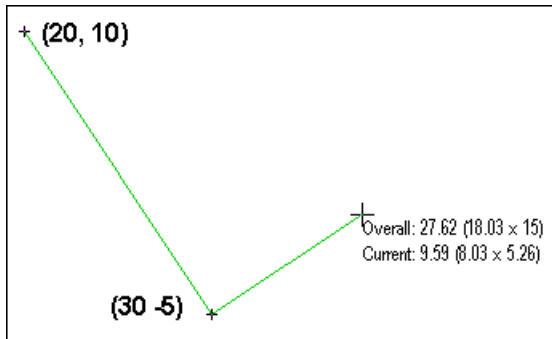
Look at the Coordinate Toolbox. X and Y contain the actual coordinates of the second point. DX and DY contain the relative distances - measured from the first point at (20, 10). D is the overall length of this segment, and A is the angle measured from horizontal.



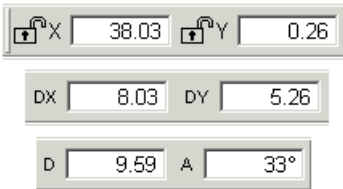
To place the second point, place the cursor in the X field and enter 30. Enter -5 for Y and press Enter or Tab to place this point.



Now you can start to draw the second segment of this line. The **Overall** length is the total of all segments. The **Current** length is the length of just the current segment.



In the Coordinate Toolbox, DX and DY reflect the length measured from the (30, -5) point. The D and A fields contain the length and angle of just the current segment.



Locking Coordinates

Menu: View / Lock X (or Y, or Z) Coordinate

Shortcut Key: Shift+X, Y, or Z

Toolbox Icon: Lock symbol in Coordinate Toolbox.

Point 1: Point to specify the desired X coordinate

Restrict points that can be set to a fixed value of X, Y or Z.

Using the Command

Select the **Lock X (or Y, or Z)** command. Set a point at the coordinate at which you want to set all points while the coordinate is locked. Snap commands and Point XYZ are useful for setting the point.

While a coordinate is locked, a lock icon appears beside that coordinate in the Coordinate Toolbox. A locked coordinate may be changed by typing a new value for that coordinate directly into the Coordinate Bar. Just remember that if you change a value in the Coordinate Bar, you need to press the Enter key to return the focus to the drawing area.

Once the coordinate no longer needs to be locked, select the **Lock** command again to disable it. Only one coordinate may be locked at one time.

Lock X Coordinate is handy if you are drawing in the Side View of your drawing and want to make sure that you are always using the same value of X.

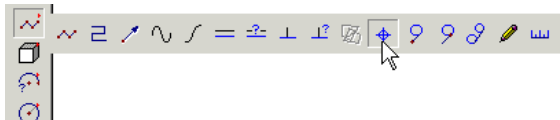
Lock Y Coordinate is very helpful when you are drawing in the Top View and want to maintain a constant Y value.

Lock Z Coordinate is useful when you are drawing in the Front View and want to draw on a fixed Z depth.

Pointmark

Menu: Draw / Pointmark

Toolbox Icon:

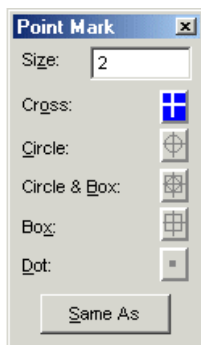


Point 1: Location for point mark.

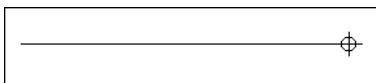
Marks a point in the drawing with a small cross, box, circle or combination of these elements.

Using the Command

Choose the **Pointmark** command in the Toolbox. Select the marker style by clicking a button in the window.



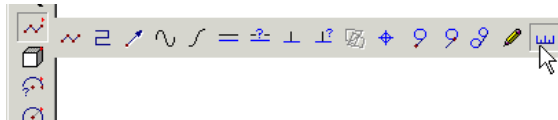
To draw the point mark the same size as another point mark in the drawing, click the **Same As** button. Then click a point mark in the drawing. Set a point where you want the point mark. A mark is placed there. Set more points for any additional point marks. Press the Enter key to end the command.



Tickmark

Menu: Draw / Tickmark

Toolbox Icon:



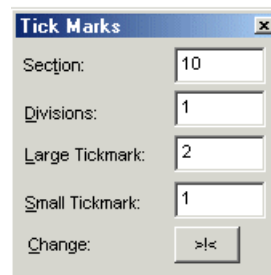
Point 1: Point on line, curve, or arc

Point 2: Point indicating on which side of line, curve, or arc tick marks are to be drawn

Draws short lines, or tick marks, along a line, curve, ellipse, or circle entity. These tick marks are spaced at specified intervals along the entity.

Using the Command

Choose the **Tickmark** command and the following window appears:



Enter the number of **Sections** of large tick marks - this determines the number of large tick marks to be evenly measured out along the length of the item being marked.

If you would rather determine the exact distance between large tickmarks click the **Change** button in the window. The **Sections** box will be replaced with the **Mark Distance** box. This box lets you

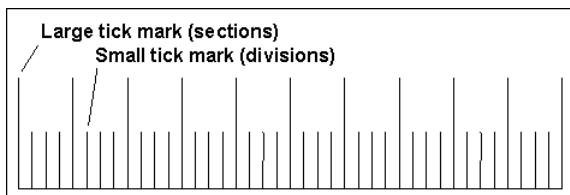
specify the exact distance between the large tick marks. To switch back to the **Sections** box, click the button again.

Enter the number of **Divisions** of tick marks - this divides each section up with small tick marks.

Enter the **Large Tickmark** and **Small Tickmark** sizes. Set a point on the existing line. Set a point to indicate on which side of the line the tick marks will be drawn. If both points are set in the same location, the tick marks will be centered on the line. Tick marks will be drawn at even intervals along the line, on the side of the second point.

Example: Draw tick marks along a 10 unit line.

Draw a 10-unit-long line using the **Line** command and choose the Tickmark command. Enter 10 in the **Sections** box and 4 in the **Divisions** box. Set a point on the line. Decide on which side of the line the tick marks should be placed. Set another point to that side of the line. The tick marks will be drawn as follows: 10 one-unit sections, 4 divisions to each section.



Snap Points

Snaps are points that the cursor can automatically jump to, such as grid points, an endpoint or midpoint of a line, or the center points of an arc. See Grid and Snapping.

To snap without setting points, use **Move Cursor Only**.

Toolbox Icon:



NOTE: When Snap commands are selected from the Snap Toolbox, you have the option of making them set a point or just move to the position; however, if you select a Snap command from the **Point** menu, a point will always be set.

Gravity

Menu: Point / Gravity

Shortcut Key: . (period)

Mouse Shortcut: Right Mouse Button

Toolbox Icon:



This very useful command moves the cursor to the nearest point in the drawing and sets a point there. It enables you to set a point exactly on another point without having to "zero in" on it.

Using the Command

When you choose the command while in 3D Selection Mode, the cursor moves to the nearest point in 3D space. In 2D Selection Mode, the cursor moves to the nearest point on the screen and does not take into account the point's location in 3D space.

This is an important difference. If you want to be able to snap to the nearest point on the screen, you should use the 2D Selection Mode. However, in 2D Selection Mode, the cursor may seem to move

in an arbitrary manner along the Z axis when you snap to a point because the cursor takes on the Z value of the point that it snaps to.

In 3D Selection Mode, the cursor may seem to jump to a point other than the nearest point on the screen. This is because it goes to the nearest point in the drawing, in 3D space. That point may not be the point that appears closest on the screen.

Example: Set a point exactly on the right endpoint of a line.

Select the **Gravity** command. Move the cursor to a point near the right end of the line. Click the mouse button. The cursor snaps to the right endpoint and sets a point.

Line Snap

Menu: Point / Line Snap

Shortcut Key: K

Toolbox Icon:



Moves the cursor to the nearest line and sets a point there. It can be very useful in conjunction with drawing commands. For example, if you are drawing a line and want one of the endpoints to lie exactly on another line, you can use **Line Snap** to accomplish this without having to "zero in" on the line.

Using the Command

Choose the **Line Snap** command and set a point near the line to which you want to snap. The cursor snaps to the nearest point on the line and sets a point there.

Example: Set a point exactly on a line in your drawing.

Choose the **Line** command again and then, before setting a point, choose the **Line Snap** command. Move the cursor close to the line and click the left mouse button. The cursor snaps to the line and sets the starting point of your second line.

Plane Snap 3D only

Menu: Point / Plane Snap

Shortcut Key: F7

Toolbox Icon:



Point 1: Point on or near the plane to which you want to snap

Moves the cursor to the nearest location on the nearest surface and sets a point there.

Using the Command

Choose the **Plane Snap** command and set a point near the plane you want a point on. The cursor snaps to that plane and sets a point there.

Intersect-1

Menu: Point / Intersect-1

Shortcut Key: F4

Toolbox Icon:



Point 1: Line on which you want to find an intersection

Finds the nearest intersection to a point.

Using the Command

Choose the **Intersect-1** command. Set a point on the line on which you want to find the intersection. The cursor will move to the nearest intersection of that line and a point will be set there.

This command works on lines, circles, arcs, planes, and curves.

Example: Find the intersection of a line and a circle.

Select the **Intersect-1** command. Set the cursor on the line and click the left mouse button. The cursor will move to the intersection of the line and the circle, and a point will be set there.

Intersect-2

Menu: Point / Intersect-2

Shortcut Key: Shift+F4

Toolbox Icon:



Point 1: First line of intersection

Point 2: Second line of intersection

Locates the intersection of two entities, using two points as references.

Using the Command

Select the **Intersect-2** command. Set a point on each entity. The cursor will move to the point at which they intersect. If the lines do not meet, then the cursor will move to a point where they would meet.

This command works on lines, circles, arcs, planes, and curves.

Example: Find where two lines intersect in a "web" of intersecting lines.

Choose the **Intersect-2** command and set a point on one of the lines. Next set a point on one of the intersecting lines. The cursor will move to the intersection of the two lines.

Line Plane ^{3D}only

Menu: Point / Line Plane

Toolbox Icon:



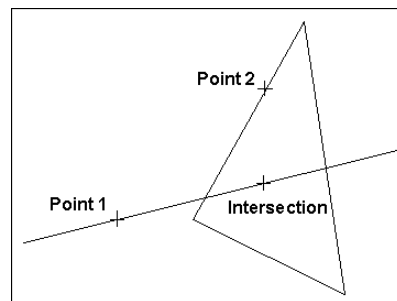
Point 1: Set a point on the intersecting line

Point 2: Set a point on the intersecting plane

Sets a point at the intersection of a line and a plane.

Using the Command

Select the **Line Plane** command. Set one point on the line and a second point on the plane. If the line does not touch the plane, a point will be set where the intersection would be if the line were extended.



Midpoint

Menu: Point / Midpoint

Shortcut Key: Ctrl+K

Toolbox Icon:



Moves the cursor to the midpoint of the closest line or line segment and sets a point there.

Using the Command

Select the **Midpoint** command, move the cursor near the line you want to snap to, and select the line.

NOTE: In 2D Mode, this command will snap to the midpoint of a line's projection on the XY plane, not to the midpoint of the actual 3D line.

Midpoint2 ^{3D}only

Menu: Point / Midpoint2

Shortcut Key: Ctrl+Shift+K

Toolbox Icon:



Moves the cursor to the midpoint of two points, and sets a point there.

Using the Command

Select the **Midpoint2** command. Set two points. The cursor snaps to a position midway between the points and sets a point there.

Center of Gravity

Menu: Point / Center of Gravity

Toolbox Icon:

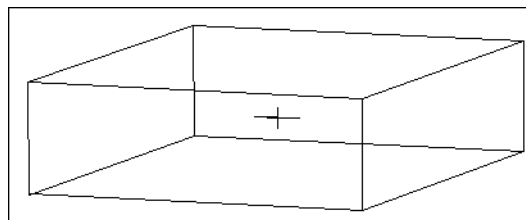


Point 1: Point on the object

Finds an object's center of gravity. For example, you can use this command to find the volumetric center of complex solids. This command is also useful for finding the geometric center of a plane.

Using the Command

Select the **Center of Gravity** command and set a point on or near the object. The cursor snaps to the object's center of gravity.



If you select this command from the **Point** menu, it will always set a point. If the **Move Cursor Only** button is pressed on the **Snap Toolbox** and you select the command from the **Snap Toolbox**, the command will just move to the position instead of setting a point there.

Tangent Snap

Menu: Point / Tangent Snap

Toolbox Icon:

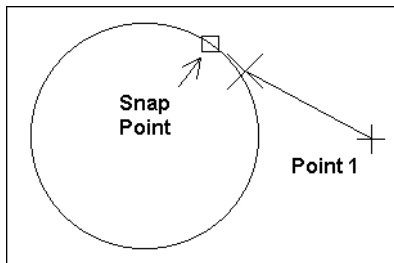


Point 1: Point near the circle or arc

Sets a point for a line or curve tangent to an existing circle or arc.

Using the Command

Set the first point for a line. Choose the **Tangent Snap** command. Move the cursor near an existing circle. A small box appears to show where the cursor will snap if the current cursor position is used. Set a point. The cursor snaps to the nearest tangent location and sets a point. Continue setting points for the line or press Enter to end the command and draw the line.



NOTE: If you are not in 2D Mode, the line and circle must lie in the same plane.

Origin

Menu: Point / Origin

Point 1: New location for the drawing's origin

Sets the origin (0,0,0) anywhere in the drawing. This can be convenient if you want to use coordinates relative to a particular point on the drawing.

Using the Command

Choose the **Origin** command. Set a point anywhere on the screen. The location of this point becomes the new origin. The origin remains the same until it is changed again. The X,Y,Z location on the coordinate bar reads 0,0,0 at the selected origin.

Point XYZ

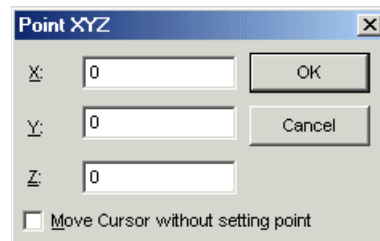
Menu: Point / Point XYZ

Shortcut Key: : (colon)

Sets a point by specifying the X, Y, and Z coordinates of that point.

Using the Command

Select the **Point XYZ** command. Enter the coordinates in the X, Y, and Z boxes and click OK. You can also move the cursor to the new location without setting a point there.



Point Relative

Menu: Point / Point Relative

Shortcut Key: ' (single quote)

Set a point relative to the last point set, the origin, a reference point, or relative to the cursor position. The position of the new point is given in horizontal (X), vertical (Y), and depth (Z) distances from the specified location.

Using the Command

Choose the **Point Relative** command. Enter the X coordinate in the DX box, the Y coordinate in the DY box, and the Z coordinate in the DZ box. Normally you'll want to use the **Last Cursor Position** option. Click OK button to set the point.

If you select the command from the pull-down menu, the **Last Cursor Position** option will not be available because the cursor is off the screen. The same occurs if you use the keyboard (Alt+P, then R) to activate the command. If you have set no points and you activate the command from the Command Menu, the point is placed relative to the origin or a reference point. If you press the ' (apostrophe) shortcut key, then you have the **Last Cursor Position**, **Origin**, and **Reference Point** options.

You can move the cursor to the new location without setting a point by selecting that option.

If the **Reset Values to Zero Next Time** option is selected, the DX, DY, and DZ values will be reset to zero the next time the command is used. Otherwise DesignCAD will remember the DX, DY, and DZ values and default to those the next time the command is used.

Example: Draw a circle with a radius of exactly 10 units.

Select the **Circle (Center, Outside)** command and set a point for the center. Then press the ' (apostrophe) key to select the **Point Relative** command. Enter 10 in the DX box. Press Enter to complete the circle.

Point Polar

Menu: Point / Point Polar

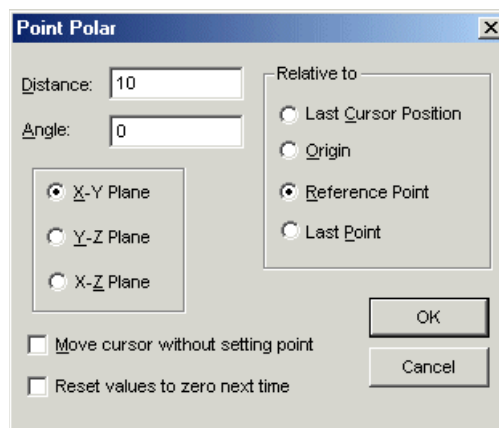
Shortcut Key: ; (semicolon)

Sets a point by specifying the distance and angle (polar coordinates) of the new point from the last point set.

This command is used inside another drawing command (such as **Line**) to set a point at a specific angle and distance from the last point set or the last cursor position.

Using the Command

Choose the **Point Polar** command. In the **Point Polar** box, enter the **Distance** and **Angle** from a given location.



You can specify the angle in either the X-Y, Y-Z, or X-Z plane. You can also specify whether you want the new point to be relative to the **Last Cursor Position**, **Origin**, **Reference Point**, or **Last Point**.

The options available depend on how you select the command. If you select the command from the pull-down menu, you will not have the option of **Last Cursor Position** because the cursor is off the screen. The same occurs if you use the keyboard (Alt+P, then A) to activate the command. If you have set no points and you activate the command from the Command Menu, the point is placed relative to the origin or a reference point. If you press the ; (semicolon) shortcut key, then you have the **Last Cursor Position**, **Origin**, and **Reference Point** options.

You can move the cursor to the new location without setting a point by selecting that option.

If **Reset Values to Zero Next Time** is selected, the **Distance** and **Angle** values will be reset the next time the command is used. Otherwise, DesignCAD will remember the **Distance** and **Angle** values and will default to them the next time the command is used.

Example: Draw a 20-unit line in the X-Y plane at an angle of 45 degrees.

Select the **Line** command and set the first point. Then press the ; (semicolon) key to choose the **Point Polar** command. Enter 20 for **Distance** and 45 for **Angle** and check the **X-Y Plane** and **Last Point** options. Press Enter to execute the command. The second point of your line will be 20 units away from the first at a 45-degree angle. Press Enter to insert the line into your drawing.

Gravity Move

Menu: Point / Gravity Move

Shortcut Key: , (comma)

Snaps the cursor to the nearest point in the drawing, but it does not set a point there.

Using the Command

This command lets you move the cursor to another point precisely without having to "zero in" on it. As soon as you choose the command, the cursor moves to the nearest point in the drawing, without setting a point.

Gravity Move is often convenient for moving the cursor to a point in the drawing to get to a known location before using the cursor keys or the **Point Relative** command.

In 3D Selection Mode, the cursor moves to the nearest point in 3D space. In 2D Selection Mode, the cursor moves to the nearest point on the screen and doesn't take into account the point's location in 3D space.

This is an important difference. If you want to be able to move to the nearest point on the screen, you should use the 2D Selection Mode. However, in 2D Selection Mode, the cursor may seem move in an arbitrary manner along the Z axis when you snap to a point because the cursor takes on the Z value of the point that is moved to.

In 3D Selection Mode, the cursor may seem to jump to a point other than the nearest point on the screen. This is because it goes to the nearest point in the drawing in 3D space. That point may not be the point that appears nearest on the screen.

Example: Move the cursor to the right endpoint of a line.

Select the **Gravity Move** command. Move the cursor to a point near the right end of the line. Click the mouse button. The cursor moves to the right endpoint but does not set a point there.

Point Control

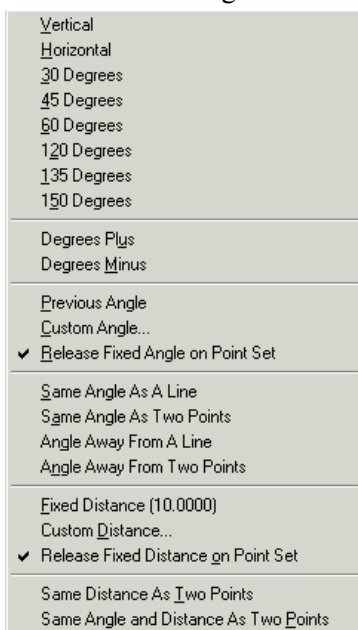
Menu: Point / Point Control

Shortcut Key: J

The **Point Control** submenu contains several simple commands. These commands can be used after the first point is set in a drawing command to insert another point relative to the first by a specified angle or distance.

Using the Command

After selecting a drawing command and setting the first point, select **Point Control** and a menu of simple commands appears. Choose the desired command from this listing. The movement of the cursor is restricted according to the command.



NOTE: The **Custom Distance** command is available in 2D mode and 3D mode. The other **Point Control** commands are available in 2D mode only.

Vertical, Horizontal: Forces the cursor to move parallel to the Y or X axis.

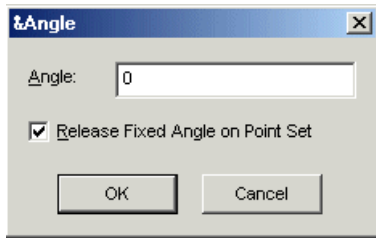
30 Degrees, 45 Degrees, etc.: Restricts movement of the cursor to a 30-degree, 45-degree, etc. angle.

Degrees Plus, Degrees Minus: Restricts the cursor movement to 5.0 degrees more or less than the previous angle. Two points must be set with one of the drawing commands before this command is accessible. These two points determine the previous angle to which the **Degrees More** or **Less** command adds or subtracts 5 degrees.

Previous Angle: Before becoming accessible, the two points must be set with one of the drawing commands. These first two points determine the angle at which the this command restricts the cursor. The third point is set so that a straight line could be drawn through all three points.

Custom Angle: Restricts cursor movement to an angle entered in the **Angle** window. Leave the **Release Fixed Angle on Point Set** option checked to release the cursor when you set a

point. Uncheck the checkbox to use the **Custom Angle** command for several points along the same angle.



Release Fixed Angle on Point Set: This is just another way to enable or disable the same option that is available in the **Custom Angle** window.

Same Angle as a Line: After setting the first point in a drawing command, choose this command and set a point on an existing line. The cursor is restricted to the angle defined by the line for the insertion of the next point.

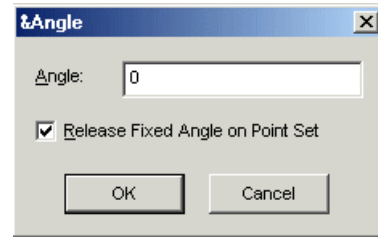
Same Angle as Two Points: After choosing this command, set two points. This command restricts the cursor to the angle defined by these two points.

Angle Away from a Line: Lets you place a point on a line. The cursor is restricted to a specified angle away from that line. Set the desired angle in the window.

Angle Away from Two Points: Allows you to set two points that define an angle. If an imaginary line were drawn through the two points, the cursor is restricted to a specified angle away from that imaginary line. Set the desired angle in the window.

Fixed Distance: Lets you set a point a fixed distance from the previous point. The distance used by this command may be changed by using the **Custom Distance** command.

Custom Distance: Lets you change the distance used in the **Fixed Distance** command. Uncheck the **Release Fixed Distance on Point Set** option if you want to set multiple points using the **Fixed Distance** command.



Release Fixed Distance on Point Set: This is just another way to enable or disable the same option available in the **Custom Distance** window.

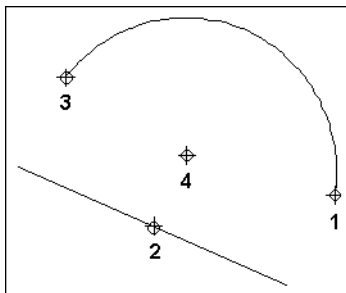
Same Distance as Two Points: Uses two points to determine the distance that the next point will be set away from the previous point in a drawing command. For instance, this command is useful if you are drawing a line and want the current segment to be the same length as a previous segment in the line or a line segment in another line in the drawing. Just select the command, and set a **Gravity** point on both ends of the existing segment. The next point is constrained to that distance from the previous point in the line currently being drawn. Move the cursor to the desired angle and click the mouse button.

Same Angle and Distance as Two Points: Uses two points to determine the angle and distance that the next point will be set away from the previous point in a drawing command. For instance, this command is useful if you are drawing a line and want the current segment to be drawn at the same angle and length as a previous segment in the line or a line segment in another

line in the drawing. Just select the command, and set a **Gravity** point on both ends of the existing segment. The next point is constrained to that angle and distance from the previous point in the line currently being drawn. Just move the cursor to determine the direction of the segment and click the mouse button.

Example: Set the second point for an arc using the **Same Angle as a Line** command.

Choose the **Arc (Endpoints, Center)** command. Set the first point for the arc. Choose the **Same Angle as a Line** command and set a point on the existing line that lies at the desired angle. The cursor is restricted to that angle. Set the second point for the arc. Set a third point for the radius of the arc. The angle between the two endpoints of the arc is the same as that of the line.



Lines

This section covers methods for creating various types of lines and curves.

Double Line Mode 2D only

Menu: Options / Double Line Mode

Toolbar Icon:

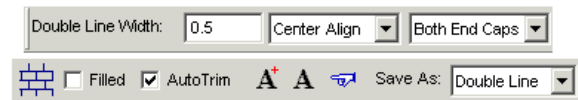


When simple 2D lines and curves are drawn in Double Line Mode, DesignCAD draws two parallel entities at the same time. Double Line Mode simplifies several common types of drawings, including house plans and land plots.

Using the Command

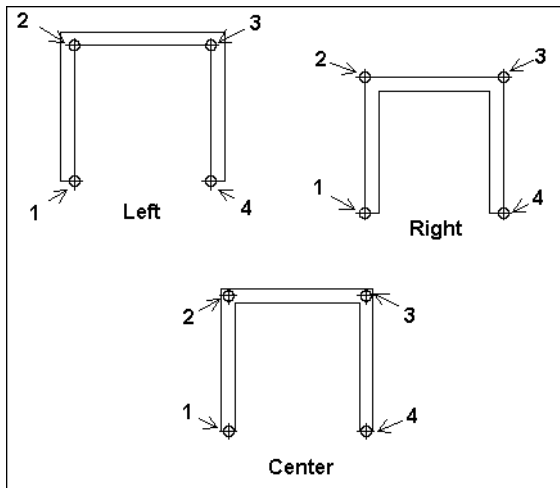
Choose the **Double Line Mode** command simple 2D entities such as arcs, circles, curves, and lines will automatically be drawn with duplicate parallel entities.

The **Double Line** Toolbar to control where and how the parallel entities are drawn. (See View Options for displaying toolbars)



Double Line Width: Sets the width of the entity (the amount of space between the two parallel entities) in Drawing Units.

Align: Select **Center**, **Left**, or **Right** to set the position of the width of the double entity. If **Left** is selected, the width of the entity will extend to what would be your left if you were standing on the first point set for the entity facing the second point set for the entity.



NOTE: If the double entity is drawn with the **Left** or **Right** option, you will be able to snap to both sides of the finished double entity. If the double entity is drawn with the **Center** option selected, you will be able to snap to both sides and the center of the double entity.

Cap: Specify which ends of the double entity are capped or left open.

Hatch: Select a hatch pattern with which to fill the entity. The **Filled** option must be checked for the hatch pattern to be used.

Filled: Draws a filled entity, otherwise the double line will be hollow.

Auto Trim: Automatically trims the entity where it overlaps itself or another double entity.

NOTE: The **Explode** command can be used to explode a double entity into vectors. If you are going to cross one double entity over another, explode the existing double entity and uncheck the **Auto Trim** box before drawing the second double entity to ensure that all four entities will be visible in the intersection.

Apply to Selected Double Lines: Applies the current options to selected double entities in the drawing.

Apply to All Double Lines: Applies the current options to all double entities in the drawing.

Same As: Match the parameters of an existing double entity. Click on the double entity already in the drawing that has the properties you want to assign to the options in the Double Line bar.

Save As: Select **Double Line**, **Plane**, or **Line** to determine how the entity will be recognized and saved in the drawing.

NOTE: You can convert existing entities to double-line entities. See Convert to Double Line. See also Double Line to Plane and Double Line to Wide Line.

MultiLine Mode 2D only

Menu: Options / MultiLine Mode

Toolbar Icon:



When separate 2D lines, curves, arcs, circles, and ellipses are drawn in "Multiline" mode, DesignCAD simultaneously draws a set of

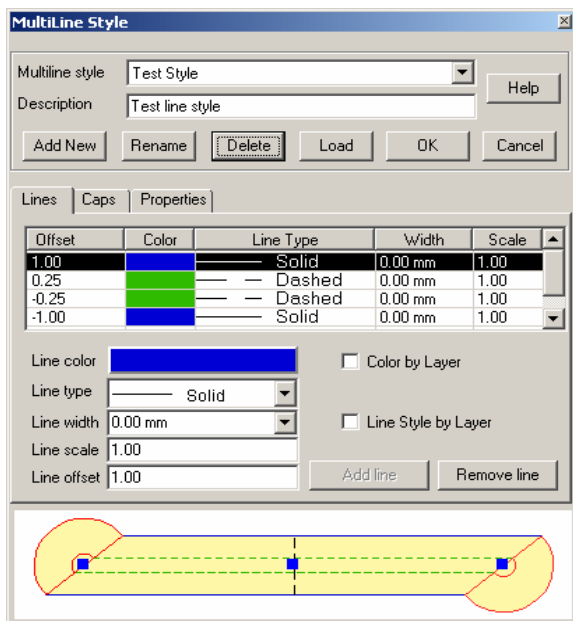
parallel objects. The same operations, as those permitted for "Double Line" objects, are also available for multiline objects. In essence, "Multiline Mode" is "Double Line Mode" with advanced capabilities for setting line representation.

When "Multiline" mode is active, the created 2D lines, curves, arcs, circles, and ellipses are displayed as multiline objects, with the current multiline style used.

Creating Multiline styles

The Multiline Style dialog

(Menu: Options / Multiline style...) is used to create and edit multiline styles:



The Multiline style list contains all multiline styles available in a given document. The Selected element of the list corresponds to the style that DesignCAD uses to draw a new 2D object in "Multiline" mode. This style is considered the current style.

Each multiline style has a Name, shown on the list, and a Description (Description field). Style name is specified at the time of creation, and can be changed by the user. Styles can have identical names. Style description is optional, and can be used, for instance, to specify more precisely the purpose and particular details of using a given style.

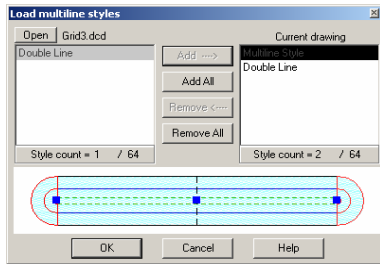
Add New: is used for adding a new style. When you add a new style, you need to enter its name and description (if desired). New style will have the current settings for Lines, Caps, and Properties. Thus, to create a new style you can select an existing one, and change it as desired. There can be a maximum of 64 styles for one drawing.

Rename: is used to change the name of the current style.

Delete: is used to remove the current style.

Load: is used to load styles from a file. Multiline styles can be loaded from any *.dcd file created in DesignCAD 16. After pressing "Load" you can

select the file, which contains the required styles, in the "Load multiline style" dialog ("Open" button).



The list under the button (on the left) will show all multiline styles that exist in the file. To transfer a style into the current document, use "Add" and "Add All" buttons, to cancel the transfer - "Remove" and "Remove All". "Remove" and "Remove All" buttons work only for the styles added from the list on the left. Existing styles cannot be removed using this dialog.

By default, when a new document is created, styles are loaded from Multiline.sys file. It is a regular *.dcd file with the extension changed, and it can be opened in DesignCAD. To set the default styles, you can open this file, create or load the required styles, save the file as *.dcd, and then change its extension to *.sys.

If the program cannot find this file, then when a new document is created, one style will be added automatically: it will have the name Double Line, and style line settings that are used in Double Line Mode.

Editing Multilines

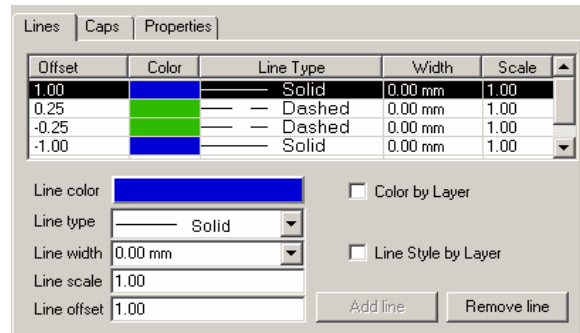
Multiline styles can be edited in the same "Multiline Style" dialog.

To work with styles of the entire document, and also to set the style to be used, this dialog can be called from (Menu: Options / Multiline style...).

To change multiline style of the current multiline object, you can invoke its InfoBox, and press "Change Style" button there. In this case, all operations for adding, removing, and loading styles will be inaccessible, and "Multiline style" list will contain, apart from all styles of the document, one more style: it will have the name of one of existing styles with the word (Current) appended to it. This is the style that is currently used to display the selected object.

All style settings are located on three pages: "Lines" - settings for positioning lines relative each other, and controlling their display properties; "Caps" - settings for multiline caps; "Properties" - general style settings.

Lines page



This page contains all settings for positioning lines relative each other, and controlling their display properties (color, style, width, scale).

All information about lines used in the current multiline style is presented in the spreadsheet.

Offset is used as the parameter for sorting.

After selecting a line in the spreadsheet, user can edit its properties and settings.

Line Color - defines line color. When "Color by Layer" flag is set, DesignCAD will use the color of the layer, where the object was created.

Line Type - defines line type. Here you can choose one of the types available in DesignCAD.

Line width - line width (in millimeters). This width does not depend on document scale, and always stays the same. It is similar to "Fixed Lineweight" for single lines and curves.

Line Scale - scale applied to the selected line type.

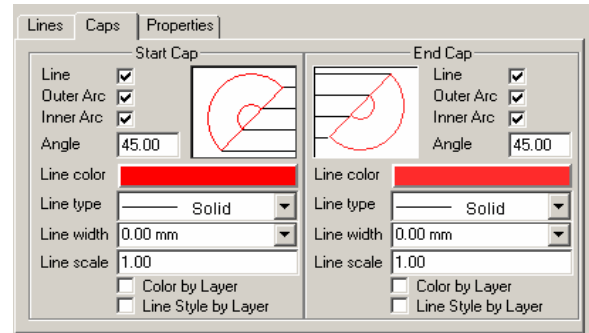
When "Line Style by Layer" flag is set, type, width and scale of the line will match those specified for the layer, where the object is located.

Line Offset - relative offset of lines, which determines the distance between them. The offset is calculated from a certain 0 axis, in scale units used in DesignCAD at the moment. For example, if a style uses 4 lines, the first of which has offset equal to 2, and the fourth - equal to 8, the width of such line will be 6.

To add a line, select an empty row in the spreadsheet, set line properties, and press "Add line" button.

To delete a line, select it, and press "Remove line" button. The only limitation for this operation - each multiline style must have at least 1 line defined.

Caps page



Defines shape, style, and color of lines for caps. There are separate settings for start and end of a multiline.

Line flag - when this flag is set, the corresponding boundaries (start or end points) of the first and last lines are connected with a line.

Outer Arc flag - when this flag is set, the corresponding boundaries (start or end points) of the first and last lines are connected with an arc.

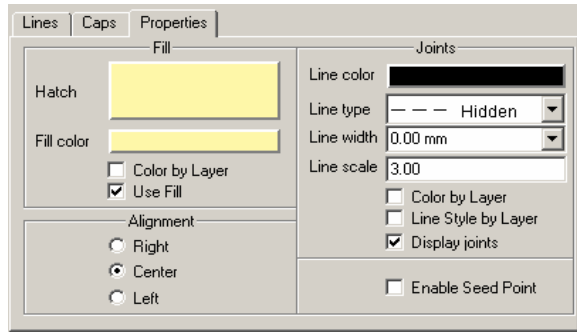
Inner Arc flag - when this flag is set, all boundaries of internal lines (between the first and the last one) are connected in pairs, with arcs. The pairs are formed from the lines on opposite sides, e.g. the second line is connected with the next to last one.

Angle field - determines the tilt angle of the cap. The angle is calculated from the lower line. The angle is specified in degrees, from 0 to 180. It is used only for multilines. For other multiline objects, it is automatically set at 90 degrees.

"Line color", "Line type", "Line width", "Line scale", "Color by Layer" and "Line Style by Layer" define the settings for style and color of

the line for the corresponding cap; they are similar in functionality to control elements of **Lines** page, bearing the same names.

Properties page



This page contains all other settings for multiline style. These include "Fill" and "Alignment" settings, as well as "Joints" - settings for representation of line inflection points.

Fill - contains the following settings for multiline object fill:

Hatch - defines the style, scale and tilt angle of the fill.

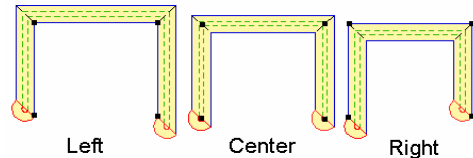
Fill color - defines the color of the fill.

Color by Layer - when it is specified, the color of the layer, where the multiline object was created, will be used for the fill.

Use Fill - determines whether fill will be used.

Enable Seed Point - links fill to a point. When this flag is set, the fill will be displayed the same all the time, regardless of line position and operations performed with it.

Alignment - settings for aligning a multiline object.



Joints - settings for representation of line inflection points.

These settings are used only for multilines; they are turned off automatically for all other multiline objects.

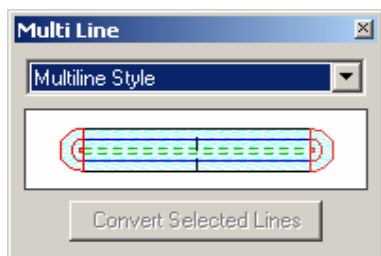
"Line color", "Line type", "Line width", "Line scale", "Color by Layer" and "Line Style by Layer" define the settings for style and color of the line; they are similar in functionality to control elements of "Lines" and "Caps" pages, bearing the same names

Display Joints - when this flag is set, a straight line (with color and style specified by abovementioned settings) will be drawn between the points on the 1st and last lines, corresponding to the point on the multiline object (except the 1st and last points, because the caps are drawn there).

Some of multiline style settings for existing objects are available in InfoBox, and can be changed without using "Multiline style" dialog.

Converting to a multiline

Single lines and curves can be converted to multiline objects using "Convert to MultiLine" dialog (Menu: Edit / Convert to MultiLine).



Here, the you can select an existing multiline style and click close to the object that he wants to convert. If conversion is possible (the object is a single 2D line, a curve, an arc, a circle or an ellipse), the selected object becomes a multiline object.

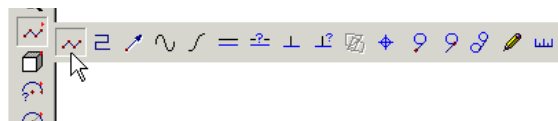
It is possible to convert a group of selected objects using "Convert Selected Lines" button. In this case, only those objects will be converted, for which conversion is possible.

Line

Menu: Draw / Lines / Line

Shortcut Key: V

Toolbox Icon:



Draws a single or multi-segment 2D or 3D line. A valid line entity must contain two or more points.

Using the Command

Choose the **Line** command and set points for the line. You will see a rubber-band line connected to each point you set. When you have set the points, press Enter or double-click the mouse to end the command.

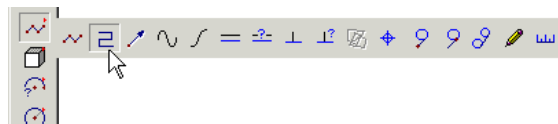
*NOTE: This command is similar to the **Plane** command except the **Plane** command creates a surface instead of a line.*

*TIP: You can use the **Make Plane** command to convert lines into planes so they can be shaded.*

Ortho Line

Menu: Draw / Lines / Ortho Line

Toolbox Icon:



Point 1: Beginning of line

Point 2: End of first segment

Point 3-n: End of second and subsequent segments (optional)

Draws lines with segments that are parallel to the X, Y, or Z axis. If you set points that are not parallel to one of these axes, the command automatically shifts the line to be parallel to the nearest matching axis.

Using the Command

Choose the **Ortho Line** command and set a beginning point for the ortho line. Set any number of points to draw the line. Then press Enter or double-click the mouse to end the command.

Example: Draw a line parallel to the X axis.

Select the **Ortho Line** command and set a point on the screen. Move the cursor to the right and up. Notice that the rubber-band line only moves along the X axis. Set a point for the body of the line and move the cursor up. Again the line is drawn exactly parallel to an axis. Set another point and press Enter to complete the command and add the line to the drawing.

Ortho Line-2

Menu: Draw / Lines / Ortho Line 2

Point 1: Beginning of the line

Point 2: Direction of 1st segment

Point 3: Length of 1st segment; length & direction of 2nd segment

Points 4-n: even number: direction of segment; odd number: previous segment length; current segment length & direction

Like the **Ortho Line** command, this command draws lines that are parallel to the X, Y, or Z axis. However, the points set in this command function differently.

With the **Ortho Line-2** command, the first point set is the beginning of the line. The second point determines only the *direction* of the first line segment, not the length. After the second point is set, the length of the first line segment continues to change when the cursor moves. The third point determines the direction of the second line segment and the length of both the first and second line segments. After the third point is set, all even numbered points function like the second point, determining only the direction of the line segment. All the following odd numbered points work like the third point and determine the length of the previous line segment and the length and direction of the current line segment.

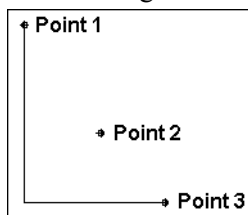
Using the command:

Choose the **Ortho Line-2** command. Set a beginning point for the line. Move the cursor in the desired direction for the first segment and set a second point. Notice how the rubber-band line is bent in the direction you specified, but the length of both segments continues to change when you move the cursor. When you have both segments the length you want them and the second segment the desired direction, set a third point. Set any number of points to draw the line. Then press Enter or double-click the mouse to end the command.

NOTE: *This command only works in 2D Mode. Also, Orthogonal Mode needs to be disabled.*

Example: Draw a bent line.

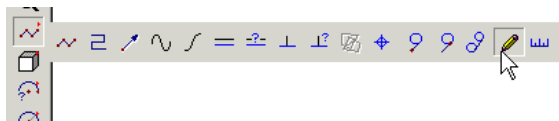
Choose the **Ortho Line-2** command. Set a point for the beginning of the line. Now, move the cursor down and to the right. Notice how the rubber-band line will only move straight along the X-Axis or the Y-Axis. Set a point so that the line goes down along the Y-Axis. Now, move the cursor up and down. Notice how the length of the first segment gets longer or shorter with the movement of the cursor. Move the cursor to the right and set a point so that the line forms an "L" shape. Press Enter to complete the command and add the line to the drawing.



Sketch

Menu: Draw / Lines / Sketch

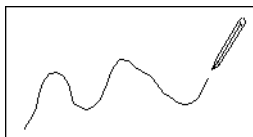
Toolbox Icon:



Draws freehand by following the cursor's movement.

Using the Command

Choose the **Sketch** command. Press and hold down the left mouse button. Move the cursor in a curve motion. Release the mouse button. A curve is drawn, following the cursor's movement.



NOTE: This command only works on the current workplane in 3D Mode. The default workplane is the XY plane. This plane can be changed in the **General Options** (see General Options).

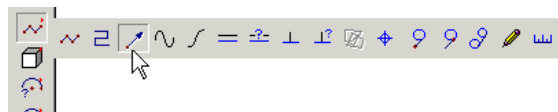
TIP: Two commands can be used to "clean up" a line drawn with the **Sketch** command. See Smooth Line by Point Reduction and Smooth Line by Slope Detection.

Arrow

Menu: Draw / Lines / Arrow

Shortcut Key: >

Toolbox Icon:



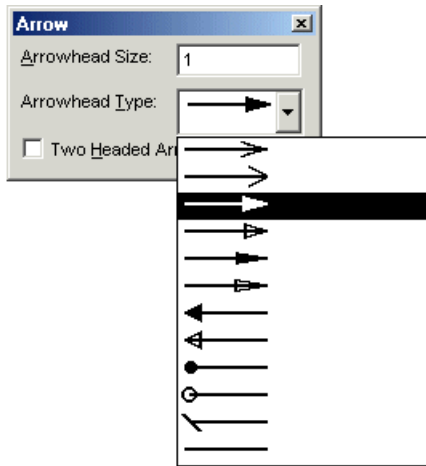
Point 1-n: Body of the arrow

Point n: Arrowhead

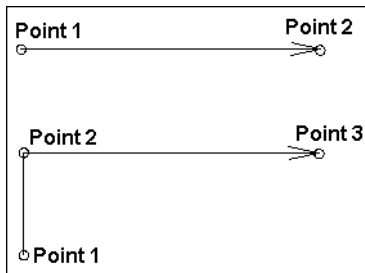
Draws an arrow using two or more points.

Using the Command

Use the options in the window to set the arrowhead size and type. A double-headed arrow can also be specified.



Set the first point at the tail and work toward the arrowhead.



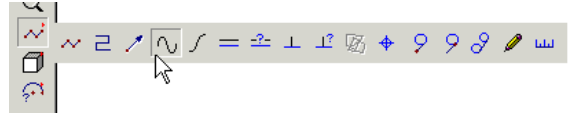
NOTE: The **Arrowhead Size** option is actually a ratio of the current text size.

Curve

Menu: Draw / Lines / Curve

Shortcut Key: C

Toolbox Icon:



Draws a cubic spline curve through three or more points. The points for a curve do not have to lie in the same plane.

The curve can be saved as a line by checking **Save as a Line**.

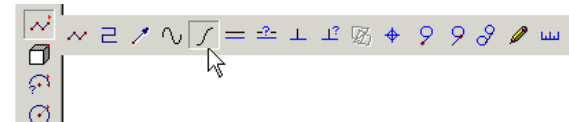


TIP: To make a "corner" in the curve, set two consecutive points in the same location. This will cause two separate curves to be drawn, one on each side of the corner.

Bezier Curve

Menu: Draw / Lines / Bezier Curve

Toolbox Icon:



Point 1: Beginning of the Bezier curve

Point 2: First control point for the curve (beginning tangent)

Point 3: Pass-through point

Point 4: Second control point (tangent for point 3)

Point 5: Pass-through point (optional)

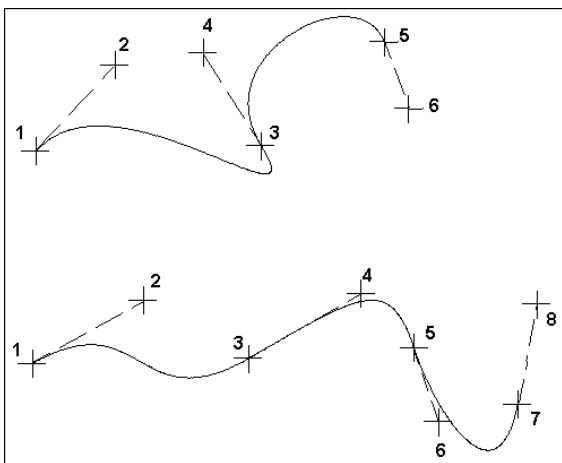
Point 6: Third control point (tangent for point 5)

Point 7-n: (Optional points in pairs)

Using the Command

The first point is the endpoint of the curve, and the second point is the control point. The curve at the first point will be tangent to the second point. The third point is the endpoint, and the fourth point is the tangent. Each subsequent pair of points is a point for the curve to pass through and a control point to define the tangent at that location.

Moving a control point further from the curve makes the curve sharper. Points must be set in pairs. If you set an odd number of points, the last point is ignored.

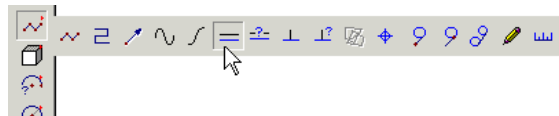


Parallel

Menu: Draw / Lines / Parallel

Shortcut Key: =

Toolbox Icon:



Point 1: Existing line

Point 2: Location for parallel line

Draws a line parallel to any existing line, curve, or arc.

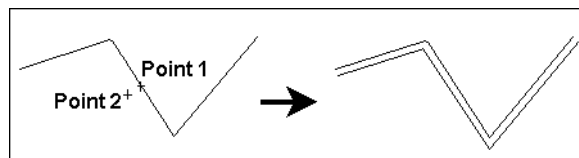
Using the Command

Choose the **Parallel** command. Set a point on the original line and a point for the location of the parallel line.

This command works on 3D lines as well as lines that lie on a single plane.

Example: Draw a line parallel to a multi-segment line.

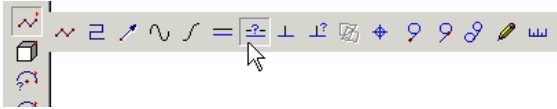
Select the **Parallel** command. Set a point on the line and move the cursor away from it. A rubber-band line is drawn parallel to the first line at the current cursor distance. When the line is where you want it, set the second point.



Parallel by Distance

Menu: Draw / Lines / Parallel by Distance

Toolbox Icon:



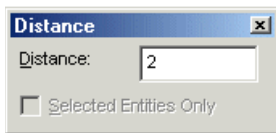
Point 1: Line to be offset

Point 2: Direction of parallel line from original line.

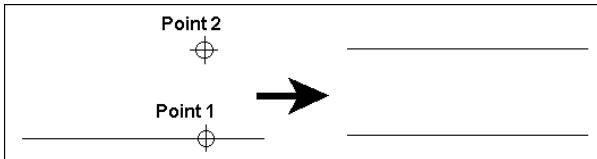
Draws a parallel line a specified distance from any other line or curve. The distance of the parallel is set in the window. The direction of the parallel from the original line is established by setting a point on the screen.

Using the Command

Choose the **Parallel by Distance** command and enter how far you want the parallel from the original line in the **Distance** box in the window.



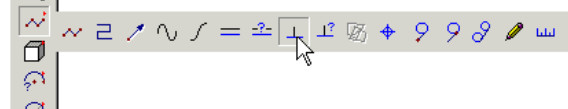
Set a point on the original line. A rubber-band line shows how the parallel line will look. Next, move your cursor to the side of the original line on which you want the parallel to be drawn and set a point. A parallel line will be drawn at the specified distance.



Perpendicular to a Line

Menu: Draw / Lines / Perpendicular to a Line

Toolbox Icon:



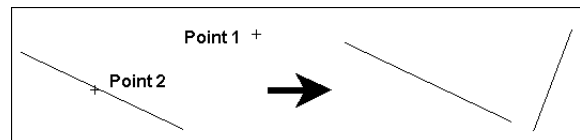
Point 1: End point of perpendicular line

Point 2: Point on line to which new line is drawn perpendicular

Draws a line perpendicular from a point to an existing line.

Using the Command

Choose the **Perpendicular to a Line** command. To restrict the command in a complex drawing so that only selected lines can be used, click the **Selected Entities Only** option. Set one point for the location and starting point of the perpendicular line and a second point on the line to which the perpendicular line will be drawn.

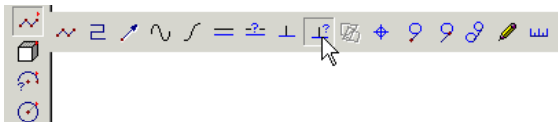


NOTE: This command is similar to the **Perpendicular From a Line** command, but this command uses the point away from the line for the perpendicular location. **Perpendicular From a Line** uses the point on the existing line for the perpendicular location.

Perpendicular from a Line

Menu: Draw / Lines / Perpendicular from a Line

Toolbox Icon:



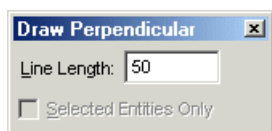
Point 1: Reference point

Point 2: Direction of perpendicular

Draws a line perpendicular to an existing line. The perpendicular line is drawn from a point on the existing line in the direction of the second point set. You can enter the length for the perpendicular line in the window.

Using the Command

Choose the **Perpendicular from a Line** command. Enter the **Length** for the perpendicular line in the window.

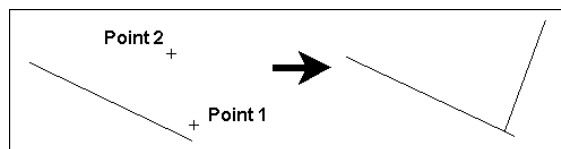


Set a reference point on or near the line from which you want the perpendicular line drawn. A rubber-band line appears. Then set a second point away from the first line. A perpendicular line is drawn from the existing line through the first point in the direction of the second point.

NOTE: *This command is similar to the **Perpendicular to a Line** command, but this command uses the point on or near the existing line for the perpendicular location. **Perpendicular to a Line** uses the point away from the line as the reference for the perpendicular location.*

Example: Draw a 20-unit line perpendicular from a certain point on another line in your drawing.

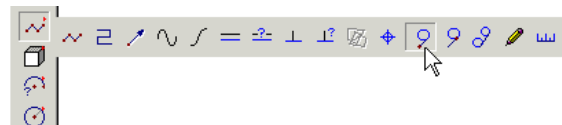
Select the **Perpendicular from Line** command. Enter 20 for the **Length** in the window. Set a point on or near the line, at the point through which you want the perpendicular line drawn. A rubber-band line, 20 Drawing Units in length, will be drawn from the line and through Point 1. If you move the cursor to the other side of the line, the perpendicular line will flip to the other side as well. Set the second point when the line extends in the desired direction.



Tangent to a Circle

Menu: Draw / Lines / Tangent to a Circle

Toolbox Icon:



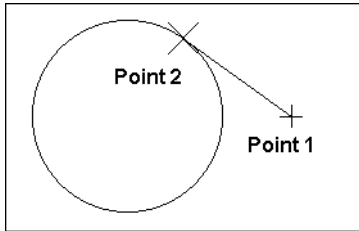
Point 1: Endpoint for the tangent line

Point 2: Point near the circle

Draws a line tangent from a point to a circle, ellipse, arc, or elliptical arc. There are two possible lines that can be drawn tangent to a circle from a single point. The line drawn is the one closest to the second point.

Using the Command

Choose the **Tangent to a Circle** command. Set a point for the beginning of the line. After the first point is set, a rubber-band line shows how the line will be drawn. Set a point on or near the circle or arc. A line will be drawn from the first point tangent to the circle or arc.

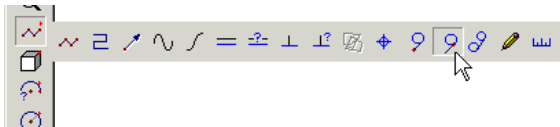


NOTE: If you are not in 2D Mode, the line and circle must lie in the same plane.

Tangent from a Circle

Menu: Draw / Lines/ Tangent from a Circle

Toolbox Icon:



Point 1: Point on or near circle

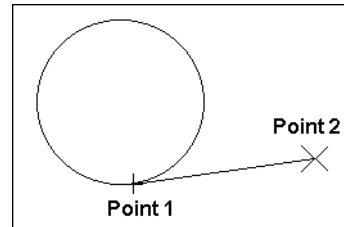
Point 2: Endpoint for the tangent line

Draws a line tangent from a point on a circle, ellipse, arc, or elliptical arc to another point.

Using the Command

Choose the **Tangent from a Circle** command. Set a point on or near the circle or arc. After the first point is set, a rubber-band line shows how the tangent will be drawn. Set a point for the end of the line on or near the line tangent to the circle or

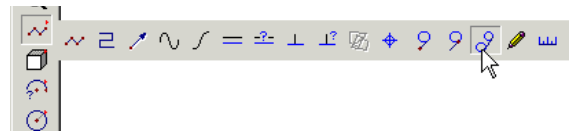
arc. The endpoint of the line will be even with the second point. A line will be drawn from the first point to the second point, tangent to the circle or arc.



Tangent between Circles

Menu: Draw / Lines / Tangent between Circles

Toolbox Icon:



Point 1: Point on or near the first circle.

Point 2: Point on or near the second circle

Draws a line tangent to two circles, two arcs, or an arc and a circle. There are four possible lines that can be drawn tangent to any two circles. The command draws a line closest to the two points that you set in the command.

Using the Command

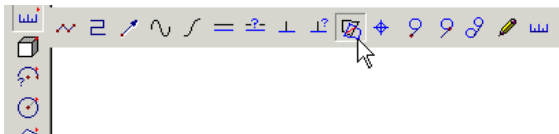
Choose the **Tangent between Circles** command. Set a point on or near the first circle or arc. After the first point is set, a rubber-band line shows how the line will be drawn. Set a point on or near the second circle or arc. A line will be drawn tangent to the two circles or arcs.

NOTE: The **Tangent** commands do not work on circles that have been saved in vector form. These entities are not true circles. They are line entities approximating circles. Also, if you are not in 2D Mode, the two lines must lie in the same plane.

Surface Intersection ^{3D}only

Menu: Draw / Lines / Surface Intersection

Toolbox Icon:



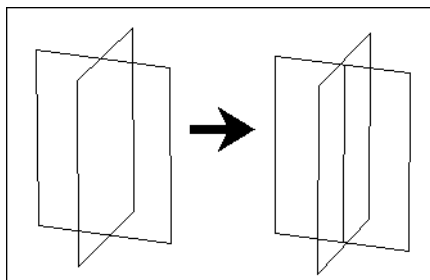
Point 1: Point on first surface or plane

Point 2: Point on second surface or plane

Draws a line at the intersection of two surfaces. This command can be used on grid surfaces as well as planes.

Using the Command

Choose the **Surface Intersection** command. Set a point on each of the intersecting surfaces. A line is drawn along the intersection.



Contour Line ^{3D}only

Menu: Draw / Lines / Contour line

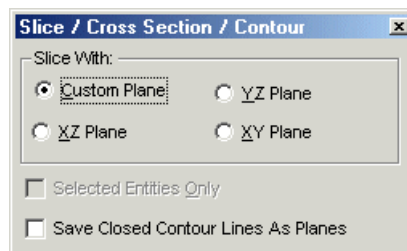
Points 1-3: Points to define the contour plane

Draws a line through all locations where solids or grid surfaces intersect a specified YZ, XZ, XY, or custom plane.

Using the Command

Make sure that the current drawing contains at least one solid or grid. (Grid surfaces are created with the **Surface Connect** and **Surface Patch** commands.)

Select the **Contour Line** command. The following window appears.



If **Custom Plane** is selected, set three points to define the "contour plane." A contour line is drawn through all locations where solids and/or grid surfaces intersect the custom plane.

If **YZ Plane**, **XZ Plane**, or **XY Plane** is selected, set a single point to define the location of the "contour plane." The contour plane incorporates the point set and lies parallel to the plane indicated in the selected option. A contour line is drawn through all locations where solids and/or grid surfaces intersect the contour plane.

You can choose to run the **Contour Line** command on the current selection only by activating that option in the window.

To save any resulting closed, contour lines as planes, enable the **Save Closed Contour Lines as Planes** option.

Construction Lines

Construction lines are used to aid in drawing. They are temporary lines that can be removed later.

Construction Line

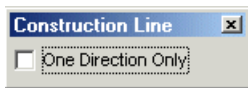
Menu: Tools / Construction Line

Point 1-2: Points through which the construction line passes.

Draws a 2D or 3D construction line.

Using the Command

Choose the **Construction Line** command. If you want the construction line to extend in only one direction, make sure the **One Direction Only** option is enabled.



Set one point for the beginning of the construction line. Set a second point through which the construction line passes. The construction line is drawn.

The command will remain active, so additional construction lines may be drawn. To end the command, press Enter.

Show Construction Lines

Menu: View / Show Construction Lines

Make lines drawn using the **Construction Line** command visible or invisible (toggle function).

Arcs

This section covers methods of creating arcs (partial circles) and partial ellipses.

Arc

Menu: Draw / Arcs / Arc

Shortcut Key: A

Toolbox Icon:



Point 1: Center of the arc

Point 2: Start of the arc

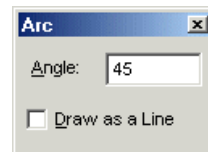
Point 3: Orientation of the arc (optional)

Draws an arc which spans a given angle.

Using the Command

Choose the **Arc** command. Enter the arc angle in the **Angle** field in the window. Finally use the mouse to specify a center point, a starting point, and an optional third point to orient the arc in 3D space.

There are two choices for the format of the arc, which you can set in the window:



If you select **Draw as a Line**, the arc will be drawn as line segments that follow the shape of an arc. Otherwise the arc will be stored as an actual arc. You should normally use the arc format

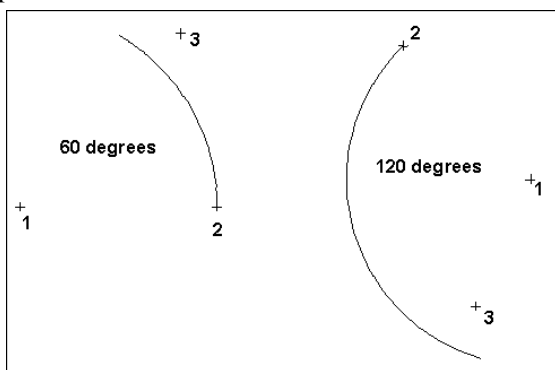
because it is more efficient and precise. You can use the line format if you need to distort the arc by scaling or stretching.

Example 1: Draw an arc in the XY plane with a radius of 10 and an angle of 60 degrees.

Change the view setting to the **Front** view. Choose the **Arc** command and enter 60 degrees in the **Angle** field. Set a point anywhere on the screen for the arc's center. Choose the **Point Relative** command from the **Point** menu or press the ' (single quote) key, and enter 10 for DX. Set a third point anywhere on the screen with the mouse or arrow keys. Your arc should look like the one in the illustration below.

Example 2: Create a 120-degree arc similar to the one in the figure.

Set the angle as described above, and place the points as shown in the illustration below.



Example 3: Create a third arc at an angle of 60 degrees which is parallel to the XZ plane.

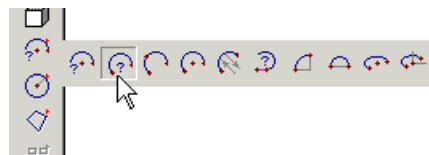
Set the angle as above. Set the center point, then a point for the radius. For the third point press ' (single quote) key for the **Point Relative** command, and set DZ to 10. In the **Front** view this arc appears to be a straight line. You will have

to look a different view to see it as an arc. In the **Top** view, this arc should look like the 60-degree arc in the illustration above.

Arc (Radius, Begin-End)

Menu: Draw / Arcs / Arc (Radius, Begin-End)

Toolbox Icon:



Point 1: Start of arc

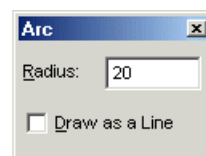
Point 2: End of arc

Point 3: Orientation (3D Mode only)

Draws an arc of predetermined radius by specifying the beginning and ending points of the arc and setting an optional third point to orient the arc in 3D space.

Using the Command

There are two choices for the format of the arc, which you can set in the window:

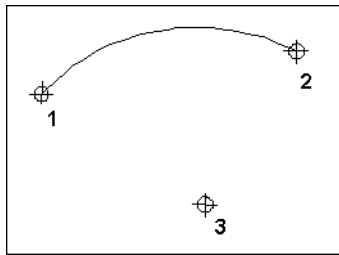


If you select **Draw as a Line**, the arc will be drawn as line segments that follow the shape of an arc. Otherwise the arc will be stored as an actual arc. The arc format stores a center point, endpoint, and orientation point in the drawing. The line format stores a series of points for the line approximating the arc.

This command automatically creates an arc with a span of less than 180 degrees, the smallest possible arc based on the supplied radius and the first two points set.

Example: Create an arc with a radius of 20.

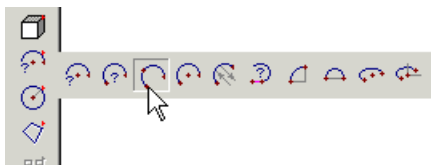
Select the **Arc (Radius, Begin-End)** command. Enter the **Radius** as 20 in the window. Set a point somewhere around the center of the screen. Move the cursor up and to the right a couple of inches and set a second point. Now, move the cursor between the two points. Notice that DesignCAD forms a rubber-band arc. Set a third point to determine the orientation of the arc. The arc is inserted into the drawing.



Arc (3-Point)

Menu: Draw / Arcs / Arc (3-Point)

Toolbox Icon:



Point 1: Beginning of the arc

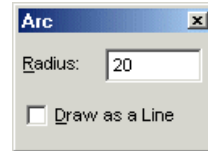
Point 2: A point along the arc

Point 3: Endpoint of the arc

Draws an arc using a specified starting point, a second point through which the arc passes, and the endpoint of the arc.

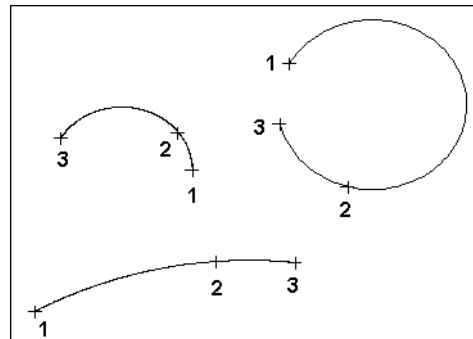
Using the Command

There are two choices for the format of the arc, which you can set in the window:



If you select **Draw as a Line**, the arc will be drawn as line segments that follow the shape of an arc. Otherwise the arc will be stored as an actual arc. The arc format stores a center point, endpoint, and orientation point in the drawing. The line format stores a series of points for the line approximating the arc.

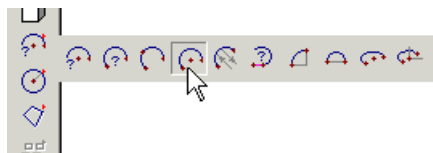
You can draw arcs with this command in either a decreasing or increasing angular direction. This means that the arc can be drawn in either direction from its starting point.



Arc (Center, Begin, End)

Menu: Draw / Arcs / Arc (Center, Begin, End)

Toolbox Icon:



Point 1: Center of arc

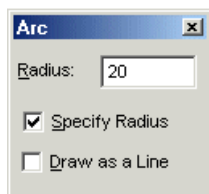
Point 2: Beginning of arc

Point 3: End of arc

Draws an arc using the center, beginning, and end points of the arc.

Using the Command

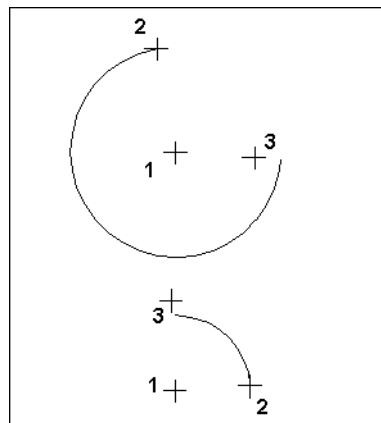
There are two choices for the format of the arc, which you can set in the window:



If you select **Draw as a Line**, the arc will be drawn as line segments that follow the shape of an arc. Otherwise the arc will be stored as an actual arc. If you **Specify Radius**, the radius is independent of the distance between the first and second points.

The arc format stores a center point, endpoint, and orientation point in the drawing. The line format stores a series of points for the line approximating the arc.

The first point you set becomes the center of the arc. The second point sets both the radius and the start angle of the arc. The third point sets the ending angle of the arc, and will not necessarily lie on the endpoint of the actual arc that is drawn.

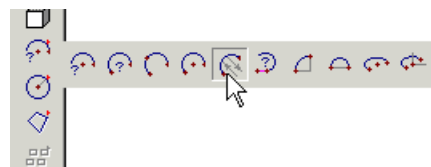


NOTE: Arcs drawn with this command are created in such a way that the arc can only span a positive angle from its starting point, so the arc will be drawn counterclockwise from point 2 to the span angle on the arc determined by point 3.

Arc (Endpoints, Center)

Menu: Draw / Arcs / Arc (Endpoints, Center)

Toolbox Icon:



Point 1: Beginning of the arc

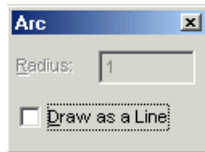
Point 2: End of the arc

Point 3: Center (radius) of the arc

Draws an arc using points set for the beginning, end, and radius of the arc. This command is similar to the **Arc (Radius, Begin-End)** command except the radius is determined by the third point rather than by entering a radius in the window.

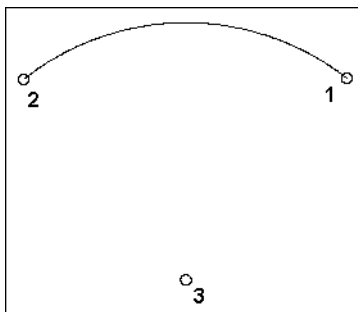
Using the Command

There are two choices for the format of the arc, which you can set in the window:



If you select **Draw as a Line**, the arc will be drawn as line segments that follow the shape of an arc. Otherwise the arc will be stored as an actual arc. The arc format stores a center point, endpoint, and orientation point in the drawing. The line format stores a series of points for the line approximating the arc.

The first point determines the beginning of the arc. The second point defines the end of the arc. After the second point is set, a rubber-band arc shows how the arc will be drawn. Set a third point for the center of the arc. An arc is then drawn counterclockwise from the first point to the second point.



Tangent Arc

Menu: Draw / Arcs / Tangent Arc

Toolbox Icon:



Point 1: First point of line tangent to arc

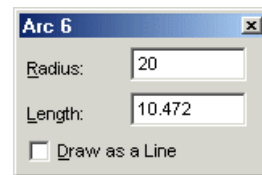
Point 2: Second point of line tangent to arc

Point 3: Direction in which the arc is to be drawn

Draws an arc with a specified radius by defining a line tangent for one end of the arc.

Using the Command

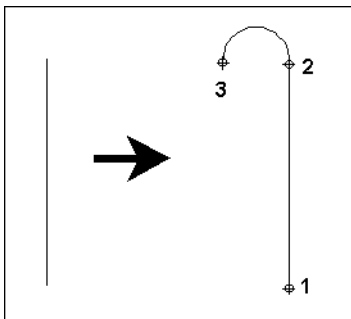
Select the **Tangent Arc** command and enter the **Radius** in the window. Enter the length for the arc in the **Length** box.



NOTE: *The length of the arc is measured along the arc and is a linear measurement. It is not an angular or chord measurement.*

Set the first point for the imaginary line to which the arc is to be drawn tangent. Set the second and final point for the line; this point will also serve as the starting point of the arc. (These first two points may be set on the endpoints of an existing line - Use the **Gravity** command for this.) After the second point is set, a rubber-band arc is drawn

and shows how the arc will be drawn. Move the mouse to determine the orientation of the arc. Set the endpoint for the arc.

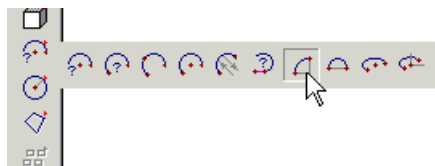


Quarter Circle

Menu: Draw / Arcs / Quarter Circle

Shortcut Key: (

Toolbox Icon:



Point 1: Beginning of the quarter circle.

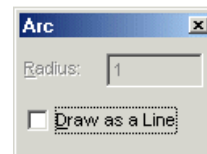
Point 2: Endpoint for the quarter circle.

Point 3: Orientation of the quarter circle (optional, 3D Mode only)

Draws a quarter circle, or 90 degree arc, between two points.

Using the Command

There are two choices for the format of the arc, which you can set in the window:



If you select **Draw as a Line**, the arc will be drawn as line segments that follow the shape of an arc. Otherwise the arc will be stored as an actual arc. You should normally use the arc format because it is more efficient and precise. You can use the line format if you need to distort the arc by scaling or stretching.

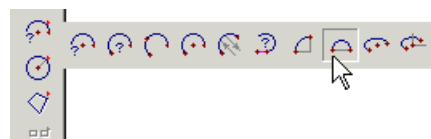
Set a point for the beginning of the quarter circle. After the first point is set, a rubber-band circle shows how the quarter circle will be drawn. Set a point for the end of the quarter circle. Set a third point to determine the orientation of the quarter circle or press Enter to draw the quarter circle using the current orientation. A quarter circle, or 90 degree arc, will be drawn counterclockwise from the first point to the second point.

Semi Circle

Menu: Draw / Arcs / Semi Circle

Shortcut Key:)

Toolbox Icon:



Point 1: Beginning of the semicircle

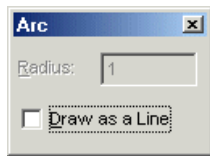
Point 2: Endpoint of the semicircle

Point 3: Orientation of the semicircle (optional, 3D Mode only)

Draws a semicircle, or 180 degree arc, between two points.

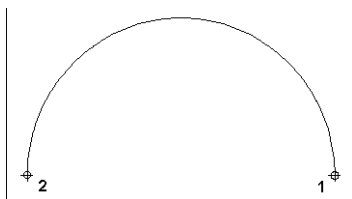
Using the Command

Choose the **Semi Circle** command. There are two choices for the format of the arc, which you can set in the window:



If you select **Draw as a Line**, the arc will be drawn as line segments that follow the shape of an arc. Otherwise the arc will be stored as an actual arc. You should normally use the arc format because it is more efficient and precise. You can use the line format if you need to distort the arc by scaling or stretching.

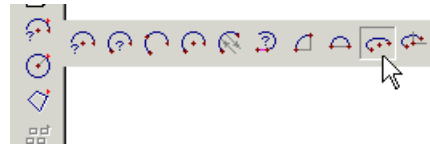
Set a point for the beginning of the semicircle. After the first point is set, a rubber-band semicircle shows how the semicircle will be drawn. Set a point for the end of the semicircle. Set a third point to determine the orientation of the semicircle or press the Enter key to draw the semicircle using the current orientation. A semicircle will be drawn counterclockwise from the first point to the second point.



Elliptical Arc

Menu: Draw / Arcs / Elliptical Arc

Toolbox Icon:



Point 1: Center of the ellipse

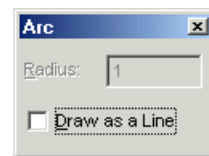
Point 2: Beginning of the arc

Point 3: End of the arc

Draws an elliptical arc, or part of an ellipse or oval, between two points.

Using the Command

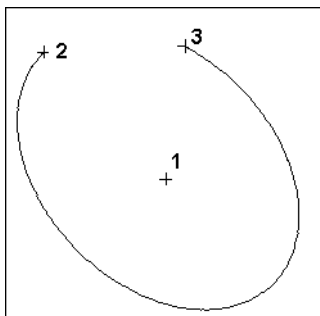
Choose the **Elliptical Arc** command. There are two choices for the format of the arc, which you can set in the window:



If you select **Draw as a Line**, the arc will be drawn as line segments that follow the shape of an arc. Otherwise the arc will be stored as an actual arc.

Set a point for the center of the arc and then set a point for the beginning of the arc. After the second point is set, a rubber-band arc shows how the arc will be drawn. Set a point for the end of the

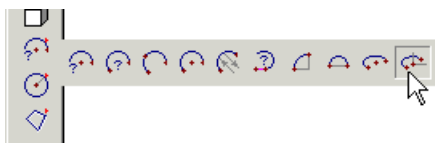
arc. An elliptical arc is drawn from the second point, counterclockwise around the ellipse to the third point.



Elliptical Arc (Center, Axes, Angles)

Menu: Draw / Arcs / Elliptical Arc (Center, Axes, Angles)

Toolbox Icon:



Point 1: Center of the elliptical arc

Point 2: Major axis

Point 3: Minor axis

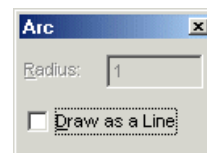
Point 4: Beginning of the arc

Point 5: End of the arc

Draws an elliptical arc, or part of an ellipse or oval, using a point for the major axis, a point for the minor axis, a point for the beginning of the arc, and a point for the end of the arc.

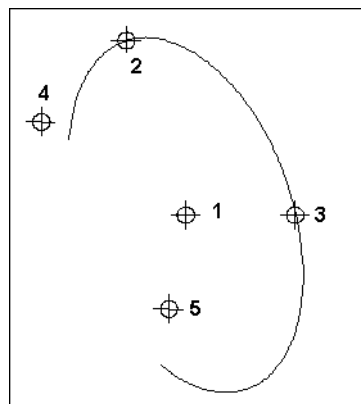
Using the Command

Choose the **Elliptical Arc (Center, Axes, Angles)** command. There are two choices for the format of the arc, which you can set in the window:



If you select **Draw as a Line**, the arc will be drawn as line segments that follow the shape of an arc. Otherwise the arc will be stored as an actual arc.

Set a point for the center of the arc. Set a point for the major axis and a point for the minor axis. Set a point for the beginning of the arc. After the fourth point is set, a rubber-band arc shows how the arc will be drawn. Set a point for the end of the arc. The elliptical arc is drawn.



Circles

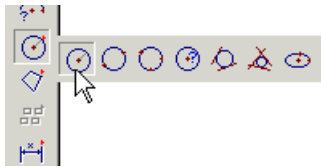
This section covers methods for creating full circles and ellipses. For partial circles and ellipses, see Arcs.

Circle (Center, Outside)

Menu: Draw / Circles / Circle (Center, Outside)

Shortcut Key: O (the letter, not the number)

Toolbox Icon:



Point 1: Center of the circle

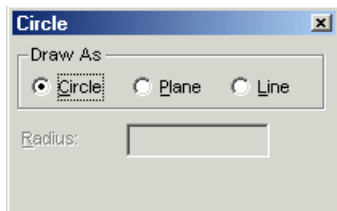
Point 2: Point on the circle

Point 3: Orientation of the circle (optional)

Draws a circle based on a point at the center and a point on the outside of the circle. A third point can be used to specify the plane on which the circle lies.

Using the Command

Select the **Circle (Center, Outside)** command. The window displays three choices for how you want to save the circle:



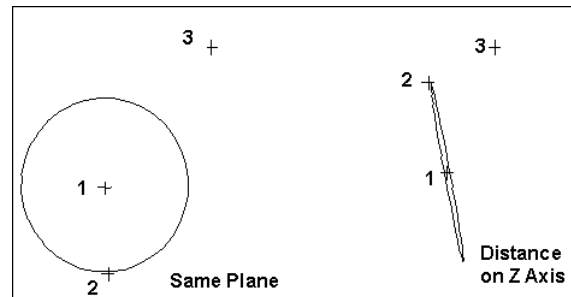
Circle: Stored as an actual circle in the drawing.

Plane: Stored as a circular plane with 120 sides; plane circles can be shaded.

Line: Stored as a line entity.

Normally, you'll want to save the circle in the **Circle** form. If you want to use the circle as you would a solid, select **Plane**. Save the circle as a **Line** if you want to be able to scale the circle or treat it as a line entity.

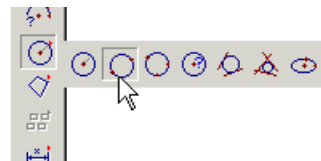
Set a point near the center of the screen. Move the cursor away from the first point in any direction. Notice that a rubber-band representation of the circle is drawn using the cursor location as a point on the circle. Set a point when the circle is of the desired radius. Now, hold down Ctrl+Shift while moving the mouse forward or backward. This moves the cursor away from or toward you on the Z axis respectively. As you do this, the circle will pivot on the hinge set by the first two points. When the circle lies in the plane that you want it to, set the third point.



Circle (Diameter)

Menu: Draw / Circles / Circle (Diameter)

Toolbox Icon:



Point 1: A point on the diameter of the circle

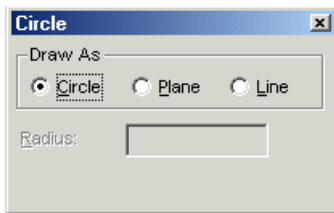
Point 2: A point for the opposite end of the diameter

Point 3: A point to align the circle in 3D space (optional)

Draws a circle between two points, so that the two points lie on the diameter of the circle.

Using the Command

Select the **Circle (Diameter)** command. The window displays three choices for how you want to save the circle:



Circle: Stored as an actual circle in the drawing.

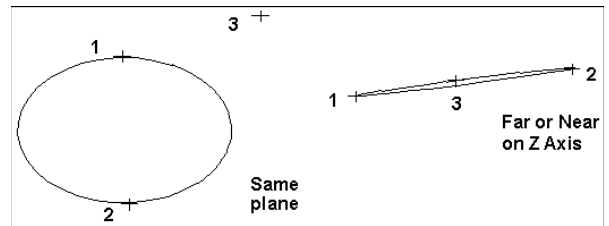
Plane: Stored as a circular plane with 120 sides; plane circles can be shaded.

Line: Stored as a line entity.

Normally, you'll want to save the circle in the **Circle** form. If you want to use the circle as you would a solid, select **Plane**. Save the circle as a **Line** if you want to be able to scale the circle or treat it as a line entity.

Set two points for the circle diameter. If necessary, you can set a third point to define the plane on which the circle lies: hold down Ctrl+Shift while moving the mouse forward or backward and notice how the circle swings on a hinge set by the first two points. When the circle

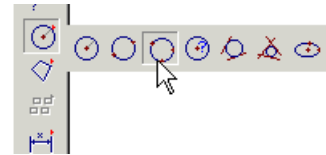
lies in the plane you want it to, set the third point. If you just want to set two points, press Enter after you have set them.



Circle (3-Point)

Menu: Draw / Circles / Circle (3-Point)

Toolbox Icon:



Point 1: Any point on the circle

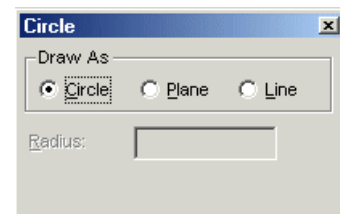
Point 2: A second point on the circle

Point 3: A third point on the circle

Draws a circle that passes through three points. The three points must not lie in a straight line.

Using the Command

Select the **Circle (3-Point)** command. The window displays three choices for how you want to save the circle:



Circle: Stored as an actual circle in the drawing.

Plane: Stored as a circular plane with 120 sides; plane circles can be shaded.

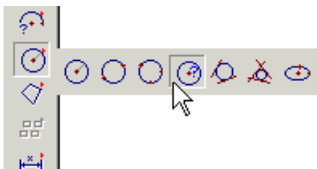
Line: Stored as a line entity. This makes it possible to scale the circle or treat it as a line entity with other commands.

Set a point on the screen through which the circle will pass. Move the cursor away from the first point in any direction and set a second point. Now, as you move the cursor, you see a rubber-band circle, showing you how the circle would look if you set the final point at the cursor location. To draw the circle in a different plane, move the cursor along the Z axis by holding down Ctrl+Shift and moving the mouse forward or backward. This will move the cursor out or in along the Z axis respectively. The circle will pivot and change size according to the location of the third point on the Z axis. When the circle lies in the desired plane and is the correct size, set the third point.

Circle (Center-Radius)

Menu: Draw / Circles / Circle (Center-Radius)

Toolbox Icon:



Point 1: Center of the circle

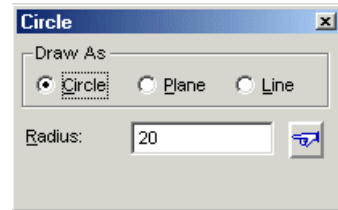
Point 2: Orientation of the circle (optional)

Point 3: Orientation of the circle (optional)

Draws a circle of a specified radius with a point set for the center. Two more points can be used to orient the circle in 3D space.

Using the Command

Select the **Circle (Center-Radius)** command. The window displays three choices for how you want to save the circle:



Circle: Stored as an actual circle in the drawing.

Plane: Stored as a circular plane with 120 sides; plane circles can be shaded.

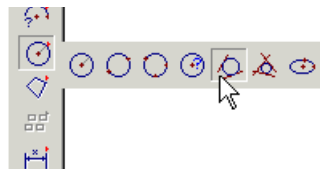
Line: Stored as a line entity. This makes it possible to scale the circle or treat it as a line entity with other commands.

Enter the **Radius** of the circle in the window. To use the same radius as another circle, click the **Same As** icon. A rubber-band circle shows how the circle will be drawn. Set a point for the center of the circle. Press Enter or set a second point to orient the circle. Press Enter or set a third point to tilt the circle using the first two points as a hinge.

Circle Tangent to 2 Lines

Menu: Draw / Circles / Circle Tangent to 2 Lines

Toolbox Icon:



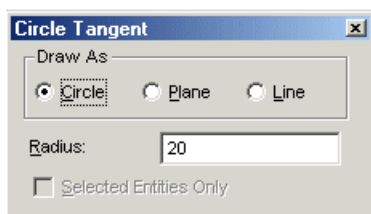
Point 1: First line tangent to circle

Point 2: Second line tangent to circle

Draws a circle of a specified radius which is tangent to two lines. The two lines must lie in the same plane unless you are currently in 2D Mode.

Using the Command

Select the **Circle Tangent to 2 Lines** command. The window displays three choices for how you want to save the circle:

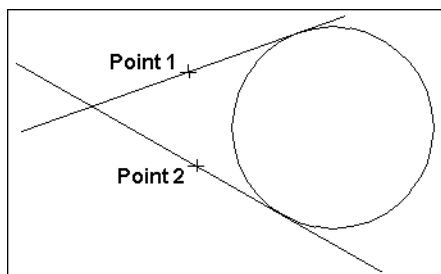


Circle: Stored as an actual circle in the drawing.

Plane: Stored as a circular plane with 120 sides; plane circles can be shaded.

Line: Stored as a line entity. This makes it possible to scale the circle or treat it as a line entity with other commands.

Enter the **Radius** of the circle in the window. Set a point on each of the two lines which are to be tangent to the circle. A circle with the specified radius is drawn tangent to the two lines.

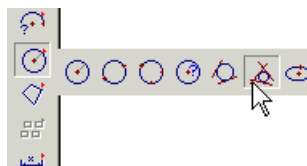


NOTE: If the two lines do not lie in the same plane, and the circle is drawn tangent to the two lines while in 2D Mode, the circle is drawn at the Z coordinate the cursor was on before the switch to 2D Mode. This will be apparent when you switch back to 3D Mode.

Circle Tangent to 3 Lines

Menu: Draw / Circles/ Circle Tangent to 3 Lines

Toolbox Icon:



Point 1: First line tangent to circle

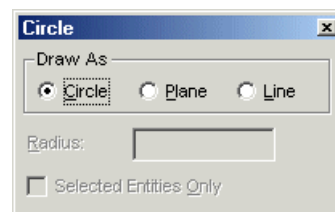
Point 2: Second line tangent to circle

Point 3: Third line tangent to circle

Draws a circle which is tangent to three lines. The three lines must lie in the same plane unless you are currently in 2D Mode.

Using the Command

Select the **Circle Tangent to 3 Lines** command. The window displays three choices for how you want to save the circle:

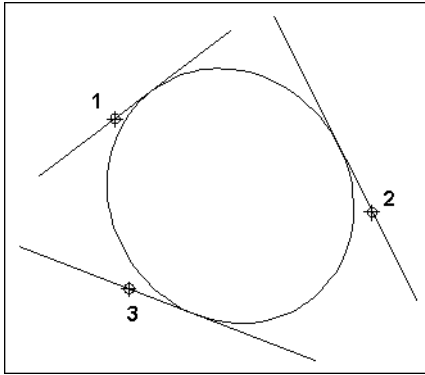


Circle: Stored as an actual circle in the drawing.

Plane: Stored as a circular plane with 120 sides; plane circles can be shaded.

Line: Stored as a line entity. This makes it possible to scale the circle or treat it as a line entity with other commands.

Set a point on the first line. Set a point on the second line. After the second point is set, a rubber-band circle shows how the circle will be drawn. Set a point on the third line. A circle is drawn tangent to the three lines.

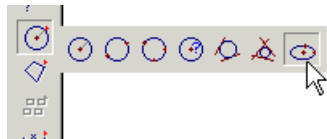


NOTE: If the three lines do not lie in the same plane, and the circle is drawn tangent to the three lines while in 2D Mode, the circle is drawn at the Z coordinate the cursor was on before the switch to 2D Mode. This will be apparent when you switch back to 3D Mode.

Ellipse

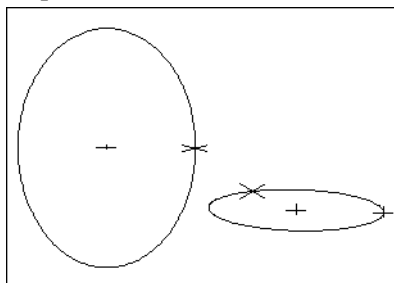
Menu: Draw / Circles / Ellipse

Toolbox Icon:



Point 1: Center of the ellipse

Point 2: One axis of the ellipse

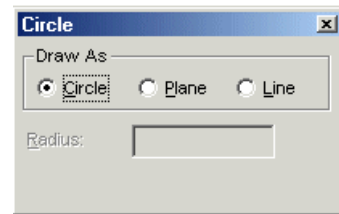


Point 3: Point through which the ellipse passes

Draws an ellipse, or oval. Points are set for the center and axes.

Using the Command

Choose the **Ellipse** command. The window displays three choices for how you want to save the ellipse:



Circle: Stored as an actual circle in the drawing.

Plane: Stored as a circular plane with 120 sides; plane circles can be shaded.

Line: Stored as a line entity. This makes it possible to scale the circle or treat it as a line entity with other commands.

Set a point for the center of the ellipse. After the first point is set, a rubber-band ellipse shows how the ellipse will be drawn. Set a point for one axis of the ellipse. Set a point through which the ellipse passes. An ellipse will be drawn passing through the second and third points with its center at the first point.

Planes

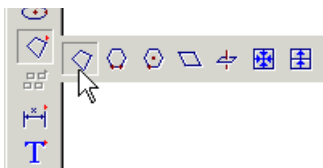
2D planes are closed boundaries that act as surfaces - they can be shaded like a 3D object. This section covers 2D planes, which are flat. (For 3D surfaces, see Surfaces.)

Plane 3D only

Menu: Draw / Planes / Plane

Shortcut Key: P

Toolbox Icon:



Points 1-2: Starting point and edge of the plane

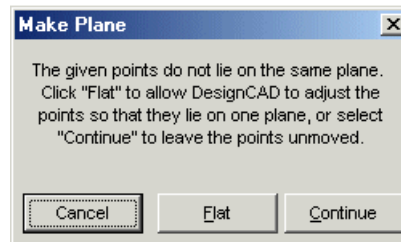
Points 3-n: Points around the perimeter of the plane

Draws a plane or flat surface by setting points around the perimeter of the surface. The points of a plane must lie on a single plane in 3D space. You cannot, for example, have a plane entity with a bump in it. (You can use surface commands such as **Surface Connect** and **Extrude** for that.)

Using the Command

Choose the **Plane** command. Set a starting point for the plane. Set another point for one edge of the plane, and then set at least one more unique point (i.e., not on Point 1 or Point 2) to set the plane. When you have set all the points you want, press Enter to end the command.

DesignCAD checks to see if the points form a valid plane. If they do not, you will have the option to allow DesignCAD to adjust the points so that they form a plane.

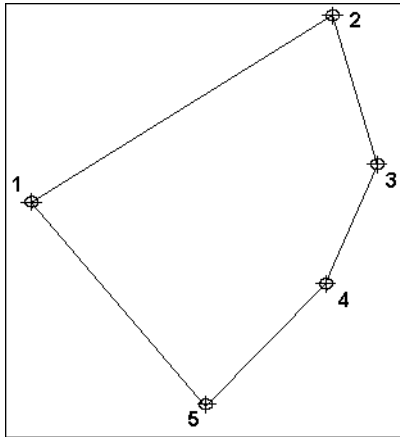


NOTE: *Be careful not to confuse the **Plane** command with the **Line** command. Plane entities have surface area, like a piece of paper, and can be shaded. Line entities have no surface area, like a piece of thin wire, and they cannot be shaded.*

Example: Create a plane along five specific points in your drawing.

Select the **Plane** command. Set the first point. After the first point is set, a rubber-band drawing of the plane appears, showing how the plane will

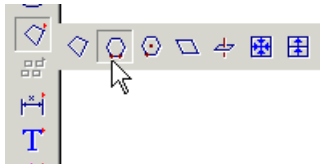
look when subsequent points are added. Press Enter after all the points have been set and a plane is drawn on the screen.



Polygon (Edge)

Menu: Draw / Planes / Polygon (Edge)

Toolbox Icon:



Point 1: Start of one side of the polygon

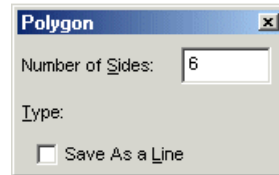
Point 2: End of one side of the polygon

Point 3: Orientation of the polygon (3D Mode only)

Draws a regular polygon - a convex shape in which each side is the same length. The number of sides can be entered in the window.

Using the Command

Choose the **Polygon (Edge)** command. Enter the number of sides for the polygon in the **Number of Sides** box.



Set two points to designate one side of the polygon. If the drawing is in 3D Mode, a third point can be used to determine the plane on which the polygon lies.

If you select the **Save as a Line** option, the polygon will be saved as a line instead of a plane entity.

This command is similar to the **Polygon (Center-Vertex)** command, but with this command you set two points for one edge of the regular polygon. With the **Polygon (Center-Vertex)** command, you set a point at the center and a point at a vertex or midpoint of a side of the polygon.

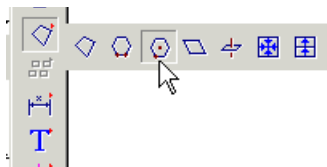
Example: Draw a polygon in your drawing.

Select the **Polygon (Edge)** command. Set a point for the beginning of one of the sides. Then move the cursor away from Point 1. A rubber-band polygon will be drawn using the cursor position as Point 2. When the polygon is the desired size, set Point 2. Hold down Ctrl while moving the mouse up or down and the polygon will swing on the axis of the first two points. When the polygon lies in the desired plane, set the last point.

Polygon (Center-Vertex)

Menu: Draw / Planes / Polygon (Center - Vertex)

Toolbox Icon:



Point 1: Center of the surface

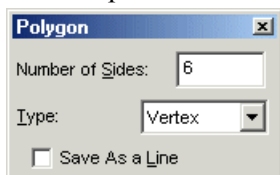
Point 2: A corner of the surface

Point 3: An orientation point to align the surface in space (3D Mode only)

Draws a regular polygon - a convex shape in which each side is the same length.

Using the Command

Choose the **Polygon (Center-Vertex)** command. Enter the number of sides for the polygon in the **Number of Sides** box. In the **Type** box you can choose to draw the polygon either from center to a vertex or to the midpoint on one of the sides.



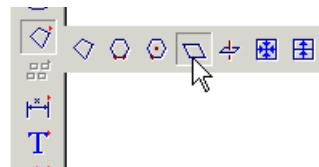
Set a point for the center of the polygon. Set a second point to designate a vertex or the midpoint of a side of the polygon. A third point can be used to determine the plane on which the polygon lies.

If you select **Save as a Line**, the polygon will be saved as a line entity instead of a plane.

Parallelogram

Menu: Draw / Planes / Parallelogram

Toolbox Icon:



Point 1: First point for the baseline

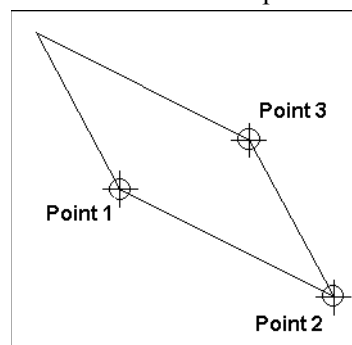
Point 2: Second point for the baseline

Point 3: Upper-right corner

Draws a parallelogram defined by three points as described above.

Using the Command

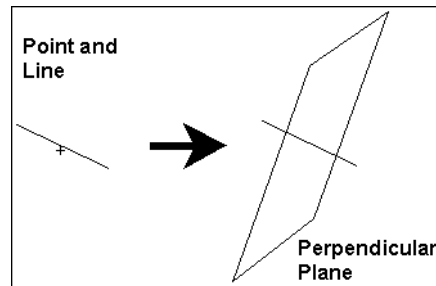
Choose the **Parallelogram** command. Set the first point for the baseline of the parallelogram. Set a point for the opposite end of the baseline. After the second point is set, a rubber-band parallelogram shows how the parallelogram will be drawn. Set a point for the upper-right corner of the parallelogram. A parallelogram is drawn with three of its corners at the three points.



Example: Draw a parallelogram.

Select the **Parallelogram** command. Set a point for the parallelogram's lower-left corner. Next, select the **Point Relative** command (' key). In the **Point Relative** box, enter 6 in the DX box and -3 in the DY box. Press Enter or click OK. Use the mouse or the arrow keys to determine the width and slant of the parallelogram. When the rubber-band parallelogram is positioned correctly, click the mouse or press the Insert key. A parallelogram is drawn with its baseline aligned with the angle of the first two points.

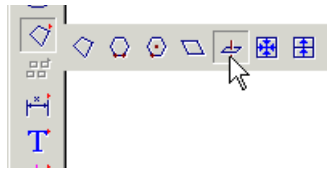
Set a point on the existing line. Then set a point to establish the location of the plane.



Perpendicular Plane ^{3D only}

Menu: Draw / Planes / Perpendicular Plane

Toolbox Icon:



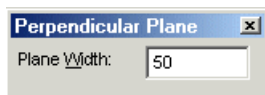
Point 1: Existing line

Point 2: Point on the line to form the center of the plane

Creates a plane perpendicular to any given point on a line or curve.

Using the Command

Choose the **Perpendicular Plane** command. You can set the width of the created plane in the **Plane Width** field.



4 3D Objects

Moving Around in a 3D Drawing

Make sure that the 2D Mode button is not pushed in; this tells you that you are in 3D Mode.



If the button is pushed in, click it to turn off 2D Mode and put DesignCAD in 3D Mode. Next, select the **Options** menu, look down the menu to the **2-D Selection Mode** command, and make sure there is not a check mark to the left of the command name. If there is a check mark, select the command to disable it.

You can move the cursor in the drawing screen using either the mouse or the keyboard.

Mouse: When you use the mouse, the cursor normally moves along two of the three axes (which two depends on your viewing angles).

In Front View or Perspective View, the mouse moves in the XY plane. To move the mouse along the third axis, first choose a drawing or point command. Then simultaneously press the Ctrl and Shift keys and continue holding down the keys as you move the mouse. Forward mouse motion moves the cursor in the positive direction along the axis, and a backward motion moves the cursor in the negative direction.

When a drawing command is active, holding down the Shift key alone forces the mouse to move along one axis in the current viewing plane

(X in the Front view). Pressing only Ctrl forces the mouse to go along the other axis in the plane (Y in the Front view).

The mouse normally moves in increments of one screen pixel. This distance varies depending on your zoom factor and the original size of the drawing. To constrain the mouse to move in increments of a particular size (1.0, 0.25, etc.), turn on the **Snap Grid** and set the **Snap Grid Size** (see Grid Options). If the change causes your mouse to move in a jerky manner, reduce the value in the **Snap Grid Size** box. You may have zoomed in on a region only slightly larger than the snap increments.

Keyboard

When using the keyboard, you move about the screen using the arrow keys. The left and right arrows move you along the horizontal axis (which of the axes is "horizontal" depends on your view angles). The up and down arrows move you along the vertical axis. Press and hold the Shift key while using the left, right, up, and down arrows to move along the horizontal and vertical axes in smaller increments or "steps." To move in the third direction, press the Ctrl key in combination with Home or End. To move the cursor in smaller increments in the third direction, press Shift in combination with the arrow keys or Ctrl+Home or Ctrl+End.

You can also specify the size of the cursor movement when using the arrow keys. Use the **Options / Cursor** command to set the **Large**

Step Size (regular arrow keys) and **Small Step Size** (Shift+arrow keys) to convenient values. (See Cursor Options.)

Solids

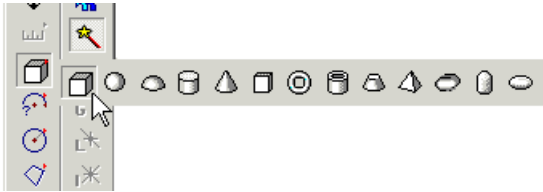
This section covers the 3D solid objects available in DesignCAD, such as boxes, spheres, and cones.

Box

Menu: Solids / Box

Shortcut Key:]

Toolbox Icon:



Point 1: First corner of the box

Point 2: Opposite corner of the box

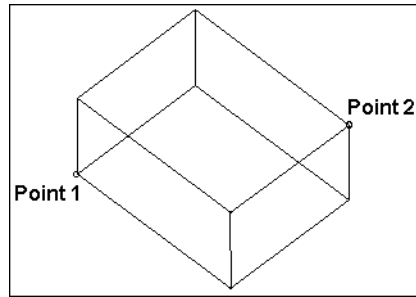
Draws a 2-D or a 3-D box. A 2-D box is defined as a Plane. A 3-D box is defined as a Solid.

Using the Command

For a 2-D box, set a point for a corner of the box. A rubber-band box shows how the box will be drawn. Set a second point for the opposite corner.

To draw a 3-D box, set a point for one corner of the box. Move the cursor until the opposite corner of the box is where you want to place it. Then

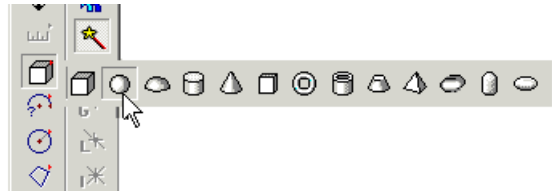
hold down the Ctrl+Shift keys and move the cursor along the third axis. Set the second point when the box is in position.



Sphere ^{3D} only

Menu: Solids / Sphere

Toolbox Icon:



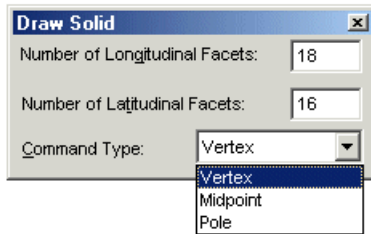
Point 1: Center of the sphere

Point 2: Point on surface of the sphere

Draws a solid sphere.

Using the Command

Choose the **Sphere** command. Specify the number of sides or facets around both the longitude and latitude of the sphere in the window. The more facets the sphere has, the more spherical it appears.

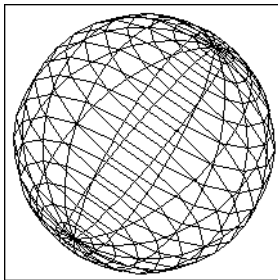


The first point is the center of the sphere. The second point determines the radius of the sphere. This point can lie in one of three places: **Pole**, **Vertex**, or **Midpoint**.

Vertex: The equator of the sphere is inscribed by a circle of that radius. (The second point lies on the equator at one of the longitudinal divisions.)

Midpoint: The equator of the sphere circumscribes a circle of that radius. (The second point will be on the equator midway between two longitudinal lines.)

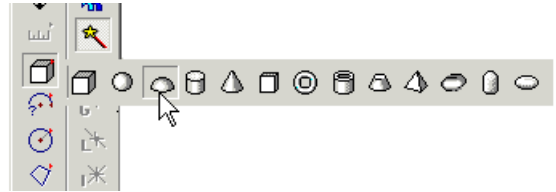
Pole: The second point is at the pole and the axis lies along the line between Points 1 and 2



Hemisphere ^{3D}only

Menu: Solids / Hemisphere

Toolbox Icon:



Point 1: Center of the flat face

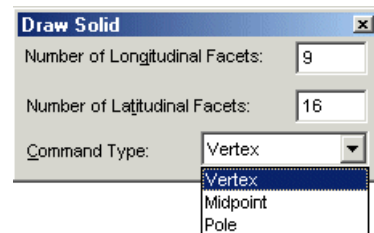
Point 2: Radius of the flat face

Point 3: Direction of the dome

Draws a solid hemisphere.

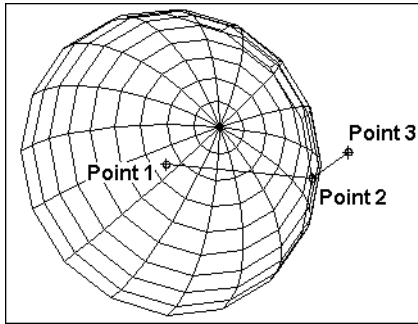
Using the Command

Choose the **Hemisphere** command. Specify the number of sides or facets around both the longitude and latitude of the sphere in the window. The more facets the sphere has, the more spherical it appears.



Choose whether the second point represents a point at a vertex of the rim, or a midpoint of one of the flats on the rim.

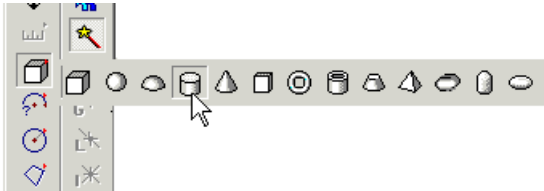
When you draw the hemisphere, you may find it helpful to think of the object as a covered bowl. Point 1 is set for the center of the cover. Point 2 is somewhere on the rim. Point 3 defines the direction of the bottom of the bowl from Point 1.



Cylinder ^{3D} only

Menu: Solids / Cylinder

Toolbox Icon:



Point 1: Center of the cylinder face

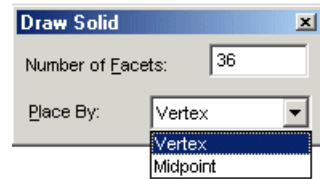
Point 2: Radius of the cylinder

Point 3: Length and direction of the cylinder

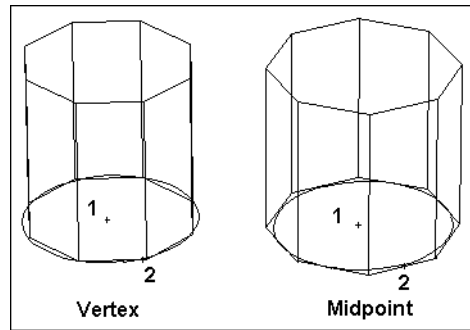
Draws a solid cylinder.

Using the Command

Choose the **Cylinder** command. You can specify the number of sides or facets around the cylinder in the window



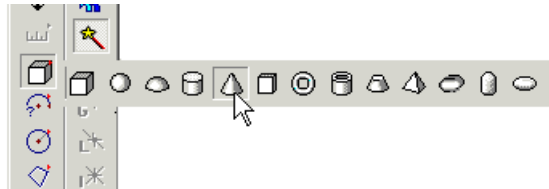
You can also choose whether the midpoint or vertex of the facets will be located at the radius defined by Point 2. If you choose **Vertex**, the base of the cylinder is inscribed by a circle of that radius. If you choose **Midpoint**, the base of the cylinder circumscribes a circle of that radius.



Cone ^{3D} only

Menu: Solids / Cone

Toolbox Icon:



Point 1: Center of the base of the cone

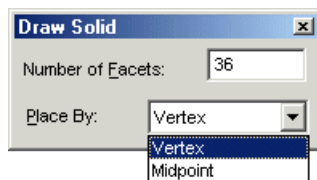
Point 2: Edge of the base of the cone

Point 3: Height and direction of the cone

Draws a solid cone.

Using the Command

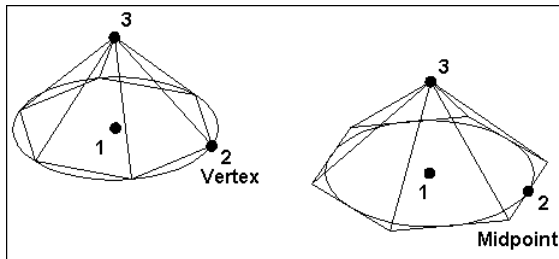
Select the **Cone** command. You can specify the number of sides or facets around the cone in the window.



Vertex: The base of the cone is inscribed by a circle of that radius.

Midpoint: The base of the cone circumscribes a circle of that radius.

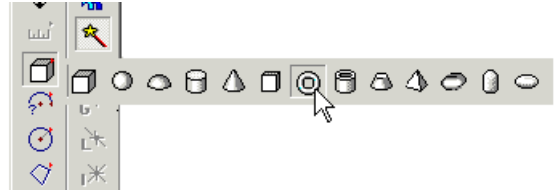
Set a point for the center of the base of the cone, a second point at the edge of the base, and a third point for the cone height.



Torus 3D only

Menu: Solids / Torus

Toolbox Icon:



Point 1: Center of the torus

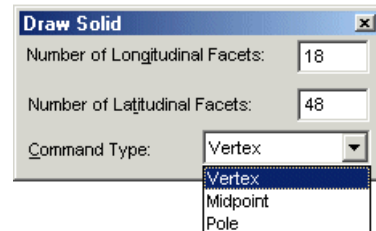
Point 2: Center of the cross-section

Point 3: Radius of the cross-section

The Torus command draws a solid torus.

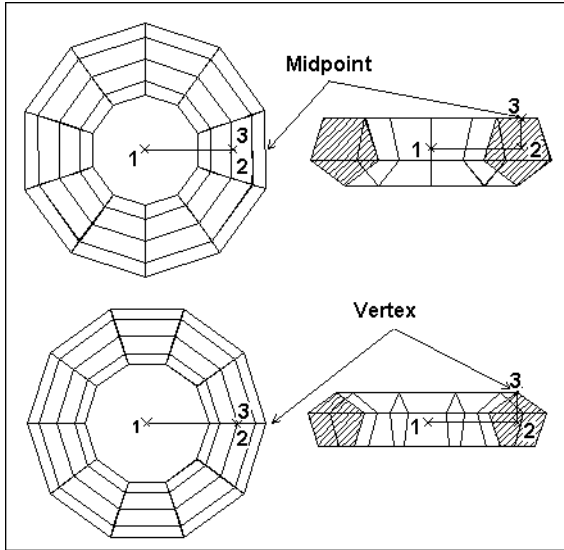
Using the Command

Choose the **Torus** command. You can specify the number of sides or facets around both the longitude and latitude of the torus in the window. The more facets the torus has, the smoother it appears.



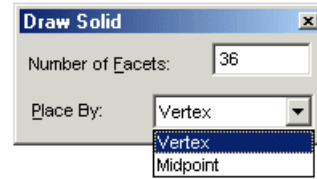
Vertex: The center for the cross-section of the torus is inscribed by a circle of the radius defined by the distance from Point 2 to Point 1. The torus is inscribed by a circle of the radius defined by the distance from Point 3 to Point 2.

Midpoint: The center for the cross-section circumscribes a circle of that radius. The cross-section of the torus circumscribes a circle of that radius.



Using the Command

Choose the **Tube** command. You can specify the number of sides or facets around the tube in the window.



Vertex: The inner/outer radius of the tube is inscribed by a circle of that radius.

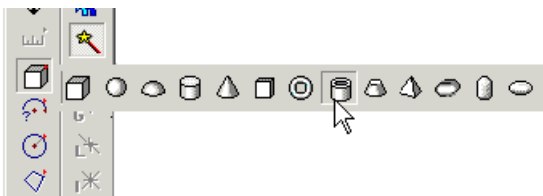
Midpoint: The inner/outer radius of the tube circumscribes a circle of that radius.

Set a point for the center of the tube and a second point for the inner or outer radius of the tube. Now set a third point for the length of the tube. Finally, set a fourth point for the other radius of the tube (if the second point was the inner radius, set the outer radius or vice versa).

Tube ^{3D} _{only}

Menu: Solids / Tube

Toolbox Icon:



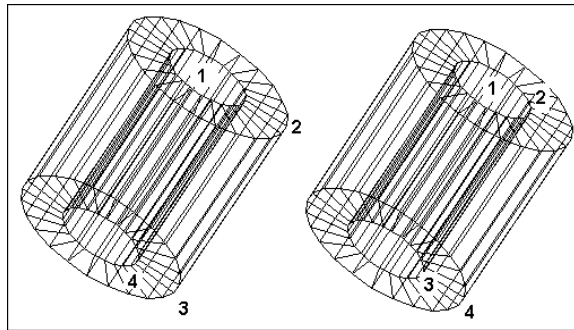
Point 1: Center of the tube

Point 2: Radius 1 (inner or outer)

Point 3: Length of the tube

Point 4: Radius 2 (outer or inner)

Draws a solid tube.



Truncated Cone 3D only

Menu: Solids / Truncated Cone

Toolbox Icon:



Point 1: Center of the base of the cone

Point 2: Edge of the base of the cone

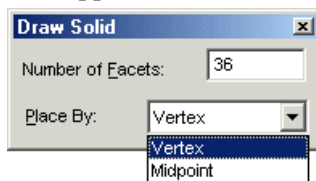
Point 3: Height of the cone

Point 4: Edge of the top of the cone

Draws a solid truncated cone.

Using the Command

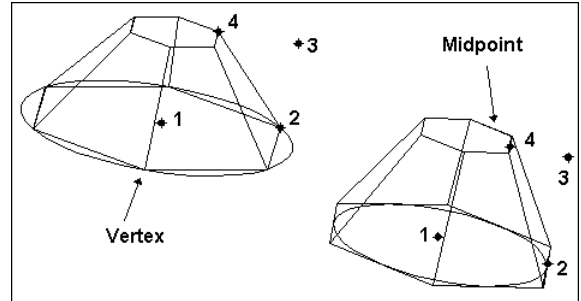
Select the **Truncated Cone** command. You can specify the number of sides or facets around the cone in the window. The more facets the cone has, the smoother it appears.



Vertex: The radius of the cone is inscribed by a circle of that radius.

Midpoint: The radius of the cone circumscribes a circle of that radius.

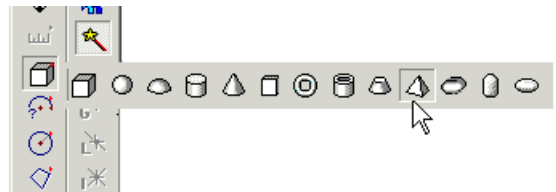
Set a point for the center of the base of the cone, a second point at the edge of the base, and a third point for the cone height. Next, move the cursor inward toward the center and set the fourth point for the edge of the truncated point of the cone.



Pyramid 3D only

Menu: Solids / Pyramid

Toolbox Icon:



Point 1: Center of the pyramid's base

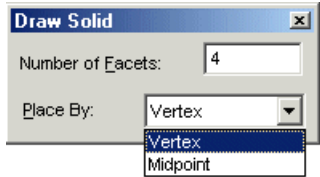
Point 2: Radius of the pyramid

Point 3: Height of the pyramid

Draws a solid pyramid.

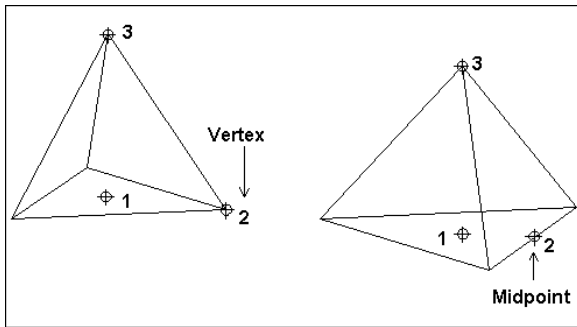
Using the Command

Select the **Pyramid** command and enter the number of sides or facets for the pyramid in the window.



Vertex: The radius of the pyramid is inscribed by a circle of that radius.

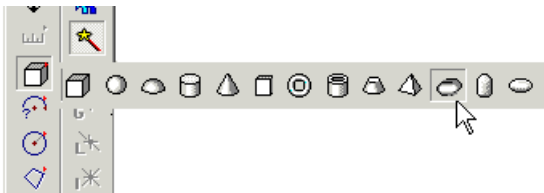
Midpoint: The radius of the pyramid circumscribes a circle of that radius.



Rounded Box 3D only

Menu: Solids / Rounded Box

Toolbox Icon:



Point 1: First corner of the box

Point 2: Opposite corner of the box

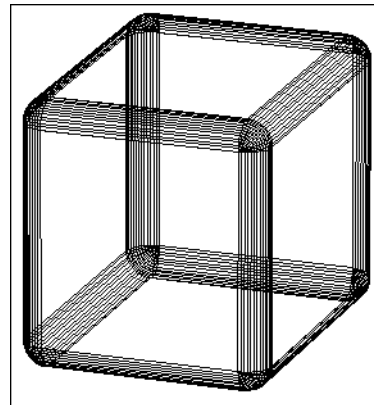
Draws a solid box with filleted (rounded) corners and edges.

Using the Command

Select the **Rounded Box** command and specify the number of facets (or "faces") for the corners and edges and the radius of these facets.



Set a point for the first corner of the rounded box. Move the cursor along the X, Y axes, and a rubber-band box appears to help you determine the size of your box. When the box is the desired size, press Ctrl+Shift to move the cursor in the Z axis. The rounded box is drawn at the size specified by the two points set and by the values entered for the options in the window.

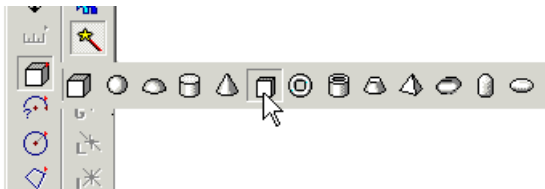


Wall ^{3D}_{only}

Menu: Solids / Wall

Shortcut Key: [

Toolbox Icon:



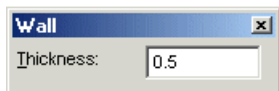
Point 1: First corner of wall

Point 2: Opposite corner of wall

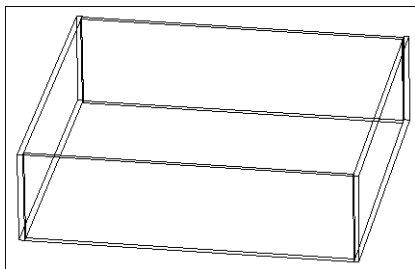
Draws a vertical wall of a specific thickness. It is useful when placing walls in a building model.

Using the Command

Choose the **Wall** command and specify the wall thickness.



All you need do is draw a 2D box; the thickness you enter is automatically applied in the third direction. Set a point for the first corner of the wall. As you move the cursor, a rubber-band box shows how the wall will be drawn. Set a second point for the opposite corner of the wall face.

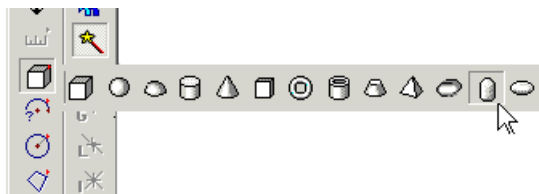


Once the first wall is set, adjacent walls are easily added.

Rounded Cylinder ^{3D}_{only}

Menu: Solids / Rounded Cylinder

Toolbox Icon:



Point 1: Center of the first end

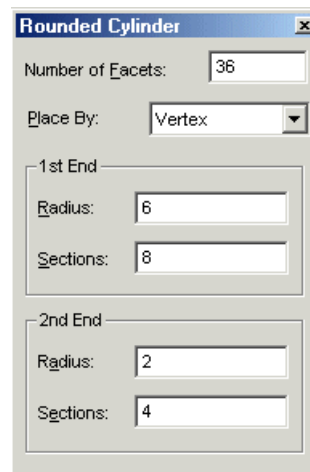
Point 2: Point on diameter of the first end

Point 3: Point on the diameter of the second end

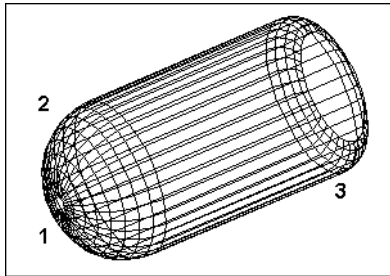
Draws a cylinder with rounding applied to one or both ends.

Using the Command

Select the **Rounded Cylinder** command and specify the number of facets (or "faces") for the corners and edges, and the radii of each cylinder end.



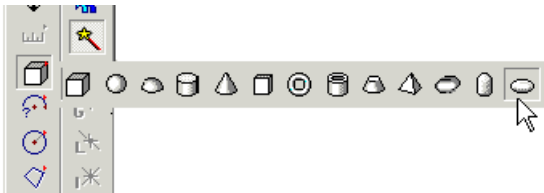
Set a point for the center of the first cylinder end. Move the cursor along the X, Y axes, and a rubber-band box appears to help you determine the diameter of your cylinder. When the cylinder is the desired size, set a second point. Then move in the perpendicular direction and set a third point to set the opposite end of the cylinder. The **Number of Sections** determines the smoothness of the rounding.



Ellipsoid ^{3D} only

Menu: Solids / Ellipsoid

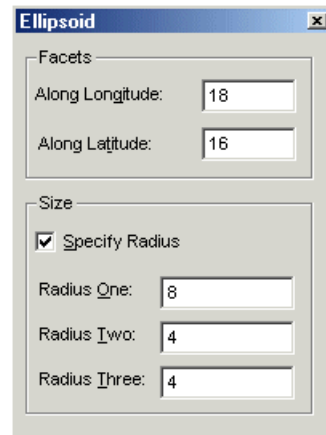
Toolbox Icon:



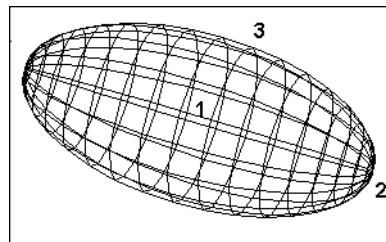
Draws a solid “football” shape - an oblong solid with radii defined for all three directions.

Using the Command

Choose the **Ellipsoid** command and set the number of facets in both directions. You can either enter the three radius values, or deselect **Specify Radius** and use the cursor to size the ellipsoid.



Set the first point at the center of the ellipsoid. If you are setting the radius points using the cursor, set two more points for the major axis and minor axis. If you entered the radius values, the second point determines the orientation of the ellipsoid.



Surfaces

This section covers 3D surfaces - zero-thickness objects equivalent to sheets or pieces of paper. (The 2D **Plane** commands also create surfaces, but these are flat. See Planes.)

These surfaces can be thought of as “grid” surfaces; they are created from a set of contour lines (rather than boundary curves) with implicit surfaces between them.

Sweep 3D only

Menu: Draw / Sweep

Shortcut Key: W

Toolbox Icon:



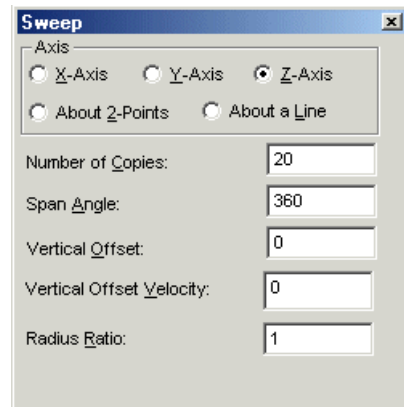
Point 1: Center of rotation

Point 2: Second point for axis of rotation (Two Point rotation only)

Makes a circular extrusion of selected objects. This command is used to create a circular Solid object from a single line outlining the silhouette of the object. You can also use this command to make a spiral-shaped object such as a spring or a bolt by specifying an offset for the sweep.

Using the Command

Select the object you want to sweep, and choose the **Sweep** command. The following window appears:



Axis: You can select from five options for the axis of rotation. The object is swept around the first point set, parallel to the axis specified here. You can choose one of the main axes, an axis defined by two points, or an axis defined by an existing line.

Number of Copies: The number of times the original shape is replicated. The more copies you make, the smoother the end result appears, but more copies also take longer to shade or edit. For most purposes you will probably want at least 10 copies per revolution (one copy every 36 degrees).

Span Angle: The number of degrees the object will sweep about its axis. For a complete, circular extrusion, enter 360°. If you are creating a spiral shape, you can enter more than 360° to achieve more than one revolution. For example, enter 1440 to get four complete revolutions.

Vertical Offset: The distance along the axis of rotation that the final copy is from the original. For normal, circular sweeps, this should be set to zero. If a value is used here, DesignCAD draws a spiral-shaped object instead of a circular object.

Vertical Offset Velocity: The ratio between the initial pitch of the offset and the final pitch of the offset. If a value of 5 is used, the final pitch will be 5 times greater than the initial pitch. If a value of 0 or 1 is used, the pitch will remain constant.

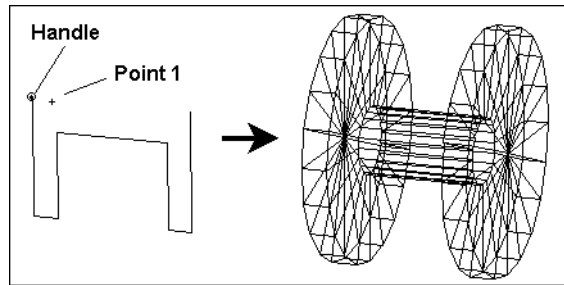
Radius Ratio: The distance from the selection handle to the axis is used for the sweep radius at the beginning of the sweep. The final radius for the sweep equals the beginning sweep radius multiplied by the Radius Ratio.

If a value of 1 is used for the Radius Ratio, the radius stays the same throughout the sweep. If a value greater than 1 is used, the distance from the axis to the extrusion increases throughout the sweep. If a value less than 1 is used, the distance from the axis to the extrusion decreases throughout the sweep.

Example: Draw a spool.

First, draw the outline or shape of the spool (shown below) and select it. Then choose the **Sweep** command. Enter 20 in the **Copies** box and 360 for **Span Angle**. Leave the offset as 0.

Choose the **X-Axis** and set a point to the right of the handle. DesignCAD sweeps the connected lines and forms the spool.



Extrude ^{3D} only

Menu: Draw / Extrude

Shortcut Key: X

Toolbox Icon:

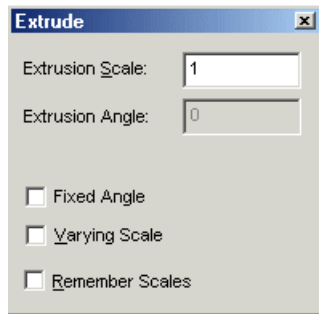


Point 1-n: Distance and direction for the extrusion path

Extends a 2D object into three dimensions, connecting it from one point to another with surfaces. For example, you can extrude a floor plan upward to make the walls of a house, or you can extrude a circle to make a pipe or bar.

Using the Command

Select the object to be extruded and choose the **Extrude** command. The following window appears:



To extrude at a fixed angle, click **Fixed Angle** and then enter an angle measurement in the **Extrusion Angle** box. Positive angle values make the extruded end larger; negative values make it smaller. Set the points for the extrusion and then press Enter.

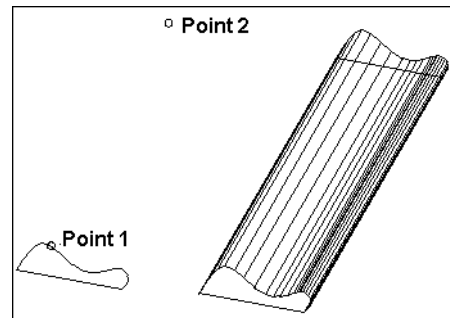
To use a different scale at points along the extrusion, click **Varying Scale**. A point list will appear in the window. Set the scale for the points of the extrusion by moving through the point numbers in the window and setting the **Extrusion Scale**. The scale is relative to the extruded object's original size and not necessarily to its size at the previous point. Set the points for the extrusion and then press Enter.

Remember Scales: Used in situations where the **Varying Scale** option is being used repeatedly. If checked, the scales for each additional point are remembered from command to command.

Set a reference point and one or more points for the object to be extruded to. The object is extruded so the selection handle is placed at each point set.

The reference point represents the location of the selected object's handle. The other points represent the distance and direction of the extrusion points. They form a parallel to the actual extrusion path.

The object does not move to the reference point. Instead, the extrusion starts at the current location of the object. Therefore, it is most convenient to just set the reference point on the handle of the object. Press Enter when you have set all the points.



Extrude Along a Curve 3D only

Menu: Draw / Extrude Along a Curve

Shortcut Key: Ctrl + Shift + X

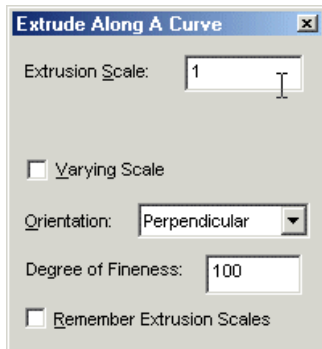
Toolbox Icon:



Extrudes an object along a path defined by a line, curve, or other drawing entity. It is possible to specify varying scales along the path by setting multiple points along it.

Using the Command

Select the object to be extruded and choose the **Extrude Along a Curve** command. The following window appears:



Select the curve to extrude along. The extrusion is created.

To use a different scale at points along the extrusion, click **Varying Scale**. A point list will appear in the window. Set the scale for the points of the extrusion by moving through the point numbers in the window and setting the **Extrusion Scale**. The scale is relative to the extruded

object's original size and not necessarily to its size at the previous point. Set the points for the extrusion and then press Enter.

Orientation: Controls the orientation of the cross-section as it moves along the path. The options here are the same as those for **Array Along a Curve** (see Array).

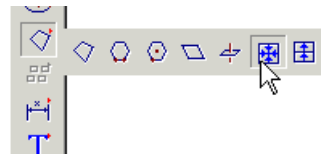
Degree of Fineness: Provides some measure of control over the number of cross-sections that are generated. A value of 0 forces the command to use the same number of cross-sections as were used in Version 14 and earlier.

Remember Extrusion Scales: Used in situations where the **Varying Scale** option is being used repeatedly. If checked, the scales for each additional point are remembered from command to command.

Surface Patch 3D only

Menu: Draw / Surface Patch

Toolbox Icon:



Point 1: First line

Point 2: Second line

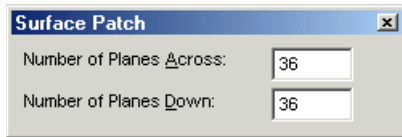
Point 3: Third line

Point 4: Fourth line (optional)

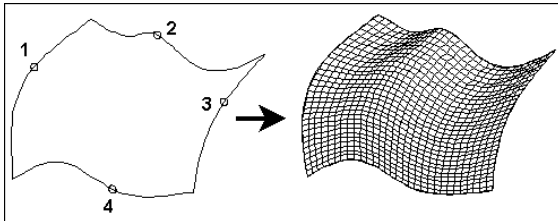
Fits a smooth surface between any three or four lines, curves, or arcs which meet at their endpoints to form a closed area.

Using the Command

Choose the **Surface Patch** command and specify the grid spacing for the surface.



Set a point on each line entity you want to patch. The line entities must share endpoints and make up a closed area or the Surface Patch command does not work properly. Press Enter when you are finished. The lines are then connected with a grid surface.

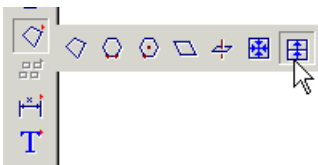


Surface Connect ^{3D} only

Menu: Draw / Surface Connect

Shortcut Key: Ctrl+Q

Toolbox Icon:



Point 1: First line to be connected with a surface

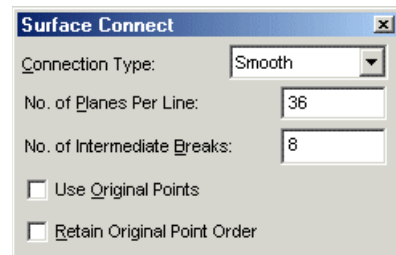
Intermediate Points: Lines or points to define surface path

Point n: Last line to be connected with a surface

Stretches a surface between two or more lines. The lines can be planes, lines, curves, arcs, or circles. The original lines to be connected can be any shape and at any orientation in 3D space.

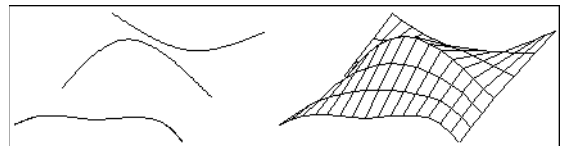
Using the Command

Choose the **Surface Connect** command. Set a point on each line entity to be connected and changed into a Solid. Press Enter to end the command, when you have the necessary points.

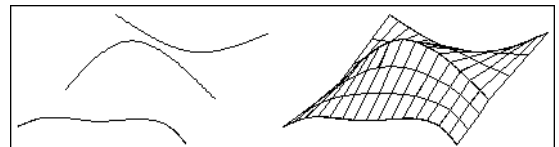


Connection Type: There are three options for the connecting surface:

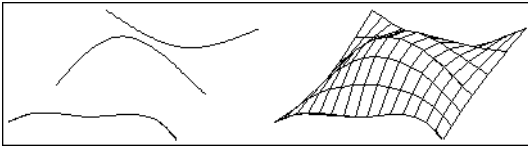
- **Normal:** Makes a ruled surface between each pair of lines.



- **Curved:** Forms a ruled surface "bent" to follow the curve defined by the points set between the original lines. It is a straight or linear fit between the original lines.



- **Smooth:** Forms the smoothest possible surface over the original lines.



No. of Planes per Line: The number of individual planes to be constructed in the connecting surface along the original lines.

No of Intermediate Breaks: The number of planes to be constructed in the connecting surface between each pair of original lines. With the **Normal** and **Smooth** options, you set a point on each of the original lines to be connected. With the **Curved** option, you can set additional points in between the original lines to force the surface to curve through those intermediate points.

Use Original Points: Forces the command to connect the original points of the cross-section lines and ignore the **No. of Planes Per Line** option. This option retains the true shape of the lines being connected. If this option is disabled, the vertices of the surface are distributed equally along the length of each line and can result in some corners being clipped off.

This option only works correctly if each line being connected has the same number of points and is useful when connecting a surface along a series of arrayed/copied lines or planes.

Retain Original Point Order: Connects points of each line to the corresponding points on the following line (i.e., Point 1 is connected to Point 1, Point 2 to Point 2, etc.). Like the **Use Original Points** option, this option is most useful when connecting a series of arrayed/copied lines or planes.

When this option is enabled, it doesn't matter which point or points are used when selecting each line to be connected; unexpected twists are eliminated in the final surface. If this option is not selected, the selected point of each line is connected to the selected point of the next one, which can result in unintended twists in the surface for closed cross-sections.

3D (Solid) Editing Operations

This section covers ways you can combine solids, otherwise known as Boolean operations.

See also: Section Delete and Section Cutoff.

Solid Add 3D only

Menu: Solids / Solid Add

Shortcut Key: Ctrl+J

Point 1: First Solid

Point 2: Solid to be added to first

Adds one Solid object to another. It makes a single Solid out of two Solids, and eliminates any unnecessary remaining surfaces.

Using the Command

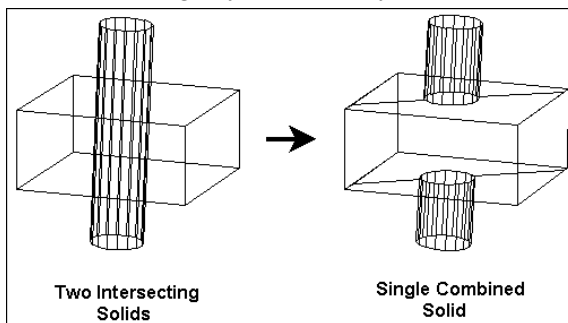
Choose the **Solid Add** command and set a point on each of the Solids to be added. The Solids are then combined into a single Solid, and any internal surfaces are removed.

NOTE: For best results, the solids should overlap rather than meet exactly at a single face. For example, if you want to stack a cube on top of another, make one

cube just a little taller, and place the other cube so that the overall height is correct. When you add the solids, the overlapping volume is removed.

Example: Make a single solid from a cylinder passing through a box.

Select the **Solid Add** command and set a point on the box. Set a second point on the cylinder. DesignCAD redraws the two objects as a single Solid, removing any unnecessary surfaces.



Solid Intersect ^{3D}_{only}

Menu: Solids / Solid Intersect

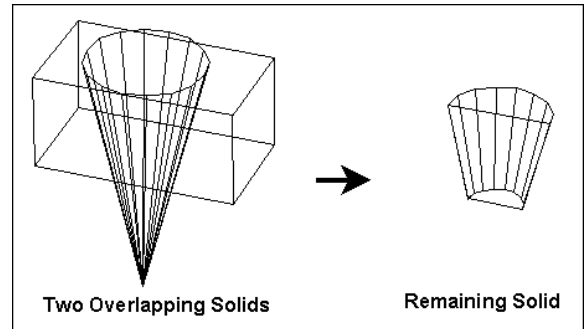
Point 1: First Solid

Point 2: Second Solid

Removes all parts of two overlapping Solids except the part that both Solids share. This part forms a new Solid.

Using the Command

Choose the **Solid Intersect** command. Set a point on each of the intersection Solids. The overlapping area remains.



Solid Subtract ^{3D}_{only}

Menu: Solids / Solid Subtract

Shortcut Key: Ctrl+U

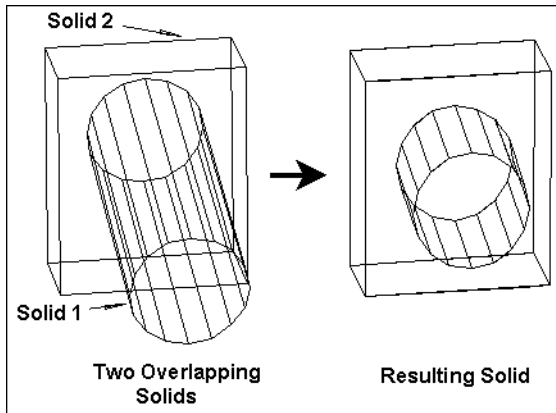
Point 1: Solid to be removed (template)

Point 2: Solid to be subtracted from

Removes one Solid from another Solid that it overlaps. the solids should overlap rather than meet exactly at a face.

Using the Command

Choose the **Solid Subtract** command. Set a point anywhere on the solid to be subtracted (the one that will be used as a template for the hole). Set a second point on the solid to be subtracted from. The first solid is removed from the second solid.



NOTE: To erase the cut lines that remain after this command, see Hidden Edge and Hidden Edge by Section.

Solid Exclusive Or ^{3D}only

Menu: Solids / Solid Exclusive Or

Point 1: First Solid

Point 2: Second Solid

Subtracts the volume in common between two solids. If a portion or portions of both of the solids remain, the portions are still recognized as a part of their original solid.

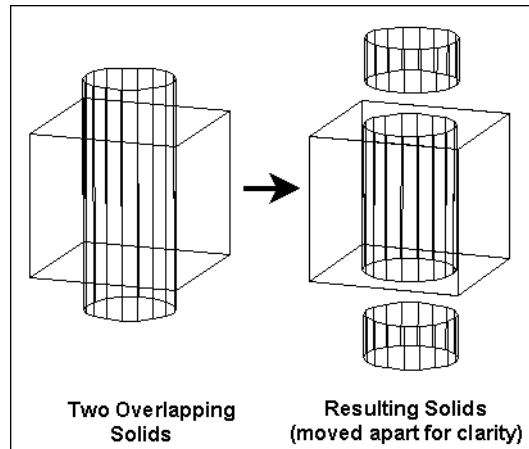
Using the Command

Choose the **Solid Exclusive Or** command. Set a point on each of the solids. The common volume between the two solids is removed. The only portions that remain are portions that were exclusive to one or the other of the two original solids.

NOTE: For best results, the solids should overlap rather than meet exactly at a face.

Example: Subtract common volume from an intersecting box and a cylinder.

Select the **Solid Exclusive Or** command and set a point on each of the solids. DesignCAD removes the shared volume and redraws the remaining portions.



Solid Segregate 3D only

Menu: Solids / Solid Segregate

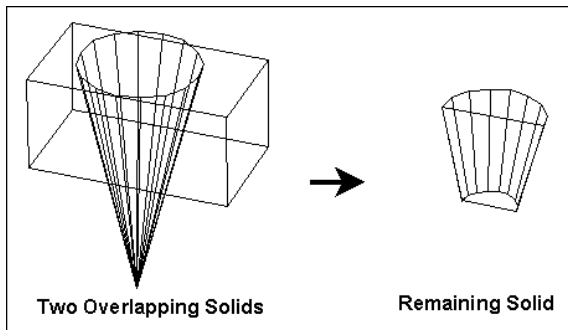
Point 1: First Solid

Point 2: Second Solid

Isolates the common volume between two solids. This command is like the **Solid Exclusive Or** command, but the common volume is cut out of the two solids and becomes a separate solid, rather than be subtracted from them.

Using the Command

Choose the **Solid Segregate** command. Set a point on each of the solids. The common volume between the two solids is cut out and becomes a third solid. The only portions that remain as the two original solids are portions that were exclusive to those solids before the command was executed..



NOTE: For best results, the solids should overlap rather than meet exactly at a face

Slice 3D only

Menu: Solids / Slice

Shortcut Key: Ctrl+L

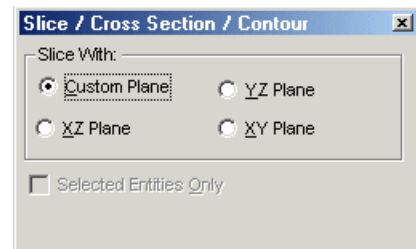
Points 1-3: Points to define the cutting plane

Point 4: Region to be removed

Deletes parts of the drawing in front of or behind a predefined or custom plane.

Using the Command

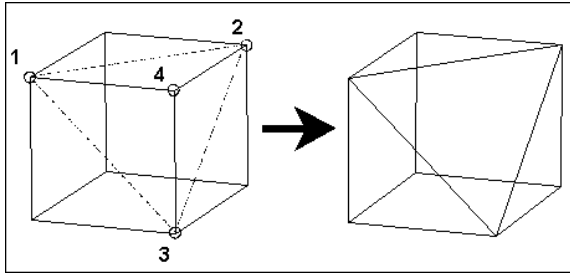
Select the **Slice** command. The Slice window appears.



If a main plane is selected, set a single point to define the location of the cutting plane. Then set a point either in front of or behind the location of the cutting plane. All parts of the drawing that lie on that side of the cutting plane will be cut and erased at the specified location using a plane that lies parallel to the plane indicated in the selected option.

If the second point is not used and the Enter key is pressed, then all entities and parts of entities will be cut at the cutting plane location, but will not be erased.

If **Custom Plane** is selected, 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.



If the fourth point is not used and the Enter key is pressed, then all entities and parts of entities will be cut at the cutting plane location, but will not be erased.

Slice by Curved Surface 3D only

Menu: Solids / Slice by Curved Surface

Point 1: Solid to be sliced

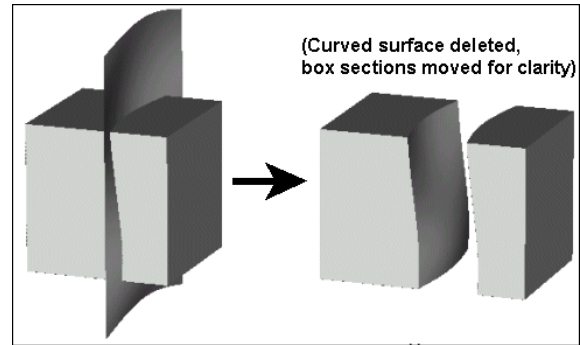
Point 2: Surface to be used to slice solid

Slices a solid using a curved surface.

Using the Command

Select the **Slice by Curved Surface** command. Set a point on the solid to be sliced. Set a point on a curved surface that intersects the solid. The

curved surface may be drawn using the **Surface Patch** or **Surface Connect** command. The solid is cut using the curved surface.



Cross Section 3D only

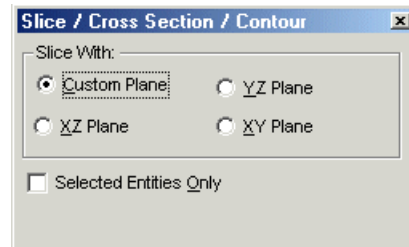
Menu: Solid / Cross Section

Points 1-3: Points to define the plane of the cross section

Draw cross sections of solids that intersect a specified YZ, XZ, XY, or custom plane.

Using the Command

Make sure that the current drawing contains at least one solid. (The Cross Section command does not work on surface grids drawn using the **Surface Connect** and **Surface Patch** commands; grids do not have volume.) Select the **Cross Section** command.



If a main plane is selected, set a single point to define the location of the cross-section plane. The cross-section plane incorporates the point set and lies parallel to the plane indicated in the selected option. A plane is drawn for each portion of a solid that intersects the cross-section plane.

If **Custom Plane** is selected, set three points to define the cross-section plane. A plane is drawn for each portion of a solid that intersects the cross-section plane.

Cross sections can be moved away from their solids and shaded since they are plane entities.

Solid Interference Check ^{3D}_{only}

Menu: Solids / Solid Interference Check

Point 1: First solid to check for interference

Point 2: Second solid to check for interference

Checks two solids to see if they overlap one another at any point.

Using the Command

To use this command, select the command and set a point on each solid. DesignCAD will display a window stating whether or not interference is detected.

Solid Define ^{3D}_{only}

Menu: Solids / Solid Define

Shortcut Key: Ctrl+D

Combines all the currently selected entities as a single Solid. This allows you to build Solids from a collection of planes, extrusions, and surfaces.

Using the Command

Select the objects you want to define as one Solid, and choose the **Solid Define** command. The Solids are defined as one Solid.

A Solid must be a *closed* set of planes and surfaces. This command does not verify whether the current selection is a legitimate Solid. If you define an invalid Solid, operations such as **Solid Add** or **Solid Subtract** command may not work correctly.

Solid Explode ^{3D}_{only}

Menu: Solids / Solid Explode

Reduces a selected Solid into its component parts: planes, surface meshes, and lines. Each component can then be manipulated individually, without affecting the rest of the former Solid.

Using the Command

Select the Solid to be exploded and choose the **Solid Explode** command. The Solids now regain their individual properties and can be manipulated as individual components.

*NOTE: Once a solid is exploded, it cannot be used in solid commands such as **Solid Add** and **Solid Subtract**. To combine objects into a solid, use **Solid Define**.*

Hammer ^{3D} only

Menu: Edit / Hammer

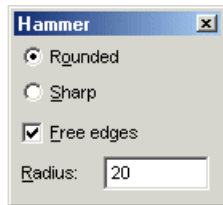
Point 1: Source point on the grid (point where the hammer hits)

Point 2: Destination point (where the hammer forces the grid)

Reshapes a surface grid.

Using the Command

Choose the **Hammer** command. In the **window** choose the options you want.

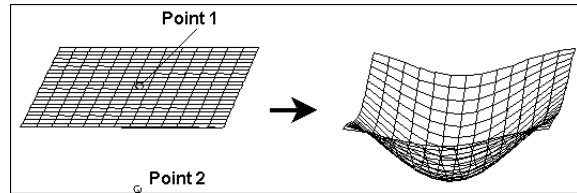


The size of the area affected by this command can be specified by entering the **Radius**. Clicking **Free Edges** lets the command warp the edges of the surface that is being hammered. You can also select whether the modified portion of the surface is to have a rounded or sharp point.

When you have specified the options you want, set a point on the surface grid at the location it is to be modified, and a point for the destination of that location. The surface is "hammered."

The **Hammer** command only moves existing points (intersections) on the grid. It does not create any new points. When you construct a grid

to be modified with the Hammer command, you should make the spacing dense enough to get a smooth result.



*NOTE: The **Hammer** command only affects surfaces created with the **Surface Connect** or **Surface Patch** commands. It does not affect planes, lines, curves, arcs, or extruded objects. It can be useful in creating specially shaped surfaces.*

Working Planes

The working plane is the plane on which 2D objects are created. In 2D mode, the working plane is always the XY plane, but when working in 3D, you can switch the working plane.

Set Working Plane

Menu: View / Working Plane / Set Working Plane

Point 1: Origin of workplane

Point 2: X axis of workplane

Point 3: Orientation of workplane

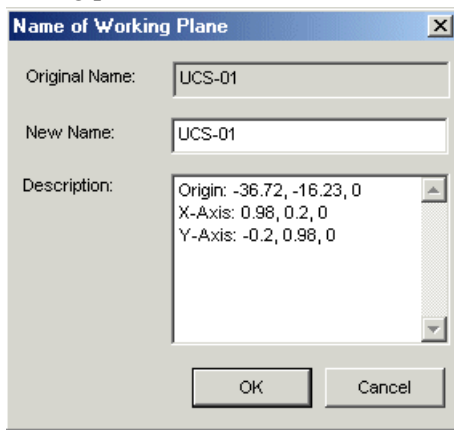
Point 4: A point defining the front of the plane

Sets a specified plane as a temporary substitute for the XY plane. It adjusts mouse and cursor movements accordingly. This feature is very convenient for doing work along oblique faces of complex objects.

Using the Command

Choose the **Set Working Plane** command. Set four points for the workplane. Point 1 becomes the new origin. Point 2 determines the X axis of the workplane. Point 3 establishes the orientation of the plane. Point 4 determines the front side of the plane.

Then set the name and confirm the orientation of the working plane.

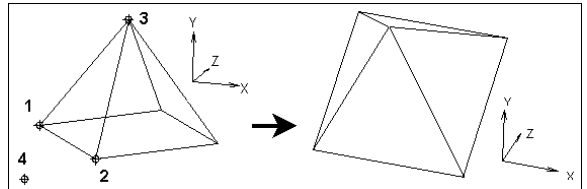


TIP: If the entire drawing is not visible after using the **Set Working Plane** command, use the **Fit to All Windows** command to readjust the views.

Example: Work on the face of a pyramid.

Set the first point on the lower left corner of one of the faces. This point sets the origin for the workplane. Set the second point at the lower right corner of the same face. This sets the X axis for the workplane. Set the third point on the apex of the pyramid. This sets the orientation, or tilt, of the plane. The last point determines which side of

the plane DesignCAD recognizes as the front. When you have done this, the program redraws the pyramid with the front of the plane facing you.

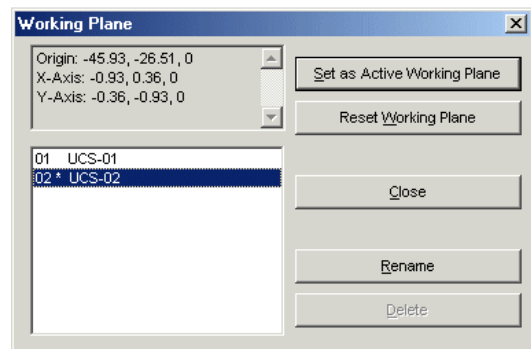


Working Plane Properties

Makes it easy to switch from one workplane to another when multiple workplanes have been defined for a drawing.

Using the Command

After at least one workplane has been defined (in addition to the default workplane) with the **Set Working Plane** command, select the **Working Plane Properties** command. The following window appears.



The lower window on the left of the window shows a list of the defined workplanes. The top window on the left gives the specifics for the selected workplane.

Set as Active Working Plane: Sets the selected workplane as the current workplane.

Reset Working Plane: Resets the default workplane as the current workplane.

Rename: Opens a window in which you can rename the workplane and/or change its description.

Delete: Deletes the selected workplane.

Reset Working Plane

Menu: View / Working Plane / Reset Working Plane

Reverses a **Set Working Plane** command. It restores the original coordinate system to the drawing, deleting any reference to the temporary coordinate axes set up by the **Set Working Plane** command.

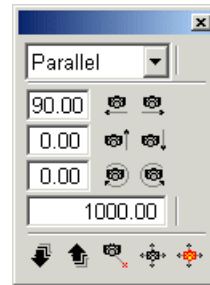
3D View Settings

When working in 3D, you will often need to change the angle or orientation at which you see the model. This section covers ways to change the view, as well as ways to save views for future reference.

View Toolbox

The View Toolbox contains useful tools for modifying the way you look at your drawing in 3D space. You can change the horizontal viewing angle (the viewer's rotation about the Y-axis), the

vertical viewing angle (the viewer's rotation about the X-axis), and the tilt (the viewer's rotation about the Z-axis).



NOTE: *For information on displaying toolboxes, see View Options.*

You can change the orientation of the drawing visually by dragging the mouse. You can choose from a set of predefined viewing angles, or you can define your own view. No matter what direction you want to approach your drawing from, DesignCAD's View Toolbox lets you get there. Like the snap tools, these tools can be used inside another command.

From the **Projection List**, select the standard view you want to see: Side View, Top View, etc. You can adjust the angle incrementally in all three directions by clicking the **Move Camera** buttons. You can also set the **View Distance** and use the arrow buttons to adjust the distance incrementally.

Set Viewer Points: Set two points - the first for the viewer position and the second for the view direction.

Set View by View Center, Set View by Drawing Center: Set the view center and drag the mouse around (keeping the left button pressed) to adjust the view.

Set View **3D only**

Menu: View / Set View

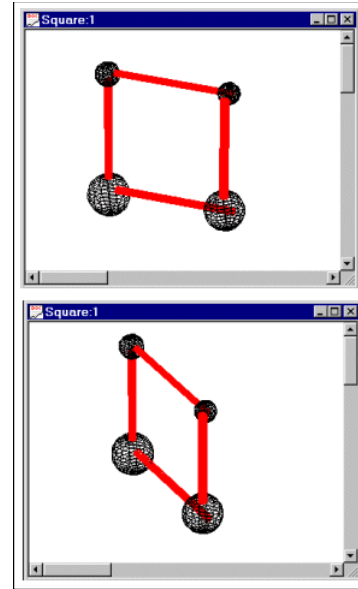
Shortcut Key: Y

Changes the viewer location.

Using the Command

Select the **Set View** command and move the cursor to the view window. Hold down the left mouse button and drag the cursor across the screen with the mouse. The view changes as you drag the mouse. To preview the drawing at a view position, release the mouse button. Press Enter when you have the desired view.

You can change the perspective of the view by moving the cursor in and out on the Z axis (by pressing Ctrl+Shift and moving the mouse). This command can be undone with the **Zoom Previous** command.



Set View by Drawing Center

3D only

Menu: View / Set View by Drawing Center

To change the view, set the view center and drag the mouse around (keeping the left button pressed) to adjust the view.

Save Current View **3D only**

Menu: View / Save Current View

Saves the current view settings as a custom view selection in the View Toolbox (see View Toolbox.)

Using the Command

When you have adjusted the view to your liking, select the **Save Current View** command. Enter a name for your custom view in the and click OK. The view is saved under the selected name in the view box in the View Toolbox. You can remove the view by selecting the view and pressing the Del key.

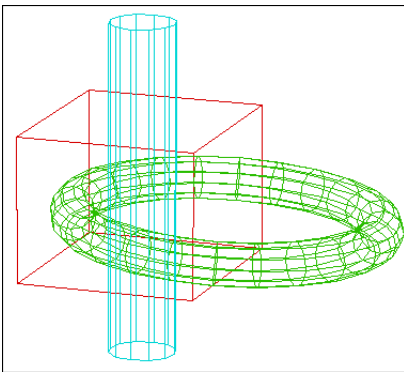
Object Display

Choose **View / Display Types (Active View)** to change the settings for only one view. Choose **Display Types (All Views)** to change settings for all displayed views.

Wireframe

Menu: View / Display Types / Wireframe

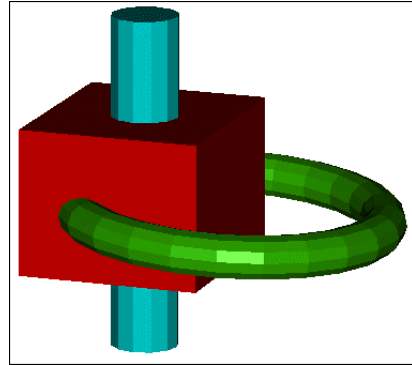
Displays the model without shading, and without hiding lines.



Flat Shading Mode

Menu: View / Display Types / Flat Shading Mode

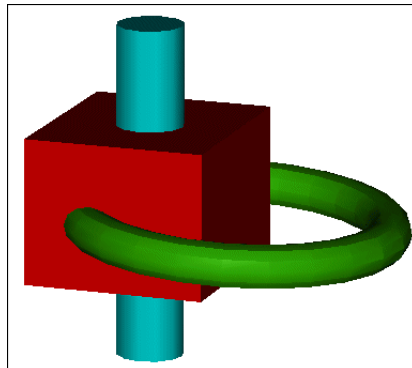
Displays non-smoothed shading -- for example, all the facets on a cylinder will look flat, instead of smoothly rounded.



Smooth Shading Mode

Menu: View / Display Types / Smooth Shading Mode

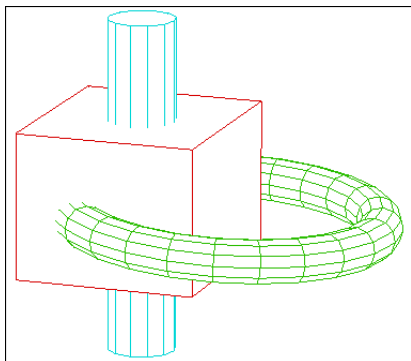
Round objects will look smoothly rounded instead of having a lot of flat faces. How smooth it looks will depend on how many facets were used to generate the object.



Quick Hidden Line Mode

Menu: View / Display Types / Quick Hidden Line Mode

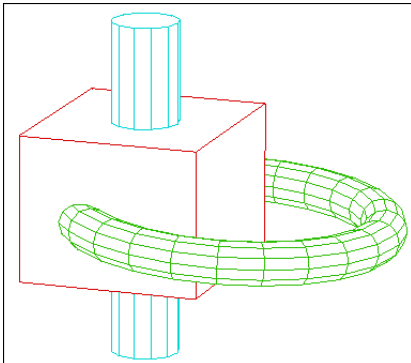
Hides lines that would be hidden by other parts of the object, but it will not show the "seams" where one solid intersects with another



Hidden Line Mode

Menu: View / Display Types / Hidden Line Mode

Hides hidden lines and also shows the boundary where two solids intersect.



Hidden Line Removal

Menu: Tools / Hidden Line Removal

Shortcut Key: Ctrl+F8

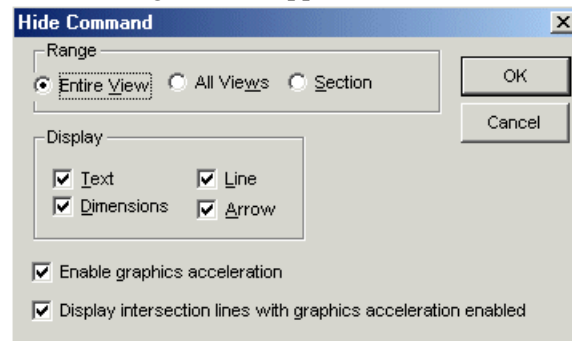
Toolbox Icon:



Performs hidden line removal on the entire drawing or on a section of the drawing. This causes all lines behind surfaces to be removed, giving the objects a more realistic image.

Using the Command

Choose the **Hidden Line Removal** command and the following window appears:



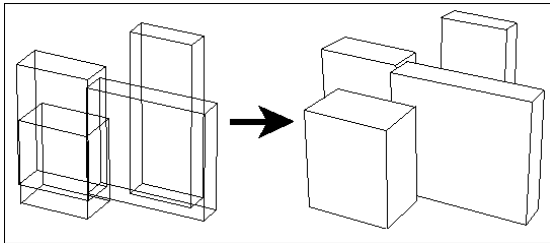
Specify whether you want to remove lines for the entire active view, all views, or only a section. If you choose the **Section** option, set two points to enclose the section on which you want DesignCAD to execute the command.

You can also specify whether you want the text and dimensions to be displayed with the resulting image.

If your graphics card supports graphics acceleration, check the **Enable Graphics Acceleration** option to improve performance.

This will enable interactive hidden line removal. Interactive hidden line removal will allow you to move and edit existing drawing items and even add new ones, all while the specified view(s) maintain hidden line removal. The view(s) will remain in hidden line removal mode until you change the status of the view(s) to wireframe or shading.

If your graphics card supports graphics acceleration, you can also check the **Display Intersection Lines with Graphics Acceleration** option to display lines at the intersections of solids and planes.



Hidden Edge

Menu: Edit / Hidden Edge

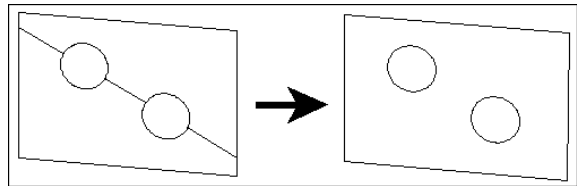
Erases cut-plane lines which are sometimes a result of the **Cut Plane**, **Polygon Subtract**, and **Solid Subtract** commands.

Using the Command

After executing a **Cut Plane**, **Polygon Subtract**, or **Solid Subtract** command, select the **Hidden Edge** command. Set a point on one of the cut-plane lines so that it will be hidden from the view or from a printout of the drawing.

Example: Draw a large square plane and two small circular planes. Draw the circular planes so that they are contained by the square plane with respect to the XY plane. Use **Polygon Subtract**

to subtract the circular planes from the square plane. Notice the cut-plane lines which are a necessity of this kind of operation. Choose the **Hidden Edge** command and set a point on one of the cut-plane lines. The cut-plane line will be removed from the view and will not be printed in subsequent printouts. Repeat the **Hidden Edge** command as many times as desired.



Hidden Edge by Section

Menu: Edit / Hidden Edge by Section

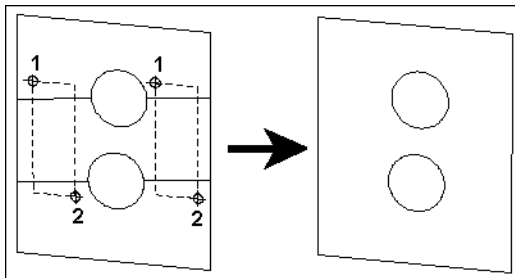
Erases cut-plane lines which are sometimes a result of the **Cut Plane**, **Polygon Subtract**, and **Solid Subtract** commands.

Using the Command

After executing a **Cut Plane**, **Polygon Subtract**, or **Solid Subtract** command, select the **Hidden Edge by Section** command. Set two points to define a bounding box that encloses or touches the cut-plane lines, so that they will be hidden from the view or from a printout of the drawing.

Example: Draw a large square plane and two small circular planes. Draw the circular planes so that they are contained by the square plane with respect to the XY plane. Use **Polygon Subtract** to subtract the circular planes from the square plane. Notice the cut-plane lines which are a necessity of this kind of operation. Choose the **Hidden Edge by Section** command and set a point at one corner of the area that contains the cut-plane lines to the left of the circular holes.

This point also defines one of the corners for a bounding (or rubber-band) box that moves with the cursor. Move the cursor so that the bounding box touches both of the edges to the left of the holes. Set another point. Both of the cut-plane lines to the left of the holes will be removed from the view and will not be printed in subsequent printouts. Now do the cut-plane lines on the right side of the holes.



Show All Edges

Menu: Edit / Show All Edges

Shows any edges that were hidden using **Hidden Edge** or **Hidden Edge by Section**.

Shading ^{3D}_{only}

Menu: Tools / Shading

Shortcut Key: F8

Toolbox Icon:



Point 1: First corner of shading region (optional)

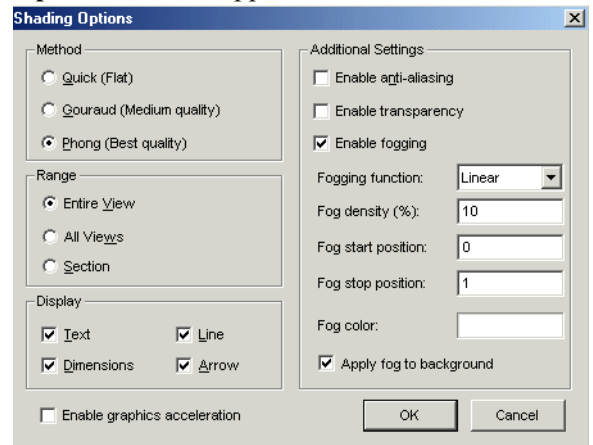
Point 2: Opposite corner of shading region (optional)

Shades all or part of a drawing. Only solids, surfaces, and planes can be shaded.

Light and material settings can be used to make the shaded rendering appear realistic. See Light and Materials.

Using the Command

Choose the **Shading** command. The **Shading Options** window appears:



Phong (Best Quality): Uses a fine shading pattern, and displays the material texture and/or texture mapping.

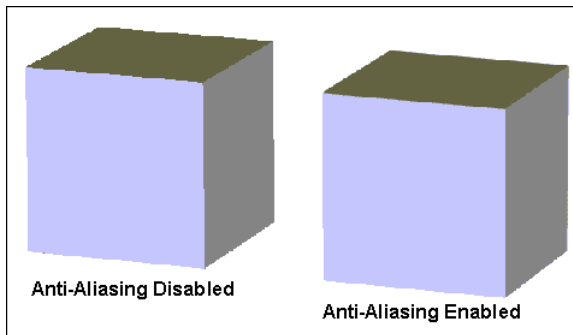
Gouraud (Medium Quality): Displays the material texture and/or texture mapping, but the resulting shading is not quite as fine as that of Phong shading.

Quick (Flat): Quite a bit faster than Phong or Gouraud shading, but the shading is coarser and less realistic. No smoothing takes place, and material textures and texture mapping are not represented.

All three shading methods can be done either for the active view, all views, or for a rectangular region of the screen. The method you choose becomes the default selection the next time you use the **Shading** command.

If your graphics card supports graphics acceleration, check the **Enable Graphics Acceleration** option to improve performance. This will enable interactive shading. Interactive shading will allow you to move and edit existing drawing items and even add new ones, all while the specified view(s) maintain shading. The view(s) will remain in shading mode until you change the status of the view(s) to wireframe or hidden line removal.

Enable Anti-aliasing: Determines whether or not to change the color of the object at its edges slightly so that all straight edges will appear smoother. If checked, the edges will be smoothed. Otherwise some of the edges will appear slightly jagged.



Enable Transparency: When Phong or Gouraud shading is used and graphics acceleration is disabled, this option can be enabled to show transparencies when a drawing contains objects that have been assigned materials that have transparent qualities and the drawing is shaded.

Enable Fogging: When Phong or Gouraud shading is used and graphics acceleration is disabled, this option can be enabled to add a fog effect to the rendered drawing.

Fogging Function: The function used to create the fog. **Linear** produces proportionally more fog the "deeper" into the drawing you go. **Exp** produces a less dense fog than Linear. **Exp2** also produces a less dense fog than Linear, following a different function than Exp.

Fog Density (%): The maximum fog density.

Fog Start Position: The point in the drawing where the fog is at its minimum. This value is a scale factor based on the depth of the drawing from the front to the back at the current view settings. A value of 0 starts the fog at the very front of the drawing. A value of -1 starts the fog one drawing-depth in front of the drawing's front. A value of 0.5 starts the fog in the middle of the drawing.

NOTE: This position and scaling will vary depending on the size of the drawing, the viewing angle, and distance settings.

Fog Stop Position: The point in the drawing where the fog reaches its maximum thickness. Like the Fog Start Position, this value is a scale factor based on the depth of the current view of the drawing.

Fog Color: The base color for the fog. To select the color, click the color box and select a color from the palette

Apply Fog to Background: If checked, the entire drawing background becomes the same color as the fog.

If Windows is configured for only 16 colors, the shaded drawing is not very realistic. If Windows is configured for 32,000 or more colors, however, the shading will be fastest and look very realistic.

You can change the number of colors and the screen resolution used by Windows in the Windows Control Panel, under Display.

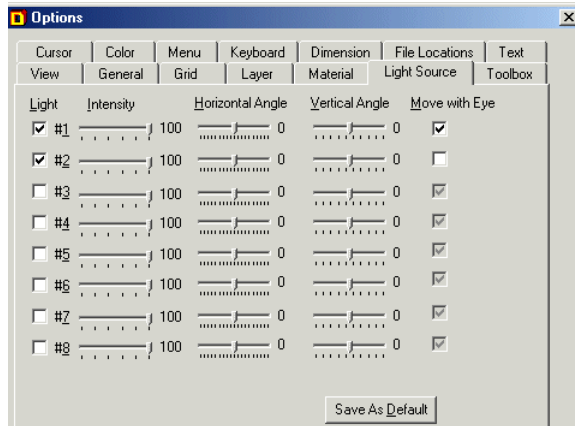
Light and Materials

These options affect how 3D entities look in shaded rendering.

Light Source Options 3D only

Menu: Tools / Light Source

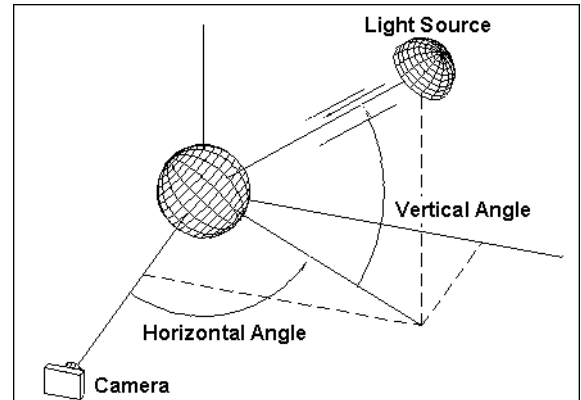
Create up to eight different light sources. Each source is activated by clicking the checkbox beside it.



Intensity: . Increasing the intensity of one light source does not increase the total amount of light. The total light is divided between each of the active light sources. If all eight light sources are turned on and set to intensity 100, each contributes 1/8 of the total light.

Horizontal Angle, Vertical Angle: Sets the horizontal and vertical angles of the light source relative to the drawn object.

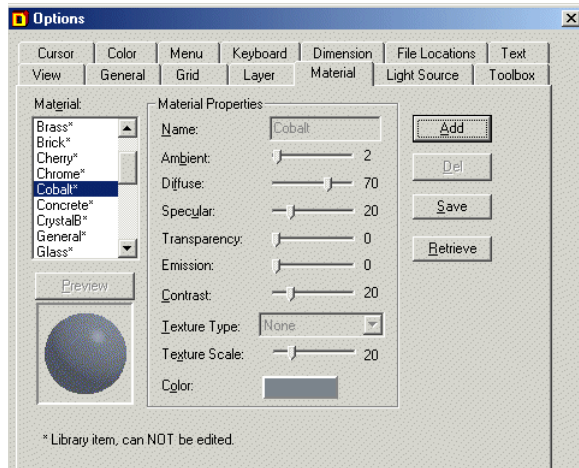
Move with Eye: The light sources move when the viewing angle changes. Otherwise the light sources remain in their locations relative to the drawing when the viewing angle changes.



Material Options 3D only

Menu: Options / Material

Specify the material properties of an object. You can choose the current material from the materials provided. In other words, you select the material in which the next object or objects will be drawn. You can also create your own materials.



After the material is selected, objects drawn will have that material property. For example, if you select the material **Walnut** and then draw a hemisphere, the hemisphere will look like walnut wood.

NOTE: *Library materials cannot be edited. To create editable materials, click **Add**.*

Ambient: The amount of background light shining on the object. This dictates how much shadow is visible on a shaded object.

Diffuse: The amount of flat (not shiny) reflection you get from a shaded object.

Specular: How shiny (not flat) the material appears.

Transparency: How "see-thru" the material appears. Materials with a value of 0 are opaque. Materials with a value of 100 can barely be seen when shaded. Transparency is only displayed when shading with Gouraud or Phong shading with graphics acceleration disabled.

Emission: Makes a shaded object look brighter, as though it is being lit from within. A value of 0 has no visible effect. A value of 100 makes the material look very bright. Emissive objects do not act as light sources. They do not cast light on other objects in the drawing.

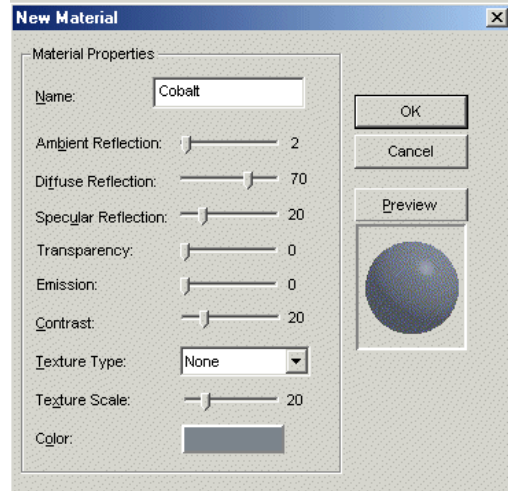
Contrast: How shiny the surface appears. The lower the setting, the shinier the surface appears.

Texture Type: The texture of the material.

Texture Scale: The amount and size of the texturing.

Color: The color of the selected material. Clicking on this bar will bring up the color palette.

Add: Creates a custom material. You can set preferences for all of the material's properties.



Del: Deletes a material from the materials selection box.

Save: Saves created materials as separate files.

Retrieve: Retrieves a specified material file and inserts it into the selection box.

Materials Toolbox

Using the Material Toolbox allows creation of new materials from a set of customizable textures and properties. Any material can be applied to any object in a drawing. For details on displaying toolboxes, see View Options“View Options” on page 33.



If you click on the down arrow, the Material box shows a list of material types available for use. The list includes any materials you have created and added to the list.

NOTE: *Don't confuse this list with the **Material List** command in the **File** menu. The **Material List** command provides a net listing of all the attributes you placed in your drawing. **Material**, as it is referred to in the **Material Toolbox**, affects the appearance of shaded drawings. **Attributes** are generally used to denote physical materials-lumber, bolts, screws, and other parts-used to build the object depicted in your drawing.*

New Material: Opens the New Material window, described above.

Material: Opens the Material Options window.

Apply: Applies the material to selected entities.

Same As: Takes the material from the object you select.

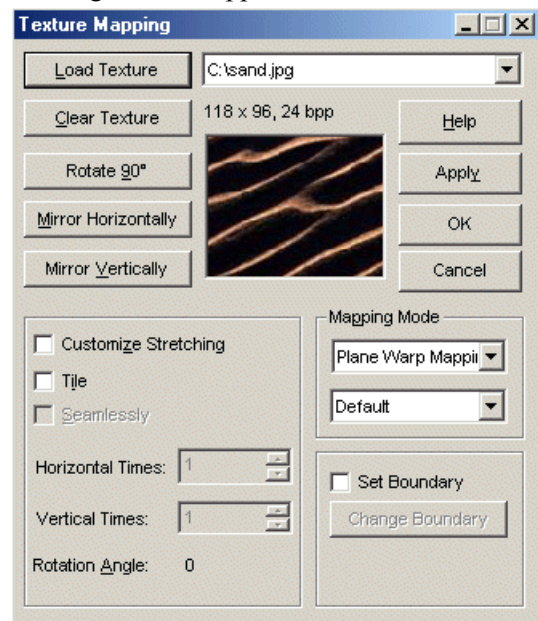
Texture Mapping 3D only

Menu: Tools / Texture Mapping

Copies a texture from an image file (bitmap, gif, jpeg, etc.) and applies that texture to one or more DesignCAD drawing objects. The next time the drawing is shaded using either Gouraud (Medium Quality) or Phong (Best Quality) method in the **Shading** Command (see Shading), all items that have been assigned a texture will be shaded accordingly. Several jpegs are included with DesignCAD as sample textures and are located in the DesignCAD directory.

Using the Command

Select the object to be assigned a texture and choose the **Texture Mapping** command. The following window appears:



Load Texture: Click this button and the **Select Texture** window appears. Make sure the directory that contains the image file (bitmap,

jpeg, etc.) is listed in the **Look In:** box. Select the file from the area below the Look In: box or type the name in the **File Name** box. Click **OK**.

Clear Texture: Removes all texture settings from the currently selected drawing objects.

Rotate 90°: Rotates the image file 90 degrees for the image's use as a texture. This button can be used two times to rotate the image 180 degrees and three times to rotate the image 270 degrees.

Mirror Horizontally: Flips the image so that all portions of the image that were on the left side will now be on the right and vice versa.

Mirror Vertically: Flips the image so that all portions of the image that were on the top of the image will now be on the bottom and vice versa.

Customize Stretching: Allows for the customization of the image's scale and aspect ratio.

Image Scale: Enter the value for the scale at which the image is to be displayed. For example, a value of .5 will display the image at half of its original size and a value of 2 will display the image at two times its original size.

Aspect Ratio: The relationship of the vertical and horizontal scale factors at which the image is to be displayed. For example, a value of .5 will display the image so that the vertical scale is twice that of the horizontal scale, and a value of 2 will display the image so that the horizontal scale is twice that of the vertical scale.

Tile: Places repeated copies (or an array) of the image appear on a single surface. When this option is selected:

Horizontal Times: Enter the number of copies of the image to be displayed from left to right on the selected item(s).

Vertical Times: Enter the number of copies of the image to be displayed from top to bottom on the selected item(s).

Rotation Angle: This value displays the current rotation angle of the image being used as a texture. This value can be increased or decreased by clicking the **Rotate 90** button repeatedly.

Seamlessly: Inverts one image everywhere two images meet, so the edges of the image copies will not be as noticeable.

Top Left	Left Top	Top Left
Left Top	Top Left	Left Top
Top Left	Left Top	Top Left

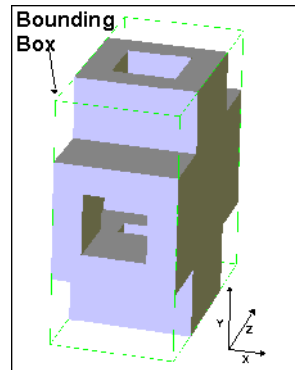
Mapping Mode: The default direction for the texture runs parallel to the Y axis; this direction can be changed for some of the Mapping Modes. Each Mapping Mode has a unique set of options available in the list box directly below the Mapping Mode list box. Each Mapping Mode and option combination works best with a different kind of entity. Some experimentation may be required to determine which Mapping Mode and option works the best for different items in a given drawing.

- **Spherical Mapping:** As its name indicates, this particular mode works best on rounded 3D surfaces. Unlike most of the other

Mapping Modes, Spherical Mapping mode has a single method: Center. Spherical Mapping mode takes the flat image and wraps and stretches it around the selected object. The direction for the texture determines the "poles" of the object. If the Y axis is said to run through the poles of the object, the top and bottom edges of the image are compressed to meet at these poles and are stretched to fit around the "equator" of the object.

- **Grid Patch Mapping:** Divides the image into the number of surfaces the selected object has. Each division of the image is then assigned to a surface. This mode is recommended for items that have a flat face but have been extruded into a 3D object or "grid." This mode has two methods: Grid Only and Grid & Plane.
- **Grid Only:** Maps the selected texture to the "depth" or "grid" portion of the object (the portion of the object that is not the front "face").
- **Grid & Plane:** Maps the selected texture to both the "face" and "grid" of the object.
- **Box Mapping:** Uses a bounding box to map the texture to the selected objects. The bounding box is an invisible box that completely surrounds the selected objects. The front and back of this box run parallel to the XY plane. The left and right sides of this box run parallel to the YZ plane. The top and bottom of this box run parallel to the XZ

plane.



This mode has six methods. Each of these six methods corresponds to a different side of the bounding box. The texture is placed parallel to the specified side of the bounding box and then projected onto the selected object(s).

- **Cylindrical Mapping:** As its name indicates, this particular mode works best on cylindrically-shaped objects. Unlike most of the other Mapping Modes, Cylindrical Mapping mode has a single method: Center Axis. Cylindrical Mapping mode takes the flat image and wraps it around the selected object. The direction for the texture determines the "poles" of the object. If the Y axis is said to run through the "poles" of the object, the image is compressed to meet at these "poles."
- **Plane Warp Mapping:** This Mapping Mode should only be used on plane entities. Plane Warp Mapping mode has two methods. Default method maps the entity using as much of the image as possible without causing a substantial amount of distortion, the Set Boundaries and Change Boundaries options can be used so the entire

image is used. The Cutoff method trims away portions of the image so that it has the same shape as the plane to which it is to be applied.

- **Default:** When this method is chosen, two options appear below it: Set Boundaries and Change Boundaries. The Set Boundaries option can be checked to make the entire image be used when the drawing entity is mapped. The perimeter of the plane is calculated and the points on the plane that make the most sense mathematically are set as the four corners for the texture. The Change Boundaries button can be used to allow four points to be set in the drawing to reset the four corners of the texture.
- **Cutoff:** The Cutoff method of Plane Warp Mapping trims away portions of the image so that it has the same shape as the plane to which it is to be applied. The direction of the texture may be set in the Rotate box.

Animation and Walk Through

These tools are used for displaying a moving view of your 3D model. This is useful for presentations.

Animation Mode

Menu: Animation / Animation Mode

Used to animate or add motion to objects in a DesignCAD drawing.

NOTE: To exit Animation Mode, select it again from the **Animation** menu.

This command is also used to produce animated AVI video files that can be played by Windows Media Player and other video player applications. Animation Command will also produce VRML files, which can be used to animate a drawing for display on the Internet.

DesignCAD animation works on the same principle used by motion pictures and animated cartoons. A series of drawing images are recorded one frame at a time, with the position of the animated object changed slightly from frame to frame. When all the frames are displayed in rapid succession, the animated drawing creates the illusion of motion and continuity for the viewer.

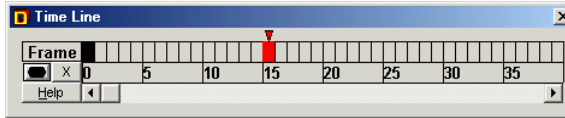
DesignCAD animation is performed by establishing keyframes of a drawing at the start, middle and end of an animated sequence. These keyframes serve as guide posts for the sequence. You must select how much time (how many frames) will pass between keyframes. You must also manipulate and move the drawing object between keyframes. This tells DesignCAD what and how much "movement" to display in relation to various times in the animation sequence.

Using the Command

You should have a drawing loaded when you choose the **Animation Mode** Command.

NOTE: Pay attention to where the selection handles on the animated object are located. The selection handle serves as a reference point for all the movement, rotation, or scale changes that will occur in your animation.

With the drawing open, choose **Animation Mode**. The **Control Panel** and **Time Line** appear.



To set the first keyframe, make sure the box in the **Time Line** above the zero is highlighted in the Time Line (there will be an arrowhead above it) and click the **Set Keyframe** button.

Begin to set the next keyframe by clicking in the box above the number for the next frame that will be a keyframe. Notice that the arrowhead moves above the box you clicked. The boxes that correspond to the keyframes you set (except for the 0 frame) are filled with red, indicating that they are keyframes.

The number of frames left between keyframes determines the size of the animation increments in the final animation. In other words, if you are rotating an object 90 degrees between two keyframes that are ten frames apart (frames 0 and 9), there will be 9 movements. In the final animation, the object will rotate 10 degrees per frame (90 divided by 9 is 10).

TIP: Animation runs at about 15 frames per second, so 30 frames would equal approximately two seconds worth of animation.

Now enter all pertinent motion information for your animated object. This is the amount and type of movement you want to occur from the first

keyframe to the second keyframe. You can use Move, Rotate, Scale or use any combination of these commands.

Once you have moved the object in the desired manner, verify that you have highlighted the box above the number you want to set as the second keyframe and click the **Set Keyframe** button to lock in the keyframe.

Repeat this series of actions as many times as necessary to complete your animation. If you make a mistake and want to delete a keyframe, highlight the box above the keyframe you want to delete in the **Time Line** and click on the **Delete Keyframe** button .

NOTE: Don't forget to record a keyframe any time you change the direction of movement, the direction of a rotation, etc. For example, if you are going to move an object to the left and then up in your animation, make sure you record a keyframe before you move the object up. If you don't record a keyframe after you move the object to the left but wait until after you have moved the object to the left and then up, DesignCAD doesn't know that you wanted to show movement to the left and then upward movement. The object will move diagonally when the animation is played.

Playing and Saving An Animation Sequence

To show or hide the paths of animated objects, click **Show Path**. Use the **Rewind**, **Back/Forward One Frame**, **Stop**, **Play**, and **Fast Forward** buttons to maneuver within the animation.

To play the animation over and over again when it is played, click on the **Loop** button.

When finished, save your work as an animation template. See **Save Animation Template**.

To play your animation, see **Load Animation Template**.

Save Animation Template

Menu: Animation / Save Animation Template

Saves an animation that has been produced in Animation Mode.

Using the Command

After producing an animation in Animation Mode, select the **Save Animation Template** command. Enter a name for the animation in the window and then click OK.

NOTE: The animation template is saved as a part of the drawing file it was created with and can only be opened with the Load Animation Template command if the drawing file is open in DesignCAD.

Load Animation Template

Menu: Animation / Load Animation Template

Opens an animation template that has been produced in **Animation Mode** and then saved as a part of the drawing file with the **Save Animation Template** command. This command is only available when DesignCAD is in Animation Mode.

Using the Command

After producing and saving an animation in Animation Mode, open the drawing file in which it was created. Choose the **Animation Mode** command and choose the **Load Animation Template** command. Select the **Template Name** and click OK.

Export

You can export an animation in two different formats: VRML and AVI.

Menu: Animation / Export / VRML

Menu: Animation / Export / AVI

AVI files are very large, so DesignCAD gives you the option to save the file uncompressed or with one of six other compression ratios. Click the menu arrow and scroll to select a compression format. Select **Full Frames** (Uncompressed) for the best results and then click OK.

Control Panel

Menu: Animation / Control Panel

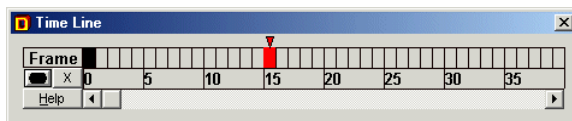
A toggle that hides or shows the Animation Control Panel.



Time Line

Menu: Animation / Time Line

A toggle that hides or shows the Animation Time Line.



Walk Through ^{3D}only

Menu: Animation / Walk Through Mode

Toggles Walk Through Mode on and off. Walk Through Mode can be used to step a viewer around a DesignCAD drawing.

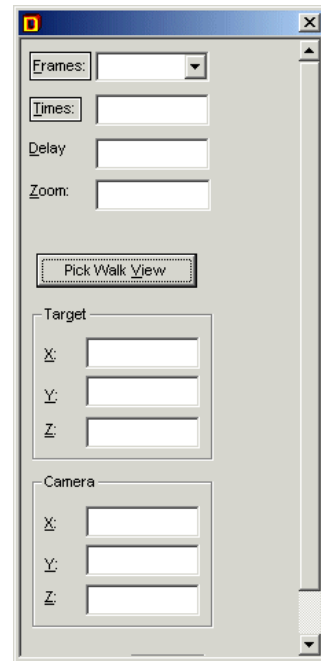
NOTE: To exit Walk Through Mode, select it again from the **Animation** menu.

Like Animation Mode, Walk Through Mode can be used to produce AVI video files that can be played by Windows Media Player and other video player applications. Walk Through Mode will also produce VRML files, which can be used to walk viewers through a drawing on the Internet.

The main difference between Walk Through Mode and Animation Mode is that Walk Through Mode is only used to change the view of a drawing; items in the drawing cannot be moved or rotated independent of one another. A series of view angles are changed slightly from one frame to another. When all the frames are displayed in rapid succession, the view angle of the viewer moves smoothly around the drawing.

Using the Command

With a drawing open, choose **Walk Through Mode**. The **Control Panel** and **Walk** window appear.



To set the first keyframe, click on the **Frames** button and select **Add Frame** from the menu that appears. Set the amount of time (in seconds) to pass between this first frame and the next one in the **Times** box. Set the amount of time to pass (in seconds) before movement toward the next keyframe begins in the **Delay** box. You may also want to change the apparent size of the drawing by entering a new value in the **Zoom** box. Then use the **Target** and **Camera** areas of the window. The **Target Area** is used to specify what area of the drawing is being focused on (displayed in the center of the view window). The **Camera** area is used to specify the viewing location and thus controls the view angle and distance. Finally set the number of frames per second to be inserted between this frame and the next one in the **Frames per Second** box.

To add the next keyframe, click on the **Frames** button and select **Add Frame** from the menu that appears. Change the amount of time to pass between this frame and the next one or leave it alone so it will be the same as the amount of time between the first frame and this one. Change any or all of the other options. When the view is the way you want it for the second keyframe, add another one.

Repeat this series of actions as many times as necessary to complete your walk through. If you make a mistake select the keyframe you want to change in the **Frames** list box and change the values.

Playing and Saving A Walk Through Sequence

Use the **Rewind**, **Back/Forward One Frame**, **Stop**, **Play**, and **Fast Forward** buttons to maneuver within the walk through.

To play the walk through over and over again when it is played, click on the **Loop** button.

When finished, save your work as an walk through template. See **Save Walk Template**. The walk through template will be saved as a part of the drawing file.

To play your animation, see **Load Walk Template**.

Export

You can export a walk through in two different formats: VRML and AVI.

Menu: Walk / Export / VRML

Menu: Walk / Export / AVI

AVI files are very large, so DesignCAD gives you the option to save the file uncompressed or with one of six other compression ratios. Click the menu arrow and scroll to select a compression format. Select **Full Frames** (Uncompressed) for the best results and then click OK.

Run Walk

Menu: Walk / Run Walk

Begins the playback of a walk template created with Walk Through Mode.

Control Panel

Menu: Walk / Control Panel

A toggle that hides or shows the Control Panel.



5 Annotation

Dimensions

Dimensions are used to indicate actual dimensions of your entities.

NOTE: If you want to measure entities without including the dimension in the drawing, see Measuring.

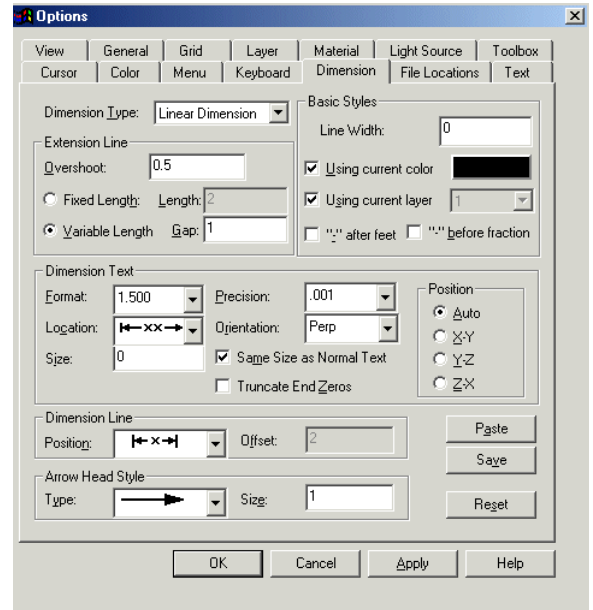
Dimension Options

Menu: Options / Options

Controls the many options in the dimension commands, plus the **Pullout** and **Balloon** commands, all from one menu.

These settings become global - they are the defaults for each dimension type. You can change settings within the dimension command itself.

Open the Dimension tab and click in the **Dimension Type** box and select the type of dimension command for which you want to set the options.



Using Current Color: Dimension text and extension lines are drawn with the current drawing color. If you are drawing with several different colors and want all dimension text to be drawn with the same color, uncheck this option and click the **Color** box to invoke the **Color Palette**.

Using Current Layer: Sets the layer in which the dimension text and extension lines are drawn. When checked, all dimension text is drawn in the active layer. To draw all of the dimension text in

the same layer of a multi-layer drawing, uncheck this option and click the **Layer** box to select the layer for dimension text.

Paste: After setting the options for a particular dimension command, click this button to apply the same options to similar dimensioning commands. If you have set the options for a dimensioning command that measures an angular dimension, the same options will be used for all dimensioning commands that make angular measurements. Likewise, if you have set the options for a dimensioning command that measures a linear dimension, the same options will be used for all dimensioning commands that make linear measurements.

Save: If you want to save the changes to the next drawing session, click **Save** before you click OK.

Reset: Restores all of the options to their default values.

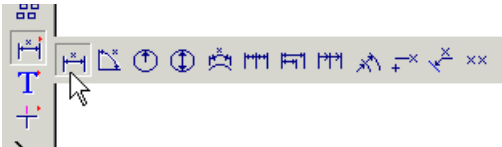
*NOTE: For more details on dimension settings, see the command description for each dimension command. Most are listed for the **Dimension** command.*

Dimension

Menu: Dimension / Dimension

Shortcut Key: @

Toolbox Icon:



Point 1: Start of distance to be measured

Point 2: End of distance to be measured

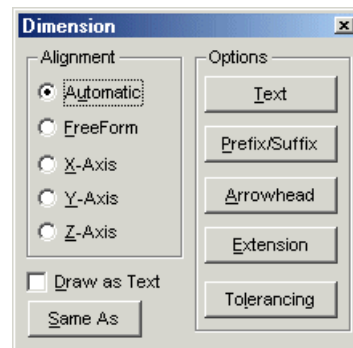
Point 3: Dimension text location

Places dimensions in your drawing.

Using the Command

Choose the **Dimension** command. To add a dimension to your drawing, set two points for the distance to be measured. Then set a third point for the dimension text location.

Several options are available in the **Dimension** window:



Automatic: DesignCAD determines automatically which axis you wish to measure, based on where you place the dimension text.

Free Form: Enables you to measure along any arbitrary direction. The dimension text will always be placed parallel to the distance measured.

X-Axis, Y-Axis, Z-Axis: Selecting any of these causes DesignCAD to measure only the distance parallel to that axis along the measured path.

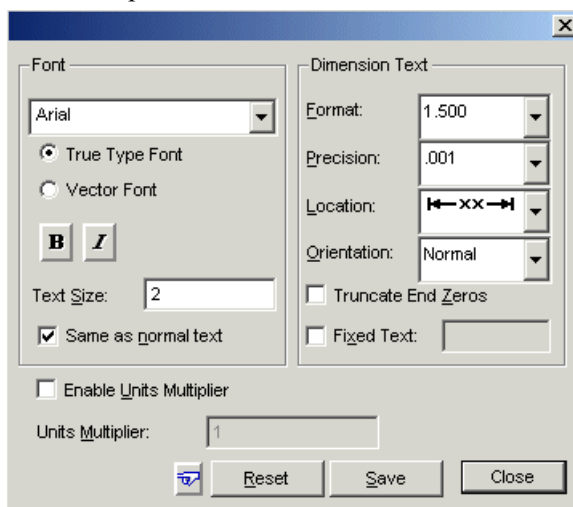
Draw as Text: The default setting (**Draw as Dimension**) causes DesignCAD to update the dimension text anytime the dimension changes size (e.g., when resetting units, or stretching an

object and its associated dimension). Checking **Draw as Text** forces the program to keep the same dimension, regardless of changes.

Same As: Uses properties from another dimension in the drawing.. Click **Save As**, then select the dimension.

Text

Click the **Text** button on the **Dimension** window for these options:



Any Windows font can be used for dimensions. The dimension text size can be specified in Drawing Units.

Format: Offers several different numeric formats for the dimensions. The Shift+F12 keys may be used to toggle through the dimension formats without opening the **Dimension Text** window.

Precision: The number of digits to the right of the decimal point.

Location: The dimension text can be placed inside or outside the extension line, to the right or left of the dimension arrows, or between the terminators. The F12 key may be used to toggle through the text locations without opening the **Dimension Text** window.

Orientation: Choose how to orient the dimension text. The Ctrl+F12 keys may be used to toggle through the text orientations without opening the **Dimension Text** window. The Shift+Spacebar keys may be used to "flip" the text.

Truncate End Zeros: Zeros at the end of a decimal will be removed. For example, 2.50 inches becomes 2.5 inches.

Fixed Text: Fixed text will not change even if you re-scale the object or resize the drawing. Fixed text is useful when you want the dimension's text to always read the same value whether you rescale it or not. It can also be useful if you have a drawing that is not to scale, but you want the dimension to show what the actual measurement should be.

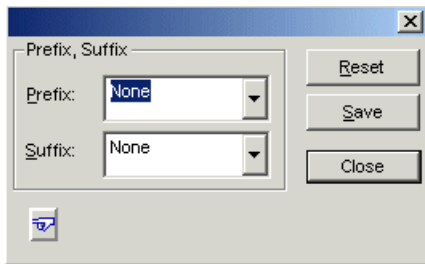
Enable Units Multiplier: Allows you to dimension in multiple base units. For example, if your current drawing is based on inches, and you want to dimension in centimeters, check the "Enable Units Multiplier" box and set the Units Multiplier value to 2.54 (because there are 2.54 centimeters per inch). Optionally you might wish to open the **Prefix/Suffix** options and specify a suffix of "cm". Now if you dimension a 10-inch line, the dimension text will report 25.4 cm.

NOTE: This works best if you choose the simple decimal or fractional text format. If you choose a feet-and-inches format in combination with the Units multiplier, you may get unexpected values.

Same As (hand icon): Click to copy the properties of an existing dimension.

Prefix/Suffix

Click the **Prefix/Suffix** button on the **Dimension** window for these options:



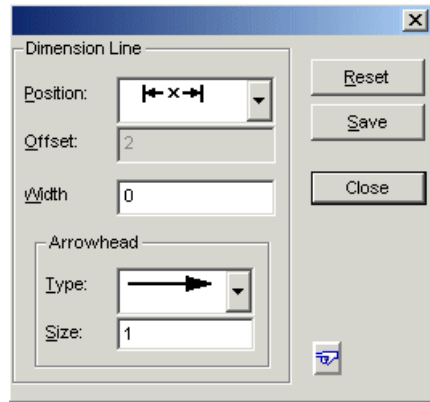
Prefix: Adds a notation before the dimension measurement. You can keep a list of up to five custom notations that are available for dimensioning.

Suffix: Adds a notation after the dimension measurement. You can keep a list of up to five custom notations that are available for dimensioning.

Same As (hand icon): Click to copy the properties of an existing dimension.

Arrowhead

Click the **Arrowhead** button on the **Dimension** window for these options:



Position: The dimension line can be positioned inside or outside the extension line.

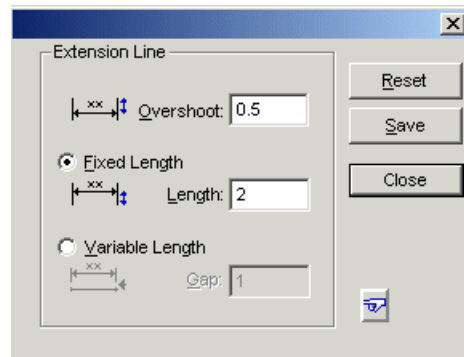
Offset: Enter the amount of offset.

Arrowhead: Use the **Type** and **Size** boxes to specify the arrowhead options you want to use.

Same As (hand icon): Click to copy the properties of an existing dimension.

Extension

Click the **Extension** button on the **Dimension** window for these options:



The extension line is perpendicular to the dimension line and is drawn at the end of the dimension line.

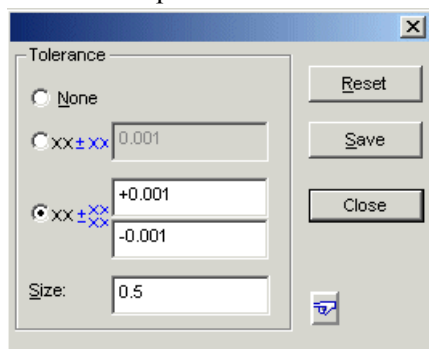
Overshoot: The distance the extension line extends past the dimension line.

Gap: The distance between the points set for dimensioning and the beginning of the extension line.

Same As (hand icon): Click to copy the properties of an existing dimension.

Tolerancing

Click the **Tolerancing** button on the **Dimensions** window for these options:



The dimension tolerance is the positive and negative values showing the amount of variation allowed from the standard measurement.

None: Allows no tolerance for the dimension, meaning that it must be exact.

The second option allows you to set one value that will be used as the positive and negative dimension tolerance.

The third option allows you to set separate values for the positive dimension tolerance and the negative dimension tolerance.

Same As (hand icon): Click to copy the properties of an existing dimension.

NOTE: You can set dimension parameters globally in the program **Options**. See *Dimension Options*.

Dimension Angle

Menu: Dimension / Dimension Angle

Toolbox Icon:



Point 1: Center of the angle to be measured

Point 2: Beginning of the angle

Point 3: Endpoint of the angle

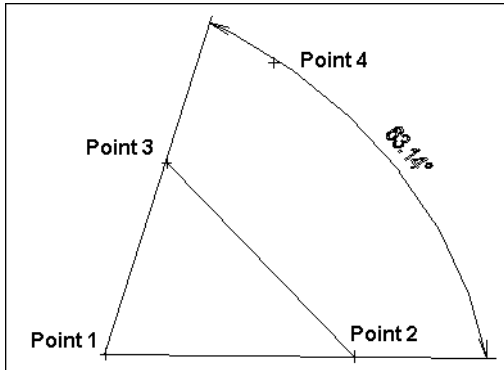
Point 4: Distance at which the dimension text is to be inserted

Adds the dimensions of angles to your drawing.

Using the Command

Choose the **Dimension Angle** command. Set a point for the center of the angle to be measured. Then set points for the beginning and end of the angle. Selection order is important, because angles are measured counter-clockwise.

Finally, set a point to fix how far away the dimension is to be from the angle. The angle is measured and the dimension is added to your drawing.



NOTE: The options available in this command are the same ones available in the **Dimension** command. See *Dimension*. You can also set dimension parameters globally in the program **Options**. See *Dimension Options*.

Dimension Arc

Menu: Dimension / Dimension Arc

Point 1: Arc to be dimensioned

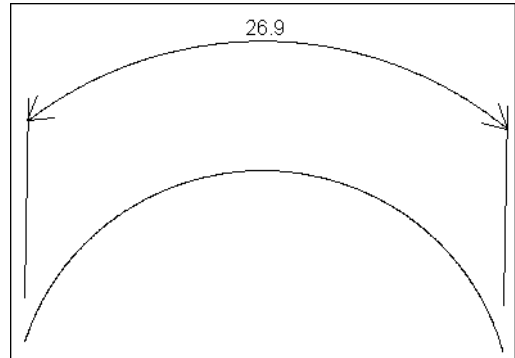
Point 2: Text location

Dimensions the arc length of an arc from endpoint to endpoint.

Using the Command

Choose the **Dimension Arc** command. Set a point on the arc to be measured. After the point is set, a rubber-band line shows how the dimension will be drawn. Set a point for the location of the

dimension text. The length of the arc will be calculated and inserted into the drawing at the last point.

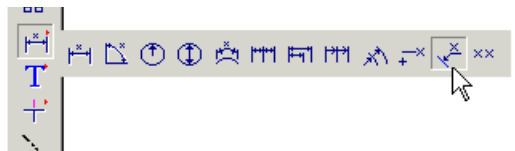


NOTE: The options available in this command are the same ones available in the **Dimension** command. See *Dimension*. You can also set dimension parameters globally in the program **Options**. See *Dimension Options*.

Dimension Chamfer

Menu: Dimension / Dimension Chamfer

Toolbox Icon:



Point 1: Line to be dimensioned

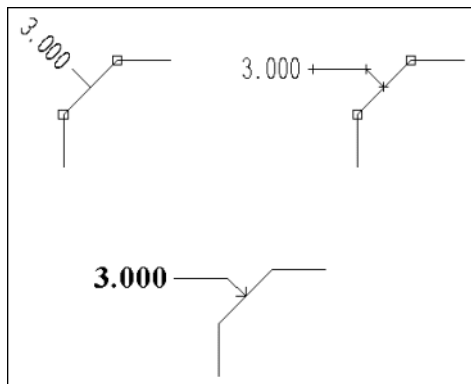
Point 2-n: Extension line

Point n+1: Text location

Draws the dimension for a chamfered line. This calculated distance is the vertical or horizontal distance between the two corners of the chamfer.

Using the Command

Choose the **Dimension Chamfer** command and set a point on the chamfer line. A rubber-band line shows how the dimension will be drawn. Set one or more points for the extension line. Set a point for the text location. Click the middle mouse button or press Enter to end the command. The dimension information for the chamfer is drawn at the last point.

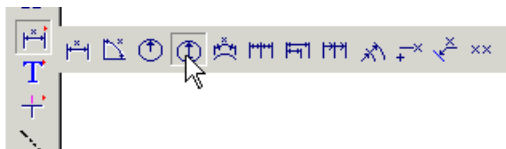


NOTE: The options available in this command are the same ones available in the **Dimension** command. See *Dimension*. You can also set dimension parameters globally in the program **Options**. See *Dimension Options*.

Dimension Diameter

Menu: Dimension / Dimension Diameter

Toolbox Icon:



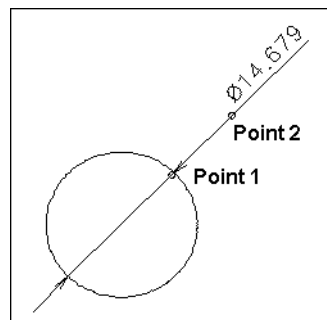
Point 1: Circle or arc to be dimensioned

Point 2: Location for dimension text

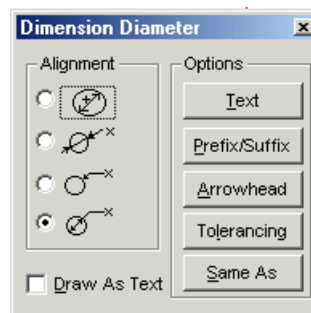
Draws a diameter dimension for a circle or arc. The text and arrowhead can be drawn outside the circle, inside the circle, or with an extension line.

Using the Command

Choose the **Dimension Diameter** command and set a point on the circle or arc to be dimensioned. After the first point is set, a rubber-band line shows how the dimension will be drawn. Set a point for the location of the text information. The diameter of the circle is displayed at the last point.



The **Dimension Diameter** window has four options for the extension lines and arrowheads.



Text Inside Circle: Draws the dimension text and arrowheads inside the circle or arc.

Pre-Defined Extension Line: Draws the dimension text and arrowheads outside the arc or circle.

Custom Extension Line: Draws the dimension text and arrowheads outside the arc or circle with a custom extension line.

Text Inside Circle: Draws the arrowheads inside the arc or circle.

NOTE: *The other options available in this command are the same ones available in the **Dimension** command. See *Dimension*. You can also set dimension parameters globally in the program **Options**. See *Dimension Options*.*

Dimension Radius

Menu: Dimension / Dimension Radius

Toolbox Icon:



Point 1: Circle or arc to be dimensioned

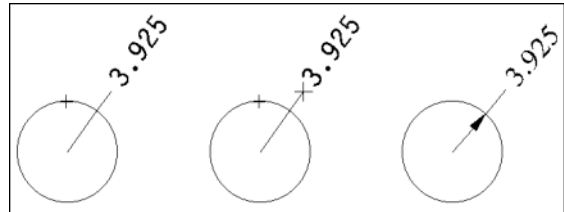
Point 2: Location for dimension text

Dimensions the radius of a circle or arc.

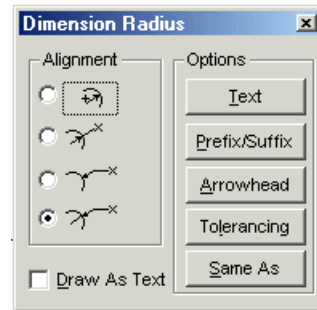
Using the Command

Choose the **Dimension Radius** command and set a point on the circle or arc to be dimensioned. After the first point is set, a rubber-band line

shows how the dimension will be drawn. Set a point for the location of the text information. The radius of the circle is displayed at the last point.



The **Dimension Radius** window has four options for the extension lines and arrowheads.



Text Inside Circle: Draws the dimension text and arrowheads inside the circle or arc.

Pre-Defined Extension Line: Draws the dimension text and arrowheads outside the arc or circle.

Custom Extension Line: Draws the dimension text and arrowheads outside the arc or circle with a custom extension line.

Text Inside Circle: Draws the arrowheads inside the arc or circle.

NOTE: The other options available in this command are the same ones available in the **Dimension** command. See *Dimension*. You can also set dimension parameters globally in the program **Options**. See *Dimension Options*.

Dimension Coordinate

Menu: Dimension / Dimension Coordinate

Shortcut Key: F11

Toolbox Icon:



Point 1: Reference point

Point 2: Point to be dimensioned

Point 3: Location for dimension text

Draws coordinate dimensions relative to a base point. The first point is the origin, or reference point, from which subsequent points are dimensioned. After the base point is set, pairs of points are set. The first point of each pair is the measurement point; the second is the text location.

If the second point in the pair of points is set above or below the first point, the X-value, or horizontal distance, from the reference point will be placed in the drawing. If the second point is set to the right or left of the first point, the Y-value, or vertical distance, from the reference point will be placed into the drawing. Finally, if the second point in a pair of points is set closer or further away along the Z-axis, the relative Z coordinate is placed into the drawing.

Using the Command

Choose the **Dimension Coordinate** and set a point for the origin, or reference point. Set a point at the coordinate to be measured. Set a point for the text location. Press Enter to end the command.

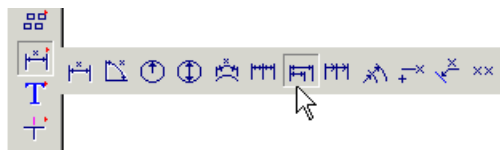
The dimension is measured horizontally, vertically, or along the Z axis from the first point to the second point depending on the relative location of the third point to the second. The dimension text is drawn at the last point.

NOTE: The options available in this command are the same ones available in the **Dimension** command. See “*Dimension*” on page 160. You can also set dimension parameters globally in the program **Options**. See “*Dimension Options*” on page 159.

Dimension Baseline

Menu: Dimension / Dimension Baseline

Toolbox Icon:



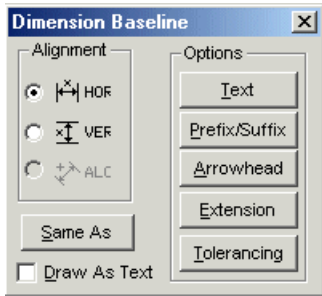
Point 1-n: Points to be measured

Point n+1: Location for dimension information

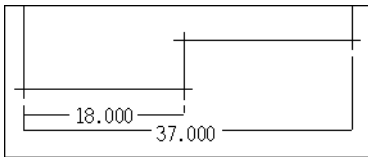
Draws dimensions using several points along a baseline. Points are set for each measurement position. The distance measured is the vertical or horizontal distance to the measurement points. Each distance is measured from the first point.

Using the Command

Choose the **Dimension Baseline** command and select **HOR** or **VER** to determine whether the horizontal or vertical distance will be measured.



Next set a point for the base measurement point. Set points for the other measurement points. A rubber-band line shows how the dimension will be drawn. Set a point for the location of the dimension text. Click the middle mouse button or press Enter. The dimension is drawn using the points.



*NOTE: The other options available in this command are the same ones available in the **Dimension** command. See *Dimension*. You can also set dimension parameters globally in the program **Options**. See *Dimension Options*.*

Dimension Extended

Menu: Dimension / Dimension Extended

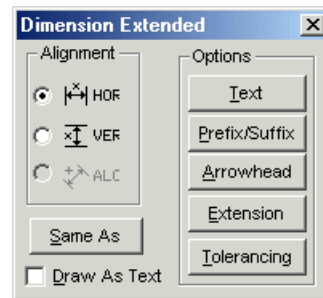
Point 1-n: Points to be measured

Point n+1: Location for dimension information

Draws successive dimensions extended along several points. Points are set for each measurement position. The distance measured is the vertical or horizontal distance between the measurement points. Each distance is measured separately between each successive pair of points.

Using the Command

Choose the **Dimension Extended** command and select **HOR** or **VER** to determine whether the horizontal or vertical distance will be measured.

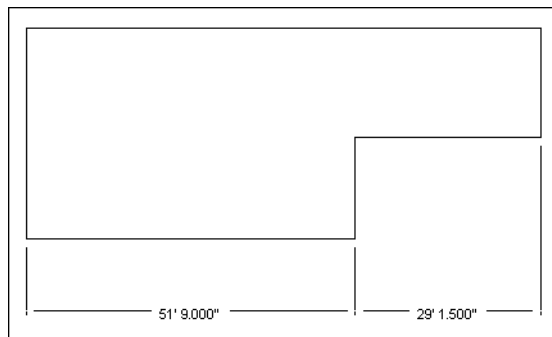


Next, set a point for each measurement position. Then set a point for the location of the dimension text. Press Enter to end the command. Dimensions will be drawn for the measurement points.

Example: Dimension a wall.

Select the **Dimension Extended** command and choose the **HOR** option. Set a point on the bottom-left corner of the wall. Set a second point near the middle of the wall, along the bottom. Set a third point on the bottom-right corner of the

wall. Set a fourth point for the text below the wall. Press Enter. The dimension is added to the drawing in two sections.



NOTE: *The other options available in this command are the same ones available in the **Dimension** command. See Dimension. You can also set dimension parameters globally in the program **Options**. See Dimension Options.*

Dimension Progressive

Menu: Dimension / Dimension Progressive

Toolbox Icon:



Point 1: Base point

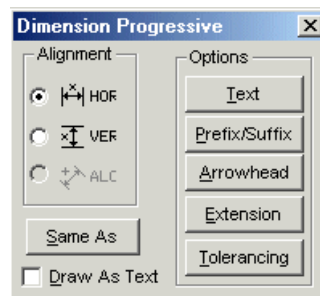
Points 2-n: Points to be measured

Point n+1: Location for text

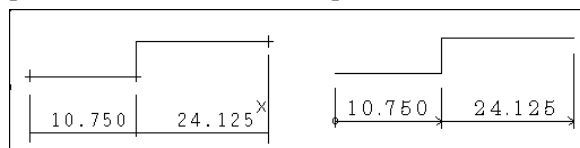
Draws the dimensions of a line progressively from a base point. The distance measured is the vertical or horizontal distance between each measurement point and the base point.

Using the Command

Choose the **Dimension Progressive** command. Select **HOR** or **VER** to determine whether the horizontal or vertical distance will be dimensioned.



Set a point for the base measurement point. Set points for the other measurement points. A rubber-band line shows how the dimension will be drawn. Finally, set a point for the location of the dimension text. Then press Enter to end the command. The dimension is drawn from the base point to each measurement point.



NOTE: *The other options available in this command are the same ones available in the **Dimension** command. See Dimension. You can also set dimension parameters globally in the program **Options**. See Dimension Options.*

Dimension Radius Progressive

Menu: Dimension / Dimension Radius Progressive

Toolbox Icon:



Point 1: Base point

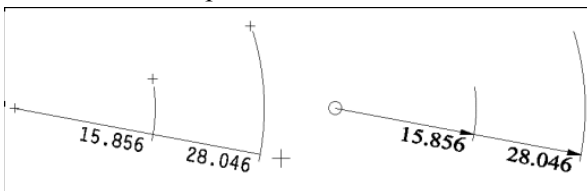
Points 2-n: Points to be measured

Point n+1: Location for text

Draws radius dimensions progressively. Points are set for each measurement position. The dimension of the radius is calculated and inserted into the drawing using progressive measurements. Each distance is measured from the base point.

Using the Command

Choose the **Dimension Radius Progressive** command and set a point for the base point. Set points for the other measurement points. Set a point for the text location. Click the middle mouse button or press Enter.

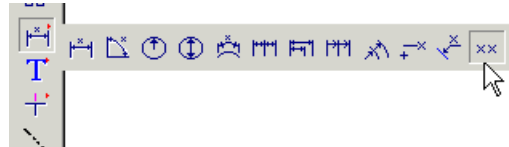


NOTE: The options available in this command are the same ones available in the **Dimension** command. See *Dimension*. You can also set dimension parameters globally in the program **Options**. See *Dimension Options*.

Dimension Distance Only

Menu: Dimension / Dimension Distance Only

Toolbox Icon:



Point 1: Beginning of distance to be measured

Point 2: End of distance to be measured

Point 3: Location of dimension text

Measures a distance horizontally, vertically, or at any angle, but does not add extension lines or arrowheads.

Using the Command

Choose the **Dimension Distance Only** command and choose which direction you wish to measure. Set a point for the beginning of the dimension. After the first point is set, a rubber-band box shows where the dimension text will be drawn. Set a point for the end of the dimension. Set a point for the location of the dimension text. The dimension is drawn without arrows or extension lines.

NOTE: The options available in this command are the same ones available in the **Dimension** command. See *Dimension*. You can also set dimension parameters globally in the program **Options**. See *Dimension Options*.

Dimension Center Mark

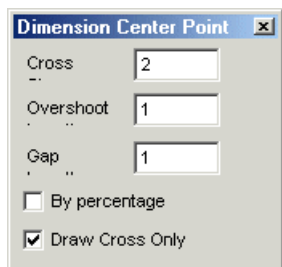
Menu: Dimension / Dimension Center Mark

Point 1: Point on circle or arc to be marked

Draws a center mark for a circle or arc.

Using the Command

Choose the **Dimension Center Mark** command.



Enter the size for the center mark in the **Cross** box. If the **By Percentage** option is enabled, the size of the center mark will be a percent value of the size of the circle/arc. If **By Percentage** is disabled, the size of the center mark will be in Drawing Units.

The **Overshoot** and **Gap** options are only available if **By Percentage** is disabled.

To draw only the center mark (no extension lines), enable the **Draw Cross Only** option.

NOTE: You can set dimension parameters globally in the program **Options**. See *Dimension Options*.

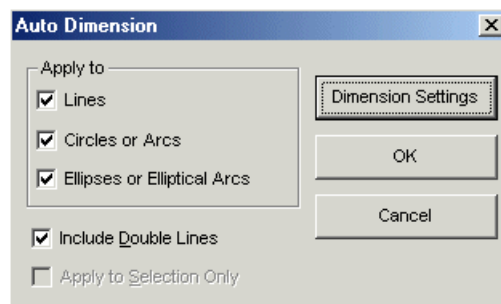
Auto Dimensions 2D only

Menu: Dimension / Auto Dimension

Automatically dimension lines, circles, arcs, ellipses, and elliptical arcs.

Using the Command

Choose the **Auto Dimension** command. In the window, select the entity types you want dimensioned.



To dimension only selected entities, check **Apply to Selection Only**.

Dimension Settings: To view or change dimension settings, click this button to open the **Dimension Options** (see *Dimension Options*).

Click **OK** to return to the drawing, and the selected entity types will be dimensioned.

Units

Menu: Dimensions / Units

Shortcut Key: U

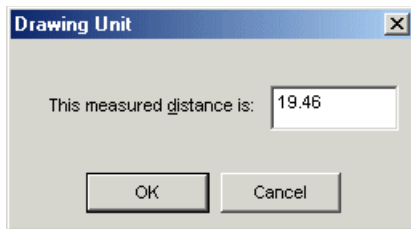
Point 1: First point on distance to measure.

Point 2: Second point on distance to measure.

Measures the distance between two points and, if you choose, can change the distance. This command can also be used to change the units of measurement in the drawing or to set up the initial drawing space.

Using the Command

Choose the **Units** command. Set two points on the screen for a known distance. In the **Units** box, you can accept the measured distance or change it.



When you change the value, it changes the entire coordinate system of the drawing, including any dynamic dimensions in the drawing.

Text and Attributes

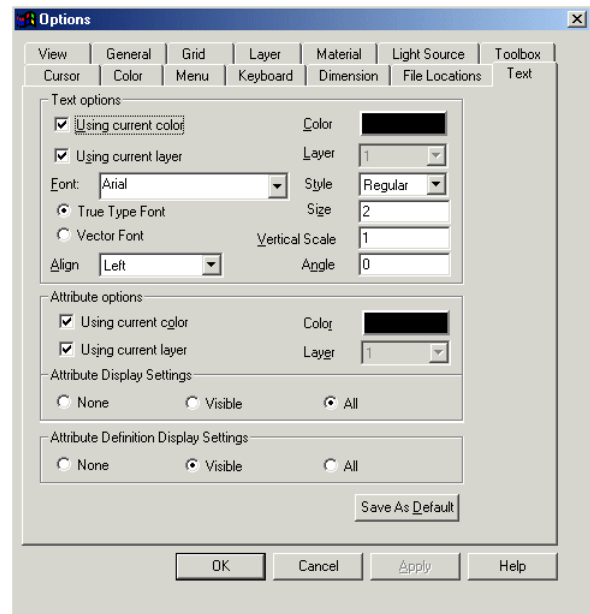
Text commands are used to place text strings in the drawing. Attributes are used to assign properties (material, cost, etc.) to objects.

Text (and Attribute) Options

Menu: Options / Options

Shortcut Key: Q

In the **Text** tab of the **Options**, you can change the color and style of regular text and Attribute text for the different layers in your drawing.



Text Options:

- **Using Current Color:** Text is drawn with the current drawing color. If you are drawing with several different colors and want all text to be drawn with the same color, uncheck this option and click the **Color** box to select the color you want to use for text.
- **Using Current Layer:** All text (excluding Dimension and Attribute) is drawn in the active layer. To draw all of the text in the same layer of a multi-layer drawing, uncheck this option and click the Layer box to select the layer you want for text.
- **Font:** Select the font you want to use for text. The available fonts depend on whether **True Type Font** or **Vector Font** is selected.

- **Align:** Placed text to by the left edge, right edge, or center.
- **Style:** Choose Regular, Bold, Italic or Bold Italic.
- **Size:** The size of the text.
- **Vertical Scale:** The height of the letters relative to their width. Enter a large value for tall, thin letters or a small value for short, wide letters.
- **Angle:** The angle at which the text will be placed if only one point is set.

Attribute Options

- **Using Current Color, Using Current Layer:** Same as for **Text Options** described above.

Attribute Display Settings:

- **None:** No attributes will be displayed, regardless of the status of their respective **Invisible** options.
- **Visible:** Visibility of attributes will be determined by the status of their respective **Invisible** options.
- **All:** All attributes will be displayed, regardless of the status of their respective **Invisible** options.

NOTE: For details on the **Invisible** option, see *Attribute*.

Attribute Definition Display Settings

- **None:** No attributes definitions will be displayed, regardless of the status of their respective **Invisible** options.

- **Visible:** Visibility of attributes definitions will be determined by the status of their respective **Invisible** options.
- **All:** All attributes definitions will be displayed, regardless of the status of their respective **Invisible** options.

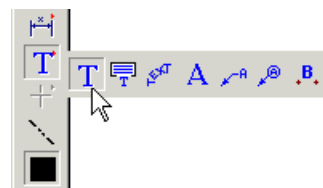
Save as Default: Saves the changes to the next session

Text

Menu: Draw / Text

Shortcut Key: T

Toolbox Icon:



Point 1: Lower-left corner

Point 2: Lower-right corner (optional)

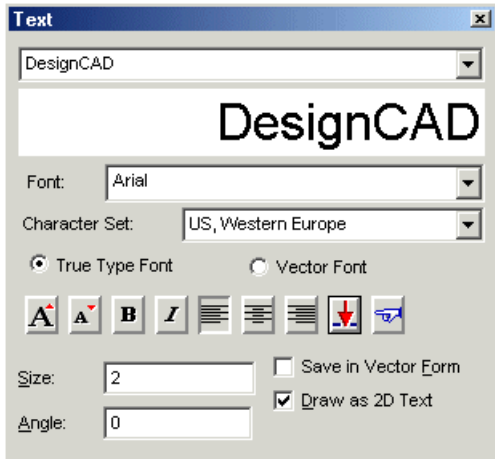
Point 3: Plane on which the text lies (optional)

Point 4: Slant and/or twisting of text (optional)

Inserts a string of text into the drawing. This text can be set to appear flat regardless of the viewing angle, or it can be placed at any 3D orientation so that its appearance varies depending on the current viewing parameters.

Using the Command

Choose the **Text** command and enter the text string in the window. DesignCAD stores a history of the last 20 entries.



Set the **Font** and **Character Set**. Available fonts depend on whether **True Type Font** or **Vector Font** is selected.

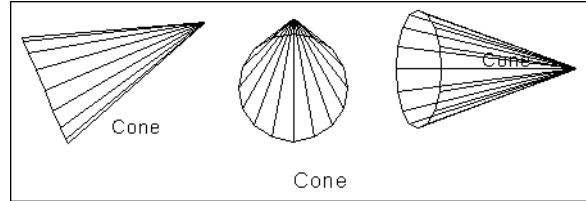
Size: Enter the font size here.

Angle: The angle of text, measured from horizontal.

Save in Vector Form: The text will be drawn as separate vector entities in the shape of each letter. Text drawn in this manner cannot be edited as normal text.

Draw as 2D Text: If used, set a point for the bottom of the text. Next, set a point to determine the angle of the text, or click the middle mouse button or press Enter to have the text drawn using the angle set in the window. The text will be


drawn according to the points and the options selected. The text will appear flat regardless of the viewing angle being used.





If **Draw as 2D Text** is not enabled, you can set one to four points. If only one point is used, the size and angle of the text is determined by the text size and angle in the window.


NOTE: To convert 2D to 3D text and vice-versa, see *Text Convert*.

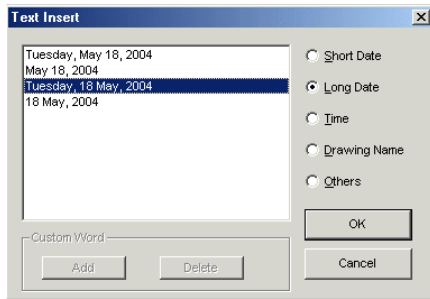
These icons provide additional options on the **Text** window:

 Increases or decreases the text size.

 Make the text bold and/or italicized.

 Placed the text by its left edge, center, or right edge.

 Inserts standard text:



If you want to insert standard text other than date, time, or drawing name, click **Others**, then click **Add**.



Uses properties from another selected text string.

Point Selection:

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.

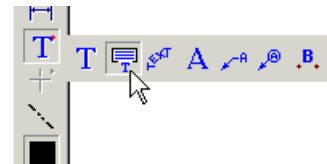
The third point, if used, defines the plane on which the text is to be drawn and partially determines the slant for the text. For example, if the third point is above the first, the text will be upright. If the third point is "behind" the first, then the text will be positioned to be read from above. It will be "flat" on the X-Z plane. If the third point is above and to the left of the first point, the top of the text will slant to the left.

The fourth point, if used, further defines the plane on which the text is to be drawn and partially determines the slant for the text. If four points are used, they essentially specify the positions of the corners of the text.

Text Block

Menu: Draw / Text Block

Toolbox Icon:



Point 1: Top of the Text Block

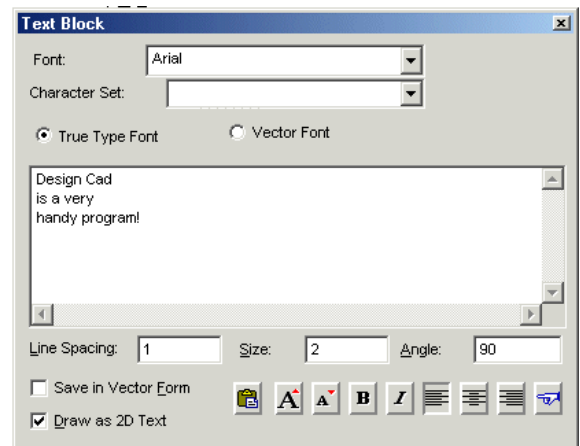
Point 2: Point determining angle of the Text Block (optional)

Point 3: Plane on which the text lies (optional)

Draws multi-line text. Points determine the position and angle of the text in the drawing.

Using the Command

Choose the **Text Block** command and enter the text block in the window. Press Enter to add a new line.



Set the **Font** and **Character Set**. Available fonts depend on whether **True Type Font** or **Vector Font** is selected.

Line Spacing: Vertical spacing between text lines.

Size: Enter the font size here.

Angle: The angle of text, measured from horizontal.

Save in Vector Form: The text will be drawn as separate vector entities in the shape of each letter. Text drawn in this manner cannot be edited as normal text.

Draw as 2D Text: If checked, set a point for the top of the text. Next, Set a point to determine the angle of the text, or click the middle mouse button or press Enter to have the text drawn using the angle set in the **Angle** field. The text will be drawn according to the points and the options selected. The text will appear flat regardless of the viewing angle being used.

If **Draw as 2D Text** is not enabled, you can set one to three points. If only one point is used, the angle of the text is determined by the **Angle** field.

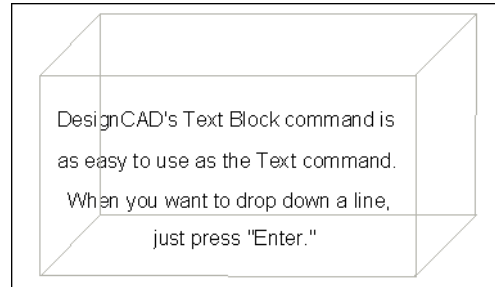
NOTE: *For other options on this window, see Text.*

The first point is used to place the location of the text.

The second point is used to specify the angle of the text.

The third point, if used, defines the plane on which the text is to be drawn. For example, if the third point is below the first, the text will be

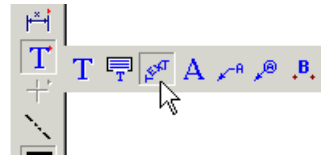
upright. If the third point is in front of the first, then the text will be positioned to be read from above. It will be "flat" on the X-Z plane.



Text Arc

Menu: Draw / Text Arc

Toolbox Icon:



Point 1: Start of the text arc

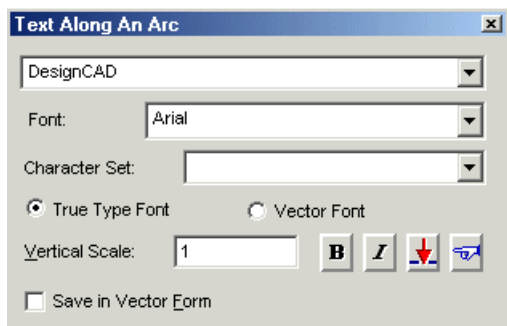
Point 2: Center of the text arc

Point 3: End of the text arc

Draws text along an arc. The arc is defined by a beginning point, a center point, and an endpoint.

Using the Command

Choose the **Text Arc** command and enter the text in the window.

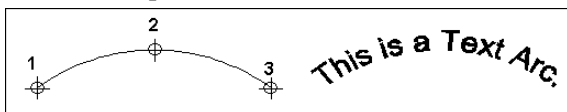


Set the **Font** and **Character Set**. Available fonts depend on whether **True Type Font** or **Vector Font** is selected.

Vertical Scale: The vertical scale factor of the text. A large value will make tall, thin letters. A small value will make short, wide letters.

NOTE: *For other options on this window, see Text.*

Set a point for the beginning of the text arc. Set a point for the center of the text arc. Set a point for the end of the text arc. The text will be drawn in an arc, beginning with the first point and ending with the last point.

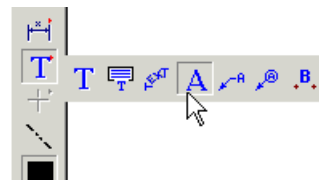


Attribute

Menu: Draw / Attribute

Shortcut Key: \$

Toolbox Icon:

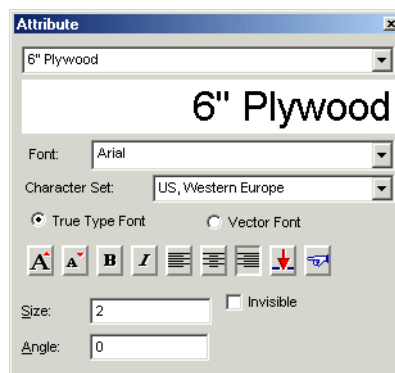


Point 1: Location of the attribute

Assigns information to objects in the drawing. Later you can use the **Material List** command to list the types and numbers of attributes you have placed in your drawing.

Using the Command

Select the **Attribute** command and enter the text string in the window. DesignCAD stores a history of the last 20 entries.



Invisible: Controls whether attributes are displayed. Visibility will be as indicated here, if the **Attribute Display Setting** in the **Text Options** (see Text (and Attribute) Options) is set to **Visible**. If disabled, the attribute can only be hidden if the **Attribute Display Setting** is set to **None**. If checked, the attribute can only be made visible if the **Attribute Display Setting** is set to **All**.

Size and Angle: Font size and angle of text.

For details on the other options on this window, see Text.

Set a point for the text. The Attribute is inserted in the drawing.

Attribute Definition

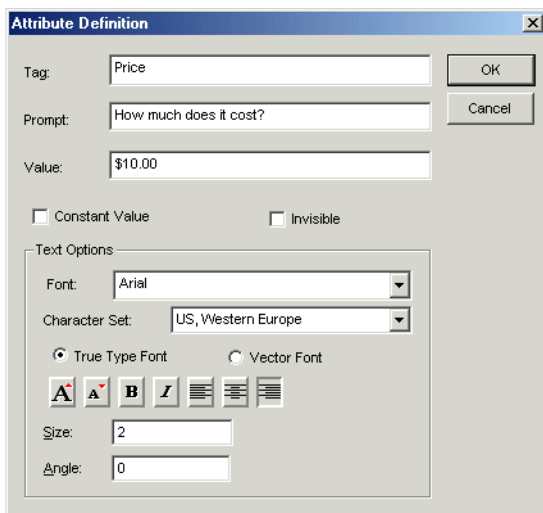
Menu: Draw / Attribute Definition

Point 1: Location of the attribute definition

Assigns constant or variable information to a block. Later you can use the **Material List** command to list the types and numbers of attributes you have placed in your drawing.

Using the Command

Select the **Attribute Definition** command. In the **Tag** field, enter an identifier for the block with which the attribute will be associated.



In the **Prompt** field, enter a question to be asked when the block is inserted in the future. In the **Value** field, enter a default value or answer to the prompt.

Constant Value: If checked, you will not be prompted and the default value will be used.

Invisible: Controls whether attribute definitions are displayed. Visibility will be as indicated here, if the **Attribute Definition Display Setting** in the **Text Options** (see Text (and Attribute) Options) is set to **Visible**. If disabled, the attribute can only be hidden if the **Attribute Definition Display Setting** is set to **None**. If checked, the attribute can only be made visible if the **Attribute Definition Display Setting** is set to **All**.

*NOTE: The **Invisible** option has no effect on stand-alone attribute definitions. Attribute definitions that have not been defined as part of a block using the **Block Define** command will be visible at all times unless they are on a hidden layer. See **Block Define**.*

For details on the other options on this window, see Text.

Set a point to place the attribute definition. The attribute definition is inserted in the drawing.

To automatically assign an attribute definition to a block when it is inserted in the drawing, the **Block Define** command must be used to define the attribute definition as part of the block. This is easier than it sounds. Just select the drawing entities to be defined as a block and the attribute definition to be associated with it. Select the **Block Define** command, name the block and click OK. Now when the block is inserted into the

drawing using **Block Insert**, the attribute definition will automatically be associated with it and you will be prompted for a value.

Material List

Menu: File / Material List

Displays a list of all the attributes and their quantities in the current drawing. This list can be printed or saved to a file.

Attributes generally refer to physical materials-lumber, bolts, screws, and other parts-used to build the object depicted in a drawing. The Material List can be used for cost estimating, parts lists, and other functions that require a list of items used in a drawing.

*NOTE: This command (in the **File** menu) should not be confused with the Material List in the Material Toolbox. "Material," as it is referred to in the Material Toolbox, affects the appearance of a shaded drawing.*

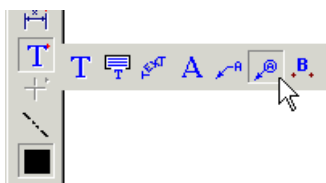
Labels

Label commands are used to assign text to specific objects.

Balloon

Menu: Dimension / Info / Balloon

Toolbox Icon:



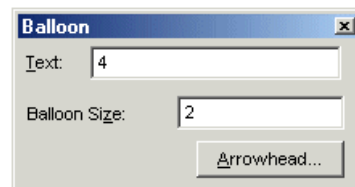
Point 1: Point of arrowhead

Points 2-n: Points along arrow. Last point is center of balloon.

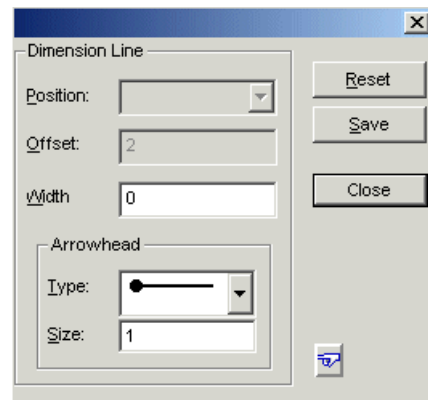
Draws a text balloon pointing to a specific object. It is useful for identifying part numbers or item numbers in diagrams. You may specify the size of the balloon and the text to include; the text is sized to fit inside the balloon.

Using the Command

Choose the **Balloon** command and enter the desired text and balloon size in the window.

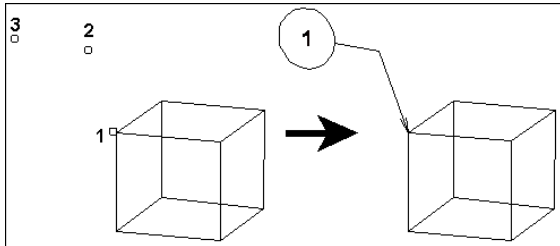


Click the **Arrowhead** button for further options.



Type and **Size** affect the arrowhead. **Width** affects the balloon pull-out line. Click **Same As** (hand icon) to copy properties from an existing balloon.

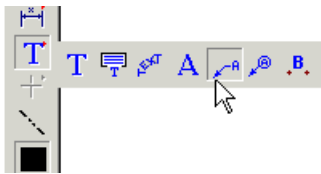
Set a point for the tip of the arrowhead and one or more points for the body of the arrow. The last point set is used as the center of the balloon.



Pullout

Menu: Dimension / Info / Pullout

Toolbox Icon:



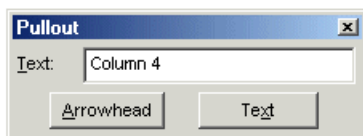
Point 1: Arrowhead

Point 2-n: Path of arrow. The last point marks the text location.

Inserts arrows and descriptive text into a drawing.

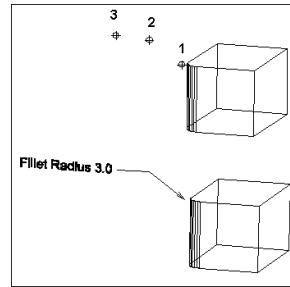
Using the Command

Choose the **Pullout** command and enter the text in the window.



Click the **Arrowhead** button for further options. These options are the same for the **Balloon** command.

Set a point for the point of the arrow, and one or more other points for the body of the arrow. Set the last point for the location of the text. Press Enter to end the command and draw the pullout.



Hatching

Hatching command are used to fill a closed area with a pattern of lines or curves. DesignCAD provides you with a library of hatch patterns, and you can define your own.

Hatch

Menu: Draw / Hatch / Hatch

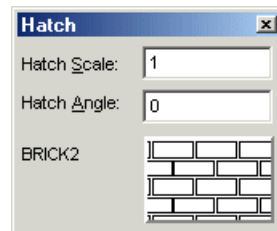
Shortcut Key: #

Points 1-n: Outline of area to be hatched

Fills an area with a hatch pattern. Points are set around the outside of the area to be hatched.

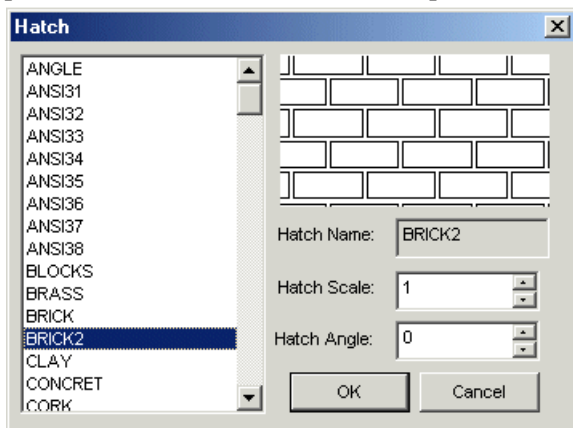
Using the Command

Choose the **Hatch** command and the following window appears:



To change the scale of the preview on the Hatch Pattern button, press and hold either the Shift or Ctrl key while you click on the thumbnail. The Shift key increases the scale of the preview. The Ctrl key decreases the scale of the preview.

Enter the desired scale and angle, and click the pattern thumbnail to choose a hatch pattern.



Set points around the area to be hatched. Click the mouse or press Enter. The area is filled with the hatch pattern. The lines making up hatch patterns are defined as a single entity. These lines can be edited or erased only as an entity.

Hatch Line ^{2D} only

Menu: Draw / Hatch / Hatch Line

Point 1-n: On the enclosed line(s) to be hatched

Fills an enclosed area with a hatch pattern. First, the hatch pattern is selected, and then points are set on the lines or curves to be hatched.

Using the Command

Choose the **Hatch Fill** command and enter the size of the hatch pattern.

Next, set a point each curve or line of the hatching boundary and press Enter. The area is filled automatically with the hatch pattern.

Hatch Fill ^{2D} only

Menu: Draw / Hatch / Hatch Fill

Point 1: In the area to be hatched

Fills a closed area with the specified hatch pattern.

Using the Command

Choose the **Hatch Fill** command and enter the size of the hatch pattern.

Next, set a point inside the area to be hatched. The area is filled automatically with the hatch pattern.

Hatch Pattern Position

Menu: Edit / Hatch Pattern Position

Lets you shift the position of a hatch pattern within its boundary. If **Selected Entities Only** is checked, the hatch pattern is shifted identically for all selected hatch entities. Otherwise, only the hatch pattern selected by the first point is shifted.

Using the Command

First set a point on a location inside a hatch pattern, then a second point indicating where that point should be shifted to. For example, you might set the first point on the lower left corner of a brick, and the second point at the lower left corner of the hatch boundary, to ensure that a brick always begins in the lower left corner of the hatched area.

*NOTE: The ability to snap to a point inside a hatch pattern is enabled or disabled in the **Preferences** command, under **Snap Settings**: Enable snap commands to snap to the lines in a hatch pattern. See **Preferences**.*

Once a hatch pattern position has been shifted with this command, the **Info Box** for that hatch pattern will have a check beside the **Enable Seed Point** setting. Remove the checkmark beside this option to restore the original unshifted hatch pattern position.

Custom Hatch Pattern

If you need to use a hatch pattern different from the ones provided by DesignCAD, you can create your own patterns. See Custom Hatch Patterns.

Save

Menu: File / Custom Hatch Pattern / Save

Saves hatch patterns contained in imported drawings.

Using the Command

Import the drawing that contains the hatch pattern(s) you want to save. Choose the **Save** command. The **Save Hatch Pattern** box appears. In the **File Name** box enter the name for the new file that will contain the custom hatch pattern(s). In the **Store In** box select the location where you want to store the file. Then click **Save**. The file is saved. The custom hatch patterns contained in the new file are saved with "AD_" appended to the beginning of the original hatch pattern name.

Load

Menu: File / Custom Hatch Pattern / Load

Loads previously saved custom hatch patterns and makes them available for use in DesignCAD.

Using the Command

Choose the **Load** command. The **Load Hatch Pattern** box appears. Browse to the pattern file (*.dch) to load, and click **Open**.

Any custom hatch patterns contained in the pattern file are loaded and become available in DesignCAD's hatch commands. The custom hatch patterns appear in the hatch commands with "AD_" appended to the beginning of the original hatch pattern name.

VRML WWW Anchor 3D only

Menu: Tools / VRML WWW Anchor

Lets you add a link for the World Wide Web to any non-animated VRML file.

Using the Command

Open a file that you are going to export as a non-animated VRML file. Select the portion you want to use as a link. Select the **VRML WWW Anchor** command.



Enter the destination URL in the **URL** box. In the **Description** box, enter the description you want displayed when the user places the cursor over the graphic. Export the VRML with the **Export** command (see Export.)

6 Editing and Measuring

Selecting Entities

Many of DesignCAD's commands work only on selected objects. Other commands work on the entire drawing or selected items only, depending on the options you choose. Below are instructions for selection tasks you'll often use in your drawings.

Once entities are selected, you can use any of the **Selection Edit** commands on them. See Selection Edit.

2D Selection Mode: In 2D Selection Mode the cursor assumes an arrow shape. To change to this mode, press Ctrl+2 or select **Options / 2-D Selection Mode**. When you are in 2D Selection Mode, you can always select an object, either by clicking it or snapping to it with point commands. In 3D Selection Mode that is not always the case.

3D Selection Mode: The 3D cursor is formed by three intersecting lines—a black one along the X-axis, a red one on the Y-axis, and a green one on the Z-axis. Press Ctrl+3 to switch to 3D Selection Mode, or select **Options / 2-D Selection Mode (toggle off)**. 3D Selection Mode only works in 3D Mode.

One point to remember about 3D Selection Mode is that the cursor moves in 3D space. Sometimes, therefore, the cursor looks as if it is on or near an object you're trying to select, but it is actually far from the object. For this reason, point snap commands and visual point placement may appear not to work very well in 3D Selection Mode. However, if you check the other views, you'll see why you missed your intended target—you just thought you were close enough!

For most cases, you'll want to use 2D Selection Mode. After you start a new drawing or open an existing one, press Ctrl+2 or use the Command Menu to change to 2D Selection Mode. In 2D Selection Mode, the cursor is a wide arrow when there is not a drawing or editing command active. 3D Selection Mode should be reserved for complex drawings that have drawing entities in close proximity.

Selecting a single object: Move the arrow-shaped cursor near the object and click. If you are close enough, the object is selected, turning magenta and showing a blue bull's-eye where you clicked. The bull's-eye is a selection handle. If you click too far away, either nothing gets selected or something closer to the cursor than the object you want gets selected. Pressing Esc will clear the current selection, as will clicking in a blank region of the screen.

NOTE: *Pressing Delete erases the current selection.*

With the **Gravity** command, it's easy to select an object and simultaneously set the handle at an exact location in the object. To set a **Gravity** point, right-click the mouse (if the **Enable Right Click Popup Menu** option in the **General Options** folder is disabled - see General Options) or press the . (period) key with the cursor near the desired point in the object you want to select. The cursor snaps to the point placing the selection handle there and selecting the composite object that contains that point.

Selecting a group of objects in a region: Move the mouse to one corner of the region. Hold down the left button and drag a selection rectangle around the region. Release the mouse button at the opposite corner. Every item that was completely enclosed in the region is now selected. The handle is placed at the center of the area that the selection rectangle encompassed.

Selecting objects enclosed in or touching a selection rectangle: Click and drag the selection rectangle, but press Ctrl before releasing the mouse button. All objects that were enclosed by or touching the selection rectangle when you released the mouse button are selected.

Adding or deselecting a single item: To add a single item to the selection set, or to deselect a single item in a selection set, move the mouse near the item, press Shift, and click the left mouse button.

Adding a group of objects to the selection set: After selecting one group of objects, drag a selection rectangle around another group as in normal selection, but press Shift before releasing the mouse button. The items must be completely enclosed by the rectangle to be affected. Any items in the region which were already selected will be deselected.

Selecting specific items: Sometimes you need to be even more specific about which items you want to select. If DesignCAD is in 3D Mode, you can select objects by dragging a 3D rectangle around them. If you press Ctrl+3 while your arrow cursor is showing, it turns into a 3D cursor like the one you see when you draw a line. This signifies that you are now in 3D Selection Mode. You select items as before, but for regional

selections, you must enclose the items in a 3D selection box rather than a simple 2D rectangle. To return to the 2D Selection Mode, press Ctrl+2.

Selecting all objects: Finally, let's not overlook the convenience of the **Select All** command. This command, located in the **Edit** menu, selects everything in the drawing.

Using Selection Handles: When you work with selected objects, you often need some way to specify how DesignCAD will manipulate the items. This is the purpose of selection handles. The selection handles are the specific points in the object with or around which DesignCAD moves, copies, and rotates the selected object or objects. A moved object or copy of an object is built around the handles according to the same relative relationships as the original object.

In many cases, you just want to move your object, or a copy of it, somewhere else in the drawing. For these tasks, one handle is enough.

At other times, however, you might need to control the size and orientation of the moved or copied object. Now you need two, or possibly even three handles by which to locate the object. The primary handle (Handle 0 if you just selected the object, the blue bull's-eye if you have used the **Set Handles** command) establishes the starting location for the object. Handle 1 provides a reference point for both the direction and scale of the object's primary axis. Handle 2 provides a second directional reference for orientation.

Placing specific handles: You can set specific selection handles on a selected object or group of objects by pressing Ctrl+H or using **Edit / Selection Edit / Set Handles**. The Status Bar

prompts you to set one, two, or three points for the handles. If you set less than three, press Enter to end the command.

You can use any of the point commands to set these handles, including **Point XYZ**, **Point Relative**, and **Gravity**. You can set or change your handles even if you have already started another command.

Selection Modes

Entity selection is slightly different when working in 2D or 3D Mode.

2D Selection Mode

Menu: Options / 2D Selection Mode

Shortcut Key: Ctrl+2

Point 1: Object to be selected or first corner of selection rectangle

Point 2: Second corner of selection rectangle (optional)

Most DesignCAD commands require drawing objects to be selected. For example, to erase an object you can select it and then press the Del key.

2D Selection Mode causes the selections to take place with respect to the drawing screen. When you click on an object, the object closest to the cursor on the screen will be selected, not the closest object in 3D space. The two are not always the same.

Using the Command

There are four ways to select objects in DesignCAD:

1. Click directly on the object.
2. Drag a selection rectangle around a region.

3. Choose the **Select All** command.
4. Choose the **Select Entity** command. This method only works while the program is in Preset Point Mode.

If you drag a selection rectangle around a region, objects inside that rectangle will be included in the selection. To include objects that touch the selection rectangle, hold the Ctrl key down during the selection.

The Shift key can be used to add items to the current selection set. For example, to select two objects in the drawing, you can select one and then hold the Shift key down while you select the other.

When you select something by dragging a selection rectangle in 2D Selection Mode, the objects inside the rectangle are selected, regardless of their "depth." With 3D Selection Mode you drag a 3D box, and objects must lie inside the box in all three directions to be selected.

NOTE: A DesignCAD drawing is always in either 2D Selection Mode or 3D Selection Mode. To change the selection mode, select this command, which is a toggle function.

You may also select the 2D or 3D Selection Mode tool in the Main Toolbox. If 3D Selection Mode is active, the 3D Selection Mode tool will be visible in the Main Toolbox. Click and hold on the tool; the tool drawer will slide out. While still holding the mouse button, move the cursor over the 2D Selection Mode tool. Release the mouse button to put the currently active drawing in 2D Selection Mode.

In 2D Selection Mode, the cursor is the familiar mouse arrow. In 3D Selection Mode, the cursor is a 3D cursor made up of three lines parallel to the X, Y, and Z axes.

3D Selection Mode

Shortcut Key: Ctrl+3

Point 1: Object to be selected or first corner of selection rectangle

Point 2: Second corner of selection rectangle (optional)

Changes the selection mode of DesignCAD so that selections are made in 3D space rather than with respect to the drawing screen.

In 3D Selection Mode, the selection takes place in three dimensions. When you select an object by clicking on it, the cursor must be on the object on the X, Y, and the Z axis. When you select an object by dragging a box around it, the object must be enclosed along X, Y, and Z axes, not just the left, right, top, and bottom.

If you drag a selection rectangle around a region, objects inside that rectangle will be included in the selection. Don't forget to use Ctrl+Shift to drag the cursor in the third direction so the objects will be encompassed in this dimension as well. To include objects that touch the selection rectangle, hold the Ctrl key down just before you release the mouse button as you complete the selection rectangle.

The Shift key can be used to add items to the current selection set. For example, to select two objects in the drawing, you can select one and then hold the Shift key down while you select the other.

When using commands like **Gravity** in 3D Selection Mode, the cursor snaps to the point that is closest in 3D space, not necessarily the one that appears to be nearest the cursor on the screen.

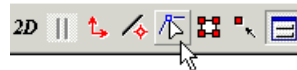
It's easy to use 3D Selection Mode using the DesignCAD Tile views. This allows you to see objects from all sides. - the main Perspective view and smaller Front, Top, and Side views. Move the cursor so that it appears outside the bottom left corner of the object to be selected in all three of the smaller views. (You may have to press Ctrl+Shift while moving the mouse to get there). Now press the left mouse button, hold it down, and move so that the cursor drags the selection box past the top right corner of the object in all three of the smaller views. Release the mouse button. If the selection box completely enclosed the object, it will be selected. (It takes some practice, so keep trying.)

Point Select Mode

Menu: Options / Point Select Mode

Shortcut Key: Ctrl+1

Toolbox Icon:



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

With Point Select Mode active, you can select one or more objects; move individual points; select a group of points and move all of them at once (even in more than one object); add or delete points to and from a single selected object; or cut a line or curve into two pieces.

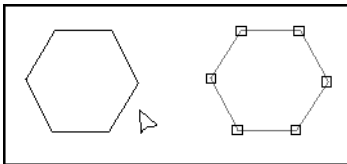
Selecting Objects

To select an individual object in Point Select Mode, simply click on the object. If another item is selected already, the first will be deselected and the second will be selected instead. You can press the Shift key and click on other items to add or remove them to or from the selection set.

NOTE: Point Select Mode will not allow you to select objects with a selection rectangle. The reason for this is that if multiple points are to be manipulated once the objects are selected, the points are specified with a selection rectangle.

Moving Individual Points in Selected Objects

Select the object. If you are not in Point Select Mode, choose the command from the **Options** menu, or press Ctrl+1. The points in the selected object are highlighted with small boxes, and the cursor turns into an arrowhead with no tail.



Move the cursor over the point you wish to move. The cursor changes shape into a circle with cross hairs through its center. Click on the point, then move the cursor to the desired position and set a point there. The advantage of this method is that once you have clicked on the point, you can use Point Polar, Point Relative, or Point XY (Point XYZ in 3D Mode) to move the point to an exact location. Alternatively, you can click on the point and hold down the mouse button and drag the point, only releasing the left mouse button when the point is where you want it.

Moving a Group of Points

Select the object(s) to be manipulated. Press and hold the left mouse button and drag a selection rectangle around the points you wish to select. Release the mouse button. The small boxes turn solid for each point selected in this manner. If the points do not all lie in a rectangular region, select some with a rectangle, then press Shift and drag a selection rectangle around the other points.

NOTE: You must select even single points using the Shift key and a selection rectangle if you wish to add them to a set of selected points.

Now that your points are all selected, move the cursor over any of the selected points. The cursor turns into a circle with cross hairs. Click once and move the cursor; DesignCAD shows a rubber-band representation of the moving points. Set a second point for your desired destination. All of the selected points are moved, and the shapes of the objects containing those points are updated.

Adding a Point to a Single Selected Entity

Select the entity and enter Point Select Mode. Move the cursor near the object you want to modify. When you are in range the cursor will turn into an arrow with a "+" sign in the arrowhead. Click at the location where you want to add a point. As you move the cursor, a rubber-band line will follow, indicating the shape the object will have. Click again to place the new point where desired. To add the point without changing the shape of the object, set the first point, then use the **Line Snap** command to snap to the same location (**Gravity** will not work, because the point is not defined yet!).

NOTE: This function only works for lines or curves and when there is only one item selected. Entities that cannot be modified this way will never show the arrow cursor with the "+."

Extending the End of a Selected Line

Select the line and switch to Point Select Mode. Move the cursor near one endpoint, and press the Shift key. If this is the first or last segment of the line, the cursor will turn into an arrow with three dots at its tip. Click the line, and release the Shift key. As you move the cursor, a rubber-band line shows the line extended to the cursor's location. You can shorten or lengthen the segment, or make it go backwards. When the rubber-band line is the desired length, click again. The extended line will be redrawn.

NOTE: This function only works on lines, and only on the first or last segment.

Deleting a Point from a Selected Object:

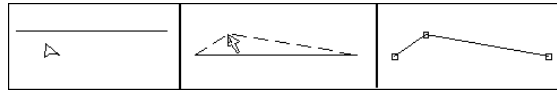
Select the object. If necessary, enter Point Select Mode. Press Ctrl+Shift. Move the cursor over the point you wish to delete. The cursor turns into a cursor with a - on it. Click on the point. The point is erased, and the object is redrawn.

Cutting a Selected Object into Two Pieces:

Select the object (this works with a line or curve, also with an arc, or circle if they are saved as a line). If necessary, enter Point Select Mode. Press Ctrl. Move the cursor to where you want to cut the object (this does not have to be a pre-defined point). The cursor becomes a tiny pair of scissors. While pressing Ctrl, click the left mouse button. The object is cut at the selected point.

Example: Add a point to a line.

Enter Point Select Mode. Move the cursor near the line. Click the left mouse button to select the entity. Once the points are showing, move the cursor onto the line. The cursor will have a + in it when it is on the line. Click the left mouse button. Move the cursor to the desired position for the new point on the line. Click the left mouse button to set a point. A point is added to the line.



Preset Point Mode

Menu: Options / Preset Point Mode

Toolbox Icon:



When Preset Point Mode is active, the points needed for a drawing can be set before the commands are chosen. This allows you to set points in precise locations without seeing a rubber-band preview of the entity you are drawing.

Using the Command

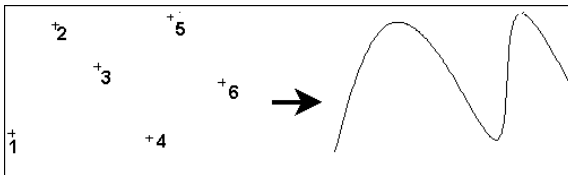
Turn on **Preset Point Mode**, and a check will appear next to the command in the **Options** in the menu. Set the points in your drawing. Select a drawing command. The entity you selected will be drawn using the points you set previously.

Drawing commands executed in this manner will use their current options. To set the options for the command, select it before setting the number of points required for the command.

If more points have been set than are necessary for the chosen entity, DesignCAD will use only as many points as needed, in the order in which they were set. The remaining points will be removed. For instance, if you set four points and choose the Box command, the first two points set will be used to draw the box. The third and fourth points will disappear.

Example: Draw a curve.

Choose **Preset Point Mode** and set six points in the drawing. Select the **Curve** command. A curve will be drawn through the six points you set.



Use Resizing Handles

Menu: Options / Use Resizing Handles

Toolbar Icon:

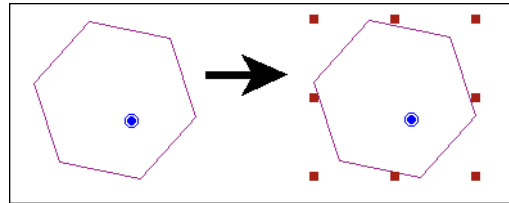


When enabled, items that are selected have scaling nodes or resizing handles around them. These nodes may be used to scale the selection along a single axis, both axes in 2D Mode, or all three axes in 3D Mode.

Using the Command

Select the items to be scaled or zoomed. Select the **Use Resizing Handles** command and resizing handles appear on and around the selection. These nodes may be used by placing the cursor over one of them, clicking the left mouse button, dragging the cursor to make the selection larger or

smaller, and then clicking the left mouse button a second time to release the selection. The resizing handles will remain visible until the command is turned off.



If you just use the left mouse button, the edge or corner opposite the resizing handle you chose maintains its original location. To scale the selection by its center, press and hold down the Ctrl key when you click on the resizing handle, and release the Ctrl key after you click the mouse.

To zoom the selection (scale the selection equally with respect to all axes at once), press and hold down the Shift key when you click on the resizing handle, and release the Shift key after you click the mouse.

To zoom the selection by its center, press and hold down the Ctrl+Shift keys when you click on the resizing handle, and release the Ctrl+Shift keys after you click the mouse.

Select All

Menu: Edit / Select All

Shortcut Key: Ctrl+A

Selects every editable object in the drawing. This makes it easy to manipulate all objects as a group rather than having to perform the same command on every object individually. Objects that reside in layers that have been defined as uneditable in the **Layer** options (see Layer Options), are not selected.

*TIP: If you have a number of objects on the screen and need to select all but a few of them, choose the **Select All** command to select them all. Then depress the Shift key and click on the objects you don't want to be selected. This is much faster than selecting each object individually.*

Select Previous

Menu: Edit / Select Previous

Shortcut Key: Shift+P

Selects the entities that were previously selected in the drawing.

Select Next

Menu: Edit / Select Next

Shortcut Key: Enter

Used to select relatively small entities that are completely overlapped by other entities. If the wrong entity is consistently selected when the mouse is used to click in an area containing a small entity, this command can be used to select the desired entity.

Using the Command

Attempt to select an entity at a point that is overlapped by another entity. One of the entities is selected and a selection handle appears at the location on which you clicked. Choose **Select Next**. The other entity that occupies the exact location of the selection handle or a location near the selection handle will be selected.

Select Entity

Menu: Edit / Select Entity

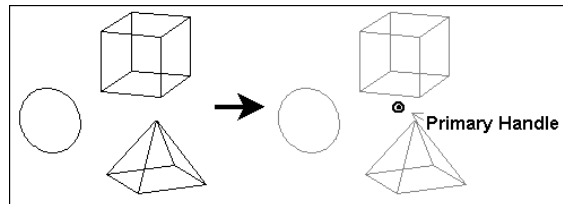
In **Preset Point Mode**, the points are set before the commands are chosen. Any time you click the mouse, a point is set. The **Select Entity** command tells DesignCAD that you want to select the entity on which you set a point.

Using the Command

In **Preset Point Mode** (found on the **Options** menu), set a point on each entity you wish to select. Choose the **Select Entity**. Each entity on which a point was set will be selected.

Example: Select several entities.

Draw a box, a pyramid and a circle. Enable **Preset Point Mode** from the **Options** menu. Set a point on the box. Next set a point on the pyramid. Set another point on the circle. Choose **Select Entity**, and the box, the pyramid and the circle will be selected.



Selection Filter

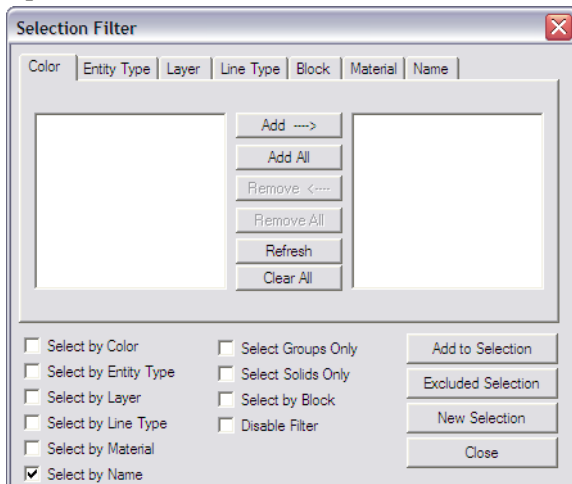
Menu: Edit / Selection Filter

Shortcut Key: Shift+F

Allows you to restrict the type of entities that are selected when a selection box is dragged around several objects in a drawing. This allows you to quickly drag a selection box around a group of entities and select only certain entities instead of everything inside the box.

Using the Command

Open the **Selection Filter**:



Three basic tasks can be performed using this window. Drawing items with given characteristics can be:

- added to an existing selection
- excluded from the current selection
- defined as a new selection.

These actions are achieved either by clicking the appropriate action button (**Add to Selection**, **Excluded Selection**, and **New Selection**) to

perform the action throughout the entire drawing, or by using standard DesignCAD selection methods (Ctrl and/or Shift, plus mouse-clicking and selection rectangle combinations) in the drawing area.

From the top of the window, open the tab (**Color**, **Entity Type**, etc.) whose selection properties you want to set. To add a property to the filter so items of that property, highlight the object in the list and click **Add**. The **Select By Color**, etc. option is automatically enabled to indicate that the particular filter is currently being used. To remove a property from the filter, highlight it on the right-hand list and then click **Remove**.

Select Groups Only: Only groups that have been defined in the drawing using the **Group Define** command can have their selection status altered.

Select Solids Only: Only solids will be affected by the Selection Filter.

Disable Filter: Filter restrictions are retained but ignored until this option is disabled or the Selection Filter is closed.

Add to Selection: All drawing entities whose characteristics are currently defined as part of a filter are added to the current selection.

Excluded Selection: All drawing entities whose characteristics are currently defined as part of a filter are removed from the current selection. If nothing is selected when this button is clicked, all entities are selected except those whose characteristics are currently being filtered.

New Selection: When this button is clicked, all selected drawing entities are deselected and the entities whose characteristics are currently defined as part of a filter are added to a new selection set.

Close: Closes the Selection Filter window. No filters remain active when the Selection Filter is closed, but the filter settings are retained until the drawing is closed.

Polygon Selection

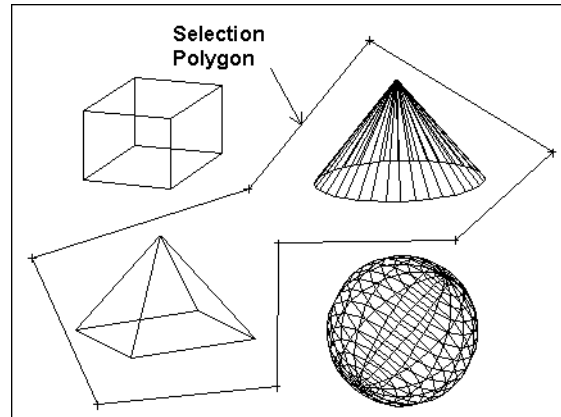
Menu: Edit / Polygon Selection

In 2D Selection Mode and 3D Selection Mode, this command works like a 2D selection rectangle. The selection polygon can consist of as many points and sides as you need. This allows you to define a more specific area for selection.

Using the Command

Select the **Polygon Selection** command and set points in your drawing forming a polygon around the entities you wish to select. At least three points must be set to form a valid polygon. If boundaries of the polygon overlap one another,

entities encompassed by the resulting area will not be selected. Press Enter. The entities which were inside the polygon are selected.



Entity Properties

All DesignCAD entities have certain properties: color, line type, and layer. This section covers ways you can identify these properties and change them.

Colors

To set the drawing color, make sure the correct color appears in the Color box in the main toolbox. See Colors‘Colors’ on page 6. Whatever color is set is the color that new entities will use.

To change an entity’s color, select the entity and open its **Info Box**.

Color by Layer

Menu: Options / Color by Layer

A toggle that determines whether or not entity colors change automatically when entities are moved to a layer for which the color has been set using the **Assign Color** option in the **Layer Options** (see Layer Options). When this

command is enabled, the color of entities on a layer will also change if the layer's assigned color is changed.

Layers

For details on creating and setting layers, see Layer Options. The active layer is the layer new entities will be placed on.

To change an entity's layer, select the entity and open its **Info Box**.

See also: Duplicate to Layer.

Line Styles

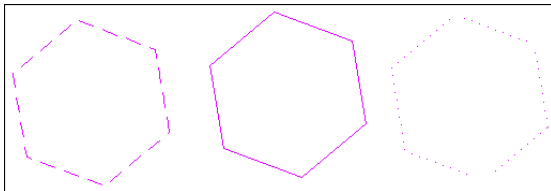
Line styles can be set using the Line Style toolbox (for details on displaying toolboxes, see View Options).



You can also open this toolbox by clicking the **Line Style** icon on the Main toolbox.



The first field contains the line style itself; DesignCAD provides several from which to choose.



To define your own, see Hacking Custom Line Styles and Custom Line Style Shapes.

Apply to Selection: Applies the line style to all selected objects.

Apply to Entire Drawing: Applies the line style to all entities in the drawing.

Fill Wide Line: Applies a solid fill to wide lines.

Change Scale Width: Adjust the width of the line style repeating pattern.

Same As: Switches the line style to that of a selected entity.

Save as Default: Sets the selected line style as the default for this and future drawings.

To change an entity's line style, select the entity and open its **Info Box**.

Line Style by Layer

Menu: Options / Line Style by Layer

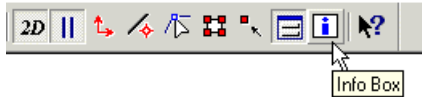
A toggle that determines whether or not entity line styles change automatically when entities are moved to a layer for which the line style has been set using the **Assign Line Style** option in the **Layer Options** (see Layer Options). When this command is enabled, the line style of entities on a layer will also change if the layer's assigned line style is changed.

Info Box

Menu: View / Info Box

Shortcut Key: Ctrl+I

Toolbar Icon:

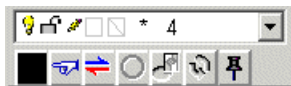


Used to view and edit properties of a selected object. To activate the command, you must select an object before executing the command.

Using the Command

Select an object and choose the **Info Box** command. The available items in the Info Box vary depending on the type of object you have selected.

All Info Boxes contain Layer (see Layer Options) and Color information.



To change any property, click its box and select the new property.

Same As: Lets you match the parameters of an existing entity. Just click on the **Same As** button, then click on the entity already in the drawing that has the properties you want for the selected entity.

Keep Visible: Controls whether or not the Info Box remains visible when no entity is selected.

Reverse Points: If enabled (only when vector entities and/or spline curves are selected), click it to change the numbering of the points. The point

designations are reordered; the first point becomes the last point and the last point becomes the first point.

Compliment Arc: If enabled (only when arcs or elliptical arcs are selected), click it to change the arc to its complimenting arc.

Texture Mapping: If enabled, click it to display the **Texture Mapping** window. See Texture Mapping.

Show Detail: If enabled, click it to display additional details on the selected object like point locations, scale, etc.

Name: Allows you specify a name for the selected object/s.

There are different kinds of Info Box configurations based on object categories. Each contains information relevant to the specific category of object.

Plane

- **Point Coordinates:** The coordinate of the current point on each axis.
- **Current Point:** The number of the point to which the information applies.
- **Number of Points:** The total number of points in the plane.
- **Area:** The area measurement of the plane in Drawing Units.
- **Length:** The measurement of the perimeter in Drawing Units.
- **Material:** The material properties of the entity.

- **Smooth:** True = Smooth enabled; False = Smooth Disabled.
- **Color by Layer:** Changes entity color automatically if moved to a layer with a preset color.

Circle/Arc

- **Center Coordinate:** The coordinate of the center on each axis.
- **Radius:** The radius in Drawing Units.
- **Start Angle:** The starting angle of the Circle/Arc.
- **Span Angle:** The span angle of the Circle/Arc.
- **Area:** The area of the Circle/Arc.
- **Length:** The circumference of the circle or length of the arc in Drawing Units.
- **Line Type:** The type of line (i.e., dashed, dotted, etc.) used to draw the Circle/Arc.
- **Line Scale:** The amount of space between dashes or dots in certain line types.
- **Line Width:** The width of the line used to draw the Circle/Arc.
- **Fill Wide Lines:** Fills wide line types in 2D Mode instead of drawing them as outlines.
- **Color by Layer:** Changes entity color automatically if moved to a layer with a preset color.
- **Line Style by Layer:** Changes line style automatically if moved to a layer with preset line style.

Dimension

- **Text Font:** The typeface of the text.

- **Text Size:** The point size of the text.
- **Accuracy:** The number of digits after the decimal point in the dimension.
- **Arrow Type:** The type of arrow used in the dimension.
- **Format:** The format of the dimension text.
- **Flip Text:** Reverses the direction of dimension text.
- **Color by Layer:** Changes entity color automatically if moved to a layer with a preset color.
- **Line Style by Layer:** Changes line style automatically if moved to a layer with preset line style.

Vector

- **Point Coordinates:** The coordinate of the current point on each axis.
- **Current Point:** The number of the point to which the information applies.
- **Area:** The area of the vector.
- **Length:** The measurement of the vector in Drawing Units.
- **Bounding Box:** The dimensions of a box that would encompass the vector.
- **Line Type:** The type of line used to draw the vector.
- **Line Scale:** The amount of space between dashes or dots in certain line types.
- **Line Width:** The width of the line used to draw the curve.

- **Fill Wide Lines:** Fills wide line types in 2D Mode instead of drawing them as outlines.
- **Color by Layer:** Changes entity color automatically if moved to a layer with a preset color.
- **Line Style by Layer:** Changes line style automatically if moved to a layer with preset line style.

Curve

- **Point Coordinates:** The coordinate of the current point on each axis.
- **Current Point:** The number of the point to which the information applies.
- **Area:** The area of the curve.
- **Length:** The measurement of the curve in Drawing Units.
- **Bounding Box:** The dimensions of a box that would encompass the curve.
- **Line Type:** The type of line used to draw the curve.
- **Line Scale:** The amount of space between dashes or dots in certain line types.
- **Line Width:** The width of the line used to draw the curve.
- **Fill Wide Lines:** Fills wide line types in 2D Mode instead of drawing them as outlines.
- **Color by Layer:** Changes entity color automatically if moved to a layer with a preset color.
- **Line Style by Layer:** Changes line style automatically if moved to a layer with preset line style.

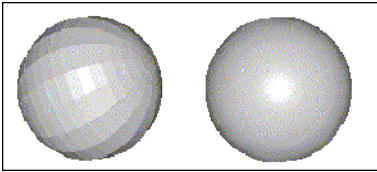
Text

- **Font:** The typeface of the text.
- **Content:** The letters, numbers, and symbols that make up the text entry.
- **Angle:** The angle at which the text entity is placed in the drawing.
- **Length:** The length of the text entity in Drawing Units.
- **Height:** The height of the text entity in Drawing Units.
- **Keep Length/Width ratio:** When checked the length/width ratio stays the same when the length or width is changed.
- **Color by Layer:** Changes entity color automatically if moved to a layer with a preset color.

Solid

- **X,Y,Z Scale:** The scale of the solid along each axis.
- **Bounding Box:** The dimensions of a box that would encompass the solid.
- **Material:** The material properties of the solid.
- **Smooth:** Enables or disables the smooth shade option. The Smooth shade option is selected by default for all solids. This causes curved surfaces to appear smooth, with their

faceted edges rounded off. Smooth shading can be disabled in order to force the facets of the shaded solid to be visible.



- **Area and Volume:** Clicking this button calculates the area and volume of the solid.
- **Color by Layer:** Changes entity color automatically if moved to a layer with a preset color.

Editing

This section covers all the ways you can edit 2D (and some 3D) objects.

Undo, Redo

Commands are stored in DesignCAD's memory and you can step backward and forward, in order, through these commands.

Undo

Menu: Edit / Undo

Shortcut Key: Ctrl+Z

Toolbar Icon:



Cancels the most recent drawing action. It can be used repetitively to "back out" of a series of commands, as it is always negating the previous drawing action. You can "back up" to the point where the drawing was last saved.

Redo

Menu: Edit / Redo

Shortcut Key: Ctrl+Y

Toolbar Icon:



Reverses any action performed by the **Undo** command. It is only available after you have used the **Undo** command in the current drawing session.

Cut, Copy, Paste

These commands interact with the Windows clipboard - a place you can store one object at a time to insert somewhere else later.

Cut

Menu: Edit / Cut

Shortcut Key: Ctrl+X

Toolbox Icon:



Removes a selected object or group of objects from the drawing screen and places them on the Windows Clipboard. Once the selected object or objects are on the Windows Clipboard, you can **Paste** the contents back into DesignCAD or other Windows applications.

Using the Command

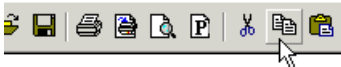
Select the object or objects you want to remove from the drawing and place in the Clipboard. Then choose the **Cut** command. The selection is cut to the Clipboard.

Copy

Menu: Edit / Copy

Shortcut Key: Ctrl+C

Toolbox Icon:



Copies selected objects from the DesignCAD drawing screen to the Windows Clipboard, leaving the original objects still in the drawing. From the Clipboard, you can **Paste** copies into DesignCAD or into other Windows applications.

Using the Command

Select the object or objects to be copied. Choose the **Copy** command. The program copies the selected items to the Clipboard.

If you want to paste the object into another drawing or Windows application, you can do so without recopying. The object remains in the Clipboard until another object is cut or copied to the Clipboard, replacing it.

Section Copy

Menu: Edit / Section Copy

Point 1: One corner of the cutoff box

Point 2: opposite corner of the cutoff box

Copies a section of the drawing that intersects a 3D box to the Windows Clipboard, so it can be pasted into the same drawing, a different drawing, or a different application.

Using the Command

After choosing the **Section Copy** command, define the section by setting two points in opposite corners of a 3D box. (If DesignCAD is in 2D Mode, a 2D box is used.) Any part of the drawing that is contained within the cutoff box boundary is pasted to the Windows Clipboard.

Example: Copy part of a circle elsewhere in the drawing.

Select the **Section Copy** command. Set the points for the cutoff box so that a section of the circle is inside the box. After the cutoff box disappears, press Ctrl+V or select the **Paste** command. A rubber-band box representing the scale of the object will follow the cursor as you move it around the screen. When you have selected a location for the copied section of the circle, set a point.

Paste

Menu: Edit / Paste

Shortcut Key: Ctrl+V

Toolbox Icon:



Point 1: First handle

Point 2: Second handle (optional)

Point 3: Third handle (optional)

Copies drawing objects from the Windows Clipboard into DesignCAD.

Using the Command

Use **Cut** or **Copy** to place one or more objects on the Windows Clipboard. Choose the **Paste** command. Set a point for the location of the object or group to be pasted into the drawing. If necessary, a second point can be used to specify the angle and size of the object, and a third point can be used to set the 3D orientation.

*NOTE: In order to use the second selection handle to change the size of the selection, you must uncheck the **Use Original Scale** option.*

Copy Image

Menu: Edit / Copy Image

Shortcut Key: Ctrl+Shift+C

Point 1: First corner of rubber-band box

Point 2: Opposite corner of the box

Copies text or graphics onto the Clipboard. Copying text or graphics to the Clipboard replaces the contents previously stored there.

This command is similar to the **Copy** command except the entities are not selected prior to selecting the command. The **Copy Image** command copies only what is enclosed in the rubber-band box. The image is treated like a Metafile because the OLE object information is bypassed.

Using the Command

Choose the **Copy Image** command and set a point for the first corner of a rubber-band box that will be used to define the image to be copied to

the clipboard. Use the mouse to enclose the image with the rubber-band box. When the rubber-band box surrounds the image you want copied, set another point for the opposite corner of the rubber-band box. A copy of the image is placed onto the Clipboard.

You can also use the **Copy Image** command to copy part of several entities onto the Clipboard. The image is treated like a drawing instead of a CAD object.

Erase

Menu: Edit / Erase

Shortcut Key: Del

Deletes selected entities from the drawing.

Using the Command

Select the entities in the drawing you want to erase. Then choose the **Erase** command. The items are erased from the drawing.

Erase Last

Menu: Edit / Erase Last

Shortcut Key: Ctrl+E

Deletes the most recently drawn entity from the drawing.

Using the Command

Choose the **Erase Last** command, and the most recently drawn entity is removed from the drawing. Each subsequent time you use this command, it erases the next most recently drawn entity, working its way from last to first entity drawn in that session.

This command pays attention only to the order in which entities were drawn originally. It does not acknowledge modifications to existing entities or undo them, but erases entire entities.

Editing Lines

These commands can be used to edit lines, arcs, circles, and curves.

Line Split by Distance

Menu: Edit / Line Edit / Line Split by Distance

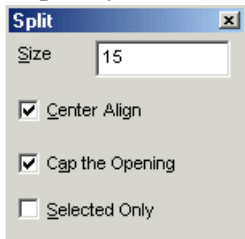
Point 1: Location of line split

Point 2: Direction of line split, if necessary

Removes a section of a specified length from a line or double line, splitting the line or double line into two segments.

Using the Command

Select the **Line Split by Distance** command.

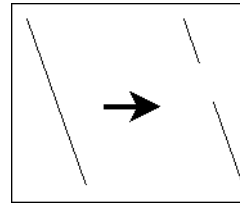


Size: The length of the segment to be removed.

Center Align: The removed segment will be centered at the point where the line is clicked. If this options is not checked, set the first point for the location of the split and then a second point to sets the direction.

Cap the Opening: For a double line, the opening will be capped at the split.

Selected Only: Runs the command only on selected lines.



Line Split by Points

Menu: Edit / Line Edit / Line Split by Points

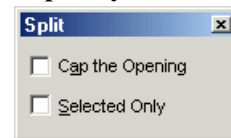
Point 1: First point of segment to be removed

Point 2: Second point of segment to be removed

Removes a section from a line or double line, splitting the line or double line into two segments.

Using the Command

Select the **Line Split by Points** command

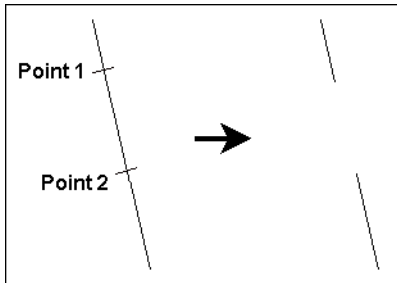


Cap the Opening: For a double line, the opening will be capped at the split.

Selected Only: Runs the command only on selected lines.

Set the first point for the segment to be removed. Set the second point for the desired length and direction of the segment to be removed. The

portion of the line between the two points is deleted, splitting the original line into two segments.



Line Connect

Menu: Edit / Line Edit / Line Connect

Point 1: Endpoint of the first line to be connected

Point 2: Endpoint of the second line to be connected

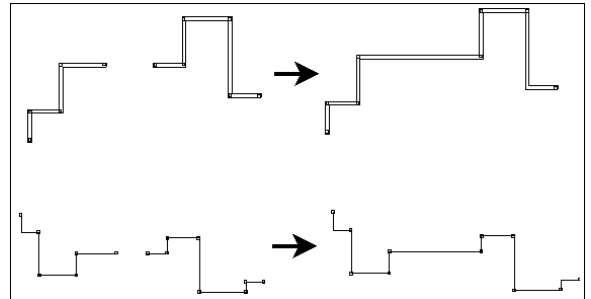
Connects two lines or double lines.

Using the Command

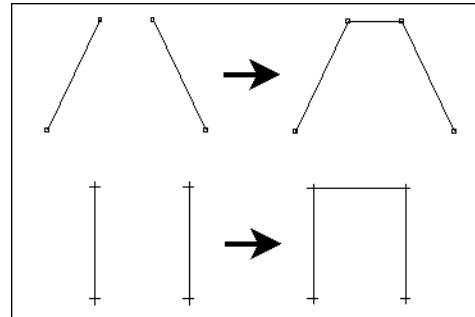
Select the **Line Connect** command and set a point on the endpoint of the first line or double line to be connected. Set a point on the endpoint of the second line or double line to be connected.

If the line segments to be connected are aligned, the gap between the two lines is seamlessly connected into a single line. The original

endpoints of the combined segments are removed, so there are no points in the line to indicate the previous gap.



If the line segments are offset, DesignCAD adds an additional segment to connect the lines, resulting in a single line. In this case, the endpoints of the line segments are not removed.



Line Cut

Menu: Edit / Line Edit / Line Cut

Point 1: Line to be cut

Point 2: Cut location

Cuts a line at a specified point.

Using the Command

Select the **Line Cut** command. Set a point on the line to be cut. Set a point at the location for the cut. The line is cut at that location. The two resulting lines can be selected and manipulated independently.

Convert to Double Line

Menu: Edit / Convert to Double Line

Point 1: Entity to be converted to a double entity

Converts existing 2D entities into double-line entities. Entities that are converted are automatically offset a specified distance and direction from the original. The ends of the entity may be capped or left open independently, and the double entity may be filled and/or automatically trimmed. Double lines simplify several common types of drawings, including house plans and land plots.

NOTE: To create double line entities, see Double Line Mode.

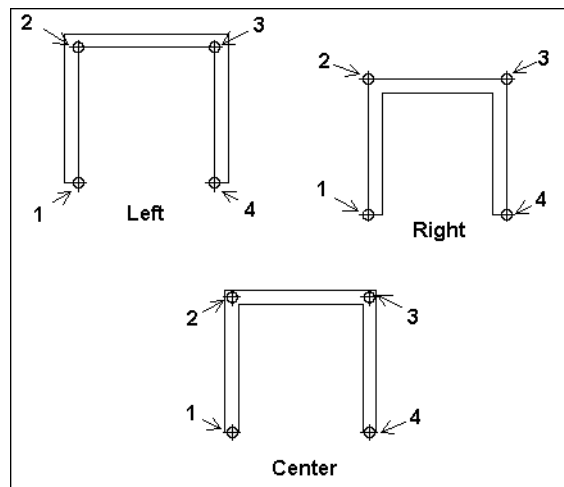
Using the Command

Choose the **Convert to Double Line** command.



Line Width: The space between the two parallel entities.

Align: Determines how the double line is created. If **Left** is selected, the width of the entity will extend to what would be your left if you were standing on the first point set for the entity facing the second point set for the entity.



NOTE: *If the double entity is converted with the **Left** or **Right** option selected, you will be able to snap to both sides of the finished double entity. If the double entity is converted with the **Center** option selected, you will be able to snap to both sides and the center of the double entity.*

Cap: Determines which ends of the double line are capped.

Filled: If checked, the double line will have solid fill.

Auto Trim: The double entity will automatically be trimmed where it overlaps itself or another double entity.

NOTE: *The **Explode** command (see Explode) can be used to explode a double entity into vectors. If you are going to cross one double entity over another, explode the existing double entity and uncheck the **Auto Trim** box before drawing the second double entity to ensure that all four entities will be visible in the intersection.*

Convert Selected Lines: Converts only selected entities.

Selection Edit

This sections refers to editing commands you can perform on entities that are already selected.

Duplicate

Edit / Selection Edit / Duplicate

Shortcut Key: N

Point 1: Handle 1 sets location for the copy

Point 2: Handle 2 sets angle and scale for the copy (optional)

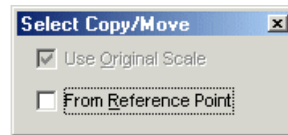
Point 3: Handle 3 sets orientation for the copy (optional)

Makes a copy of the current selection. The command does not use the Windows Clipboard, so any contents in the Clipboard are not erased or changed.

Using the Command

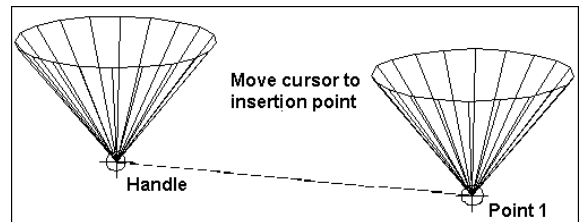
Select an object or objects, and then choose the **Duplicate** command. You can also activate this command by moving the cursor onto the primary handle of an object, then holding down Ctrl and clicking the left mouse button.

The **Select Copy/Move** window appears.



Use Original Scale: If this option is enabled and two or more points are used to place the duplicate, the second point can be used to change the orientation and size of the duplicate. If this option is disabled, the second point only affects the orientation of the duplicate.

One, two, or three points may be used to specify the location where the copy is to be placed. The first handle of the selection is positioned at the first point set with this command.



If you have set two or three selection handles using the **Set Handles** command, or if the **Enable Sticky Handles** option in the **General Options** (see General Options) is checked, then you can place and scale the duplicate by those points. Otherwise, the first point you place will make a duplicate with the same orientation and size as the selection.

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

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

From Reference Point: When enabled, this option lets you place the duplicate using a reference point. Just set a reference point relative to the original. Then set another reference point relative to the desired location for the duplicate. The duplicate is inserted into the drawing at the same scale as the original.

Duplicate to Layer

Edit / Selection Edit / Duplicate to Layer

Copies selected entities to a different layer. See Layer Options.

Using the Command

Select the entities you want to copy, and choose the Duplicate to Layer command. Enter the layer to which the entities copied, and click OK.

Mirror

Edit / Selection Edit / Mirror

Shortcut Key: /

Toolbox Icon:

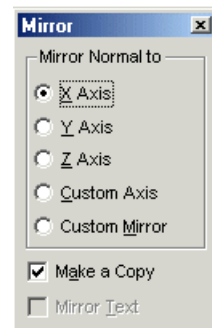


Point 1: Location of mirrored object

Makes a mirror image of a selection.

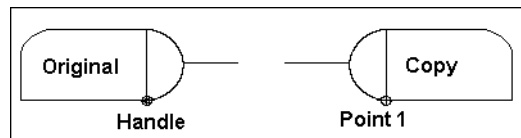
Using the Command

Choose the **Mirror** command. The window contains the following options:

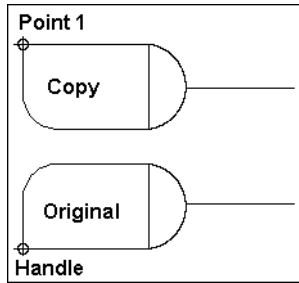


Mirror Normal to: Selecting an axis places a mirror which faces in the direction of the axis. That is, the mirror lies on the axis and is perpendicular to it.

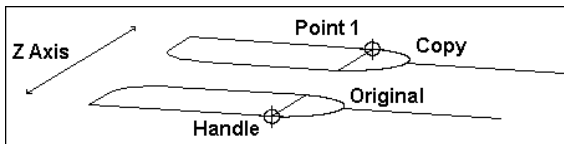
- **X-Axis:**



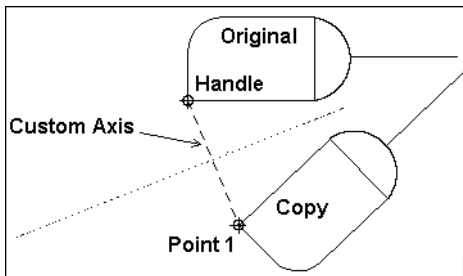
• **Y-Axis:**



• **Z-Axis:**



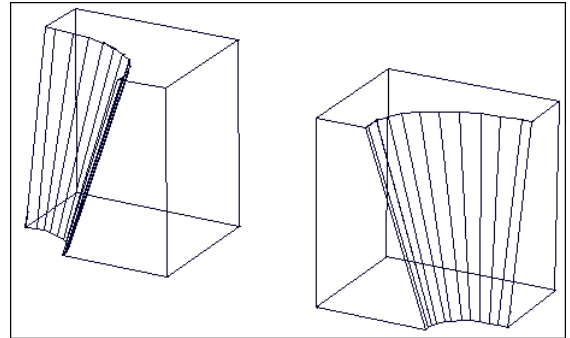
• **Custom Axis:** Specifies a 3D line perpendicular to the face of the mirror. You can use the option to create custom mirror effects, such as producing a mirror image facing at an angle 90 degrees away from the original.



• **Custom Mirror:** Uses a mirror defined by two points.

Set a point for the location of the mirror image. The mirror image will be located at that point, and mirrored according to the option selected. If

Custom Axis is selected, the mirror will be normal (perpendicular) to the line between the selection handle and the point that was set.



Move

Edit / Selection Edit / Move

Shortcut Key: M

Point 1: New location of the handle

Point 2: New location of second handle (optional)

Point 3: New location of third handle (optional)

Moves a selected entity or group to another location. One, two, or three points may be used to specify the new location.

Using the Command

Select an object or objects, and then choose the **Move** command. The options are the same as for the **Duplicate** command (see Duplicate).

You can also activate this command by moving the cursor over the primary handle. When the cursor changes into a four-sided arrow, you can move the object in one of two ways:

1. Press the left mouse button and hold. Drag the selection to the new location.

or:

2. Click the left mouse button once to lock the **Move** command. Drag, or use any of the point commands, to select a new location. If more than one handle has been set for the selection, you can set more than one point when you move the object.

Four-Point Copy

Edit / Selection Edit / Copy

Point 1: location for lower left corner of copy

Point 2: location for lower right corner of copy

Point 3: location for upper left corner of copy

Point 4: location for upper right corner of copy

Copies a selected 2D entity or group of entities to another location. Four points are used to specify the new location and any scaling or skewing that is to occur.

Using the Command

Select the 2D entity or group of entities to be copied. Choose the **Four-Point Copy** command. Set four points for the four corners of the new copy of the selection. When the fourth point is set, the new copy of the selection is inserted in the drawing.

Four-Point Move

Edit / Selection Edit / Move

Point 1: new location for lower left corner of selection

Point 2: new location for lower right corner of selection

Point 3: new location for upper left corner of selection

Point 4: new location for upper right corner of selection

Moves a selected 2D entity or group of entities to another location. Four points are used to specify the new location and any scaling or skewing that is to occur.

Using the Command

Select the 2D entity or group of entities to be moved. Choose the **Four-Point Move** command. Set four points for the new locations of the four corners of the selection. When the fourth point is set, the selection is moved.

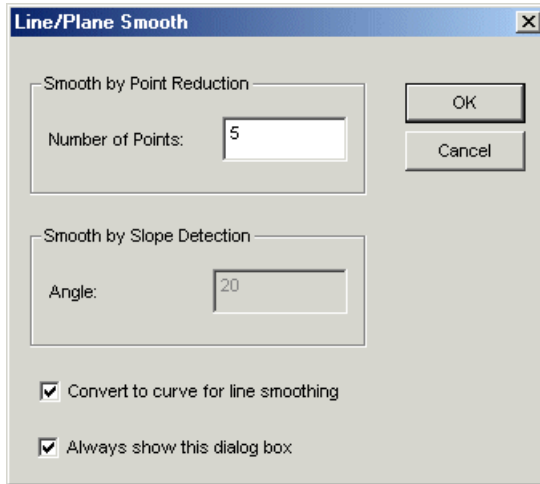
Smooth Line by Point Reduction

Edit / Selection Edit / Smooth Line by Point Reduction

Smooths complex lines and curves by reducing the number of points in them. DesignCAD divides the points of the line(s) and/or curve(s) into groups of a specified number of points. The locations of the points in each group are averaged, and the group is replaced with a single point.

Using the Command

Select the lines and/or curves to be smoothed. Choose the **Smooth Line by Point Reduction** command.



Enter a value for the **Number of Points** for each group. To convert the line to a curve, enable the **Convert to Curve for Line Smoothing** option.

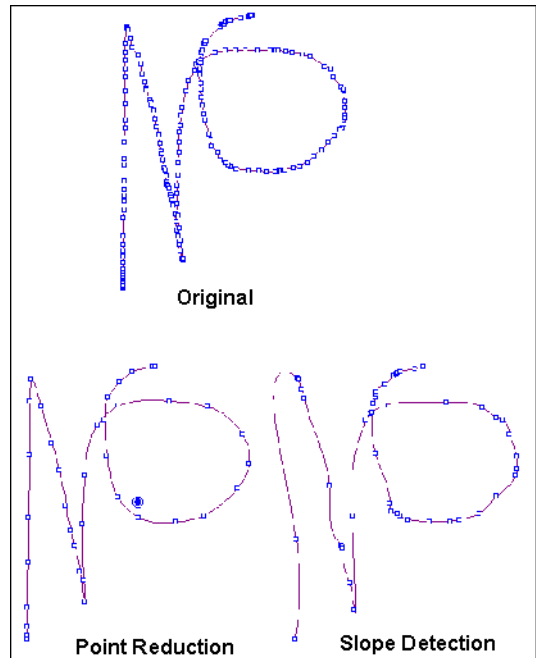
Always show this dialog box: If disabled, you can reset it using the **Smooth Options** command.

A single point is set for the average location of each group. A smoothed object results.

Example: Smooth a complex line.

Select the line. Choose the **Smooth Line by Point Reduction** command. Click OK. The line is converted to a smooth curve. The figure below is shown with **Point Select Mode** turned on to illustrate the reduced number of points in the

result. The original line is shown along with the results of a **Smooth Line by Point Reduction** and a **Smooth Line by Slope Detection**.



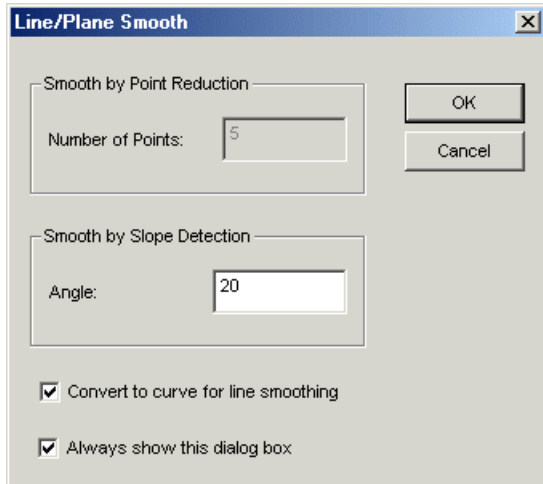
Smooth Line by Slope Detection

Edit / Selection Edit / Smooth Line by Slope Detection

Smooths complex lines and curves. A point is removed if the two segments that it joins do not deviate from a straight line by a specified angle.

Using the Command

Select the lines and/or curves to be smoothed. Choose the **Smooth Line by Slope Detection** command.



Enter a value for the **Angle** option. To convert the line to a curve, enable the **Convert to Curve for Line Smoothing** option.

Always show this dialog box: If disabled, you can reset it using the **Smooth Options** command.

For an example, see **Smooth Line by Point Reduction**.

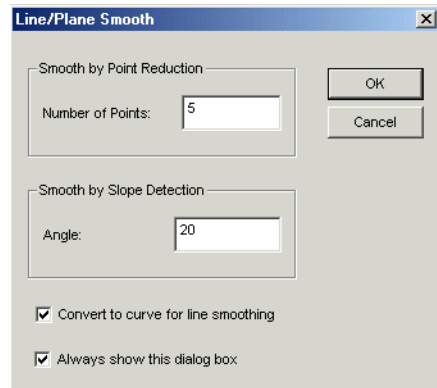
Smooth Options

Edit / Selection Edit / Smooth Options

Sets the default values for the **Smooth Line by Point Reduction** and **Smooth Line by Slope Detection** commands.

Using the Command

Choose the **Smooth Options** command.



Enter the values you want to use as defaults for both Smooth commands and click OK.

Combine Lines

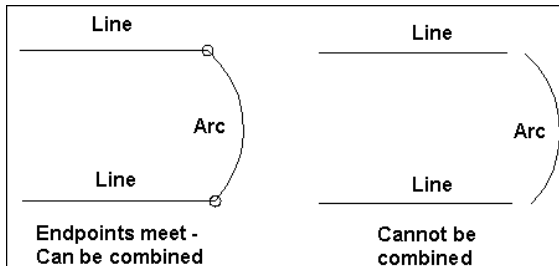
Edit / Selection Edit / Combine Lines

Shortcut Key: B

Merges connecting lines, arcs, and curves into a single line entity.

Using the Command

Select the entities you wish to combine into a single line. All selected entities must share endpoints to form a continuous line. That is, each line entity has to be connected to the next line. This command affects currently selected entities.



This command is similar to **Make Plane** (see Make Plane), but **Make Plane** converts the selected lines into a plane (surface) instead of a line entity.

NOTE: Since this command converts arcs into lines, any arcs will lose their center points. Curves are also converted to vectors for this operation.

Break Line

Edit / Selection Edit / Break Line

Shortcut Key: |

Breaks a line entity consisting of two or more line segments into separate line entities.

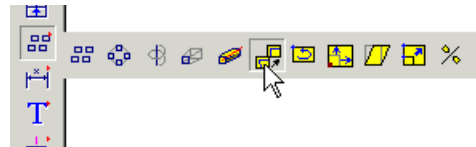
Using the Command

Select the entity (or entities) to be broken. Choose the **Break Line** command. You can use the command on a line created with the **Line** or **Polygon** commands, or created with **Arcs**, **Circles**, or **Curves** drawn as lines. The line selected will be broken into several separate lines, depending on how many vertices the line has.

Ortho

Edit / Selection Edit / Ortho

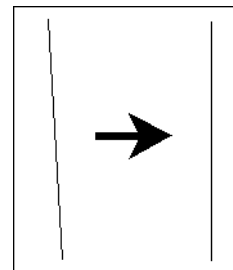
Toolbox Icon:



Forces all line segments within 10 degrees of parallel with the X, Y, or Z axis to lie exactly parallel to that axis.

Using the Command

Select the lines to be straightened, and choose the **Ortho** command. The line will be converted to a vertical or horizontal entity.



Rotate

Edit / Selection Edit / Rotate

Shortcut Key: R

Toolbox Icon:



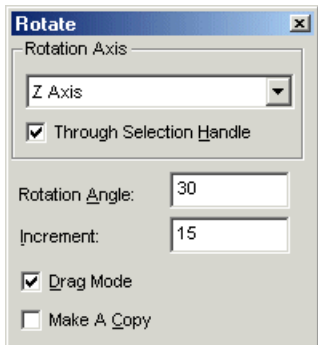
Point 1: Location of rotation axis

Point 2: Second point on rotation axis (2-point rotation only)

Rotates a selected object or group of objects.

Using the Command

Select the object or objects you want to rotate, and then choose the **Rotate** command. The selection can be rotated on any axis at any angle. The window displays several options for the axis of rotation:



Rotation Axis: Set the rotation about one of the three main axes, an axes defined by two points, or a line. You can also rotate about an axis normal to a plane.

Through Selection Handle: The rotation axis always shifts so that it passes through the selected object's primary selection handle. Otherwise the rotation axis passes directly through the first point(s) that you set.

Rotation Angle: After entering the angle, press Enter to return to the drawing area. Press Enter again to rotate the selection.

Increment: Sets the increments of rotation used in Drag Mode.

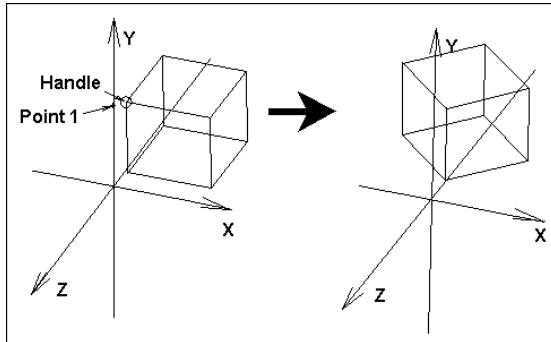
Drag Mode: Rotation can be controlled using the mouse. To rotate the selection using the mouse, move the cursor into the drawing area. Click and hold the left mouse button. Drag the cursor to rotate the selected object(s). When the selection reaches the desired rotation angle, release the mouse button.

*NOTE: If **Through Selection Handle** is disabled when using **Drag Mode**, one or two points (depending on the rotation axis) are required to establish the axis of rotation before you drag the mouse for the rotation.*

Make a Copy: A copy of the selection is rotated and the original remains in its current location.

Example: Rotate a box 45 degrees on the Y axis.

Select a box. Choose the **Rotate** command. Enter 45 in the **Rotation Angle** field in the window. Select the Y Axis for **Rotation Axis**. Make sure **Drag Mode** is disabled. and press Enter. The box rotates 45 degrees around the Y axis.



Scale

Edit / Selection Edit / Scale

Shortcut Key: S

Toolbox Icon:

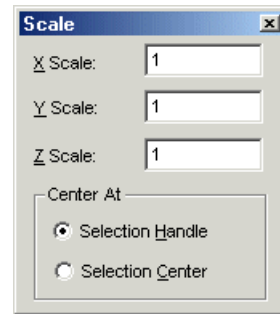


Point 1: Scale center

Scales a selection along the X, Y, or Z axis. In other words, you can "stretch" the selected objects to make them taller, shorter, longer, wider, etc.

Using the Command

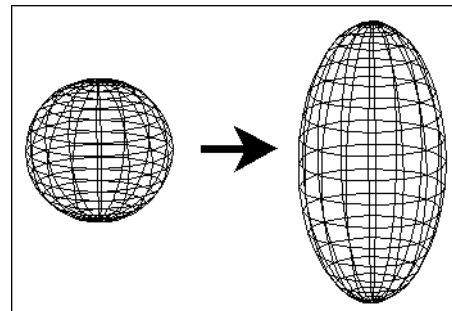
Select the object and choose the **Scale** command. In the window enter the scale factors for the X, Y, and Z directions.



Center At: Determines whether the selection handle or selection center will remain in its current location when the object is scaled.

Example: Scale an object so that it is twice as large along the Y axis.

Select the object and choose the **Scale** command. Enter 1 for the X and Z scales, and 2 for the Y scale. Press Enter. The object is redrawn with the new scale factors.



NOTE: To use resizing handles to scale objects, see Use Resizing Handles.

Skew

Edit / Selection Edit / Skew

Toolbox Icon:



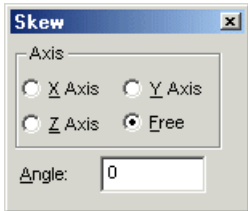
Point 1: Reference point

Point 2: New position for reference point

Slants one or more selected entities along the X, Y, Z or "free" axis.

Using the Command

Select the entities to be skewed and choose the **Skew** command. Choose the axis on which the selected objects are to be skewed.

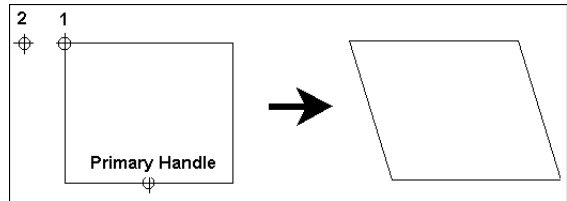


Set a reference point in the drawing. A rubber-band representation of the entity being skewed shows how the entity will be drawn. Set a second point to relocate the reference point.

*NOTE: The Primary Selection Handle of the selected object serves as an anchor during the **Skew** command. If this handle is located in the center of the object, both sides of the object move from their original positions.*

Example: Skew a box.

Select the box. Set the Primary Selection Handle on the center of the bottom side of the box so it will not be moved. Choose the **Skew** command. Select the X axis in the window. Set a reference point at the top left corner of the box. A rubber-band box is drawn to show how the box will be drawn. Set a second point 5 Drawing Units to the left of the reference point. The box is skewed so the top left corner of the box is 5 Drawing Units to the left of its original position.



Selection Edit Zoom

Menu: Edit / Selection Edit / Zoom

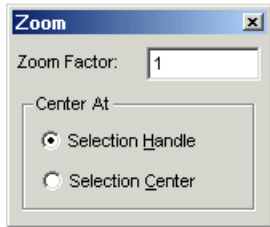
Toolbox Icon:



Zooms, or scales, selected entities to be larger or smaller. It is similar to the **Scale** command, but it changes the scales for all axes the same amount. This keeps the proportions of the object intact.

Using the Command

Select the object to be zoomed. Choose the **Select Edit / Zoom** command and enter the **Zoom Factor** in the window:



The **Center At** option determines whether the selection handle or selection center will remain in its current location when the object is zoomed.

Press the Enter key. The object will be redrawn to the new Zoom Factor.

NOTE: To use resizing handles to increase or decrease an object's, scale see Use Resizing Handles.

Set Handles

Edit / Selection Edit / Set Handles

Shortcut Key: Ctrl+H

Point 1: First handle

Point 2: Second handle (optional)

Point 3: Third handle (optional)

Selection Handles are reference points for one or more entities that have been selected. The selection handles are used to help you move, copy, or manipulate the selection.

Using the Command

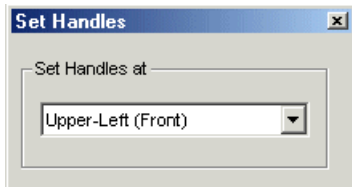
Select the object on which you want to set the handles. Choose the **Set Handles** command. Set from one to three handles on the object. The first handle set is the primary handle and is designated by a bull's eye. The secondary handle is represented by a small 1 enclosed in a circle. The tertiary handle is a small 2 enclosed in a circle. Press Enter to end the command if you set one or two handles. Now you can manipulate the object using the handles.

For example, when you copy an object with the **Duplicate** command, the first selection handle is located at the destination point you set. If you place the selection handle at a convenient location, it can be much easier to position the copy.

The second selection handle is used with some commands to set the size and angle of a selection. The **Duplicate** and **Move** commands use the second handle.

The third handle is sometimes used to position the selection in 3D space. For example, the **Move** command locates the first two handles on the first two points set, and then rotates the selection so that the three handles lie on the same plane as the three points set.

If you are only going to set one selection handle at a commonly used location for the purpose of moving or copying the selection, just select the location from the window. A single handle is placed at the location you selected.



Example: Set handles on a box and move it.

Select the box. Choose the **Set Handles** command. Set the first handle on the upper right corner, the second on the lower left corner, and the third on the lower left rear corner. Next, choose the **Move** command and deselect the **Use Original Scale** option. Move the cursor to the desired new location for the box and set a point for the first handle. Move the cursor away from the first point. A rubber-band scaling box is drawn using the cursor location as Point 2. When you have scaled the object to your liking, set the second handle. Set a point for the third handle to orient the object in 3D space.

Display Order

Edit / Selection Edit / Display Order

Changes the display order of entities so that the correct portions of overlapping entities are displayed and printed. This command affects entities in 2D Mode and entities that lie on the same XY plane in 3D Mode.

Using the Command

Select the entity to be moved and choose:

Move to Front: brings the entity to the very front

Move to Back: brings the entity to the very back.

Move After: moves the selected entity behind another entity (select the entity to move behind).

Move in Front of: moves the selected entity in front of another entity (select the entity to move in front of).

Example: Send a filled circle in front of a filled box.

Select a filled circle from a stack of entities and choose the **Move in Front of** command. Set a point on the filled box. The filled circle is sent directly in front of the filled box.

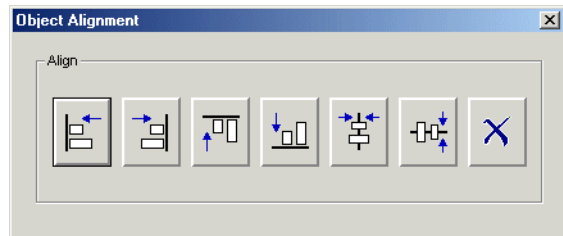
Object Alignment

Edit / Selection Edit / Object Alignment

Adjusts the alignment of selected items.

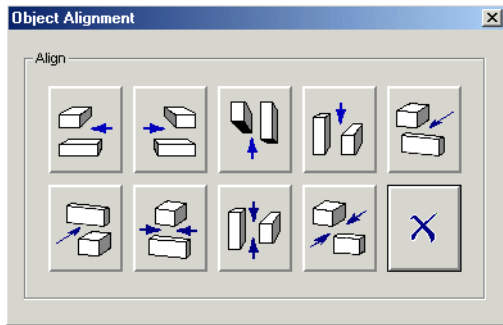
Using the Command

Select the items to be rearranged. Select the **Object Alignment** command. In 2D, these are the alignment options:



Select to align along top edges, bottom edges, left edges, right edge, or to center vertically or horizontally. Click the X to cancel.

In 3D, these are the alignment options:



You can align along any of the six edge options, or to center in X, Y, or Z.

Multiple Copies

These are commands that you can use to make multiple copies of objects, either as a grid or in a circular pattern.

Array

Menu: Draw / Array

Toolbox Icon:



Point 1: Location and direction of first copy (relative to handle)

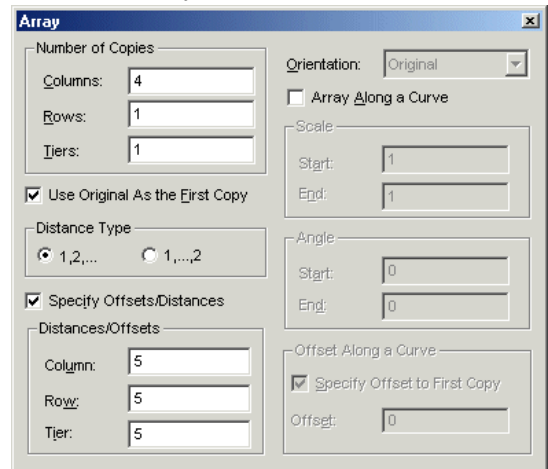
Point 2: Axis of Rows (relative to handle on the original)

Point 3: Axis of Tiers (relative to handle on the original)

Copies a selected object a specified number of times in as many as three directions. The object you want to copy must be selected before you use this command.

Using the Command

First select the object you want to copy. Then choose the **Array** command.



To copy in a single direction or dimension, enter the number of copies in **Columns**, and enter 1 for **Rows** and **Tiers**. Set a point for the location of the second copy of the object. (The first copy is the original unless the **Use Original as the First Copy** option is disabled.) The copy is positioned so that the point that corresponds to the selection handle on the original is located on that point. Subsequent copies are placed at the same direction and distance as the first point from the selection handle of the original. In other words, all copies are evenly spaced based on the spacing of the first two copies.

To copy in two dimensions, enter the number of columns and rows, leaving **Tiers** set to 1. Set a point for the offset (the distance and direction) of

copies in the first column. Then set a second point for the offset of the columns themselves. For example, you might want to set the first point to the right of the selection handle, and the second point directly below the selection handle.

To make a 3D array, add a number for tiers and add a third point for the offset of each tier.

Use Original as the First Copy: Determines whether or not the selection is used as the first copy for the directions of the array. If disabled, the first point set for the command determines the position of the first copy of the array. This copy and the subsequent array can be set far from the original. The second point set for the command determines the columns, the third determines the rows, and the fourth determines the tiers.

Distance Type: Changes the way that distances are determined for the array. When the 1,2,... distance type is used, the distance between the points set determine the distance between the first and second copies of a given direction. When the 1,...,2 distance type is used, the distance between the points set determine the distance between the first and last copies of a given direction.

Specify Offsets/Distances: Lets you set the distance from one copy to another in Drawing Units. The points set then only dictate the direction of the copies from the original. Enter the offset distances for the copies in the appropriate fields in the window.

Orientation: The alignment of the object to be arrayed. For most of these choices to be used properly, you must correctly place your handles. Imagine the object you are copying is a man. The first handle is the man's feet. The second handle is the direction he is facing. The third handle is his

head. His feet always touch the curve. His head is always in the "up" direction relative to the plane of the curve. He can walk forward or backward along the curve, or he can step along the curve sideways while facing to the left or the right of the curve. He can also slide along the curve while always maintaining his original orientation.

NOTE: The curve's sense of "up" is calculated once for the entire curve; it is not evaluated separately at each point on the curve, so curves that do not lie in a plane may give unexpected results. Also, it may sometimes go in an unexpected direction. What you expected to be "up" might in fact be "down." Be prepared to change your selection handles accordingly if you get unexpected results.

- **Original:** Maintains the orientation of the original object, regardless of which direction the curve twists and turns. For this orientation, only the first selection handle is used; the copied object is never rotated.
- **Forward, Backward:** The object along the curve so that it is always "facing" in a forward or backward direction along the curve. Objects rotate as they move along the curve to maintain their forward/backward orientation relative to the curve.
- **Left, Right:** Moves each copy along the curve so that it is "facing" to the left or right of the curve's direction of travel.
- **Perpendicular:** Treats the handles differently than other options. For this option, the plane created by the selection handles is always rotated to be perpendicular to the curve's direction of travel. This option gives the user very little control over the

final orientation of the copy, and is retained for backward compatibility with older versions of DesignCAD.

Array Along a Curve: Curve arrays always start with a fresh copy of the selected object, so the **Use Original as the First Copy option** is disabled. Set the number of copies to be drawn along the curve in the **Columns** field and set a point on the curve the array is to be drawn along. The easiest way to set the point on the curve for the array is to use the **Gravity** command to snap to one of the endpoints of the curve.

Scale: Specify the starting and ending scales for the arrayed object(s). Intermediate objects are scaled smoothly between the starting and ending scales.

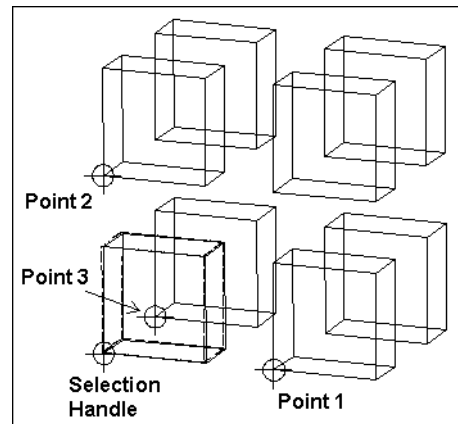
Angle: If arraying along a curve, and an orientation of **Forward**, **Backward**, **Left**, or **Right** has been chosen, the selected object can also be rotated about the direction of travel. This lets the array spin around the curve. The start angle is how much the imaginary man is leaning relative to "up" when he first starts his walk along the curve. The end angle is how much he is leaning at the end of his walk. Intermediate copies are tilted proportionately.

Offset Along a Curve: If the **Specify Offsets/Distances** and **Array Along a Curve** options are both enabled, this option can be enabled and used to determine the distance from the beginning of the curve to the first copy in the array. Two points need to be set on the curve when this option is used. The first point specifies the curve to be used for the array. The second point determines the direction of travel of the array

along the curve. The end of the curve that designated as the beginning is determined by the direction travel.

Example: Make two columns, two rows, and two tiers of a box.

Select the box and choose the **Array** command. Enter 2 for each direction (**Columns**, **Rows**, and **Tiers**). Set a point directly to the right of the selection handle, and a second point above it. Finally, set a point on the third axis, away from the handle on the original box. The box will be duplicated in a 2 x 2 x 2 formation.



Circular Array

Menu: Draw / Circular Array

Toolbox Icon:



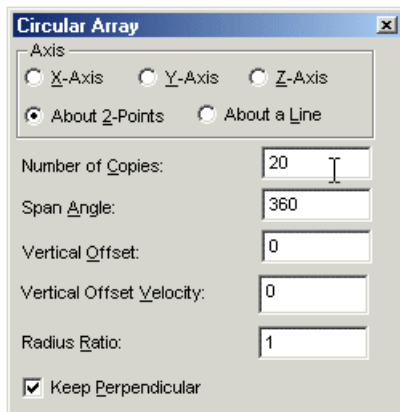
Point 1: Center of rotation

Point 2: Axis of revolution (Used only with Two-Point Axis mode)

Copies a selected object a specified number of times in a circular pattern.

Using the Command

Select the object to copy, then choose the **Circular Array** command. The following window appears:



Axis: The axis of revolution. You can set the rotation along the X, Y, or Z axis, an axis defined by two points, or one defined by an existing line. Only the **About 2Point** option requires you to set a second point. With the **About a Line** option you set a point on an existing line and the command

uses that line as the axis of rotation. The other options will use your first point as the location of the axis.

Number of Copies: The total number of copies, including the original.

Span Angle: The angle in which the copies are to be placed. For example, you can use 360° to copy an object in a complete circle, such as the bolts on a wheel. You can use 180° to copy in a semicircle.

Vertical Offset: This is the distance along the axis of revolution between the first copy and the last copy. The offset can be used to copy objects in a spiral. To make a spiral with more than one revolution, use a span angle of more than 360° . For example, to make a spiral with two complete revolutions, you can enter 720° for the span angle. Remember, the number of copies and the offset are the total for the entire operation, not just one revolution.

Vertical Offset Velocity: The ratio between the initial pitch of the offset and the final pitch of the offset. If a value of 5 is used, the final pitch will be 5 times greater than the initial pitch. If a value of 0 or 1 is used, the pitch will remain constant.

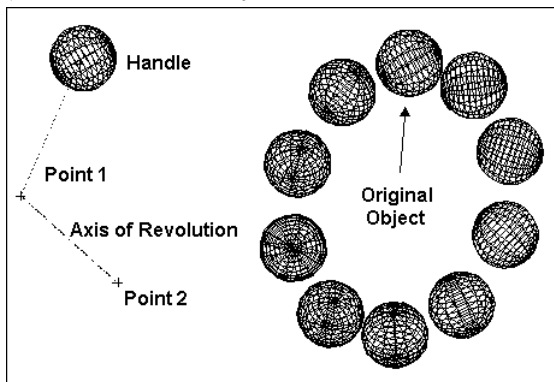
Radius Ratio: The distance from the selection handle to the axis, used for the array radius at the beginning of the array. The final radius for the array equals the beginning array radius multiplied by the Radius Ratio. If a value of 1 is used, the radius stays the same throughout the array. If a value greater than 1 is used, the distance from the axis to the duplicated items increases throughout the array. If a value less than 1 is used, the distance from the axis to the duplicated items decreases throughout the array.

Keep Perpendicular: The copies will rotate with the array. If disabled, the copies will still be copied in a circular pattern, but they will maintain the same orientation as the original.

Example: Copy an object 10 times in a circular pattern.

Select the object you want to copy. Next, choose the **Circular Array** command. Enter the following settings: **Number of Copies** = 10, **Span Angle** = 360, **Vertical Offset** = 0.

Use the **About 2 Points** option. Set a point a few inches below the object for the center of rotation and another, offset from the first, for the axis of revolution. It may help to think of this as swinging a ball on a string: The selection handle is where the string is connected to the ball, the center of rotation (Point 1) is where your hand holds the string, and the axis of revolution (Point 2) is the direction of your arm.



Trimming and Extending

This section covers all the commands used for trimming objects with respect to other objects. The edited object is either trimmed at the trimming object, or extended to meet the trimming object.

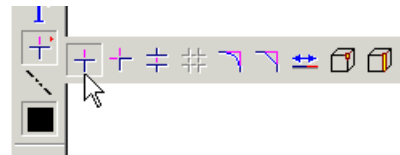
*NOTE: In 2D Mode, the **Trim** commands ignore the 3D aspects of any existing lines, and treat them as flat projections onto the XY plane. This means you can trim lines that would never meet in 3D space against each other's Front-View projections in 2D space. This is a powerful feature, but you must pay careful attention to what you are doing.*

Trim One Line

Menu: Edit / Trim/Extend / Trim One Line

Shortcut Key: E

Toolbox Icon:



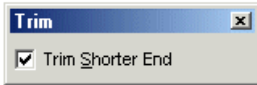
Point 1: Line to be trimmed

Point 2: Line to trim against

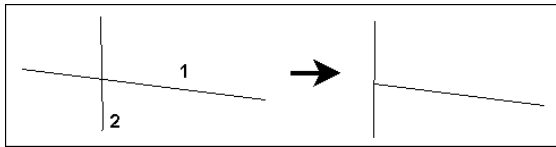
Trim (or extends) a line to its intersection with another line. This command also works with curves and arcs.

Using the Command

Choose the **Trim One Line** command. Click **Trim Shorter End** if you want to trim the shortest end of the line. Otherwise, the part of the line that you click is kept, and the other part is removed.



Set a point on the line to be trimmed and a point on the line to be trimmed against. The first line is cut off at its intersection with the second. If the first line does not intersect the second, it is extended until they meet.



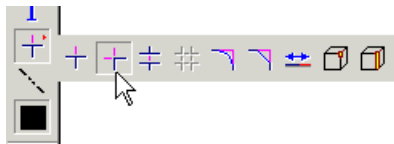
NOTE: If you are in 2D Mode, you can trim two lines that don't intersect by trimming their projections on the XY plane.

Trim Two Lines

Menu: Edit / Trim/Extend / Trim Two Lines

Shortcut Key: Ctrl+E

Toolbox Icon:



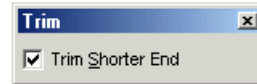
Point 1: First line to be trimmed

Point 2: Second line to be trimmed

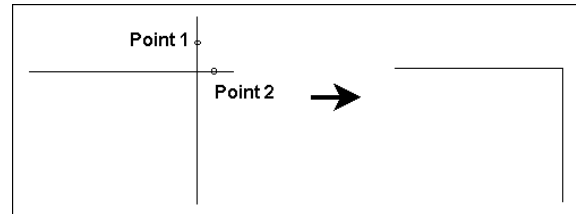
Trims (or extends) two lines at their intersection, forming a clean corner with no overlap. This command trims lines, curves, and arc entities.

Using the Command

Select the **Trim Two Lines** command. Click **Trim Shorter End** if you want to trim the shortest ends of each line. Otherwise, the part of the line that you click is kept, and the other part is removed.



Set a point on each line to be trimmed. The lines will be trimmed at their intersection. If the lines do not intersect, they will be extended to the point of intersection.

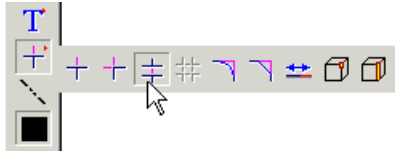


NOTE: If you are in 2D Mode, you can trim two lines that don't intersect by trimming their projections on the XY plane.

Trim Between Two Lines

Menu: Edit / Trim/Extend / Trim Between Two Lines

Toolbox Icon:



Point 1: Line to be trimmed

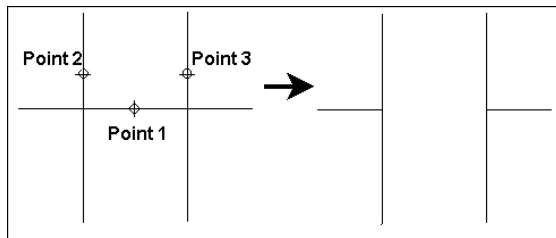
Point 2: First intersecting line

Point 3: Second intersecting line

Removes a segment of a line between its intersections with two other lines.

Using the Command

Choose the **Trim Between Two Lines** and set a point on the line to be trimmed. Set a point on the first intersecting line. Set a point on the second intersecting line. The line is trimmed between the other two lines.

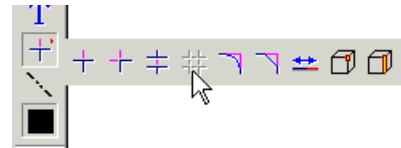


NOTE: *In 3D mode, all the lines must lie in the same plane for this command to trim them. In 2D Mode, the command will trim the lines along their XY projections.*

Trim Double Lines 2D only

Menu: Edit / Trim/Extend / Trim Double Lines

Toolbox Icon:



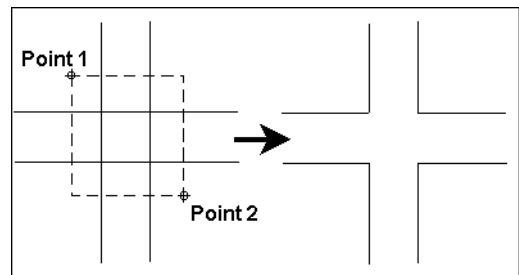
Point 1: Corner of area containing lines to be trimmed

Point 2: Opposite corner of area containing lines to be trimmed

Trims two sets of double lines at their intersection. See Double Line Mode.

Using the Command

Choose the **Trim Double Lines** command. Set a point in the corner of the area containing the lines to be trimmed. Set a point in the opposite corner. The two sets of lines will be trimmed to their intersection. Depending on the configuration of the lines, they will be trimmed to a "+," a "T," or an "L" shape.



Trim Multiple Lines

Menu: Edit / Trim/Extend / Trim Multiple Lines

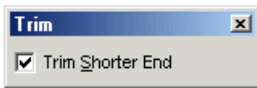
Point 1-n: Lines to be trimmed

Point n+1: Line to trim against

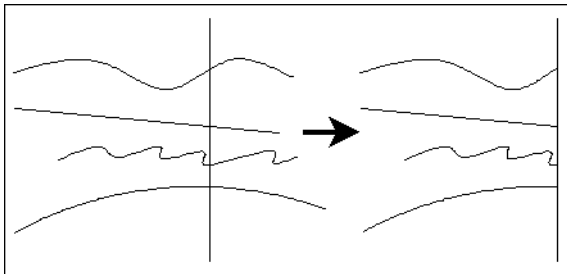
Trims multiple lines at their intersection with another line. This command trims arc, curve, and lines.

Using the Command

Select the **Trim Multiple Lines** command. Click **Trim Shorter End** if you want to trim the shortest end of each line. Otherwise, the part of the line that you click is kept, and the other part is removed.



Set a point on each line to be trimmed. Then set a point on the line to trim against. Press Enter. The lines will be cut off at their intersection with the line on which the last point was set.



NOTE: If you are in 2D Mode, you can trim two lines that don't intersect by trimming their projections on the XY plane.

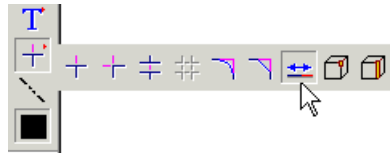
Extend

Menu: Edit / Trim/Extend / Extend

Point 1: Endpoint

Point 2: New Location

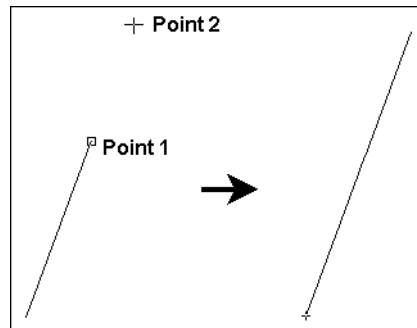
Toolbox Icon:



Extends or shortens a line by moving one of the points defining the end of the line.

Using the Command

Choose the **Extend** command and set a point on the endpoint to be extended. Move the cursor to the new location for the endpoint.



Extend by Distance

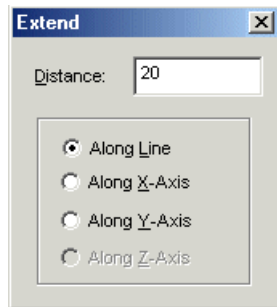
Menu: Edit / Trim/Extend / Extend by Distance

Point 1: Endpoint

Extends or shortens a line by moving one of the points defining the end of the line a specified linear distance, to a relative X coordinate, to a relative Y coordinate, or to a relative Z coordinate. The line maintains the same angle after it has been extended.

Using the Command

Choose the **Extend by Distance** command and enter the distance to be added to the line.



Specify whether the extension for the line is to be measured along the line or with respect to the X-Axis, the Y-Axis, or the Z-Axis Now, set a point on the endpoint to be extended.

Point Move

Menu: Edit / Trim/Extend / Point Move

Shortcut Key: * (asterisk)

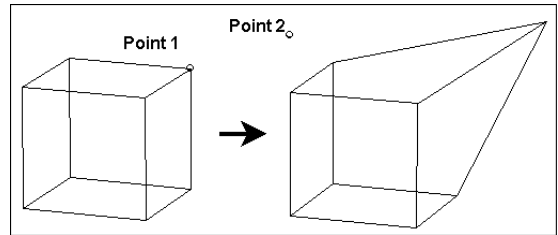
Point 1: Set a point on the point to be moved

Point 2: Set a point for the new location of that point

Moves a point in the drawing.

Using the Command

Choose the **Point Move** command. Set a point directly on a point in the drawing. 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.



Join Endpoints

Menu: Edit / Trim/Extend / Join Endpoints

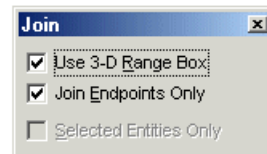
Point 1: First corner of area to be joined

Point 2: Second corner of area to be joined

Takes all of the points enclosed within a bounding box and moves them to a single point at the geometric center of the original endpoints. This command works with lines, and curves, as well as arcs drawn as lines.

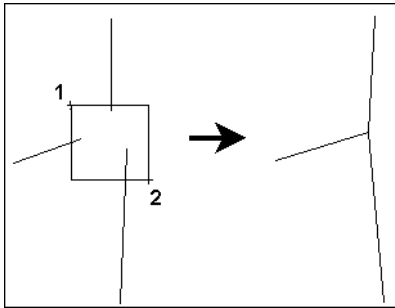
Using the Command

Choose the **Join Endpoints** command.



Join Endpoints Only: Joins only the endpoints of the lines. Otherwise all the points included in the bounding box will be joined.

Drag a box around the endpoints of the lines to be joined. Do not completely enclose the lines or they will be reduced to a single, tiny point. The bounding box should enclose only the points of the lines you want to join, not the lines themselves.



Chamfering / Filleting

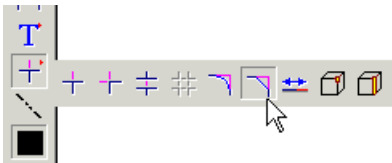
These commands affect object corners. You can round corners using Fillet tools or make angular corners using Chamfer.

Chamfer

Menu: Edit / Trim/Extend / Chamfer

Shortcut Key: Ctrl+F

Toolbox Icon:



Point 1: One of the lines on the corner to be chamfered

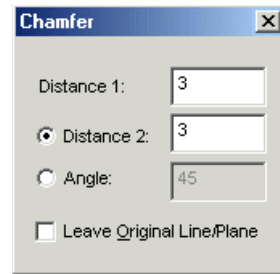
Point 2: The other side of the corner

Cuts a segment off a corner and replaces the corner with a flat face of specified depth. This command works on 2D line or plane entities.

NOTE: To chamfer the corner of a solid object, see Cut Corner or Slice.

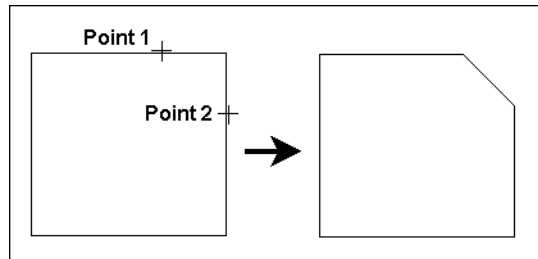
Using the Command

Choose the **Chamfer** command. Enter the chamfer depths for both sides of the corner, or enter one distance and **Angle**.



Leave Original Line/Plane: Draws the chamfer but leaves the lines that formed the corner.

Set a point on one of the lines to be chamfered, then a point on the other. If the lines are not connected, DesignCAD extends them as needed.

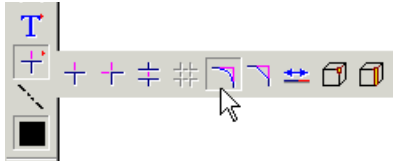


Fillet

Menu: Edit / Trim/Extend / Fillet

Shortcut Key: F

Toolbox Icon:



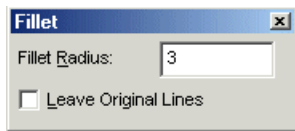
Point 1: A point on one of the lines to be filleted

Point 2: A point on the second line to be filleted

Rounds a corner of a plane or two lines.

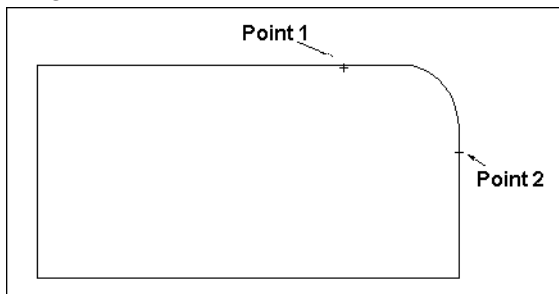
Using the Command

Choose the **Fillet** command and enter the fillet radius.



Leave Original Line/Plane: Draws the fillet but leaves the lines that formed the corner.

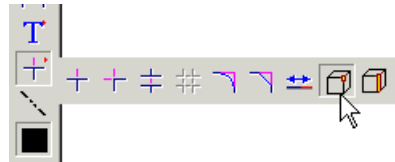
Set a point on one of the lines to be filleted, then a point on the other. If the lines are not connected, DesignCAD extends them as needed.



Fillet Corner ^{3D}only

Menu: Edit / Trim/Extend / Fillet Corner

Toolbox Icon:

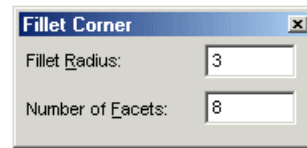


Point 1: Corner to be filleted

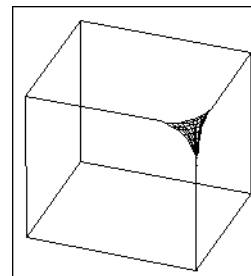
Rounds rectangular corners of solids.

Using the Command

Choose the **Fillet Corner** command and specify the radius and number of facets.



Then set a point on the corner to be filleted.

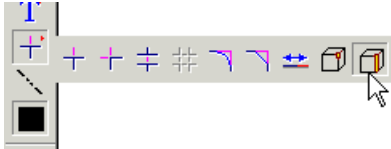


NOTE: You cannot use this command to round a corner where more than three faces meet, such as the point of a cone.

Fillet Edge 3D only

Menu: Edit / Trim/Extend / Fillet Edge

Toolbox Icon:

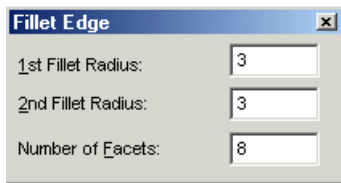


Point 1: Edge to be filleted

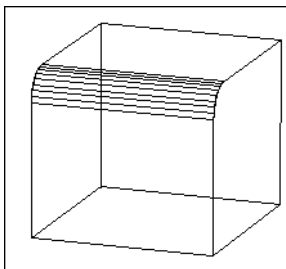
Rounds off rectangular edges of solid objects. You may specify the number of facets for the rounded edge and the radius for each end of the edge to be filleted.

Using the Command

Choose the **Fillet Edge** command. Enter the radius for the start and end of the edge, and number of facets, in the window.



Set a point on the edge (not at a corner) you want to fillet.



NOTE: This command requires sharp edges. Once you have filleted an edge of a cube, for example, the adjacent edges no longer meet at a flat face but at a curved surface. So the edges adjacent to a filleted edge cannot also be filleted.

Cut Corner 3D only

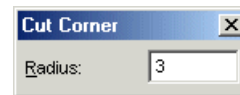
Menu: Edit / Trim/Extend / Cut Corner

Point 1: Corner to be cut

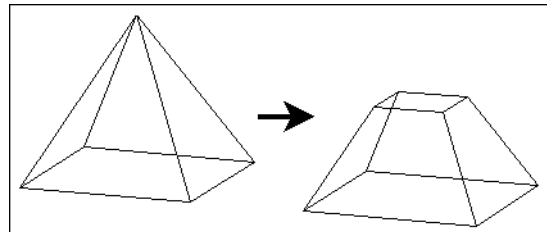
Cuts a portion off of the corner of a solid and replaces the corner with a flat face at a specified depth.

Using the Command

Choose the **Cut Corner** command. In the **Radius** field, enter the depth - the distance from the corner along all of the edges that form the corner - to the flat surface.



Set a point on the corner. The corner is cut off to the specified depth, or radius.



Cut Edge 3D only

Menu: Edit / Trim/Extend / Cut Edge

Point 1: Edge to be cut

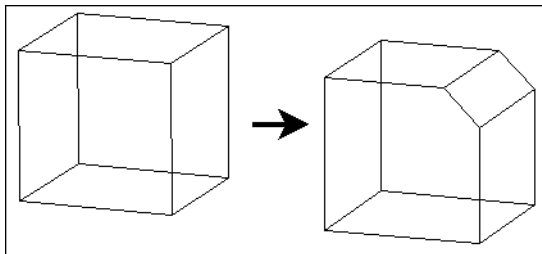
Cuts a portion off of the edge of a solid (chamfering the edge) and replaces the edge with a flat face at a specified depth.

Using the Command

Choose the **Cut Edge** command. In the **Radius** field, enter the depth - the distance from the edge along the two surfaces that form the edge - to the flat surface.



Set a point on the edge. The edge is chamfered to the specified depth, or radius.



Dividing Entities

These commands are used for breaking entities into sets of smaller entities.

Segment

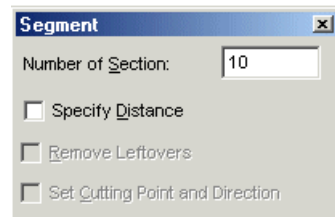
Menu: Edit / Trim/Extend / Segment

Point 1: Line or curve to be segmented

Divides a line or curve into specified segment lengths or a specified number of segments of equal length. Curves will automatically be subjected to **Vector Convert** (see Vector Convert) before being segmented.

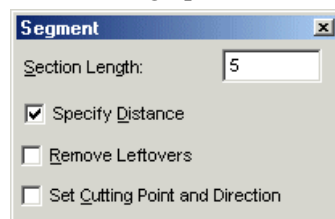
Using the Command

Choose the **Segment** command and enter the values in the window:



Number of Section: The number of segments.

Specify Distance: The **Segment** command can also divide the line or curve into segments of a specified length. If this is checked, the window will have the following options:



Remove Leftovers: Any bits of the line or curve that are shorter than the desired length are removed from the ends.

Set Cutting Point and Direction: Specify an exact point on the line or curve which should be the start of a segment, and an optional second point for the direction to cut. If only one point is set, the entire line or curve is broken into segments of the desired length, with cuts running in both directions from the point set. If the second point is set, only part of the line is cut into segments, beginning at point 1 and running in the direction of point 2. The remainder of the line on the other side of point 1 is left intact.

Set a point on the line or curve to be segmented. The object is broken into the requested number of sections or section lengths.

Cut Plane 3D only

Menu: Edit / Cut Plane

Point 1: Cutting line

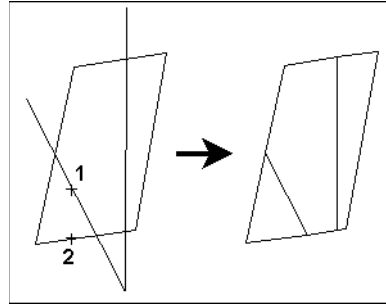
Point 2: Plane to be cut

Slices a plane into two or more pieces along a line.

Using the Command

The **Cut Plane** command requires a plane to cut and a line to cut it with. Choose the **Cut Plane** command and set a point on the cutting line, and then a point on the plane to be cut. The cutting line, or part of it, is erased during this command,

so make a copy if necessary. If you forget, you can **Undo** the command, copy the line, and re-cut the plane.



NOTE: To erase the cut lines that remain after this command, see Hidden Edge and Hidden Edge by Section.

Converting Entities

These commands are used to convert entities of one type to another type.

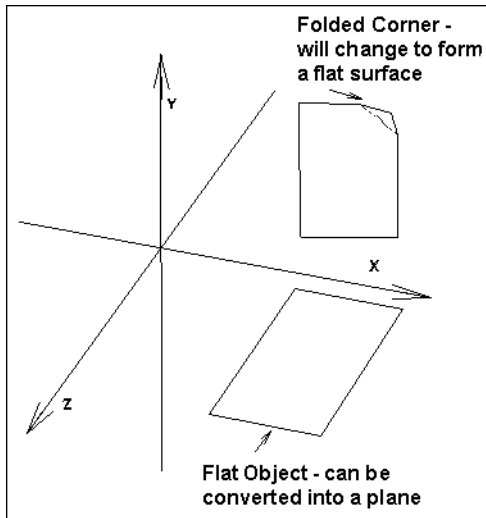
Make Plane

Edit / Selection Edit / Convert / Make Plane

Converts lines, curves, circles, and arcs into a plane. Planes can be shaded, whereas lines, circles, and arcs cannot.

Using the Command

Select the line or lines to be converted to a plane. The selected lines should all form a flat surface, and have common endpoints.



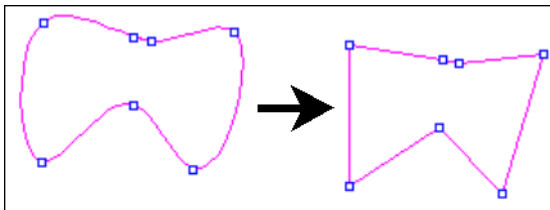
Curve to Line

Edit / Selection Edit / Convert / Curve to Line

Changes a curve entity into a line using the same points as those that defined the curve.

Using the Command

Select the curve that you wish to convert to a line. Choose the **Curve to Line** command. The selected curve is changed to a line.



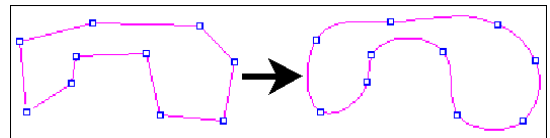
Line to Curve

Edit / Selection Edit / Convert / Line to Curve

Converts all selected lines to curves, using the vertices of the lines as defining points for the curves.

Using the Command

Select the line to convert. Choose the Line To Curve command. The line is converted to a curve.



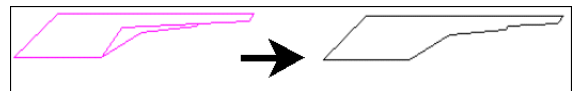
Weld

Edit / Selection Edit / Convert / Weld

Combines the area of two or more closed objects that are not solids. These objects must touch or overlap. The combined area may then be manipulated as if it had been drawn that way.

Using the Command

Select the entities to be welded together. Choose the **Weld** command. The area of the objects is combined, and any lines that separated the objects are deleted.



Explode

Edit / Selection Edit / Convert / Explode

Converts any selected composite objects into separate entities. This command affects grids, dimensions, and symbols. Afterwards, the parts of the former composite objects can be manipulated individually.

Symbols are broken apart into separate drawing entities, just as if they had been drawn originally right in the current drawing. Grids are converted into planes, and planes are converted to vectors. Dimensions are converted into text, vectors, and arrows.

If a group is selected when the **Explode** command is chosen, objects in the group will be exploded as if they were not a part of a group, but the group will remain intact. To select individual entities, it will be necessary to use the **Group Explode** (see Group Explode) command to explode the group. To explode a group so individual objects may be selected, but still remain composite objects, use **Group Explode** instead of **Explode**.

Using the Command

Select the object(s) to be exploded and choose the **Explode** command.

Vector Convert

Edit / Selection Edit / Convert / Vector Convert

Point 1-n: Entities to be changed to line entities

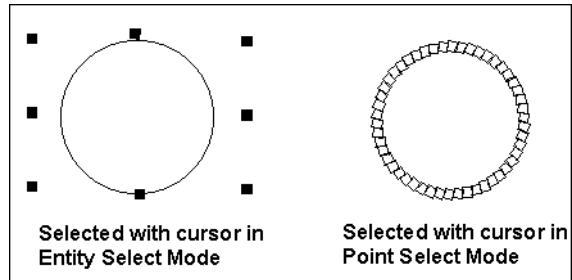
Converts selected entities such as arcs, circles, curves, and planes to vector entities.

Using the Command

Select the entity to be converted and choose **Vector Convert**. The entity is converted to a series of vectors, or short line segments. The new entity has more points than the original but looks the same.

Example: Convert a circle to vectors.

Select the circle you want to convert to vectors and choose **Vector Convert**. DesignCAD automatically converts the entity to vectors. Although the entity does not change appearance, you can see the effect of conversion to vectors by choosing **Point Select Mode** and selecting the entity.



Text Convert

Edit / Selection Edit / Convert / Text Convert

Converts selected 2D text to 3D text and vice versa.

Using the Command

Draw some 2D text using the **Text** command (see Text) with the **Draw as 2D Text** option selected. Select the 2D text. Now select the **Text Convert** command. The selected 2D text is changed to 3D text. Leave the text (now three-dimensional) selected. Select the **Text Convert** command again; the 3D text is converted back to 2D text.

Double Line to Plane

Edit / Selection Edit / Convert / Double Line to Plane

Converts double-line entities into planes (surfaces). Planes can be shaded, whereas double lines, circles, and arcs cannot. See Double Line Mode.

Using the Command

Select the double line entity to be converted to a plane. Select the **Double Line to Plane** command. The double line entity is converted to a plane.

Double Line to Wide Line

Edit / Selection Edit / Convert / Double Line to Wide Line

Converts double-line entities into wide lines. See Double Line Mode.

Using the Command

Select the double line entity to be converted to a wide line. Select the **Double Line to Wide Line** command. The double line entity is converted to a wide line.

NOTE: Regardless of which alignment was used when creating the double line, DesignCAD has to use these points to create a center-aligned wide line. This can cause the appearance of the resulting wide line to vary slightly from the original double line.

2D Boolean Operations

These are combination commands that are used on closed 2D objects, or 2D planes.

Polygon Add

Menu: Edit / Polygon Edit / Polygon Add

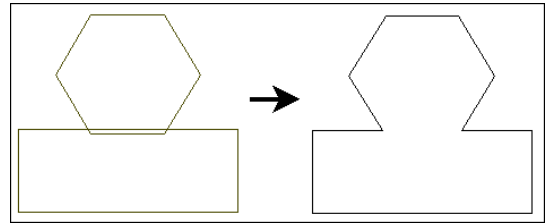
Point 1: First Polygon

Point 2: Polygon to be added to first

Adds two or more polygons together and eliminates the overlapping edges.

Using the Command

Choose the **Polygon Add** command. Set a point on the two polygons to be added together. The polygons are combined into a single polygon, and the boundaries "internal" to the resulting polygon are removed.



NOTE: For best results, the polygons should overlap, rather than meet exactly at an edge or point.

To combine more than two polygons, select the polygons first and then select the **Polygon Add** command. Click **Use Selected Objects**.

Polygon Subtract

Menu: Edit / Polygon Edit / Polygon Subtract

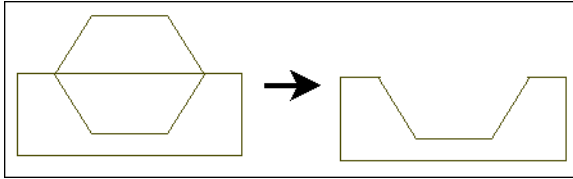
Point 1: Polygon to be removed (template)

Point 2: Polygon to be subtracted from

Removes one or more polygons from another polygon.

Using the Command

Choose the **Polygon Subtract** command and set a point on the polygon to be subtracted. Set a second point on the polygon to be subtracted from. The first polygon is removed from the second polygon.



To subtract more than one polygon, select the polygons that you want to subtract first. Then choose the **Polygon Subtract** command and click on the target polygon.

NOTE: To erase the cut lines that remain after this command, see Hidden Edge and Hidden Edge by Section.

Polygon Intersect

Menu: Edit / Polygon Edit / Polygon Intersect

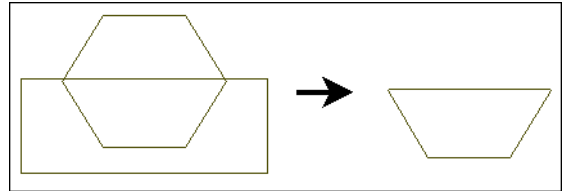
Point 1: First polygon

Point 2: Second polygon

Removes all parts of two overlapping polygons except the part that both polygons share. This part forms a new polygon.

Using the Command

Choose the **Polygon Intersect** command. Set a point on each of the intersecting polygons. The overlapping area remains.



Polygon Exclusive Or

Menu: Edit / Polygon Edit / Polygon Exclusive Or

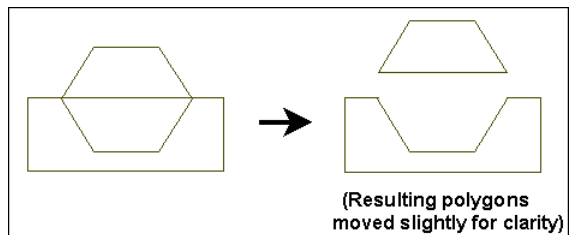
Point 1: First polygon

Point 2: Second polygon

Subtracts the area in common between two polygons. If a portion or portions of both of the polygons remain, the portions are still recognized as a part of their original polygon.

Using the Command

Choose the **Polygon Exclusive Or** command. Set a point on each of the polygons. The common area between the two polygons is removed. The only portions that remain are portions that were exclusive to one or the other of the two original polygons.



Section Edit

These are editing commands that are performed on a selected section of the drawing.

Stretch

Menu: Edit / Section Edit / Stretch

Point 1: 1st corner of area to be stretched

Point 2: 2nd corner of area to be stretched

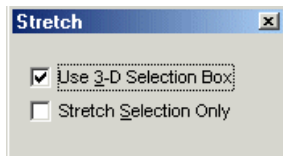
Point 3: Reference point

Point 4: Stretched position of reference point

Stretches a section of a drawing from one location to another.

Using the Command

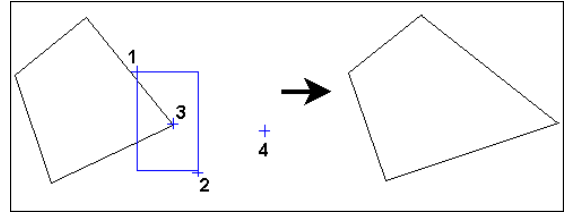
Choose the **Stretch** command. You can choose to use a 2D selection box or a 3D selection box.



The 3D selection box allows you to be more specific when you stretch 3D objects. For instance, you can use a 3D selection box to specify just the front left corner of a box.

Four points are used with the command. The first two points define a 2D or 3D bounding box that contains the part of the drawing to be stretched. The third and fourth points determine the direction and distance that part of the drawing will be stretched.

For instance, if you move the fourth point 10 Drawing Units along the X axis from the third point, then all points in the stretch region will be moved 10 Drawing Units along the X axis.



Section Delete

Menu: Edit / Section Edit / Section Delete

Shortcut Key: D

Point 1: First corner of the section to be deleted

Point 2: Opposite corner of the section to be deleted

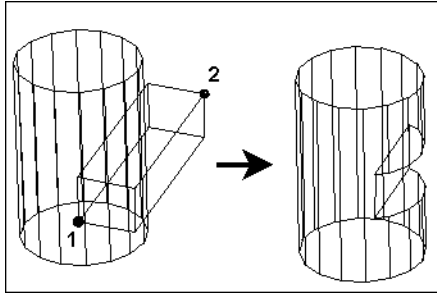
Deletes a section of the drawing. A section is a 3D box defined by two points in opposite corners. This command is useful for cutting doorways and windows out of walls.

Using the Command

Choose the **Section Delete** command. Define the section by setting two points in opposite corners of a 3D box. (If DesignCAD is in 2D Mode, a 2D box is used.) Drawing entities within the section are deleted from the drawing. An entity partly inside the box is cut off, and only the part inside the box is erased.

Example: Erase a rectangular section of a cylinder.

Select the **Section Delete** command. Set the points of the bounding box so that part of the cylinder is enclosed in it. The cylinder will be redrawn without the section that was inside the bounding box.



Section Cutoff

Point 1: One corner of the cutoff box

Point 2: Opposite corner of the cutoff box

Cuts off a section of the drawing that intersects a 3D box. It essentially separates everything inside the box from everything outside the box by cutting planes and breaking lines. Even though the drawing looks the same after the **Cutoff** command, any entity crossing the cutoff box will be cut off.

It is convenient to use this command to cut out a section of an object so it can be moved or deleted.

Using the Command

After choosing the **Section Cutoff** command, define the section by setting two points in opposite corners of a 3D box. (If DesignCAD is in 2D Mode, a 2D box is used.) Any part of the drawing that crosses the cutoff box boundary will be cut off at the intersection.

Example: Convert a circle into two separate entities.

Select the **Section Cutoff** command. Set the points for the cutoff box so that a section of the circle is inside the box. After the cutoff box disappears, place the cursor on the section of the circle that was inside the box and click the left mouse button to select it. Notice that only the section is highlighted. This is because it has been redefined as a separate entity. You can manipulate it using any command without affecting the rest of the circle.

Section Cut by Line 2D only

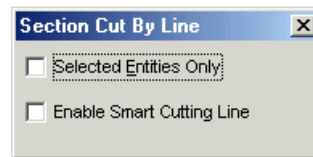
Menu: Edit / Section Edit / Section Cut by Line

Point 1: Cutting line

Cuts all 2D entities at all of their intersections with a cutting line. This command is only available in 2D Mode.

Using the Command

Choose the **Section Cut by Line** command and the following window appears:

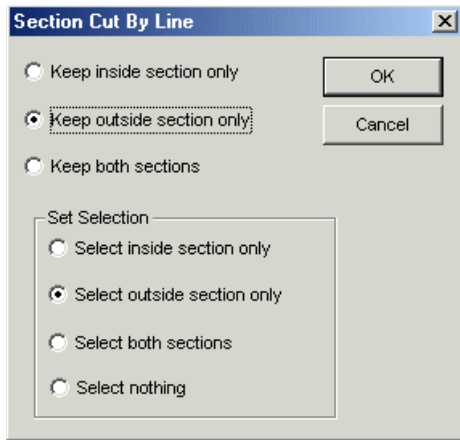


To cut only selected entities, click **Selected Entities Only**.

To cut using the cutting line and any lines that are connected to it by their endpoints, click **Enable Smart Cutting Line**.

Set a point on a line. This line is recognized as the cutting line. All entities that intersect the cutting line are cut at all intersections with it.

If the cutting line forms a closed entity (e.g., a closed box constructed of one or more lines), an additional window will appear. This window contains options for deleting and selecting portions of the cut entities.



Select the desired settings and click OK.

Measuring

These commands can be used to take measurements of entities on your drawing.

NOTE: If you want to include measurements on the drawing itself, see Dimensions.

If you want to include a measurement in the drawing, without creating a dimension, you can use the **Text** command. When the measurement calculation appears in the window, select it and **Copy** it (Ctrl+C). Then choose the **Text** command (see Text) and **Paste** (Ctrl+V) the text into the Text window.

Line Distance

Menu: Dimension / Info / Line Distance

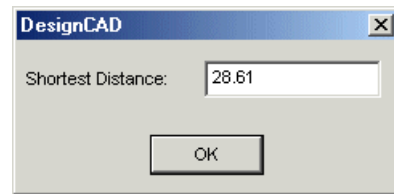
Point 1: First line

Point 2: Second line

Measures the shortest distance between two lines.

Using the Command

Select the **Line Distance** command and set points on the two lines to be measured. The shortest distance between the lines will be displayed on the screen.



Area

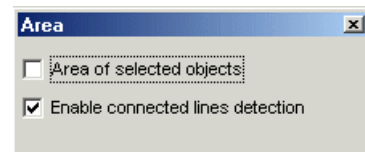
Menu: Dimension / Info / Area

Points 1-n: Determine the area to be calculated

Calculates the area of selected object or the area within a closed area defined by points.

Using the Command

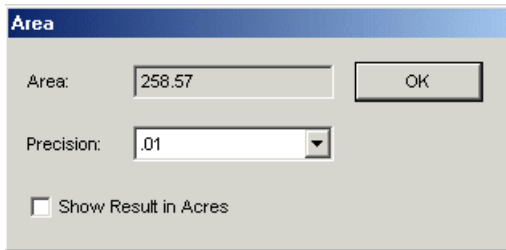
If you want to calculate the area of a selected object, select it first. Select the **Area** command. If an object is selected, you will see this window:



Area of Selected Objects: Calculates the area of any selected objects. Overlapping regions are not subtracted from the total area. If not selected, you can select the object after choosing the **Area** command.

Enable Connected Lines Detection: Calculated area between selected lines that are connected. Check this option and then check **Area of Selected Objects**.

If no objects are selected, just set points around the area you want to measure. The points will be connected with a temporary line. Press Enter when you have set all of the points. The area of the shape will be calculated and displayed in drawing units.



Precision: Controls how many digits appear after the decimal point.

Show Results in Acres: Calculates the area in acres instead of the drawing units.

Bearing

Menu: Dimension / Info / Bearing

Point 1: First point of the distance to be measured

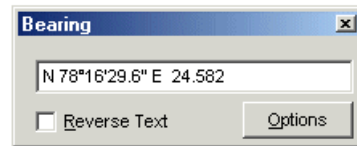
Point 2: Second point of the distance to be measured

Point 3: Insertion point for the text

Measures the bearings between two points. The bearing text is automatically inserted into the drawing.

Using the Command

Choose the **Bearing** command and set a point at one end of the distance to be measured. Set a second point at the opposite end of the distance. The bearing text is placed in the text box of the window and a rubber-band box appears at the text insertion point.



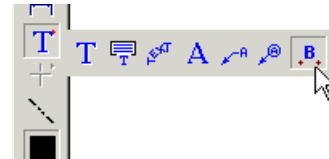
Reverse Text: If you are measuring from right to left, the text will be inserted upside-down. Checking this option will insert it rightside-up.

Click the **Options** button to change properties of the bearing text.

Angle & Distance between Points

Menu: Dimension / Info / Angle & Distance between Points

Toolbox Icon:



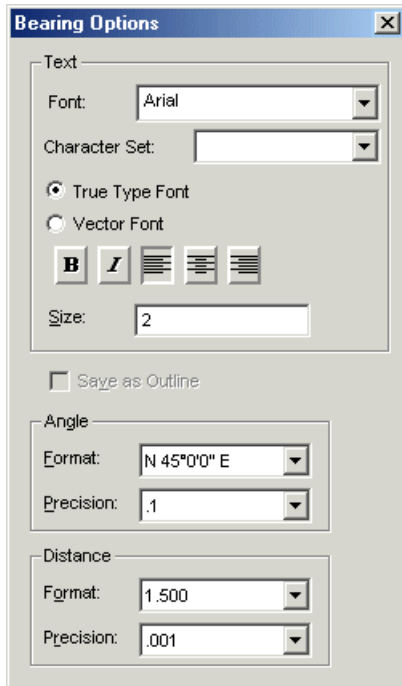
Point 1: First point of the measurement

Point 2: Second point of the measurement

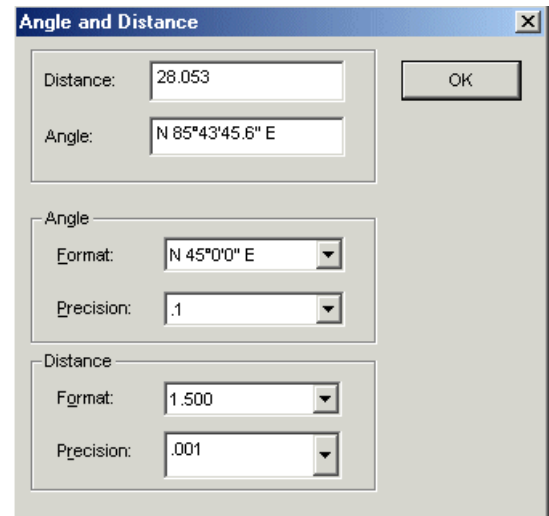
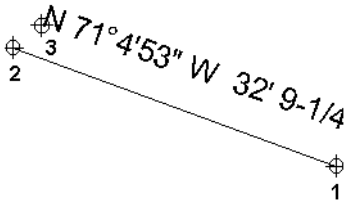
Calculates the angle and distance between two points.

Using the Command

Choose the **Angle & Distance between Points** command. Set two points for the angle and distance to be measured. The results appear in a window:



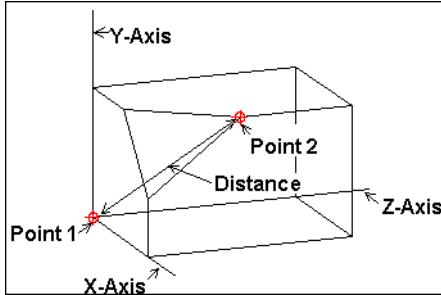
Position the rubber-band box where you want the text and set a location point.



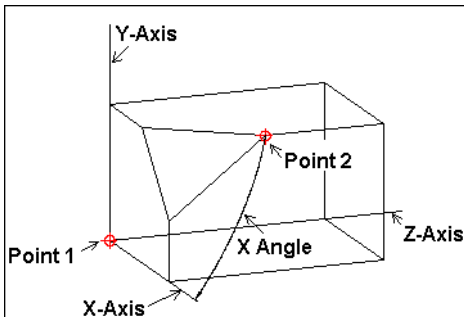
XYZ Coordinate System

When the XYZ Coordinate System is used, this command uses the location of the first point as the origin for the angular measurements.

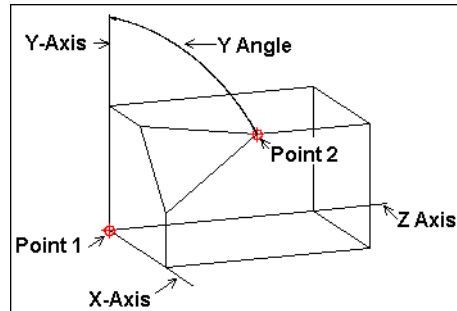
Distance: The linear distance between the two points.



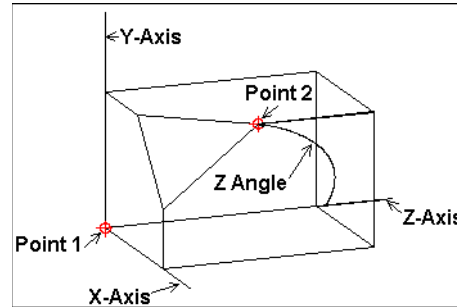
X Angle: The angle from the X-Axis to the second point.



Y Angle: The angle from the Y-Axis to the second point.

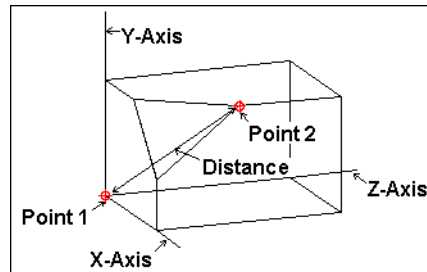


Z Angle: The angle from the Z-Axis to the second point.



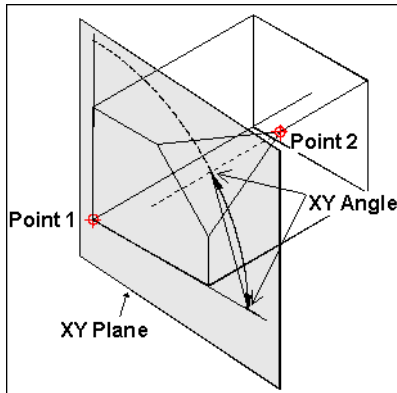
Sphere Coordinate System

Distance: The linear distance between the two points.

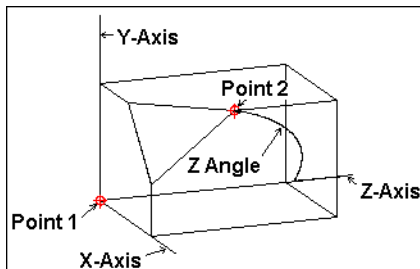


XY Angle: The angle from Point 1 to Point 2 relative to the XY Plane. In other words, if the XY plane were a projector screen, and the front view

of the drawing was projected on to that screen, the angle would be measured from Point 1 to Point 2 on the screen.



Z Angle: The angle from the Z-Axis to the second point.



Angle between Two Lines

Menu: Dimension / Info / Angle between Two Lines

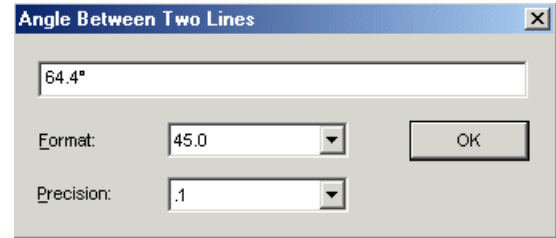
Point 1: The first line of the angle to be measured

Point 2: The second line of the angle to be measured

Measures the angle between two non-parallel lines and displays the result.

Using the Command

Choose the **Angle between Two Lines** command. Set a point on the first line of the angle to be measured. Set a point on the second line. The result is displayed in the window.



Format: Determine how you want the angle displayed.

Precision: Controls how many digits appear after the decimal point.

Surface Area ^{3D}only

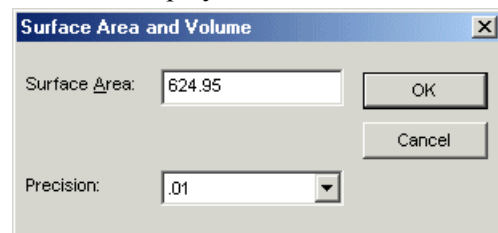
Menu: Dimension / Info / Surface Area

Point 1: Object for which surface area is to be calculated

Calculates the surface area of an object.

Using the Command

Choose the **Surface Area** command and set a point on the object for which you want DesignCAD to calculate the surface area. The calculation is displayed in the window.



Precision: Controls how many digits appear after the decimal point.

NOTE: For a 3D solid, the **Volume** command calculates both surface area and volume.

Volume ^{3D}_{only}

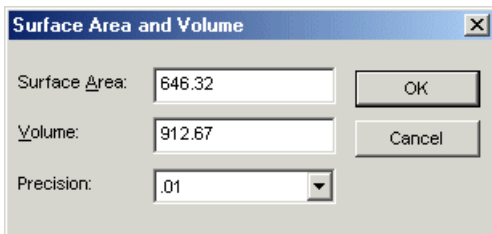
Menu: Dimension / Info / Volume

Point 1: Object for which to calculate volume

Calculates the volume and surface area of a Solid object.

Using the Command

Choose the **Volume** command, and then set a point on the Solid. The calculation is displayed in the window.



Precision: Controls how many digits appear after the decimal point.

7 Groups, Blocks, and Symbols

Groups

A group is a set of entities that act as one entity.

Group Define

Menu: Tools / Group Define

Puts all of the currently selected drawing entities into one group. The next time you select any part of the group, the entire group is selected. You can have many different groups defined in a drawing. **Group Define** makes it easy to keep related items together for copying, moving, scaling, and other operations.

Using the Command

Select all the objects you want to include in a group, and choose the **Group Define** command. DesignCAD treats the objects as a group and continues that way until the objects are ungrouped with the **Group Explode** command.

Group Explode

Menu: Tools / Group Explode

Breaks up the currently selected group so that its members become single entities and can again be selected individually. Drawing entities are not affected except that they lose their association with the group.

Using the Command

With the group selected, choose the **Group Explode** command.

Undo does not cancel the effect of **Group Explode**. To restore group status, select all of the members again and use the **Group Define** command.

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Blocks

A block is a set of entities that act as one entity. They are different than groups in that each instance of a group refers to the same embedded entity; adding multiple blocks does not greatly increase file size. Blocks are internal to the drawing, whereas symbols are linked externally.

symbols are linked to the file externally. Groups are embedded in the drawing, and if you use multiple copies of a group, each instance increases the file size.

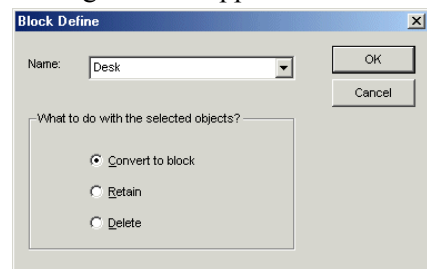
Block Define

Menu: Tools / Block Define

Puts all of the currently selected drawing entities into one group. The next time you select any part of the block, the entire block is selected. Blocks behave in much the same way that symbols do. The main advantage of blocks is that you don't have to save an external file.

Using the Command

Select all of the objects you want to include in a block and choose the **Block Define** command. The following window appears.



Enter a name for the block in the **Name** box.

Select **Covert to Block** to define a block and convert the selected entities to a block. Select **Retain** to define a block but leave the selected entities as the original entities. Select **Delete** to define a block and delete the selected entities from the drawing.

Additional copies of the block can be added to the drawing using the **Block Insert** command.

NOTE: Blocks can also be copied and pasted; the difference is that pasted blocks will not prompt you for new values for attribute definitions. Instead they will use the values from the block that was copied to the Clipboard.

An important difference between blocks and symbols is that the visibility of a block's entities depends on the visibility of each part's host layer; therefore, a block can be drawn on different layers so only part of it is visible. The visibility of an entire symbol depends on the visibility status of the layer where the symbol is inserted.

Block Insert

Menu: Tools / Block Insert

Point 1: Block placement

Point 2: Orientation and scaling (optional)

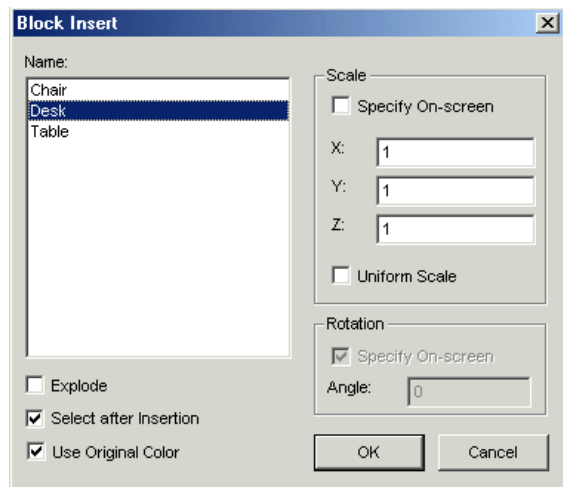
Point 3: Additional orientation (optional, only used for a 3D block)

Inserts a block into the drawing. Blocks are created using the **Block Define** command.

Blocks that are copied and pasted or inserted into the drawing with **Block Insert** are defined as single entities. That is, the entire block is selected, moved, and erased as a single entity.

Using the Command

Choose the **Block Insert** command and the following window appears:



Select the name of the block you want to insert in the drawing.

Explode: The components that make up the block are inserted into the drawing as individual entities. They may be selected and edited separately, but they are no longer recognized as a block.

Select After Insertion: The block is automatically selected when it is added to the drawing.

Use Original Color: The color of the inserted block will be the same as the original. If this option is unchecked, the current drawing color is used.

Scale

- **Specify On-screen:** Set a second point that controls the scale of the block when the block is inserted in the drawing. If this

option is disabled, the scale of the block is controlled by the values entered in the X, Y, and Z boxes.

- **Uniform Scale:** Only the value in the X box is enabled, and the values in the Y and Z boxes will automatically be updated to match the value entered in the X box. This maintains the aspect ratio of the block.

Rotation: If **Specify On-screen** is enabled, you can set a second point that controls the rotation angle of the block when the block is inserted in the drawing. If this option is disabled, the value entered in the **Angle** box is used.

OK: Click this button to accept the current options and start setting points to insert the block. A rubber-band box appears in the drawing. This shows the area of the block. Depending on the options that have been selected, set one to three points to establish the location, size, and orientation of the block (the third point is only available in 3D Mode). To accept the block's size and orientation, set a point to establish the location, and then press Enter.

Manipulating a Block

A block must be exploded in order to change its characteristics such as color, line style, etc. There are two ways to explode a block: the **Explode** option in the **Block Insert** window (before the block is loaded); or use **Edit / Selection Edit / Explode**.

An important difference between blocks and symbols is that the visibility of a block's entities depends on the visibility of each part's host layer; therefore, a block can be drawn on different

layers so only part of it is visible. The visibility of an entire symbol depends on the visibility status of the layer where the symbol is inserted.

Purge Unused Blocks

Menu: Tools / Purge Unused Blocks

Removes unused block information from the drawing file.

Symbols

A symbol is an externally linked file that is inserted in the drawing. DesignCAD provides you with a library of 2D and 3D symbols, and you can create your own symbols by saving entities or entire drawings as a symbol.

Drawing Handles

When you save a drawing with handles, you can then load that drawing as a symbol. The handles are used to place and orient (and optionally scale) the symbol in its host drawing.

Set Drawing Handles

Menu: Point / Set Drawing Handles

Point 1: First handle

Point 2: Second handle (optional)

Point 3: Third handle (optional)

Sets handles for the drawing. Use these handles to help you manipulate a drawing when you merge it with a new or existing drawing using the **Load Symbol** command.

Using the Command

Select the **Set Drawing Handles** command. Set one to three points in the drawing for the handles. If you set fewer than three points, press Enter to end the command.

Next, choose **File / Save As** and rename the file or **File / Save** and save it with the same name. The drawing is saved with the handles in place. The next time you merge the drawing into another one using the **Load Symbol** command, you can position the drawing using the handles you have set.

Example: Attach a drawing of a wing to a fuselage in another drawing.

Open the drawing of the wing. Choose the **Set Drawing Handles** command. Set two points at the base of the wing, where it will connect to the fuselage. Press Enter.

Next, use **Save As** to rename the file as SETWING.DCD and click OK. When you merge the wing into the drawing of the fuselage using the **Load Symbol** command, you can attach the wing precisely using the handles you have set and the Gravity command.

Remove Drawing Handles

Menu: Point / Remove Drawing Handles

Makes it easy to remove the existing drawing handles from the drawing. Once the drawing handles have been removed, the default handles at lower left and lower right (and top right in 3D Mode) will be used.

Show Drawing Handles

Menu: Point / Show Drawing Handles

Makes it easy to find the drawing handles for the current drawing.

Show Symbol / Block Handles

Menu: Point / Show Symbol/Block Handles

Makes it easy to see the handle location for any symbols or blocks that are present in the drawing. The primary handle of each symbol will be shown as a small red dot, the secondary handle (if any) as a green dot, and the third handle as a blue dot. Note that blocks only have one handle, shown as a red dot.

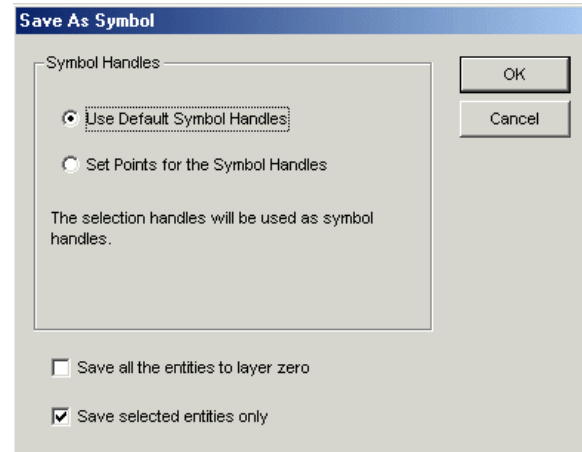
Save As Symbol

Menu: File / Save as Symbol

Allows you to quickly and easily save a file, or the selected part of a file, as a symbol with a separate filename.

Using the Command

If you want to save only selected entities, select them and choose the **Save as Symbol** command. The following window appears:



Use Default Symbol Handles: If saving only the selected entities to the symbol, the selection handles will be used as the symbol handles. If the entire file is being saved as a symbol, the drawing

handles will be used if there are any, or the lower left and right corners of the drawing if no handles have been set.

Set Points for the Symbol Handles: You will be prompted to set up to two points as symbol handles (up to three points in 3D mode)

Save all the entities to layer zero: All entities will be moved to Layer 0 in the symbol. This affects the way the symbol is imported into another drawing if it is exploded.

Save selected entities only: Only the selected entities will be saved to the symbol file, and the selection handles will be used as Default Symbol Handles. Otherwise, the entire file is saved, and the drawing's handles will be used as Default Symbol Handles.

If you want to save the entire drawing as a symbol, simply choose the **Save as Symbol** command. DesignCAD will ask you to specify the name and DesignCAD version of the symbol file.

Load Symbol

Menu: File / Load Symbol

Shortcut Key: Ctrl+F9

Point 1: First handle for symbol placement

Point 2: Second symbol handle (optional)

Point 3: Third handle (optional, only used for a 3D drawing or symbol)

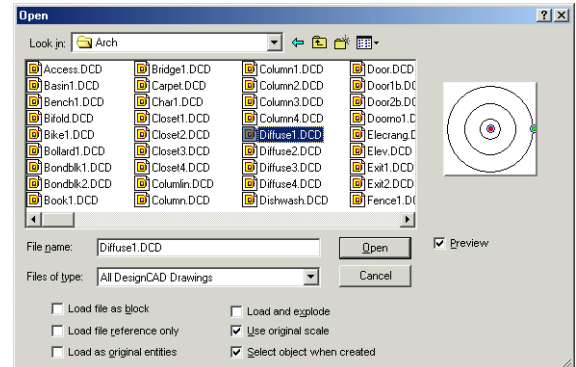
Loads a drawing symbol or merges an existing drawing with the current drawing. Symbols inserted into the drawing with the **Load Symbol** command are defined as single entities, whether

they are saved as part of the drawing or linked to the drawing. That is, the entire symbol is selected, moved, and erased as a single entity.

An important difference between blocks and symbols is that the visibility of a block's entities depends on the visibility of each part's host layer; therefore, a block can be drawn on different layers so only part of it is visible. The visibility of an entire symbol depends on the visibility status of the layer where the symbol is inserted.

Using the Command

Choose the **Load Symbol** command and the **Open** window appears. From the file list, select the name of the symbol you want to load. Select the desired options. Then click the **Open** button.



A rubber-band box appears in the drawing. This shows the area of the symbol. Set one to two points to establish the location, size, and orientation of the symbol. To accept the symbol's size and orientation, set a point to establish the location, and then press Enter.

Load file as block: Loads a block from an external file, and adds the block definition to the list of defined blocks.

Load file reference only: The contents of the symbol is not added to the drawing's data, but must always be loaded from the external file.

Load as original entities: Think of this as "Paste From File" or "Merge". The entities from the selected drawing are added to the current drawing (possibly scaled and repositioned). Entities in Layer 0 of the selected drawing may be moved to the current layer, depending on the "Merge Layer 0 items onto the current layer" setting in Options -> Preferences. Entities from any other layers are placed in their original layers in the current drawing.

Load and explode: Loads the drawing as a symbol, then explodes it. The results of this option are affected by the **Symbol Explode Settings** in the **Preferences** (see Preferences).

Use original scale: Controls whether the loaded entities can be dynamically rescaled as they are being loaded.

Select object when created: If checked, the loaded symbol/block/entities are selected after being loaded into the current drawing.

Manipulating a Symbol

A symbol must be exploded in order to change its characteristics such as color, line style, etc. There are two ways to explode a symbol: the **Load and Explode** option in the **Load Symbol** window (before the symbol is loaded); or use **Edit / Selection Edit / Convert** (after the symbol is loaded).

When a referenced symbol is exploded with the **Explode** command, the symbol file is not read from an external file when the drawing is loaded.

A copy of the drawing information for the symbol is placed into the drawing instead of an insertion entity.

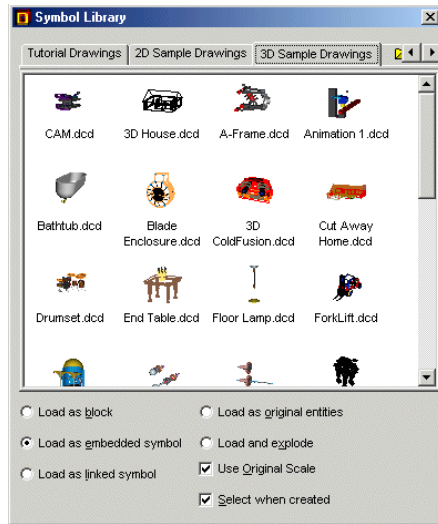
When a non-referenced symbol is exploded with the **Explode** option or command, there is no change to the drawing unless it contains multiple copies of the same symbol. If several, non-referenced copies of a symbol exist in the same drawing, they share drawing information even though that drawing information is contained in the drawing file that contains the symbols (not an external drawing file). In order to manipulate one of the copies of the symbol, it must be exploded so that it will have its own set of drawing information.

If the **Group Object when Created** option in the **General** options (see General Options) is enabled, the symbol will still be recognized as a group once it has been exploded. This way it is not necessary to select many small items just to change the color, line style, etc. of the former symbol. The **Group Explode** command may then be used if individual items need to be manipulated.

Symbol Library

Menu: File / Symbol Library

Works like the **Load Symbol** command, but it brings up a window that shows small preview images, or "thumbnails," of the symbols that are available in the different symbols libraries that come with DesignCAD.



Use the scroll bar on the right side of the window to scroll through the various symbols. Click the tabs or use the scroll bar at the top of the window (just under the title bar) to switch libraries.

DesignCAD allows you to view the DesignCAD files contained in a directory and all of the DesignCAD files in one layer of subdirectories of that directory. To view a different set of symbols click on the tab with a folder symbol on it. The Path window opens. Select a new folder and click OK.

Load as Block: Loads the symbol so it will be recognized as a block. Once the symbol has been loaded as a block, additional copies can be added to the drawing using the **Block Insert** command. This option makes it easy to use a block/attribute definition combination that you use frequently in other drawings. Just create a drawing that contains only the block and its attribute definition. Save the drawing. Open the file in which you want to use the block. Using the **Load**

Symbol command, load the block and attribute definition drawing you just saved, making sure that the **Load as Block** option is enabled.

The block will still be recognized as a block. When additional copies are placed using the Block Insert command, the attribute definition will function just like it did in the original drawing.

Load as Embedded Symbol, Load as Linked Symbol: Determines whether the drawing being added or merged with the current drawing will be saved as a part of the drawing, or just linked by a reference. If the symbol is saved as part of the current drawing, the file size will be larger, and the drawing will not be updated when the symbol file is modified. The advantage is that the drawing will not be affected if the symbol file is deleted, or the drawing is opened on a computer that doesn't have a copy of the symbol file.

If the box is checked, the program adds an "insertion entity" to the drawing file. The symbol file is read every time the drawing is loaded. The symbol file must be present and in its original location. If the symbol file is modified, the change will be reflected in all the drawings using that symbol.

Load as Original Entities: Entities are added and embedded as individual entities, not grouped as one symbol.

Load and Explode: When a referenced symbol is exploded, the symbol file is not read from disk when the drawing is loaded. A copy of the symbol is placed into the drawing instead of an insertion entity, just like a symbol that was loaded without using the Load File Reference Only option.

Select When Created: The symbol is automatically selected when it is added to the drawing.

Placing a Symbol in Your Drawing

When you find the symbol that you want to use, there are two different methods for bringing it into the drawing:

1. Move the cursor over the desired symbol, press and hold the left mouse button, drag the cursor to the desired location in your drawing, position the cursor and click the mouse button to drop the symbol into the drawing;

or,

2. Move the cursor over the desired symbol, double-click the left mouse button, move the cursor to the desired location in your drawing, and click the left mouse button again to drop the symbol into the drawing.

Door and Window Symbols

DesignCAD provides some standard architectural symbols for doors and windows.

Door

Menu: Draw / Door

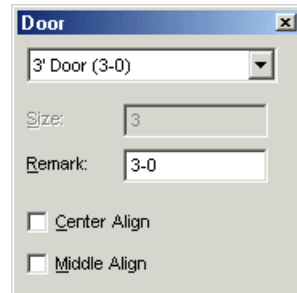
Point 1: Location of the center of the door or one end of the door

Point 2: Orientation of the door

Inserts a standard architectural symbol for a door.

Using the Command

Choose the **Door** command, and the following window appears.



Select the desired door size and type from the list box. If you choose **Custom Door** or **Custom Sliding Door**, specify the size in the **Size** box. By default, the door size appears in the **Remark** box unless **Custom** is selected. However, any information can be entered in the **Remark** box to be inserted into the drawing with the door. Check the **Center Align** box to align the door by the center or uncheck it to align the door by one end. Check the **Middle Align** box to align the door in the middle of a double line.

Set the first point for the door. A rubber-band door appears, showing how the door would look with the second point set at the current cursor location. When the door is positioned to your liking, set the second point. The Line or Double Line into which the door is placed will automatically be cut and capped.

Window

Menu: Draw / Window

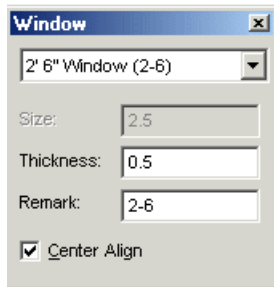
Point 1: Location of the center of the window or one end of the window

Point 2: Placement of dimension text and orientation of the window, if necessary

Inserts a standard architectural symbol for a window.

Using the Command

Choose the **Window** command, and the following window appears:



Select the desired window size, or **Custom**, from the list box. If you choose the **Custom** option, enter the size in the **Size** box. By default, the window size appears in the **Remark** box unless **Custom** is selected. However, any information can be entered in the **Remark** box to be inserted into the drawing with the window. The **Thickness** is 0.50 by default. Change the Thickness if desired. Check the **Center Align** box to align the window by the center or uncheck it to align the window by one end.

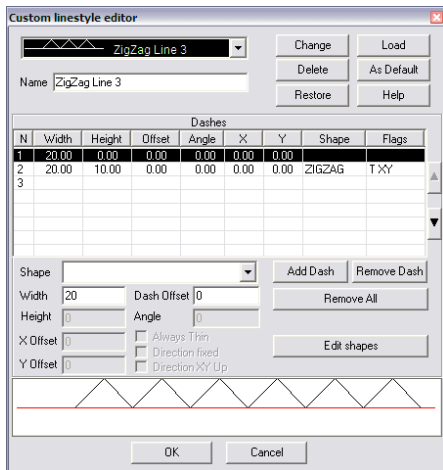
Set the first point for the window. A rubber-band window appears. A rubber-band window appears, showing how the window would look with the second point set at the current cursor location. When the window is positioned to your liking, set the second point. The Line or Double Line into which the window is placed will automatically be cut and capped.

Appendix A

Custom Hatch Patterns and Line Styles

Custom Line Styles

Menu: Option / Custom LineStyle



Enables you to edit an existing line style or define a new one.

For each type of line style component you can set various parameters. For Shape you can choose the actual shape and its size and spacing. For Text, you can choose the font and angle. The Preview window helps you see how the line style will look; use it as a guide when adjusting the parameters.

Hacking Custom Line Styles

Some of the line styles used by DesignCAD are found in the DCLSTYLE.SYS file located in the DesignCAD directory. You can create your own custom line styles for use in DesignCAD by adding them to this file. The file DCLSTYLE.SYS is an ASCII file that can be edited with a text editor or a word processor in ASCII mode. Be sure you have a second copy of the file DCLSTYLE.SYS before you modify it.

The file DCLSTYLE.SYS consists of a series of line styles. Each line style is in the following format:

```
Line Style Name (no more than 30
characters long); Text Comment
(optional)
```

```
Number of Lines and Shapes
```

```
Line or shape definition (there
will be a separate line of values
for each line or shape)
```

The Number of Lines and Shapes indicates the number of separate line segments and shapes that make up the line style.

Line Definition values:

A. PD, PU

1) PD = length of pen down (line segment)

2) PU = length of pen up (space after line segment)

Shape Definition values:

A. [S(width, height) R(z angle) P(x,y) ; %%?%|Shape Name]

1) S = size of shape; original size is used if size is not specified; if only one value is specified, width and height are the same. (optional)

2) R = rotation; original angle is used if rotation is not specified. (optional)

3) P = position of shape relative to the end of the previous Line or Shape definition; if only one value is specified, it is assigned to x. (optional)

NOTE: *When these commands are present, **Scale** should be used first, then **Rotation**, and finally **Position**.*

4) %^ = shape orientation won't be affected by the order of points of the line. (optional)

5) %- = ignores the bold flag of the line. (optional)

6) %| = shape does not rotate with the line segment.

NOTE: *If more than one of %^, %-, and %| are used, there should not be a space between them.*

7) Shape Name = shape to be used in line style; must be defined in DCSHAPE.SYS.

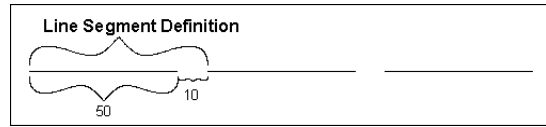
The following is an example of a simple line style that has only one line segment definition.

```
Long Dash
```

```
1
```

```
50, 10
```

This simple line style will look like this:



The next example is a little more complicated. Notice that the only thing that changed in the first three lines between this example and the last one are the Line Style Name and the Number of Lines and Shapes has changed from "1" to "2." The same long dash (50) and small space (10) are still in the line style.

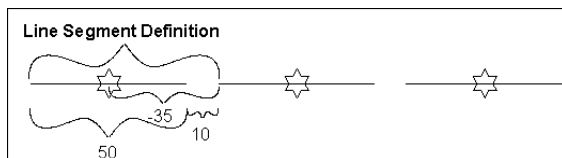
```
Long Dash and Star
```

```
2
```

```
50, 10
```

```
[P(-35,0) ; Star], 0
```

The additional line of the "Long Dash and Star" line style inserts a "Star" shape (defined in the DCSHAPE.SYS file). All of the information for the "Star" shape is entered as the fourth line of the "Long Dash and Star" line style [P(-35,0) ; Star], 0. The center of the star is placed so that its center is centered on the line segment defined in the third line of the "Long Dash and Star" line style (-35 = 50/2+10, the value 50 = length of the original dash; the value 10 = amount of space that follows the dash in the line segment definition). This line style will look like this:



Now a more complicated line style from scratch.

```
DCAD
```



```
2
2, 1
[; DCAD], 20
```

Doesn't look that complicated does it? It's really not. It just requires that you add a shape to the DCSHAPE.SYS file.

NOTE: *Be sure that the last line of both the DCLSTYLE.SYS and DCSHAPE.SYS is an empty carriage return.*

Custom Line Style Shapes

The shapes that can be used by DesignCAD line styles are found in the DCSHAPE.SYS file located in the DesignCAD directory. You can create your own custom shapes for use in DesignCAD line styles by adding them to this file. The file DCSHAPE.SYS is an ASCII file that can be edited with a text editor or a word processor in ASCII mode. Be sure you have a second copy of the file DCSHAPE.SYS before you modify it.

Each shape definition is in the following format:

```
Shape Name (no more than 30
characters long); Text Comment
(optional)
Command and its number of points
Points for command (points appear
on separate lines and consist of x
and y coordinates; x and y
coordinates may be separated by a
space or a comma)
Additional commands and points
* (indicates the end of the shape)
```

Commands allowed in the DCSHAPE.SYS file are:

1) A = Three-point arc with center, starting, and ending points

NOTE: *Identical starting and ending points make a circle.*

2) C = Spline curve

3) V = Polyline

The best way to make a new shape is to record a macro and use the **Arc (Center, Begin, end)**, **Curve**, and **Line** commands to draw the shape. Once the drawing of the shape has been recorded as a macro, you can open the macro file, delete the information that is not needed for the DCSHAPE.SYS file, and change the command names to the DCSHAPE.SYS format.

Here is a portion of the recorded macro and the equivalent portion of what was added to the DCSHAPE.SYS file to create the DCAD shape (in this comparison, blank lines in the DCAD shape definition are used only to keep the code comparison parallel; there should not be blank lines in the actual DCSHAPE.SYS file):

```
>SetStartPoint DCAD
{
}
>Line v 2
{
  <PointXYZ 0.0000,0.0000,0.0000
0.0000,0.0000
  <PointXYZ 0.0000,4.0000,0.0000
0.0000,4.0000
}
```

```
>Curve          c 3
{
  <PointXYZ 0.0000,4.0000,0.0000
0.0000,4.0000
  <PointXYZ 1.7500,3.8000,0.0000
1.7500,3.8000
  <PointXYZ 2.7500,3.0000,0.0000
2.7500,3.0000
  <Type 0
}
```

```
>Line          v 2
{
  <PointXYZ 2.7500,3.0000,0.0000
2.7500,3.0000
  <PointXYZ 2.7500,1.0000,0.0000
2.7500,1.0000
}
```

```
>Curve          c 3
{
  <PointXYZ 0.0000,0.0000,0.0000
0.0000,0.0000
  <PointXYZ 1.7500,0.2000,0.0000
1.7500,0.2000
  <PointXYZ 2.7500,1.0000,0.0000
2.7500,1.0000
  <Type 0
}
```

... ..

Don't forget to end the shape definition with an asterisk (*).

Custom Hatch Patterns

Some of the hatch patterns used by the **Hatch**, **Hatch Line**, and **Hatch Fill** commands are found in the file DCHATC.C.SYS. You can use your own hatch patterns with DesignCAD by adding them to this file. The file DCHATC.C.SYS is an ASCII file that can be edited with a text editor or a word processor in ASCII mode. Be sure you have a second copy of the file DCHATC.C.SYS before you modify it.

This file does not contain definitions for the "non-scaleable" hatch patterns such as WIN_SOLID. These are defined by MS Windows.

The file DCHATC.C.SYS consists of a series of hatch patterns. Each hatch pattern is in the following format:

```
Hatch Pattern Name
Number of Line Definitions Pattern
Scale for Preview
A B C D E F1 F2 F3 F4 F5 F6
```

(there will be a line of these values for each line definition)

The Number of Line Definitions indicates the number of separate line segments that make up the pattern. The Pattern Scale for Preview is the pattern scale that particular hatch pattern uses in the Preview box.

Line Definitions values:

- A. Angle (0=horizontal, 90=vertical)
- B. X offset of first occurrence (relative to an arbitrary starting point)
- C. Y offset of first occurrence (relative to an arbitrary starting point)

D. X change from first to second occurrence.
(relative to B,C)

E. Y change from first to second occurrence.
(relative to B,C)

F. six numbers defining the line pattern:

- 1) length on
- 2) length off
- 3) length on
- 4) length off
- 5) length on
- 6) length off

"X" and "Y" are relative to the angle of the line. This means that if the angle is 90, then X is actually the relative vertical displacement, and Y is the relative horizontal displacement.

For example, the hatch pattern definition for the BRASS pattern is:

```
BRASS
2 45
0 0 0 0 20 40 0 0 0 0 0
0 0 10 0 20 10 5 0 0 0 0
```

There are two lines in the pattern. The first line definition is oriented at an angle of zero, so it is a horizontal line. It is a solid line, since there is only one non-zero value in the last six numbers. This first line starts at 0, 0 (an arbitrary position), and it will repeat 20 "units" above this line. This line ends 40 "units" from this starting point.

The next line is also a horizontal line, but it starts 10 units above the first line. It is a dashed line, repeating a pattern of 10 "on" and 5 "off."

A more complicated example is the BRICK2 pattern:

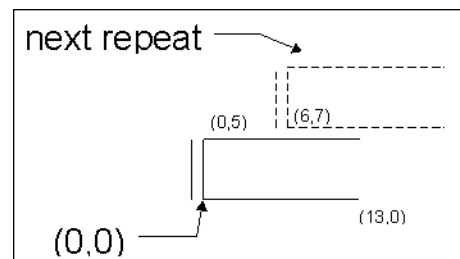
```
BRICK2
4 20
0 0 0 7 6 13 1 0 0 0 0
0 0 5 7 6 13 1 0 0 0 0
90 0 0 6 7 5 7 0 0 0 0
90 0 1 6 7 5 7 0 0 0 0
```

This pattern consists of 4 lines, 2 horizontal, and 2 vertical.

The first line is a horizontal, dashed line, repeating 13 on, and 1 off. It starts at 0,0, and will repeat at 7 units over and 6 units up. The second line is the same as the first, except that it starts 5 units above the first line.

The third line is a vertical line, starting at 0,0, repeating with 5 on and 7 off. Note that since it is vertical, the line actually repeats at 7 over and 6 up, not 6 over and 7 up. Because the fourth line is at a 90 degree angle, it starts at 1 unit to the left of the starting point.

If viewed by themselves, the four line segments will look something like this:



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