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COBALT XENON ARGON

DesignerElements

3D Modeling User Guide

Cobalt, Xenon & Argon™

January 2003

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Table of Content

User Guide Documentation

This *User Guide* section of the manual is written for both Windows 98/NT/2000/Me/Xp and Power Macintosh platforms. Before using this *User Guide* section, however, you will need to install your Designer Elements 3D modeling program (Cobalt™, Xenon™ and Argon™). Instructions and System Requirements are contained in the *Getting Started* manual. This manual is divided into sections which group chapters according to the topic. Each chapter provides information about tools, commands and other features.

Menus and Submenus

As is standard for all programs, Ashlar's Designer Elements 3D modeling programs provide menus and submenus for choosing commands and performing other operations.

Choosing Commands

As you proceed through the manual, you will be directed to choose commands contained in submenus of other menus, like the pull down menu. For example, you might be asked to select *Lock* in the Group submenu of the Layout menu. That appears in this manual as ***Layout>Group>Lock***.

Margin Notes

Your Designer Elements 3D program includes margin notes that provide you with information that may help you use this program. There are three types of margin notes: Tip, Tech Note and Referral. These notes are given special treatment so that you can instantly recognize their significance and locate them for future reference.

Tip

A tip provides instructions for getting the most out of this Designer Elements program. Tips may show you how to speed up an operation or how to perform some timesaving drawing technique.

Tech Note

A technical note provides additional technical information that may help when using a tool.

Referral

A referral directs you to related information contained somewhere else in the manual for the particular topic being addressed.

Style Conventions

This manual uses various style conventions which highlight certain terms or phrases. The list below includes and explanation and an example in parentheses. The conventions are as follows:

Bold	Tool palette names (**Light** palette); Tool names (**Single Line** tool); Keyboard-entered text; Definition terms (as shown in these style conventions)
Italic	Terms used for the first time in a chapter; (*Iso Lines*); Drafting Assistant notations (*midpoint*); tool and dialog box options (*Angle* box); book references (*User Guide*); Message Line directions (*Single Line: Pick the beginning point.*); margin note headings (*Tip*); menu commands (*Zoom Previous*); filenames (*prefs.ini*); stand alone extensions (*.dwg*); directory names; drawing names
Bold and Italic	Command series (***Layout>Group>Lock***)
ALL CAPITALS	Key names on the keyboard (ENTER, RETURN)
Title Capitalization	Dialog box names (Edit Objects); menu names (Pen menu); special Vellum phrases (the Drafting Assistant)

all lower case	File names (prefs.ini); stand-alone file extensions (.dwg)

Terminology

For those of you new to surface and solids modeling, there are many terms or phrases that might be unfamiliar to you or used differently from your experience. These terms are defined the first time they are used in the manual as well as in the Glossary.

Some important terms include:

Curves	Refer to all lines, arcs, ellipses, conics, circles, splines and polygons created using one of the wireframe tools.
Instance	Refers to an object that's moved to or placed in a different location after performing an operation on it. If you create a solid, add a blend and move it, the solid is now an instance. The original is still located in the previous position although it is not displayed. If you create a master symbol, an instance occurs when you place the symbol in your drawing. An instance is associative to the original geometry. Any change made to the original is reflected in the instance (Cobalt™ and Xenon™ only). If you copy and pasted the object, the associativity is broken.
Parent/child	Refers to objects that are related to each other in such a way that changing one (the parent) affects the other (the child). This is an important feature in Cobalt™ and Xenon™, **Argon™ does not have this functionality**. This associativity works much the same way Vellum 3D or Vellum Solids worked with dimensioning objects and editing. When you change an object in Vellum, the dimension changes. Cobalt™ and Xenon™ have taken this idea to a higher level involving geometry. After creating a curve, extruding and then filleting it, you can go back to the first curve and edit it and the entire object adjusts. Cobalt™ and Xenon™

Cobalt and Xenon Only

have the intelligence to remember the relationship between the parent curve and all other operations performed on the resulting object.

Using the Mouse

The mouse is your communication device; you use it to tell the computer what you want to do. Use the mouse to indicate locations, choose commands, select tools and construct objects.

This manual uses the following terms for mouse activities:

Pointer — An arrow or any other graphic symbol that allows selection or creation of an object. Move the pointer to point to a command or an object on the screen. Depending on its location, the pointer is an arrow or may look like the current tool.

Arrow Pointer *Selection Arrow* *Center-Point Circle*

To move the pointer, move the mouse on the mouse pad.

Point — Move the mouse until the pointer is over the item you want.

Press — Press and hold down the mouse button.

Click — Quickly press and release the mouse button once.

Double-click — Click the mouse button twice, quickly in succession.

Drag — Press and hold down the mouse button, move the mouse, then release the mouse button.

Chapter Breakdown

Referral:

Specific page information on a particular tool or command can be found in the index.

The chapters are grouped into nine sections dealing with a specific area.

Sections

1. Overview — Contains chapters that introduce you to some basic features of your Designer Elements 3D modeling

	program, including the program Window, the Drafting Assistant and information on selecting objects.
2. Setting the Environment	Contains chapters on pen and preference settings and drawing techniques.
3. Wireframe Modeling	This section contains chapters on using such curve tools as lines, arcs, circles, splines, polygons, etc.
4.Surface Modeling	Contains an introduction to surface modeling and information about surface modeling tools such as 3 Point Mesh, Mesh Curve, Infinite Plane, Skin, Net, Tube, Tangent, etc.
5. Solids Modeling	Contains an introduction to solids modeling and information on such solids modeling tools as the Sphere Primitive, Block Primitive, Cylinder Primitive, Lathe Solid, Pipe Solid, Blend Solid, Hole, Boss, Protruded Feature, etc.
6. Editing	Contains tools and commands for editing curves, surfaces and solids, including such tools as 2 Entity Fillet, Simple Trim, Connect Curve, Plane Surface Intersection, Boolean Surface, Subtract Solid, Shell Solid, Taper Solid, Deform Face, etc.
7. Adding Details	This section includes information on text, dimensions, patterns and fills, crosshatching and symbols.
8. Drawing Display	Contains information on viewing your geometry, layers, planes and rendering.
9. Documents	Contains information on file management, importing, exporting, page setup and producing production drawings.

Appendices

Operators	Describes all operators accepted by the data fields.

Special Characters	Lists all special characters not directly available from the keyboard and symbols that you can use in this Designer Elements program.
DXF/DWG Translator	Includes notes to help the user in successful .dxf/ .dwg translation.
IGES Translator	Includes notes to help the user in successful *.igs* translation.
Short Cuts	Contains a list of the default Short Cuts
Spline Text Files	Provides instructions for creating spline text files to import.
Shader Attribute Definitions	Contains the definitions of shader attributes used when using the advanced shader rendering capabilities.

Other

Glossary	Defines terminology used in CAD drafting and in this Designer Elements program.
Index	List this Designer Elements program tools, features and actions and their page location.
Tips	Contains additional tips for using this Designer Elements program.

Graphics

Most of the graphics in the manuals apply to both platforms. In those instances that require a platform and software reference, a Windows graphic is used. When necessary, both Windows and Macintosh graphics are included.

On-Line Help

Designer Elements program Help (Windows only) provides a complete description of the program's many features, commands, and tools. The Help index is organized by menus and tools. The Macintosh Help uses the UserGuide PDF file that was installed when you installed the application.

First Look

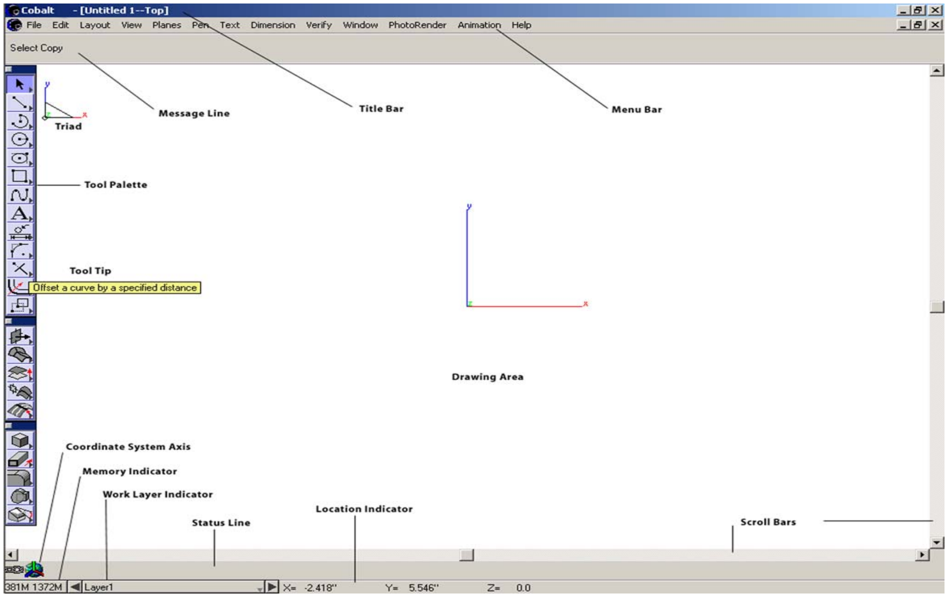
This chapter describes the “first look” components of your Designer Elements program. A brief overview of useful features may be all you need to know if you are familiar with CAD software. The following topics are covered:

- Parts of the your Designer Elements program Window
- Menu Bar, including the dialog boxes

For more information about standard elements such as menus, scroll bars, File menu commands, and dialog boxes, refer to the Windows or Macintosh User's Guide that came with your computer.

Parts of the Designer Elements Window

When you start your Designer Elements program, the following window appears.



Title Bar	Includes the title of the active document and buttons for controlling the window including boxes for zooming and closing the program.
Menu Bar	Contains the Designer Elements program menus of commands and settings. You can make choices from the menus with the mouse or by using special key combinations.
Tool Palette	Contains drawing and editing tool icons you will use for constructing, editing and annotating geometry. This is the most basic tool palette. The **Surface** tool palette and **Solids** tool palette are not automatically displayed on start up, unless specified as such through Preferences. See Chapter 6.
Tool Tip	As you hold your cursor over one of the tools, a brief description of the tool is displayed. The Tool Tips function can be activated or deactivated by choosing ***File>Preferences>General***. If you assign a short cut to a tool, the short cut displays with the tip.
Pointer	Shows the active position on the screen. For wireframe tools, when the pointer is in the drawing area, its shape represents the current tool.
Axis	Displays the current view orientation of the x, y and z axis in the center of your screen.
Triad	Displays the current view orientation of the x, y and z axis with a representation of the current work plane.
Location Indicator	Shows the x, y and z coordinates of the pointer location at the top of your drawing area.
Message Line	Displays the name of the current tool and step-by-step instructions for using the tool. For some tools, the Message Line includes an additional subpalette.
Drawing Area	Consists of multiple layers where you construct and annotate geometry.

Status Line Shows the coordinate location and other geometric characteristics of the current construction based on the World coordinate system.

Coordinate System Axis Shows the current coordinate system. You can choose between the default global or world coordinate system and a user-defined coordinate system.

Scroll Bars Allow you to move around a drawing so you can see different sections of it through the Designer Elements program window. The scroll buttons allow you to move one line at a time.

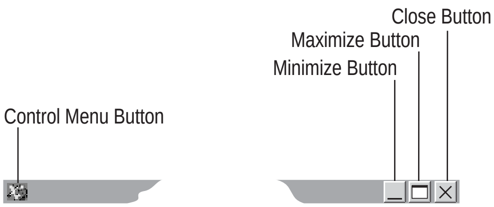
Work Layer Indicator Displays the name of the current layer and provides a menu for changing the work layer, creating a new layer and accessing the Layer Manager.

Memory Indicator Displays two numbers. The first is the RAM available on your system and the second is the amount of virtual memory available (RAM plus swap).

Title Bar

The Title Bar includes the name of the current document, the Control Menu, Minimize and Maximize/Restore buttons (Windows) or the Close and Zoom boxes (Macintosh).

Windows



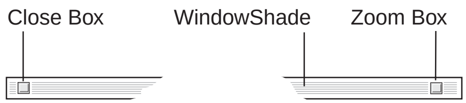
Control Menu Button Allows you to close, move, and change the size of the window. This button is available on all windows and many dialog boxes.

Double-clicking this button closes the window without displaying the menu. If you want to choose a different option from the Control menu, click the button once to display the menu and then make your choice.

Minimize Button Reduces the Vellum window to a task bar near the lower-left corner of the screen. This action does not close or save the document, it only shrinks the window to a task bar so you can perform some other Windows-related task. To display the window again, click once on the same button.

Maximize/Restore Displays the window, full or partial screen. Once the window appears full screen, click the button again to restore it to its previous size.

Macintosh

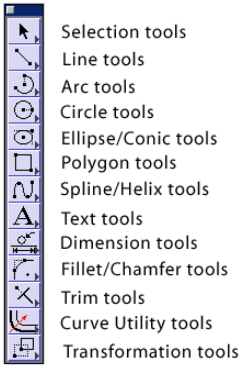


WindowShade This WindowShade contains the Close Box and Zoom Box. You can set your computer to allow you to reduce your file to just the WindowShade. See the *User Guide* that came with your computer.

Close Box Closes the window when you click on it. If you attempt to close the window without saving your work, your Designer Elements program displays a message so you can decide whether to save or not.

Zoom Box Toggles the window size between the previous size and full size.

Tool Palette



A tool palette is a group of tool icons that represent tools for drawing, editing and annotating geometry. This graphic here is the main tool palette.

Selecting a Tool from the Tool Palette

1. Position the arrow pointer on the icon of the tool you want to use.

2. Click the mouse button.

   The icon appears highlighted to indicate its selection.

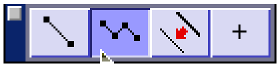
Floating Tool Palettes

All tool palettes in your Designer Elements program are floating tool palettes. Once a palette is displayed, you can move it to any location in your drawing area simply by dragging the palette by its title bar. You can save your palette to that location for future files by choosing ***File>Preferences>General*** and click the Save Now button.

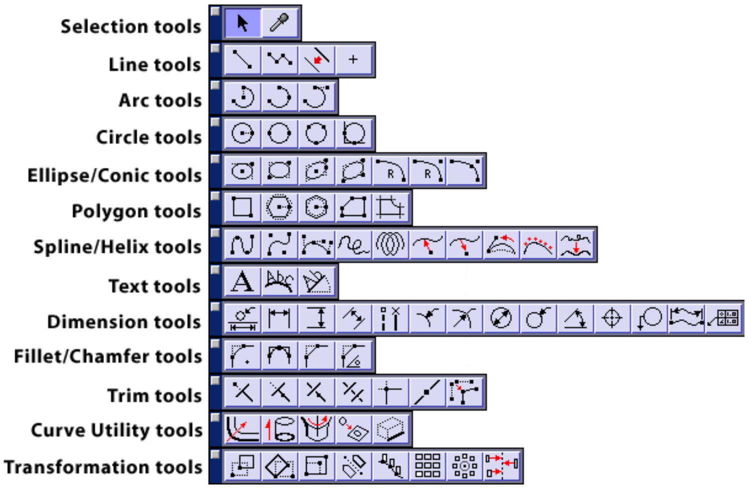
Tech Note:

Macintosh only: If you are using more than one monitor with your computer, you can save your palettes to the other monitor.

Tool Subpalettes

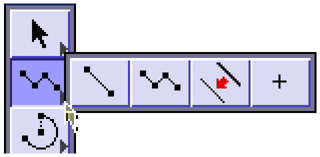


Most of the tools in tool palettes contain a subpalette of tools with related functions. The (arrow) in the lower-right corner of the tool icon represents the presence of a subpalette which contains related tools. The next graphic shows the **Wireframe** tool palette.



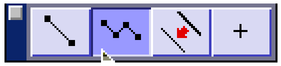
Viewing and selecting from a subpalette are similar to choosing a command from a menu.

Selecting a Tool from a Subpalette



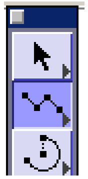
1. Position the arrow pointer on the tool.
2. Press the mouse button.

   The subpalette appears to the right of the tool.
3. Drag the pointer to highlight the desired tool.



4. Release the mouse button.

   The selected tool replaces the previous tool in the tool palette. The highlighted icon in the tool palette shows that your selection from the subpalette is the active or current tool.



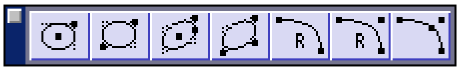
The new tool is visible in the tool palette until you select another tool from the same subpalette. The tools in the subpalette remain in the same order; only the tool displayed in the tool palette changes.

Once you select a tool, additional information appears to help with your construction. The Pointer, Pointer Locator, Message Line, and Status Line all provide feedback about the active tool.

If you want to select a tool already displayed in the tool palette, you only need to click it; you don't need to select it from the subpalette.

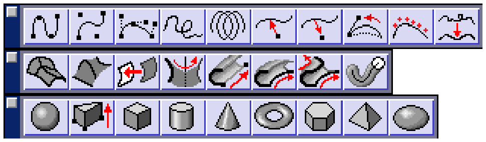
Tear Away Palettes

All tool palettes containing subpalettes are capable of "tearing away" from the parent palette located anywhere in your drawing area. The graphic here is the **Ellipse/ Conic** tool palette.



When you drag the mouse to the right across the subpalette beyond the last tool, the subpalette "tears away" from the parent tool palette. You can save the palette in this location by choosing ***File>Preferences>General*** and click on the Save Now button. Close the palette by clicking on the close button in its title bar.

When you tear away multiple tool palettes and drag them near each other they snap to a left alignment.

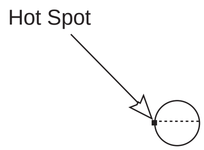


Smart Pointer & Wireframe Tools

When you select a tool and move the pointer into the drawing area, the pointer shape represents the tool.

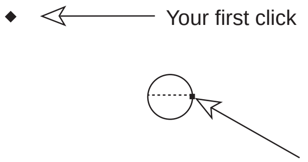
Some of the pointers, like the Single Line pointer, are simple cross-hairs. Others, such as the Opposite-Point Circle pointer, resemble the tool.

The pointer, called a smart pointer, displays indicators for multi-step procedures. Each smart pointer has a dot, the hot spot, showing the next point you should specify. The dot changes position on the pointer during each step of the construction.



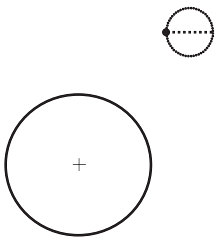
The smart pointer shows you where to click next.

For example, the Opposite-Point Circle pointer, illustrated above, shows that the first click of the mouse places a point on one edge of the circle you're creating. After you click a location, the hot spot moves to the other side of the pointer, showing that the next click places a point on the opposite edge of the circle. See the graphic below.



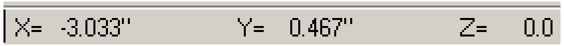
The hot spot moves to the other side of the smart pointer to indicate the next step.

After you click the second location, the circle appears. The hot spot moves back to its original position on the pointer so that you can create another circle.



Location Indicator

The Location Indicator is located next to the Layer indicator at the bottom of the drawing area.



This indicator continuously tracks the pointer location when the pointer is in the drawing area, displaying the X, Y and Z coordinates of the current location relative to the origin. The origin (0,0,0) appears in the center of the screen when you open a new document. If you are displaying the Axis, the origin is at the intersection of the x, y and z axes.

Message Line

The Message Line across the top of the drawing area provides concise instructions for the use of the current tool.

For example, after selecting the **Center-Point Circle** tool, the Message Line appears as illustrated here.

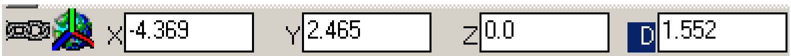
The instructions in the Message Line for some tools also indicate optional activities. For example, if you hold down the CTRL (Windows) or OPTION (Macintosh) key while using the **Center-Point Circle** tool, the next mouse click creates a copy of the last circle with the center placed where you clicked.

Status Line

Tech Note:

The number of decimal places displayed in the Status Line fields is determined by the decimal setting in the Units page of Preferences.

The Status Line provides measurements, angles, X, Y and Z coordinates and delta values for the current construction. The current tool determines the number of status fields and which of the status fields highlights after the construction. For example, if you select the Center-Point Circle tool, the Status Line shows the X,Y and Z coordinates for the center of the circle and the length of the diameter.



When you click the last point of the circle, the diameter (D) entry field highlights in the Status Line to indicate that it is active. It shows the diameter of the circle you just created. If you type a new number, and press the ENTER (Windows) or RETURN (Macintosh) key, the diameter of the circle you just created changes.

You can change any or all entries in the Status Line. If you change the entry before the tool operation, the values automatically register when you click in the drawing area for the particular operation. If you enter a value after the geometry is created and then press ENTER (Windows) or RETURN (Macintosh), the changes are made to the geometry and you can't make any more changes in the Status Line.

The number of decimal places displayed in the data fields is determined by the Precision setting in the Units page of Preferences.

The Status Line uses the *World coordinate system* for all values entered in the data fields. Values can be entered in inches, feet, millimeters, centimeters, meters and mathematical expressions (ex. 10”+2.54 cm).

When the units in Preferences are set to feet and inches, it's important to be aware of the following rules:

- All numbers are assumed to be feet unless accompanied by the unit symbol, like “ for inches. Entering a 1.5 in the field is read as 1.5 feet. If you want 1.5 inches, enter 1.5”, 1.5i, 1.5in, 1.5 inch, etc.
- If you want to enter fractions of inches, each entry must include the unit symbol. For example, 5 feet 6 5/8 inches must be entered 5’ 6” 5/8”. Internally this is converted as 5’ + 6” + 5/8”. If the inch symbol is not included with the fraction, 5/8 will be interpreted as a fraction of a foot.

Copying and Pasting Status Line Entries

You can copy and paste Status Line text for use in another data field.

For Windows, hold down the right mouse button and use the *Copy* and *Paste* commands available in the menu. You cannot use the *Copy* and *Paste* commands in the Edit menu.

For Macintosh, use z +C to copy and z +V to paste text. You cannot use the *Copy* and *Paste* commands in the Edit menu. These function only for your Designer Elements program data.

Moving between Status Fields

You can use the TAB key to move to the right, highlighting the next field. When you press ENTER (Windows) or RETURN (Macintosh), the entry either changes or the construction redraws according to the new specifications in the Status Line. You can also use your mouse to activate a Status Line field.

Tip:

You can also change objects with the Edit Objects command or double-clicking on the object to display the Edit Objects dialog box.

Use the Status Line arrows to scroll if any of the fields are off screen.

Expression Parsing

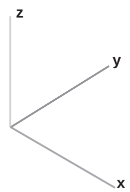
The status line entry fields also accept mathematical, trigonometric, and exponential operators. Position the cursor in the text field and type in the additional operation. See Appendix A for the list of supported operators.

Coordinate System Axis

The coordinate system axis, at the left of the Status Line, displays the coordinate system currently set for the file. You can choose either the Global (world coordinate system) or a user-defined coordinate system. Clicking on the axis displays a menu from which you set the coordinate system.

The default system is the Global coordinate system. See Chapter 7 for more information.

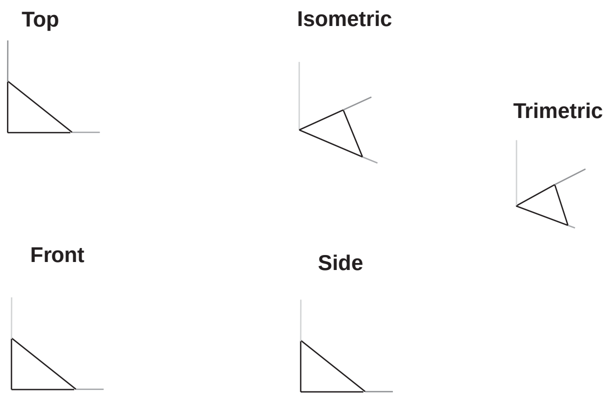
Axis



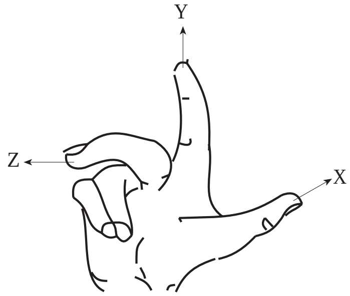
The Axis displays the current view orientation of the x, y and z axis in the center of your screen. When you change views, the axis will adjust accordingly. You can turn off the display by choosing ***View>Show Axis***. If you do not want the Axis to display at start up, choose ***File>Preferences>General*** and deselect the Axis option.

Triad

The Triad displays in the upper-left corner of your drawing area to illustrate the orientation of the x, y, z axis and the work plane. The example below uses the default view definitions.



The Triad represents the principle of the right-hand rule—a memory aid for the relative directions of the positive axes. With your right palm upturned, the thumb (x) points right, the index finger (y) points straight ahead, and the middle finger (z) points up. If you move your hand to indicate the x and y-axes, you can easily see the direction of the z-axis.



The Right Hand Rule

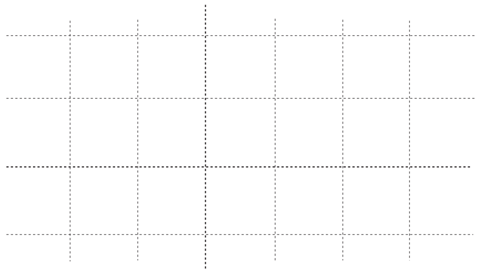
You can turn off the Triad by choosing, ***Planes>Show Triad***. If you do not want the Triad to display at start up, choose ***File>Preferences>General*** and deselect the Triad option.

Drawing Area

You use the drawing area for all construction, editing and annotation of geometry. Think of the drawing area as a sheet of paper of unlimited size that you use to construct full-size unscaled drawings. You use the scroll bars to move the sheet so the portion you want to work on is visible in the window.

Displaying the Grid

If you wish to work with a grid in the drawing area, choose ***Planes>Show Grid***.



If you choose ***Window>Snap***, the Snap Options dialog box appears. Choose To Grid and your constructions snap to the grid, meaning that any geometry point that you click snaps onto the closest grid point.

Scroll Bars

The scroll bars allow you to move the sheet up and down or right and left. You can display different parts of the drawing sheet by dragging the slider of a scroll bar to the approximate location. For example, the right, center, or left position in the horizontal scroll bar displays the right side, middle, or left side of the drawing, respectively.

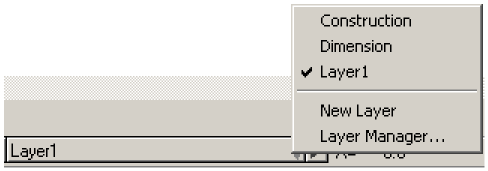
You can also click the arrows at the end of the scroll bars to move the sheet one line at a time.

Work Layer Indicator

The Work Layer Indicator in the lower-left corner of the screen shows which layer is the current work layer. New geometry goes on the work layer. If you want your construction to go on a specific layer, first make it the current layer.

You can select the work layer in two ways:

- Click on the arrow to the left or right of the layer name and the layer will move backward or forward to another layer.
- Position the pointer over the Work Layer Indicator, then press the mouse button. All available layers are then displayed in a pop-up menu from which you can select a different layer to be the current work layer. Drag to the new work layer. All new geometry will be placed on that layer. You can also create a new layer and display the Layer Manager from the Work Layer Indicator.



Memory Indicator

The Memory Indicator field displays two numbers, the amount of RAM available on your system and the amount of virtual RAM (RAM plus swap). For Macintosh users, when the value reaches 5 MB use extreme caution because your Designer Elements program may crash. You are advised to save your files, close the program and then restart it.

Remember that the minimum required RAM for your Designer Elements program is 256 MB. The recommended RAM is 512 MB.

Menu Bar

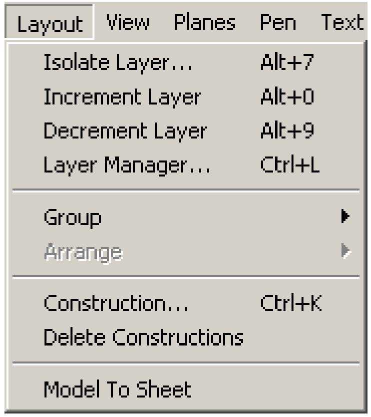
Your Designer Elements program menus contain related commands and settings.

File	Contains commands that affect entire documents (files), including opening and closing files, setting

	preferences and defining your Print Setup (Windows) or Page Setup (Macintosh).
Edit	Contains commands to select and manipulate objects. These include such things as copying and pasting as well as changing an object's direction, resolution, layer and type.
Layout	Contains commands and settings that specify the drawing area and provide program features and functionality such as the Grid, Layer Manager, Group and Align.
View	Contains commands for displaying your document and includes choosing and setting views and zoom options.
Planes	Contains commands for choosing and defining the work plane.
Pen	Contains commands to specify pen characteristics (color, weight, and pattern), polygon patterns, polygon fill, crosshatching and arrows.
Text	Contains commands to set the font, size, style and case of your text (lower, upper and title capitals).
Dimension	Contains commands that specify dimensions, their format and tolerance.
Verify	Contains commands to obtain information about your file and specific objects in your file. These include properties of an object, direction, curvature and object counts.
Window	Contains commands for displaying tool palettes, the Design Explorer, Trackball, etc.
PhotoRender	Contains commands for rendering your geometry, setting options for rendering and placing lights.
Animation	Contains commands for generating Quicktime movies.

Help Contains commands to access the on-line help file (Windows only).

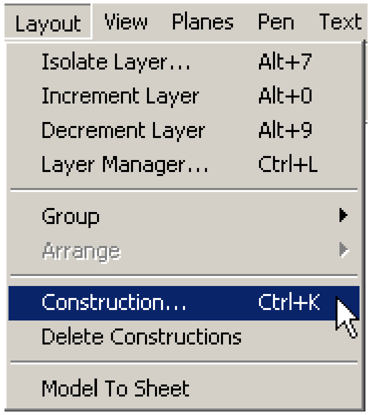
Displaying a Designer Elements program Menu



1. Point to the menu name.
2. Click on the name.

   The menu appears. If you want to dismiss the menu without making a choice, click outside the menu.

Choosing a Command from a Menu



1. Point to the menu name.
2. Click on the name.

   The menu appears.
3. Click on the command.

   The command executes, or the setting, such as *Snap To Grid*, toggles on or off.

Mouse versus Keyboard

Designer Elements program menu items can be chosen with the mouse or with a combination of keys on the keyboard. For example, you can use various methods for displaying the Edit menu.

Windows and **Macintosh**:

- Click on Edit in the menu bar.

Windows only:

- Press the ALT key and then type **E**.

- Press the ALT key and then press the RIGHT ARROW key until Edit is highlighted in the menu bar; then press ENTER.

You also have various methods for choosing commands with the keyboard. For example, you can use any of the following methods to choose ***Planes>Show Grid***.

Windows:

- Press ALT and **L** and then type **G**.
- Press ALT and then use the RIGHT ARROW key to highlight Layout and press ENTER. Then press the DOWN ARROW key to move the highlighted area to *Show Grid* and press ENTER.
- Hold down the CTRL key and type **G**.

The first method is the mnemonic method. Press the ALT key with the appropriate letters for the menu and command as indicated by the underlined character in the names.

Macintosh:

- Hold down the z (command) key and type **G**.

The third method for Windows and the only one available for Macintosh is a keyboard accelerator. When available it is denoted by the key sequence listed on the menu.

While keyboard functionality is always available, this manual generally describes making choices with the mouse.

Submenus

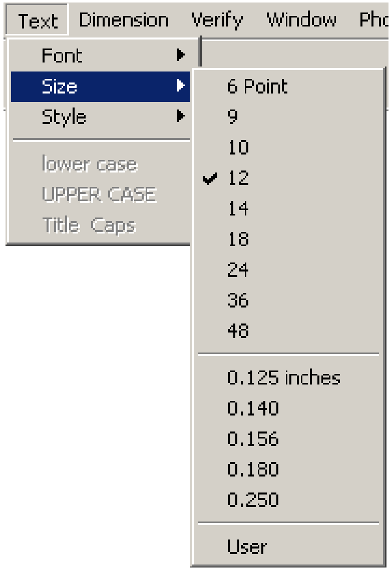
Commands followed by an arrow symbol have submenus which display when the command is highlighted.

1. Pull down the menu.
2. Click on a command followed by an arrow symbol.

   The submenu displays.
3. Click on the submenu.
4. Click the desired command.

Tip:

Windows only: You can also hold down the ALT key and type a letter to select the first item that begins with that letter, then use the arrow keys to move to the selection you want. Once your choice is highlighted, press the ENTER key.



Dialog Boxes

When you choose a command followed by an ellipsis (…), such as *Edit Objects* in the Window menu, a dialog box appears.

Dialog boxes allow you to qualify the command you chose by adding information. For example, in the Edit Objects dialog box here, you can change the specifications of the selected object.

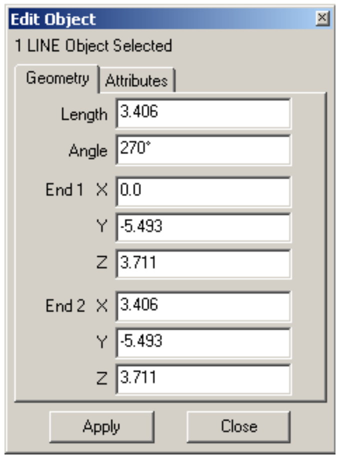
If a dialog box obscures your view of the drawing area, you can move it to a new location by dragging it with the pointer on the Title bar.

Tech Note:

Windows only: Designer Elements program dialog boxes do not support using the large font setting in the Setting page of the Display Properties dialog box for your Operating System.

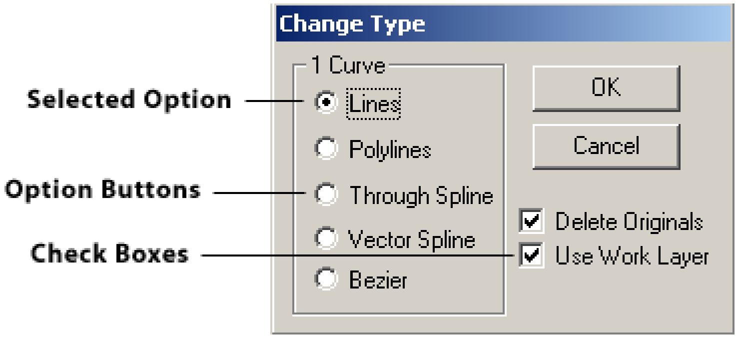
Tech Note:

If you want to copy and paste data between data fields in dialog boxes, you cannot use the Copy and Paste commands. For Windows, press the right mouse button and use the commands in the menu. For Macintosh, use ctrl+C to copy and ctrl+V to paste.



Option Buttons

Option buttons indicate mutually exclusive choices; you can select only one option at a time. Click the option you want and the button turns black, as shown by the inches option below.



Check Boxes

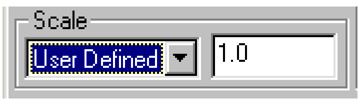
Check boxes, as shown above, provide options you can switch on and off and which are not mutually exclusive. A check mark shows the option that is set.

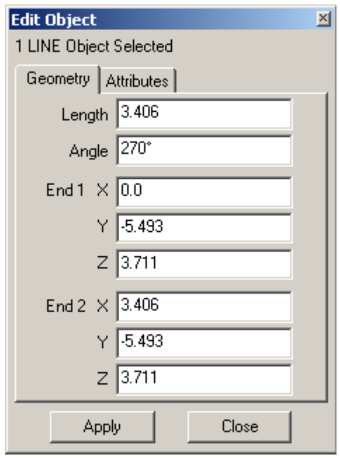
List/Entry Fields

Some dialog boxes contain lists of options, displaying an arrow to provide access to the list.

If the entry includes an arrow, you can display a menu which works like a submenu on the menu bar but the item you specify appears in the field once you select it.

Some list fields also allow you to type an entry. For example, you can type a value in the Scale entry field in the Drawing Size dialog box or choose from the pull-down menu, indicated by the arrow. See the graphic here.

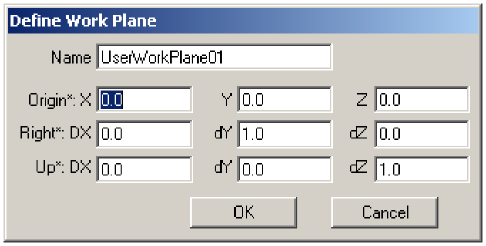




To type an entry, select the current entry (if it isn't already selected), then type a new entry. In most cases, clicking OK, saves the changes.

Asterisks

When an item in the dialog box displays an asterisk (*), you can specify a value by clicking or dragging in the drawing area. This feature is particularly useful for specifying location because you don't need to know any x, y and z coordinates.



Apply Buttons

Some dialog boxes have an Apply button that allows you to apply the specification you just set. You can leave the dialog box open to set other specifications.

For example, once you make a change to an object in the Edit Objects dialog box, you can click apply and leave the dialog box open for future changes.

Closing a Dialog Box

If a dialog box contains an OK or Cancel button or an action button such as Open, the dialog box closes when you click the button. Otherwise, you dismiss the dialog box manually by double-clicking the Control Menu (Windows) or clicking the Close Box (Macintosh) in the upper-left corner of the box.

Toggling Commands

Commands that set a condition (such as *Snap To Grid* and *Arrow At Start*) display a check mark in the menu to indicate that they are active. To turn a command off, choose it and the check mark will disappear.

In the case of pen patterns and text characteristics, the check shows the current setting.

The Drafting Assistant

The Drafting Assistant is the feature that makes your Designer Elements program unique among design and drafting software products. The Drafting Assistant thinks like a drafter; it automatically knows where you typically want construction lines and displays them temporarily when you need them.

The Drafting Assistant also makes it easy to select existing points for construction by displaying information about the pointer's location in the drawing area. If a Drafting Assistant notation displays when you click, the construction snaps onto the geometry precisely, without requiring finely tuned eye-hand coordination or tedious selection of special modifiers, modes, or other specialized construction tools.

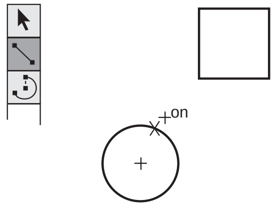
The following topics are covered in this chapter:

- Snapping onto Geometry
- Drafting Assistant Construction Lines
- Permanent Construction Lines
- Drafting Assistant and Display

Snapping onto Geometry

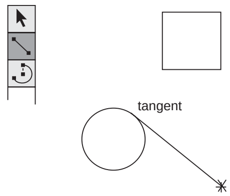
When the pointer is in the drawing area, it has a snap point function. The snap point locks onto specific points on existing objects as you move the pointer near them.

The Drafting Assistant tells you when the snap point is *on* an object.



The Drafting Assistant snapping onto a circle

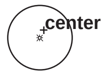
The Drafting Assistant displays information about the location of the snap point. This information appears either beside the pointer or next to the object itself.



The Drafting Assistant displaying the relationship between the circle and the line that is being constructed.

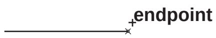
The Drafting Assistant tells you when a snap point locks onto object points.

center



The center of an arc or circle. Move the pointer across the arc or circle to display *on* for the arc or circle, then move the pointer near the center to display the *center* point notation.

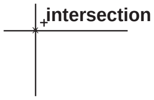
endpoint



The endpoint of lines, arcs, circles, ellipses and splines.

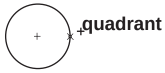
midpoint

The midpoint of lines, arcs, circles, ellipses and splines.



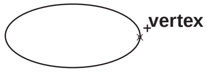
intersection

The intersection of two curves (permanent and the Drafting Assistant's dynamic construction lines).



quadrant

Quadrant points on an arc or circle displayed at 3 o'clock, 6 o'clock, 9 o'clock and 12 o'clock.

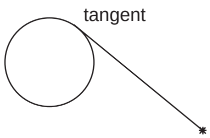


vertex

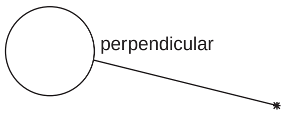
The vertices of an ellipse, spline, or dimension point.

Using Tangents and Perpendiculars

If you click a point *on* an arc or circle and drag the pointer away at about a 45° angle, the Drafting Assistant locks onto the *tangent*. If you drag away at a 90° angle the Drafting Assistant locks onto a *perpendicular*.



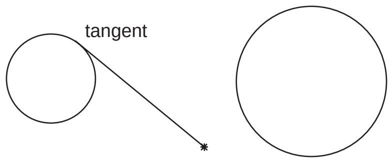
Tangent



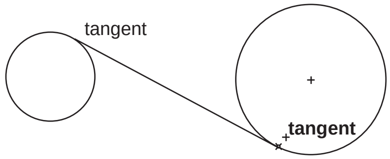
Perpendicular

If you continue holding down the mouse button, the line remains *tangent* or *perpendicular* while you drag the ending point around the object.

This is a useful feature if, for example, you want to create a line from and tangent to an existing circle to the tangent point of another circle.



Once a line is *tangent* to the circle, you can drag it to the tangent point on the other circle, with the tangency maintained at both ends.



The Drafting Assistant locks onto a *tangent* or *perpendicular* only when the Drafting Assistant starts from the *on* notation. You cannot begin from a specific point, such as *endpoint*, *quadrant* or *vertex*.

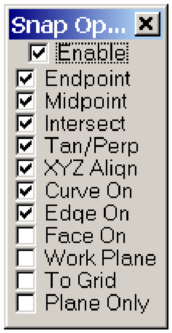
Drafting Assistant Axis Locking

The Drafting Assitant support locking to the X, Y or Z axis. To lock to one of these axis hold down the corresponding key on your keyboard (X, Y or Z). The Drafting Assistant will only show alignment points in that axis.

Customizing the Drafting Assistant

When you first start your Designer Elements program, the Drafting Assistant automatically activates. You can customize the Drafting Assistant in a few different ways by using the *Snaps* command or changing your preferences in the *Snap* page of the Preferences dialog box.

Snap Command



This command, found in the Window menu, displays the Snap option dialog box and allows you to activate and deactivate the different snaps modes of the Drafting Assistant.

When an option activates the Drafting Assistant operates in the following manner as a pointer moves across geometry:

Snaps Enabled	The Drafting Assistant is enabled. Snaps that are checked display. (De-selecting this option is the same as turning the DA off.)
Endpoints	Object endpoints display.
Midpoints	Object midpoints display.
Curve Intersections	Intersections between objects display.
Curve On	The point nearest to the pointer displays with an *on* notation.
XYZ Align	Alignments along the x, y and z axes display.

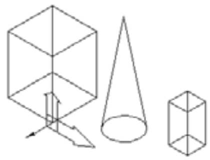
Tangents/Perpendiculars Tangents and perpendiculars for your geometry displays.

Face On A face notation displays for solid objects.

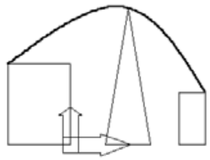
Edge On All notations normally associated with curves, such as endpoint, midpoint, and vertex display for solid objects.

Work Plane This snap option allows you to create a curve using referencing points on 3D objects but confined to your current 2D work plane. This snap works best when you are viewing your current work plane. See the example here for clarification.

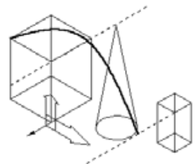
Example: You have three objects from which you want to reference points. Display the Work Plane icon for the Front plane.



Change the view to front and draw a curve using the points from the objects with the help of the Drafting Assistant.



Change the view and you can see the curve aligned with the current work plane. The dotted lines show that the curve aligns with the object points.



To Grid When using the grid, this option snaps to grid alignment points.

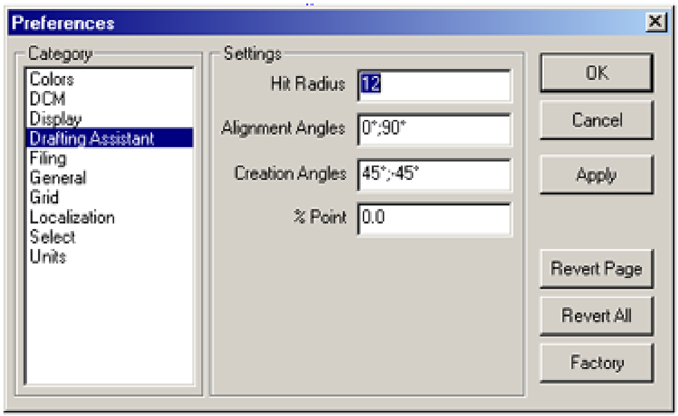
Plane Only Snap points are given for objects **only** on the current work plane.

Each of these snap options have Short cut equivalents. See the “Short Cuts” section in Chapter 6 for more information on using Short cuts.

To turn off an option, click the check box to remove the mark next to the option name. By default, *Solid Face Projections*, *Project to Work Plane*, *To Grid* and *Plane Only* are turned off.

Drafting Assistant Preferences

When you choose, ***File>Preferences>Drafting Assistant***, the following dialog box appears.



The *Drafting Assistant* page of Preferences includes the following options:

Hit Radius

This setting determines the detection distance in pixels. When the pointer is within the specified Hit Radius, the Drafting Assistant notations are displayed and the object is selected when you click the mouse. The default Hit Radius is 12 pixels.

If you can't specify locations that are close together because the Drafting Assistant snaps to an existing control point, you can do any of the following:

- Decrease the Hit Radius.
- Zoom in so more pixels separate the existing point and the point you want to select.
- Lock on a point by pressing the mouse button and typing the letter that represents the point (*m* for midpoint, for example).

If you set the Hit Radius to zero, you disable the single click selection of the **Selection** tool (dragging a selection fence and double-clicking to select all objects will still work). Instead, you

might consider using the *Select Mask*
in the Window menu to specify that some objects cannot be selected.

Alignment Angles

These angles define the dynamic construction lines that the Drafting Assistant automatically uses. If you want to change the orientation of your drawing, you can change these specifications.
For example, you could set these angles to 30°, 90° and 150° for an isometric drawing. The defaults are 0° (horizontal) and 90° (vertical). Use a semicolon to separate the values.

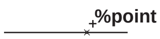
To display a dynamic construction line through a point, move the pointer to the point to activate it (a diamond appears); construction lines automatically display through the active point. You can have as many as eight active points. When you activate the ninth point, the first one in the series deactivates.

Creation Angles

These lines are used by the Drafting Assistant only when you are creating geometry and they are not part of the list of lines generated from the eight active points. The defaults are 45° and -45°. Use a semicolon to separate the values.

% Point

The divisions of a line for Drafting Assistant notations. If you want to divide the line into quarters, use a 25 specification. The default is 0.0.



For example, entering 25 instructs the Drafting Assistant to tell you when the pointer is 25% of the distance along a line.

Drafting Assistant Construction Lines

In addition to snapping onto geometry, the Drafting Assistant displays *dynamic construction lines*. The three types of construction lines you use most frequently—

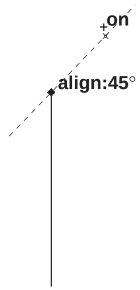
vertical, horizontal and 45° angle lines—display automatically during construction. These construction lines appear temporarily to help you align geometry. Once you've set a point, the Drafting Assistant construction line disappears—your drawing is not cluttered with extraneous lines.

Dynamic construction lines extend automatically from the last point you created. You may want to activate other points so the Drafting Assistant displays construction lines relative to them. Simply move the pointer over the geometry to activate or "wake-up" its control points, then move away horizontally or vertically.

Of course, you also have the option to create *permanent construction lines* and other shapes, as described later in this chapter.

Using the Drafting Assistant's Construction Lines

After you've indicated the first endpoint of a line, move the pointer horizontally, vertically, or in a 45° direction, the dynamic construction line appears. The figure here illustrates a 45° construction line relative to the endpoint of an existing line.



Displaying Dynamic Construction Lines while Constructing Geometry

Tech Note:

You can have as many as eight active points; activating the ninth point in a series deactivates the first point.

1. Click a point to begin new geometry.
2. Move the pointer away from the point horizontally, vertically, or at a 45° angle.
3. While the construction line is visible and the Drafting Assistant displays *on*, click the next point.

The point is placed exactly on the construction line, even though the pointer wasn't exactly on that line when you clicked. The dynamic construction line disappears.

Displaying Dynamic Construction Lines with Existing Geometry

1. On selected geometry, without pressing the mouse button, move the pointer over an existing point.

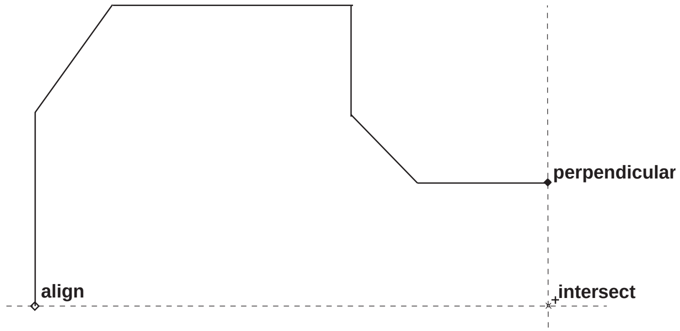
   The point notation (*endpoint* or *midpoint*, for example) shows that the point is active.
2. Move the pointer horizontally or vertically.

   A construction line appears through the point.

3. While the construction line is visible and the Drafting Assistant displays *on*, click the desired point in the construction.

   The point is placed exactly on the construction line, even though the pointer wasn't exactly on that line when you clicked. The dynamic construction line disappears.

The figure here illustrates intersecting construction lines drawn through two existing, active points.



Once you are familiar with the Drafting Assistant, you'll see how much your Designer Elements program streamlines design and drafting tasks!

Setting New Drafting Assistant Construction Angles

You can add to or change the angles that the Drafting Assistant uses for dynamic construction lines by choosing ***File>Preferences>Drafting Assistant***. Enter the new construction line angles, separated by semicolons, in the Alignment and Creation Angles fields.

Locking the Drafting Assistant to Work Plane

By holding down the Shift key, the Drafting Assistant locks to the x, y and z axes of the work plane. Only *align:x*, *align:y* and *align:z* construction lines appear.

Permanent Stroke Construction Lines

In addition to the Drafting Assistant's dynamic construction lines, you may want to create construction lines that display until you hide or remove them.

Stroke construction lines are lines that you create with the mouse. Hold down the SHIFT+CTRL keys (Windows) or the z key (Macintosh) and drag the mouse horizontally or vertically. Since you are working in the drawing area, the Drafting Assistant helps you place the stroke precisely.

Tip:

You can use strokes to create construction lines while you are in the process of using a tool.

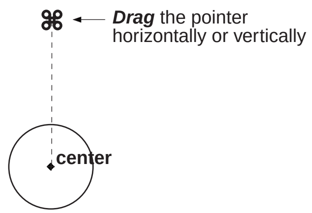
Windows users: When both the CTRL and SHIFT keys are used by a tool, Stroke construction lines cannot be created while in the tool.

Construction lines automatically appear on the Construction layer, not the work layer of your drawing. If you have inadvertently deleted the Construction layer, it recreates the next time you create a construction line.

You can hide the construction layer to view or print the drawing without construction lines. You also can select construction lines in the usual manner from any layer; the Construction layer doesn't have to be the work layer. (More information on layers can be found in Chapter 31).

When you want to get rid of all construction lines, choose ***Layout>Delete Constructions***. Everything on the construction layer deletes and not just the construction lines.

Holding down the SHIFT+CTRL keys (Windows) or the z key (Macintosh) changes the mouse pointer to the Stroke pointer (z).



Drag	Result
Vertically	A vertical construction line appears through the first point of the stroke.
Horizontally	A horizontal construction line appears through the first point of the stroke.

Using Stroke Construction Lines

Construction lines are as long as the dimensions of the viewing area of the screen or the printing region (as designated in *Print Setup* (Windows) or *Page Setup* (Macintosh) in the File menu), whichever is larger.

For example, if you are using the **Connected Lines** tool, you can create a construction line that extends through the center of a circle:

1. Hold down the SHIFT+CTRL keys (Windows) or the z key (Macintosh). The pointer becomes the stroke pointer (z).
2. Move the pointer near the center of the circle.

   The Drafting Assistant snaps onto the *center* point.
3. Drag the mouse vertically or horizontally away from the *midpoint*.

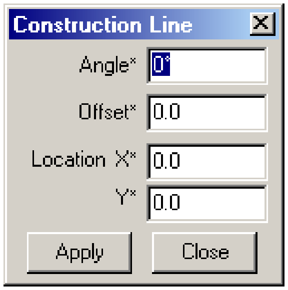
   The construction line appears through the center and you are still in the process of creating connected lines after releasing the SHIFT+CTRL keys (Windows) or the z key (Macintosh).

The Construction Command

Stroke construction lines are useful for creating lines through existing points. If you want to create a construction line at a location other than an existing point or at a particular angle, you can use the *Construction* command.

Using the Construction Command - CTRL+K (Windows); z +K (Macintosh)

This command in the Layout menu creates a construction line on the construction layer of the current document.



Tip:

Vellum 3D Users: You cannot enter multiple angles or offsets to create multiple construction lines.

You can specify the angle of the construction line or the offset from a reference point defined by the X, Y coordinates. The asterisk shows that the values can be specified by clicking or dragging the mouse; values can also be typed into the fields.

The distance dragged will always be entered in the Offset data field as a positive value, regardless of the direction dragged.

You can only enter one set of values for a new construction line at a time. You cannot enter multiple angles to create multiple construction lines.

Specifying the Construction Line Angle with the Mouse

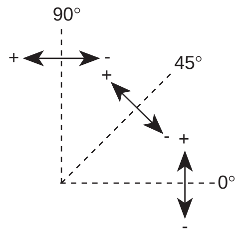
1. Click the *Angle* field.
2. Drag a vector in the drawing area.

The angle of the vector line appears in the *Angle* field.

Specifying the Construction Line Offset with the Mouse

1. Click the *Offset* field.
2. Drag the offset distance in the drawing area.

The distance you drag appears in the *Offset* field. Be aware that the offset is determined by the angle of the construction line as shown.



Specifying X,Y Coordinates with the Mouse

The coordinates of the last point you specified appear in the *X* and *Y* fields, but you can change them by doing the following:

1. Click the *X* field.

2. Enter new coordinates

   or

   In the drawing area, click the location of the point through which the construction line should pass. The coordinates are entered automatically for both the *X* and *Y* fields.

Creating Parallel Construction Lines

You can create parallel construction lines by creating one construction line using this field, then creating new lines with the **Parallel Line** tool.

Tech Note:

If you create construction geometry on the construction layer, be sure you change layers before creating your actual geometry. When you choose Layout>Delete Constructions, everything on the construction layer is deleted and not just construction lines. Choose Undo to restore your geometry and move it to another layer

Creating Construction Geometry

Non-construction geometry is placed on the work layer. You can create temporary construction geometry, such as arcs or circles, by making the construction layer the work layer, creating the geometry and switching to another layer to continue your work.

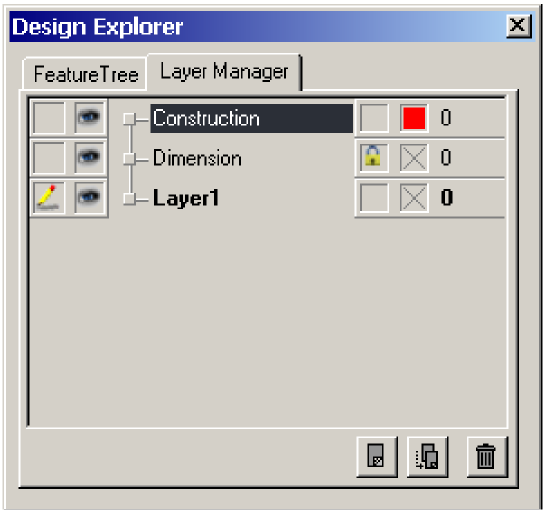
Creating Construction Geometry

1. In the Layout menu, choose *Layer Manager*. (The dialog box will say Design Explorer because to two boxes [Design Explorer and Layer Manager] are combined.)

2. Click in the work layer column just to the left of Construction. (Move the pencil to that layer.)

3. Click OK. The Layer Manager closes and the Construction layer is now the work layer. You will note this in the *Work Layer Indicator* at the bottom left of your screen.

4. Create the geometry you will use for construction.



5. When the construction geometry is complete, make another layer the work layer by changing the layer in the Layer Manager or the Work Layer Indicator.
6. Continue your work.

Once you no longer need the construction geometry, choose ***Layout>Delete Constructions*** to remove all geometry on the construction layer.

Removing Construction Lines

If you have many construction lines and you want to delete only one or two of them, select the lines you want to remove, and then choose the BACKSPACE (Windows) or DELETE (Macintosh) key. You can remove all the construction lines that you've created by choosing ***Layout>Delete Constructions***.

Delete Constructions

This command in the Layout menu deletes all construction lines and any geometry on the construction layer. The Drafting Assistant's dynamic construction lines appear only temporarily and are not affected by this command. Any geometry on the construction layer (regardless of the pen style used) deletes by this command.

You can retrieve deleted construction geometry with the *Undo* command.

Drafting Assistant and Display

The Drafting Assistant enables you to accurately create and place objects. When you create an object that shares a common control point with another object, both control points are in the same X, Y, Z location.

If you zoom in on that specific control point, depending on the resolution capabilities of your monitor, it may appear that the objects are not connected at the control point. This is a limitation of your hardware. The Drafting Assistant has correctly located your geometry.

Selecting Objects

Once you've constructed objects, you may want to make changes. To do so, you will have to select the desired object. In your Designer Elements program, the step at which you select the object depends on the operation you want to perform. In both cases it, modifying an object is always a two-step process.

Modifying with a Tool

1. Select the tool.
2. Select the object to be modified.

For example, use the **Divide** tool to divide a curve into multiple pieces.

Modifying with a Menu Command

1. Select the object.
2. Specify the action for the selected object.

For example, you could select a circle and then change the pen pattern to Center to indicate a bolt-hole circle.

The following topics are covered:

- Objects
- Indicating Selection Preferences
- Selection Process
- Eye Dropper tool

Objects

A line is an object

Single Object

A single piece of geometry is an object.

Several objects that have been grouped with the *Group* command are also an object and are selected when you click on any member in that group.

Referral:

Selecting points is described later in this chapter.

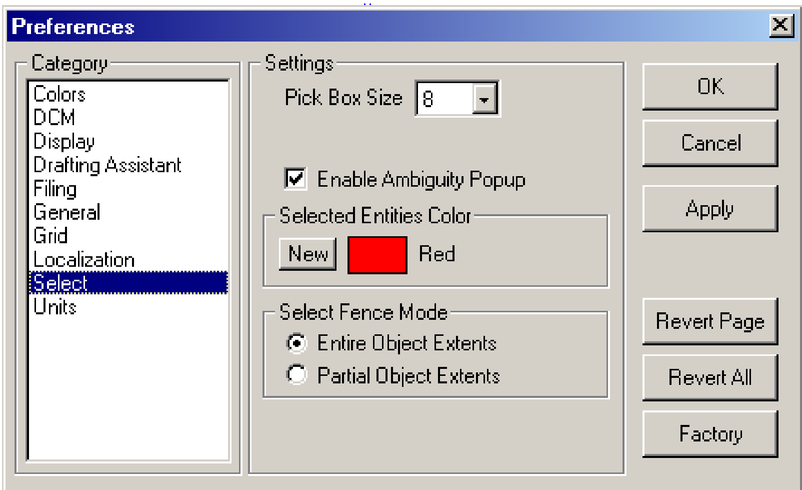
A line without points displayed

A line with points displayed

A point is an object, too. Every type of geometry contains one or more points, sometimes called *control points*. A line has two control points, one at the beginning and one at the end of the line. If you select an object and choose ***Edit>Show Points***, the points are visible on the selected object. If you select a point without selecting the geometry it defines, Your Designer Elements program treats the point as an object.

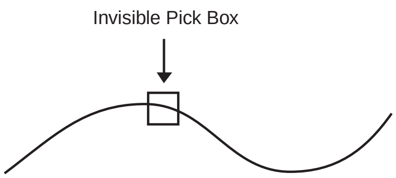
Indicating Selection & Preferences

When you select an object, its appearance on the screen shows that it is selected in a specific color. The default color chosen is red for your Designer Elements program. You can choose you own color as well as choose other settings for selecting objects by choosing ***File>Preferences>Select***. The following selection page is displayed.



Pick Box

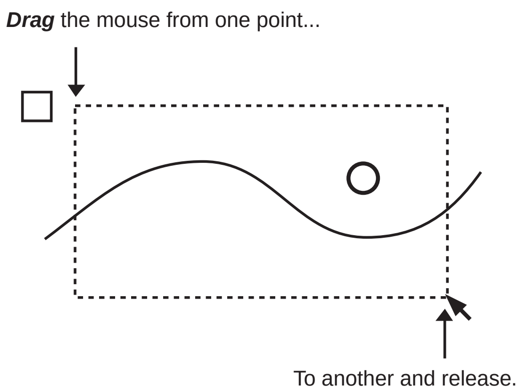
The pick box is an invisible box centered about the pointer tip. The default size of the box is 8 x 8 pixels. The pick box requires you to move the pointer over the desired object and click the mouse button. Objects that are within the bounds of the pick box are selected. See the graphic below.



Select Fence

The selection fence is a rubber banding selection window. You can choose between one of two settings, *Entire Object Extents* or *Partial Object Extents.*

Entire Object Extents Only objects that lie completely within the pick window are selected. In the graphic below, only the circle is selected.



Partial Object Extents Any object that lies within the selection fence window is selected. In the graphic above, both the circle and the spline are selected.

Selection Color

You can choose any color for the selection color. In the *Selected Entities Color* section, click New and your standard color display appears. Choose the color, click OK and you are returned to the *Select* page of the Preferences display. Once the selection color changes, all current and future selected items appear in the new color.

Do not use black for indicating selection because black is the default color for all curves.

Selection Process

Selecting an object does not affect the properties of the object. A selected object highlights, but this highlighting goes away once the object is deselected. You can select an object by using either the **Selection** tool or by the hollow selection tool that appears temporarily when using one of the editing tools.

You can use the **Selection** tool to select both objects and points. While the selection process for objects and points are quite similar, there is enough difference to merit separate attention.

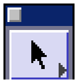
Selecting Objects with the Selection Tool

To select an object, begin by clicking the **Selection** tool in the main tool palette. You can select one object, SHIFT-click more than one object, or drag a selection fence around a group of objects.

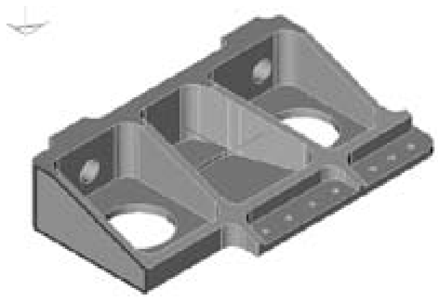
Tip:

Having trouble selecting objects when displaying many objects? Try using the Selection Mask, a selection fence, or rotate the view slightly and try again.

Selection Tool



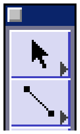
The **Selection** tool in the main tool palette selects one or more objects (curves, solid edges, faces, etc.) or points in the drawing area. The graphic here shows the selection of a solid face.



You can use the **Selection** tool to edit previously created geometry. Click on the object to be edited and use the **Status Line** to edit the parameters of the object.

Selecting a Single Object

1. Click the **Selection** tool in the tool palette. The Message Line reads: *Selection: Select [Shift=Extend, Ctrl* (Windows) or *Option* (Macintosh) = *Copy]*.



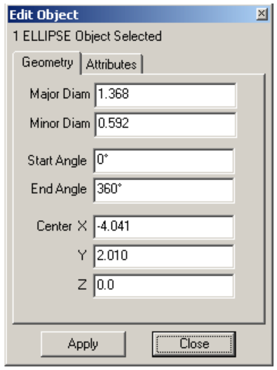
2. Move the pointer to an object and click. The object is selected, and any previously selected objects are deselected.

Referral:

The Pick Objects command in the Planes menu is for setting the work plane not for selecting objects on which to perform an operation. See Chapter 32, “Planes” for more information.

Double-selecting a Single Object

Double-click on an object to display an Object’s characteristics. This will bring up the Edit objects dialog box. You can also make changes to the object through this box. See Chapter 24 for more information.



Selecting Multiple Objects by SHIFT-clicking

1. Choose the **Selection** tool in the tool palette.
2. Move the pointer to an object and click.
3. Press down the SHIFT key.
4. While holding down the SHIFT key, click other objects to be selected.

   The objects you clicked are selected. If you then click one of the selected objects, it is deselected.
5. Release the SHIFT key.

Selecting Multiple Objects by Dragging

If you want to select more than one object, you can drag a selection fence around the objects.

1. Click the **Selection** tool in the tool palette.
2. Drag a selection fence around the objects you want to select.

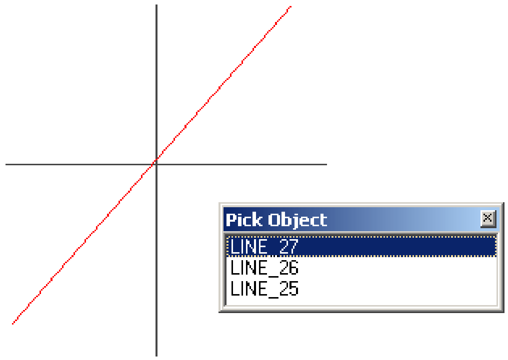
If you want to select most of the objects within an area, you can drag a selection fence to select all the objects, and then deselect the objects you do not want by holding down the SHIFT key and clicking them.

Selecting One or More Objects when they Overlap (Ambiguity Popup box)

Tip:

You can specify that only certain objects, layers, or colors are selected by setting a selection mask with the Select Mask command. Select All is useful if you want to make a global change in a drawing, such as changing the width of all lines.

Often times with more complex drawings, geometry overlaps to such an extent that it's difficult to select one object without zooming very close to it. This Designer Elements program has made this easier with the Ambiguity Popup box. When you attempt to select one object among many objects close together, the popup menu appears.



As you move the pointer over an object name in the popup, the object it represents highlights in the selection color but is not yet selected. Click on the object name in the popup to accept that choice. If you move the pointer beyond the extents of the popup window, the window will automatically move, enabling you to see what is underneath the window.

The Ambiguity Popup box lists faces for such tools that require the selection of a face (draft, shell, fillet and extrude).

You can select only one object at a time in the popup box.

You have two options for closing the Ambiguity box without selecting an object:

- Click the Close button or box in the title bar
- Click outside the popup box

Selecting All Objects

There are two ways to select everything in the your Designer Elements program document.

- Select the **Selection** tool and choose ***Edit>Select All*** (CTRL+A (Windows) or z +A (Macintosh).
- Double-click the **Selection** tool.

Copying with the Selection Tool

The **Selection** tool can be used to copy wireframe, surface and solid objects. Be careful when copying surfaces and solids. By holding down the CTRL (Windows) or OPTION (Macintosh) key and dragging a surface or solid you create an instance of the original object. All changes made to the original object are automatically reflected in the instance (Cobalt™ and Xenon™ only). If you do not want to create an instance, select the original object, choose ***Edit>Copy*** and then ***Edit>Paste***.

Selecting Points

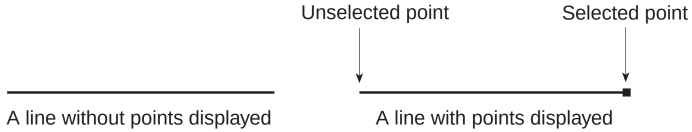
Selecting points differs from selecting objects because points are not always visible.

Being able to select points is useful in two situations: stretching selected geometry (described later in this chapter) and control point selection for transformations.

Show/Hide Points Command

This command in the Edit menu toggles the display of the *control points* (*endpoints*, *midpoints*, *center* points and *vertex* points) for selected objects. When points are displayed, you can select a point by clicking it. If points are not displayed, you can select a point by dragging a selection fence around the location of the point.

To show points for an individual selection, choose ***Window>Edit Objects*** and change the control points setting from Hidden to Visible.



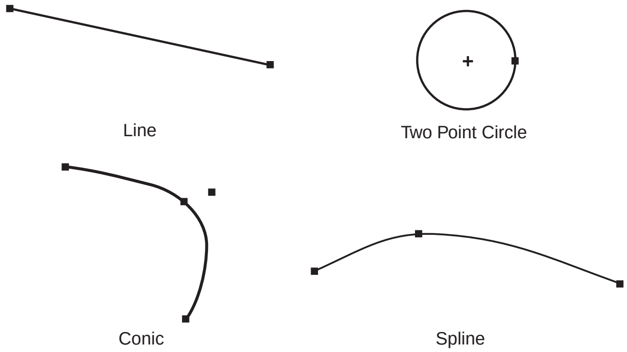
If you select and drag a line, the line and the endpoints move. If you select and drag only an endpoint of the line, the endpoint moves and the line length changes while the other endpoint of the line remains fixed.

The appearance of a selected point is not affected by the zoom scale or the line width of your geometry.

Displaying Points

1. Select the geometry.
2. Choose ***Edit>Show/ Hide Points*** to toggle the display of points on and off.

To turn off the point display once the points of an object are showing, you must select the object again and choose either ***Edit> Hide Points*** or the *Control Points* option in the Edit Objects dialog box. The graphic illustrates some curves with their associated points.



Showing and Hiding Points with Stroke

When you hold down the SHIFT+CTRL keys (Windows) or the z key (Macintosh) and click an object, the display of the object's points toggles on or off. If the points are hidden when you click the object, the points display.

Selecting Points

You can select a control point whether the points are visible or not. To select points:

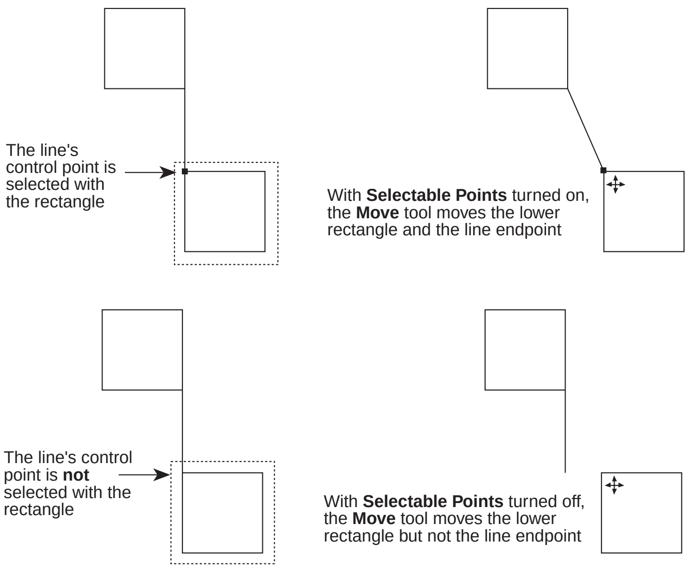
1. Chose the **Selection** tool.
2. Drag a selection fence around the location of the point.

   The selected point displays as a square.

Selectable Points Command

This command in the Edit menu gives you the ability to select points that aren't displayed. When *Selectable Points* is not set, you cannot select points by dragging a fence.

If points are displayed, click the point to select it. If points are not displayed, you can select a point by dragging a selection fence around it. The following example illustrates the use of *Selectable Points*.



Selecting Objects using Other Tools

With the editing tools, you will be asked to select the objects for performing an operation. As was mentioned earlier in the chapter, you select the tool first and then the objects.

When you select the tool and move the pointer into the drawing area, it becomes a hollow selection arrow.

When you finish selecting, the pointer becomes a general tool cursor.

This outline arrow must not be confused with the **Selection** tool.

Selecting Objects with Commands

Your Designer Elements program provides you with three selection commands, *Select All*, *Selection Mask* and *Select Chain*.

Select All - CTRL+A (Windows); z +A (Macintosh)

When the **Selection** tool is chosen, this command in the Edit menu selects all objects except those on a hidden layer or excluded by the Selection Mask. You can also double-click the **Selection** tool to select all objects.

Select Mask Command

This command in the Window menu allows you to limit selection by object type, layer and color. You can only select objects that are highlighted in the dialog box. The object type list includes such things as points, lines, splines, mesh, surfaces, solids, text, dimensions, images, symbols, groups, lights, decals, draw view, cross hatch, etc.

For example, if circles are not highlighted, when you choose *Select All*, everything but the circles is selected. In this way, you can select such combinations as only blue splines or only red objects on a particular layer.

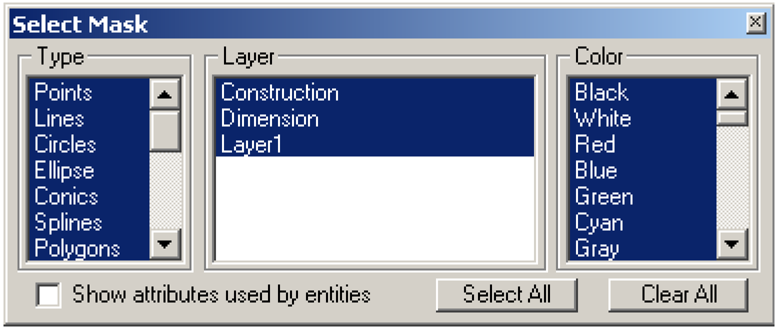
Tech Note:

Vellum 3D Users: In Vellum 3D when you closed the Selection Mask, the setting remained in effect until you selected another creation tool.

Using the Selection Mask

1. Choose ***Window>Select Mask***.

   The Selection Mask dialog box appears.



   The highlighted items respond to all selection methods and can be detected by the Drafting Assistant.

2. Click the items you want to be able to select so they are highlighted.

While the dialog box is visible, you can select, create and edit geometry. You can move the dialog box if it covers geometry you want to work on. The Selection Mask resets itself to the default setting of *Select All* after you close the box.

When an item in the dialog box is not highlighted, the Drafting Assistant and all the tools cannot detect it, even though it is visible to you on the screen.

The Selection Mask is useful when you have created a complex drawing and want to change particular groups of objects. You can use this feature to export some but not all geometry.

If layers are added when the Selection Mask dialog box is displayed, the Selection Mask dialog box automatically updates.

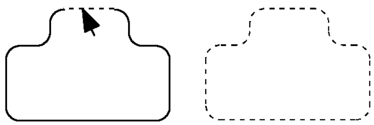
Selecting or Deselecting Listed Items

- To select one item - *Windows*: With all items selected, click on the item and the rest of the list will be deselected. *Macintosh*: Press the Clear All button and click on the desired item.
- To deselect a list quickly, press the Clear All button.
- To select a contiguous group of items, click on the item at the top or bottom of the desired group list and drag up or down to select the other items in the group.
- To select or deselect non-contiguous items, hold down the CTRL (Windows) or the z key (Macintosh) and click on the items.

Select Chain Command

This command selects curves that are connected to the selected object. To use the *Select Chain* command:

1. Select the beginning of a curve as shown by the arrow in the left graphic below.
2. Choose ***Edit>Select Chain***.
3. All objects connected to the start or end are selected, as in the right graphic below.



Deselecting

To deselect an object, click anywhere in the drawing area where there is no object, or click any of the creation tools in the tool palette.

You can deselect an object that was selected in a multiple selection operation. While the objects are still selected, hold down the SHIFT key and click the objects you want deselected.

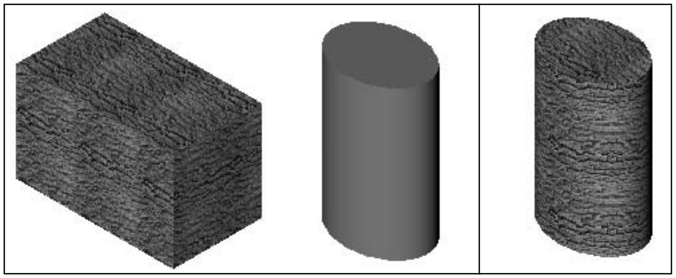
Eye Dropper Tool

This tool allows you to copy object characteristics such as line font, color, pattern, arrow at start and end and render materials from one object to another.

You can specify which object characteristic you want to copy through the Eye Dropper Filter dialog box. To display this box, press the CTRL (Windows) or OPTION (Macintosh) key.

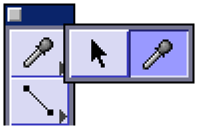
The listed characteristics include: line font, color, layer and render material.

This tool does not affect the resolution, dimensions, iso lines or the size and shape of the object. In the graphic below, the left block characteristics were applied the cylinder with no material applied to it (middle graphic). The right graphic shows the cylinder with the same render material as the block after using this tool.



Using the Eye Dropper Tool

1. Select the **Eye Dropper** tool from the **Selection** tools palette. The Message Line reads: *Eye Dropper: Select objects to modify [Shift=Extend].*



2. Select the objects whose characteristics you want to change.

   The Message Line now reads, *Eye Dropper: Select object to copy from [Shift=Extend, Ctrl* (Windows) or *Option* (Macintosh) = *Filter].*

3. If you do not want all object characteristics copied, press the CTRL (Windows) or OPTION (Macintosh) key to display the Eye Dropper Filter dialog box. Otherwise, skip to next step 5.
4. Deselect those items in the dialog box that you do not want copied.
5. Select the object whose characteristics you want to copy.

The object's characteristics change to those of the referenced object.

You can also select the object before you select the tool. If you do this, you only select the reference entity and the object automatically changes. If the change involves applying a material, render your drawing to confirm the change.

There are no Status Line entries.

Pen Settings

This chapter describes various options for adapting your Designer Elements program to your needs.

Default versus Selected Object Settings

When no object is selected, any changes made to the settings become the default for all open files. When an object is selected, any change made will only affect the object.

The following topics are covered:

- Style
- Color
- Weight
- Pattern

- Arrowheads

You have three options for setting pen characteristics, pen color, weight and pattern.

You can change the pen characteristics for selected objects or choose one or more of these characteristics as the default setting for your file.

Changing the Pen Characteristics of an Object

1. Select the object.
2. Choose ***Pen>Color***, Weight or Pattern and select the one you want for the selected object.

   This change only affects the selected objects. It does not affect the default setting for future objects created.

Changing the Default Pen Characteristics

If you want to change a pen characteristic so that all future objects will have that characteristic, choose the characteristic without having any object selected.

Style

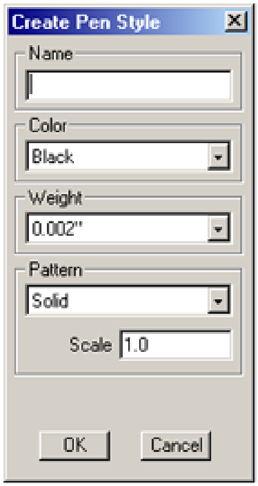
The ***Pen>Style*** menu contains commands for creating and modifying pen styles. A pen style is a collection of pen attributes such as color, weight and pattern.

Pen Styles are associative. This means that after you create and assign pen styles to a drawing you can go back and modify the pen style and have all objects using that pen style update.

The first three menu option provide tools to create, delete and modify user defined pen styles. The next nine pen styles are hard coded into the program and therefore not modifiable.

Creating a new Pen Style

To create a new pen style choose ***Pen>Style>New*** from the drop down menu. Specify the name, color, weight, pattern and scale, clicking "Ok" creates the new style. All pen styles that you create are listed below the non modifiable pen styles.

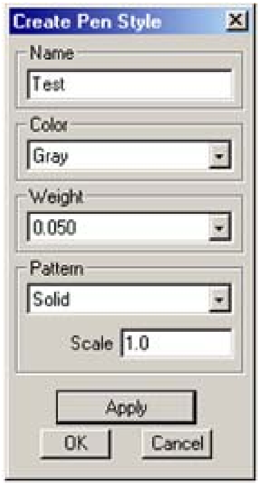


Deleting a Pen Style

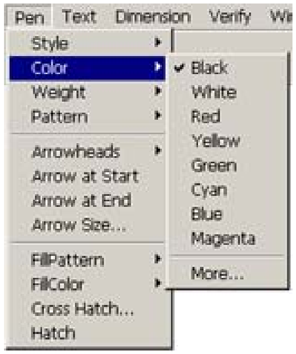
To delete a pen style choose ***Pen>Style>Delete*** from the drop down menu and choose one of the styles you created. The non modifiable pen styles cannot be deleted.

Modifying a Pen Style

To modify a pen style choose ***Pen>Style>Modify*** from the drop down menu. This provides you the means to change one of the pen styles you created. Hit "Apply" for your changes to take effect.



Color



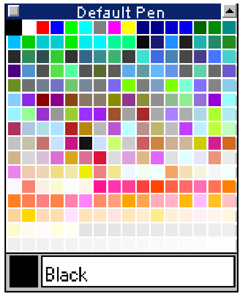
There are 256 color options including 239 user definable colors and 17 predefined colors. Each definable color can be assigned independently from the 16.7 million colors available.

The color submenu displays eight of the defined colors by name, including black, white, red, yellow, green, cyan, blue and magenta. The submenu also includes the option, More which displays a color palette with more color options.

Tip:

The fill color for polygons is set in the Pen Color. See Chapter 28.

Displaying the Color Palette



1. Choose ***Pen>Color>More***.

   The Color palette appears.

2. You can choose to display 8, 16, 144 or the full 256 color palette. The default color palette displays 256 colors. Click the button in the upper right corner of the title bar.

   You can cycle through the number of colors displayed by continuing to click the same button.

Specifying the Color of the Current Pen

1. Choose ***Pen>Color***. The submenu appears.
2. Choose the color you want.

   The pen takes on the new color, selected in the submenu.

This Color command specifies the color of the current pen and any other selected geometry without changing any other pen characteristics.

Defining Colors

You can define as many as 239 colors. When you define a color, you are defining it both for the file and for your Designer Elements program.

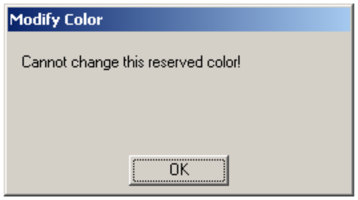
Because the color displays are different for Windows and Macintosh machines, the process for defining a color varies slightly.

Windows:

1. To define a new color, choose ***Pen>Color>More***. The color palette displays.

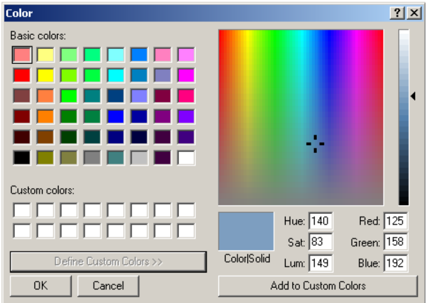
2. Double-click on one of the definable colors.

   The first 17 are not definable. If you attempt to redefine one of those you receive the following error message.



   If the color is definable, the following color display appears.

3. Click the Define Custom Colors button to display the full color display.



The color display contains the following elements:

Basic Colors	This section displays the 64 default colors available in the palette.
Custom Colors	You can display 16 more user-defined colors in this area for a particular file. These will be saved with the particular file but will not save as a default setting.
Add to Custom Colors	Use this button to place a color in the Custom colors section.
Color Palette	Select your color by using this palette.
Color Palette Solid	Once you select a color, it appears in this swatch.
Hue, Saturation, Luminance	These fields automatically fill in when you select a color. You can also enter values into these fields.
Define Custom Colors	Click on this button when you see a partial color display to show the full color display.

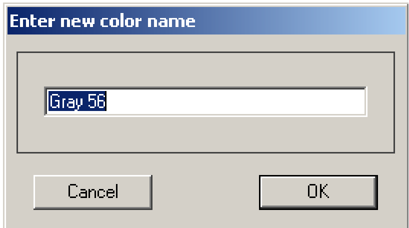
Red, Green, Blue	These fields automatically fill in when you select a color. You can also enter values into these fields.
Cancel, OK	Click Cancel to close the color display without saving the color selection. Click OK to accept the color selection.

4. In the color palette on the right, place the pointer within the target cursor and drag to the color desired. The color displays in the Color/Solid area below the palette.

   You can also adjust the Hue, Saturation, Luminance, and the percentage of Red, Green and Blue by entering your own values. The % of black can be adjusted by moving the cursor over the arrow in the scroll bar to the right of the color palette and drag up or down to the desired location. All the values in the Luminance, Red, Green and Blue data fields change as well. Hue and Saturation do not change.

5. Once you have chosen the color, click OK, the color display closes and you are returned to color palette.

6. Double-click on the color name field, where the old name currently displays. The following dialog box appears.



7. Enter the new name for the color and click OK.

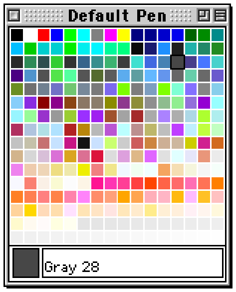
   You are returned to the color palette with the new color name shown.

8. Repeat this process to define up to 239 colors.

These colors save with the file and the program for future use.

Macintosh:

1. To define a new color, choose ***Pen>Color>More***. The color palette displays.

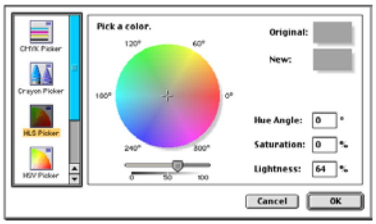


2. Double-click on one of the definable colors.

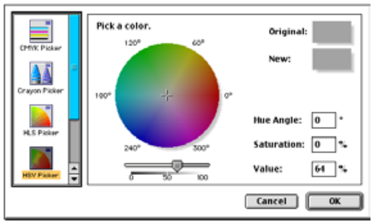
The first 17 are not definable. If you attempt to redefine one of those you receive the following error message.

If you click a definable color, one of six color displays appears: HLS Picker, HSV Picker, CYMK Picker, Crayon Picker, HTML Picker or the RGB Picker. You have a choice for the default display.

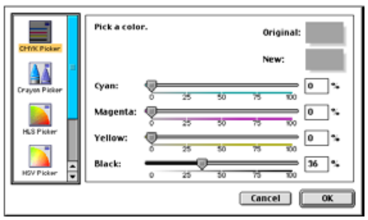
HLS Picker



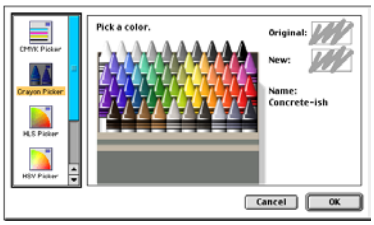
HSV Picker



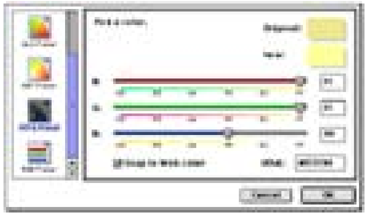
CYMK Picker



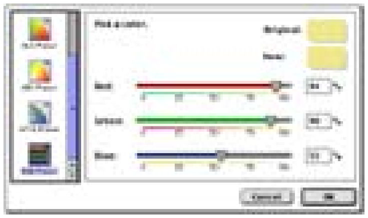
Crayon Picker



HTML Picker



RGB Picker



The *HLS Picker, HSV Picker, CYMK Picker, Crayon Picker, HTML Picker and RGB Picker* contain the following elements:

Color Wheel (HLS & HSV) Contains the color spectrum from which you choose a color. Click or drag within the wheel to select the color. The color values appear in the Hue and Saturation data fields.

Lightness Slide Bar (HLS & HSV) Allows you to set the lightness for a particular color. Place the cursor over the slide and drag to the desired lightness. The lightness value appears in the data field.

Hue Angle, Saturation Lightness fields (HLS & HSV) Values can be entered into these fields to select a color.

Color Slide Bar (RGB) There are three bars; Red, Green and Blue. Place the cursor over the slide and drag it to adjust the color value for each bar. Values are entered in the data field.

Red, Green and Blue (RGB) Values can be entered into these fields to select a color.

Name (Crayon) The color name displays when you choose a crayon color.

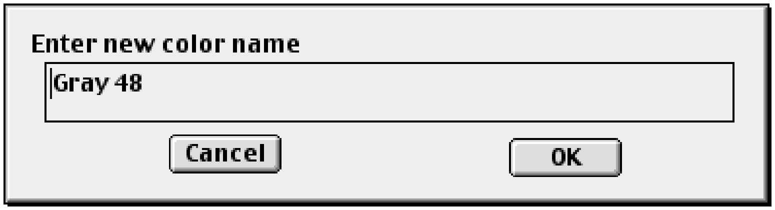
Crayon Box (Crayon) Displays the colors available for selecting.

More Choices/Fewer Choices This toggles between the two options. Choosing More Choices displays two icons for the HLS Picker, HSV Picker, CYMK Picker or Crayon Picker displays. Choosing Fewer Options turns off the icon display.

Original/New fields Original displays the current color setting for the color you are defining. When you define a new color, the swatch is gray. New displays the new color you have selected.

Cancel, OK — Click Cancel to close the color display without saving the color selection. Click OK to close the dialog box and save the new settings.

3. Choose a color from the color display.
4. Once you have chosen the color, click OK, the color display closes and you are returned to the color palette.
5. Double-click on the color name field, where the old name currently displays. The following dialog box appears.



6. Enter the new name for the color and click OK.

   You are returned to the color palette with the new color name shown.

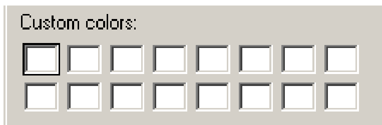
7. Repeat this process to define up to 239 colors.

These colors save with the file and the program for future use.

Add to Custom Colors (Windows only)

This feature is standard for the Windows platform and allows the user to save 16 user-defined colors for a particular file and display in the Custom colors section of the Color display when you first double-click on a definable color.

1. Repeat steps 1 and 2 in the previous section, “Defining Colors,” to show the partial color display.
2. Place the pointer over one of the available boxes in the Custom colors area and click. A dotted line appears around that box to show that it is selected.



3. Click Define Custom Colors and the full color display appears.

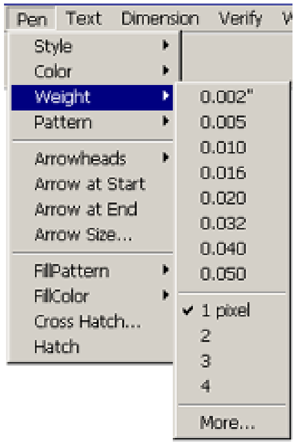
   Note in the display that the Color/Solid area is black and the target icon in the palette above it is at the bottom edge.

4. Move the pointer to the color palette and place it within the target area. Drag this to the color you desire. You can also adjust the % of black in the color by dragging the arrow, in the scroll bar to the right, to the desired location.

5. Click Add to Custom Colors. The color appears in the Custom colors patch.
6. To fill in the others boxes, go through the same process.

Weight

This command in the Pen menu sets the pen width of the current pen pattern and selected lines without changing any other pen characteristics.



You also have the option of setting a line weight based on units, pixel thickness and through the More option by the point size or model space.

Units

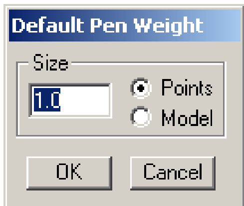
The line weights shown depend on the units selected in the Preferences dialog box. If you have selected metric units, the line weight shows in millimeters instead of inches.

Pixels

The pixel weight depends on the screen resolution and so remains constant from zoom to zoom.

More

When you choose More, the Pen Weight dialog box appears.



By selecting the Points, you can enter a value in the size field of the point size you want for the current pen weight. By selecting the Model, you can enter a thickness value based on the units of your model set in the Preferences dialog box. A one inch value entered in the field, prints one inch thick, for a drawing scale of 1:1. After you choose your setting, click OK to accept the pen weight or Cancel to close the More dialog box.

Note: Any line thickness of less than .016 inch appears one pixel wide on the screen. When you print or plot such lines, you can see the different weights.

Specifying a New Weight for the Current Pen

1. Choose ***Pen>Weight***. The submenu appears.
2. Drag to the weight you want.

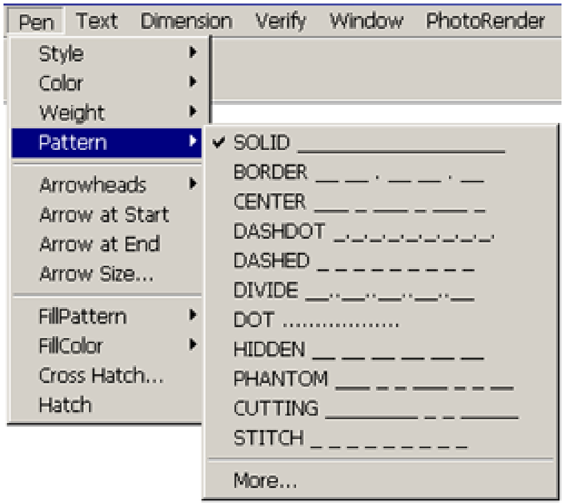
   The pen takes on the new weight, selected in the submenu.

Pattern

The pen pattern determines the appearance of lines on the screen and during plotting. Any line thickness of less than .016 inch appears one pixel wide on the screen. When you print or plot such lines, you can see the different weights.

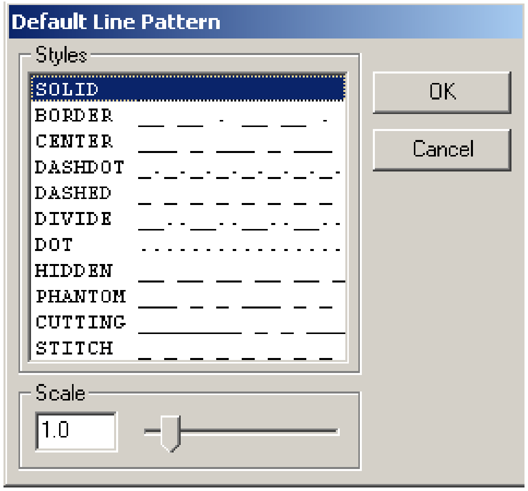
The default pen pattern is Solid. The default color for all patterns is black. The default weight for all patterns is 1 pixel.

There are nine default pen patterns in the Pattern submenu of the Pen menu.



You can also create your own pen patterns and access them through the Line Pattern dialog box by choosing ***Pen>Pattern>More***.

Line Pattern Manager



The Line Pattern manager appears when you choose ***Pen>Pattern> More***. If nothing is selected the dialog box is called Default Line Pattern and the pattern becomes the default pattern for the drawing. If an object is selected the dialog box is called Object Line Pattern and the pattern selected only applies to the selected object. Both dialog boxes display the pen patterns that are available to you for creating and editing geometry. You can perform some basic editing on these patterns using the options provided.

The Line Pattern manager contains the following items:

Patterns This section contains all the patterns available.

Scale This field contains the value by which the pattern dashes and spaces are multiplied to obtain the same pattern at a different size. You can either enter a specific value or use the slide to set the scale. The graphic below is an example of the phantom line with scale values of 1, 2 and 3.

(Scale 1) ___ _ _ ___ _ _ ___ _ _ ___ _ _ ___ _ _

(Scale 2) ______ __ __ ______ __ __ ______ __ __

(Scale 3) ____________ ____ ____ ____________

New Patterns

Each line pattern displays according to two description lines of the Cadd.lin file in the Environ folder.

```
;;
;;      Version 1.0 - CSI-CADD Linetype file
;;
;;      Date    6/12/95
;;
;;      Line font descripters using the AutoCAD ACAD.LIN format
;;
;;
BORDER,__ __ . __ __ . __ __ . __ __ . __ __ . __ __ . __ __
A,.25,-.125,.25,-.125,0,-.125
CENTER,___ _ ___ _ ___ _ ___ _ ___ _ ___ _ ___ _ ___ _ ___ _ ___ _ _
A,.75,-.125,.125,-.125
```

You can add new patterns to your Designer Elements program by opening the file in an ASCII text editor and creating two new description lines per pattern.

The first line contains the pattern line name plus a descriptor string. The second line contains the length attribute for dashes, spaces and dots. A zero (0) value indicates a dot and a negative value indicates a blank space.

Creating a New Pattern

1. Quit your Designer Elements program, if it is running.
2. Open the Cadd.lin file from the Environ folder in an ASCII text editor.
3. After the last line pattern, enter the name of your new pattern. Give it a name not already used.
4. In the same line, enter a descriptor string. This string is simply a visual representation of the pattern that will display in the Line Pattern Manager next to the name. (See the other patterns for an example). Use the periods, dashes and the space key on your keyboard to create the descriptor.
5. Begin the second line with a capital “A” followed by the length of the attributes. Each attribute should be followed by a comma and contain no spaces.
6. Save the file.
7. Restart your Designer Elements program.

Editing Patterns

You can edit any pattern in two ways:

- Make changes to the Cadd.lin file in an ASCII editor.

Tip:

If you want to make changes to the default patterns, copy the original file to another location or rename it and keep it in the same location before making changes. Then, if you want to return to the original factory settings, you simply have to replace the Cadd.lin file with this

- Choose settings in the Line Pattern Manager.

Making Changes in the Cadd.lin file

1. Quit your Designer Elements program, if it is running.
2. Open the Cadd.lin file, from the Environ folder, in an ASCII text editor.
3. Make the desired changes to the patterns. If you have changed a default pattern, name or descriptor, it will be reflected in the Pattern submenu.
4. Save the file.
5. Restart your Designer Elements program.

Choosing Settings in the Line Pattern Manager

1. Choose ***Pen>Pattern>More***. The Line Pattern Manager displays.
2. Select the pattern you want to modify.
3. Enter a value in the Scale field.
4. Click OK to accept the changes or Cancel to close the Line Pattern Manager. All selected will reflect the changes.

   If any objects are selected, they will reflect the changes. The current pen pattern will not be affected unless it's the pattern you changed.

   If no objects are selected, clicking OK will change the current pen pattern to the edited pattern.

   Important: Changes made here affect all open files.

Specifying a New Pattern for the Current Pen

1. Choose ***Pen>Pattern***. The submenu appears.
2. Drag to the pattern you want. The pen takes on the new pattern, selected in the submenu.

Specifying a New Pattern for a Selected Object

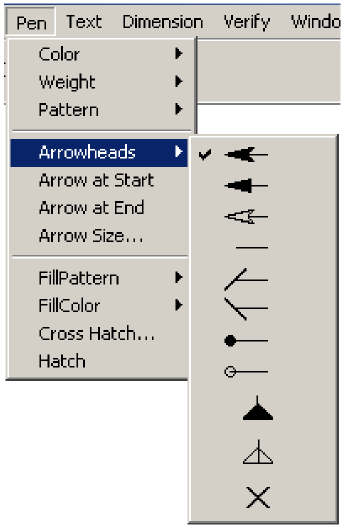
1. Select the object.
2. Choose ***Pen>Pattern***.
3. Drag to the pattern you want. The pen takes on the new pattern, selected in the submenu.

Construction Line Pattern

A construction line uses a dotted line pattern. It cannot be modified and is not affected by changes to the dot pattern. The default color is magenta. If you want to create construction geometry on the construction layer, you can use the dotted line pattern.

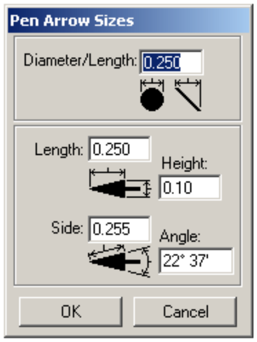
Arrowheads

When you want to use arrowheads on lines or circular arcs that are not a part of dimensions, specify the placement of arrowheads in the Pen menu. You can have an arrowhead at the beginning or end of a line or circular arc, or at both the beginning and end.



Arrowheads Type

The Arrowheads submenu allows you to choose one of eight arrowheads for arrow lines. Select the style that you prefer.



Arrow Size Command

This command in the Dimension menu lets you specify the size of the arrowhead that you have selected.

Diameter/Length — This value is the diameter or length of circular and length of all slash or standard arrow styles. The value here affects all Length, Side and Angle fields. The value displays in the current units specified in the Units page of the Preferences dialog box.

Length — This value is the length of the arrowhead as the horizontal distance from its tip to the furthest extension of its base.

Height — This value is the height of the arrowhead as the vertical distance of its base.

Side — This value is the length of the edge of the arrowhead.

Angle — This value is the angle of the tip of the arrowhead.

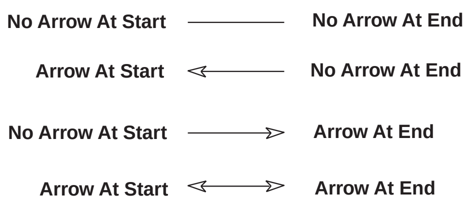
If you change any value in the Length, Height, Side, or Angle entry fields, this Designer Elements program changes the values in the other entry fields accordingly.

Arrow At Start

This command in the Pen menu places an arrowhead at the beginning of selected and subsequent lines and circular arcs. You can choose the type of arrowhead from the Pen menu. A check indicates the current arrowhead setting. You can change the default setting by saving changes in the preferences file.

Arrow At End

This command in the Pen menu places an arrowhead at the end of selected and subsequent lines and circular arcs. You can choose the type of arrowhead from the Pen menu. A check indicates the current arrowhead setting. You can change the default setting by saving changes in the preferences file.



The start and end of a line and an arc are determined by the point that was created first.

Preference Settings

All designers develop a particular style when creating their models and parts. This style includes specific standards that unify their work and may include such things as measurement units, line color, drawing layout, short cut keys and more. It also unifies work within companies and industries. Preferences set in the Preferences dialog box relating to object display and creation affect only the entities created after the preference is set.

Default versus Selected Object Settings

When no object is selected, any setting changes made to *Selectable Points*, *Grid*, *Axis*, *Triad*, *Show Points*, *Construction Lines*, *User-defined* plane, *Pen*, *Text*, *Dimension*, tool palettes and *Render* become the default for all open files and the current Designer Elements program session. When an object is selected, any change made will only affect the object.

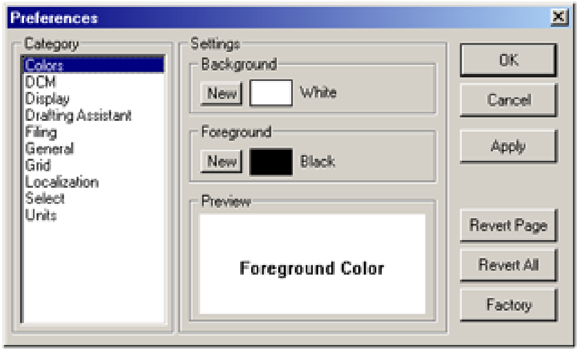
This chapter covers the following topics:

- Preferences
- Short Cuts

Preferences

Your Designer Elements program allows you to save preferences for a particular session. However our program do not support saving preferences with a specific file.

When you have more than one file open during a particular Designer Elements program session, menu settings like pen pattern, *Hide/Show Axis* and the status and location of palettes are the same for all open files. Commands dealing with the view orientation and work plane are file specific.



To save preferences, choose ***File>Preferences*** to display the dialog box. Preferences are saved in *prefs.ini*.

The Preferences dialog box contains a *Category* list of Preference groups, the *Settings* section and a series of operation buttons. The buttons include:

OK	Saves preference settings you have specified in this session and closes the Preferences dialog box.
Cancel	Closes the dialog box without saving all changes.
Apply	Applies the change instantly.
Revert Page	Undoes changes made to the current preference group.
Revert All	Undoes changes made to any preference group.
Factory	Resets all preference groups to the factory settings.

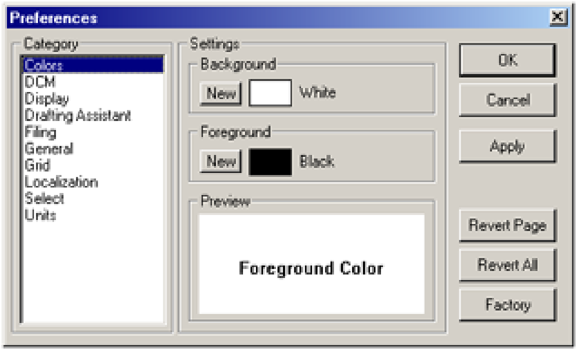
The *Category* list includes:

- Colors
- DCM
- Display
- Drafting Assistant
- Filing
- General
- Grid
- Localization

- Select
- Units

Selecting an item from the *Category* list displays its preference options in the *Settings* section.

Colors



Choosing the Color category displays the Color preferences page. This page controls the foreground and background color of the drawing area. The current (or proposed) settings are indicated by the color rectangle, color name and the *Preview* section. Press the appropriate New button to display the color selection dialog box and change the color.

Background

This setting allows you to set the background color for your drawing area. The current color is displayed in the window.

1. Click New. The color palette is displayed.
2. Choose the background color.
3. Click OK. The new background displays in the view window with the color name. For colors other than the standard colors, the color values display to the right of the view window.

Foreground

This setting allows you to set the foreground color for your drawing area (specifically the location indicator and the indicator separator lines). The current color is displayed in the window.

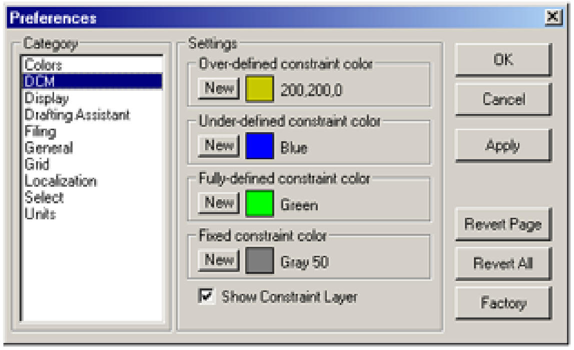
1. Click New. The color palette is displayed.
2. Choose the foreground color.

3. Click OK. The new foreground displays in the view window with the color name. For colors other than the standard colors, the color values display to the right of the view window.

Preview

The Preview window displays your background and foreground choices.

Dimensional Constraint Manager (DCM) (Cobalt™ Only)



This option controls how the Dimensional Constraints Manager gives feed back to the user.

Over-Defined Constraint Color

You have the ability to set the color of an over-defined sketch. Set this field to the desired color.

Under-Defined Constraint Color

You have the ability to set the color or an under-defined sketch. Set the field to the desired color.

Fully-Defined Constraint Color

You have the ability to set the color of a sketch that is fully defined. Set this field to the desired color.

Fixed Constraint Color

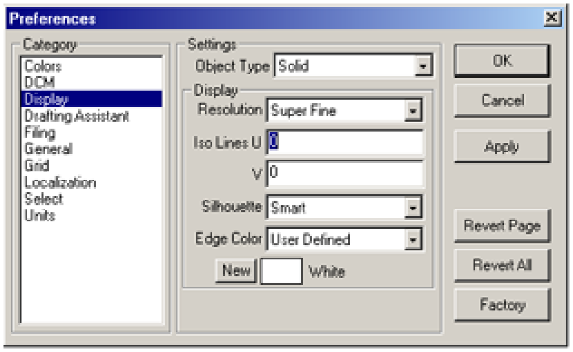
You have the ability to set the color of fixed constraints. Set this field to the desired color.

Show Constraint Layer

This check box tells the program whether or not to show constraints as they are being applied. If the box is not checked, constraints will be visible on the active work layer even though the constraints are being applied.

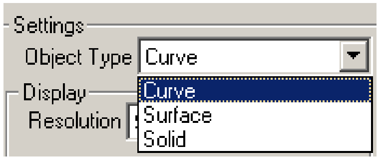
Display

This option controls the individual default display parameters for curve, surface and solid object types.



Object Type

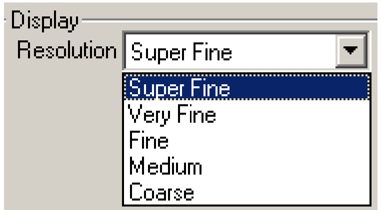
You have the ability to set the appearance of Curves, Surfaces and Solids. Each object type offers different display options.



Display

For each object type (curve surface and solid), you can choose a display option. Display options vary according to the object type and may include: *Resolution*, *Iso Lines*, *Silhouette* and *Edge Color*.

Resolution (Available for all object types) Controls how accurately an object's curves appear. You can set the curve resolution to Coarse, Medium, Fine, Very Fine and Super Fine. An object with a Coarse resolution draws quickly but may be visually less appealing. An object with a Fine resolution draws more slowly but may be visually more appealing.



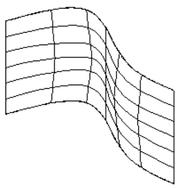
Tech Note:

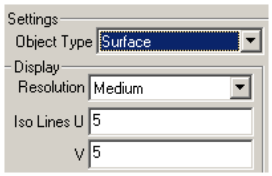
Parameter space is where objects are defined in a 2D coordinate system. Typically, a surface is mathematically defined in parameter space. Each surface has a mathematical function that maps 2D parameter space into 3D model space. A U/V coordinate of U=0.5 and V=0.25 in parameter space maps to X=100, Y=300, Z=255 in 3D model space.

Iso Lines

(Available for Surface and Solid object types) *Iso* (isopram) *Lines* control the number of U and V lines displayed for a surface or solid object. Iso Lines are constant parameter curves that lie on an object. U and V are letters used to define these lines (and their coordinates) in parameter space where U is for horizontal and V is for vertical. These are standard for the industry. A zero (0) in both fields turns off Iso Lines. U/V values may enhance the visual appearance of a surface or solid at the expense of drawing speed.

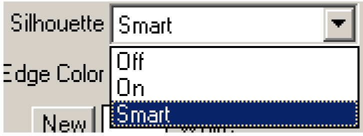
The left graphic below shows a surface with both U and V Iso lines set to five (5).





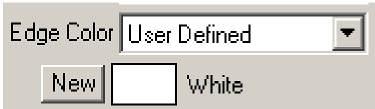
Silhouette

(Available for Surface and Solid object types) Controls the silhouette edge draw mode. There are three options, Off, On and Smart. Silhouette edges are view dependent and can cause a significant reduction in drawing speed. If the Smart mode is selected, silhouettes will be dynamically drawn based on performance considerations.



Edge Color

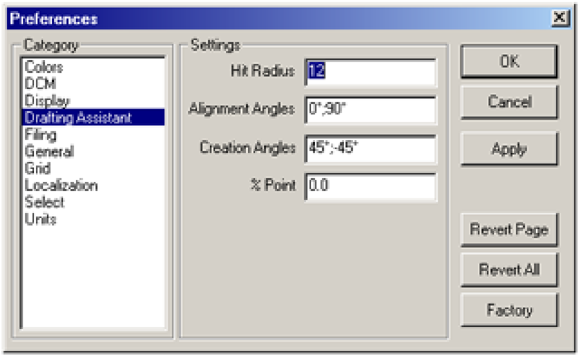
(Available for Surface and Solid object types) This option allows you to set the edge color of rendered mesh objects separate from the entity itself.



You can select one of four options from the pull-down menu, Foreground, Background, Entity and User Defined. To specify a User Defined color, click on New, choose a color in the palette and click OK. The new color is displayed in the Edge Color window with its RGB values.

Drafting Assistant

This category controls the low level snapping behavior of the Drafting Assistant. The settings include the *Hit Radius*, *Alignment Angles*, *Creation Angles* and *% Point*.



Hit Radius

Determines the detection distance in pixels. When the pointer is within the specified *Hit Radius*, the

Drafting Assistant notations display and the object is selected when you click the mouse.

Alignment Angles

Define angles for the Drafting Assistant's dynamic construction lines. If you want to change the orientation of your drawing, you can change these specifications. For example, in a 2D drawing, you could set these angles to 30°, 90° and 15° for isometric drawing. The defaults are 0° (horizontal) and 90° (vertical). Values should be separated by semicolons.

To display a dynamic construction line through a point, move the pointer to the point to activate it (a diamond appears); construction lines automatically display through the active point. You can have as many as eight active points. When you activate the ninth point, the first one in the series deactivates.

Creation Angles

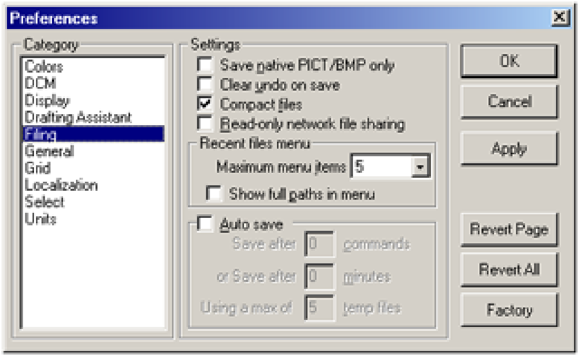
Define angles for Draft Assistant dynamic construction lines displayed when you are creating geometry. (These lines are not part of the list of lines generated from the eight active points.) The defaults are +45° and -45°. Values should be separated by semicolons.

% Point

Controls the Drafting Assistant's notations for divisions of a curve. For example, entering 25 instructs the Drafting Assistant to tell you when the pointer is 25% of the distance along a line.

Filing

This category controls the filing behavior of your Designer Elements program. Settings include *Save Native Picture Formats Only*, *Compact Files*, *Read-only Network File Sharing*, *Recent Files* and *Auto Save*.



Save Native Picture Formats Only

Activating this option allows you to disable multiple platform picture support. Saving only the native format will reduce file size.

Clear Undo on Save

Activating this option allows you to flush the undo stack after a file has been saved. If the box is not checked, the undo stack is maintained after saving, increasing the file size.

Compact Files

If this option is checked, files compact when saved. Display facets are not written out when this option is selected. This file size reduction is especially noticeable on files containing solid models with many creation parts.

Read-only Network File Sharing

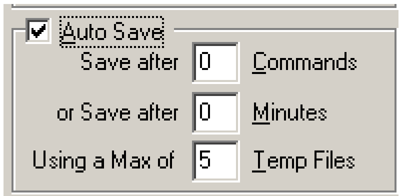
If this option is checked, the file can only be edited by the current user. No one else will be able to edit the file until the first person closes the file.

Recent Files

This option allows you to set the number of recent files that display in the File menu. Selecting a file from the menu immediately opens the file and bypasses the Open File dialog box.

Max Files in Menu	The drop down list allows you to specify 0 to 20 file names. The default number is five.
Show Paths in Menu	Checking this box displays the full path along with the recently used file name in the File menu.

Auto Save



Checking this box enables the automatic file saving options and directs your Designer Elements program to save a backup of your work periodically. If this box is not checked, automatic file saving does not occur.

If auto save is triggered, a backup file is created in the Backup folder within the Designer Elements program folder. Backup files are numbered sequentially. If you have not yet saved your file, the backup will be named untitled.

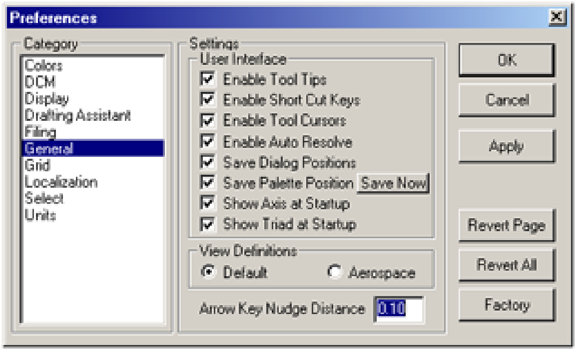
If auto save is on but you have not modified any geometry in your drawing since the period that the last auto save operation occurred, auto save does not activate.

There are three auto save options:

Save after "N" Commands	Entering a value in this field specifies the number of drawing modifications (creations, edits or deletions) that must occur before an auto save triggers.
Save after "M" Minutes	Entering a value in this field specifies the number of minutes that must pass after the first drawing modification before an auto save triggers.
Using a Max of Temp Files	Entering a value in this field specifies the number of backup files to be created before reusing a backup file name. A large value will consume more disk space. The default value is five.

General

This category controls the general user interface behavior, view definitions and arrow nudge distance.



User Interface

This section provides the following check boxes for choosing interface options:

Enable Tool Tips Enables the floating tool tip help windows.

Enable Short Cut Keys Enables setting the keyboard short cut keys through ***File>Short Cuts***.

Enable Tool Cursors Enables the display of tool specific icons when using the tool. Without this enabled, the cursor appears as crosshairs when moved into the drawing area.

Enable Auto Regen Enables the automatic regeneration of child objects when parent objects are modified (Cobalt™ and Xenon™ only).

Save Dialog Positions Saves the location of dialog boxes. The next time the dialog box displays, it is positioned at its most recent location.

Save Palette Positions Saves the tool palette positions and displays its status when exiting. The next time your Designer Elements program launches, the palettes display in their previous positions. Click the Save Now button to immediately save tool palette positions, pen

	color, fill color, dialog box locations and display status.
Show Axis at Startup	Shows the coordinate axis when your Designer Elements program launches.
Show Triad at Startup	Shows the coordinate triad when your Designer Elements program launches.

View Definitions

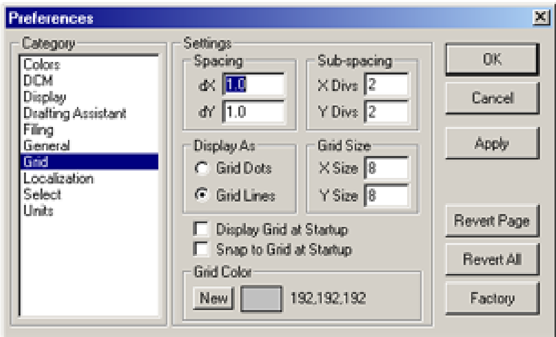
Your Designer Elements program supports two different common view definitions, *Default* and *Aerospace*. Select the *Default* option to use view definitions commonly used for mechanical drafting. Select the *Aerospace* option to use view definitions commonly used for aerospace lofting.

Arrow Key Nudge Distance

The *Arrow Key Nudge* distance specifies how far the drawing will scroll when a keyboard arrow key is pressed in the **Selection** tool. The units for this option are based on the units chosen on the *Units* page of this dialog box.

Grid

This category controls the grid display and snapping behavior. The options include grid spacing, appearance, start up preference and color.



Spacing

The *DX* and *DY* values set the grid spacing. Units are based on those set on the *Units* page of the dialog box.

Sub-spacing

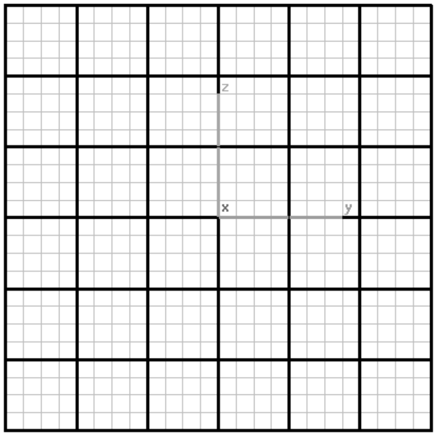
X Divs and *Y Divs* specify the number divisions for the grid to which geometry snaps in your drawing when the Snap to Grid command is activated.

Display As

The grid can be displayed as either dots or lines. Check either the *Grid Dots* or *Grid Lines* option.

Grid Size

Enter a value in these fields to set the grid size. The values represent the number of grid sections that appear in the positive and negative X and Y directions. A value of three entered into each field results in six grid sections in the X direction and six grid sections in the Y direction. The graphic here is an example of this.



The grid also follows the current work plane allowing you to use it when creating objects in any plane. For example, if you change the plane from the Top plane to a user-defined plane, the grid displays as it did in the Top plane.

Display Grid at Startup

Check this box to display the grid when you first launch your Designer Elements program.

Snap to Grid at Startup

Check this box to activate the snapping function when you first launch your Designer Elements program.

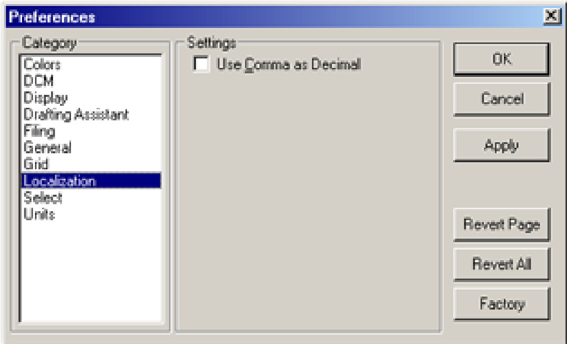
Grid Color

You can select any color for the grid. The current color displays in the window.

1. Click New to display the color palette.
2. Select a color from the palette.
3. Click OK to accept the color and return to the *Grid* page. The new color now displays in the grid view window with the color name. For colors other than the standard colors, the color values display to the right of the view window.

Localization

This category controls the usage of decimals versus commas in your Designer Elements program.

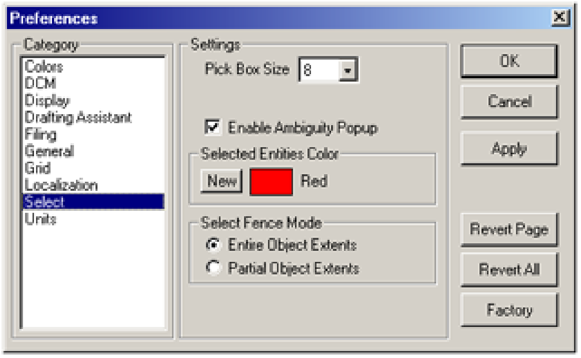


Use Commas as Decimal

Checking the *Use Comma as Decimal* option allows international users to display numbers according to their numerical standards.

Select

This category controls object selection behavior, including *Pick Box Size*, *Ambiguity Popup*, *Selection Color* and the *Selection Fence* mode.



Pick Box Size

When you select an object, you place the cursor on the object and click the mouse. The *Pick Box* is the area around the cursor in which an object must be located to be selected. You can specify the area using the pull-down menu (ranges from 2 through 16, even numbers only). The *Pick Box* does not display.

Ambiguity Popup

The Ambiguity Popup displays when there are multiple objects near the vicinity of your selection and offers choices of which object to choose.

A check mark in the box enables the popup. This is the default setting.

Selected Entities Color

This option allows you to set the selection color. The current color displays in the window. To change the color:

1. Click New to display the color palette.
2. Select a color from the palette.
3. Click OK to accept the color and be returned to the *Select* page. The new color now displays in the view window with the color name. For colors other than the standard colors, the color values display to the right of the view window.

Select Fence Mode

Your Designer Elements program supports two mode when dragging to select one or more objects, *Entire Object Extents* or *Partial Object Extents*. Selecting the *Entire Object Extents* option allows only the selection of objects that fall completely within the selection fence. Selecting the *Partial Object Extents* option allows the selection of any object that has a portion within the selection fence.

Tech Note:

The decimal places entered here only affect the Status Line and Location Indicator. They do not affect the decimal places used for dimensions. These must be set in the Dimension menu.

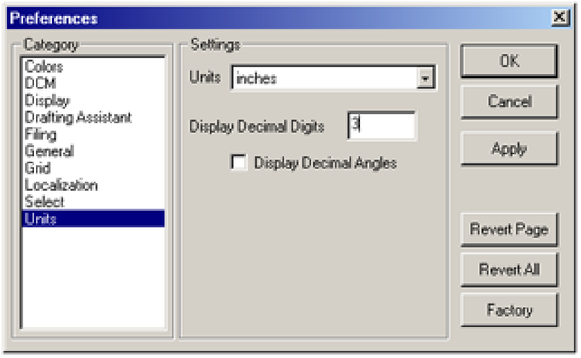
Tech Note:

If you choose feet/inches for units, you can enter the combination of feet and units in the Status Line and Edit Objects dialog box. However, these values will convert to decimals after they are applied to your geometry. For example: A value of 1”3” converts to 1.25 feet.

Note: Be aware that control points will affect what is selected when using the *Partial Objects Extents* option. For example: With this option selected, if your selection fence covers a control point for a circle, only the center point is selected rather than the entire circle.

Units

This category controls the units and the number of decimal points displayed for your geometry.



Units can be set to inches, feet, feet/inches, millimeters, centimeters and meters.

In the *Display Decimal Digits* data field, enter the number of decimal places (between 1 and 8) you want to display. Three decimal places is the default.

Changing the Preference Settings with the Preferences Command

1. Choose ***File>Preferences***.
2. Select the category you want to set.
3. Make your desired changes.
4. Select another category or click OK to close the dialog box.

Changing the Preference Settings Manually

1. Quit your Designer Elements program.
2. Open the Preferences file, *prefs.ini*, in a text editor. The file is located in the Environ folder - the same folder as the executable file for your Designer Elements program.

3. Change the characteristics you want.
4. Save and close the file.

   The file must be stored in the original folder.
5. Relaunch your Designer Elements program.

The preferences are set.

Preferences and Object Creation

When you change any of the preference settings, the changes are only reflected for new objects.

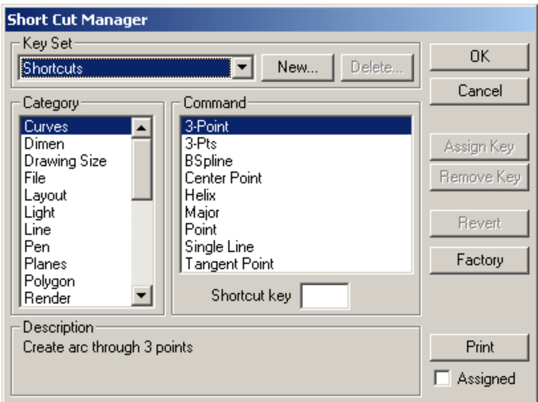
Pen and Dimension Preferences

When you launch your Designer Elements program, you can set the default pen, text and dimension characteristics. Without anything selected, specify the settings. You do not have to save any file to save these preferences. Simply, exit the program and the pen, text and dimension settings are saved to the preferences *.ini* file.

Short Cuts

The *Short Cuts* command in your Designer Elements program allows you to create keyboard combinations, providing alternative ways to invoke commands. You can create short cuts for activating tools and commands, setting the work plane, changing layers, switching between wireframe and render modes and more. If you assign a short cut to a tool, the short cut displays with the tool tip when you move the pointer over the tool.

Short Cuts are organized by the *Short Cut Manager*.

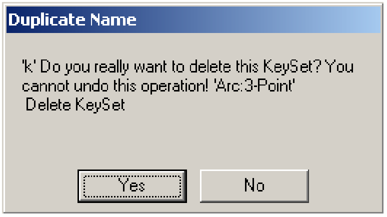


The Short Cut Manager dialog box includes the following elements:

Category	Contains all available operations in your Designer Elements program.
Command	Displays the commands assigned to the selected operation in the category list.
Shortcut key	Displays the key combination for the selected operation. If you want to use the function keys, type **F** and the number (F3 for example) rather than pressing the function key itself.
Description	Describes the action that results from the selected command.

The buttons in the Short Cut Manager include:

OK	Saves the new Short Cuts and closes the Short Cut Manager.
Cancel	Closes the Short Cut Manager without saving any changes.
Print	Prints a hard copy of the current short cut key assignments. The factory default Short Cuts are included in Appendix E.
Assigned	If this box is checked, when you print the Short Cut list, only assigned keys are included.
Assign Key	Assigns the Short Cut key combination for future use. If the short cut key is already assigned, a warning box displays asking if you want to remove the command assignment already using that key combination.



Click Yes to accept the reassignment or No to be returned to the Short Cut Manager to enter another key combination.

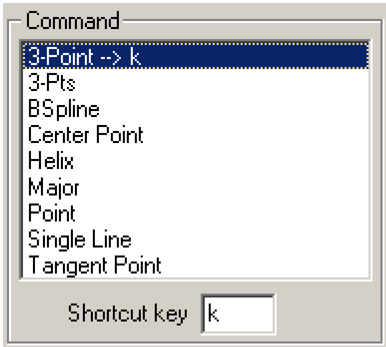
Remove Key Removes the key combination from the selected command.

Revert Undoes any changes made to any groups.

Factory Resets all groups to the factory settings.

Creating a New Short Cut

1. Choose ***File>Short Cuts***.
2. Select the desired category and command in the appropriate sections.
3. Enter the key combination into the *Shortcut key* data field.
4. Click Assign Key. The key combination displays next to the selected command.
5. Continue entering all key combinations as desired.
6. Click OK to save the new short cut keys and close the dialog box.



Drawing Techniques

This chapter provides techniques for creating geometry and setting up your drawing area. The following topics are covered:

- Object Creation Methods
- Status Line
- Message Line
- Coordinate System
- Trackball
- Drawing Display Commands
- Drawing at Full Scale
- Default versus Selected Object Settings
- Escape Key
- Progress Bar
- Right Mouse Button

Object Creation Methods

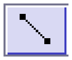
In Vellum 3D and Vellum Draft, you had two methods for creating objects, clicking and dragging. In your Desiger Elements program, objects are created by clicking only.

Tech Note:

Rubberband images only appear when geometry is displayed in the Wireframe rendering mode.

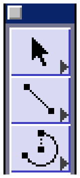
As you create geometry, a rubberband image of the geometry appears showing you how the object will look when you click the last point.

To make object creation easier, your Desiger Elements program's wireframe tools contain a feature we call Smart Pointers. As you work with a tool, the cursor icon shows you the points you must indicate to create an object with the tool.



Each dot on the icon represents a point you must place by clicking. The smart pointer indicates the order for designating points.

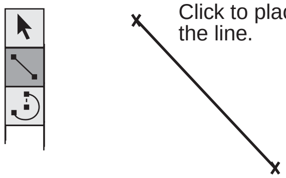
Creating a Line by Clicking



1.Click the **Single Line** tool in the main tool palette. The Message Line reads: *Single Line: Pick beginning point [Ctrl = Copy Previous* (Windows) or *Option = Copy Previous* (Macintosh)*]*.

2.Move the pointer to the drawing area and click to set the starting point of the line.

3. Move the mouse to the desired location for the endpoint of the line. As you do so, a rubberband image of the line appears.



4. Click to set the endpoint.

Immediately after construction, you can make changes in the Status Line at the bottom of the drawing area to alter the length or position of the line. If the geometry is not satisfactory, just press the BACKSPACE (Windows) or the DELETE (Macintosh) key

Status Line

Whenever you select a tool from a tool palette, the Status Line appears along the bottom of the drawing area. It contains data fields that provide information about the current construction. For example, when the **Single Line** or **Connected Lines** tool is active, the Status Line contains data fields for the X, Y and Z coordinates of the beginning point and the change of X, Y and Z values. It also contains the value of the length and angle of the line.

You can use the Status Line in three ways:

- To create an object with keyboard entries only.
- To edit an object that was just created and is still selected.
- To create additional wireframe objects using the current tool.

Whenever you construct an object, the status data field containing the specification that you are most likely to change is active (the data field is highlighted). For example, when you draw a line with the **Single Line** tool, the Length data field is highlighted so you can enter a value for the length. Just type a value. When you press ENTER (Windows) or RETURN (Macintosh), the line is redrawn at the new length.

You can set the number of decimal places for Status data field entries to display by going to ***File>Preferences>Units***.

Using the Status Line

If you want to make an entry in a different status data field, you can use one of four selection methods:

- Press the TAB key to cycle the selection highlight through the status data fields from left to right.
- Click inside the status data field and the pointer becomes an *I-beam* text cursor.
- Double-click inside the status data field to select the entire contents of the data field.
- Click the data field label to select the entire contents of the data field.

Units and the Status Line

Values can be entered in inches, feet, feet and inches, millimeters, centimeters, meters and mathematical expressions (ex. 10”+2.54 cm).

When the units in Preferences are set to feet and inches, it’s important to be aware of the following rules:

- All numbers are assumed to be feet unless accompanied by the unit symbol, like " for inches. Entering a 1.5 in the field is read as 1.5 feet. If you want 1.5 inches, enter 1.5", 1.5i, 1.5in, 1.5 inch, etc.
- If you want to enter fractions of inches, each entry must include the unit symbol. For example, 5 feet 6 5/8 inches must be entered 5' 6 5/8". Internally this is converted as 5' + 6" + 5/8". If the inch symbol is not included with the fraction, 5/8 will be interpreted as a fraction of a foot.

Copying and Pasting Status Line Entries

You can copy and paste Status Line text for use in another data field.

For Windows, select the Status Line text, hold down the right mouse button and use the *Copy* and *Paste* commands available in the menu that appears. You cannot use the *Copy* and *Paste* commands in the Edit menu.

For Macintosh, use z +C to copy and z +V to paste text. You cannot use the *Copy* and *Paste* commands in the Edit menu. These function only for your Desiger Elements program data.

Tech Note:

There are two methods for making Status Line entries in Your Desiger Elements program.The first is Select a tool, create the object, type the new values and press ENTER (Windows) or RETURN (Macintosh).The selected geometry changes. In the second method, you select the tool, type the desired values in the Status Line and press ENTER (Windows) or RETURN (Macintosh).The values are now registered. Then you create the geometry with the tool.

Status Line Entries and Tool Operations

Information entered in the Status Line is registered in two different ways in your Desiger Elements program depending on what step you are on in creating your geometry.

- After you select a tool, you can immediately enter values in the Status Line data fields. When you perform the tool operation, those values will be used to create your object.
- After you create your object with a tool and the object is still selected, you can enter new values in the Status Line data fields. To move from field to field you use the TAB key. When you press ENTER (Windows) or RETURN (Macintosh), you object changes to reflect those values. Once you hit ENTER (Windows) or RETURN (Macintosh), future changes can only be made to the object through the Edit Objects dialog box.
- When entering values in the Status Line, the accuracy of your geometry depends on the number of decimal places you enter.
- The Status Line references the current coordinate system current work plane. All values in the Status Line are based on the current coordinate/work plane system.

Using the Status Line with Drawing Tools

Once you have drawn an object, you can adjust the specifications —the length, angle and location—in the Status Line. You can make these changes immediately before you construct another object, select a different tool or choose a command however it is not necessary.

You can make only one series of changes in the Status Line (in as many fields as necessary); after you press ENTER (Windows) or RETURN (Macintosh) your Desiger Elements program redraws the object to your specifications. After this, you must make subsequent changes using the *Edit Objects* command in the Edit menu.

Try the following exercises to create and change a single line with the Status Line.

Altering Geometry in Progress with Status Line

1. Select the **Single Line** tool. The Message Line reads: *Single Line: Pick beginning point [Ctrl = Copy Previous* (Windows) or *Option = Copy Previous* (Macintosh)*]*.
2. Click two locations in the drawing area. The length (L) field automatically highlights in the Status Line.
3. Type **3**. The 3 is entered directly in the L field.
4. Press the TAB key to select the next status field. The angle (A) field now highlights. (If your monitor does not display the Angle field, click on the scroll arrow at the right to display it and then press the Tab key to highlight the data field.)
5. Enter **15**.
6. Press ENTER (Windows) or RETURN (Macintosh).

   Pressing ENTER (Windows) or RETURN (Macintosh) completes the data entry for this object. The line is redrawn 3 units long and at a 15° angle.

Remember that when you press ENTER (Windows) or RETURN (Macintosh), your Desiger Elements program constructs the object based on the specifications in the Status Line.

Tip:

Vellum 3D Users: If you pressed ENTER (Windows) or RETURN (Macintosh) a second time after creating a line in Vellum 3D, you would create an identical object in the same location. This is not the case with your Desiger Elements program.

Creating Additional Geometry with the Status Line

1. With the **Single Line** tool still selected from the previous example, click 2 more points.
2. Type **4**.
3. Press the TAB key and type **25** in the Angle field.

4. Press ENTER (Windows) or RETURN (Macintosh). Another line is drawn.

Creating New Geometry with the Status Line

1. Choose the **Selection** tool so the Status Line clears.
2. Click the **Single Line** tool again.

   The X status field is active, showing that you can enter a value for the X coordinate of the line's beginning point.

   Do not press ENTER (Windows) or RETURN (Macintosh) until you reach step #14.
3. Type **0**.
4. Press the TAB key. The Y data field highlights.
5. Type **0**.
6. Press the TAB key. The Z data field highlights.
7. Type **0**.
8. Press the TAB key.

   The dX data field highlights. The dX value is the delta X, the numeric difference between the beginning and ending X coordinate.
9. Type **2**.
10. Press the TAB key. The dY data field highlights.
11. Type **2**.
12. Press the TAB key. The dZ data field highlights.
13. Type **0**.
14. Press ENTER (Windows) or RETURN (Macintosh). The line is drawn.

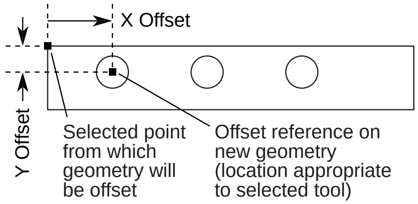
Creating Geometry Offset from a Point

If you want to create geometry offset from an existing point, you can use the Status Line to specify the offset.

Creating Geometry Offset from a Point

1. Select the tool you want to use.
2. Move the pointer over the control point from which you want the offset.
3. Click once to lock onto that point.
4. Finish creating the geometry.

5. With the geometry still selected, click in the appropriate X, Y, or Z data field in the Status Line, placing the text cursor at the end of the entry.
6. Type the offset (such as **+ 3**) and press ENTER (Windows) or RETURN (Macintosh).
7. Continue with your construction.



Coordinate System Axis

The coordinate system axis, at the left of the Status Line, displays the coordinate system currently set for the file. You can choose either the Global (world coordinate system) or a user-defined coordinate system. The graphic below represents the global coordinate system. The default system is the Global coordinate system.

The Global System aligns with the X, Y and Z axes (X = 1, 0, 0; Y = 0, 1, 0; Z = 0, 0, 1). The user-defined coordinate system is set by you. Planes labeled as *DynWorkPlane* or *UserWorkPlane#* are part of the user-defined coordinate system. See Chapter 32 for information on the Work Plane Manager and defining your own work planes.

Choosing a Coordinate System

1. Choose the coordinate system by clicking on the coordinate system axis icon at the far left of the Status Line.

Referral:

See Chapter 32 for information on defining a work plane.

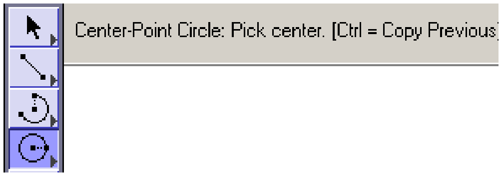
A menu displays.

2. Choose the GlobalWorkPlane, DynWorkPlane or any user-defined work plane.

   The work plane and coordinate system are now set.

3. Continue designing.

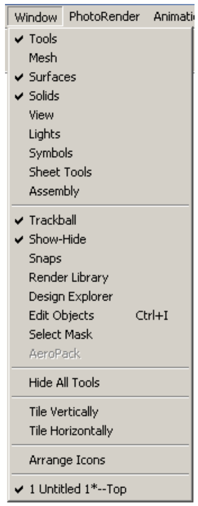
Message Line



The *Message Line* is an important feature when drawing. After selecting a tool, the line displays the tool name and the first step in its use.

Some tools display a subpalette and a pull-down option list. As you finish each step, the next step displays until all steps are completed. The Message Line may also display additional commands that can be used with the tool.

Trackball



The *Trackball* gives you the ability to rotate the view orientation of your geometry in the drawing area. You can also rotate the view around an object by selecting it before using the Trackball. The objects include features such as holes. You have to select the hole through the Design Explorer to do the rotation. (Selecting a hole results in a view rotation around the hole center by referencing the faces and edges introduced by the hole.)

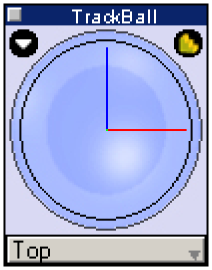
To display the Trackball, choose ***Window>Trackball***. When the Trackball displays a check mark appears in front of the command in the Windows menu. You can drag the Trackball to any location in your drawing area.

You can toggle the Trackball between two display types, the sphere display and the step display. Both displays include a views pull-down menu.

Axis Locking

The **Trackball** will lock rotations to an X, Y or Z axis. Hold down the X, Y or Z key on your keyboard while using the mouse in the Trackball window and the Trackball will rotate only in the direction of the key being pressed.

Sphere Trackball



The *Sphere Trackball* gives the ability to drag your view to the desired rotation. You can drag beyond the boundary of the trackball and continue the rotation. The sphere trackball is the default display.

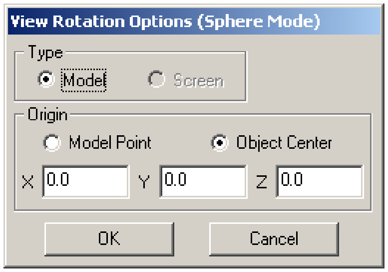
Using the Sphere Trackball

1. Choose ***Window>Trackball***.

2. Drag the pointer on the Trackball to rotate the view.

   The model rotates as you drag. See a later section for setting view rotation options. The model continues rotating if you drag the pointer past the edge of the trackball display and until you release the mouse button.

Setting View Rotation Options



You can to set view rotation options for the Sphere Trackball in the View Rotation Options dialog box.

To display the dialog box, double-click on the gray area of the Sphere Trackball.

The View Rotation Options dialog box appears containing the following:

Type Allows you to set the rotation type, *Model* or *Screen*.

The *Model* option rotates the view around the x, y and z axis as displayed by the Axis icon.

The *Screen* option rotates the view around the screen axis with the x axis oriented horizontally, the y axis oriented vertically and the z axis oriented normal to the screen.

Origin Allows you to set the origin at either the *Model Point* or the *Object Center.*

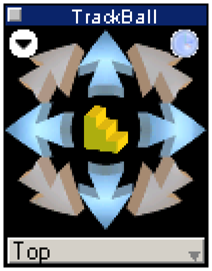
The *Model Point* option gives you the ability to determine the rotational point yourself. You can either enter the values in the x, y and z data fields or click on your geometry to set the location (values are entered automatically).

The *Object Center* option gives you the ability to rotate your geometry around the center of the objects in your drawing. This center is calculated by your Desiger Elements program. When you select this option, the x, y, and z data fields are unavailable.

You can choose only one of the four rotational options, *Model Type*, *Screen Type*, *Model Point Origin* or *Object Center Origin*.

Click OK to accept your settings and close the dialog box or Cancel to close the dialog box without accepting the changes.

Step Trackball



The *Step Trackball* gives you have ability to rotate your view in regular angle increments or in a continuous movement and provides you with addition options in the View Rotation Options dialog box.

Change the Sphere Trackball to the Step Trackball by clicking the arrow button on the right side of the Trackball title bar.

The Step Trackball includes the following icons:

Directional Arrows These arrows allow you to rotate your view in a specific direction. Clicking the vertical arrows rotates your geometry about the x axis. Clicking the horizontal arrows rotates your geometry about the y axis. Clicking the angled arrows rotates your geometry about the z axis. (In the View Rotation Options dialog box, you can choose either the model or screen axis to reference the rotation when using these arrows. See a later section, “Setting View Options,” for more information.)

Step Rotation This display, represented by the stair icon, toggles with Continuous Rotation. With the step icon displayed, rotation moves

through stepped increments in the direction you select.

You can set the degree increment for the steps in the View Rotation Options dialog box. See a later section, “Setting View Options,” for more information.

Continuous Rotation

This display, represented by the circular arrow, toggles with Step Rotation. With this icon displayed, rotation is a continuous motion in the direction you select.

You can temporarily halt the rotation by placing the cursor over the arrow and pressing. When you release the mouse, the rotation continues. To stop the continuous rotation, click the circular arrow icon.

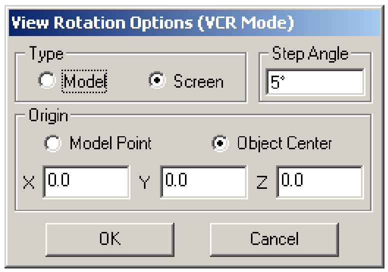
Using the Step Trackball

1. Choose ***Window>Trackball***.
2. Click on the arrow button on the right side of the Trackball title bar to change the display to the Step Trackball.
3. Click one of the directional arrows. The model rotates a specified number of degrees. Or...

   Click on the *Step Rotation* icon to toggle the display to the *Continuous Rotation* icon and click one of the directional arrows. Click on the circular arrow icon to stop the rotation.

Setting View Rotation Options

Your Desiger Elements program allows you to set view rotation options for the Step Trackball in the View Rotation Options dialog box.

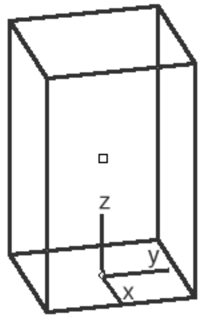


To display the dialog box, double-click on the black area of the Step Trackball. View Rotation Options containing these options:

Type

This option sets the rotation type, *Model* or *Screen*.

The *Model* option rotates the view around the x, y and z axis as displayed by the Axis icon. The option allows you to rotate the model around one stationary axis. See the graphic here.



The *Screen* option rotates the view around the screen axis with the x axis oriented horizontally, the y axis oriented vertically and the z axis oriented normal to the screen.

Step Angle

Allows you to set the rotation angle for the Step Trackball.

Origin

Allows you to set the origin at either the *Model Point* or the *Object Center.*

The *Model Point* option gives you the ability to determine the rotational point yourself. You can either enter the values in the x, y and z data fields or click on your geometry to set the location (values are entered automatically.

The *Object Center* option gives you the ability to rotate your geometry around the center of the objects in your drawing. This center is calculated by your Desiger Elements program. When you select this option, the x, y, and z data fields are grayed out.

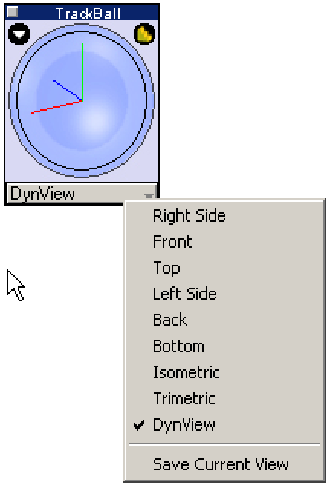
You can choose only one of the four rotational options, *Model Type*, *Screen Type*, *Model Point Origin* or *Object Center Origin*.

Click OK to accept your settings and close the dialog box or Cancel to close the dialog box without accepting the changes.

Trackball View Menu

The Trackball has a pull-down menu for specifying the view or saving the current view. The views available in this menu include the default views and any user-defined views.

Using the Trackball View Menu



1. Move the pointer to the current view name displayed at the bottom of the Trackball window.
2. Press the mouse button.

   The Trackball views menu displays.
3. Choose the view orientation you want to display in the current window.

   The view orientation changes to your specification in the view window. A check mark appears next to the selected view, as shown here.

View Rotation Short Cut

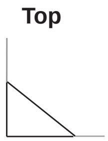
You can rotate your view by holding down the SHIFT key and pressing one of the keyboard arrows. This rotates the view a specified number of degrees.

You can also change your view with the following keyboard short cuts: a - Side View, s - Front View, d - Top View, f- Isometric View and g - Trimetric View.

Drawing Display Commands

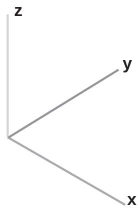
There are a number of commands that display features that may assist you in creating geometry. These include: *Show/Hide Triad*, *Show/Hide Axis*, *Show/Hide Work Plane*, *Show/Hide Points* and the Grid commands.

Show/Hide Triad



This command in the Planes menu toggles the display of the *Triad* symbol in the upper-left corner of the view windows. The Triad illustrates the orientation of the x, y, z axes and the work plane. See Chapter 3 for more information on the Triad.

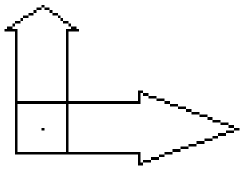
Show/Hide Axis



This command in the View menu toggles the display of the Axis symbol at the drawing origin. The Axis establishes the direction of the x, y and z axes. The Axis can clarify the geometry location when rotating the view.

Show/Hide Work Plane

This command in the Planes menu toggles the display of the Work Plane icon on your geometry. The graphic below shows the work plane set to the Top plane.



Show/Hide Points

This command in the Edit menu toggles the display of the control points for selected objects. See Chapter 4, “Selecting Objects,” for more information.

Grid

The grid overlays your drawing and can help you create and align geometry. The grid is made up of horizontal and vertical lines of dots. The grid commands, *Show/ Hide Grid* and *Snap to Grid* are found in the Planes menu. Set your Grid preferences (spacing, the number of divisions, display appearance and startup options by choosing ***File>Preferences>Grid***.

Show/Hide Grid Command

This command in the Planes menu toggles the display of the grid. The grid is always aligned with the work plane's x and y axis. You can display the grid when viewing any plane.

Snap to Grid Command

This command in the ***Window>Snaps*** menu toggles on and off and is normally used with the grid. However, this command does not require that the grid is displayed.

When *Snap to Grid* is on (a check mark appears next to the name), the Drafting Assistant snaps all geometry to the grid. In other words, if the grid is set to .25 inch spacing, you can't construct an object closer than .25 inch to another object.

Tip:

If you notice a gray rectangle in your drawing area when you first start your Desiger Elements program, choose ***File>Page Setup***. The option, *Show Page Breaks in Drawing Windows* may be selected. The gray rectangle is the page bounds.

Drawing at Full Scale

Whether you are designing or drafting a highly detailed blueprint, you should create the geometry at its actual size. Your Desiger Elements program allows you to construct the part using full-scale specifications and then set the visual scale of the drawing. In this way, the part dimensions to its true-to-life measurements. Drawing at full scale has the following advantages:

- Scaling mistakes are eliminated
- Dimensions are automatic (you must dimension manually if you do not draw at full scale)
- Associative dimensions update when the object is edited (manual dimensions do not)
- The size relationship of imported parts is compatible.

Once your project is drawn, you can dimension it and scale it visually to fit into a standard drawing size, if you wish with the Print Setup command (Windows) or Page Setup command (Macintosh) in the File menu. Regardless of how you set up your drawing, the actual size of the geometry remains constant unless you edit it.

If you want to create ensure that your geometry is contained within a specified page bounds at a 1:1 scale, you can display the page bounds. See Chapter 37 for more information.

When you open a new Desiger Elements program document, the drawing area is an infinitely large sheet so that you can design anything at full size. For a simplistic example, here's how to draw and view a line 83 feet long:

1. Draw a line, specifying **83'** for the length.

   The line extends off the screen.

2. Choose ***View>Zoom All***.

   The entire 83 foot line is visible on the screen.

Using the draw to scale/Zoom All method, you can create accurate full-scale drawings which are displayed at the magnification you choose. The actual size of an object is not affected by zoom magnification or reduction.

Zoom All magnifies or reduces all objects on your drawing to fill the screen—regardless of the size of the objects.

Default versus Selected Object Settings

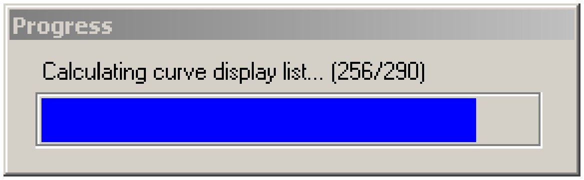
When no object is selected, any setting changes made with respect to *Selectable Points*, *Grid*, *Axis*, *Triad*, *Show Points*, *Construction Lines*, *User-defined plane*, *Pen*, Text, *Dimension*, tool palettes (choice, status and location) and *Render* become the default for all open files and the current Desiger Elements program session. When an object is selected, any change made will only affect the object.

Escape Key

When geometry becomes quite complex, the time required for operations to complete will inevitably lengthen. You can interrupt the command by pressing the ESC key.

Progress Bar

The progress bar provides you with feedback when opening files, importing or exporting files or performing complex operations. The bar provides you with feedback about the status of the operation.



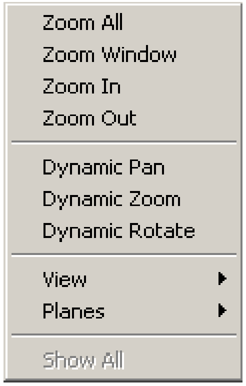
Right Mouse Button

Your Desiger Elements program gives you access to commands though the right mouse button. If you are a Macintosh user and do not have a right mouse button, these same commands are available by holding down the CONTROL key and pressing the mouse button.

Different sets of commands display depending on whether the button is pressed while on or off an object or when a drawing view is activated.

Over No Object

When you click the right mouse button (Windows) or Control + mouse button (Macintosh), the following popup menu appears:



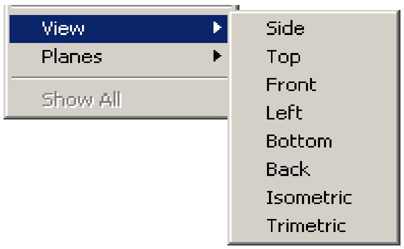
The menu includes the following commands:

Zoom All, **Zoom Window**, **Zoom In**, **Zoom Out**
These commands are the same as those in the View menu.

Dynamic Pan, **Dynamic Zoom**, **Dynamic Rotate**
These commands perform the same function as the **Dynamic Pan**, **Dynamic Zoom** and **Dynamic Rotate** tools in the **View** tool palette. When you select one of these commands, the pointer icon becomes that of the tool. You can perform the operation once and then the icon reverts back to the selected tool.

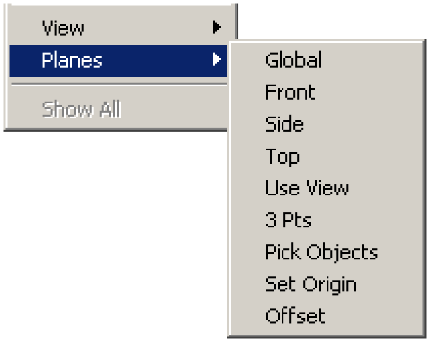
If you want to perform multiple operations, hold down the Shift key before selecting the command. When you release the Shift key, the icon reverts back to the selected tool.

View
This command displays all default views and any user-defined views.



Planes

This command displays the same commands as those in the Planes menu, with the exception of the *Show Work Pln* command.

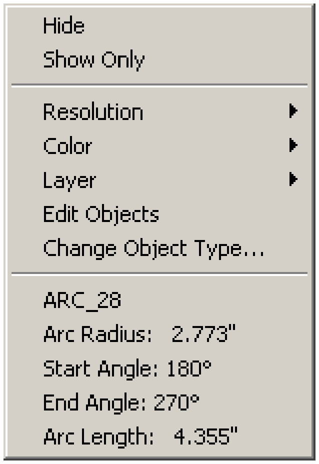


Show All

This command is enabled when objects are hidden in the drawing and is the same as the *Show ALL* command in the Show-Hide dialog box.

Over An Object

When you select an object. place the pointer over the selected object and click the right mouse button (Windows) or Control + mouse button (Macintosh), the following popup menu appears:



The menu includes the following:

Hide, Show Only

These commands are the same as those in the Show-Hide dialog box allowing you to hide or show the object.

Resolution

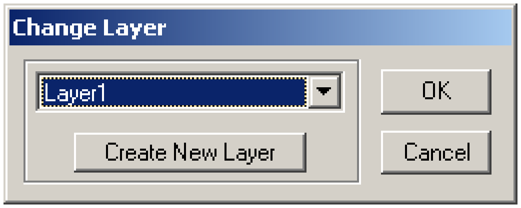
This command displays a menu allowing you to change the object's resolution to Super Fine, Very Fine, Fine Medium or Coarse.

Color

This command displays the color menu allowing you to change the color or the object. These are the same color options available when choosing ***Pen>Color***.

Layer

This command allows you to change the object's layer to any layers in the file, whether or not a layer is hidden. It also includes The *More* command. Selecting More displays the following dialog box:



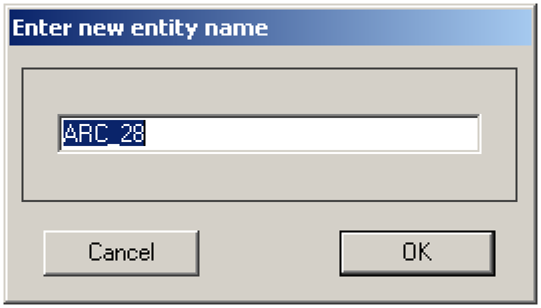
Click on the Create New Layer button and then click OK to move the object to the new layer. The new layer is titled with a default name (for example, Layer3). You can change the name through the Layer Manager (***Layout>Layer Manager***).

Edit Objects

This command displays the Edit Objects dialog box.

Object Name

The name of the object appears in the menu. You can change the object's name by clicking on the name. The following dialog box appears:



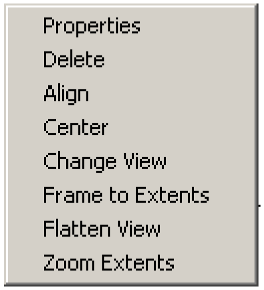
Type a new name and click OK to make the change.

Object Characteristics

For a wireframe object, the menu also provides its geometric characteristics. For example, an ellipse displays the major radius, the minor radius and circumference.

Selected Drawing View

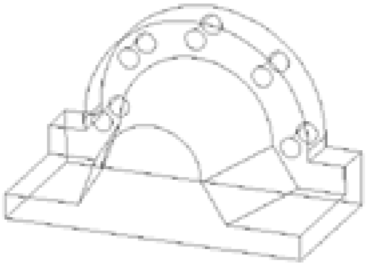
When a drawing view is selected and you click the right mouse button (Windows) or Control + mouse button (Macintosh), the following popup menu appears:



These commands are identical to those in the Drawing View pull-down menu. See Chapter 36 for information on these commands.

Introduction to Wireframe Modeling

In your Desiger Elements program you have the ability to create wireframe, surface and solid models. The most basic model is the wireframe model. A wireframe consists of the geometry that makes up the edges of the object. The word “wireframe” relates to the idea of a wire that is bent to follow an object's edges. A wireframe model is the simplest mathematical representation of an object.



Wireframes in your Desiger Elements program consist of points, lines, arcs, circles, ellipses, conics, splines or a combination of any of these. These particular wireframes are also collectively referred to as curves. Although wireframes are limited in the amount of model content they represent, they are powerful building blocks for creating more complex models composed of surfaces and solids.

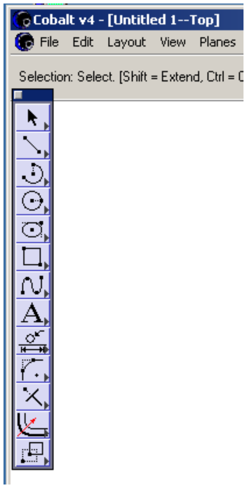
A wireframe model can often be used in place of a prototype (to run simulations and tests on the computer rather than in the laboratory). Models can be used for checking visual specification, measuring distances between points within the model and observing the visual and real intersections of lines.

The topics discussed in this chapter include:

- Wireframe tools
- Wireframes and the Drafting Assistant
- Wireframes, Selection and Display
- Object Types and Edit Objects

Wireframe Tools

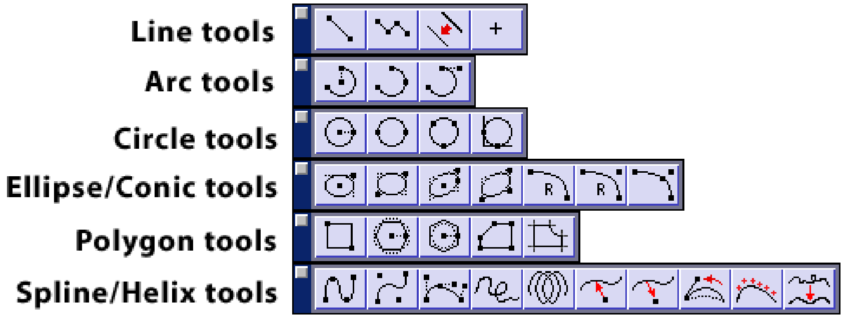
These chapters in this section describe how to create wireframe geometry with your Desiger Elements program tools. They also provide information on how the Drafting Assistant helps you design faster and easier.



Wireframe tools are contained in the main tool palette, located at the left side of the Desiger Elements program drawing area.

You can change the default status (open or closed) and location of the main tool palette by choosing your status and location and quitting your Desiger Elements program. The next time you launch the program, the palette will retain the status and location you chose.

Each icon in the palette is the first in the subpalette of tools grouped together by purpose, as shown here.



Information on using each tool is contained in the chapters that follow.

Wireframes and the Drafting Assistant

The Drafting Assistant recognizes several useful snap locations associated with wireframe objects. These dynamic snap locations include:

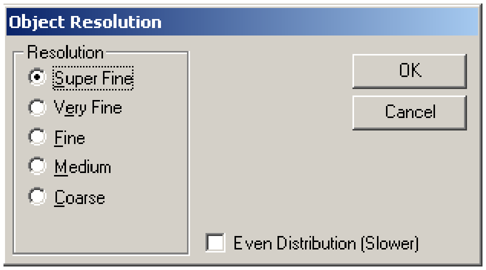
- Endpoints
- Midpoints
- Intersections
- Projections
- Centers
- Vertices
- Tangencies

The Drafting Assistant recognizes such points in all three dimensions, making 3D drafting much easier.

Wireframes, Selection and Display

Wireframes are selected when they are within the selection fence or the boundaries of a single pick box as defined in Preferences (see Chapter 6, “Preference Settings”). Conics, ellipses, splines and circles have resolution attributes that impact the screen display and printer output. You can change the resolution of the selected object by choosing ***Edit>Change Resolution***.

Changing an Object's Resolution

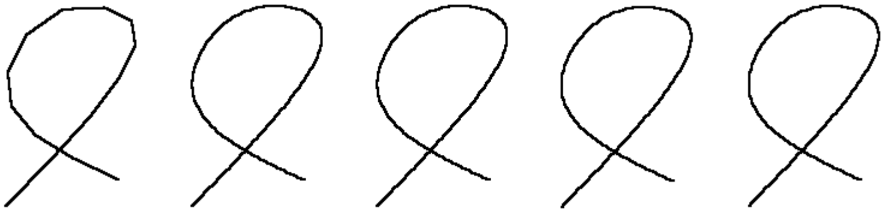


1. Select the object.
2. Choose ***Edit>Change Resolution***.

   The Object Resolution dialog box appears.

   You have five resolution options: *Super Fine*, *Very Fine*, *Fine*, *Medium* and *Coarse*. *Medium* is the default option.

.



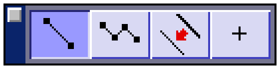
Objects Characteristics and Edit Objects

All wireframe objects you create with a Desiger Elements program tool are defined by their own geometric, display and attribute characteristics. The Edit Objects dialog box provides information for these three categories. Where the display and attribute characteristics are identical for wireframe objects, the geometric characteristics differ according to a particular object.

For example: A line created by the **Single Line** tool includes the following geometric categories: Length, Angle, End 1 (X, Y and Z values) and End 2 (X, Y and Z values).

Chapters 8 through 13 introduce you to the wireframe tools. With every tool description there is a list of the geometric characteristics displayed in the Edit Objects dialog box. For information on using the *Edit Objects* command and the dialog box, see Chapter 24.

Line Tools



The Line tools in the **Line** tool palette create line segments, connected lines, lines parallel to existing lines and points. As you create a line, the coordinate locations, line length and angle from horizontal appear in the Status Line. All geometry appears in the current pen specifications for color, weight and pattern.

For each tool you can enter values in the Status Line to define a line, either before or after you create it. If you enter the values after you select the tool but before you create the line, your first click in the drawing area automatically registers all Status Line values. If you enter values in the selected Status Line data field after creating the line and while the line is still selected, pressing ENTER (Windows) or RETURN (Macintosh) updates the line to reflect the new values.

Tech Note:

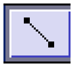
Vellum 3D Users: your Designer Elements program, unlike Vellum 3D, will not create a second entity if you hit ENTER (Windows) or RETURN (Macintosh) while still in a tool.

The topics explained here include

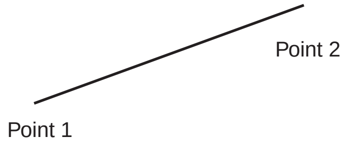
- **Single Line** Tool
- **Connected Lines** Tool
- **Parallel Line** Tool
- **Point** Tool
- Modifying Lines

Your Designer Elements program does not support Smart Wall entities. All imported Smart Walls are converted into individual curves.

Single Line Tool



This tool draws a line between two points. You can click points to draw a line.

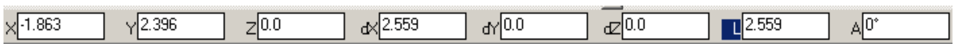


Using the Single Line Tool

1. Select the tool. The Message Line reads: *Single Line: Pick the beginning point. [Ctrl = Copy Previous]* (Windows) or *[Option = Copy Previous* (Macintosh)*]*.
2. Click to place the first endpoint of the line.
3. Move the cursor to a new location. As you do, a rubberband image appears that previews your construction.
4. Click to place the last point of the line.

To copy the line just created, as the Message Line indicates, hold down the CTRL (Windows) or the OPTION (Macintosh) key and click once in the drawing area to set the beginning point. An identical line appears beginning where you clicked and on the current work plane.

The Status Line contains the X, Y, and Z coordinates of the beginning, the relative location of the end (delta X, delta Y and delta Z), the line length, and the angle from horizontal. (If any of the fields do not display, click the arrow at the right end of the Status Line). Once a line is drawn, the Length field is active in the Status Line.



Drawing a Line Perpendicular to Another Object

1. Construct the object.
2. Move the pointer to the object until a Drafting Assistant *on* notation appears (*endpoint*, *midpoint*, *quadrant* etc., will not work, only “*on*”).
3. Drag straight away from the object in a perpendicular direction. A *perpendicular* line will appear attached to the object. Notice that the line stays perpendicular but slides along the object.

4. Drag to the desired length.

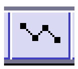
Drawing a Line Tangent or Perpendicular to an Arc, Circle or Ellipse

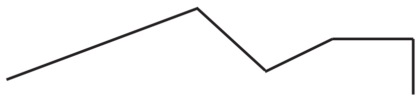
1. Construct an arc, circle, or ellipse.
2. Choose the **Single Line** tool.
3. Move the pointer to the arc until a Drafting Assistant *on* notation appears (*end-point*, *midpoint*, *quadrant* etc., will not work, only “*on*”).
4. Drag in the appropriate direction (straight out for perpendicular or at an angle for tangent) until the Drafting Assistant perpendicular or *tangent* notation appears.
5. When the Drafting Assistant locks on to *perpendicular* or *tangent*, drag the line around the arc to the desired location and extend the line to the desired length.

Geometric Characteristics

A single line is created by clicking two points and is made up of the following characteristics: Length, Angle, End 1 (X, Y and Z values) and End 2 for X, Y and Z values). This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the line and choose ***Window>Edit Objects*** or double-click on the line.

Connected Lines Tool

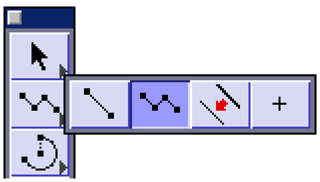




This tool draws lines in which the endpoint of one line segment is the beginning point of the next.

Using the Connected Lines Tool

1. Select the tool. The Message Line reads: *Connected Lines: Pick the beginning point.*

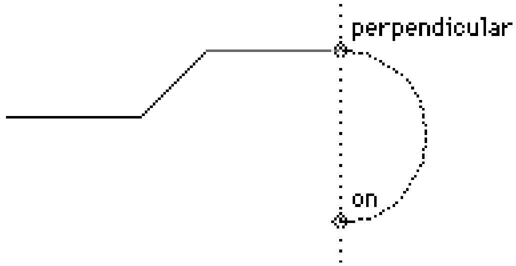


2. Click to indicate the endpoints of the line segments.

   If you click a point and then change your mind, press the ESC key, or choose *Undo* to remove the last segment.

   After the beginning point is set, the Message Line changes to reflect the next step.

3. Indicate the last point by double-clicking, hitting the ESC key or by choosing another tool.



After completing at least one segment with the **Connected Lines** tool, you can create a tangent arc off of the last line by holding down the CTRL (Windows) or the OPTION (Macintosh) key (the pointer temporarily changes to an "arc" icon) and clicking or dragging to the next point. The Message Line notes this added feature. Several tangent arcs can be strung together by continuing to hold down the CTRL (Windows) or OPTION (Macintosh) key.

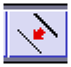
The radius of arcs created with this method cannot be edited in the Status Line.)

The Status Line contains the X, Y and Z coordinates of the beginning, the relative location of the end (delta X, delta Y and delta Z), the line length and the angle from horizontal. Once a line segment is drawn, the Length field is active in the Status Line.

Geometric Characteristics

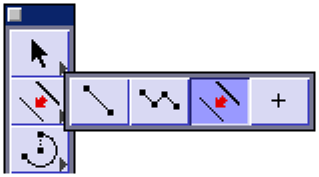
A connected line is multiple single lines connected at their endpoints. Lines are made up of the following characteristics: Length, Angle, End 1 (X, Y and Z values) and End 2 (X, Y and Z values). This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the line and choose ***Window>Edit Objects*** or double-click on the line.

Parallel Line Tool



This tool constructs a line parallel to an existing line in the current work plane.

Using the Parallel Line Tool



1. Select the tool. The Message Line reads: *Parallel Line: Drag new line off existing line.*

2. Drag off the desired line and release to indicate the position for the new parallel line. A rubber-band line moves with the pointer. The distance field is active in the Status Line and displays distance from the original line that you dragged.

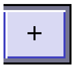
   If you type in your own value and hit ENTER (Windows) or RETURN (Macintosh), the parallel line will move the offset distance specified.

The Status Line contains the Offset data field.

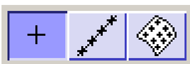
Geometric Characteristics

A parallel line is a duplicate of a line created by the **Single Line** tool or the **Connected Lines** tool. The line is made up of the following characteristics: Length, Angle, End 1 (X, Y and Z values) and End 2 (X, Y and Z values). This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the line and choose ***Window>Edit Objects*** or double-click on the line.

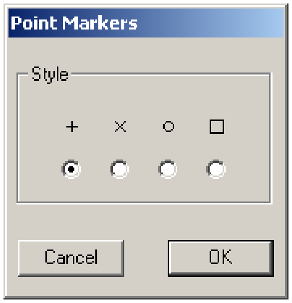
Point Tool



When you select the **Point** tool, a subpalette appears in the Message Line, with three **Point** tools, **Point at an XYZ Location**, **Points on a Curve** and **Points on Surface**.

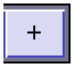


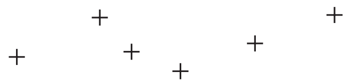
You can choose the appearance of the points using any of these tools. Press the CTRL (Windows) or the OPTION (Macintosh) key to display the Point Markers dialog box.



Click the radio button for the desired marker (crosshairs plus, crosshairs x, empty diamond or empty square) and click OK to close the dialog box. When you click to place the point in the drawing area, the point is represented by the new marker. The Point Marker style cannot be saved as a preference.

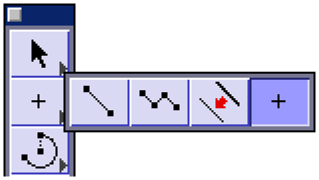
Using the Point at an XYZ Location Tool





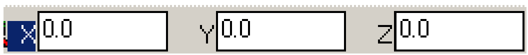
You create individual points in your drawing area with this tool.

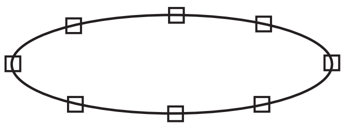
1. Select the **Point** tool from the palette.



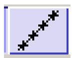
2. Select the **Point at an XYZ Location** tool in the Message Line. The Message Line reads: *Point: Enter point. [Ctrl = Marker* (Windows) or *Option = Marker* (Macintosh)*]*.
3. Click a point in the drawing area.

The Status Line contains the X, Y and Z location of the point.



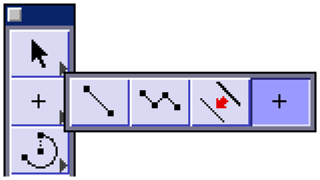


Using the Points on a Curve Tool



This tool places a series of points distributed equally along a curve (remember that curves include lines, circles, ellipses, arcs and splines). You cannot use this tool on polygons created using one of the **Polygon** tools.

1. Select the **Point** tool from the palette.

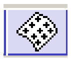


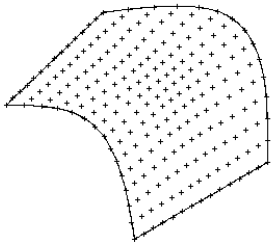
2. Select the **Point on a Curve** tool in the Message Line. The Message Line reads: *Point on Curve: Select curves to distribute points on. [Ctrl = Marker* (Windows) or *Option = Marker* (Macintosh), *Shift = Extend].*
3. Enter the number of points you want distributed along the curve in the #Pts data field of the Status Line.
4. Select the curve. If you want to select more than one curve, hold down the SHIFT key before selecting the first curve and while selecting succeeding curves.

   Points are equally distributed along the curve. For closed curves, the start and endpoint are the same.

The Status Line contains the number of points to distribute along a selected curve.

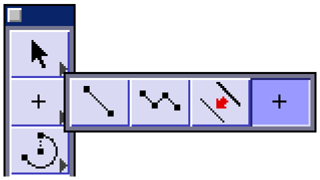
Using the Points on a Surface Tool





This tool places a series of points distributed equally along a surface.

1. Select the **Point** tool from the palette.



2. Select the **Points on a Surface** tool in the Message Line. The Message Line reads: *Point on Surface: Select surface(s) to distribute points on. [Ctrl = Marker* (Windows) or *Option = Marker* (Macintosh), *Shift = Extend].*
3. Enter the number of points you want distributed along the surface in two perpendicular directions in the #U and #V data fields of the Status Line. *U* and *V* represent perpendicular coordinate directions along the surface. (The letters U and V are standard identifiers for surface coordinates.)
4. Select the surface. If you want to select more than one surface, hold down the SHIFT key before selecting the first surface and while selecting succeeding surfaces.

   Points are equally distributed in a grid like pattern across the surface.

The Status Line contains the number of points to distribute in the U and V direction for the surface.

Geometric Characteristics

A Point is created by a single click placed in one or more locations depending on the tool used. Points are made up of the following characteristics: X, Y and Z coordinates and the Display Style of the Point (crosshair plus, crosshair x, empty diamond, empty square). This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the point and choose ***Window>Edit Objects*** or double-click on the point.

Modifying Lines

You can modify a line in a number of ways. See Chapter 21 for information on the Editing tools. After creating a line, you can change any value including length if the line is still selected.

For example, you can type a new length, press ENTER (Windows) or RETURN (Macintosh) and the length will change. If you decide that the length is not correct, you can choose ***Edit>Undo*** and the line will return to the previous length. If you decide to change the length of the line again you may either select the line with the selection tool and edit information in any of the status line boxes or you may double click on the line to bring up the Edit Objects dialog box and edit all aspects of the line there.

Arc & Circle Tools

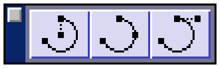
This chapter contains information on the Arc and Circle tools available in your Designer Elements program. As you create an arc or circle, the coordinate locations and radius/diameter appear in the Status Line. The arc or circle is also drawn with the current pen specifications for color, weight and pattern.

For each tool you can enter values in the Status Line to define an object, either before or after you create the object. If you enter the values after you select the tool but before you create the object, your first click in the drawing area automatically registers all Status Line values. If you enter values in the selected Status Line data field after creating the object and while the object is still selected, pressing ENTER (Windows) or RETURN (Macintosh) updates the object to reflect the new values.

The tools explained here include:

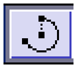
- **Center-Point Arc**
- **3-Point Arc**
- **Tangent-Point Arc**
- **Center-Point Circle**
- **Opposite-Point Circle**
- **3-Point Circle**
- **Tangent Circle**

Arc Tools

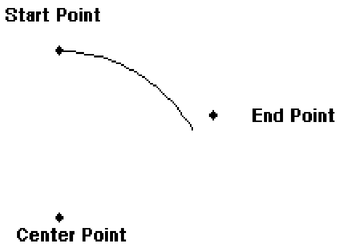


Your Designer Elements program features three arc tools: **Center-Point Arc**, **3-Point Arc** and **Tangent Point Arc**.

Center-Point Arc Tool

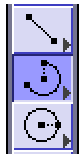


This tool draws an arc based on three points: the center point, the beginning point and the end-point of the arc.



Using the Center-Point Arc Tool

1. Select the tool. The Message Line reads: *Center-Point Arc: Pick Center.*
2. Click the center point of the arc. The Message Line now tells you the next step in using the tool.
3. Click the beginning point of the arc which will also define the radius.
4. As you move the cursor to place the final point, a rubberband arc appears. Click the final point.



To construct an arc greater than 180**°**, you must create one of 180° or less, double-click the arc to bring up the Edit Objects dialog box and enter the desired degrees.

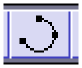
The Status Line contains the X, Y and Z coordinates of the center of the arc, the Radius, the starting angle from horizontal, A, and the delta angle from the start, dA. The Radius field is active in the Status Line.

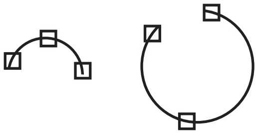
Geometric Characteristics

The center-point arc is created from a center point and two arc points. Arcs are made up of the following characteristics: Diameter, Start Angle, End Angle, Start (X, Y and Z values) and End (X, Y and Z values). This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the arc and choose ***Window>Edit Objects*** or double-click on the arc.

The Start and End Angles refer to the angles tangent to the arc point relative to the work plane.

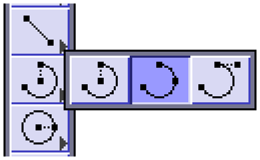
3-Point Arc Tool





This tool draws an arc through the points you select.

Using the 3-Point Arc Tool

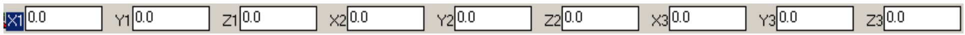


1. Select the tool. The Message Line reads: *3-Point Arc: Pick first point. [Ctrl = Tangent to object* (Windows) or *Option = Tangent to object* (Macintosh)*]*.
2. Click the first endpoint of the arc.

   Notice that the Message Line tells you the next step for using the tool.
3. Click the second point of the arc. A rubberband arc appears as you move the cursor to a third position.
4. Click the last point of the arc. The arc is created.

The arc is drawn from the first position that you indicate, through the second position and ends at the third position. For each of the three clicks that define the arc, if the CTRL (Windows) or the OPTION (Macintosh) key is held down and you click on some other object, your Designer Elements program will define the arc to be *tangent* to that object at the nearest tangency point.

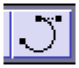
The Status Line contains the X, Y and Z coordinates for each of the three points.

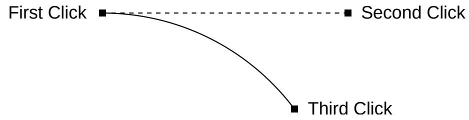


Geometric Characteristics

A 3-point arcs is created by placing three points and are made up of the following characteristics: Diameter, Start Angle, End Angle, Start (X, Y and Z values) and End (X, Y and Z values). This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the arc and choose ***Window>Edit Objects*** or double-click on the arc.

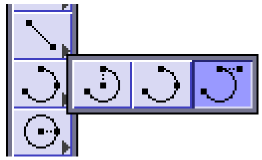
Tangent-Point Arc Tool





This tool draws an arc beginning at the first point you specify. The second point you specify is the direction vector and the third point you specify indicates the endpoint of the arc. Essentially, the **Tangent-Point Arc** tool first creates a line then creates an arc tangent to the line and erases the line.

Using the Tangent-Point Arc Tool



1. Select the tool. The Message Line reads: *Tangent-Point Arc: Pick beginning point of arc (tangent line)*.
2. Click the starting point. This is both the starting point of the arc and the starting point of the temporary tangent line.

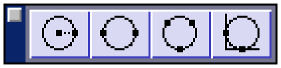
   The Message Line displays the next step for using the tool.
3. Click the endpoint of the tangent line.
4. Click the endpoint of the arc. The arc is drawn between the first and last point you click, tangent to the line between the first and second points.

The Status Line contains the X, Y and Z coordinates of the endpoints of the arc and the angle of the tangent line.

Geometric Characteristics

A tangent-points arc is created by clicking points for the tangent line and the arc and are made up of the following characteristics: Diameter, Start Angle, End Angle, Start (X,Y and Z values) and End (X, Y and Z values). This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the arc and choose ***Window>Edit Objects*** or double-click on the arc.

Circle Tools

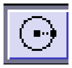


The Circle tools include the **Center-Point Circle**, **Opposite-Point Circle**, **3-Point Circle** and **Tangent Circle**.

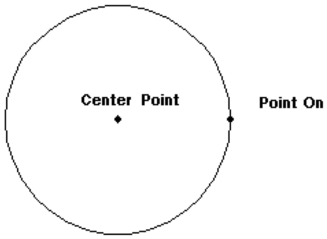
The **Center-Point** tool uses the center and diameter of the circle. The **3-Point** tool uses three points and can be tangent to existing objects. The **Tangent-Point** tool draws a circle tangent to two objects, using the diameter specified by the user.

The circle is drawn with the current pen specifications for color, weight and pattern.

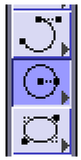
Center-Point Circle Tool



This tool draws a circle specified by the center point and diameter.



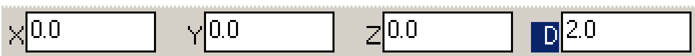
Using the Center-Point Circle Tool



1. Select the tool. The Message Line reads: *Center-Point Circle: Pick center. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*]*.
2. Click the center of the circle. A rubberband circle appears guiding your construction as you move the cursor.
3. Click the second point which determines the circle's radius.

You can create a copy of the last circle by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking once in the drawing area to set the center point. An identical circle appears center at the point you clicked on the current work layer.

The Status Line contains the X, Y and Z coordinates of the center and the diameter of the circle. Diameter is the active status field.



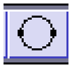
Geometric Characteristics

A center-point circle is created by clicking the center point and a point on the circle. Circles are made up of the following characteristics: Diameter and Center (X, Y and Z values). This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the circle and choose ***Window>Edit Objects*** or double-click on the circle.

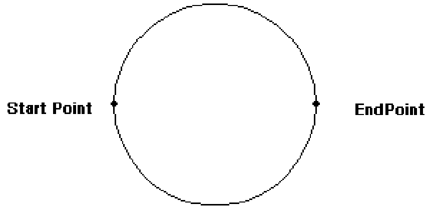
Tip:

You can select the center point of an opposite point circle if you display its control points using the *Show Points* command in the Layout menu.

Opposite-Point Circle Tool

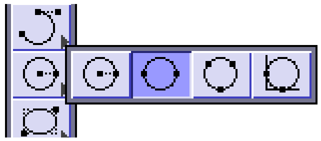


This tool draws a circle specified by the diameter.



Using the Opposite-Point Circle Tool

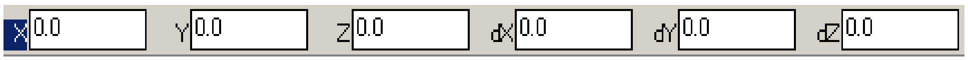
1. Select the tool. The Message Line reads: *Opposite-Point Circle: Pick first point on circle. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*]*.



2. Click two locations to indicate the diameter. After you click the first point, a rubberband circle appears to guide you in the construction.

You can create a copy of the last circle by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking once to place the center point of the new circle. An identical circle appears center at that point on the current work layer.

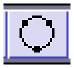
The Status Line allows you to specify the X, Y and Z coordinates representing the endpoints of the diameter.

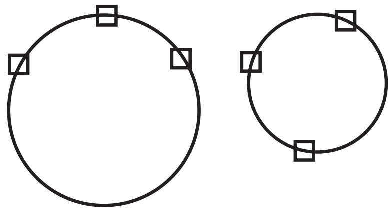


Geometric Characteristics

An opposite-point circle is created by clicking two points to establish the diameter. Circles are made up of the following characteristics: Diameter and Center (X, Y and Z values). This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the circle and choose ***Window>Edit Objects*** or double-click on the circle.

3-Point Circle Tool





This tool draws a circle through the points you select.

Using the 3-Point Circle Tool

1. Select the tool. The Message Line reads: *3-Point Circle: Pick first point. [Ctrl = Tangent to object* (Windows) or *Option = Tangent to object* (Macintosh)*]*.

2. Click the first point on the circle.

   Notice that the Message Line displays the next step for using the tool.

3. Click the second and third points. After the second point a rubberband circle appears guiding your construction.

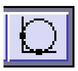
If you place any of the three points on an existing object, the circle is drawn through that point. If you click an object while holding down the CTRL (Windows) or OPTION (Macintosh) key, the circle is drawn tangent to the object rather than through the indicated point. You may combine the placement of these points to create a circle through a specific point of one object and tangent to another object, or tangent to three objects, etc.

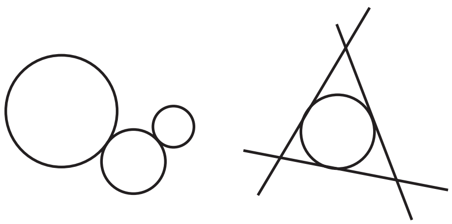
The Status Line contains the X, Y and Z coordinates for each of the three points.

Geometric Characteristics

A 3-point circle is created by clicking three points to set the boundary of the circle. Circles are made up of the following characteristics: Diameter and Center (X, Y and Z values). This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the circle and choose ***Window>Edit Objects*** or double-click on the circle.

Tangent-Point Circle Tool





This tool draws a circle tangent to two objects.

Using the Tangent-Point Circle Tool

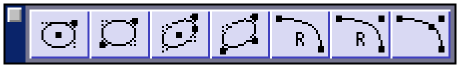
1. Select the tool. The Message Line reads: *Tangent Circle: Enter diameter then pick first tangent object.*
2. You can enter a diameter for the circle in the Status Line data field.

3. Click the objects to which the circle is to be tangent.

Geometric Characteristics

A tangent-point circle is created by entering the circle diameter and clicking on objects to establish tangency. Circles are made up of the following characteristics: Diameter and Center (X, Y and Z values). This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the circle and choose ***Window>Edit Objects*** or double-click on the circle.

Ellipse & Conic Tools



This chapter contains information on using the **Ellipse** tools and **Conic** tools. As you create an ellipse, the coordinate locations, lengths of the control rectangle and angles appear in the Status Line. As you create a conic, the rho value appears in the Status Line. These objects are also drawn with the current pen specifications for color, weight and pattern.

Referal:
The rho value definition can be found on pages 11-6 and 11-7.

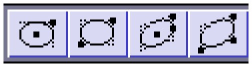
For each tool you can enter values in the Status Line to define an object, either before or after you create the object. If you enter the values after you select the tool but before you create the object, your first click in the drawing area automatically registers all Status Line values. If you enter values in the selected Status Line data field after creating the object and while the object is still selected, pressing ENTER (Windows) or RETURN (Macintosh) updates the object to reflect the new values.

The tools explained here include:

- **2-Point Center Ellipse**
- **Opposite Corner Ellipse**
- **3-Point Center Ellipse**
- **3-Corner Ellipse**
- **2-Point Conic**
- **3-Point Conic**

- **4-Point Conic**

Ellipse Tools



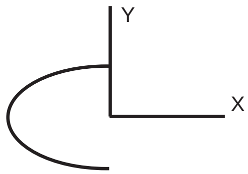
You can create ellipses with four tools: **2-Point Center**, **Opposite-Corner**, **3-Point Center** and **3-Corner**. These tools in the main tool palette construct ellipses inscribed within an invisible rectangle or parallelogram. The **2-Point Center Ellipse** tool uses the center point and one corner location of the rectangle. The **Opposite-Corner Ellipse** tool uses opposite corners of a rectangle. The **3-Point Center Ellipse** tool uses the center point, the midpoint of a side and the corner of the parallelogram. The **3-Corner Ellipse** tool uses three corners of a parallelogram.

Ellipses are defined by their *Major* and *Minor* Diameters, the Start and End Angles and their centers. The Start and End Angles specify the start and end location of the ellipse measured from the major axis.

Tip:

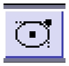
Vellum 3D Users: Ellipses are enhanced in your Designer Elements program and are now defined by centers.

The graphic here shows an ellipse with a Start and End Angle of 90° and 180°.

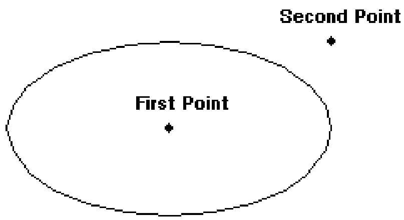


The ellipse is drawn with the current pen specifications for color, weight and pattern.

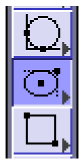
2-Point Center Ellipse Tool



This tool constructs an ellipse inscribed within a rectangle defined by two specified points: the center point and one corner of the rectangle.



Using the 2-Point Center Ellipse Tool



1. Select the tool. The Message Line reads: *2-Point Center Ellipse: Pick center of ellipse. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*].*

2. Indicate the center of the ellipse.

   Notice that the Message Line displays the next step for using the tool.

3. Indicate the corner of the rectangle defining the ellipse.

   If the two points are on the vertical or horizontal axis, a straight line is drawn.

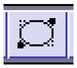
You can create a copy of the last ellipse by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking where you want the center. The copy is placed on the current work layer.

The Status Line allows you to specify the X, Y and Z coordinates of the center point and the length and angle of the semi-major and semi-minor axes of the ellipse.

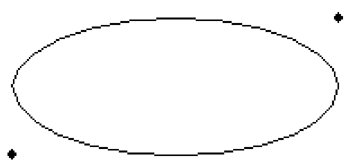
Geometric Characteristics

A 2-point center ellipse is created by placing the center point and corner of the control rectangle. It is made up of the following characteristics: Major Diameter, Minor Diameter, Start Angle, End Angle, and Center (X, Y and Z values). This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the ellipse and choose ***Window>Edit Objects*** or double-click on the ellipse.

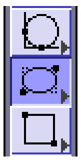
Opposite-Corner Ellipse Tool



This tool draws an ellipse inscribed in a rectangle specified by opposite corners.



Using the Opposite-Corner Ellipse Tool



1.Select the tool. The Message Line reads: *Opposite-Corner Ellipse: Pick first corner of rectangle. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*].*

2.Indicate one corner of the rectangle defining the ellipse.

Notice that the Message Line displays the next step for using the tool.

3. Indicate the opposite corner of the defining rectangle.

   If the two points are on the vertical or horizontal axis, a straight line is drawn.

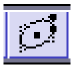
You can create a copy of the last ellipse by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking to place the lower-left point. The copy is placed on the current work layer.

The Status Line allows you to specify the X, Y and Z coordinates of the lower-left point, the length of the major and minor axes, L1 and L2 and angle of the major and minor axes, A1 and A2 of the ellipse.

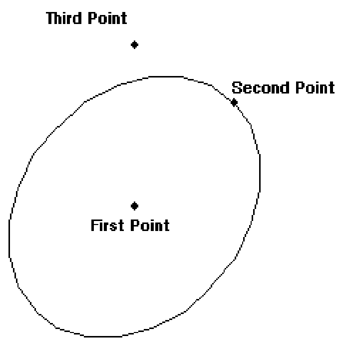
Geometric Characteristics

An opposite-corner ellipse is created by placing two points that represent the corners of the control rectangle and is made up of the following characteristics: Major Diameter, Minor Diameter, Start Angle, End Angle and Center (X, Y and Z values). This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the ellipse and choose ***Window>Edit Objects*** or double-click on the ellipse.

3-Point Center Ellipse Tool

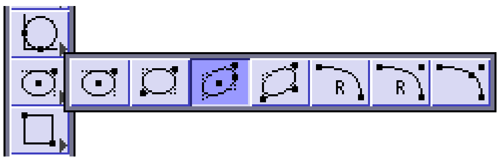


This tool constructs an ellipse inscribed within a parallelogram calculated from three specified points: a center point, a midpoint of a side and a corner of the parallelogram.



Using the 3-Point Center Ellipse Tool

1. Select the tool. The Message Line reads: *3-Point Center Ellipse: Pick center of the ellipse. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*]*. The Message Line will guide you through each successive step.



2. Pick the center of the ellipse.

3. Pick the midpoint of the side of the control parallelogram defining the ellipse.
4. Pick the corner of the control parallelogram defining the ellipse.

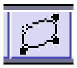
If the three points are on the vertical or horizontal axis, a straight line is drawn. You can create a copy of the last ellipse by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking where you want the center. The copy is placed on the current work layer.

The Status Line allows you to specify the X, Y and Z coordinates of the center point and the length and angle of the sides of the parallelogram.

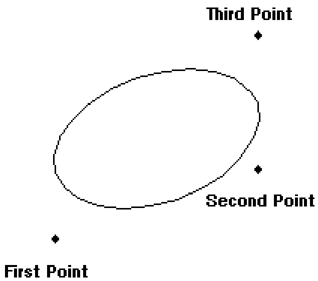
Geometric Characteristics

A 3-point center ellipse is created by placing the center point, midpoint of the side and the corner of the control parallelogram. It is made up of the following characteristics: Major Diameter, Minor Diameter, Start Angle, End Angle and Center (X, Y and Z values). This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the ellipse and choose ***Window>Edit Objects*** or double-click on the ellipse.

3-Corner Ellipse Tool

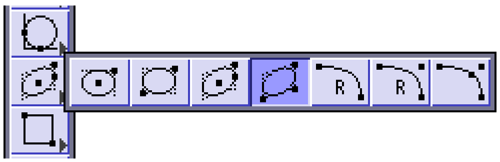


This tool draws an ellipse inscribed in a parallelogram defined by three corners.



Using the 3-Corner Ellipse Tool

1. Select the tool. The Message Line reads: *3-Corner Ellipse: Pick first corner of control parallelogram*. *[Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*]*. The Message Line will guide you through each successive step.



2. Pick one corner of the parallelogram defining the ellipse.
3. Pick another corner of the defining parallelogram.

4. Pick the final corner of the defining parallelogram.

If the three points are on the vertical or horizontal axis, a straight line is drawn. You can create a copy of the last ellipse by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking to place the lower-left corner of the parallelogram. The copy is placed on the current work layer.

The Status Line allows you to specify the X, Y and Z coordinates of a corner and the length and angle of the sides of the parallelogram.

Geometric Characteristics

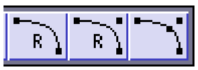
A 3-corner ellipse is created by placing the three points that represent the corners of the control rectangle and is made up of the following characteristics: Major Diameter, Minor Diameter, Start Angle, End Angle and Center (X, Y and Z values). This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the ellipse and choose ***Window>Edit Objects*** or double-click on the ellipse.

Modifying Ellipses

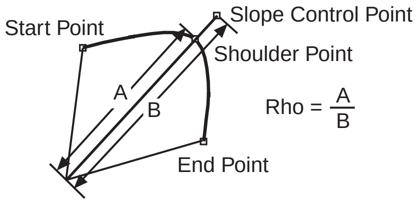
You can modify any ellipse in a number of ways:

- After it's created and while it's still selected by entering new values in the Status Line
- Using the **Transformation** tools (see Chapter 25)
- Using Edit Objects
- Select a control point and dragging it to a new location. The number of control points available to edit is based on the tool used to create the ellipse. A 2-Point Center ellipse has 2 control points.

Conic Tools

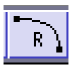


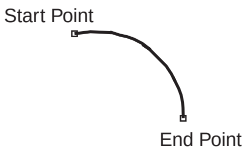
You can create conics with three tools: **2-Point Conic, 3-Point Conic** and **4-Point Conic**. The **2-Point Conic** tool uses the start point and end point to define the conic. The **3-Point Conic** tool uses the start point, end point and slope control point to define the conic. The **4-Point Conic** tool uses the start point, end points, slope control point and shoulder point to define the conic. The *rho* value is the dimension ratio of the distance from the center point to the shoulder point and the center point and the slope control point. The graphic below illustrates the definitions of the conic characteristics.



When using a conic tool a rubberband image appears after you place all but the last point. This allows you to see the ellipse before it is drawn. The conic is drawn with the current pen specifications for color, weight and pattern.

2-Point Conic Tool

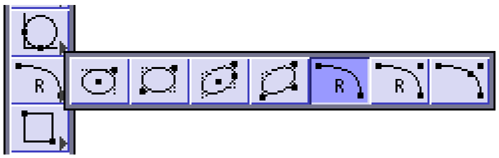




This tool constructs a conic calculated from three specified points: the start point, end point and slope control point.

Using the 2-Point Conic Tool

1. Select the tool. The Message Line reads: *2-Point Conic: Pick start point. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*]*. The Message Line will guide you through each successive step.



2. Pick the start point of the conic. As you move the cursor to place the second point, a rubberband image appears.
3. Pick the end point to complete the conic.

   You can modify the Rho value in the Status Line and press ENTER (Windows) or RETURN (Macintosh) to accept the new rho value. Rho values must be between .501 and .999. A rho value of .501 will create a straight line. A rho value of .999 will create a conic with a 90° angle.

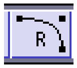
You can create a copy of the last conic by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking to indicate the start point. The copy is placed on the current work layer.

The Status Line allows you to specify the Rho value of the conic.

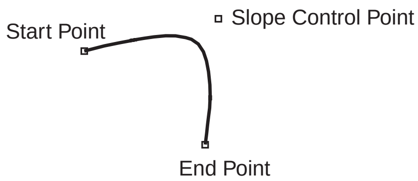
Geometric Characteristics

A 2-point conic is created by placing two points; the start point and end point and is made up of the following characteristics: Rho, End 1 (X, Y and Z values) and End (X, Y and Z values). This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the conic and choose ***Window>Edit Objects*** or double-click on the conic.

3-Point Conic Tool

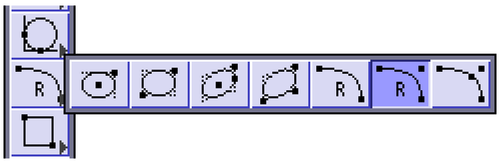


This tool constructs a conic calculated from three specified points: the start point, end point and slope control point.



Using the 3-Point Conic Tool

1. Select the tool. The Message Line reads: *3-Point Conic: Pick start point. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*]*. The Message Line will guide you through each successive step.



2. Pick the start point of the conic.
3. Pick the end point of the conic. Once you place this third point, a rubberband image appears.
4. Pick the shoulder point to complete the conic.

   You can modify the Rho value in the Status Line and press ENTER (Windows) or RETURN (Macintosh) to accept the new rho value. Rho values must be between .501 and .999. A rho value of .501 will create a straight line. A rho value of .999

will create a conic with a 90° angle.

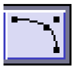
You can create a copy of the last conic by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking to indicate the start point. The copy is placed on the current work layer.

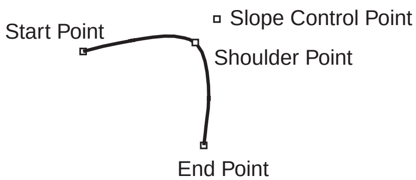
The Status Line allows you to specify the Rho value of the conic.

Geometric Characteristics

A 3-point conic is created by placing three points; the start point, end point and slope control point and is made up of the following characteristics: Rho, End 1 (X, Y and Z values) and End 2 (X, Y and Z values). This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the conic and choose ***Window>Edit Objects*** or double-click on the conic.

4-Point Conic Tool

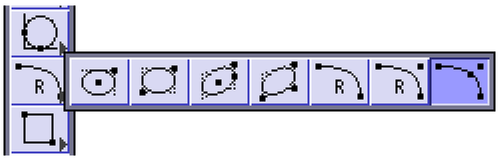




This tool constructs a conic calculated from four specified points: the start point, end point, slope control point and shoulder point.

Using the 4-Point Conic Tool

1. Select the tool. The Message Line reads: *4-Point Conic: Pick start point. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*]*. The Message Line will guide you through each successive step.



2. Pick the start point of the conic.
3. Pick the end point of the conic.
4. Pick the slope control point of the conic. Once you place this third point, a rubberband image appears.
5. Pick the shoulder point to complete the conic.

You can modify the rho value in the Status Line and press ENTER (Windows) or RETURN (Macintosh) to accept the new rho value. Rho values must be between .501 and .999. A rho value of .501 will create a straight line. A rho value of .999 will create a conic with a 90° angle.

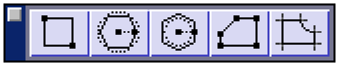
You can create a copy of the last conic by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking to indicate the start point. The copy is placed on the current work layer.

The Status Line allows you to specify the Rho value of the conic.

Geometric Characteristics

A 4-point conic is created by placing four points; the start point, end point, slope control point and shoulder point and is made up of the following characteristics: Rho, End 1 (X, Y and Z values) and End 2 (X, Y and Z values). This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the conic and choose ***Window>Edit Objects*** or double-click on the conic.

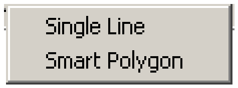
Polygon Tools



These tools draw rectangles, inscribed polygons, circumscribed polygons, arbitrary polygons and polygons from curves. The polygon is drawn with the current pen specifications for color, weight and pattern.

For each tool you can enter values in the Status Line to define a polygon, either before or after you create the polygon. If you enter the values after you select the tool but before you create the polygon, your first click in the drawing area automatically registers all Status Line values. If you enter values in the selected Status Line data field after creating the polygon and while the polygon is still selected, pressing ENTER (Windows) or RETURN (Macintosh) updates the polygon to reflect the new values.

Every polygon tool but the **Polygon from Curves** tool, provides two options for creating polygons, *Single Line* and *Smart Polygon*. These options are available through a pull-down menu in the Message Line.



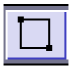
Polygons created with the *Single Line* option, are composed of individual lines that can be modified independently. These polygons can be filleted/chamfered, extruded, revolved, offset or relimited.

Polygons created with the *Smart Polygon* option are true polygons, whose width and height can be modified. These objects cannot be filleted/chamfered, extruded, revolved, offset, or relimited with the 2D wireframe tools however you can perform 3D operations such as extrudes, blends and revolutions on these entities. To perform these operations on a smart polygon with 2D wireframe tools convert it into individual lines by choosing ***Edit>Change Object Type*** and select the line option. See Chapter 24, “Editing Commands,” for more information.

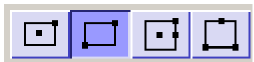
The tools explained here include:

- **Rectangle**
- **Inscribed Polygon**
- **Circumscribed Polygon**
- **Arbitrary Polygon**
- **Polygon from Curves**

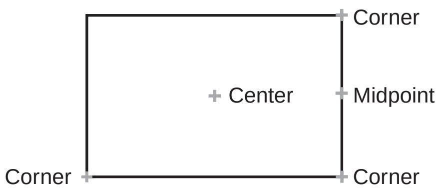
Rectangle Tool



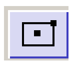
When you select the **Rectangle** tool, a subpalette appears in the Message Line containing four tools for creating rectangles, Polygon by center & point, Polygon by diagonals, Polygon by center & axis and Polygon by major & minor axes



The rectangle tools create rectangles from the selected points as directed by the specific tool.



Center/Corner Rectangle Tool



This tool draws a rectangle, using the center and corner you specify.

Using the Center/Corner Rectangle Tool

1. Select the **Rectangle** tool from the palette.

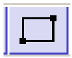
2. Select the **Center/Corner Rectangle** tool in the Message Line. The Message Line reads: *Rectangle: Pick center point of rectangle. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*].*
3. Pick the center point of the rectangle. As you move the cursor, a rubberband image appears.
4. Pick rectangle corner.

With the rectangle still selected, you can change the width and height of the rectangle by entering the values in the Status Line fields and pressing ENTER (Windows) or RETURN (Macintosh). You can also change the width and the height of a rectangle in the Edit Objects dialog box.

You can create a copy of the last rectangle by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking where you want the upper-left corner.

The Status Line allows you to specify the X, Y and Z coordinates of the first point, as well as the width and height of the rectangle. Width is the active status field.

Diagonal Rectangle Tool



This tool draws a rectangle based on two corners along a diagonal.

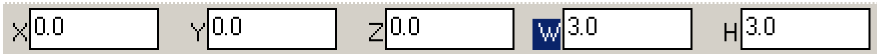
Using the Diagonal Rectangle Tool

1. Select the **Rectangle** tool from the palette.
2. Select the **Diagonal Rectangle** tool in the Message Line. The Message Line reads: *Rectangle: Pick first corner of rectangle. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*].*
3. Click one corner of the rectangle. As you move the cursor a rubberband image appears.
4. Click the opposite corner of the rectangle.

With the rectangle still selected, you can change the width and height of the rectangle by entering the values in the Status Line fields and pressing ENTER (Windows) or RETURN (Macintosh).You can also change the width and the height of a rectangle in the Edit Objects dialog box.

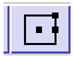
You can create a copy of the last rectangle by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking where you want the upper-left corner.

The Status Line allows you to specify the X, Y and Z coordinates of the first point, and the rectangle's width and height. Width is the active field.



You can create a square by aligning the second point on the 45° construction line. If the two points are on the vertical or horizontal axis, a straight line is drawn. If you want to draw a square from the center rather than opposite corners, use one of the other polygon tools, specifying four sides. You can't create a rectangle from the center.

Center/Axis Rectangle Tool



This tool draws a rectangle using the center, midpoint and corner.

Using the Center/Axis Rectangle Tool

1. Select the **Rectangle** tool from the palette.
2. Select the **Center/Axis** tool in the Message Line. The Message Line reads: *Rectangle: Pick center point of rectangle. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*].*
3. Click the center point of the rectangle.
4. Click the midpoint of the rectangle side. As you move the cursor a rubberband image appears.
5. Click the corner of the rectangle.

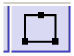
With the rectangle still selected, you can change the width and height of the rectangle by entering the values in the Status Line fields and pressing ENTER (Windows) or RETURN (Macintosh).

You can also change the width and the height of a rectangle in the Edit Objects dialog box.

You can create a copy of the last rectangle by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking where you want the upper-left corner.

The Status Line allows you to specify the X, Y and Z coordinates of the first point, as well as the width and height of the rectangle. Width is the active status field.

Major/Minor Axis Rectangle Tool



This tool draws a rectangle using the corners you specify.

Using the Major/Minor Rectangle Tool

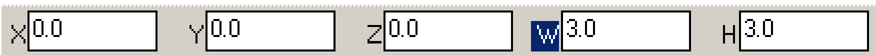
1. Select the **Rectangle** tool from the palette.
2. Select the **Major/Minor Axis** tool in the Message Line. The Message Line reads: *Rectangle: Pick center point of rectangle. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*]*.
3. Click the first corner of the rectangle.
4. Click the second corner of the rectangle. As you move the cursor, a rubberband image appears.
5. Click the third corner of the rectangle.

With the rectangle still selected, you can change the width and height of the rectangle by entering the values in the Status Line fields and pressing ENTER (Windows) or RETURN (Macintosh).

You can also change the width and the height of a rectangle in the in Edit Objects dialog box.

You can create a copy of the last rectangle by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking where you want the upper-left corner.

The Status Line allows you to specify the X, Y and Z coordinates of the first point, as well as the width and height of the rectangle. Width is the active status field.

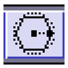


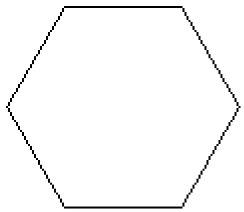
Geometric Characteristics

A rectangle polygon is created by picking the appropriate points as directed by the selected tool in the rectangle subpalette. A rectangle is made up of the following characteristics according to the Edit Objects dialog box: Length and Width and includes the option to Show Frame and Show Fill.

Show Frame refers to the display of the polygon. Show Fill refers to the placement of a fill within the polygon. Setting Show Frame to No, turns off the wireframe display. (If the wireframe is not displayed and does not contain a fill, your geometry will become invisible when your click Apply.) Setting Show Fill to Yes, places a fill within the polygon, based on the default pattern and fill currently selected in the Pen menu. This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the polygon and choose ***Window>Edit Objects*** or double-click on the polygon.

Inscribed Polygon Tool

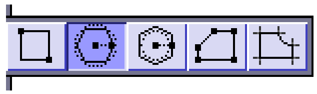




This tool creates a polygon where the radius of the circumscribing circle determines the location of the polygon's vertices.

The default polygon is a hexagon, but you can specify the number of sides in the Status Line. The Status Line shows a diameter for the circle, the standard way of describing a polygon inscribed in a circle.

Using the Inscribed Polygon Tool



1. Select the tool. The Message Line reads: *Inscribed Polygon: Pick center of polygon. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*].*
2. Click the center of the polygon. The Message Line reads: *Inscribed Polygon: Pick vertex of polygon*. As you move your cursor a rubberband polygon appears guiding your construction.
3. Pick a point on the circumference of the circumscribing circle to complete the polygon.

You can create a copy of the last inscribed polygon by holding down the CTRL (Windows) or OPTION (Macintosh) key and clicking where you want the center.

The Status Line allows you to specify the X, Y and Z coordinates of the center, the diameter of the circle defining the polygon, and the number of sides. Diameter is the active Status Line selection, and the default number of sides is six.

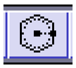
Geometric Characteristics

An inscribed polygon is created by picking the center and the vertex of the polygon. It is made up of the following characteristics according to the Edit Objects dialog box: Diameter, Sides, Scribe Type and includes the option to Show Frame and Show Fill.

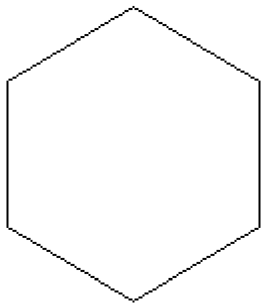
Scribe Type refers to circle type used to define the polygon. Choosing Circle Outside places the polygon within the circle, defining an inscribed polygon. Choosing Circle Inside places the polygon on the outside of the circle, defining a circumscribed polygon.

Show Frame refers to the display of the polygon. Show Fill refers to the placement of a fill within the polygon. Setting Show Frame to No, turns off the wireframe display. (If the wireframe is not displayed and does not contain a fill, your geometry will become invisible when your click Apply.) Setting Show Fill to Yes, places a fill within the polygon, based on the default pattern and fill currently selected in the Pen menu. This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the polygon and choose ***Window>Edit Objects*** or double-click on the polygon.

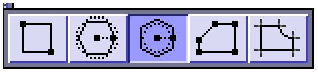
Circumscribed Polygon Tool



This tool draws a polygon in which the radius/diameter of the circle determines the midpoint of the sides. The default shape is a hexagon but you can specify the number of sides in the Status Line. Notice that the Status Line shows a diameter for the circle, the standard way of describing a polygon circumscribed around a circle.



Using the Circumscribed Polygon Tool



1. Select the tool. The Message Line reads: *Circumscribed Polygon: Pick center of polygon*. [*Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)].
2. Click the center of the polygon and the midpoint of one of the sides (as directed by the Message Line).

   After you place the first point, a rubberband polygon appears guiding construction.

You can create a copy of the last circumscribed polygon by holding down the CTRL (Windows) or OPTION (Macintosh) key and clicking where you want the center.

The Status Line allows you to specify the X, Y and Z coordinates of the center, the diameter of the circle defining the polygon, and the number of sides. The default number of sides is six. Diameter is the default Status Line selection.

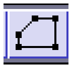
Geometric Characteristics

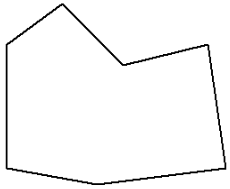
A circumscribed polygon is created by picking the center and the midpoint of the polygon side. It is made up of the following characteristics according to the Edit Objects dialog box: Diameter, Sides, Scribe Type and includes the option to Show Fill and Show Frame.

Scribe Type refers to circle type used to define the polygon. Choosing Circle Outside places the polygon within the circle, defining an inscribed polygon. Choosing Circle Inside places the polygon on the outside of the circle, defining a circumscribed polygon.

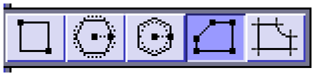
Show Frame refers to the display of the polygon. Show Fill refers to the placement of a fill within the polygon. Setting Show Frame to No, turns off the wireframe display. (If the wireframe is not displayed and does not contain a fill, your geometry will become invisible when your click Apply.) Setting Show Fill to Yes, places a fill within the polygon, based on the default pattern and fill currently selected in the Pen menu. This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the line and choose ***Window>Edit Objects*** or double-click on the polygon.

Arbitrary Polygon Tool





This tool draws a polygon with the number of sides determined by the points you choose.



Using the Arbitrary Polygon Tool

1. Select the tool. The Message Line reads: *Arbitrary Polygon: Pick first point. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*]*.
2. Click the next point (as directed by the Message Line).
3. Continue clicking points to complete the desired polygon. Single click the last point to close the polygon at its starting point. (If you try to double-click at this point you will receive an error message.)

   You can also double-click to place the last point (that is not the starting point of the polygon and your Designer Elements program will close the polygon automatically.

You can create a copy of the polygon by holding down the CTRL (Windows) or OPTION (Macintosh) key and clicking where you want the center.

The Status Line allows you to specify the X, Y and Z coordinates of the first point. X is the default Status Line selection.

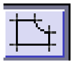
Geometric Characteristics

An arbitrary polygon is created by placing the desired points for the polygon. It is made up of the following characteristics according to the Edit Objects dialog box: Defining Point and the options to Show Fill and Show Frame.

Defining Point refers to the chosen’s point X, Y and Z location. The section below Defining Points lists the points with the X, Y and Z location. The selected point is displayed in the Defining Point fields. Each point can be edited individually.

Show Frame refers to the display of the polygon. Show Fill refers to the placement of a fill within the polygon. Setting Show Frame to No, turns off the wireframe display. (If the wireframe is not displayed and does not contain a fill, your geometry will become invisible when your click Apply.) Setting Show Fill to Yes, places a fill within the polygon, based on the default pattern and fill currently selected in the Pen menu. This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the polygon and choose ***Window>Edit Objects*** or double-click on the polygon.

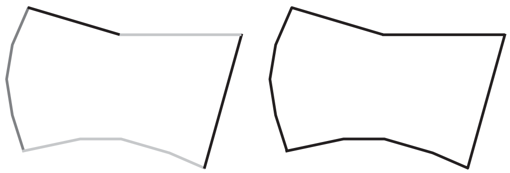
Polygon from Curves Tool



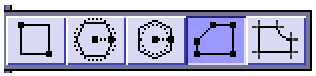
This tool creates a polygon from the curves you choose. The curves do not have to be connected to create a polygon. If these curves are not connected, the tool adds curves to complete the polygon. The polygon created when a curve is missing depends on the order of the curve selection.

Once the polygon is created a parent/child relationship has been established between the original curves and the polygon. The original curves selected are still available for editing and any change made will affect the polygon. Due to this same parent/child relationship, in order to move the polygon and retain the relationship, you must select the curves and the polygon. You can break the relationship by selecting them and choosing ***Edit>Remove Links***. See Chapter 24, “Editing Commands” for more information.

The graphic below on the left represents five different curves. The graphic on the right is the one polygon created after using the tool.



Using the Polygon from Curves Tool



1. Select the tool. The Message Line reads: *Polygon from Curves: Select a collection of curves [Shift = Extend]*.
2. Select all the desired curves. If you are selecting the curves individually, hold down the SHIFT key before beginning your selection.

   A polygon is created from those curves.

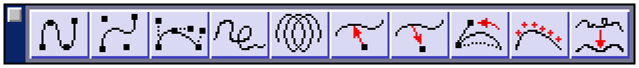
There are no entries in the Status Line.

Geometric Characteristics

Using this tool a polygon is created by selecting specific curves. No new geometry is created. The following characteristics are listed in the Edit Objects dialog box: Show Fill and Show Frame.

Show Frame refers to the display of the polygon. Show Fill refers to the placement of a fill within the polygon. Setting Show Frame to No, turns off the wireframe display. (If the wireframe is not displayed and does not contain a fill, your geometry will become invisible when your click Apply.) Setting Show Fill to Yes, places a fill within the polygon, based on the default pattern and fill currently selected in the Pen menu. This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the polygon and choose ***Window>Edit Objects*** or double-click on the polygon.

Spline Tools



The spline tools in the main tool palette create *NURBS* (Non-Uniform Rational B-Splines). These NURB splines are smooth curves created by a complex mathematical formula.

NURB splines provide designers with two interrelated functions. First, curvature continuity remains intact when the curve is changed. Kinks don't develop as the spline is altered. Second, NURB splines provide localized control of a complex curve by changing the control points.

Tech Note:

Your Designer Elements program creates B-Splines of the third and fourth order.

These properties are essential in aerodynamic designs. Air molecules moving over a wing surface must flow smoothly for maximum aero-dynamic lift. If the surface does not maintain curvature continuity, the air molecules separate from the wing surface and cause a vacuum. Such a vacuum causes an eddy as the molecules try to fill it. This disruption of air flow increases the drag, which is not a part of an effective design.

Complete curvature continuity also improves styling. The appearance of a car is one of the major sales factors. The potential buyer would not be impressed if the showroom lights' reflection on the car rippled and wavered. It is complete curvature continuity that makes a smooth reflection.

Localized control of complex curves allows you to make minor modifications without adversely affecting the shape. For example, if a new, bigger engine wouldn't fit under a perfectly-designed hood, you could use a NURB spline to raise the center of the hood without changing the basic design. NURB splines are also valuable for injection mold designs to eliminate the swirl of plastic as it is injected into the mold. Such designs provide better surface finishes and allow thinner cross-sections in the die.

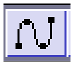
The spline creation tools are the **Through-Points B-Spline** tool, **Vector Spline** tool, **Bezier Spline** tool and Sketch Spline tool. At the most basic level these splines are the same. For the Through-Points B-Spline and the Bezier Spline, your Designer Elements program has the additional step of interpolating the points to calculate a control polygon which is hidden from you. For the Vector Spline the interpolation step is eliminated because you define the control polygon to which you have access.

Spline control points automatically lock after placement. You can select one point and move it without affecting the other control points.

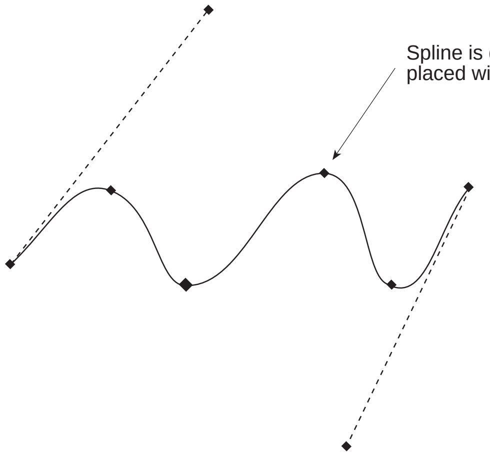
You can enter values in the Status Line to define a spline either before or after you create the spline. If you enter the values after you select the tool but before you create the spline, your first click in the drawing area automatically registers all Status Line values. If you enter values in the selected Status Line data field after creating the spline and while the spline is still selected, pressing ENTER (Windows) or RETURN (Macintosh) updates the spline to reflect the new values. The tools explained here include:

- **Through-Points B-Spline**
- **Vector Spline**
- **Bezier Spline**
- **Sketch Spline**
- **Helix Curve**
- **Add Control Point**
- **Remove Control Point**
- **Modify Slope**
- **Elevate Curve**
- **Fair Curve Spline**

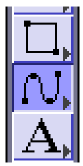
Through-Points B-Spline Tool



The **Through-Points B-Spline** tool draws a spline through your specified points. Use this tool when you want the B-Spline to interpolate a collection of points. The interpolation algorithm is based on predefined B-Spline blending functions, control point locations and imposing curvature continuity across the curve length. This results in a smooth spline created through your points.



Using the Through-Points B-Spline Tool



1. Select the tool. The Message Line reads: *Through-Points B-Spline: Pick control points. (End = ESC, Double-click)*.
2. Click the points for the vectors of the spline.
3. Double-click the last point or hit the ESC key.

The Status Line shows the X, Y and Z coordinates for each point as it is placed.

If you create a spline that crosses over itself, the Drafting Assistant will not be able to find that intersection. This is by design.

Geometric Characteristics

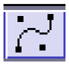
A Through-Points B-Spline is created by picking the desired points. A spline is made up of the following characteristics according to the Edit Objects dialog box: Start and End Angles (relative to work plane) and Defining Point.

Angles refer to the degrees of the tangent relative to the work plane for the spline. You can change the Start and End Angles by clicking within the check box and entering the angle.

Defining Point refers to the X, Y and Z location of the active point. X, Y and Z values for each point can be edited individually using these fields.

This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the spline and choose ***Window>Edit Objects*** or double-click on the spline.

Vector Spline Tool

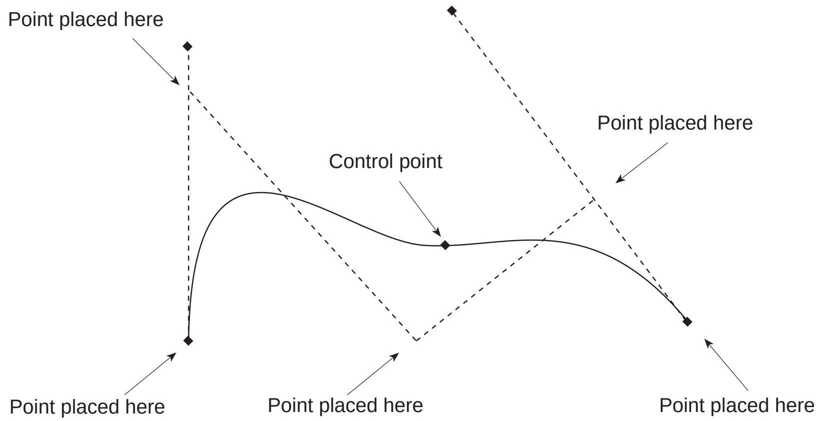


Tip:

: With the **Vector Spline** tool all original points used to create the spline will display when the spline is selected.

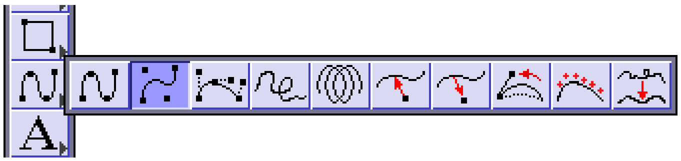
This tool draws a spline using vectors determined by the points you specify. The **Vector Spline** tool uses each point that you place as the vertex of a vector for the spline it creates. Vector splines are smooth curvature, continuous B-Splines created from a control polygon that you define. The polygon influences the shape (position and tangencies) of the spline. Except for the start and end points, the control polygon does not necessarily lie on the spline.

Choosing ***Edit>Show Points*** will display all the control points used to define the spline and can be selected to edit the spline. This provides the additional benefit of being able to smooth your spline to eliminate inflections. To determine whether or not your spline has inflections, choose ***Verify>Curvature***. An inflection is located where the curvature changes from one side of the spline to the other.



Using the Vector Spline Tool

1. Select the tool. The Message Line reads: *Vector Spline: Pick control points (End = ESC, Double-click)*.



2. Click the points for the vectors of the spline.
3. Double-click the last point.

This Designer Elements program uses these vectors to calculate the control points (two fewer than the number of vertices you specify). The spline is tangent to the first and last vectors and passes through the calculated control points.

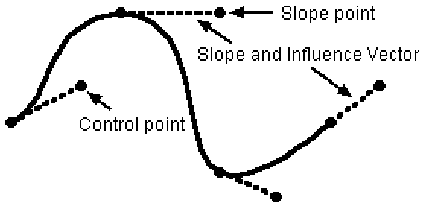
The Status Line shows the X, Y and Z coordinates of each point as they are placed.

Geometric Characteristics

A vector spline is created by picking control points and is made up of the one characteristic according to the Edit Objects dialog box: the Defining Point (X, Y and Z location). Defining Point refers to the active point's X, Y and Z location. Each point can be edited using the X, Y, Z fields. This information is listed in the Edit

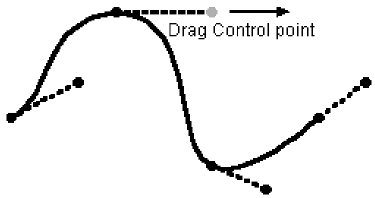
Objects dialog box under the Geometry tab. To display the dialog box, select the spline and choose ***Window>Edit Objects*** or double-click on the spline.

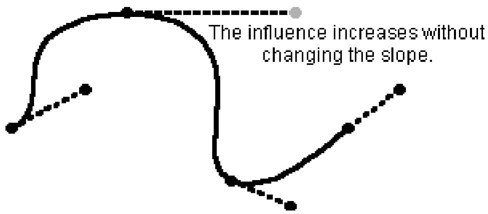
Bezier Spline Tool



This tool creates curvature continuous (C2) splines through a collection of user defined points. At each control point, you can control the direction and the magnitude of the tangent. After creating a spline with this tool, you can move the control points, change the slope (direction) and the influence of a control point using the spline vector.

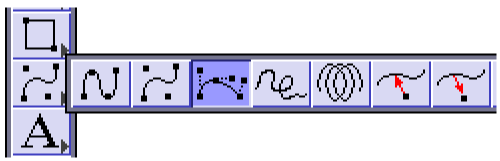
The influence of a control point is determined by the length of the control point's vector. The longer the vector, the greater the influence a control point has on the spline. You can adjust the length of control point vector but maintain the slope. The left graphic here shows a Bezier spline, a selected control point and the direction it will be moved to change the influence but maintain the slope. The right graphic shows the spline with a longer vector.





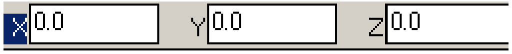
You can also use the **Add Control Point**, **Remove Control Point** and **Modify Slope** tools to change the spline.

Using the Bezier Spline Tool



1. Select the tool. The Message Line reads: *Bezier Spline: Pick control points (End = ESC, Double-click)*.
2. Click the points of the spline.
3. Double-click the last point.

The Status Line shows the X, Y and Z coordinates of each point as they are placed.

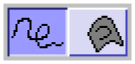


Geometric Characteristics

A Bezier spline is created by picking control points and is made up of the following characteristics according to the Edit Objects dialog box: the Defining Point (X, Y and Z location) and the points list. This list includes the spline control points and the slope points (if displayed). Display the slope points by selecting the spline and choosing ***Edit>Show Points***. Defining Point refers to the active point's X, Y and Z location.   Each point can be edited using the X, Y, Z fields. This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the spline and choose ***Window>Edit Objects*** or double-click on the spline.

Sketch Spline Tool

The **Sketch Spline** tool samples points as the cursor moves and creates a smooth spline throught the sampled data. Once selected this tool has two options located in the Message Line, **Sketch Spline** tool and **Sketch on a Surface** tool.



Using the Sketch Spline Tool

1. Select the tool. The Message Line reads: Sketch Spline: Press and drag mouse (Option=AutoClose).
2. Click and drag the mouse on the screen.
3. Release the mouse button to create the sketch spline.

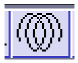
Using the Sketch on a Surface Tool

1. Select the tool. The Message Line reads: Sketch on Surface: Select surface or solid to sketch on (Option=AutoClose).
2. Select the surface or solid to sketch on.
3. Click and drag the mouse over the selected surface, keeping the points on the surface.
4. Release the mouse button to complete the sketch.

Tech Note:

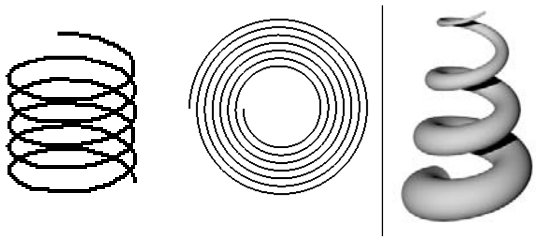
If you want to sweep a surface on a helix, the helix must be created using this tool. See Chapter 16 for information on using the **Sweep Surface** tool.

Helix Curve Tool



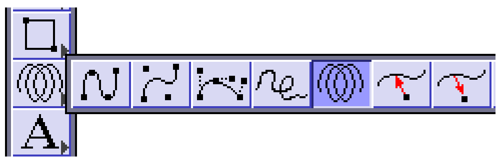
The **Helix Curve** tool draws a curve based on its start point, end point, *Pitch*, diameter, length and draft angle. The endpoint defines the length and orientation of the helix. Pitch is the distance between helix peaks (two adjacent turns of the helix). The number of turns in the helix is equal to the length of the helix divided by the pitch. The draft angle must be a value less than 90°.

You can create a standard helix (left graphic below), a spiral (middle graphic below) by entering zero for the helix length, or a swept solid using a helix with a draft as the sweep path (right graphic below).



Using the Helix Curve Tool

1. Select the **Helix Curve** tool. The Message Line reads: *Helix Curve: Pick start and end of helix [Ctrl* (Windows) or *Option* (Macintosh) = *Left handed helix]*.



2. The Status Line contains the Pitch, Diameter, Length and Draft Angle data fields.

Enter the desired values in the appropriate fields. Tab between fields.

3. Pick the start and end point of the helix. A right handed helix is created.

While the helix is still selected you can change the Status Line values. Type the new values and press ENTER (Windows) or RETURN (Macintosh) and the helix updates.

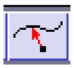
Geometric Characteristics

A helix is created by picking the start and end point and specifying the pitch and diameter. A helix is made up of the following characteristics according to the Edit Objects dialog box: the Diameter, Pitch, Length and Draft Angle.

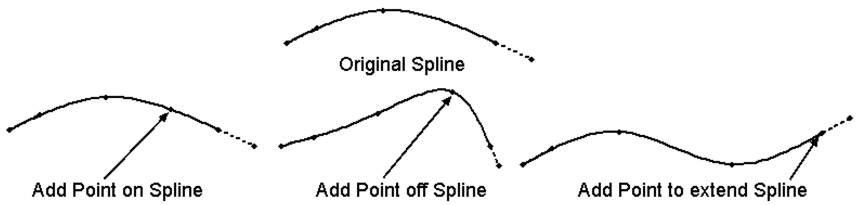
You also have a check box to change the helix direction. By default, using this tool creates a right-handed helix. You can change the helix to a left-handed helix by deselecting the *Right handed helix* option and clicking Apply or Close. If you already created a left-handed helix, the box is not checked.

This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the helix and choose ***Window>Edit Objects*** or double-click on the helix.

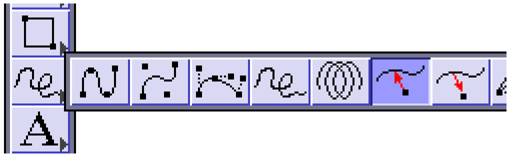
Add Control Point Tool



This tool allows you to add another control point to an existing spline both within the spline and outside of the spline to extend it.



Using the Add Spline Control Point Tool

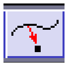


1. Select the tool. The Message Line reads: *Add Control Point: Select the spline to add point to (Shift = Extend)*.
2. Click on the spline to which you want to add a point.
3. Pick the desired location(s) for the new control point(s).

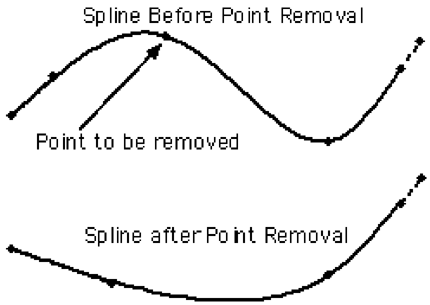
Note: To see the new control point(s), as well as the existing control points, you must first select the spline and then choose ***Edit>Show Points*** (or use the Edit Objects dialog box to specify whether the Control points are visible or hidden).

There are no Status Line entries for this tool.

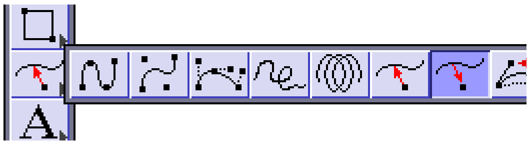
Remove Control Point Tool



This tool allows you to remove a control point from an existing spline. A spline must have more than two control points to use this tool.



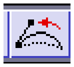
Using the Remove Control Point Tool



1. Select the tool. The Message Line reads: *Remove Control Point: Select the spline point to remove (Shift = Extend)*.
2. If the control points of the spline are not displayed, select the spline and choose ***Edit>Show Points***.
3. Pick the control point you want to remove.

There are no Status Line entries for this tool.

Modify Slope Tool

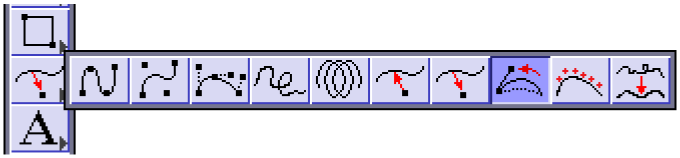


This tool allows you change the endpoint slope of an existing spline created using the **Through-Points B-Spline** tool. You have four options for changing the slope: *Free*, *Reference*, *Reverse* and *Explicit*.

Free	Frees the spline slope and returns it the original slope as defined by this Designer Elements program for the chosen slope control point. This is helpful if you've adjusted the slope of a spline in numerous locations along the spline and want to return to the spline that was originally created.
Reference	Uses the slope of other curves or surfaces as a reference to make the selected curves tangent.
Reverse	Reverses a curve 180°.
Explicit	Allows you to set the exact slope of the curve relative to the work plane (between 0° and 360°). A slope of 0° will make the curve tangent to the plane. Choose ***Planes>Show Work Plane*** when using this option to assist you.

Using the Modify Slope Tool - Free Option

1. Select the tool. The Message Line reads: *Modify Slope: Select spline slope(s) to free (Shift = Extend)*.



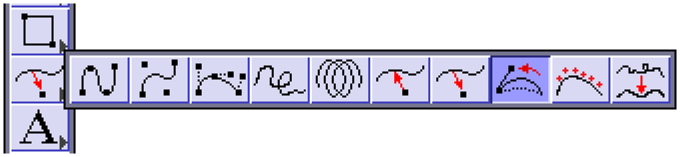
2. Choose the *Free* option in the pull-down menu of the Message Line.
3. If the control points of the spline are not displayed, select the spline and choose ***Edit>Show Points***.
4. Pick the spline slope control endpoint you want to free.

   The slope control endpoint returns to its original position.

There are no Status Line entries for this tool.

Using the Modify Slope Tool - Reference Option

1. Select the tool. The Message Line reads: *Modify Slope: Select the spline end to set slope at (Shift = Extend)*.

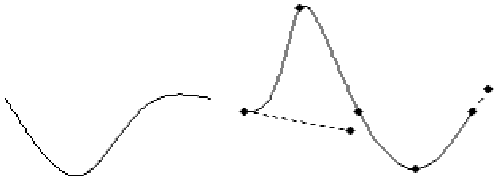


2. Choose the *Reference* option in the pull-down menu of the Message Line.
3. If the control points of the spline that you want modified are not displayed, select the spline and choose ***Edit>Show Points***.
4. Pick the spline want to modify. In the graphic here, it's the spline displaying points.

5. Pick the curve, surface or solids whose slope you want to reference.

   The icon now becomes a target icon.

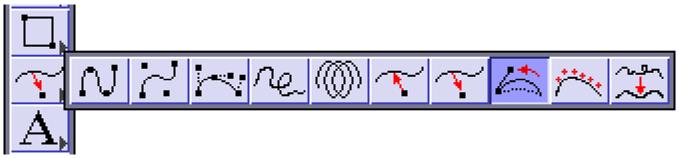
6. Click on the endpoint of the curve you want to modify, (The curve and the reference object do not have to connect.) The curve slope adjusts.



There are no Status Line entries for this tool.

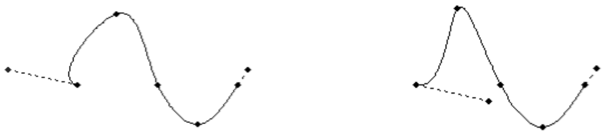
Using the Remove Control Point Tool - Reverse Option

1. Select the tool. The Message Line reads: *Modify Slope: Select spline slope point to reverse (Shift = Extend)*.



2. Choose the *Reverse* option in the pull-down menu of the Message Line.

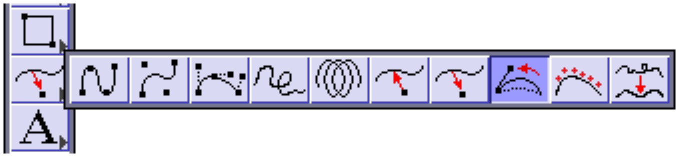
3. If the control points of the spline are not displayed, select the spline and choose ***Edit>Show Points***.
4. Pick the spline slope control endpoint you want to reverse, as in the left graphic below. The slope control endpoint reverses 180° (right graphic).



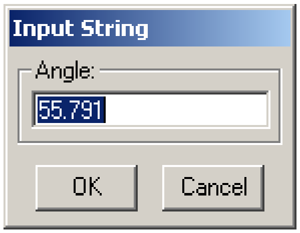
There are no Status Line entries for this tool.

Using the Modify Slope Tool - Explicit Option

1. Select the tool. The Message Line reads: *Modify Slope: Select spline slope point for explicit set (Shift = Extend)*.



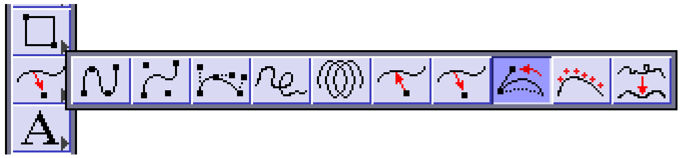
2. Choose the *Explicit* option in the pull-down menu of the Message Line.
3. If the control points of the spline are not displayed, select the spline and choose ***Edit>Show Points***.
4. Pick the spline control endpoint you want to set. An Input String dialog box appears.
5. Enter the Angle for the slope in the data field and click OK. The dialog box closes and the slope of the spline is adjusted. (Click Cancel if you want to close the dialog box without changing the angle value.)



There are no Status Line entries for this tool.

Using the Modify Slope Tool - Free Magnitudes Option

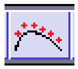
1. Select the tool. Choose the Free Magnitudes option in the pull down menu. The Message reads: Modify Slope: Select Spline Magnitude to free (Shift=*Extend)*.



2. If the control points of the spline are not diplayed, select the spline and choose **Edit>Show Points**.
3. Select the bezier curve control point you want to free.

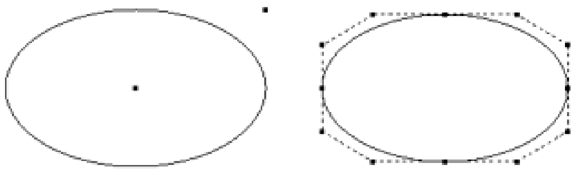
The Bezier slope resets to an optimally calculated slope.

Elevate Curve Tool

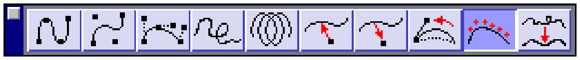


This tool elevates the degree of a curve, introducing more control points. These control points can be used to modify the curve.

Curves are defined by NURB polynomial equations. The most basic question possible is used to define the curve. Shapes defined by higher degree polynomials have more control points than those defined by lesser degree polynomials. This toolraises the degree level for a shape, providing more control points for curve manipulation. Curves can be elevated up to the 22nd degree. However, it is recommended that you never elevate a curve highter than 9 degrees. The left side of the graphic below shows the original ellipse. The right side of the graphic below shows the ellipse elevated one degree.



Using the Elevate Curve Tool



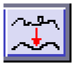
1. Select the Elevate Curve tool. The Message Line reads: Elevate Curve: Pick curve to elevate [Shift=Extend].

2. Select the curve to elevate.

The curve elevates one degree.

There are no entries in the Status Line.

Fair Spline Tool



The **Fair Spline** command provides a tool to globally smooth a curve. Fairing a spline optimally moves control vertices to locations that minimize large curvature variations. Curves eligible for fairing are limited to vector splines. It's helpful to watch the control vertices move as you fair the spline with the **Edit>Show Points** command. You can also use the **Verify>Curvature** tool to visually inspect the curvature graph of a curve.

Using the Fair Spline Tool

1. Select the tool. The Message Line reads: *Fair Spline: Select curve to fair.*

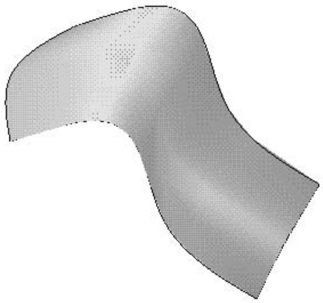
2. Click on the spline which you want to fair.

The status line displays a move tolerance value for the tool. The move tolerance is the maximum distance a spline control vertex will be allowed to move towards its optimal position.

Introduction to Surface Modeling

A surface/mesh model is a more complete and less ambiguous representation than a wireframe model. Surface models take the representation of an object one step beyond wireframe models by defining the area between selected boundaries with smooth equations. Mesh models also go beyond wireframe models but are defined by nodes or vertices. Consequently, surface and mesh modeling provides a means for the designer to create complex shapes such as cars, ships and aircraft.

Your Designer Elements program surfaces are based on Spatial Technology's ACIS Geometry Engine. ACIS surfaces are composed of Non Uniform Rational B-Splines (NURBs). NURBs provide a highly precise yet flexible mathematical definition for modeling even the most demanding free form shapes. Surfaces generated with your Designer Elements program are suitable for precise geometric analysis and can be passed to computer aided manufacturing applications that support SAT or IGES file formats.

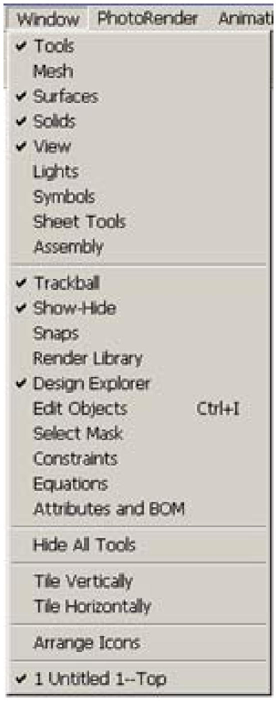


The topics discussed in this chapter include:

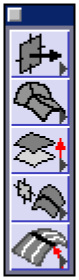
- **Surface Modeling** tools
- Surfaces, Selection and Display
- Surfaces and Instances
- Cutting and Pasting Surfaces
- Exporting Surfaces
- Object Types and Edit Objects

Surface Modeling Tools

The surface modeling tools are accessible from the **Surfaces** tool palette. The **Surfaces** tool palette automatically displays when you first launch your Designer Elements Program. If you close the palette and need to display the palette again, choose ***Window>Surfaces***.



The palette will appear under the main tool palette on the left side of your screen.

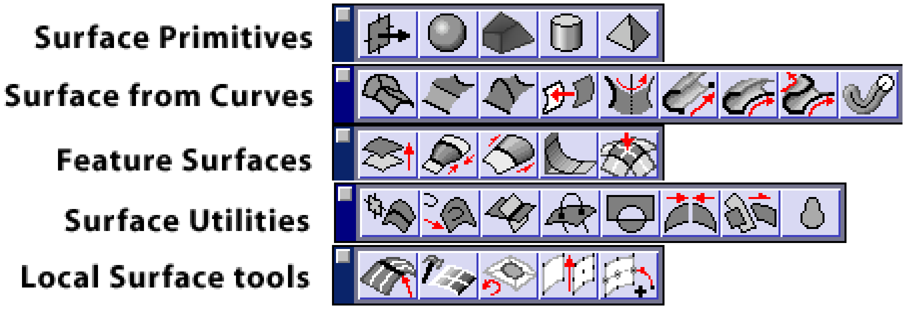


You can save the open status of the palette and its location, simply by quitting your Designer Elements program. The next time you launch the program, the palette will be open and placed in the last location.

Each icon in the palette is the first in a subpalette of tools grouped together with a similar purpose. From top to bottom they include:

- **Surface Primitive** tools
- **Surface From Curves** tools

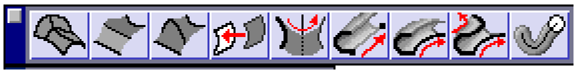
- **Feature Surface** tools
- **Surface Utility** tools
- **Local Surface** tools



Surface Primitive Tools

The Surface Primitives tool palette provides tools for quickly creating surfaces from simple shapes and parameters. This tool palette provides a means to create **Infinite Planes**, **Speheres**, **Blocks**, **Cylinders** and **Pyramids**. (For more information on these tools see chapter 16.)

Surface From Curves Tools

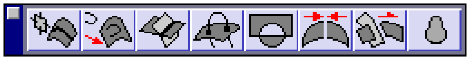


Use the Surfaces From Curves tools to construct surfaces built upon curves. This tool palette is home to the **Net**, **Ruled**, **Skin**, **Cover**, **Revolution** (lathe), **Extruded**, **Swept**, **Swept 2 rail** and **Pipe** surface tools.(For more information on these tools see chapter 16.)

Feature Surface Tools

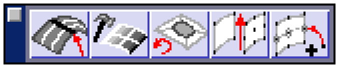
The Feature Surface tool palette constructs surfaces built upon other surfaces. The tools in this palette include **Offset**, **Loft**, **Draft**, **Fillet** and **Tangent Cover Surface**. (For more information on these tools see chapter 16.)

Surface Utilites Tools



After you've constructed or imported surfaces into your model, you may need to refine or combine the surfaes with other entities. Your Designer Elements program provides several tools which make these tasks easy to complete. The Surface Utilities palette is home to tools such as **Entity Intersections**, **Projections**, **Joining**, **Trimming** and **Silhouette Curves** commands. (For more information on these tools see chapter 21.)

Local Surface Tools



This tool palette provides commands to modify the internal shape or definition of a surface. Example tools located in this palette include, **Surface Slope Matching**, **Rebuilding**, **Untrimming**, **Degree Elevation** and **Knot Insertion**. (For more information on these tools see chapter 21.)

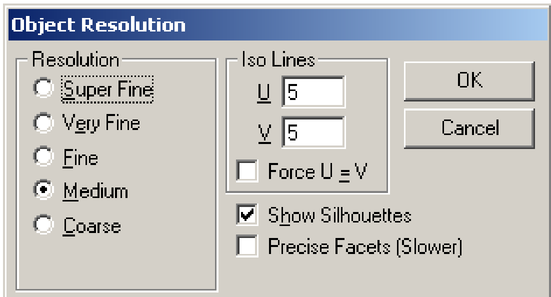
Surfaces, Selection and Display

You can select a surface by clicking anywhere within its boundary. However, your Designer Elements program will find the surface fastest if you select the surface's edge. A surface is displayed by its boundary edges and any existing interior holes. You can increase a surface's edge resolution with the *Change Resolution* command in the Edit menu.

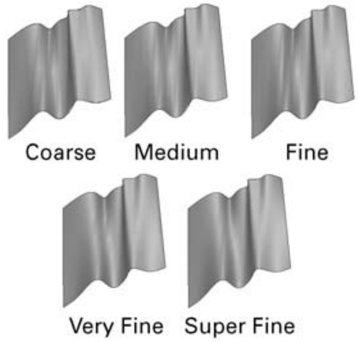
Changing an Object's Resolution

1. Select the object.
2. Choose ***Edit>Change Resolution***.

   The following Object Resolution dialog box appears.



You can choose one of five resolutions, Super Fine, Very Fine, Fine, Medium and Coarse. Select the desired resolution and click OK to close the dialog box. The object changes accordingly. The graphic illustrates each option.



3. Enter the desired number of Iso lines. Check the *Show Silhouettes* box if desired.

   The *# Iso Lines* allow you to control the *isopram* lines drawn for a surface. These Iso (isopram) lines are constant parameter curves that lie on a surface, typically defined in parameter space. The parameter space coordinate system uses *U* and *V* coordinates. A 0 (zero) in both fields turns off Iso lines. The appropriate U/V values may enhance the visual appearance of the surface at the expense of drawing speed. The letters, U and V are space coordinate identifiers (U = horizontal, V = vertical) that are the industry standard. The Force U=V check box automatically sets equal number of U and V lines. Entering a new value in one automatically changes the other isoline value.

   The *Silhouette* check box controls the silhouette edges of objects. Silhouette edges are view dependent and can cause a significant reduction in drawing speed. A check in the box turns the silhouette on.

4. Click OK to close the dialog box and save the new settings.

Surface Associativity

Cobalt and Xenon Only

Surfaces are associative to the original curves used to create them. Changing the curve automatically changes the dependent surface. For example, if you create a skin surface from three arcs and change the radius of one of the arcs, the skin surface automatically updates to reflect the change. Use the ***Edit>Remove Links*** command to remove all associativity with the surface if you do not want curve changes to affect the surface. Another important issue to remember with the curve/surface associativity is that if the curve is deleted, the surface deletes as well. Since it is defined by the curve, it is no longer valid.

Cutting and Pasting Surfaces

When you cut a surface into the paste buffer, two sets of data get stored. The first set of data is a Designer Elements program object that will precisely paste into the current file or into another Designer Elements program drawing. The second set of data is a collection of display vectors in the bitmap format. You can paste this data into other Windows or Macintosh programs that support the clipboard.

Exporting Surfaces

Your Designer Elements program provides several methods for exporting surfaces into other applications. Our primary goal is for you to preserve as much data a possible during the export. Use the following export option order when exporting surfaces:

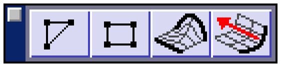
- ASIS (.SAT)
- Parasolids (.x_t) (Windows only)
- ProE (.g)(Windows only)
- STEP
- DXF with imbedded .sat
- IGES
- StereoLithography (.stl)(Mesh surfaces only)

Objects Types and Edit Objects

Every surface or mesh object you create with a Designer Elements program tool is defined by its own characteristics, and include geometric characteristics and attributes. These are displayed in the Edit Objects dialog box. While the characteristics listed in the Attributes tab are identical, the characteristics listed in the Geometry tab vary with each object.

Chapters 15 and 16 introduce you to the surface tools of your Designer Elements program. Included with every tool description is a list of the characteristics displayed for the object when Edit Objects is chosen. For information on using the *Edit Objects* command and the dialog box, see Chapter 24.

Mesh Creation Tools -



A mesh is a collection of planar elements typically with three or four sides. It is equivalent to the 3D Mesh element found in DXF files.

You can use meshes to model bodies that span areas and for calculating wetted areas and volume attributes. Although less accurate than surface modeling, mesh modeling uses less memory, and still provides considerable flexibility for those who prefer to design only with cross sections.

Unlike a surface, a mesh object is not defined by a mathematical formula but by nodes and 3D vertices. The resolution or precision of a mesh is determined by the number of vertices.

A mesh object can be used to create output to rapid prototyping machines and for compatibility with AutoCAD R12 files. A mesh object is helpful to people who want to control the exact number of facets created for each model.

Of the mesh tools available in your Designer Elements program, the **Mesh Curve** tool is the most powerful. With this tool, you can create sections and “skin” them together to obtain areas. Skins can be combined into bodies to calculate body area curves. Calculations of the area underneath the curve also yield the body’s volume.

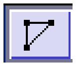
For each tool you can enter values in the Status Line to define a mesh, either before or after you create the mesh. If you enter the values after you select the tool but before you create the mesh, your first click in the drawing area automatically registers all Status Line values. If you enter values in the selected Status Line data field after creating the mesh and while the mesh is still selected, pressing ENTER (Windows) or RETURN (Macintosh) updates the mesh to reflect the new values.

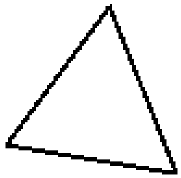
Your Designer Elements program supports four tools for creating mesh objects, **Mesh 3 Pts**, **Mesh 4 Pts**, **Mesh Curve** and **Mesh Extrude**. The topics explained in this chapter include:

- **Mesh 3 Pts** tool
- **Mesh 4 Pts** tool
- **Mesh Curve** tool
- **Mesh Extrude** tool
- Mesh and Rendering
- Mesh Notations

To access the **Mesh** tools go to ***Window>Mesh***.

Mesh 3 Pts Tool





The **Mesh 3 Pts** tool creates a flat plate triangular object from three points that you select.

Using the Mesh 3 Pts Tool

Tech Note:

To select a mesh object immediately after you create it, choose the **Selection** tool. Otherwise, you will continue to create new mesh objects.

1. Select the tool. The Message Line reads: *Mesh 3 Pts: Enter 3 locations for mesh*.

   If you would like to create the mesh in a plane different than the one currently set, change the plane before moving on to the next step.
2. Click three points to define the mesh boundary.

   The mesh is created.

There are no entries in the Status Line.

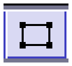
Geometric Characteristics

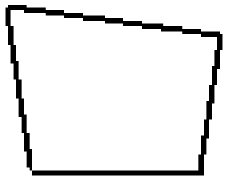
A 3 points mesh is created by picking the three desired points. A mesh is made up of the following characteristic according to the Edit Objects dialog box: the Defining Point for the X, Y and Z location.

Defining Point refers to the X, Y and Z location of the chosen point. The section below Defining Points lists the points with the X, Y and Z location. The selected point is displayed in the Defining Point fields. Each point can be edited individually using these fields.

This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the mesh and choose ***Window>Edit Objects*** or double-click on the mesh.

Mesh 4 Pts Tool





The **Mesh 4 Pts** tool creates a flat plate rectangular object from four points that you select.

Using the Mesh 4 Pts Tool

1. Select the tool. The Message Line reads: *Mesh 4 Pts: Enter 4 locations for mesh*.

   If you would like to create the mesh in a plane different than the one currently set, change the plane before moving on to the next step.

2. Click four points to define the mesh boundary.

   The mesh is created.

There are no entries in the Status Line.

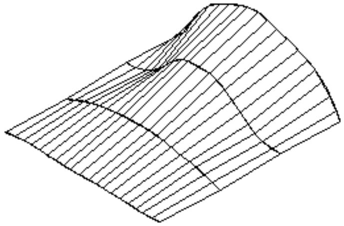
Geometric Characteristics

A 4 points mesh is created by picking the four desired points. A mesh is made up of the following characteristics according to the Edit Objects dialog box: the Defining Point for the X, Y and Z location.

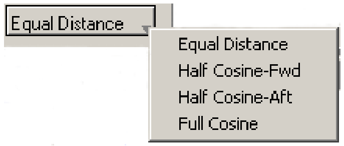
Defining Point refers to the X, Y and Z location of the chosen point. The section below Defining Points lists the points with the X, Y and Z location. The selected point is displayed in the Defining Point fields. Each point can be edited individually using these fields.

This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the mesh and choose ***Window>Edit Objects*** or double-click on the mesh.

Mesh Curve Tool



The **Mesh Curve** tool creates an *M* by *N* mesh from a collection of curves. M is the number of curves you select to define the mesh boundary. The mesh is created from the order in which you select them. N is number of segment lines you want displayed on the mesh. The graphic above shows a mesh curve with M = 4 (curves) and N = 20 (grid lines).



The **Mesh Curve** tool provides several options that control mesh spacing. You can pick them from the drop down list in the Message Line. The options include:

Equal Distance	Mesh points are spaced equally.
Half Cosine (Fwd)	Mesh points are spaced more closely at the forward end of the mesh, according to a half cosine.
Half Cosine (Aft)	Mesh points are spaced more closely at the back end of the mesh, according to a half cosine.
Full Cosine	Mesh points are spaced more closely at both ends of the mesh, according to a full cosine.

Using the Mesh Curve Tool

1. Select the tool. The Message Line reads: *Mesh Curve: Select curves in order of meshing [Shift=Extend].*
2. Select a mesh spacing option from the pull-down menu.
3. The Status Line displays the N data field which represents the number of mesh segments lines that will be distributed across the mesh.

If you want a different number of segments, enter the new value in the data field. The default value is 25.

4. Click two or more curves to define the mesh boundary. These curves should not be connected. If you need to select more than two curves, hold down the SHIFT key before you select the first curve.

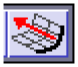
   Select the curves in the order that you want the mesh created.

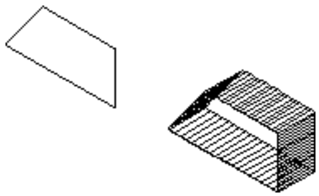
Geometric Characteristics

A mesh curve is created by selecting the desired curves. A mesh curve is made up of the following characteristics according to the Edit Objects dialog box: the Distribution setting (Equal, Half Cosine (Forward), Half Cosine (Aft) or Full Cosine) and Segments (N) which make up the mesh.

This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the mesh and choose ***Window>Edit Objects*** or double-click on the mesh.

Mesh Extrude Tool



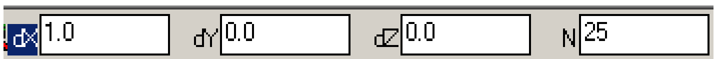


The **Mesh Extrude** tool sweeps a collection of curves along a vector, creating *N* mesh segments.

Using the Mesh Extrude Tool

1. Select the tool. The Message Line reads: *Mesh Extrude: Select objects to extrude [Shift=Extend]*.

2. The Status Line displays data fields allowing you to enter the delta values for the extrusion and the number of N segments for the mesh.



3. Click one or more curves to sweep. If you need to select more than one curve, hold down the SHIFT key before you select the first curve.

4. If you want a different value for the N segments than currently listed, select the field and type the new value. After you selected the curves, The Message Line reads: *Mesh Extrude: Enter two points to extrude*.

5. Click two points to set the direction and the distance of the extrusion. The points do not have to be on the selected curves.

   The mesh is created.

If you want to modify the mesh while it's still selected, type the value in the appropriate field tabbing between fields as necessary. Press ENTER (Windows) or RETURN (Macintosh) and the mesh curve will be modified.

Geometric Characteristics

An extruded mesh is created by selecting the desired curves and the extrusion values that define the distance and direction. An extruded mesh is made up of the following characteristics according to the Edit Objects dialog box: Vertex Count, Facet Count and Segments. This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the mesh and choose ***Window>Edit Objects*** or double-click on the mesh.

Mesh and Rendering

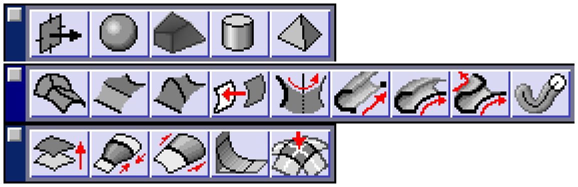
Mesh is displayed according to the Static Render setting in the Rendering Option dialog box. You can display the mesh as Wireframe, Flat, Gouraud, Gouraud w/ Edges, Phong, Phong w/Edges, Hidden and Hidden with Dimmed. See Chapter 33, “Rendering,” for an explanation of these options.

Mesh Notations

The following notations will be helpful as you begin creating mesh objects.

- Mesh objects are not surfaces. They are defined by nodes or 3D vertices.
- Since mesh objects are not surfaces, you cannot use the **Surface Editing** tools contained in the third set of tools within the **Surface** tool palette.
- You can convert surfaces and solids into meshes and convert a closed mesh back into a solid.

Surface Creation Tools



Your Designer Elements program has 11 tools for creating surfaces. Every tool but the **Infinite Plane** tool can display a surface with *Iso Lines*. Iso (or isopram) lines are constant parameter curves that lie on a surface (defined mathematically in parameter space). U/V Iso lines refer to parameter space coordinates where U and V are letter identifiers. U Iso lines run perpendicularly to V Iso lines on a surface.

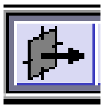
For each tool you can enter values in the Status Line to define a surface, either before or after you create the surface. If you enter the values after you select the tool but before you create the surface, your first click in the drawing area automatically registers all Status Line values. If you enter values in the selected Status Line data field after creating the surface and while the surface is still selected, pressing ENTER (Windows) or RETURN (Macintosh) updates the surface to reflect the new values.

Many of the tools use a two step process. The first step after choosing the tool may be to select one or more objects. The second step is to perform the editing operation. Once you the complete the first step (which may involve holding down the SHIFT key to select multiple objects), you cannot select more objects to include in the same operation. To select additional objects, reset the tool and start again.

The tools are explained in the order of the palettes and include:

- **Infinite Plane**
- **Sphere Primitive**
- **Block Primitive**
- **Cylinder Primitive**
- **Pyramid Primitive**
- **Net Surface**
- **Ruled Surface**
- **Skin Surface**
- **Cover Surface**
- **Revolved Surface**
- **Sweep 2pts Surface**
- **Sweep 1 Rail Surface**
- **Sweep 2 Rail Surface**
- **Tube Surface**
- **Offset Surface**
- **Lofted Surface**
- **Draft Surface**
- **Fillet Surface**
- **Tangent Surface**

Infinite Plane Tool



The *Infinite Plane* is the simplest surface supported by your Designer Elements program. An infinite plane is defined by a location and a normal. The plane surface is useful for generating cross-sections through meshes, surfaces, or solids.

The infinite plane has six creation methods listed in the pull-down menu of the Message Line.

X-Station Creates a plane with normal X=1, Y=0, Z=0.

Y-Station Creates a plane with normal X=0, Y=1, Z=0.

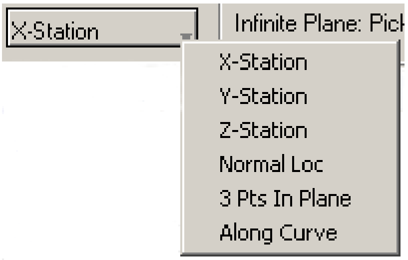
Z-Station Creates a plane with normal X=0, Y=0, Z=1.

Normal Location Creates an arbitrary plane by specifying the location and normal.

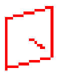
3 Pts in Plane Specifies a plane by three points. The normal is calculated from the three points.

Along Curve Lays an evenly spaced series of planes along a selected curve. The plane's normals lie parallel to the curve.

The choices above are accessed from the Message Line, in the pull down pictured below.



After creating an infinite plane, it is represented on the screen by the following icon:



Using the Infinite Plane Tool for X-,Y- and Z-Station Planes

1. Select the **Infinite Plane** tool.
2. Pick *X-*, *Y-* or *Z-Station* from the drop down list on the Message Line. The Message Line reads: *Infinite Plane: Pick location for plane.*

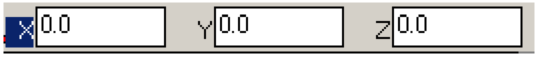
   If you would like to create the surface in a plane different from the one currently set, change the plane before moving on to the next step.

Tip:

You can determine the distance between two infinite planes by selecting the planes and choosing **Verify>Minimum Distance.**

3. Click the point on the screen where you want the plane. An icon for the plane appears at that location.

The Status Line displays the X, Y and Z location for the plane.



You can modify the location by typing new values in the appropriate fields. Tab to move from field to field. Pressing ENTER (Windows) or RETURN (Macintosh) and the plane location changes to reflect those values. All future changes must be made in the Edit Objects dialog box.

Using the Infinite Plane Tool to create a Normal and Location Plane

1. Select the **Infinite Plane** tool.
2. Pick *Normal Loc* from the drop down list on the message line. The Message Line reads: *Infinite Plane: Enter 3 points (1-Normal End, 2-Normal Start, 3-Loc)*.

   If you would like to create the surface in a plane different from the one currently set, change the plane before moving on to the next step.
3. Click the point where you want the plane's normal to end, the point where you want the plane's normal to start, and the point where you want the plane to be placed.

There are no entries in the Status Line.

Using the Infinite Plane Tool to create a Three Point Plane

1. Select the **Infinite Plane** tool.
2. Pick *3 Pts In Plane* option from the drop down list on the Message Line. The Message Line reads: *Infinite Plane: Enter 3 points in the plane*.

   If you would like to create the surface in a plane different from the one currently set, change the plane before moving on to the next step.
3. Click three points. The plane icon at the specified location.

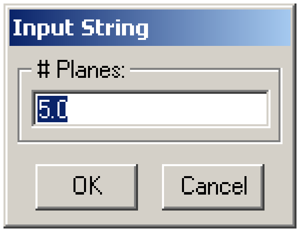
There are no entries in the Status Line.

Using the Infinite Plane Tool to create a Plane along a Curve

1. Select the **Infinite Plane** tool.

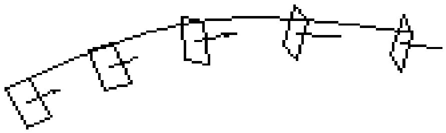
2. Pick *Along Curve* from the drop down list on the Message Line. The Message Line reads: *Infinite Plane: Pick curve to distribute planes along [Shift=Extend]*.

3. Click a curve on the screen. The Input String dialog box appears.



   Enter the number of planes to place along the curve. Click OK to accept the change and close the dialog box. Click Cancel to close the dialog box without accepting the changes.

   The planes are created along a curve. The example here shows 5 planes along a curve.



If you select the curve, the Input String dialog box reappears allowing you to change the number of planes placed along the curve.

There are no entries in the Status Line.

Infinite Plane Tool - Along the Curve Option and the Plane/Surface Intersection Tool

Referral:

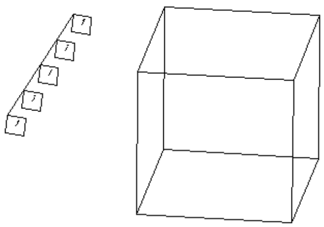
See Chapter 22, "Surface Editing Tools," for information on using the **Plane/Surface Intersection** tool.

You can use the *Along Curve* option of the **Infinite Plane** tool with the **Plane/Surface Intersection** tool to quickly create cross sections through solid models.

1. Select the **Infinite Plane** tool and select the *Along Curve* option.

2. Create the planes along a curve adjacent to the solid model.

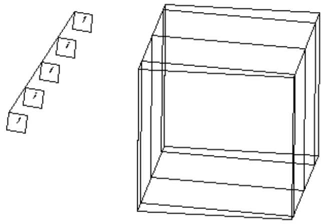
3. Choose the **Plane/Surface Intersection** tool and pick the planes on the curve with which you want to cut the solid. The graphic here shows one example.



4. Select the solid. The **Plane/Surface Intersection** tool projects the planes through the solid. Your Designer Elements program puts each projected plane on a separate layer if you click the *AutoLayers* box on the Message Line before you create them.

5. The intersections displays.

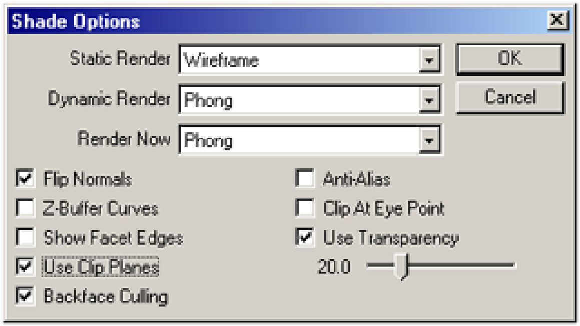
There are no entries in the Status Line.



Clipping Planes

Dynamic display clipping planes provide a means to slice through the model and hide the display to either side of the plane.

In order to use an **Infinite Plane** as a **Clipping Plane** you must bring up the **Shade Options** dialog box located in the **View Menu** and select the “Use Clip Planes” option.

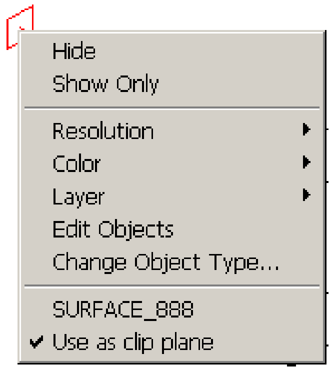


Using an Infinite Plane as a Clipping Plane

1. Select the Infinite Plane tool. The Message Line reads: *Infinite Plane: Pick location for plane.*

2. Place the Infinite Plane in your drawing.

3. Right-Click on the infinite plane and select the “Use a Clip Plane” menu option.



4. Use the **Selection** tool to dynamically drag the clip plane through the model.
5. To change the direction that the plane clips choose **Edit>Change Direction**.

This is also referred to as changing the surface normal or flipping normals.
To turn off the clipping plane option go to the **View>Shade Options** menu and deselect the Use Clip Plane option. By default this option is off.

Geometric Characteristics

An infinite plane is created by choosing points or curves as directed by the Message Line. An infinite plane is a surface made up of the following characteristics according to the Edit Objects dialog box: the Location for the X, Y and Z location and the Normal for the DX, DY and DZ location.

This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the plane and choose ***Window>Edit Objects*** or double-click on the plane.

Sphere Primitive

This tool creates a sphere surface using a center and radius point.

Using the Sphere Primitive Surface tool

1. Select the Sphere Primitive tool. The Message Line reads: *Sphere Primitive: Enter center and radius point for sphere*.
2. In the drawing area click the center point and radius point.

The Status Line contains X,Y and Z data fields for the sphere center point and dX, dY and dZ fields to reference distance and direction of the radius.

Block Primitive

The Block Surface Primitive creates a cubic or rectangular shaped object.

Using the Surface Block Primitive

1. Select the Block Primitive tool. The Message Line reads: *Block Primitive: Enter center and length points*.

2. In the drawing area click the center and length points for the block.

The first point you click is the center point. This is the bottom center of the resulting block. The second point you click controls not only length, but orientation of the block.

The Status Line contains X, Y and Z data fields for the block base center point plus dX, dY and dZ for the height. In addition, you can specify L and W for the length and width of the block.

Cylinder Primitive

This tool creats a cylindrical surface primitive.

Using the Surface Cylinder Primitive

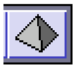
1. Select the Cylinder Primitive tool. The Message Line reads: *Cylinder Primitive: Enter start and end points for cylinder.*

2. In the drawing area click the start and end points for the cylinder.

The first point you click is the cylinder base center point. The second point you click controls not only length, but orientation of the cylinder.

The Status Line contains X, Y and Z data fields for the cylinder base center point plus dX, dY and dZ for the height. In addition, you can specify D for the diameter of the cylinder.

Pyramid Primitive



This tool creates a pyramid surface.

Using the Surface Pyramid Primitive

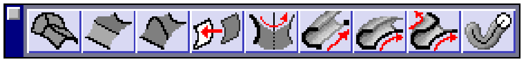
1. Select the Surface Pyramid Primitive tool. The Message Line reads: *Pyramid Primitive: Enter start and end points for pyramid.*

2. In the drawing area click the start and end points for the pyramid.

The first point you click is the pyramid base center point. The second point you click controls the length and orientation of the pyramid.

The Status Line contains X, Y and Z data fields for the pyramid base center point plus dX, dY and dZ for the height. In addition, you can specify D for the pyramid base diameter and the #Sides for the number of sides

Surface From Curves Tools.



Net Surface Tool

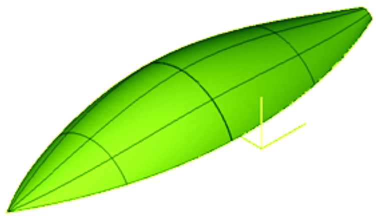
The **Net Surface** tool creates a surface defined by a grid of *M* rows and *N* columns, where M and N are letter identifiers referring to the number of rows or columns. Curves suitable as boundaries for net surfaces include arcs, circles, lines, splines, ellipses and conics. These curves cannot be grouped.

Tech Note:

A net surface is not the same as a mesh. Net surfaces are nurb surfaces. A mesh is defined by nodes or 3D vertices.

The curves may be in different planes. If the curves lie in the same plane, our Designer Elements program creates a simple planar surface within the curve boundary. If the curves lie out of plane, our Designer Elements program fits a curved net surface similar to a *Coons patch* mesh to the boundary curves. A Coons patch is a three or four sided nurb surface. The left graphic below shows four curves. The right graphic shows the resulting net surface with the referenced curves in gray.

You can also specify a point entity for the start or end of the net surface, creating a degenerative surface.

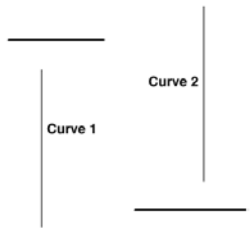


Using the Net Surface Tool

1. Select the **Net Surface** tool. The Message Line reads: *Net Surface: Pick (M) curves defining surface [Shift=Extend].*

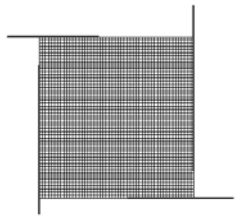
2. Hold down the SHIFT key and pick two or more curves along one direction. The objects appear selected as in the graphic here (Curves 1 and 2).

   When you release the SHIFT key, the objects are deselected. The Message Line now reads, *Net Surface: Pick (N) curves defining surface [Shift=Extend].*



3. Hold down the SHIFT key once again, and pick two or more curves to complete the boundary for the surface.

   When you release the mouse button the net surface is created. The graphic here shows the net surface from the example above.



As the example shows here, the curves defining the surface do not have to be connected. Our Designer Elements program calculates the intersecting area of the selected curves and creates the net surface from that area.

There are no entries in the Status Line.

Geometric Characteristics

A net surface is created by choosing curves as directed by the Message Line. A net surface has no specific geometry characteristics according to the Edit Objects dialog box. There are tabs for both Display and Attributes.

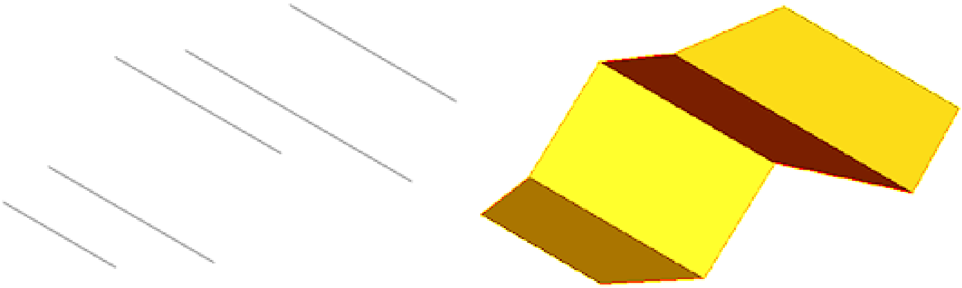
Display contains settings for Iso Lines and Silhouette. You can display Iso Lines on your surface by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on your surface. You have options for the Silhouette setting, Smart, Off or On.

Ruled Surface Tool

The ruled surface tool is used to create surfaces that are linear between the selected spans. Ruled surfaces are developable surfaces. Developable surfaces can be unrolled into flat patterns without stretching or shrinking.

Using the Ruled Surface Tool

1. Select the **Ruled Surface** tool. The Message Line reads: *Ruled Surface: Pick curves for defining surface. [Shift=Extend]*
2. If there are more than two curves being used for the ruled surface hold down the SHIFT key and select all of your curves. When you release the SHIFT key the surface is created



As the example shows, the curves defining the surface do not have to be connected. Our Designer Elements program calculates the intersecting area of the selected curves and creates the ruled surface

Geometric Characteristics

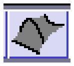
A ruled surface is created by choosing curves as directed by the Message Line. A ruled surface has no specific geometry characteristics according to the Edit Objects dialog box. There are tabs for both Display and Attributes.

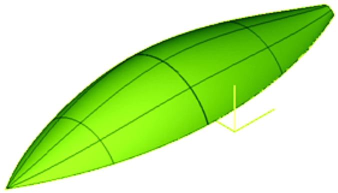
Display contains settings for Iso Lines and Silhouette. You can display Iso Lines on your surface by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on your surface. You have options for the Silhouette setting, Smart, Off or On.

Tech Note:

Do not confuse the Skin Surface tool with the Cover Surface tool. The curves you select for this tool are used as cross-sections. They cannot be connected or intersecting. A warning message will appear if you choose invalid curves for the operation. To surface curves that are connected or closed use the **Cover Surface** tool.

Skin Surface Tools

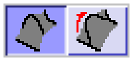




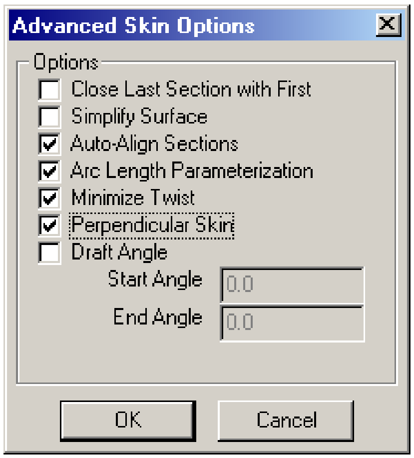
A skin surface fits a network of NURB patches to a collection of curves that act as *cross-sections* for the final surface. The curves that are skinned can be open, closed or grouped. Curves suitable for skinning include lines, arcs, circles, ellipses, conics and splines. You can create a standard skin surface or closed (tangent) skin surface. You can also specify a point entity for the start or end of the surface, creating a degenerative surface.

Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when creating a skin surface.

There are two tools in the Message Line, **Skin Curve** and **Skin with Guide Curves**.



There is also an Advanced Skin Options dialog box that gives you more control of how your object looks when skinned.



It contains the following options:

Close Last Section with First The close option may be used when the user needs to construct a body closed in one direction (v direction). The default is an open (not closed) body. The surface is continuous at each profile. If the user provides a set of closed profiles, the face normals of the skin or loft body point outside, away from the body material. When the user provides a set of open profiles, the surface normals of the skin surface is oriented along the surface normals and no attempt is made to change the surface normal orientation. When using this option you do not provide profiles in the same position.

Simplify Surface The simplify option simplifies the created surface to a conical surface, if applicable. If all of the cross sections lie on a conical surface (plane, cylinder, cone, sphere or torus), the conical surface is created instead. The value of 0.000001 is used to determine whether or not the cross section lies on an analytical surface (planar, conical, spherical or toroidal). The default is not simplified.

Auto-Align Sections The align sections option may be used to allow the skinning algorithm to align the direction of the curves in the selection list. Closed loops of wires can also be aligned. The default is aligned.

Arc Length Parameterization The arc length option is used to choose arc length or isoparametric parameterization of the skinning surfaces. In isoparametric parameterization the surface parameter in the v direction follows the cross section curves. In arc length parameterization the surface parameter follows lines of constant length. The default is arc length parameterization.

Minimize Twist The twist option may be used to minimize the twist of the surface produced. Twist minimization aligns closed curves such that the start of the second curve is aligned to the start of the first curve. Even if a body's shape is unaffected by twisting, a surface with a twist could produce unexpected results when faceting and rendering. By default, twist minimization is on. Twist minimization is also an involved calculation that some users may not want to have carried out.

Perpendicular Skin

The take–off vector is a tangent vector going out of the starting edge or surface and into the skinned surface. The perpendicular option is used to specify the direction of the take–off vector, perpendicular to the edge. The default is in the loft direction because a perpendicular take–off vector can cause self–intersections to the surface.

Draft Angle

Skinning with draft angles provides the ability to control the take–off vectors of the two outer skinning profiles. The “draft” angle is defined as an angle off the plane of the wire at every point along the skinning profile. In addition to the user supplying the angle itself, one may also supply a magnitude for the take–off vector. The draft angle and magnitude is constant for the entire profile. However, one may apply different draft angles to the two outer profiles. In addition skinning with draft angles supports open and closed profiles and skinning to a point. When skinning to a point the algorithm constructs its own normal vector. The outer profiles must be planar when not degenerate. Use the Edit Object dialog box to access the skin magnitude.

Standard Skin Surface

An open skin surface is a surface that has open ended in all directions. You select the curves in the order that you want them skinned. When you select the successive curves (or *n#* sections), you create a surface between the curves. You are not surfacing the individual curves. If you want to surface curves connected end to end use the **Cover Surface** tool. See a later section in this chapter.

The curves must have the same direction. If the curves have different directions the resulting surface will twist from section to section, as shown here.

Tech Note:

The term, *N# sections*, refers to the curve sections you select to create a surface.The letter N is the generic identifier.

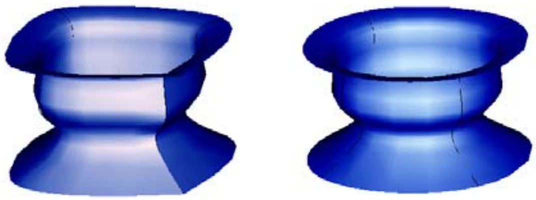
To avoid creating twisted surfaces, check the curves' directions by choosing ***Verify>Direction***. If the curves do run in different directions you can adjust or redraw them before creating the surface. The graphic here shows curves with the same direction and the resulting surface.

This tool also provides an option for creating closed tangents.

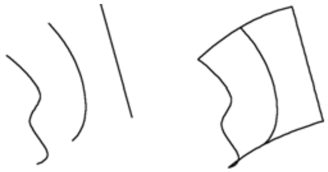
Closed Skin Surface

A closed or tangent skin surface is continuous end to end. When you use this option, your Designer Elements program automatically closes the skin such that the first section equals the last section making the slopes along the edge tangent continuous. The graphic here shows spline curves used to create a closed skin surface with the **Skin with Guide Curves** tool.

When these splines are skinned normally, the left model here results. When skinned using the closed skin option, the right model results.



Skin Curve Surface Tool



This tool creates a skin surface between two or more curves. The surface edges are defined by the limits of the selected curves. You can create a standard skin surface or a closed skin surface. The figure here shows a open skin surface created from three curves.

Creating a Standard Skin Surface

1. Check curve directions by choosing ***Verify>Direction***. If all of the curves do not run in the same direction, adjust them so that they do.
2. Select the **Skin Surface** tool. The Message Line reads: *Skin Surface: Pick curves for defining surface [Ctrl = Advanced* (Windows) or *Option* (Macintosh) *Shift=Extend].*

3. Select the **Skin Curve Surface** tool (the left tool) in the Message Line.
4. Select the curves in the order that the surface is to be skinned. To select more than two curves hold down the SHIFT key. You cannot use a selection fence to select the curves.

There are no entries in the Status Line.

Creating an Closed Skin Surface

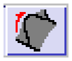
1. Check curve directions by choosing ***Verify>Direction***. If all of the curves do not run in the same direction, adjust them so that they do.
2. Select the **Skin Surface** tool. The Message Line reads: *Skin Surface: Pick curves for defining surface [Ctrl = Closed* (Windows) or *Option* (Macintosh) *Shift=Extend].*
3. Select the **Skin Curve Surface** tool (the left tool) in the Message Line.
4. (Windows) Select the curves in the order that the surface is to be skinned. To select more than two curves hold down the SHIFT key. (You cannot use the selection fence to select the curves.) After you select the last curve and while still holding down the SHIFT key, hold down the CTRL key. Release the SHIFT key and then the CTRL key.

   (Macintosh) Hold down the OPTION key and select the curves in the order that the surface is to be skinned. To select more than two curves hold down the SHIFT+OPTION keys. You cannot use the selection fence to select the curves.

The tangent surface is created.

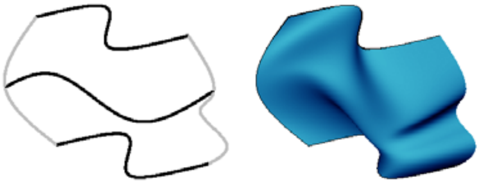
There are no entries in the Status Line.

Skin with Curve Guides Surface Tool



This tool creates a skin surface between two or more curves using curves as guides to define the edges. Guide curves give you control over the skin surface. You can use one or more guide curves to influence the surface.

In the left graphic below, the profile curves are black and the guide curves are gray. The right graphic shows the skinned surface.



Guide Curve Rules

- Curves can go in any direction and need not be consistent with the others.
- Curves cannot loop and must be "well-behaved."
- The curves must connect with each profile that you want to use for surface creation.

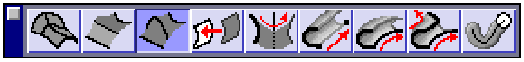
Note: Guide curves only affect the geometry of the surface created between the profiles to which the guide curves are attached. Also, if the guide curve passes through vertices, the two adjoining surfaces follow the curve profile.

You can create a standard skin surface or a closed skin surface.

Creating a Standard Skin Surface

1. Check curve directions of the profile curves by choosing ***Verify>Direction***. If all of the curves do not run in the same direction, adjust them so that they do.
2. Create one or more guide curves.

3. Select the **Skin Surface** tool. The Message Line reads: *Skin Surface: Pick curves for defining surface [Ctrl = Advanced* (Windows) or *Option* (Macintosh) *Shift=Extend]*.



4. Select the **Skin with Guide Curves Surface** tool (the right tool) in the Message Line.
5. Select the curves in the order that the surface is to be skinned. To select more than two curves, hold down the SHIFT key. You cannot use a selection fence to select the curves.

   The Message Line now reads, *Skin Surface: Pick guide curves [Shift = Extend]*.
6. Select the guide curves.

   The skin surface is created.

There are no entries in the Status Line.

Creating a Closed Skin Surface

1. Check curve directions by choosing ***Verify>Direction***. If all of the curves do not run in the same direction, adjust them so that they do.
2. Create one or more guide curves.
3. Select the **Skin Surface** tool. The Message Line reads: *Skin Surface: Pick curves for defining surface [Ctrl = Advanced* (Windows) or *Option* (Macintosh) *Shift=Extend]*.
4. Select the **Skin with Guide Curves Surface** tool (the right tool) in the Message Line.
5. (Windows) Select the curves in the order that the surface is to be skinned. To select more than two curves hold down the SHIFT key. (You cannot use the selection fence to select the curves.) After you select the last curve and while still holding down the SHIFT key, hold down the CTRL key. Release the SHIFT key and then the CTRL key.

   (Macintosh) Hold down the OPTION key and select the curves in the order that the surface is to be skinned. To select more than two curves hold down the SHIFT+OPTION keys. You cannot use the selection fence to select the curves.

   The Message Line now reads, *Skin Surface: Pick guide curves [Shift = Extend]*.
6. Select the guide curves.

The tangent surface is created.

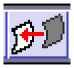
There are no entries in the Status Line.

Geometric Characteristics

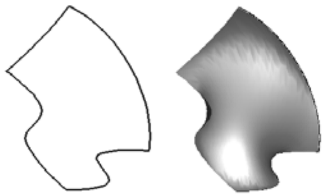
A skin surface is created by choosing curves as directed by the Message Line. A skin surface has no specific geometry characteristics according to the Edit Objects dialog box. There are tabs for both Display and Attributes.

Display contains settings for Iso Lines and Silhouette. You can display Iso Lines on your surface by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on your surface. You have options for the Silhouette setting, Smart, Off or On.

Cover Surface Tool



The **Cover Surface** tool creates a surface from curves connected end to end. The tool tip calls these curves *n-sided*. Curves suitable as boundaries for covered surfaces include arcs, circles, lines, splines, ellipses, conics and grouped curves.



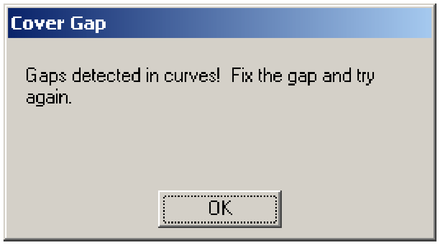
The curves may be in different planes. If the curves lie in the same plane our Designer Elements program creates a simple planar surface within the curve boundary. If the curves lie out of plane, our Designer Elements program fits a *Gregory surface* to the boundary curves. A Gregory surface is a nurb surface with five or more sides.

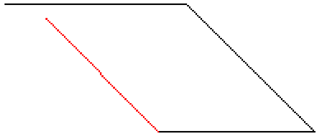
Using the Cover Surface Tool

1. Select the **Cover Surface** tool. The Message Line reads: *Cover Surface: Pick curves for cover surface [Shift=Extend].*
2. If you want to select more than one curve hold down the SHIFT key before selecting the first curve.
3. Select the curves. Do not worry about the order you select them.

The cover surface is created.

If not all of the curves that you select connect end to end, as shown in the figure below, our Designer Elements program warns you with a Cover Gap dialog box and marks the gaps in the selected curves.





Either move the endpoints together or redraw the curves so that they connect.

There are no entries in the Status Line.

Geometric Characteristics

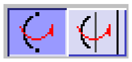
A cover surface is created by choosing curves as directed by the Message Line. A cover surface has no specific geometry characteristics according to the Edit Objects dialog box. There are tabs for both Display and Attributes.

Display contains settings for Iso Lines and Silhouette. You can display Iso Lines on your surface by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on your surface. You have options for the Silhouette setting, Smart, Off or On

Revolved Surface Tool

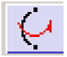
The **Revolve Surface** tool allows you can create a surface by revolving a set of curves (called the profile) around a central axis.

In the Message Line, you have a subpalette displaying two tools for revolving profiles: **Revolved Surface about Two Points** and **Revolved Surface about a Curve**.

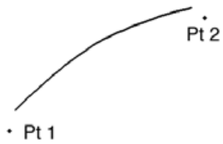


If you revolve curves connected at their endpoints the resulting revolution creates one surface.

Revolved Surface about Two Points Tools



This tool allows you to revolve a curve about two points that you choose. The direction of the Angle of Revolution is determined by the order in which you pick the points that define the axis. Use the Right Hand Rule to determine the direction that the surface will revolve.



See Chapter 2 for more information on the Right Hand Rule.

Using the Surface of Revolution Tool - About Two Points

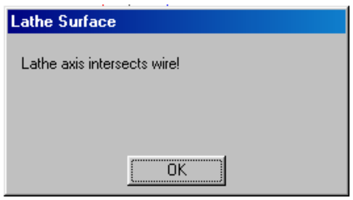
1. Select the **Revolved Surface** tool.
2. Select the **Resolve Surface About Two Points** tool in the Message Line. The Message Line reads: *Revolved Surface: Pick surface to revolve [Shift=Extend].*
3. Enter the angle through which to revolve the curve in the Angle data field of the Status Line.

4. Select the curve or curves to revolve.

   The Message Line directs you to enter two points for lathe (or revolution) axis.

5. Click two points to define the Axis of Revolution.

   If the Axis of Revolution crosses any of the profile curves this error message appears:

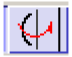


   Pick a new non-intersecting axis to complete the surface of revolution.

   The surface is created.

You can change the Angle while the surface is still selected by highlighting the Angle field, entering a new value and pressing ENTER (Windows) or RETURN (Macintosh). The surface adjusts accordingly. Any future modification to angle must be made through the Edit Objects dialog box.

Revolved Surface about a Curve Tool



This tool allows you to revolve a curve about an existing curve that you choose. The direction of the angle of revolution is determined by the direction in which the axis of revolution curve was drawn. Use the Right Hand rule to determine the direction for the revolution. See Chapter 2 for more information about the Right Hand Rule.

Using the Revolved Surface Tool: About a Curve

1. Select the **Revolved Surface** tool.
2. Select the **Resolve Surface About Curve** tool in the Message Line. The Message Line reads: *Revolved Surface: Pick surface to revolve [Shift=Extend].*
3. Enter the angle through which to revolve the curve in the Angle data field on the Status Line.

4. Select the curve or curves to revolve.
5. Select the curve to use as the Axis of Revolution. If the Axis of Revolution crosses any of the profile curves and error message appears. Choose another curve or adjust the curve as required.

   The surface is created.

You can change the Angle while the surface is still selected by highlighting the Angle field, entering a new value and pressing ENTER (Windows) or RETURN (Macintosh). The surface will adjust accordingly. Any future modification to angle must be made through the Edit Objects dialog box.

Geometric Characteristics

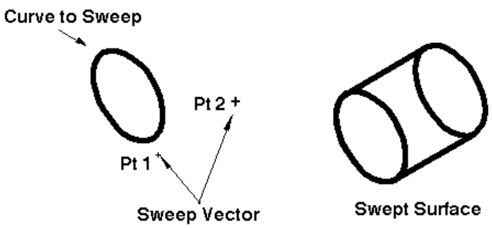
A revolved surface is created by choosing a curve to revolve and the axis of revolution. A revolved surface has the following geometry characteristics according to the Edit Objects dialog box: Rot. Angle, End 1 (X, Y and Z values) and End 2 (X, Y and Z values).

Display contains settings for Iso Lines and Silhouette. You can display Iso Lines on your surface by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on your surface. You have options for the Silhouette setting, Smart, Off or On.

Sweep Surface Tool - Along 2 Points

The **Sweep Surface** tool sweeps a curve along a path defined by the user. The path chosen depends on the **Sweep Surface** tool used. This Designer Elements program automatically aligns, orders and sets the direction of your curves for the sweep path.

This tool sweeps a curve along a two point vector. The sweep surface starts from the curve and moves in the direction and distance indicated by two points that you choose. The resulting surface is associative to the original sweep curve. Change the curve and the surface automatically updates.

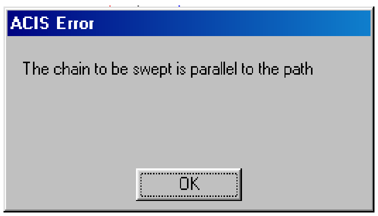


You can check information on the sweep using commands in the Verify menu. You can sweep grouped curves with this tool.

Using the Sweep Surface Tool: Along 2 Points

1. Select the **Sweep Surface Along 2 Points** tool. The Message Line reads: *Pick curve(s) to sweep [Shift=Extend].*
2. Select the curve that you want to sweep. The Message Line reads: *Sweep 2 Pts Surface: Enter two points for sweep direction*.
3. Click two points to define the vector. The order you click the points specifies the direction of the sweep.

   The vector cannot run in the same direction as the curve. If you define a vector in that direction this message appears:

   Select the curve again and click two other points to define the vector.

   The sweep surface is created.

Tip:

To sweep a curve at an angle not perpendicular to the curve, change the view to specify the sweep points. Example: If you draw a rectangle in the top plane, change the view to front and specify the points at the desired angle.

You can change the values in these data fields while the surface is still selected by highlighting the desired field and entering in a new value. Use the Tab key to move from field to field. Press ENTER (Windows) or RETURN (Macintosh) and the surface will adjust accordingly. Any changes you want to make after this must be made in the Edit Object dialog box.

The Status Line displays delta values for the X, Y and Z coordinates of the vector and the distance of the sweep.

Geometric Characteristics

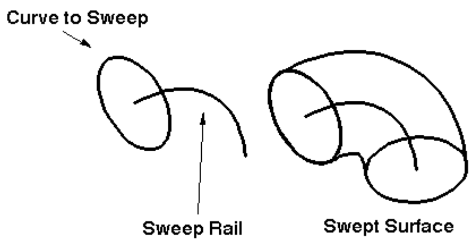
A swept surface is created by choosing a curve to sweep and the points for the two points defining the vector. A swept surface along two points has the following geometry characteristics according to the Edit Objects dialog box: Distance and Vector (dX, dY and dZ values).

Display contains settings for Iso Lines and Silhouette. You can display Iso Lines on your surface by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on your surface. You have options for the Silhouette setting, Smart, Off or On.

Sweep Surface Tool - Along 1Rail

This tool creates a surface by sweeping an open, closed or grouped curve along another curve. The resulting surface is associative to both the path and the sweep curve.

Modifying either automatically updates the associative sweep surface. You can only sweep one curve at a time.



Using the Sweep Surface Tool: Along 1 Rail

1. Select the **Sweep Surface Along 1 Rail** tool. The Message Line reads: *Sweep 1 Rail Surface: Pick curve to sweep [Shift=Extend].*

2. Click the sweep curve. The Message Line reads: *Sweep 1 Rail Surface: Pick sweep axis [Shift=Extend].*
3. Click the rail curve. The surface is created.

There are no entries in the Status Line.

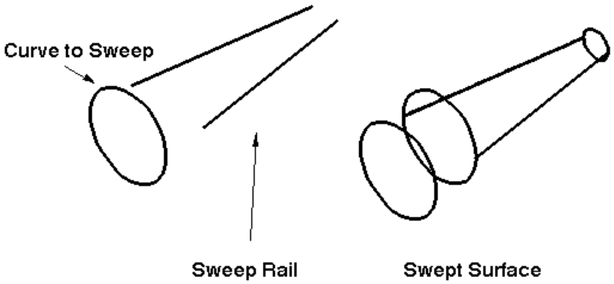
Geometric Characteristics

A swept surface is created by choosing a curve to sweep and a curve that acts as the rail for the sweep. A swept surface along one rail has no specific geometry characteristics according to the Edit Objects dialog box. There are tabs for both Display and Attributes.

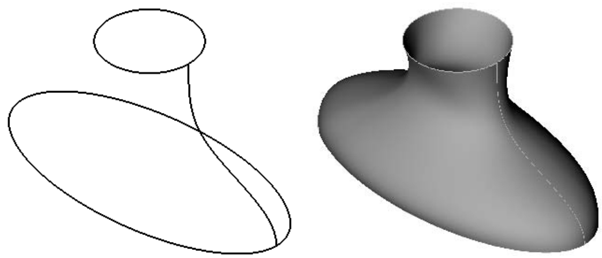
Display contains settings for Iso lines and Silhouette. You can display Iso Lines on your surface by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on your surface. You have options for the Silhouette setting, Smart, Off or On.

Sweep Surface Tool - Along 2 Rails

The sweep along two rails method creates a swept surface by sweeping an open, closed or grouped profile between two rail curves. A rail can also be closed, back tracking over itself like a circle. This tool supports the maintaining of the profile height along the rail by holding down the CONTROL (Windows) or OPTION (Macintosh) key. This tool does not support multiple or grouped curves for the rails. A simple two rail sweep is shown in the graphic here.



As mentioned, you can also use closed curves or curves that backtrack as the rails. In the left graphic below, the top circle and bottom ellipse are the rails and the spline is the curve being swept. The right graphic shows the swept surface.



The rail curves define the orientation and scale of the swept surface as it moves between them. The resulting surface is associative to both rails and the sweep curve. Modifying any of the three will automatically update the associative sweep surface.

Using the Sweep Surface Tool: Along 2 Rails

1. Select the **Sweep Surface** tool.
2. Select the **Sweep Surface Along 2 Rails** tool in the Message Line. The Message Line reads: *Sweep 2 Rail Surface: Pick curves or group to sweep. [Ctrl or Option = Maintain Height][Shift = Extend]*
3. Click the profile you want to sweep and then choose the two rails for the sweep path.

   A swept surface is created.

   Hold down the Ctrl or Option key if you want the profile height to be maintained.

There are no entries in the Status Line.

Geometric Characteristics

A swept surface is created by choosing curves or grouped profiles that act as the rails for the sweep. A swept surface along two rails has no specific geometry characteristics according to the Edit Objects dialog box. There are tabs for both Display and Attributes.

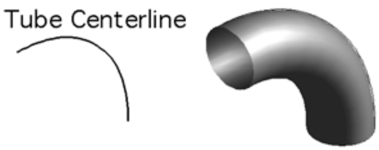
Display contains settings for Iso Lines and Silhouette. You can display Iso Lines on your surface by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on your surface. You have options for the Silhouette setting, Smart, Off or On.

Sweep Rules

- You must double-click the surface to display the Edit Objects dialog box. If you pick the surface and the original sweep curve with ***Edit>Select All***, or by dragging a selection fence, the Edit Objects dialog box will list only the attributes of the two entities and will omit the sweep surface's geometry information. Therefore, always select the surface by double-clicking it. With the second click, the Edit Objects dialog box appears automatically.
- If you want to perform a helical sweep, the helix must originate in this Designer Elements program. Imported helix curves do not follow the helix curve law and the sweep operation requires this for a proper sweep.
- If you sweep curves connected at their endpoints, the resulting sweep creates one surface.

Tube Surface Tool

With the **Tube Surface** tool you can quickly and efficiently create a tube or pipe along a reference curve. This tool support grouped curves. You must specify a radius small enough to prevent a self-intersecting surface.



Using the Tube Surface Tool

1. Create a curve that would act as the center line for the tube.
2. Select the **Tube Surface** tool. The Message Line reads: *Tube Surface: Pick tube center line [Shift=Extend].*
3. Click the curve for the tube's center line.
4. Enter a new tube radius in the data field on the Status Line, if required. Press ENTER (Windows) or RETURN (Macintosh) and the radius of the tube will change accordingly.

If you want to specify a tube radius before the tube is created, type the value in the Radius data field and then select the curve and the tube surface will be created with the specified radius.

Geometric Characteristics

A tube surface is created by choosing a curve that will act as the center line for the tube. A tube surface has the following geometry characteristic according to the Edit Objects dialog box: Diameter of the tube.

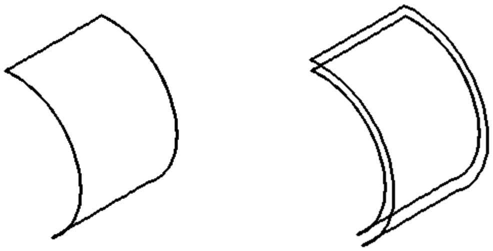
Display contains settings for Iso Lines and Silhouette. You can display Iso Lines on your surface by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on your surface. You have options for the Silhouette setting, Smart, Off or On.

Feature Surface Tools

Offset Surface Tool

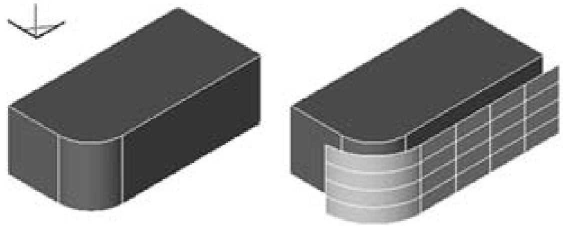
With the **Offset Surface** tool you can make a new surface based on an existing or parent surface, offset by an amount that you choose. You can also offset an infinite plane.

The offset surface is created by first selecting the parent surface and specifying the offset surface. Offset surfaces maintain a constant distance normal to the parent surface. The direction of the offset is the surface normal direction. You can specify negative offsets. Do not offset a surface a distance greater than the smallest radius of curvature or you can create a degenerate surface.



If you specify a zero offset, this Designer Elements program creates an associative copy of the selected surface at the same location.

You can also offset solids faces. You may single select a particular face or box select the entire solid to offset all of its faces. A positive offset for a solid, creates a surface outside the solid. A negative offset creates a surface towards the inside of the solid.



Using the Offset Surface Tool

1. Select the **Offset Surface** tool. The Message Line reads: *Offset Surface: Pick surfaces to offset [Shift=Extend].*
2. Enter an Offset distance in the data field of the Status Line.

3. Click the parent surface. A duplicate surface appears at the offset distance.

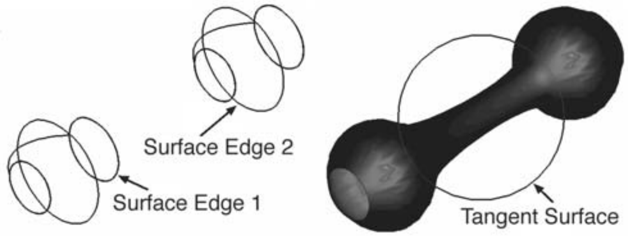
Geometric Characteristics

An offset surface is created by choosing the parent surface and specifying an offset distance. An offset surface is composed of the offset value according to the Edit Objects dialog box. There are tabs for both Display and Attributes.

Display contains settings for Iso Lines and Silhouette. You can display Iso Lines on your surface by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on your surface. You have options for the Silhouette setting, Smart, Off or On.

Lofted Surface Tool

The **Lofted Surface** tool creates a smooth, transitional surface between surfaces edges and any number of interior non-surface edges. To create a tangent surface select the edge of the start surface and the edge of the end surface.



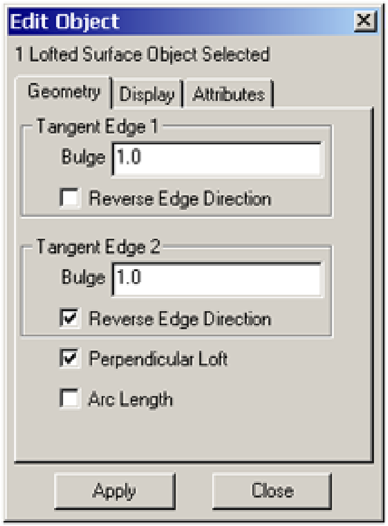
Using the Lofted Surface Tool

1. Select the **Lofted Surface** tool. The Message Line reads: *Lofted Surface: Pick edges for lofted surface.[Shift=Extend].*

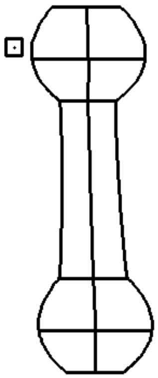
2. Enter a new bulge factor in the Bulge data field in the Status Line, if desired. The bulge factor controls the magnitude of the slope of the surface edges.

3. Click an edge on each surface to be joined by the tangent surface. When you create a tangent surface, the screen shows only one connecting line. Choose ***View>Shade Now*** to see the tangent surface. If *Shade Now* is set to anything other than wireframe in the Shade Options dialog box, your lofted surface appears in the chosen mode.

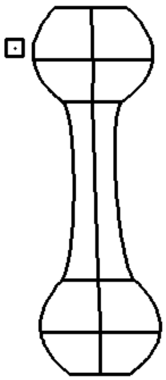
Once created, the lofted surface can be modified in the Edit Objects dialog box accessed through the **Design Explorer**. The start bulge factor controls the magnitude of the slope from the starting edge, whereas the end controls the slope at the end curve. The **reverse Edge Direction** options are provided to fix twisting of the Tangent surface resulting from different edge directions.



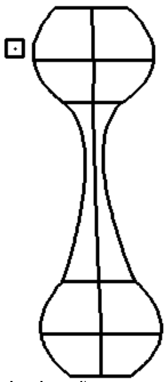
The figure shows two bulb surfaces joined with a tangent surface using different bulge factors. The left figure uses a 0.25 bulge factor; the center figure uses a 10.0 bulge factor; the right figure uses a 3.0 start bulge factor and a 0.5 end bulge factor.



A bulge factor of .25 is used at both ends.



A bulge factor of 10 is used at both ends.



A bulge factor of 3 is used at the top end and 0.50 factor at the bottom end.

Geometric Characteristics

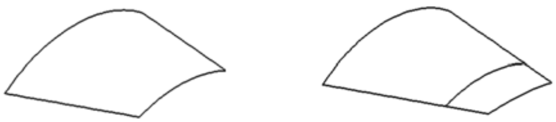
A lofted surface is created by choosing the two connecting surfaces. You may edit the bulge factor for the lofted surface edges in the Edit Objects dialog box. Tangent Edge 1 controls the first edge chosen and Tangent Edge 2 controls the last edge chosen.

Display contains settings for Iso Lines and Silhouette. You can display Iso Lines on your surface by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on your surface. You have options for the Silhouette setting, Smart, Off or On.

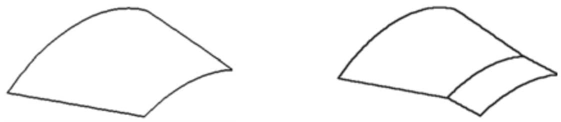
Draft Surface Tool

The **Draft Surface** tool extends a surface to create a new surface attached to the end of the original surface. This Designer Elements program provides data fields for angle and length that give you control over how the surface is generated. Entering 0° in the Angle data field extends the surface tangent to the original. Entering 90° extends the surface perpendicularly.

You have two options to set the new surface tangency; *Default* and *Align Curve*. The default option is automatic when using this tool. It creates a draft surface aligned with the original surface derivatives, based on the product of the surface tangent and normal. The graphic below shows the original geometry and the resulting draft surface.



By holding down the CTRL key (Windows) or OPTION key (Macintosh) you choose the *Align Curve* option listed in the Message Line. When you use this option a draft surface is created based on the product of the alignment curve tangent and the surface normal. The graphic below shows the original geometry and the resulting draft surface.



Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when creating a draft surface.

Using the Draft Surface Tool

1. Select the **Draft Surface** tool. The Message Line reads: *Draft Surface: Select curve on surface [Ctrl=Align Curve].*
2. Enter a new angle and/or length in the data fields in the Status Line, if you so desire.

3. Click a curve (edge) on the surface to draft or extend the surface. When you select the edge, the whole surface is highlighted until this Designer Elements program creates the draft.

You can change both the angle and the length of the Draft by entering new values in the data field while the geometry is still selected. Pressing ENTER (Windows) or RETURN (Macintosh) will make the change. Any other changes must be made in the Edit Objects dialog box.

Geometric Characteristics

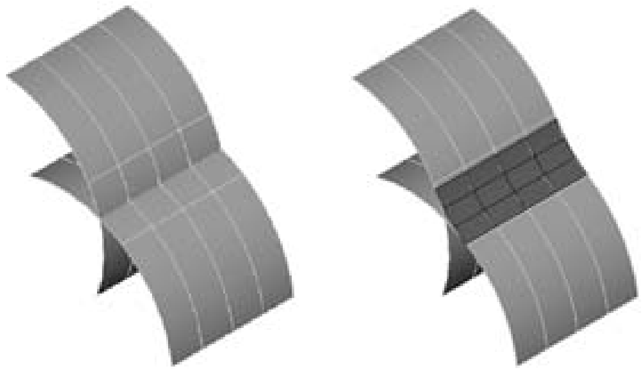
A draft surface is created by choosing a curve to draft and entering the desired angle and length of the draft. A draft surface has the following geometry characteristics according to the Edit Objects dialog box: Angle and Length.

Display contains settings for Iso Lines and Silhouette. You can display Iso Lines on your surface by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on your surface. You have options for the Silhouette setting, Smart, Off or On.

Fillet Surface Tool

The **Fillet Surface** tool blends two surfaces that share an edge or intersection. This tool does not extend surfaces.

When filleting two intersecting surfaces, there are four places where the fillet can be placed. The fillet is created where the surface directions are positive for both surfaces. The surface direction can be determined by selecting and choosing ***Verify>Direction***. You can change the direction by choosing ***Edit>Change Direction***. The left graphic here shows two intersecting surfaces. The right graphic shows the surfaces blended.



Using the Fillet Surface Tool

1. Select the **Fillet Surface** tool. The Message Line reads: *Fillet Surface: Pick two surfaces to blend [Shift=Extend].*

2. Select the two intersecting or connected surfaces that share an edge

   The surfaces are filleted.

If you want to change the fillet radius, type the value in the Radius data field in the Status Line and press ENTER (Windows) or RETURN (Macintosh).

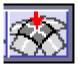
The Status Line contains the Radius for the blend.

Geometric Characteristics

A fillet surface has the following geometric characteristic according to the Edit Objects dialog box: Radius of the fillet.

The Display page of Edit Objects contains settings for Iso Lines and Silhouette. You can display Iso Lines on your surface by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on your surface. You have options for the Silhouette setting, Smart, Off or On.

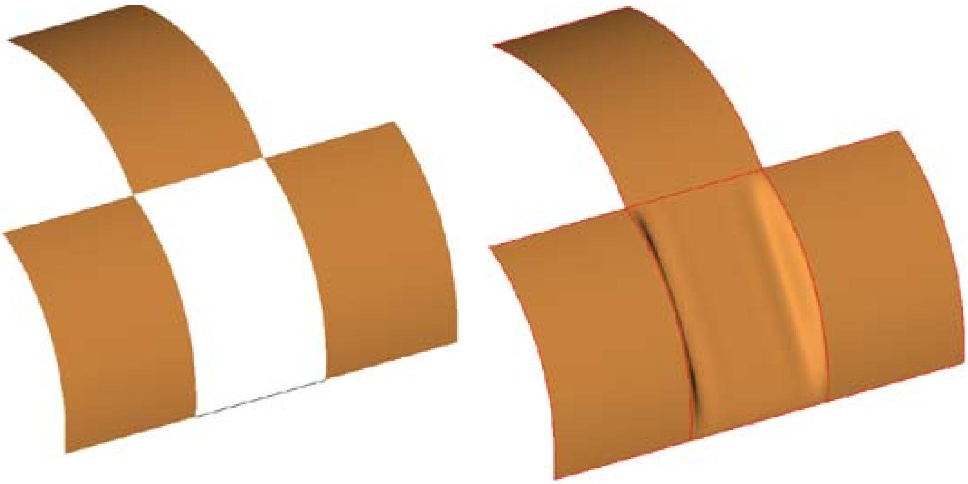
Tangent Cover Surface Tool



The **Tangent Cover** surfaced creates a surface that is tangent to four neighboring surfaces. The surfaces must form a closure. You may also select curves for some of the boundaries however if you have a choice, use surface edges for the boundary of the tangent cover surface.

Using the Tangent Cover Surface Tool

1. Select the Tangent Cover Surface tool. The Message Line reads: *Tangent Cover Surface: Pick four edges/curves for cover surface.*
2. Select the surface edges and or curves. Select them inhead to tail fashion. The tangent cover surface is created.



Note: The first surface selected is the driving surface. Changing the order of the surface selection can change the final results.

Introduction to Solid Modeling

A solid model is an unambiguous 3D representation of an object, composed of selectable faces. A solid model is unambiguous because their mathematical descriptions completely define the inside and outside of a 3D object.

Unlike wireframes and surfaces which define lengths and areas, solids accurately provide volume, mass, moments of inertia, centroids and interference information. Solids also generate stereolithography models useful for demonstrating prototype concepts or for creating forms for molds or castings.

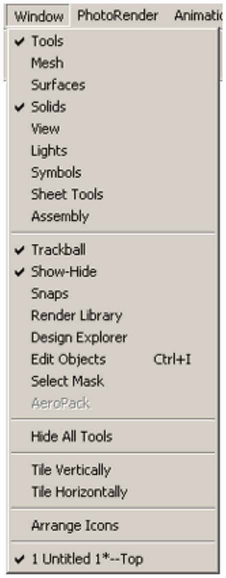
Our Designer Elements program is based on Spatial Technologies' ACIS Geometry Engine. ACIS provides a precise boundary representation of solids suitable for design projects sensitive to accuracy. In ACIS, linear and quadric geometry is exactly represented analytically while free-form geometry is represented as Non Uniform Rational B-Splines (NURBs). Unlike faceted solid modelers, ACIS maintains a high level of accuracy as you perform boolean operations or add features. Because the ACIS engine is so accurate, parts modeled with it are suitable for applications that demand precision, like numerical control. Our Designer Elements program provides several sets of tools with which to create or modify solids: Primitives, Profiles, Features and Editing.

The topics explained here include:

- Solid Modeling tools
- Drafting Assistant and Solids
- Solids, Selection and Display
- Solid Associativity

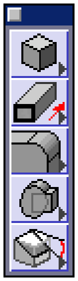
- Solids and Instances
- Cutting and Pasting Solids
- Exporting Solids
- Object Types and Edit Objects

Solid Modeling Tools

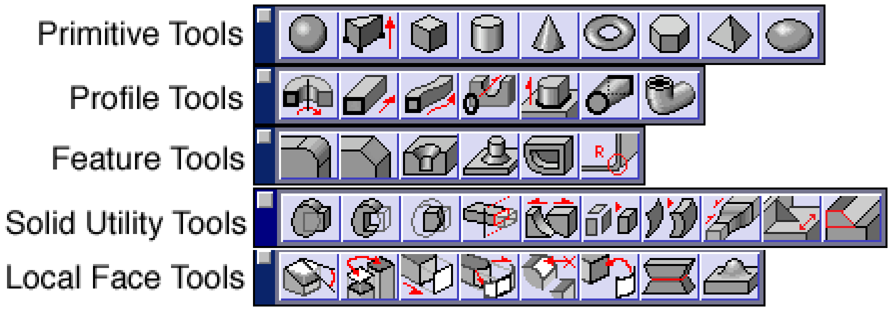


The solid modeling tools are accessible from the Solids tool palette. The Solids tool palette displays automatically when you first launch your Designer Elements program. To display the palette, choose ***Window>Solids***. Selecting the Solids option again from the Window menu or the close box in the its palette's title bar, closes the palette.

The palette will appear under the main tool palette on the left side of your screen.

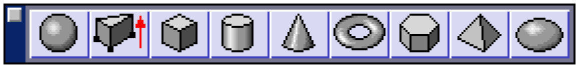


The five solid modeling subpalettes contain groups of tools for creating primitives, profiled solids, features and editing solids.



Primitives

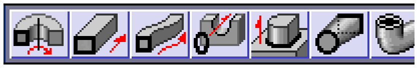
A primitive is a simple shape such as a sphere, slab, block, cylinder, etc.



The Primitives subpalette contains the following tools: Sphere, Slab, Block, Cylinder, Cone, Torus, Prism, Pyramid and Ellipsoid. Chapter 18 explains these tools.

Profiles

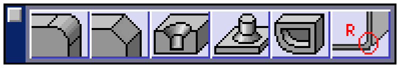
A profile is a curve or collection of curves that enclose an area. You can create a solid from a profile by revolving, extruding, or sweeping a curve or pipe profile along a path.



The Profiles subpalette's contains the following tools: Lathed, Extrude, Sweep, Cutout, Protruded Feature, Skinned Solid and Pipe Solid. Chapter 19 explains these tools.

Features

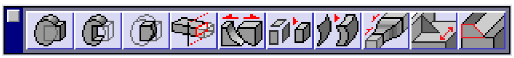
A feature simplifies building or modifying a solid by constructing common design elements with a single command.



The Features subpalette's contains the following tools: Blends (constant, linear, variable), Chamfer (constant, angle, linear) Holes (simple hole, counter bore, counter sink), Boss, Shell and Bend. Chapter 22 explains these tools.

Solid Utility Tools

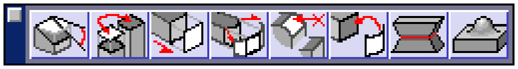
As you create your geometry you may want to perform various operations on them such as splitting a solid or placing a draft. The Solid Utility tools provide you with various ways of editing solid objects.



The Solid Editing tool subpalette contains the following tools: Boolean (Union, Subtract, Intersect), Trim, Split, Stitched, Thicken, Lofted, Rib Feature and Lip Feature (Remove and Add). Chapter 23 explains these tools.

Local Solid Face Tools

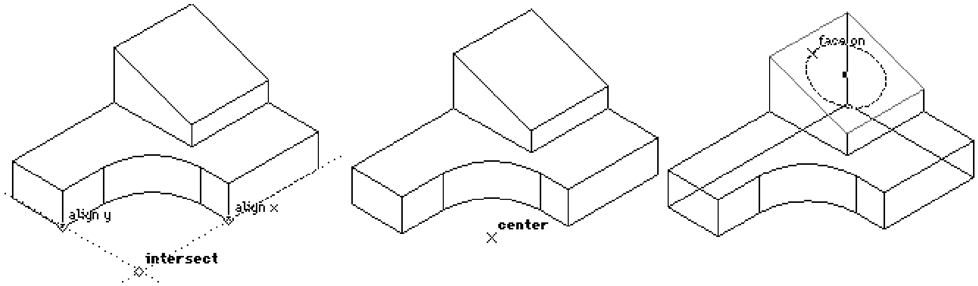
As you create or import geometry you may want to perform various operations on it. The Local Solid Face Tools provide a means to modify individual faces of your solid geometry.



The Local Solid Face subpalette contains the following tools: Draft Solid, Match Face, Move Face, Offset Face, Remove Face, Replace Face, Parting Line and Deform Face (by disatance, to a point or to a curve). Chapter 23 explains these tools.

Drafting Assistant and Solids

The Drafting Assistant recognizes a variety of intelligent snap locations for solids including all hard vertices, user supplied points (grips), hole centers, cylinder centers, alignments, alignment intersections and fillet centers. The Drafting Assistant works the same for a solid as it does for a curve.



Solids, Selection and Display

You can select a solid by clicking on an edge or within a face. Selecting the solid on its edge is considerably faster than selecting within a face (our Designer Elements program has to cast a ray to see if it pierces the solid).

Displaying Solid Edges

Solids display three types of edges, Hard, Silhouette and Smart Silhouette. Hard edges are permanent edges and are present in all views. A Silhouette edge is a temporary edge that displays when the surface normal makes a 90 degree angle with the view normal. Smart Silhouettes display a silhouette only if it does not degrade the display performance.

You can set one of these as the default by choosing ***File>Preferences>Display*** and selecting the option.

Object Resolution

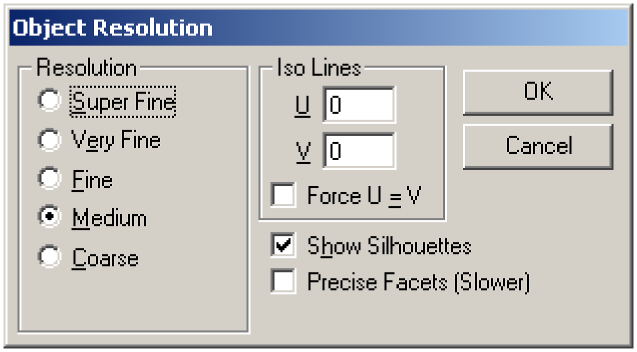
You can increase a solid's edge resolution with the Change Resolution command in the Edit menu.

You can choose one of five resolution settings, Super Fine, Very Fine, Fine, Medium or Coarse. Click a radio button next and then click OK to close the dialog box. The object will change accordingly.

Changing an Object's Resolution

1. Select the object.
2. Choose ***Edit>Change Resolution***.

   The Object Resolution dialog box appears.

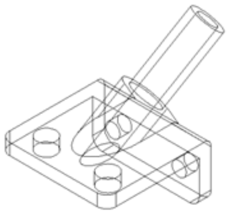


3. Enter the desired number of Iso lines. Check the Show Silhouettes box if desired.

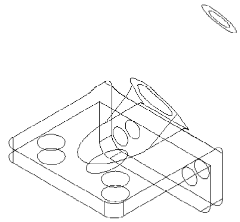
   The # Iso Lines allow you to control the isopram lines drawn for a solid face. These Iso (isopram) lines are constant parameter curves that lie on a surface, typically defined in parameter space. The parameter space coordinate system uses U and V coordinates. A 0 (zero) in both fields turns off Iso lines. The appropriate U/V values may enhance the visual appearance of the face at the expense of drawing speed. The letters, U and V are space coordinate identifiers (U = horizontal, V = vertical) that are the industry standard. The Force U=V check box automatically sets equal number of U and V lines. Entering a new value in one automatically changes the other isoline value.

   The Silhouette check box controls the silhouette edges of objects. Silhouette edges are view dependent and can cause a significant reduction in drawing speed. A check in the box turns the silhouette on.

4. Click OK to close the dialog box and save the new settings.



Silhouette On



Silhouette Off

See Chapter 24 for more information on resolution.

Solid Associativity

Cobalt and Xenon Only

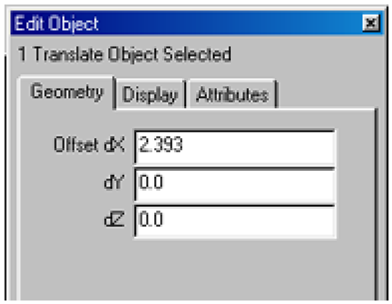
Solid objects are associative to the original curves used to create them. Changing the curve automatically changes the dependent solid. For example, if you create a lathed solid from four curves and change the length of one of them, the lathed solid automatically updates to reflect the change. Use the ***Edit>Remove Links*** command to remove all associativity with the solid if you do not want curve changes to affect the solid. Another important issue to remember with the curve/solid associativity is that if the curve is deleted, the solid deletes as well. Since it is defined by the curve, it is no longer valid.

Solids and Instances

When you perform an operation on a solid and move it to another location or copy a solid you create an instance of the original. When you display the Edit Objects dialog box, the object is referred to as an instance.

Instances and Moving Solids

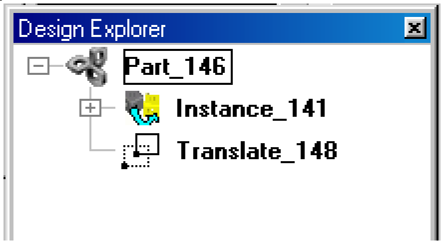
When you create an instance by moving the modified solid to another location, the information appears as an instance operation in the Design Explorer. The original object is still located in previous position, although it is not displayed.



When you select the Instance Translate item in the Design Explorer and open the Edit Objects dialog box the only geometry characteristics provided the offset distance from the original solid.

Instances, Copying Solids and Associativity

Holding down the CTRL (Windows) or OPTION (Macintosh) key while moving the object with the Selection tool creates an instance copy of the original solid. If you display the history tree in the Design Explorer, you see two instance operations. The first, called Instance, is the copying operation. The second, called Instance:Translate is the translation operation moving the copied solid to a different location.



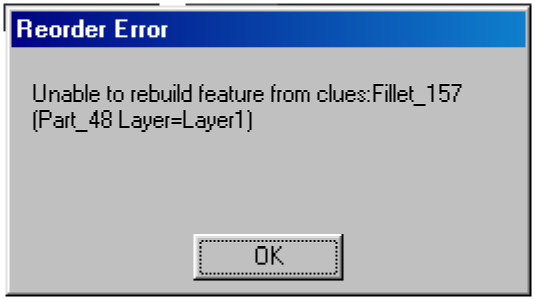
Displaying the Edit Objects dialog box for the Instance item shows there are no unique geometric characteristics. If you open the history tree further you expose the original solid and can view its characteristics through the Edit Objects dialog box.

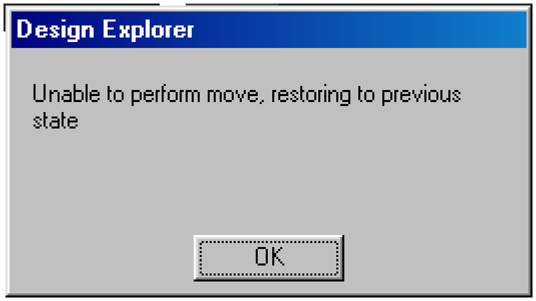
Cobalt and Xenon Only

When you copy a solid, creating an instance, this instance is associative to the original solid. All changes made to the original solid will automatically be made on all instances created from the original. For example, if you added a hole to the original after creating the instance, the hole is also added to all instances. You may need to choose ***Edit>Resolve Links*** for this to occur.

If an operation you perform on the original after creating an instance conflicts with an operation already performed on that instance, you receive two successive errors, a Reorder

Error and a Resolve Links error (shown below). Choose **Edit>Undo** to return to part to the original state.





Cobalt and Xenon Only

When you perform operations on an instance, they are listed in the Design Explorer and do not affect the original. If you break the link between the original and the instance, changes made to the original do affect the instance.

Be aware when you are creating multiple instances whether you are instancing the original part or another instance. This is important if you later decide to break the link between the original and an instance. If there are any instances created from that instance, you are breaking their associativity to the original as well. So, if you only want to break the link between one instance and the original, all other instances should be created from the original solid.

If you do not want to create an instance, select the original solid, choose ***Edit>Copy*** and then ***Edit>Paste***. You can also select the instance and choose ***Edit>Remove Links***. However, the object geometry cannot be edited using the technique.

Cutting and Pasting Solids

When you cut and paste a solid all associativity relationships are lost. After cutting and pasting a part, you will not be able to edit any of its features. Cutting and pasting a solid does, however, provide an easy way to remove a part's history tree, thereby minimizing the amount of memory it consumes.

Cutting and Pasting on the Macintosh

When you cut and paste a solid object into our Designer Elements program on a Macintosh computer, the object is represented in both PICT2 and VS Object form. When both forms are present, you can cut and paste a representation of the solid into other applications and cut and paste a true solid back into our Designer Elements program drawing.

Exporting Solids

If you want to export a solid model created in our Designer Elements program to another application, you must convert the model to a standard format. Our Designer Elements program supports three formats:

1. You can convert the solid into polylines or meshes with the Change Object Type command in the Edit menu. After converting the model, use the DXF export command to create an ASCII DXF file.
2. If you are exporting to an application that supports ACIS, you can generate a SAT file. SAT files contain precise geometry and topology, so the other application can import the solid's data as it was created in our Designer Elements program.
3. You can export solids through the IGES translator as solid entities or as a collection of trimmed surfaces.

Object Types and Edit Objects

Every solid object you create with our Designer Elements program tools are defined by their own characteristics and include geometric characteristics and attributes. These are displayed in the Edit Objects dialog box. While the categories listed in the Attributes tab are identical, the categories listed in the Geometry tab vary with each object.

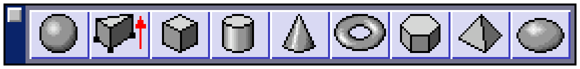
For example: A line created by the Single Line tool includes the following categories: Length, Angle, End 1 for X, Y, and Z, and End 2 for X, Y and Z.

Chapters 18 and 19 introduce you to the solids tools of our Designer Elements program that create the objects. Included with every tool description is a list of the categories displayed for the object when Edit Objects is chosen. For information on using the Edit Objects command and the dialog box, see Chapter 24.

Error Messages

Error Messages provide more feedback when trying to perform operations such as shelling, local face modeling and so on. Our Designer Elements program tries to determine why the operation failed and provides a hint to the feature that may have caused the problem.

Solids Creation Tools - Primitives



Primitives are simple solid shapes. You can create primitives by picking one point, two points or diagonals. Each Designer Elements program primitive is defined by a unique set of characteristics which you can edit.

For each tool you can enter values in the Status Line to define a solid either before or after you create the solid. If you enter the values after you select the tool but before you create the solid, your first click in the drawing area automatically registers all Status Line values. If you enter values in the selected Status Line data field after creating the solid and while the solid is still selected, pressing ENTER (Windows) or RETURN (Macintosh) updates the solid to reflect the new values.

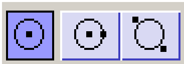
The **Primitive** tools are explained in the order of the palette and include:

- **Sphere**
- **Slab**
- **Block**
- **Cylinder**
- **Cone**
- **Torus**
- **Prism**
- **Pyramid**

- **Ellipsoid**

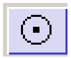
Sphere Primitive Tool

Our Designer Elements program creates a sphere from a center point and radius or from two diagonal points on a bounding box. When you select the **Sphere Primitive** tool, a subpalette appears in the Message Line containing three tools for creating spheres: **Sphere 1 Point**, **Sphere 2 Point** and **Sphere by Diagonals**.

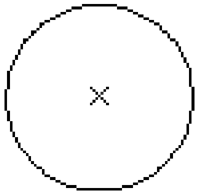


The Drafting Assistant recognizes the points that you pick to define the sphere.

Sphere 1 Point Tool

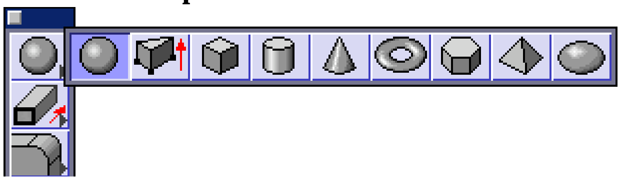


This tool draws a sphere using the center point that you specify and the radius entered in the Status Line.



Using the Sphere 1 Point Tool

1. Select the **Sphere Primitive** tool.

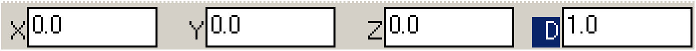


2. Select the **Sphere 1 Point** tool in the Message Line. The Message Line reads: *Sphere Primitive: Enter center point for sphere.*
3. Click a point in the drawing area.

   The sphere is created in the specified location.

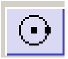
While the sphere is still selected, change the X, Y, Z location of the center point and the radius, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

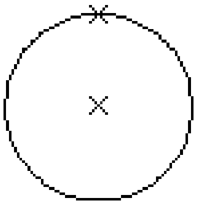
The Status Line contains X, Y and Z data fields for the sphere center point and the Radius data field.



Our Designer Elements program creates one point spheres independently of the construction plane's orientation.

Sphere 2 Point Tool





This tool draws a sphere from the center point and radius that you specify.

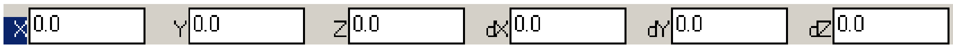
Using the Sphere 2 Point Tool

1. Select the **Sphere Primitive** tool.
2. Select the **Sphere 2 Point** tool in the Message Line. The Message Line reads: *Sphere Primitive: Enter center and radius point for sphere.*
3. Click the center point and the radius point in the drawing area.

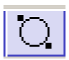
   The sphere is created.

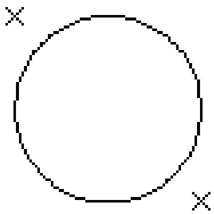
While the sphere is still selected you can change the X, Y, Z location of the center point and dX, dY and dZ values for the radius, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains X, Y and Z data fields for the sphere center point and dX, dY and dZ fields to reference distance and direction to the radius.



Sphere by Diagonals Tool





This tool draws a sphere based on the two corners of the sphere's bounding box that you specify.

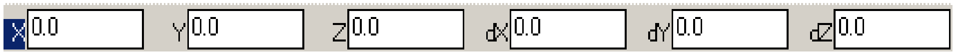
Using the Sphere By Diagonals Tool

1. Select the **Sphere Primitive** tool.
2. Select the **Sphere by Diagonals** tool in the Message Line. The Message Line reads: *Sphere Primitive: Enter diagonal corners for sphere.*
3. Click two points in the drawing area.The sphere is created.

   This Designer Elements program places the sphere's center midway between the two points and calculates the radius from the smallest length of the enclosing box.

While the sphere is still selected you can change the X, Y, Z location of the first corner point and dX, dY and dZ values to the second corner, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains X, Y and Z data fields for the first point and the dX, dY and dZ fields for the distance and direction to the second point.



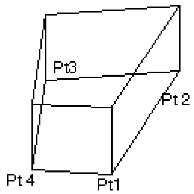
Geometric Characteristics

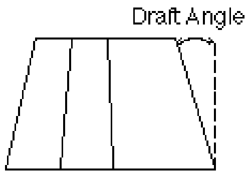
A sphere primitive is created by picking one or two points to specify the sphere's center point, radius or bounding box corners, and is made up of the following characteristics: Center (X, Y, and Z values) and Diameter. This information appears in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the sphere and choose ***Window>Edit Objects*** or double-click on the sphere.

Slab Primitive Tool

You create a solid slab primitive by extruding a series of closed planar points. If the points do not close, our Designer Elements program automatically closes them. Our Designer Elements program also checks for invalid or overlapping slab points. If one or more are detected, no slab is created.

Our Designer Elements program determines the extrusion direction from the order in which you enter the points. Our Designer Elements program constructs the slab in the direction of the right hand rule (see Chapter 2 for information on the right hand rule). Although the slab's points must be coplanar, they do not have to be in the current construction plane. You can also specify a draft angle for the slab.





Using the Slab Tool

1. Select the **Slab Primitive** tool. The Message Line reads: *Slab Primitive: Enter 3 or more points for slab. (Double-click last point).*

2. Click the three or more points to describe the slab. (Be careful that the points are not overlapping.)

   The slab is created with the height and draft angle listed in the Status Line.

While the slab is still selected you can change the height and draft angle in the Status Line, if desired. Press ENTER (Windows) or RETURN (Macintosh) to adjust the slab to the new values.

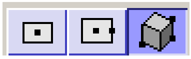
The Status Line contains the Height and Draft Angle data fields.

Geometric Characteristics

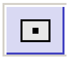
A solid slab primitive is created by picking three or more points for the slab's corners and is made up of the following characteristics: the Height and Draft Angle, and the Defining Points (X, Y, and Z values) of the slab's corner points. This information is listed in the Edit Objects dialog box under the Geometry tab. The section below Defining Points displays the selected point. Each point can be edited individually using the X, Y and Z fields. To display the dialog box, select the slab and choose ***Window>Edit Objects*** or double-click the slab.

Block Primitive Tool

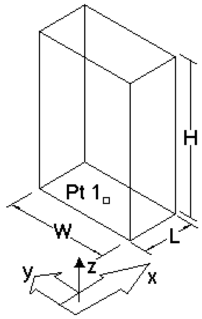
The block primitive is a cubic or rectangular solid. When you select the **Block Primitive** tool a subpalette appears in the Message Line containing three block tools: **Block 1 Point**, **Block 2 Point** and **Block by Diagonals**.



Block 1 Point Tool



This tool draws a block using the center point that you specify and the Height entered in the Status Line.



Using the 1 Point Block Tool

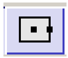
1. Select the **Block Primitive** tool.
2. Select the **Block 1 Point** tool in the Message Line. The Message Line reads: *Block Primitive: Enter center point for block.*
3. Click a point in the drawing area.

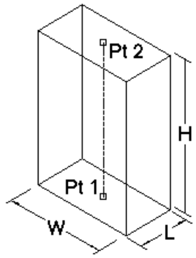
   The block is created with its base centered at the point you chose. The block's orientation is determined by its length, width, and height along the x, y, and z axes of the work plane.

While the block is still selected you can change the X, Y, Z location of the base center point and length, width and height of the block, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z location of the block base center point and Length, Width and Height of the block.

Block 2 Point Tool





This tool draws a block from the center point and radius.

Using the 2 Point Block Tool

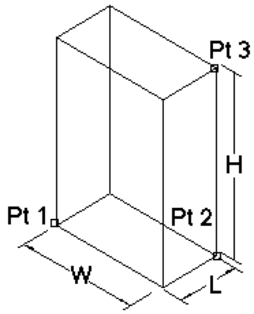
1. Select the **Block Primitive** tool.
2. Select the **Block 2 Point** tool in the Message Line. The Message Line reads: *Block Primitive: Enter center and length points for block.*
3. Click two points in the drawing area.

   The block is created centered on the point chosen with a height equal to the distance between the two points that you chose.

While the block is still selected you can change the X, Y, Z location of the base center point, the dX, dY and dZ to the second point for defining the width and height of the block and the length, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z location of the block base center point, the dX, dY and dZ to the second point for defining the width, height and length of the block.

Block by Diagonals Tool



This tool draws a block based on the two base corners and the height that you specify.

Using the Block by Diagonals Tool

1. Select the **Block Primitive** tool.
2. Select the **Block by Diagonals** tool in the Message Line. The Message Line reads: *Block Primitive: Enter start diagonal point [Shift=Square].*
3. Click two points in the drawing area to specify two diagonal corners of the block's base.

   Hold down the SHIFT key to draw a square. After you click your first point, while holding down the key, The Message Line reads: *locking....* These two points set the block's width and length.
4. After the first two points are placed The Message Line reads: *Block Primitive: Enter height.*
5. Drag to set the height of the block.

   The block is created. If you are working in the top, front or side view, pick a point, and our Designer Elements program will make the block's height equal to its width. If you are working in 3D view, you can select the point to set the desired height.

While the block is still selected you can change the X, Y, Z location of the first corner or the length, width and height of the block, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

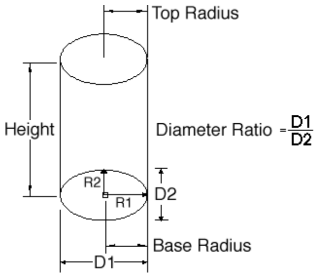
The Status Line contains the X, Y and Z values for the first corner that you chose and the solid's Length, Width and Height.

Geometric Characteristics

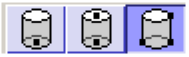
A solid block primitive is created by picking one or two points to specify the center point of the block's base, the block's height, or the locations of two of the block's corners. The block is made up of the following characteristics: Length, Width and Height. This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the block and choose ***Window>Edit Objects*** or double-click the block.

Cylinder Primitive Tool

With our Designer Elements program, you can make circular and elliptical cylinder primitives. The cylinder is defined by a base radius ratio and a height. The base radius ratio defines the ratio between the major radius (R1) and minor radius (R2). A circular cylinder's base ratio equals one. Any other value yields an elliptical cylinder. The graphic gives you a visual representation of these definitions.

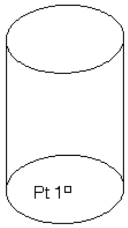


When you select the **Cylinder Primitive** tool, a subpalette appears in the Message Line containing three cylinder tools: **Cylinder 1 Point**, **Cylinder 2 Point** and **Cylinder by Diagonals**.



Cylinder 1 Point Tool

This tool draws a cylinder using the center point that you specify and the radius and height entered in the Status Line.



Using the Cylinder 1 Point Tool

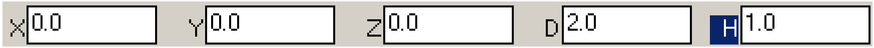
1. Select the **Cylinder Primitive** tool.

2. Select the **Cylinder 1 Point** tool in the Message Line. The Message Line reads: *Cylinder Primitive: Enter center point for cylinder.*
3. Click a point in the drawing area.

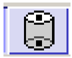
   The cylinder is created with its base centered at the point you chose. The cylinder extends along the z axis of the current construction plane.

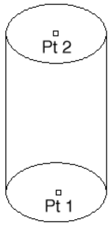
While the cylinder is still selected you can change the X, Y, Z location of the base center point and radius and height of the cylinder, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z values for the cylinder's base center point and its Radius and Height.



Cylinder 2 Point Tool





This tool draws a cylinder from the center point of the base, the second point for the height and radius.

Using the Cylinder 2 Point Tool

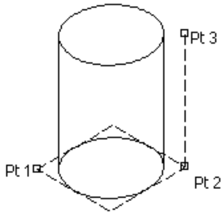
1. Select the **Cylinder Primitive** tool.
2. Select the **Cylinder Two Point** tool in the Message Line. The Message Line reads: *Cylinder Primitive: Enter start and end points for cylinder.*
3. Click two points in the drawing area.

   The cylinder is created with its height is equal to the distance between the two points that you chose and with its top and bottom centered on those points.

While the cylinder is still selected you can change the X, Y, Z location of the base center point, the dX, dY and dZ to the second point for defining the distance and direction of the cylinder and the radius, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z values for the base center point, the dX, dY and dZ values for the distance and direction to the second point, and the Radius.

Cylinder by Diagonals Tool



This tool draws a cylinder based on the two base corners and the height.

Using the Cylinder by Diagonals Tool

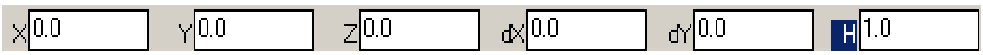
1. Select the **Cylinder Primitive** tool.
2. Select the **Cylinder by Diagonals** tool in the Message Line. The Message Line reads: *Cylinder Primitive: Enter start diagonal point [Shift=Square]*.
3. Click two points in the drawing area to specify two diagonal corners of the cylinder base's bounding box.

   Hold down the SHIFT key to draw a circular base. After you click your first point, The Message Line reads: *locking...*. The distance between the two points are used to calculate the cylinder's diameter.
4. The Message Line reads: *Cylinder Primitive: Enter height*.
5. Drag to set the height of the cylinder.

   The cylinder is created. If you are working in the top, front or side view, pick a point and our Designer Elements program will make the cylinder's height equal to that of the last cylinder you drew (or equal to one inch if this is the first cylinder you draw). If you are working in a 3D view you can select the point to set the desired height.

While the cylinder is still selected you can change the X, Y, Z location of the cylinder's base center point, the dX, dY, and dZ values for the distance and direction to the second point, and the cylinder's height, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z values for the cylinder's base center point, the dX, dY and dZ values for the distance and direction to the second point and the Height.



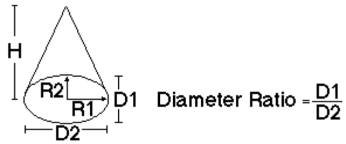
Geometric Characteristics

A cylinder primitive is created by picking one point to specify the center point of the cylinder's base, two points for the center point and height, or three points for the corners of the bounding box and the cylinder's height. It is made up of the following characteristics: Height, Diameter (Major axis) and Diameter Ratio (D1/D2 or R1/R2). This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the cylinder once and choose ***Window>Edit Objects*** or double-click the cylinder.

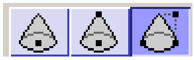
Cone Primitive Tool

The cone primitive has an elliptical base at one end and a point at the other. Like the cylinder, the cone is defined by a base radius ratio and a height.

The ratio defines the relationship between the major radius (R1) and minor radius (R2). A circular cone's base ratio equals one. Any other value yields an elliptical cone.

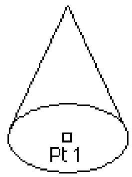


When you select the **Cone Primitive** tool a subpalette appears in the Message Line containing three cone tools: **Cone 1 Point**, **Cone 2 Point** and **Cone by Diagonals**.



Cone 1 Point Tool

This tool draws a cone using the center point that you specify and the radius and height entered in the Status Line.



Using the Cone 1 Point Tool

1. Select the **Cone Primitive** tool.

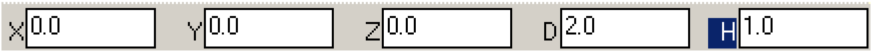
2. Select the **Cone 1 Point** tool in the Message Line. The Message Line reads: *Cone Primitive: Enter center point for cone.*

3. Click a point in the drawing area.

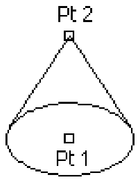
The cone is created with its base centered at the point you chose. The cone extends along the z axis of the current work plane.

While the cone is still selected you can change the X, Y, Z location of the base center point and radius and height of the cone, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z values for the cone's base center point and its Radius and Height.



Cone 2 Point Tool



This tool draws a cone from the center point of the base at the height and radius you specify.

Using the Cone 2 Point Tool

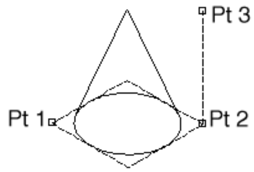
1. Select the **Cone Primitive** tool.
2. Select the **Cone 2 Point** tool in the Message Line. The Message Line reads: *Cone Primitive: Enter start and end points for cone.*
3. Click in the drawing area to set the start (center) point and the end (tip) point of the cone.

   The cone is created with its height equal to the distance between the two points that you chose and with its top and bottom centered on those points.

While the cone is still selected you can change the X, Y, Z location of the base center point, the dX, dY and dZ to the second point for defining the distance and direction of the cone and the radius, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z values for the base center point, the dX, dY, dZ values for the distance and direction to the second point (tip) and the Radius.

Cone by Diagonals Tool



This tool draws a cone based on the two base corners and the height that you specify.

Using the Cone by Diagonals Tool

1. Select the **Cone Primitive** tool.
2. Select the **Cone by Diagonals** tool in the Message Line. The Message Line reads: *Cone Primitive: Enter start diagonal point [Shift=Square].*
3. Click two points in the drawing area to specify two diagonal corners of the cone base's bounding box.

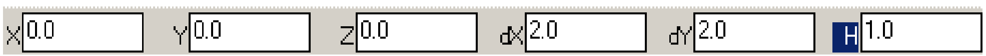
   Hold down the SHIFT key to draw a circular base. After you click your first point, while holding down the key, The Message Line reads: *locking....*" The distance between the two points are used to calculate the cone's diameter.

   After doing this step, The Message Line reads: *Cone Primitive: Enter height*.
4. Drag to set the height of the cone.

   The cone is created. If you are working in the top, front or side view, pick a point, and our Designer Elements program will make the cone's height equal to that of the last cone you drew (or equal to one, if this is the first cone you drew). If you are working in a 3D view, you can select the point to set the desired height.

While the cone is still selected you can change the X, Y, Z location of the cone's base center point, the dX, dY, and dZ values for the distance and direction to the second point (tip), and the cone's height, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

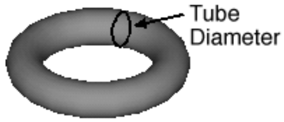
The Status Line contains the X, Y and Z values for the cone's base center point, the dX, dY and dZ values for the distance and direction to the second point, and the Height.

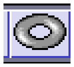


Geometric Characteristics

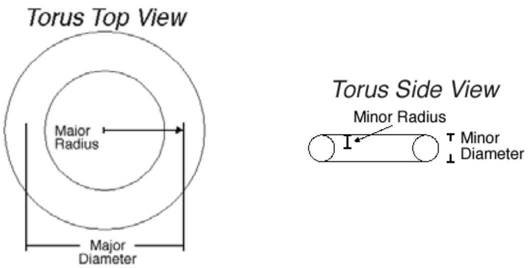
A cone primitive is created by picking one point to specify the center point of the cone's base, two points for the center point and height, or three points for the corners of the bounding box and the cone's height. It is made up of the following characteristics: Height, Top Diameter, Diameter Ratio (D1/D2 or R1/R2) and Bottom Diameter. This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the cone once and choose ***Window>Edit Objects*** or double-click the cone.

Torus Primitive Tool

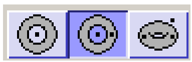




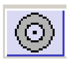
The torus primitive looks like a doughnut. It is generated by sweeping a circle around another implied circle. The torus is defined by a center point, the diameter of the torus, called the Major Diameter, and the tube’s diameter, called the Minor Diameter. The graphics below illustrate these dimensions.

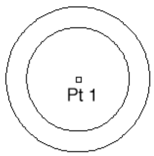


When you select the **Torus Primitive** tool, a subpalette appears in the Message Line with three cone tools: **Torus 1 Point**, **Torus 2 Point** and **Torus by Diagonals**.



Torus 1 Point Tool





This tool draws a torus using the center point you specify and the Major Diameter (D1) and the Minor Diameter (d2) of the Status Line.

Using the Torus 1 Point Tool

1. Select the **Torus Primitive** tool.

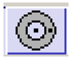
2. Select the **Torus 1 Point** tool in the Message Line. The Message Line reads: *Torus Primitive: Enter center point for torus.*
3. Click in the drawing area to set the center.

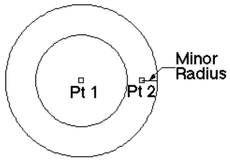
   A torus is created with its center at the point you chose. The torus lies perpendicular to the z axis of the current work plane.

While the torus is selected you can change the X, Y, Z location of the base center point, the Major diameter (D1) and the Minor diameter (d2), if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z values for the torus' center point, D1 (Major diameter) and d2 (Minor diameter).

Torus 2 Point Tool





This tool draws a torus from the center point and radius.

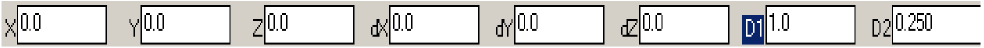
Using the 2 Point Torus Tool

1. Select the **Torus Primitive** tool.
2. Select the **Torus 2 Point** tool in the Message Line. The Message Line reads: *Torus Primitive: Enter center and radius point for torus.*
3. Click the center point and the radius point in the drawing area.

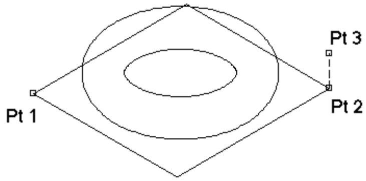
   A torus is created with a center point at the first point and the major diameter at the second point.

While the torus is still selected you can change the X, Y, Z location of the base center point, the dX, dY and dZ to the second point for defining the distance and direction, the Major Diameter (D1) and Minor Diameter (d2), if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the values.

The Status Line contains the X, Y and Z values for the torus's center point, the dX, dY and dZ values for the distance and direction to the second point, the D1 (Major Diameter) and d2 (Minor diameter).



Torus by Diagonals Tool



This tool draws a torus from the center point, major radius and minor radius.

Using the Torus by Diagonals Tool

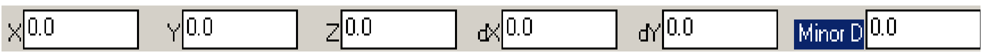
1. Select the **Torus Primitive** tool.
2. Select the **Torus by Diagonals** tool in the Message Line. The Message Line reads: *Torus Primitive: Enter start diagonal point [Shift=Square]*.
3. Click twice to specify two diagonal corners of the torus' bounding box.

   Hold down the SHIFT key to draw a square bounding box. After you click your first point The Message Line reads: *locking...*. The distance between the two points are used to calculate the torus' diameter.
4. The Message Line reads: *Torus Primitive: Enter height*.
5. Drag to set the height of the torus.

   If you are working in the top, front or side view, pick a point. Our Designer Elements program will calculate the torus's Minor Diameter based on the location of that point. If you are working in a 3D view, you can select the point to set the Minor Diameter.

While the torus is still selected you can change the X, Y, Z location of the torus' center point, the dX, dY, and dZ values for the distance and direction to the second point (Major Diameter), and the torus' Minor Diameter, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

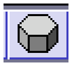
The Status Line contains the X, Y and Z values for the torus center point, the dX, dY and dZ values for the distance and direction to the second point D1 (Major Diameter) and the d2 (Minor Diameter).

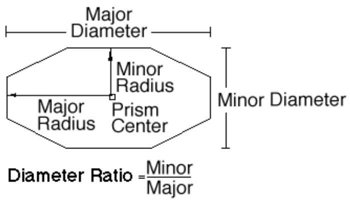


Geometric Characteristics

A torus primitive is created by picking one point to specify the center point of the torus, two points for the center point and Major Diameter, or three points for the corners of the bounding box and the torus' Minor Diameter. It is made up of the following characteristics: Major Diameter, Minor Diameter, and Center (X, Y, and Z values). This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the torus and choose ***Window>Edit Objects*** or double-click the torus.

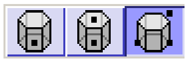
Prism Primitive Tool



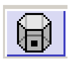


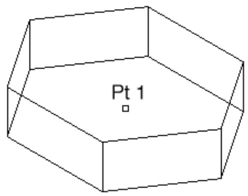
The prism primitive is composed of three or more sides, all perpendicular to the solid's base.

When you select the **Prism Primitive** tool a subpalette appears in the Message Line containing three prism tools: **Prism 1 Point**, **Prism 2 Point** and **Prism by Diagonals**.



Prism 1 Point Tool





This tool draws a prism using the center point that you specify and the radius, height and number of sides listed of the Status Line.

Using the Prism 1 Point Tool

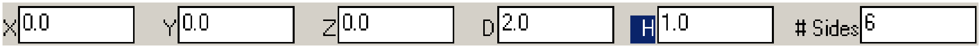
1. Select the **Prism Primitive** tool.

2. Select the **Prism 1 Point** tool in the Message Line. The Message Line reads: *Prism Primitive: Enter center point for prism*.
3. Click in the drawing area to set the base's center point.

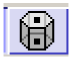
   A prism is created with its base centered at the point you chose. The prism extends along the z axis of the current work plane.

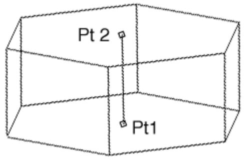
While the prism is still selected you can change the X, Y, Z location of the base center point, the radius, height and number of sides, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z values for the prism's base center point and its Radius, Height, and # of sides.



Prism 2 Point Tool





This tool draws a prism from the center point and at the height you specify.

Using the 2 Point Prism Tool

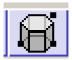
1. Select the **Prism Primitive** tool.
2. Select the **Prism 2 Point** tool in the Message Line. The Message Line reads: *Prism Primitive: Enter start and end points for prism*.
3. Click in the drawing area to set the start point (center) and the end point (height).

   A prism is created with its height equal to the distance between two points of your chosing and its top and bottom centered on those points.

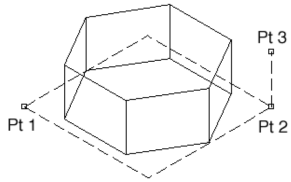
While the prism is still selected you can change the X, Y, Z location of the base center point, the dX, dY and dZ to the second point for defining the distance and direction, the radius and the number of sides, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z values for the prism's base center point, the dX, dY and dZ values for the distance and direction to the second point and the prism's Radius and # of sides.

Prism by Diagonals Tool



This tool draws a torus from two corner points, representing the prism's bounding box and a third point for the height.



Using the Prism by Diagonals Tool

1. Select the **Prism Primitive** tool.
2. Select the **Prism by Diagonals** tool in the Message Line. The Message Line reads: *Prism Primitive: Enter start diagonal point [Shift=Square].*
3. Click two points in the drawing area to specify two diagonal corners of the prism base's bounding box.

   Hold down the SHIFT key to draw a square bounding box. After you click your first point, The Message Line reads: *locking....* The distance between the two points are used to calculate the prism's diameter.
4. The Message Line reads: *Prism Primitive: Enter height.*
5. Drag to set the height of the prism.

   If you are working in the top, front or side view, pick a point, and our Designer Elements program will make the prism's height equal to that of the last prism you drew (or equal to one inch if this is the first prism you draw). If you are working in 3D view you can select the point to set the desired height.

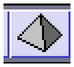
While the prism is still selected you can change the X, Y, Z location of the prism's base center point, the dX, dY, and dZ values for the distance and direction to the second point and the prism's height, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

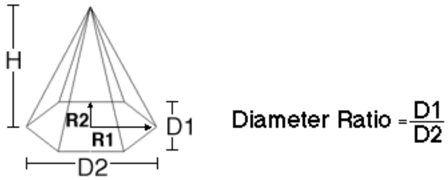
The Status Line contains the X, Y and Z values for the prism's base center point, the dX, dY and dZ values for the distance and direction to the second point and the prism's Height.

Geometric Characteristics

A prism primitive is created by picking one point to specify the center point of the prism's base, two points for the center point and height, or three points for the corners of the bounding box and the prism’s height. It is made up of the following characteristics: Height, Diameter, Diameter Ratio (D2/D1 or R2/R1) and Sides. This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the prism and choose ***Window>Edit Objects*** or double-click the prism.

Pyramid Primitive Tool

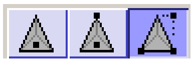




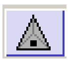
A pyramid contains three or more flat faces which converge to a common vertex at the tip.

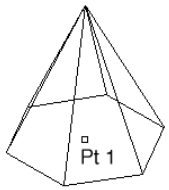
A pyramid is created by defining the diameter of the base, the height and the number of sides.

When you select the **Pyramid Primitive** tool a subpalette appears in the Message Line containing three pyramid tools: **Pyramid 1 Point**, **Pyramid 2 Point** and **Pyramid by Diagonals**.



Pyramid 1 Point Tool





This tool draws a pyramid using the center point that you specify and the radius, height and number of sides listed of the Status Line.

Using the 1 Point Pyramid Tool

1. Select the **Pyramid Primitive** tool.

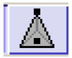
2. Select the **Pyramid 1 Point** tool in the Message Line. The Message Line reads: *Pyramid Primitive: Enter center point for pyramid*.
3. Click in the drawing area to set the center.

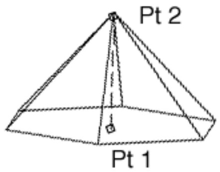
A pyramid is created with its base centered at the point you chose. The pyramid extends along the z axis of the current work plane.

While the pyramid is still selected you can change the X, Y, Z location of the base center point, the radius, height and number of sides, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z values for the pyramid's base center point and its radius, height and **#** of sides.

Pyramid 2 Point Tool





This tool draws a pyramid from the base center point and the height.

Using the 2 Point Pyramid Tool

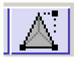
1. Select the **Pyramid Primitive** tool.
2. Select the **Pyramid Two Point** tool in the Message Line. The Message Line reads: *Pyramid Primitive: Enter start and end points for pyramid*.
3. Click two points in the drawing area.

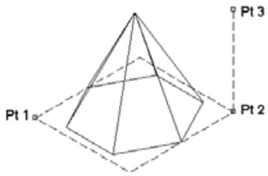
   A pyramid is created whose height is equal to the distance between the two points that you chose and whose top and bottom are centered on those points.

While the pyramid is still selected you can change the X, Y, Z location of the base center point, the radius, height and number of sides, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z values for the pyramid's base center point, the dX, dY and dZ characteristics for the distance and direction to the second point and the pyramid's radius and **#** of sides.

Pyramid by Diagonals Tool





This tool draws a pyramid from the corner of the pyramid's bounding box and the height that you specify.

Using the Pyramid By Diagonals Tool

1. Select the **Pyramid Primitive** tool.
2. Select the **Pyramid by Diagonals** tool in the Message Line. The Message Line reads: *Pyramid Primitive: Enter start diagonal point [Shift=Square].*
3. Click two points in the drawing area to specify two diagonal corners of the pyramid base's bounding box.

   Hold down the SHIFT key to draw a square bounding box. After you click your first point, The Message Line reads: *locking....* The distance between the two points are used to calculate the pyramid's diameter.
4. The Message Line reads: *Pyramid Primitive: Enter height.*
5. Drag to set the height of the pyramid.

   If you are working in the top, front or side view, pick a point, and a pyramid is created with a height equal to that of the last pyramid you drew (or equal to one inch if this is the first pyramid you draw). If you are working in a 3D view you can select the point to set the desired height.

While the pyramid is still selected you can change the X, Y, Z location of the pyramid's base center point, the dX, dY, and dZ values for the distance and direction to the second point and the pyramid's height, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z values for the pyramid's base center point, the dX, dY and dZ values for the distance and direction to the second point and the pyramid's Height.

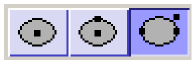
Geometric Characteristics

A pyramid primitive is created by picking one point to specify the center point of the pyramid's base, two points for the center point and height, or three points for the corners of the bounding box and the pyramid's height. It is made up of the following characteristics: Height, Top Diameter, Diameter Ratio (D2/D1 or R2/R1), Bottom Diameter and Sides. This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the pyramid and choose ***Window>Edit Objects*** or double-click the pyramid.

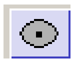
Ellipsoid Primitive Tool

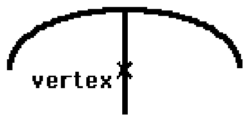
An ellipsoid primitive is created from a center point, center point and radius, or two diagonal points on a bounding box.

When you select the **Ellipsoid Primitive** tool a subpalette appears in the Message Line containing three tools for creating ellipsoids: **Ellipsoid 1 Point**, **Ellipsoid 2 Point** and **Ellipsoid by Diagonals**.



Ellipsoid 1 Point Tool





This tool creates an ellipsoid using the center point that you specify and the X, Y and Z diameters entered in the Status Line. The Drafting Assistant displays a *vertex* notation at the center point.

Using the Ellipsoid 1 Point Tool

1. Select the **Ellipsoid Primitive** tool.

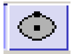
2. Select the **Ellipsoid 1 Point** tool in the Message Line. The Message Line reads: *Ellipsoid Primitive: Enter center point for ellipsoid*.
3. Click a point in the drawing area.

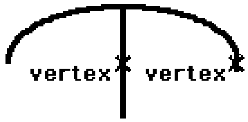
   The ellipsoid appears.

While the ellipsoid is still selected you can change the X, Y and Z location of the center point and the diameters. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z values for the center point and DiamX, DiamY and DiamZ for the ellipsoid diameters.

Ellipsoid 2 Point Tool





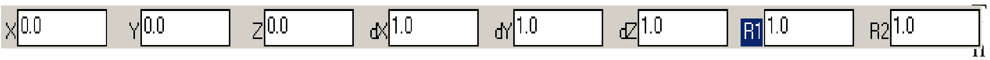
This tool creates an ellipsoid using the center point and the radius you specify. The Drafting Assistant displays *vertex* notations at the center point and radius.

Using the Ellipsoid 2 Point Tool

1. Select the **Ellipsoid Primitive** tool.
2. Select the **Ellipsoid 2 Point** tool in the Message Line. The Message Line reads: *Ellipsoid Primitive: Enter center and radius point for ellipsoid.*
3. Click the center point and radius point in the drawing area.

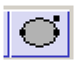
   The ellipsoid appears.

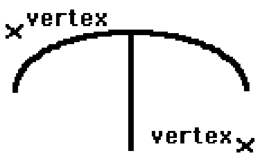
While the ellipsoid is still selected you can change the X, Y and Z location of the center point, the dX, dY and dZ values for the first radius and the two radii not specified in the operation (R1 and R2). Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.



e Status Line contains the X, Y and Z values for the center point, the dX, dY, dZ, R1 and R2 values.

Ellipsoid by Diagonals Tool





This tool creates an ellipsoid based on two corners of the ellipsoid's bounding box and the height you specify. The Drafting Assistant displays *vertex* notations at the bounding box corners.

Using the Ellipsoid by Diagonals Tool

1. Select the **Ellipsoid Primitive** tool.

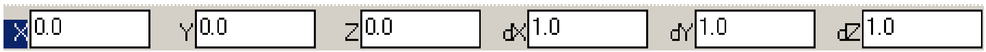
2. Select the **Ellipsoid by Diagonals** tool in the Message Line. The Message Line reads: *Ellipsoid Primitive: Enter diagonal corners for ellipsoid. Enter start diagonal point [Shift = Square].*
3. Click two points in the drawing area to specify two diagonal corners of the ellipsoid's bounding box.

   Hold down the SHIFT key to draw a rectangular base. After you click your first point The Message Line reads: *locking...* The distance between the two points are used to calculate the ellipsoid's diameter.
4. The Message Line reads: *Ellipsoid Primitive: Enter height.*
5. Drag to set the height of the ellipsoid.

   The ellipsoid appears. If you are working in the top, front or side view, pick a point, and our Designer Elements program will make the ellipsoid's height equal to that of the last ellipsoid you drew (or equal to one inch if this is the first ellipsoid you draw). If you are working in a 3D view you can select the point to set the desired height.

While the ellipsoid is still selected you can change the X, Y, Z location of the ellipsoid's base center point, the dX, dY, and dZ values for the distance and direction to the second point, and the ellipsoid's height. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains the X, Y and Z values for the cylinder's base center point, the dX, dY and dZ values for the distance and direction to the second point.



Geometric Characteristics

An ellipsoid primitive is created by picking one point to specify the center of the primitive's base, two points for the center point and height, or three points for the corners of the bounding box and the ellipsoid's height. It is made up of the following characteristics: Center X, Y and Z and Diameter 1, Diameter 2 and Diameter 3.

For ellipsoids created using the **Ellipsoid 1 Point** tool, DiamX, DiamY and DiamZ correspond to Diameter1, Diameter2 and Diameter3. For ellipsoids created with the **Ellipsoid 2 Point** tool, the dX, dY, dZ values are used to determine the Diameter 1 value. R1 corresponds to Diameter 2 and R2 corresponds to Diameter 3.

This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the cylinder and choose ***Window>Edit Objects*** or double-click on the cylinder.

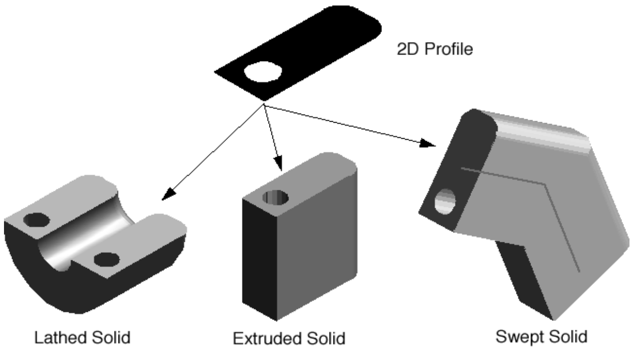
Solids Creation Tools - Profiles

A profile is a closed section created from curves, polygons or surfaces (planar and non-planar). You can lathe, extrude, or sweep a profile to create a solid.

Tech Note:

The ASIC kernal has been improved such that you may now use grouped profiles as well as grouped curves for profiles..

When you make a solid from a profile, our Designer Elements program creates a parent/child relationship **(Cobalt™ and Xenon™ only)** between the profile (the parent) and the solid (the child). If you change the parent, our Designer Elements program automatically regenerates the child **(Cobalt™ and Xenon™ only)**. For example, if you lathe a rounded rectangle polygon, you can change the polygon's radius and the solid will update to the new size **(Cobalt™ and Xenon™ only)**. The graphic here shows how you can use the same 2D profile to create three different objects.



Our Designer Elements program supports five tools for creating solids from profiles. For each tool you can enter values in the Status Line to define a solid, either before or after you create the solid. If you enter the values after you select the tool but before you create the solid, your first click in the drawing area automatically registers all Status Line values. If you enter values in the selected Status Line data field

Tech Note:

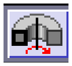
These tools do not support using the *Select Chain* command to select multiple grouped profiles. However can can extrude multiple grouped profiles at one time.

after creating the solid and while the solid is still selected, pressing ENTER (Windows) or RETURN (Macintosh) updates the solid to reflect the new values.

The topics explained include the following:

- **Lathed Solid** tool
- **Extrude Solid** tool
- **Swept Solid** tool
- **Swept Solid 2 Rail** tool
- **Cutout Feature** tool
- **Protruded Feature** tool
- **Skinned Solid** tool
- **Pipe Solid** tool
- Solids from Profile Curve Checks

Lathed Solid Tool



The Lathed Profile tool creates a solid by revolving a profile around an axis line. You can lathe profiles composed of individual, grouped or nested group curves. You cannot lathe a profile that intersects itself, as can happen with splines and polygons.

Using the Lathed Solid Tool

Before you can use this tool you must first create a profile and a curve for lathe axis.

1. Click the Lathe Solid tool. *The Message Line reads: Lathed Solid: Select objects to lathe [Shift=Extend].*

2. The Status Line contains the Degree and Draft Angle fields. Enter the desired values for your lathed solid. Tab between data fields.

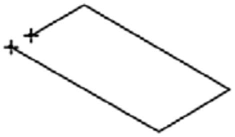
   When the draft angle is zero, the sides of the solid lie parallel to the profile normal. If the draft angle is positive, the sides will be forced outward from the normal. Negative draft angles force the sides inward towards the normal.

3. Pick a curve or curves to lathe. If you are selecting more than one curve, hold down the SHIFT key before selecting the first curve.

   The curves making up the profile must be closed. If they are not, this message appears:

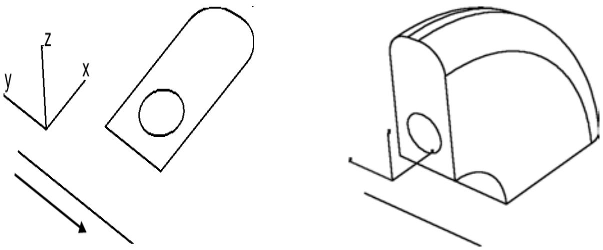
   Our Designer Elements program places markers at the gap locations so you can easily find and close them.

   The Message Line now reads, Lathed Solid: Pick line for lathe axis [Shift=Extend].

4. Pick the line around which to rotate the curves and make the solid object.

   Our Designer Elements program determines the direction of revolution from the direction in which the line was drawn. The left graphic shows a view of a profile and a line drawn in the direction of the arrow. The right graphic shows the profile lathed 90°.

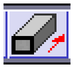
   After you click the line, the object is lathed.

While the object is still selected you can edit the number of degrees in the revolution angle and the draft angle for the entire solid. Enter the new values and press ENTER (Windows) or RETURN (Macintosh) and the solid updates.

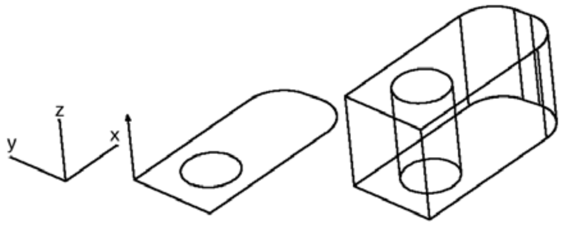
Geometric Characteristics

A lathed profile solid is created by picking one or more curves from which to construct the solid and a line about which to rotate the curves. The lathed solids is made up of the following characteristics: Lathe Angle and Draft Angle. This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the lathed solid and choose **Window>Edit Objects** or double-click the lathed solid.

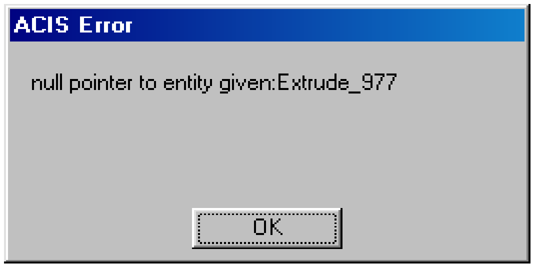
Extrude Solid Tool



The Extrude Solid tool extends the selected profile along a vector. You can extrude profiles composed of lines, circles, ellipses, closed splines, polygons, grouped curves (including nested groups) and surfaces.



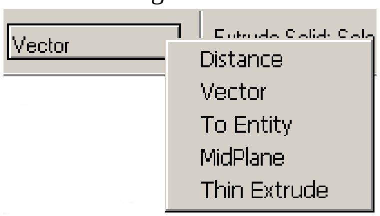
Be careful when selecting circles for extrusion. If one of them is actually a sphere you will receive the following error message:



This tool does not support extruding a non-planar surface with a non-zero draft angle. This message displays if you attempt to do so.

In our Designer Elements program, this tool uses the plane of the selected profile for the extrusion rather than the work plane normal, providing greater flexibility.

In the Message Line there are five extrusion options:



Distance Extrudes normal to the plane of the profile a specified distance. The distance can be positive or negative.

Vector Extrudes a profile along a vector whose distance is defined by the user supplied points.

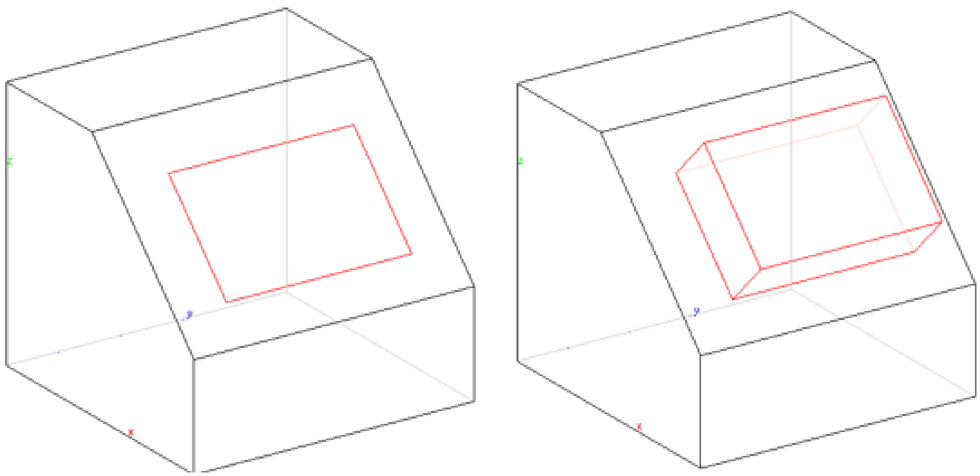
Mid Plane Extrudes a profile in both directions using the plane of the profile.

To Entity Extrudes along a vector terminating at a surface or solid. The terminating body is not unioned with the extruded body.

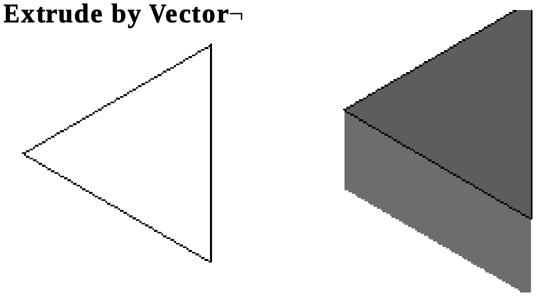
Thin Extrudes an open or closed profile by a vector and thickness.

Below are examples of each type of extrusion

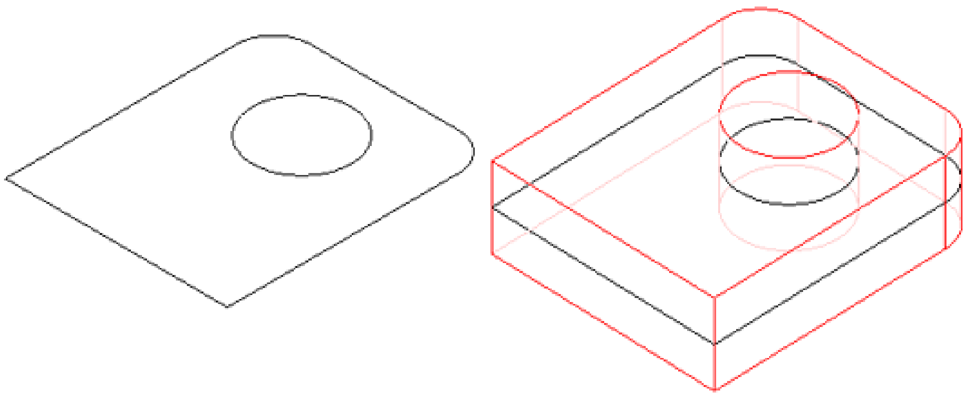
Extrude by Distance



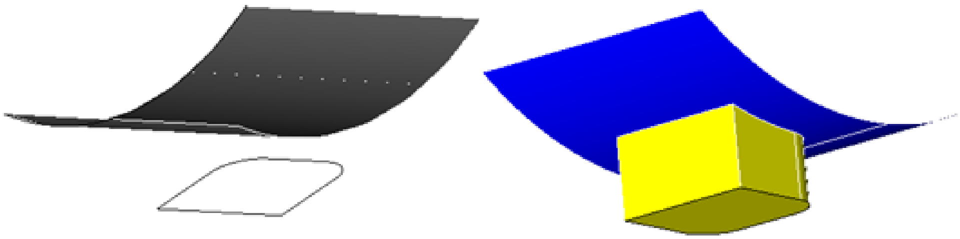
Extrude by Vector¬



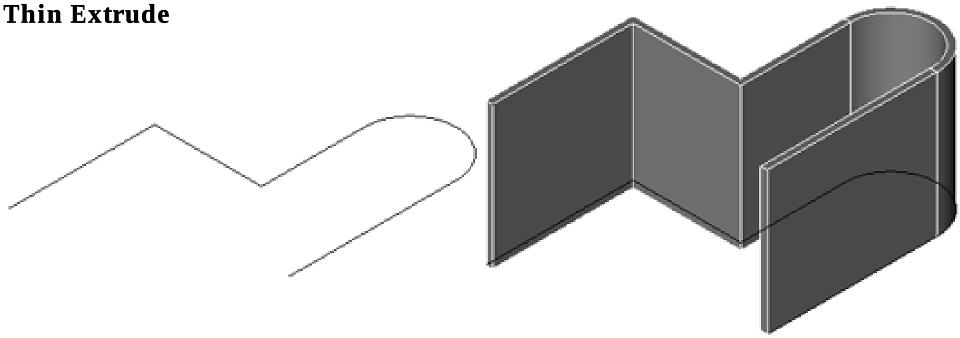
Extrude by Mid Plane



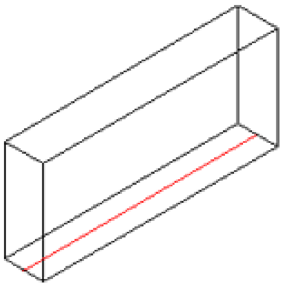
Extrude to Entity



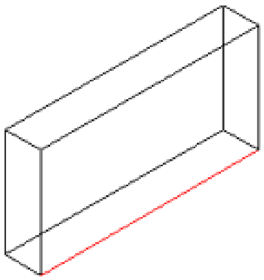
Thin Extrude



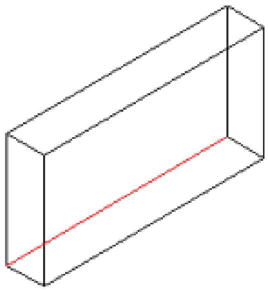
Note: The Thin Extrude tool has a toggle key that changes the location of the extrude. Use the CRTL key (Windows) or the OPTION key (Macintosh) to toggle the extrusion between the right side, left side or offset (mid plane) directions. Examples of the toggle directions are below.



Offset/Mid Plane

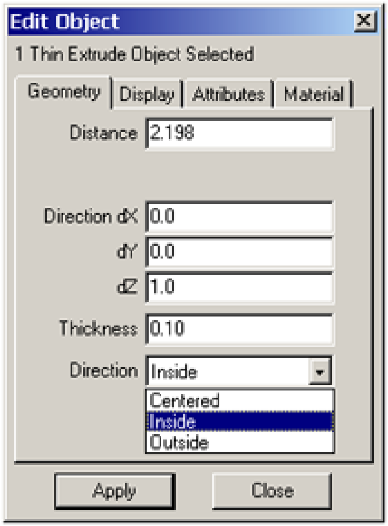


Right Side Extrude



Left Side Extrude

You have access to these toggle directions in the Edit Objects dialog box. However you must access the Edit Objects box through the Design Explorer in order to change these directions.



Tech Note:

Some draft angles and extrusion heights will cause the solid to self intersect. If this happens you must reduce the draft angle or increase the extrusion height.

If you select multiple profiles and one of them is not planar, a warning will appear. You must correct this or you will receive an ACIS error.

Using the Extrude Solid Tool

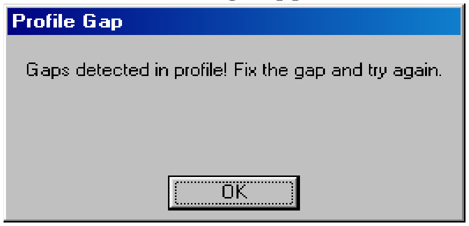
Before you can use this tool you must first create a profile to extrude.

1. Click the Extrude Profile tool. *The Message Line reads: Extrude Solid: Select close objects to extrude by vector [Shift=Extend].*

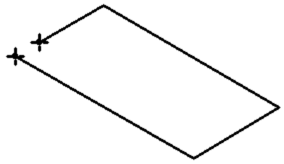
2. The Status Line contains the dX, dY and dZ values, the Distance of the extrusion vector and the Draft Angle. Enter the desired Distance and the Draft Angle for your extruded solid. Tab between data fields.

When the draft angle is zero, the sides of the solid lie parallel to the profile normal. If the draft angle is positive, the sides will be forced outward from the normal. Negative draft angles force the sides inward towards the normal.

3. Pick one or more curves to extrude. As the Message Line indicates, they must be closed. If they are not, this message appears:



Our Designer Elements program places markers at the gap locations so you can easily find and close them.



The Message Line now reads, Extrude Solid: Specify two points for extrusion direction and length.

4. Pick two points on the screen. The extruded solid is created.

   If the extrusion vector defined by is points travels in a positive direction, the extrusion lies along the profile's normal. If the extrusion vector is negative, the extrusion direction lies opposite the profile's normal.

   The following error message appears if you choose points that are tangential to the profile:

While the object is still selected you can edit the dX, dY, and dZ values for the extrusion vector's of the extrusion vector's distance and the draft angle for the entire solid. Enter the new values and press ENTER (Windows) or RETURN (Macintosh) and the solid updates.

Tech Note:

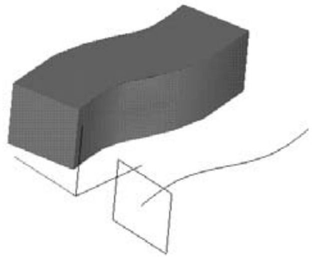
You must enter the extrusion distance for each new profile recreated even if the distance is the same as the previous extrusion.

Geometric Characteristics

An extruded profile solid is created by picking one or more curves from which to construct the solid, and then picking two points to indicate the extrusion's distance and direction. It is made up of the following characteristics: Distance, Draft Angle, and the Direction of the vector (DX, DY and DZ). This information is listed in the

Edit Objects dialog box under the Geometry tab. To display the dialog box, select the extruded solid and choose **Window>Edit Objects** or double-click the extruded solid.

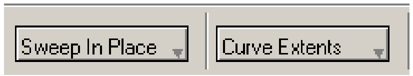
Swept Solid Tool



The Swept Solid tool creates a solid by sweeping the profile along a curve. You can extrude profiles composed of lines circles, ellipses, closed splines, polygons, grouped (including nested groups) curves and surfaces.

Important: Be careful that there are no gaps or overlapping curves in the profile.

The Swept Solid tool has two pull down options in the Message Line. The first pull down controls how the profile orientation as it is swept along the path. This pull down has three options:



- **Sweep In Place** - The profile is not translated or aligned to the path.
- **Sweep Perp** - The profile is translated and aligned perpendicular to the path
- **Sweep Rigid** - The profile orientation is maintained regardless of path tangency.

The second pull down controls how the profile is terminated. This pull down has three options:

- **Curve Extents** - The profile is swept across the entire curve extents.
- **To Body** - The profile is terminated at a surface or solid.
- **Between Points** - The user specifies two points on the path.

Note: Sweep Perpendicular was the default sweep behaviour in previous version of Cobalt and Vellum Solids.

Using the Swept Solid Tool

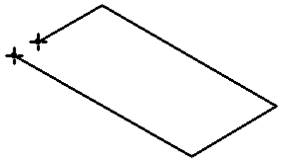
Before you use this tool you must first create a profile and a curve for the sweep.

1. Click the Swept Solid tool. The Message Line reads: *Swept Solid: Select closed profile to sweep [Shift=Extend].*

2. The Status Line contains the Twist Angle and Draft Angle data fields. Enter the twist angle through which you want your solid to twist from beginning to end and a draft angle if desired.

3. Pick one or more curves for the profile. As the Message Line indicates, they must be closed. If they are not, this message appears:

Our Designer Elements program places markers at the gap locations so you can easily find and close them.



The Message Line now reads, *Swept Solid: Enter location for sweep profile origin*.

4. Pick the point on the screen where you want the sweep to begin.

The Message Line now reads, *Swept Solid: Pick curve(s) for sweep path [Shift=Extend]*.

5. Pick the curve or curves along which to sweep the profile.

The swept solid is created.

While the object is still selected you can edit the twist and draft angles. Enter the new values and press ENTER (Windows) or RETURN (Macintosh) and the solid is adjusted.

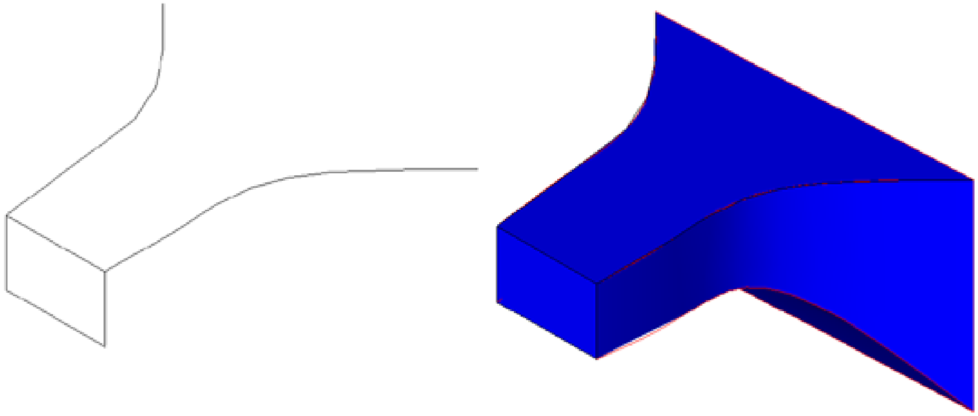
Geometric Characteristics

A swept profile solid is created by picking one or more curves from which to construct the solid, picking the point where the sweep will start, and then picking the curve along which the profile will sweep. It has two characteristics, Twist Angle and Draft Angle. This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the swept solid once and choose **Window>Edit Objects** or double-click the swept solid.

Sweep 2 Rail (Paths) Solid

The Sweep 2 Rail Solid tool creates a solid by sweeping the profile along two curve. You can extrude profiles composed of lines circles, ellipses, closed splines, polygons, grouped curves (including nested groups) and surfaces.

Important: Be careful that there are no gaps or overlapping curves in the profile.



Using the Sweep 2 Rail Solid Tool

Before you use this tool you must first create a profile and a curve for the sweep.

1. Select the Sweep 2 Rail Solid tool. The Message Line reads: *Sweep 2 Rail Solid: Pick curves or group to sweep. [Ctrl = Maintain Height][Shift = Extend]*
2. Pick the curves or group that defines the profile you want to sweep.
3. Choose the first rail that your profile will be swept along.
4. Choose the second rail that your profile will be swept along.

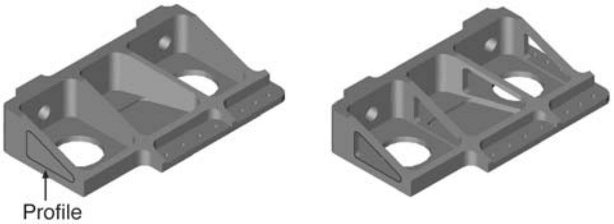
There are not status line entries for this tool.

Geometric Characteristics

A swept 2 rail profile solid is created by picking closed curves or a group from which to construct a solid. Picking the two rails (curves) along which the profile will be swept. There are only to tabs in the Edit Objects dialog box for this tool, Display and Attributes.

Cutout Feature Tool

This tool subtracts material from a solid. The cutout is formed from a 2D profile. This profile can be located on or in the solid and composed of grouped curves. You can specify a Draft Angle in the Status Line.



Using the Cutout Feature Tool

1. Create a 2D profile with a curve or polygon tool and place it on or in the solid where the cutout will begin.
2. Select the Cutout Feature tool. The Message Line reads: *Cutout Feature: Pick solid for cutout.*

3. Select the solid to be cut. The Message Line now reads, *Cutout Feature: Pick closed curves or polygon for cutting [Shift=Extend].*
4. Select the profile previously created.
5. Specify two points in the drawing area to indicate the direction and length of the cutout.

While the solid is still selected you can change the distance, draft angle and direction of the cutout in the Status Line. Type the desired values in the data fields and press ENTER (Windows) or RETURN (Macintosh).

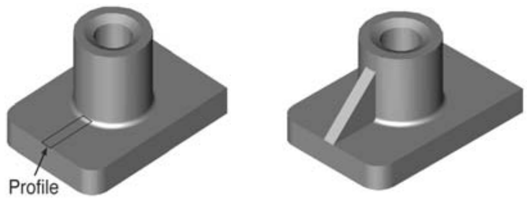
The Status Line contains the extrusion dX, dY and dZ values, the extrusion Distance and the Draft Angle.

Geometric Characteristics

A cutout is created by selecting a profile, the solid through which to cut and the distance and direction of the cut. It is made up of these characteristics according to Edit Objects: Distance, Draft Angle and Direction (DX, DY, DZ values). To modify the hole characteristics, select the hole using the Design Explorer and choose **Window>Edit Objects** or double-click on the cutout name. See Chapter 24 for more information on the Design Explorer.

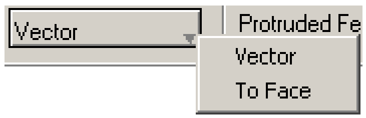
Protruded Feature Tool

The tool adds a piece to an existing solid. The protrusion is formed from a 2D profile. The profile does not need to be attached to the original solid and can be grouped curves.



Termination Types

The Message Line also contains a pull-down menu allowing you to choose where the protrusion will end. You have two options: Vector and To Face.



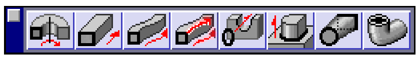
Vector Defined by two points that you choose the vector determines the length and direction of the protrusion.

To Face Extends the protrusion to a face on the solid.

Using the Protruded Feature Tool - Vector Option

1. Create a 2D profile where the protrusion will begin.

2. Select the Protruded Feature tool. The Message Line reads: Protruded Feature: Pick solid for protrusion.



3. Select the Vector Termination Type from the pull-down menu in the Message Line.
4. Select a solid. The Message Line reads: *Protruded Feature: Pick closed curves or polygon for adding [Shift=Extend].*
5. Select the profile previously created.
6. Specify two points for protrusion direction and length.

While the solid is still selected, you can change the direction and distance of the protrusion and the draft angle in the Status Line. Type the desired values in the data fields and press ENTER (Windows) or RETURN (Macintosh).

Using the Protruded Feature Tool - Face Option

1. Create a 2D profile where the protrusion will begin. (It need not be on the solid.)
2. Select the Protruded Feature tool. The Message Line reads: *Protruded Feature: Pick solid for protrusion*.
3. Select the To Face Termination Type from the pull-down menu in the Message Line.
4. Select a solid. The Message Line reads: *Protruded Feature: Pick closed curves or polygon for adding [Shift=Extend].*
5. Select the profile previously created.
6. Select the face on the solid where you want the protrusion to end.
7. Specify two points for the protrusion direction. Since you are specifying the direction, not the length, you do not need to reference the curve or the solid.

While the solid is still selected you can change the direction and distance of the protrusion and the draft angle in the Status Line. Type the desired values in the data fields and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the protrusion dX, dY and dZ values, the protrusion's Distance and Draft Angle.

Geometric Characteristics

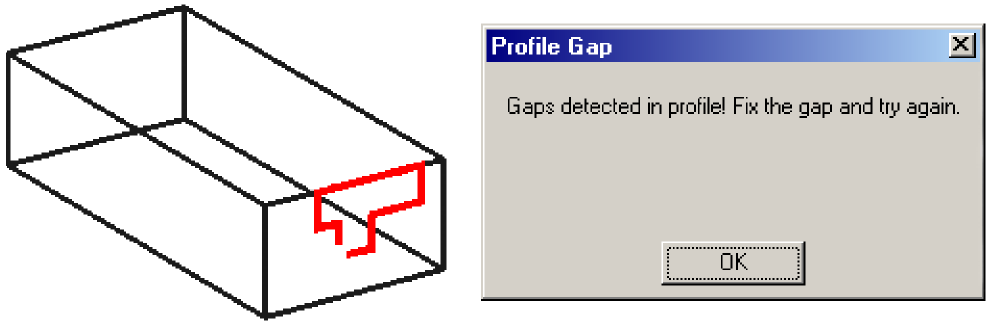
A protrusion is created by selecting a profile, the solid from which you want to extend the profile and the distance and direction of the protrusion. It is made up of these characteristics according to Edit Objects: Distance, Draft Angle and Direction (DX, DY, DZ values). It also contains the Offset Both Directions check box. To modify the protrusion characteristics, select the protrusion using the Design Explorer and choose **Window>Edit Objects** or double-click on the protrusion name. See Chapter 24 for more information on the Design Explorer.

Cutout and Protruded Curve Checks

To assist you in performing cutout and protrude operations, this Designer Elements program provides you with two curve checks: **gaps** and **planar**.

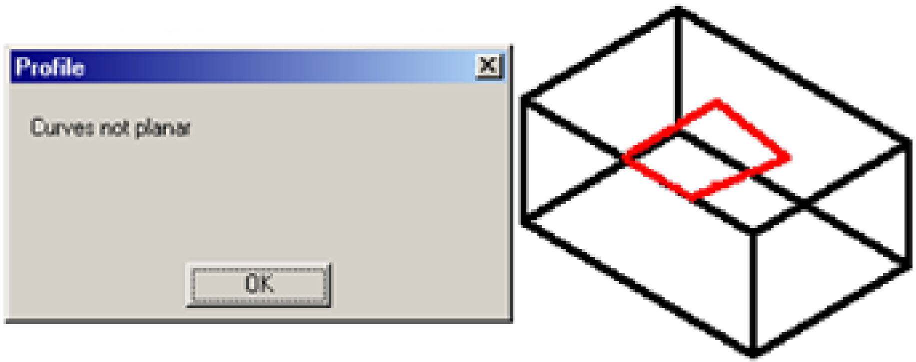
Curve Gap Check

The gap check warns you when the curves for the cutout or protrusion are not closed.



Planar Curves Check

The planar check warns you if the selected curves for the cutout or protrusion are not planar.



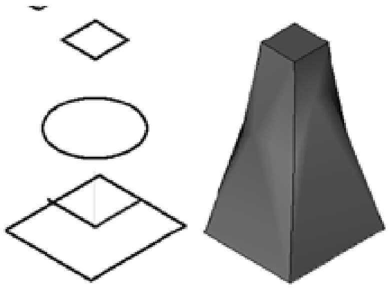
Skinned Solid Tool

The Skinned Solid tool creates a solid from a collection of closed profiles. You can create skin solids using profiles composed of individual curves (such as circles or ellipses) or grouped curves. A profile composed of individual curves does not have to be grouped for use with this tool.

There are two tools in the Message Line: Skin Curve and Skin with Guide Curves.

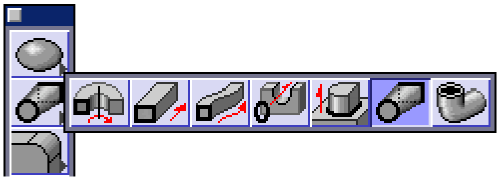
Skin Curve Solid Tool

This tool creates a skin solid between two or more curves. The solid edges are defined by the limits of the select curves. The figure here shows a skin solid created from three curves.



Using the Skin Curve Solid Tool

1. Select the Skinned Solid tool. The Message Line reads: *Skinned Solid: Pick closed curve(s) or polygons for solid from sections [Shift = Extend].*



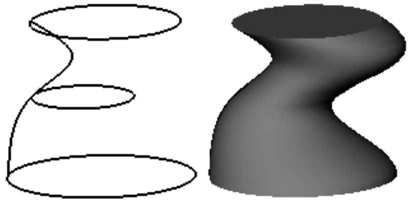
2. Select the Skin Curve Solid tool in the Message Line.
3. Hold down the SHIFT key and select the closed curves.

   The solid is created.

There are no entries in the Status Line.

Skin with Guide Curves Solid Tool

This tool creates a skinned solid between two or more curves using curves as guides to define the edges. Guide curves give you control over the skinned solid. You can use one or more guide curves to influence the solid. In the graphic the circles are skinned using the spline as a guide.



Guide Curve Rules

- Curves can go in any direction and need not be consistent with the others.
- Curves cannot loop and must be “well-behaved.”
- The curves must connect with each profile that you want to use for the solid creation.

Using the Skin with Guide Curves Solid Tool

1. Create the profile curves and curve guides.
2. Select the Skinned Solid tool. The Message Line reads: *Skinned Solid: Pick closed curve(s) or polygons for solid from sections [Shift = Extend].*
3. Select the Skin with Guide Curves Solid tool (the right tool) in the Message Line.
4. Hold down the SHIFT key and select the closed curves.

   The Message Line now reads, *Skinned Solid: Pick guide path for skinned solid [Shift = Extend].*
5. Select the guide curves.

   The solid is created.

There are no entries in the Status Line.

Geometric Characteristics

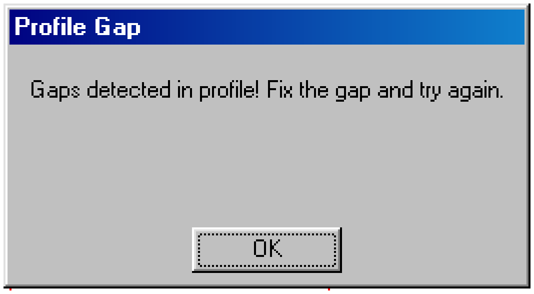
A skinned solid object is a solid created from profiles and is made up of the same characteristics as any solid object.

Solids from Profiles Curve Checks

In order to create objects using the Lathed Solid, Extrude Solid and Swept Solid tools, the profiles must be closed and planar.

Close Curves Check

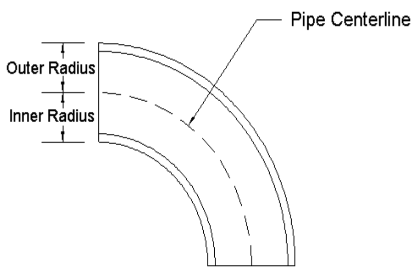
The closed curves check warns you if your profile contains gaps.

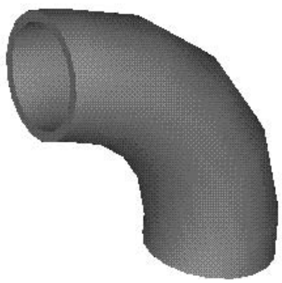


Pipe Solid Tool

The Pipe Solid tool provides a quick and easy way to create pipes and tubes.You select a curve for the pipe's center line and specify the pipe's inner and outer diameter values. You can create a pipe by selecting curves connected end to end. You cannot create more than one pipe at a time with this tool.

When you make a pipe solid, this Designer Elements program creates an intelligent link between the solid and its parent center line. When you move or modify the center line, the pipe updates automatically. A pipe with an inner diameter of zero is a solid cylinder.





This tool does not support grouped curves for creating a pipe solid.

Using the Pipe Solid Tool

Before you can use this tool you must first create a curve for the pipe.

1. Click the Pipe Solid tool. The Message Line reads: *Pipe Solid: Select curve for pipe path [Shift=Extend].*

   The Status Line contains the Outside Diameter and Inside Diameter data fields.

2. Enter values in the Status Line data fields.

   An Inside Diameter of zero creates a cylinder. Tab between the fields.

3. Select the curve. If you want to select more than one curve, hold down the SHIFT key before selecting the first curve.

   A pipe is drawn.

While the object is still selected you can edit the Outside Diameter and the Inside Diameter. Enter the new values and press ENTER (Windows) or RETURN (Macintosh) and the solid is adjusted.

Geometric Characteristics

A pipe solid is created by selecting one or more curves and entering specifying the inner and outer diameters. A pipe solid is made up of the following characteristics: Outer Diameter, Inner Diameter and Wall Thickness. This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the extruded solid and choose **Window>Edit Objects** or double-click the pipe.

General Editing Tools

Once you create an object you can select and alter it by choosing a command or using a tool to edit it. This chapter describes common editing activities and compares different methods for performing the same action. You can use the editing tools to change an object's geometry, size, location or orientation. They are collected in three subpalettes in the main tool palette.

For each tool you can enter values in the Status Line to perform the editing operation before or after the operation is complete. If you enter the values after you select the tool but before you perform the operation, your first click in the drawing area automatically registers all Status Line values. If you enter values after performing the operation and while the curve is still selected, pressing ENTER (Windows) or RETURN (Macintosh) updates the curve to reflect the new values.

Tech Note:

These editing tools cannot be performed on grouped objects, only curves.

Many of the editing tools require a two step process for their usage. The first step after choosing the tool may be to select one or more objects. The second step performs the editing operation. Once you the complete the first step you cannot select more objects to include in the same operation. To select additional objects reset the tool and start again.

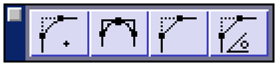
There are no unique geometric characteristics listed in the Edit Objects dialog box for an object after using these tools. For example, a fillet between two lines creates an arc. Edit Objects lists the parameters for an arc.

The following topics are explained:

- **Fillet/Chamfer** tools
- **Trim** tools

- **Curve Extras** tools

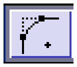
Fillet and Chamfer Tools



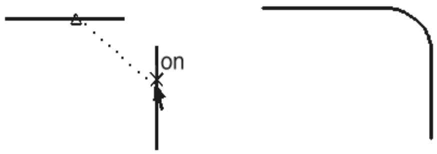
These tools on the main tool palette construct fillets and chamfers from corners formed by nonparallel lines or curves. This Designer Elements program automatically trims fillets and chamfers by default. If you do not want them trimmed hold down the CTRL (Windows) or the OPTION (Macintosh) key while you are selecting the objects to fillet or chamfer.

All curves that you want to fillet must be in the same plane.

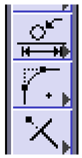
2-Entity Fillet Tool



This tool constructs an arc tangent to the two objects you specify.



Using the 2-Entity Fillet Tool

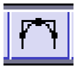


1. Select the **2-Entity Fillet** tool. The Message Line reads: *2-Entity Fillet: Pick first entity*. *[Shift=Corner, Ctrl=No trim* (Windows) or *Option=No Trim (Macintosh)]*.
2. Enter the arc radius you want in the Status Line. The default radius is .50 inch.
3. Click the first curve. The Message Line reads: *2-Entity Fillet: Pick second entity*. *[Ctrl=No trim]* (Windows) or *[Option=No Trim* (Macintosh)*]*.

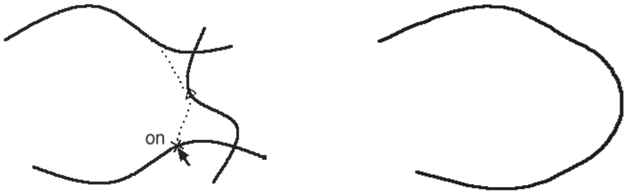
You can also hold down the SHIFT key and click once inside the corner you want to fillet. (See the Message Line at the top of your drawing area for a reference to this feature.) If you hold down the CTRL (Windows) or OPTION (Macintosh) key while you select the objects to fillet the objects are not trimmed. (See the Message Line). Fillets create the smallest possible arc between the selected geometry.

The Status Line contains the Radius of the fillet.

3-Entity Fillet Tool

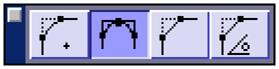


This tool constructs a fillet tangent to the three objects you choose.



Using the 3-Entity Fillet Tool

1. Select the **3-Entity Fillet** tool. The Message Line reads: *3-Entity Fillet: Pick three entities [Ctrl=No Trim* (Windows) or *Option=No Trim* (Macintosh)*]*.

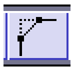


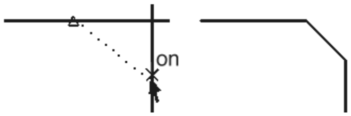
2. Click the three objects you want to fillet.

If you hold down the CTRL (Windows) or OPTION (Macintosh) key while you select the objects to fillet the objects are not trimmed (see the Message Line).

There are no Status Line entries.

2-Entity Chamfer Tool



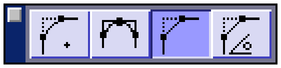


This tool creates a chamfer across a corner at the specified distance from the intersection of two lines.

Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when creating a chamfer with this tool.

Using the 2-Entity Chamfer Tool

1. Select the **2-Entity Chamfer** tool.

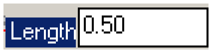


2. In the Status Line specify the chamfer's distance from the corner. (The default distance is .50 inch.)
3. Click the first curve. The Message Line reads: *2-Entity Chamfer: Pick second entity. [Ctrl=No trim* (Windows) or *Option=No Trim* (Macintosh)*]*.

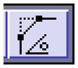
You can also hold down the SHIFT key and click once inside the corner you want to chamfer. (See the Message Line at the top of your drawing area for a reference to this feature.)

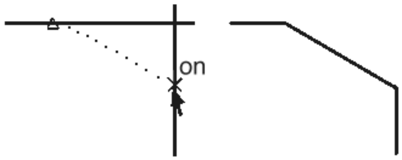
The lines are automatically trimmed or extended. If you hold down the CTRL (Windows) or the OPTION (Macintosh) key while you select the objects to chamfer, the objects are not trimmed (see the Message Line).

The Status Line allows you to specify the distance (Length) from the chamfer to the intersection of the corner lines.



Angular Chamfer Tool

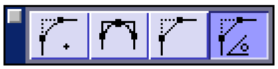




This tool creates a chamfer at the specified angle and distance from the corner. The specified angle is the angle between the chamfer and the second line of the corner. The specified length is the distance between the "*corner and the intersection of the chamfer*" and the "*second line of the corner.*" (The *second line of the corner* refers to the second line chosen when creating the chamfer.)

Using the Angular Chamfer Tool

1. Select the **Angular Chamfer** tool. The Message Line reads: *Angular Chamfer: Pick first entity [Ctrl = No trim* (Windows), *Option = No trim* (Macintosh)*]*.



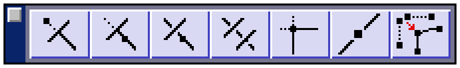
2. In the Status Line, enter the length for the intersection of the chamfer and the *second line of the corner* from the corner. The default length is .50 inch.
3. In the Status Line also enter the angle you want between the chamfer and the second side. The default Angle is 45°.
4. Click the first curve. The Message Line reads: *Angular Chamfer: Pick second entity. [Ctrl=No trim* (Windows) or *Option=No Trim* (Macintosh)*]*.

You can also hold down the SHIFT key and click once inside the corner you want to chamfer. (See the Message Line at the top of your drawing area for a reference to this feature.)

The lines are automatically trimmed or extended by default. If you hold down the CTRL (Windows) or the OPTION (Macintosh) key while you select the objects to chamfer, the objects are not trimmed. (See the Message Line).

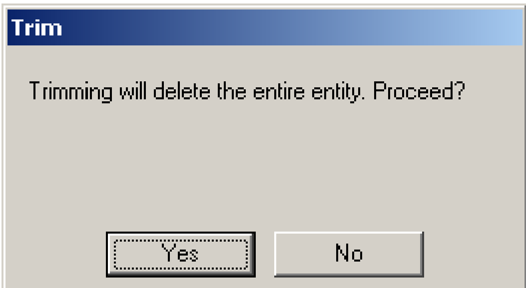
The Status Line allows you to set the distance (Length) from the intersection as well as the Angle.

Trim Tools



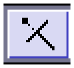
The **Trim** tools on the tool palette lengthen or shorten lines, curves and polylines. You select the limiting object(s) before or after you select the tool. If you don't select everything you need you can hold down the SHIFT key (the tool becomes a temporary **Selection** tool) and click additional objects after the **Trim** tool is selected (see the Message Line).

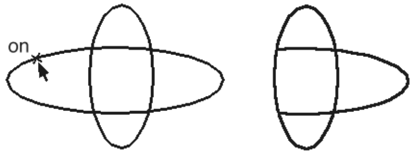
When trimming curves using one of these tools the following warning will appear if the operation will delete the entire curve.



Click Yes to proceed with the operation or No to exit the operation.

Simple Trim Tool





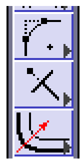
This tool shortens a curve to the specified boundary. If you hold down the CTRL (Windows) or the OPTION (Macintosh) key while using this tool it becomes the **Relimit** tool.

When you trim a Through Point B-spline or Bezier spline an editable vector spline results. You can move its control points and change its shape and slope. In some situations trimming a curve results in a spline. This spline is also an editable vector spline.

Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when trimming with this tool.

Rule: Point to what you want to remove.

Using the Simple Trim Tool



1. Select the **Simple Trim** tool. The Message Line reads: *Simple Trim: Select boundaries for simple trim [Shift = Extend]*.

   If you already selected the boundaries, proceed to step 3.

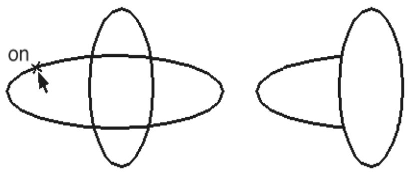
2. Select the object that limits the trim.

   If necessary, use SHIFT-click to select more boundary objects. The Message Line reads: *Simple Trim: Pick section to trim [Shift=Select Boundary, Ctrl=Relimit* (Windows) *Option=Relimit* (Macintosh)*]*.

3. Click the section of the object to be discarded.

There are no Status Line entries.

Relimit Tool



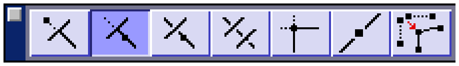
This tool lengthens or shortens a curve to the specified boundary. If you hold down the CTRL (Windows) or the OPTION (Macintosh) key while using this tool it becomes the **Trim** tool. The **Relimit** tool will not relimit arcs.

When you relimit a Through Point B-spline or Bezier spline an editable vector spline results. You can move its control points and change its shape and slope. In some situations relimiting a curve results in a spline. This spline is also an editable vector spline.

Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when relimiting geometry with this tool.

Rule: Point to what you want to keep.

Using the Relimit Tool



1. Select the **Relimit** tool. The Message Line reads: *Relimit: Select boundaries for relimit [Shift = Extend].*

   If you already selected the boundaries, proceed to step 3.

2. Select the object that limits the change.

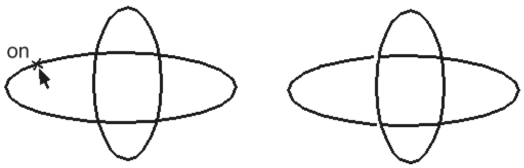
   If necessary, use SHIFT-click to select more boundary objects. The Message Line reads: *Relimit: Pick section to keep [Shift=Select Boundary, Ctrl=Trim* (Windows) *Option=Trim* (Macintosh)*].*

3. Click the section of the object that you want to retain.

There are no Status Line entries.

Segment Tool

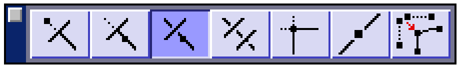
This tool divides a curve at intersections with other lines or curves.



When you segment a Through Point B-spline or Bezier spline an editable vector spline results. You can move its control points and change its shape and slope. In some situations segmenting a curve results in a spline. This spline is also an editable vector spline.

Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when segmenting curves with this tool.

Using the Segment Tool



1. Select the **Segment** tool. The Message Line reads: *Segment: Select boundaries for segment [Shift = Extend]*.

   If you already selected the boundaries, proceed to step 3.

2. Select the objects that limits the segmentation.

   If necessary, use SHIFT-click to select more boundary objects. The Message Line reads: *Segment: Pick Entity [Shift=Select Boundary, Ctrl=Current Pen* (Windows) or *Option=Current Pen* (Macintosh)*]*.

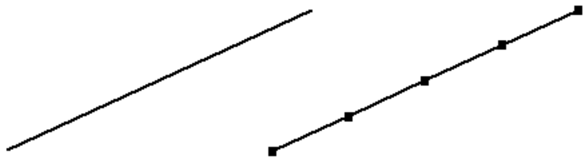
3. Click the object to be segmented.

The selected object is segmented at the boundary objects. Even though you cannot see the segmentation on the screen, you can select parts of the segmented line by choosing it with the **Selection** tool. In order to see the segmentation on the screen, first select the boundaries and the object you want to segment, then choose ***Edit>Show Points***. The endpoints of the segmented parts are displayed.

If you hold down the CTRL (Windows) or OPTION (Macintosh) key while selecting the line to be segmented the new segment appears in the characteristics of the current pen style (see the Message Line).

There are no Status Line entries.

Divide Tool

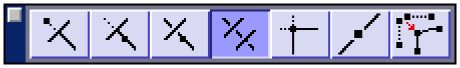


This tool subdivides a curve into a specified number of equal-length segments. The left graphic above shows a single line. The right graphic shows the line divided into four segments with the control points displayed.

When you divide a Through Point B-spline or Bezier spline an editable vector spline results. You can move its control points and change its shape and slope. In some situations dividing a curve results in a spline. This spline is also an editable vector spline.

Using the Divide Tool

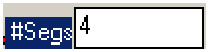
1. Select the **Divide** tool. The Message Line reads: *Divide: Pick curve to divide [Shift=Extend].*



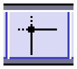
2. Set the #Segs (number of segments) value in the Status Line.
3. Select the object to divide.

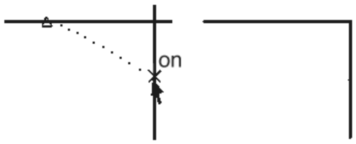
   The object divides. The individual segments is not indicated unless you turn on their control points.

The Status Line allows you to set the #Segs in the curve.



Corner Trim Tool



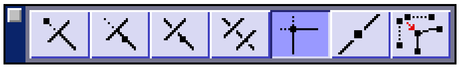


This tool creates a corner from two objects that you specify. Lines are extended or shortened to create the corner.

When you conduct a corner trim on a Through Point B-spline or Bezier spline an editable vector spline results. You can move its control points and change its shape and slope. In some situations trimming a curve results in a spline. This spline is also an editable vector spline.

Using the Corner Trim Tool

1. Select the **Corner Trim** tool. The Message Line reads: *Corner Trim: Pick portion of first entity to retain. [Shift=Corner, Ctrl=No trim* (Windows) or *Option=No trim* (Macintosh)*].*

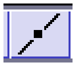


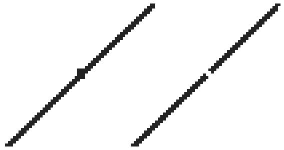
2. Select the first curve. The Message Line reads: *Corner Trim: Pick portion of second entity to retain. [Ctrl=No trim* (Windows) or *Option=No trim* (Macintosh)*].*
3. Select the second object.

As an alternative to picking two entities, you can press and hold the SHIFT key and click inside the about-to-be-created corner.

There are no Status Line entries.

Divide at Location Tool



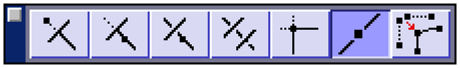


This tool breaks one curve into two at a point that you choose.

When you divide a Through Point B-spline or Bezier spline using this tool an editable vector spline results. You can move its control points and change its shape and slope. In some situations dividing a curve results in a spline. This spline is also an editable vector spline. You can also divide 360° circles and ellipses.

Using the Divide at Location Tool

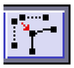
1. Select the **Divide at Location** tool. The Message Line reads: *Divide at Location: Pick curve to divide at location: [Shift=Extend].*



2. Select the curve. The Message Line reads: *Divide at Location: Pick divide point.*
3. Choose the point on the curve where you want it divided.

There are no Status Line entries.

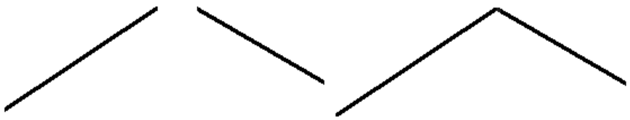
Connect Curve Tool



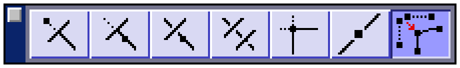
This tool allows you to connect curves. This Designer Elements program provides you with two options, *Move Curve Point* and *Join*. The *Join* option also includes the ability to set a tolerance. (The tolerance field does not apply to the default *Move Curve Point* option.)

Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when connecting curves with this tool.

Using the Connect Curve Tool - Move Curve Point Default



Move Curve Point is the default option when using the tool. Using this option moves the endpoint from the second selected curve to the endpoint of the first selected curve. This option is useful for creating closed profiles that you can extrude, revolve, or sweep to create solid models.



1. Select the **Connect Curve** tool. The Message Line reads: *Connect Curve: Select two curves to connect end points [Ctrl=Join* (Windows) or *Option=Join* (Macintosh), Shift=Extend*]*.
2. Select the first curve near the endpoint where you want to connect the curve.
3. Select the second curve near the endpoint that you want to move.

   The second curve's endpoint moves to the endpoint of the first curve. The other endpoint of the second curve remains fixed.

The Status Line contains a Tolerance data field. This field only applies to the Join option.

Using the Connect Curve Tool - Join Option

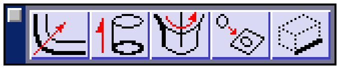
The *Join Curve* option replaces the two curves you select with a new spline curve. The selected lines must share a tangent. This tool will also join two polylines within a specified tolerance distance. This is helpful when you've converted curves into polylines using the *Change Object* command and later want to join them. Select the *Join Curve* option as you pick the two curves.

1. Select the **Connect Curve** tool. The Message Line reads: *Connect Curve: Select two curves to connect end points [Ctrl=Join* (Windows) or *Option=Join* (Macintosh), *Shift=Extend]*.
2. If desired, specify a tolerance distance in the Status Line to which the resulting curves will fit.
3. Hold down the CTRL key (Windows) or the OPTION key (Macintosh) and select the two curves.

   Any discontinuities between the selected lines are eliminated.

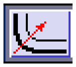
The Status Line contains the Tolerance data field.

Curve Utility Tools

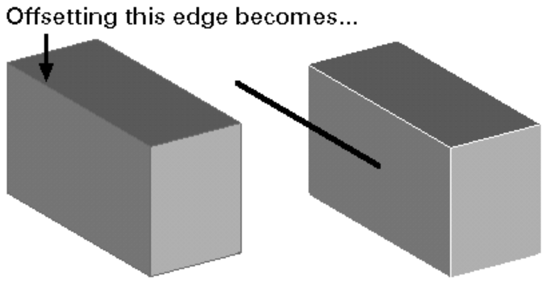


The **Curve Utility** tool palette contains tools for offsetting and exploding curves.

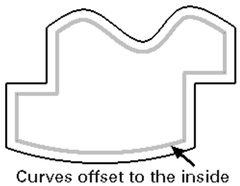
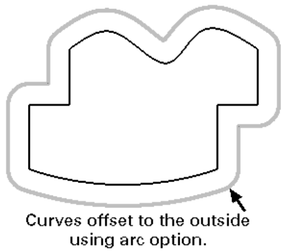
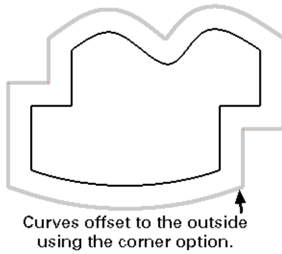
Offset Curve Tool



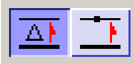
The **Offset Curve** tool offsets curves by a location or a distance that you specify. The curves can be part of a wireframe object, surface edge (**Offset to Value** tool only) or solid edge. Groups cannot be offset. The graphic here is an example of an offset solid ege.



You can offset one or more curves and supports the following options.

Automatic trimming for curves offset inside-curves.	*Curves offset outside of curves with the arc corners for the offset.*	*Curves offset outside of curves with extended corners for the offset.*
		

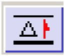
The **Offset Curve** tool includes a subpalette with two tools, **Offset to Value** and **Offset by Point**.



When you offset a Through Point B-spline or Bezier spline an editable vector spline results. You can move its control points and change its shape and slope. In some situations offsetting a curve results in a spline. This spline is also an editable vector spline.

Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when using this tool.

Offset to Value Tool



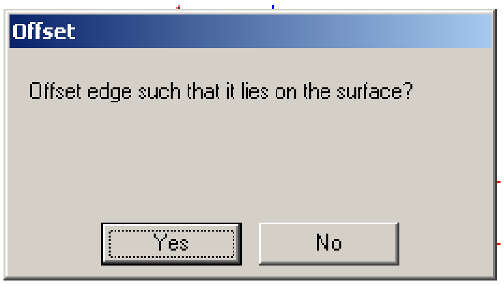
This tool creates an offset curve at a specified distance from the original curve, surface edge or solid edge. When offsetting surface edges you also have the option of placing the offset on the surface. You can offset one curve or edge or connected curves or edges.

Using the Offset to Value Tool - One Curve, Surface Edge or Solid Edge

1. Select the **Offset Curve** tool.
2. Select the **Offset to Value** tool in the Message Line. The Message Line reads: *Offset Curve: Pick curves to offset distance [Ctrl=Extend Corner* (Windows) or *Option=Extend Corner* (Macintosh), *Shift=Extend].*
3. Enter an offset distance in the Offset data field of the Status Line.
4. Select the curve or edge to offset. The Message Line reads: *Offset Curve: Enter offset direction*.

   If you select a surface edge, the Offset dialog box appears asking if you want the offset to lie on the surface.

   Click Yes to place the offset on surface. Click No and proceed with the next step.



5. Pick a point on either side of the curve or edge to specify the offset

direction. The offset direction must be parallel to the curve or edge. Our Designer Elements program offsets the curve by the distance set in the data field.

While the offset curve is selected you may enter a new offset distance in the Status Line. Press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the Offset distance.

Using the Offset to Value Tool - Connect Curves or Solid Edges

For connected curves or edges you have multiple offset options: extending exterior offsets so they connect with an arc intersection, extending exterior offsets with a corner intersection or trimming intersecting offsets for interior offsets.

1. Select the **Offset Curve** tool.
2. Select the **Offset to Value** tool in the Message Line. The Message Line reads: *Offset Curve: Pick curves to offset distance [Ctrl=Extend Corner* (Windows) or *Option=Extend Corner* (Macintosh), *Shift=Extend].*
3. Enter an offset distance in the Offset data field of the Status Line.
4. Select the curves or edges to offset. The Message Line reads: *Offset Curve: Enter offset direction*.
5. Select the type of offset you desire.

   Exterior Offset - Arc intersection: Pick a point on the outside of the curves or edges.

   Exterior Offset - Corner intersection: Hold down the CTRL key (Windows) or the OPTION key (Macintosh) and pick a point on the outside of the curves or edges.

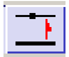
   Interior Offset: Pick a point on the inside of the curves or edges.

   Our Designer Elements program offsets the curves or edges by the distance set in the Status Line data field.

While the offset curves are selected you may enter a new offset distance in the Status Line. Press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the Offset distance.

Offset by Point Tool



This tool creates an offset at the point specified. You can offset one curve or solid edge or connected curves or solid edges.

Using the Offset by Point Tool - One Curve or Solid Edge

1. Select the **Offset Curve** tool.
2. Select the **Offset by Point** tool in the Message Line. The Message Line reads: *Offset Curve: Pick curves to point [Ctrl=Extend Arc* (Windows) or *Option=Extend Arc* (Macintosh), *Shift=Extend].*
3. Select the curve or edge to offset. The Message Line reads: *Offset Curve: Enter offset point*.
4. Pick the desired point for the offset. Our Designer Elements program offsets the curve to that point. The offset value is displayed in the Offset data field in the Status Line.

While the offset curve is selected you may enter a new offset distance in the Status Line. Press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the offset distance.

Using the Offset by Point Tool - Connected Curves or Solid Edges

1. Select the **Offset Curve** tool.
2. Select the **Offset by Point** tool in the Message Line. The Message Line reads: *Offset Curve: Pick curves to point [Ctrl=Extend Arc* (Windows) or *Option=Extend Arc* (Macintosh), *Shift=Extend].*
3. Select the curves to offset. The Message Line reads: *Offset Curve: Enter offset point*.
4. Select the type of offset you desire.

   Exterior Offset - Arc intersection: Pick a point on the outside of the curves.

   Exterior Offset - Corner intersection: Hold down the CTRL key (Windows) or the OPTION key (Macintosh) and pick a point on the outside of the curves.

   Interior Offset: Pick a point on the inside of the curves.

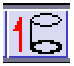
   Our Designer Elements program offsets the curves at that point. The offset value is displayed in the Offset data field in the Status Line.

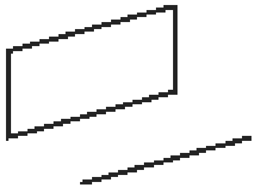
5. Pick the desired point for the offset. Our Designer Elements program offsets the curve to that point. The offset value is displayed in the Offset data field in the Status Line.

While the offset curves are selected you may enter a new offset distance in the Status Line. Press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the offset distance.

Extrude Curve Tool





This tool copies a curve at a distance and in the direction that you specify. It connects its endpoints by two lines back to the original curve. In the graphic to the right the lower curve is extruded to form the polygon.

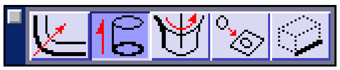
Using the Extrude Curve Tool

Tip:

To extrude an object you must click to define the extrusion rather than drag.

To extrude a curve at an angle not normal to the curve, change the view. Example: Create a rectangle in the top plane. Change the plane to the front plane and place the extrude points to define an angle and length.

1. Click the **Extrude Curve** tool. The Message Line reads: *Extrude Curve: Pick curves to extrude [Shift=Extend].*

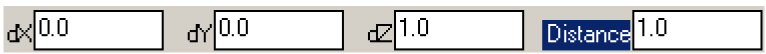


2. Select the curve(s) to extrude. The Message Line reads: *Extrude Curve: Specify two points for extrusion direction*.

3. Click two points to indicate where the new extruded curve will be placed. The direction of the extrusion is specified by the order in which you pick the points. This Designer Elements program applies the distance and direction you specify.

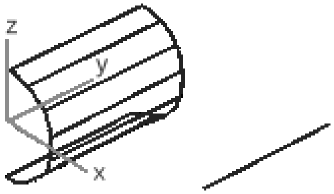
While the object is selected you can change the extrusion distance in the Status Line. Press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the extrusion's dX, dY and dZ values and the Distance.



Revolve Curve Tool

This tool revolves a curve around an axis.



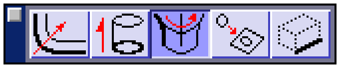
Tech Note:

You cannot revolve a *Smart Polygon*. It must first be converted into single lines using the Change Object Type command. See Chapter 12 for more information on polygon types.

To define the axis of revolution you must click the points rather than drag.

Using the Revolve Curve Tool

1. Select the **Revolve Curve** tool. The Message Line reads: *Revolve Curve: Pick curves to revolve [Shift=Extend].*

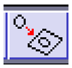


2. Select the curve(s) to revolve. The Message Line reads: *Revolve Curve: Specify revolution origin*.
3. Click a point about which to revolve the curve. The Message Line reads: *Revolve Curve: Specify revolution axis.*
4. Click two points to indicate the axis of revolution. The direction of the revolution is specified by the order you pick the points. This Designer Elements program rotates the curve about the axis by the number of steps and through the angle set in the data fields on the status line.

While the object is still selected you can change the number of steps and the angle of revolution in the Status Line. Press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the number of steps and the angle of revolution.

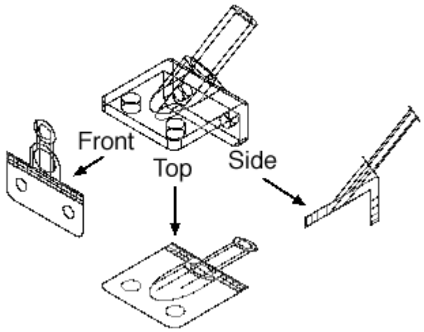
Project Curve Tool



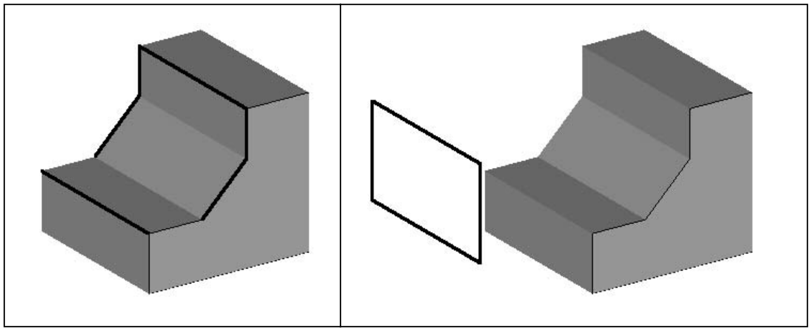
The **Project Curve** tool projects 3D curves, including solid edges onto a 2D plane. Arcs, circles and ellipses maintain their precise shape when you project them normal to their definition. If you project them otherwise a spline is used with a default curve tolerance of 0.001 inches. This spline is an editable vector spline.

When you project a Through Point B-spline or Bezier spline an editable vector spline results. You can move its control points and change its shape and slope. In some situations projecting a curve results in a spline.

The select curves are created on the plane and replace the original curves the curves unless the CTRL key (Windows) or OPTION key (Macintosh) is held down. The graphic here shows a model projected to XY, XZ and YZ planes.

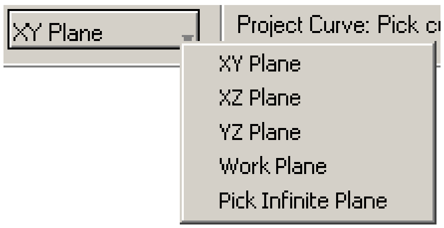


The left graphic below shows a model with four solid edges selected (displayed in black). The right graphic shows the edges projected to the XZ Plane.

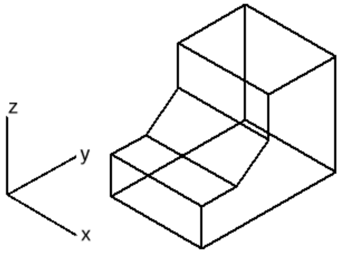


Plane Options

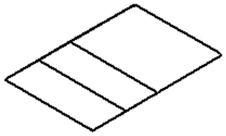
The pull-down menu in the Message Line provides five options for choosing the plane: *XY Plane*, *XZ Plane*, *YZ Plane*, *Work Plane* and *Pick Infinite Plane*.



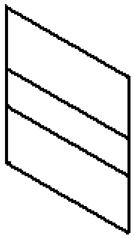
The following figures, displayed in the Isometric view, explain the projection options. All graphics used for the individual planes are also displayed in the Isometric view.



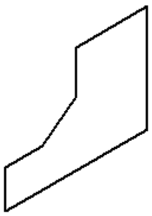
XY Plane Choosing this option projects the selected curves into the XY plane (Top) where X=0 and Y=0.



XZ Plane Choosing this option projects the selected curves into the XZ plane (Front) where X=0 and Z=0.

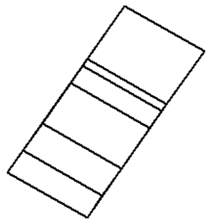


YZ Plane Choosing this option projects the selected curves into the YZ plane (Side) where Y=0 and Z=0.



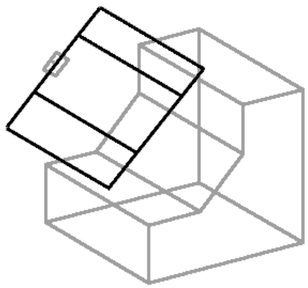
Work Plane

Choosing this option projects the selected curves into the work plane. For this example, the work plane was defined using the sloped face of the object.



Pick Infinite Plane

Choosing this option project the selected curves to the specified infinite plane.



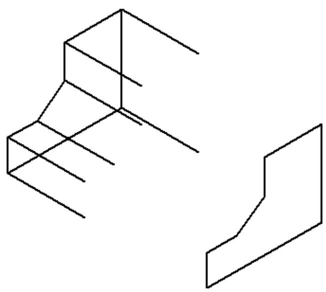
Use the Plane commands in the Plane menu to define your own plane. You must set the work plane to the desired plane before using this option.

Projection Options

When you project curves this Designer Elements program takes those curves and creates new geometry on the specified plane. You have two options, Default and Copy (see the Message Line).

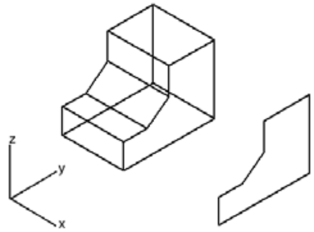
Project Curves - Default Option

This option takes the selected geometry. See the graphic here. Notice that the selected geometry no longer appears on the original part.



Project Curves - Copy Option

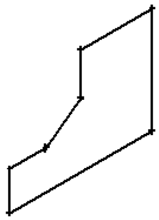
This option copies the selected geometry. To choose this option hold down the CTRL key (Windows) or OPTION (Macintosh). See the graphic here.



Projecting & Multiple Curve Selection

After you select the tool you can select multiple lines for projection by holding down the SHIFT key and selecting the desired lines, as noted in the Message Line.

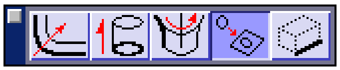
You can also select multiple lines or an entire object by dragging a selection fence around the entire object. When you project the object all curves that are normal to the projection plane are recreated as points in the new curve. The graphic here shows a YZ projection. The cross hairs represent the points.



Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when projecting curves with this tool.

Using the Project Curve Tool

1. Select the **Project Curve** tool. The Message Line reads: *Project Curve: Pick curves to project [Ctrl=Copy* (Windows) or *Option=Copy* (Macintosh), *Shift=Extend]*.



2. From the pull-down menu in the Status Line choose the projection plane.
3. Select the curves to project. Hold down the SHIFT key or drag a selection fence to select multiple lines. The Message Line now reads, *Project Curve: Specify point in projection plane [Ctrl=Copy* (Windows) or *Option=Copy* (Macintosh)*]*.
4. Pick the point on the projection plane where the curves should be projected. The selected lines are replaced with lines projected on the selected plane.

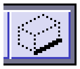
   To project a copy of the lines hold down the CTRL key (Windows) or OPTION key (Macintosh) before selecting the point.

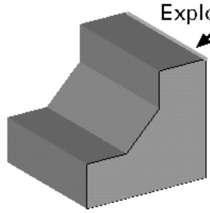
Tech Note:

If you hit ENTER (Windows) or RETURN (Macintosh) instead of specifying a point, the projection is placed at the origin.

There are no Status Line entries.

Explode Edge Tool

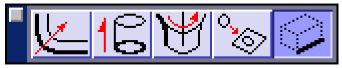




This tool enables you to create a curve from a surface or solid edge. For surfaces and solids, you can also explode Iso lines when displayed on your geometry. This ability is useful when constructing geometry off an object.

When you explode a solid edge, solid Iso line, surface edge or surface Iso line the new curve is placed on the work layer and assumes its color characteristic. If the layer has no specific color characteristic, the curve displays in the color selected in the Pen menu. If the exploded edge or Iso line results in a spline, a vector spline results. You can move its control points and change its shape and slope.

Using the Explode Edge Tool

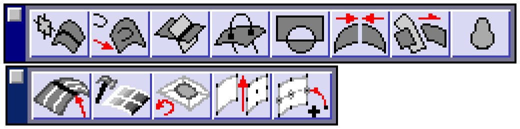


1. Select the **Explode Edge** tool. The Message Line reads: *Explode Edge: Pick edges to explode [Shift=Extend]*.
2. Select the surface or solid edge or Iso line from which you want to create a curve.

   The curve is created at the same location as the selected edge.

There are no entries in the Status Line.

Surface Editing Tools and Features



Our Designer Elements program provides several editing tools in the **Surface Editing** tool palettes. The tools allow you to perform such operations as intersecting existing surfaces and solids with other entities, projecting curves onto surfaces, adding to or subtracting from and intersecting 2D planar surfaces as well as joining or trimming surfaces and matching surfaces. Our Designer Elements program also includes features for reshaping surfaces through control vertex modification.

The following topics are covered in this chapter:

- **Solid Editing** tools
- Surface Control Vertex Modification
- Surface Evaluation

Solid Editing Tools

Many of the editing tools require a two step process for their usage. The first step after choosing the tool may be to select one or more objects. The second step performs the editing operation. Once you the complete the first step you cannot select more objects to include in the same operation. To select additional objects reset the tool and start again.

Note: These tools do not support the use of grouped objects.

The geometric characteristics listed in the Edit Objects dialog box after editing an object using these tools are the same as the object's standard characteristics. For example, if a curve/surface intersection is performed, the Edit Objects dialog would display characteristics of the resulting curve.

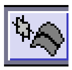
Selecting Objects within the Editing Tools

With these editing tools you are asked to select the objects for performing an operation. When you select the tool and move the cursor into the drawing area it becomes a hollow selection arrow (shown here).

When you finish selecting the pointer becomes a general tool pointer (shown here.)

This hollow selection arrow must not be confused with the **Selection** tool.

Plane/Surface Intersection Tool



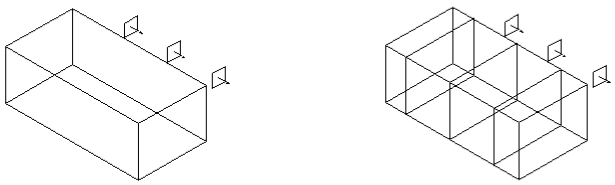
The **Plane/Surface Intersection** tool calculates the intersection of an infinite plane with a curve, surface or solid. The resulting cut is automatically associative with the plane and the object so that if you modify either entity, the cut will update (**Cobalt™ and Xenon™ only**).

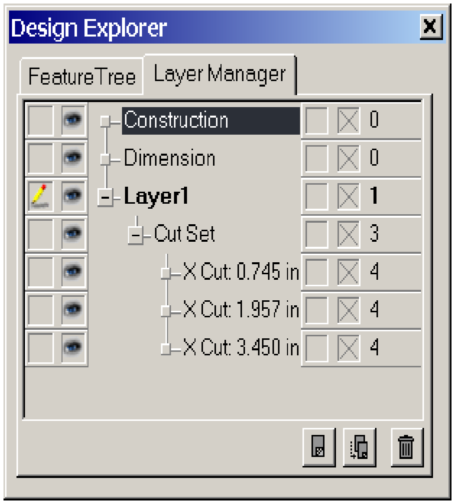
Auto Layers Option

Tip:

When the *Auto Layers* option is used with the *Increment Layer* command (ALT+0 - Windows, z +0 - Macintosh) and Decrement Layer (ALT+9-Windows, z +9-Macintosh), to change layers, you have a convenient way of examining every 2D cut of a 3D model.

This tool provides an *Auto Layer* option in the Message Line. When this option is enabled, our Designer Elements program puts each section cut on a separate layer and names the layer according to the location of the cut. For example, the graphics here show a solid block with three intersecting planes.



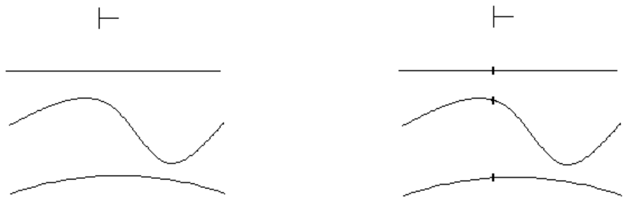


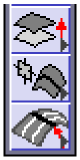
With the *Auto Layers* option enabled each section is placed on a separate layer. The Layer Manager graphic below shows the new layers.

Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when using this tool.

Intersecting a Plane with a Curve

When you intersect a plane with a curve a point intersection results. The left graphic here shows a plane and three curves. The right graphic displays the intersection points for the plane and the curves.





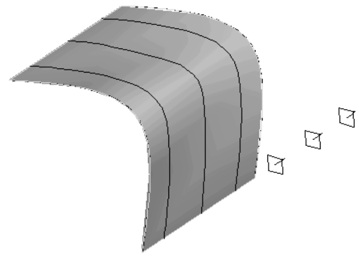
1. Select the tool in the tool palette. The Message Line reads: *Plane/Surface Intersection: Pick the planes to slice objects with [Shift+Extend].*
2. Click the plane you want to use to intersect the curve. Hold down the SHIFT key to pick more than one plane. The Message Line reads: *Plane/Surface Intersection: Pick objects to intersect with plane [Shift=Extend].*
3. Click the curves that you want to intersect with the plane.

   The intersection point displays. The point is associative to the plane and curve. Modifying either the plane or the curve automatically updates the location of the point.

There are no entries in the Status Line.

Intersecting a Plane with a Surface

When you intersect a plane with a surface a curve intersection results. If the curve is a spline, it is a vector spline capable of being edited.



1. Select the tool in the tool palette. The Message Line reads: *Plane/Surface Intersection: Pick planes to slice objects with [Shift=Extend].*
2. Click the plane you want to use to intersect the surface. Hold down the SHIFT key to pick more than one plane. The Message Line reads: *Plane/Surface Intersection: Pick objects to intersect with plane [Shift=Extend].*
3. Click the surface that you want to intersect with the plane.

   The intersection curve displays. The curve is associative to the plane and surface. Modifying either the plane or the surface automatically updates the location of the curve (**Cobalt™ and Xenon™ only**).

There are no entries in the Status Line.

Intersecting a Plane with a Solid

When you intersect a plane with a solid a curve intersection results. If the curve is a spline, it is a vector spline capable of being edited.

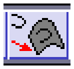
1. Click the tool in the tool palette. The Message Line reads: *Plane/Surface Intersection: Pick planes to slice objects with [Shift=Extend].*

2. Click the plane you want to use to intersect the solid. Hold down the SHIFT key to pick more than one plane. The Message Line reads: *Plane/Surface Intersection: Pick objects to intersect with plane [Shift=Extend].*
3. Click the solid that you want to intersect with the plane.

   The intersection curve displays. The curve is associative to the plane and solid. Modifying either the plane or the solid automatically updates the location of the curve (**Cobalt™ and Xenon™ only**).

There are no entries in the Status Line.

Curve/Surface Projection Tool



The **Curve/Surface Projection** tool projects a curve along a direction vector onto a surface or a solid. You specify the direction vector by clicking two points. The new curve is associative to the object. When you modify the object the projected curve will automatically update (**Cobalt™ and Xenon™ only**).

Projecting a Curve to a Surface

With this tool you can project a curve onto a surface. If the curve is a spline, it is a vector spline capable of being edited.

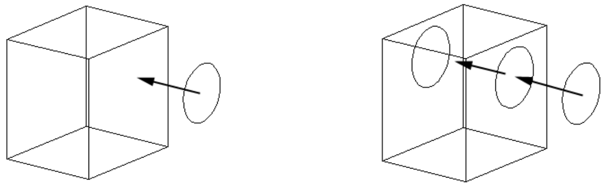
1. Select the tool in the tool palette. The Message Line reads: *Curve/Surface Projection: Pick curves to project onto surface [Shift=Extend].*
2. Click the curve to project onto the surface (it does not highlight). Hold down the SHIFT key to pick more than one curve. The Message Line reads: *Curve/Surface Projection: Pick ONE surface or solid for curve projections [Shift=Extend].*
3. Click the surface or solid onto which you want to project the curve.
4. Enter 2 points for the projection vector as directed by the Message Line.

   The curve projects onto the surface.

There are no entries in the Status Line.

Projecting a Curve to a Solid

With this tool you can project a curve onto a solid. If the curve is a spline, it is a vector spline capable of being edited.



1. Click the tool in the tool palette. The Message Line reads: *Curve/Surface Projection: Pick curves to project onto surface [Shift=Extend].*
2. Click the curve to project onto the solid (it does not highlight). Hold down the SHIFT key to pick more than one curve. The Message Line reads: *Curve/Surface Projection: Pick ONE surface or solid for curve projections [Shift=Extend].*
3. Click the solid onto which you want to project the curve.
4. Enter 2 points for the projection vector as directed by the Message Line.

   The curve projects onto the solid.

There are no entries in the Status Line.

Surface/Surface Intersection Tool

The **Surface/Surface Intersection** tool calculates the intersection of two existing surfaces or a surface and a solid. Select the two objects and the tool creates one or more curves. The curves are associative with both objects (**Cobalt™ and Xenon™ only**). When you change either object the intersection curve will update. To remove the associative link, use ***Edit>Remove Links***.

Intersecting a Surface with a Surface

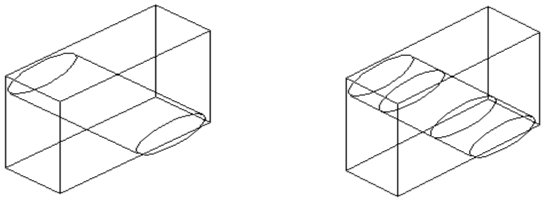
Use this tool to intersect two surfaces producing a curve intersection. If the curve is a spline, it is a vector spline capable of being edited.

1. Select the tool in the tool palette. The Message Line reads: *Surface/Surface Intersection: Pick two surfaces or solids for intersection*.
2. Click on the two surfaces. An intersection curve displays.

There are no entries in the Status Line.

Intersecting a Surface with a Solid

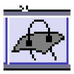
Use this tool to intersect a surface with a solid.



1. Click the tool in the tool palette. The Message Line reads: *Surface/Surface Intersection: Pick two surfaces or solids for intersection [Shift=Extend]*.
2. Click the two surfaces. An intersection curve displays.

There are no entries in the Status Line.

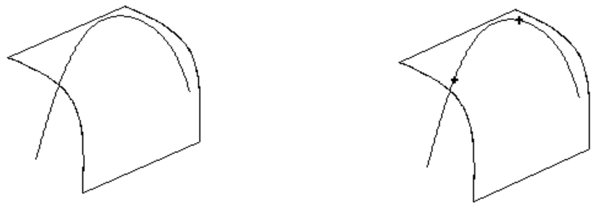
Curve/Surface Intersection Tool



The **Curve/Surface Intersection** tool calculates the intersection of a curve and a surface or solid. The tool creates a collection of point objects. The points are associative to both the curve and object (**Cobalt™ and Xenon™ only**).

Intersecting a Curve with a Surface

You can intersect a curve with a surface with this tool.

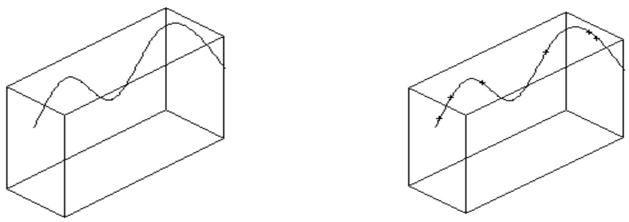


1. Select the tool in the tool palette. The Message Line reads: *Curve/Surface Intersection: Select curves to intersect surface/solids [Shift=Extend].*
2. Click the curve to intersect the surface. Hold down the SHIFT key to pick more than one curve. The Message Line reads: *Curve/Surface Intersection: Select surfaces or solids to intersect with curve [Shift=Extend].*
3. Click the surface to intersect with the curve. The intersection points display.

There are no entries in the Status Line.

Intersecting a Curve with a Solid

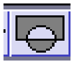
You can intersect a curve with a solid with this tool.



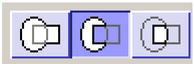
1. Select the tool in the tool palette. The Message Line reads: *Curve/Surface Intersection: Select curves to intersect surface/solids [Shift=Extend].*
2. Click the curve to intersect the solid. Hold down the SHIFT key to pick more than one curve. The Message Line reads: *Curve/Surface Intersection: Select surfaces or solids to intersect with curve [Shift=Extend].*
3. Click the solid to intersect with the curve. The intersection points display.

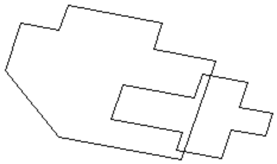
There are no entries in the Status Line.

Boolean Surface Tools



Our Designer Elements program supports 2D boolean operations on planar surfaces created with the cover surface command. With the **Boolean Surface** tool you can add surfaces together, subtract surfaces from one another, and find the intersection of two or more surfaces. You can use the new curves created from boolean operations for 2D property analysis or as profiles for solid extrusions, sweeps and lathe operations.

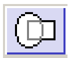


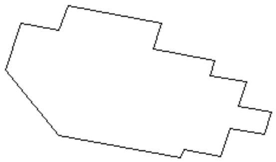


You have three tools for performing boolean operations: **Add Boolean Surface**, **Subtract Boolean Surface** and **Intersect Boolean Surface**.

For the purposes of explaining these tools the objects to the right are used.

Add Boolean Surface Tool





With this tool you add two objects to create a third object.

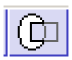
1. Select the **Boolean Surface** tool.
2. Select the **Add Boolean Surface** tool in the Message Line. The Message Line reads: *Boolean Surface: Pick planar surface to add to [Shift=Extend].*
3. Select the surface to which you will add another. The Message Line reads: *Boolean Surface: Pick planar surface to add [Shift=Extend].*
4. Select the surface to be added.

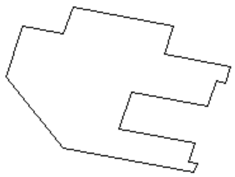
There are no entries in the Status Line.

Tip:

If you want to get volume, density and other properties of geometry in your drawing, you can combine them with this tool whether or not they are connected.

Subtract Boolean Surface Tool





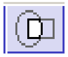
With this tool you subtract one surface from another.

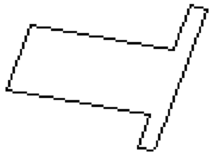
1. Select the **Boolean Surface** tool.

2. Select the **Subtract Boolean Surface** tool in the Message Line. The Message Line reads: *Boolean Surface: Pick planar surface to subtract from [Shift=Extend]*.
3. Select the surface from which you will subtract another. The Message Line reads: *Boolean Surface: Pick planar surface to subtract [Shift=Extend]*.
4. Select the surface to be subtracted. One surface is subtracted from the other.

There are no entries in the Status Line.

Intersect Boolean Surfaces Tool



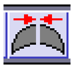


With this tool you retain the intersecting portion of two surfaces.

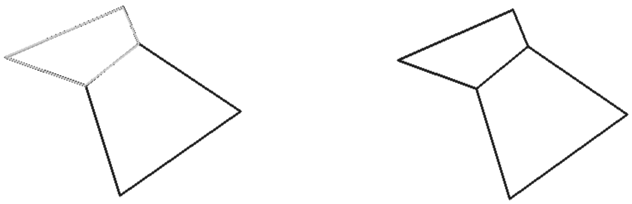
1. Select the **Boolean Surface** tool.
2. Select the **Intersect Boolean Surface** tool in the Message Line. The Message Line reads: *Boolean Surface: Pick planar surface to intersect with [Shift=Extend]*.
3. Select the surface with which you will intersect another. The Message Line reads: *Boolean Surface: Pick planar surface to intersect [Shift=Extend]*.
4. Select the surface to be intersected. The intersecting portion of the surfaces is displayed.

There are no entries in the Status Line.

Connect Surface Tool



The **Connect Surface** tool merges two surfaces into one. The resulting surface retains the original surfaces mathematical definitions, but a selection operation treats both surfaces as one. Unlike the **Add Boolean Surface** tool the **Connect Surface** tool can be used with non-planar surfaces. The left graphic below displays two separate but connecting surfaces. The graphic on the right displays one surface that retains the characteristics of the individual surfaces.



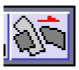
Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when using this tool.

Using the Connect Surface Tool

1. Click the **Connect Surface** tool in the tool palette. The Message Line reads: *Connect Surface: Pick two surfaces to connect [Shift=Extend].*
2. Click the two surfaces that you want to connect. The surfaces become one surface but retain their individual mathematical definitions.

There are no entries in the Status Line.

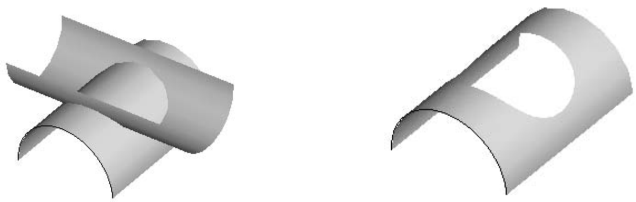
Split Surface Tool



Use the **Split Surface** tool to cut away portions of a surface. You can trim a surface to another surface, curve or solid. Unlike the **Subtract Boolean Surface** tool the **Split Surface** tool can be used with non-planar surfaces.

*Using the Tool to **Split** a Surface with a Surface*

You can use this tool to trim a surface to a surface.

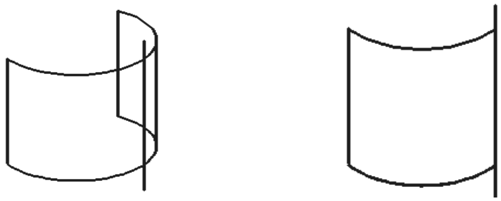


1. Select the **Split Surface** tool in the tool palette. The Message Line reads: *Split Surface: Pick surface to split [Shift=Extend].*
2. Select the surface that you want to split. The Message Line reads: *Split Surface: Pick curves, surface, or solid to split with [Shift=Extend].*
3. Select the entity that defines the trim area. Our Designer Elements program splits the surface at the intersection of the two entities. Your objects won't appear differently on your screen. Delete any portions that you don't need.

There are no entries in the Status Line.

Using the Tool to Split *the Surface to a Curve*

You can use this tool to trim a surface to a curve.



Tech Note:

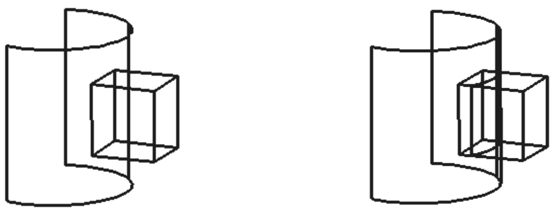
The Split Surface Tool will not work with grouped surfaces.

1. Select the **Split Surface** tool in the tool palette. The Message Line reads: *Split Surface: Pick surface to split [Shift=Extend].*
2. Select the surface that you want to trim. The Message Line reads: *Split Surface: Pick curves, surface, or solid to split with [Shift=Extend].*
3. Select the entity that defines the trim area. Our Designer Elements program splits the surface at the intersection of the surface and curve. Your objects won't appear differently on your screen. Delete any portions that you don't need.

There are no entries in the Status Line.

Using the Tool to Split *a Surface with a Solid*

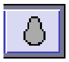
You can use this tool to split a surface to a solid.



1. Select the **Split Surface** tool in the tool palette. The Message Line reads: *Split Surface: Pick surface to split [Shift=Extend].*
2. Select the surface that you want to split. The Message Line reads: *Split Surface: Pick curves, surface, or solid to split with [Shift=Extend].*
3. Select the entity that defines the split area. Our Designer Elements program splits the surface at the intersection of the surface and the solid. Your objects won't appear differently on your screen. Delete any portions that you don't need.

There are no entries in the Status Line.

Silhouette Curves Tool

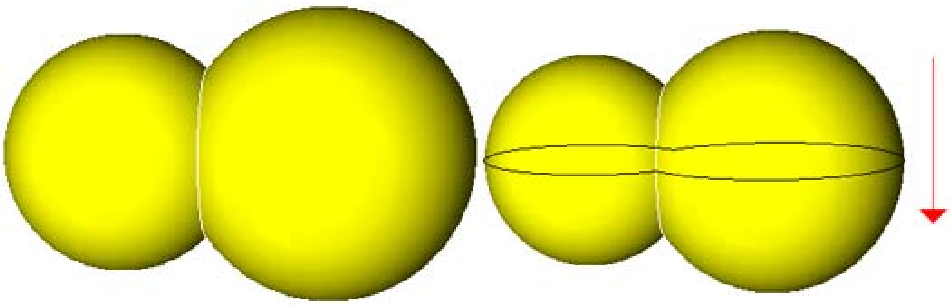


The **Silhouette Curves** tool creates curves from the bounding edge of an object relative to a user-defined projection direction. The curves produced by this tool are, in most cases, non-planar. This tool is usefule for producing injection mold drawings.

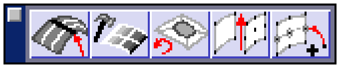
Using the Silhouette Curves Tool

1. Select the Silhouette Curves tool, The Message Line reads: *Silhouette Curves: Pick surface/solids for silhouettes. [Shift = Extend]*
2. Select the objects on which you want silhouette curves. The message line now reads, *Silhouette Curves: Specify two points for view direction*.

The example below shows the progression of use with the Silhouette Curves tool.



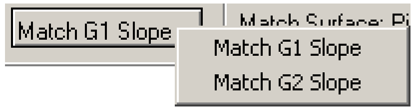
Local Surface Tools



Match Surface Tool

This tool gives you the ability to impose a smooth transition from one surface to another neighboring surface. It also allows you perform greater surface manipulation by inserting a knot and elevating the degree of the surface.

There are five options available for this tool: *Match G1 Slope*, *Match G2 Slope*, *Insert Knot*, *Elevate Degree* and *Untrim Surface*. These tools are accessed through the pull-down menu in the Message Line.



Rules and Notations for using the Match Surface Tool and Options

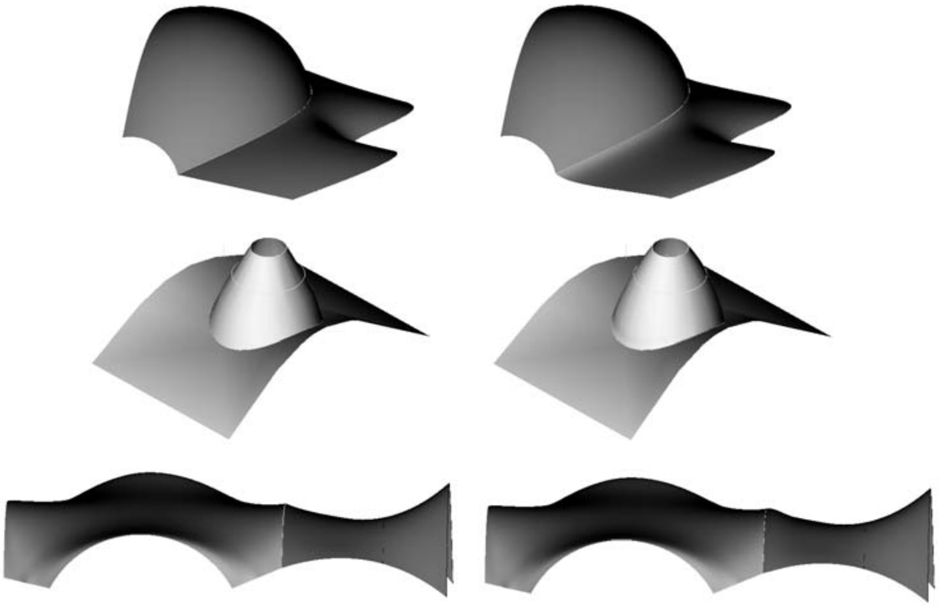
- Surfaces can not be trimmed surfaces. (Any surface that has had a relimit or another editing operation conducted on it is a trimmed surface.) If you want to match untrimmed surfaces, you must first untrim them using the *Untrim Surface* option of this tool before matching them. See a later section about this option.
- Surfaces must share the same edge being within 0.001 inches or 0.0254 mm of each other.
- The direction and slope of the surfaces do not matter.
- Matching two surfaces with the *G1 Slope* or *G2 Slope* options adjusts the slope of the first surface to that of the second. To reduce the influence of the matched slope on the surface, elevate its degree using the *Elevate Degree* option of this tool before matching the surfaces. See a later section about this option.

Match G1 Slope Option

Use this option to match two untrimmed surfaces with G1 continuity. G1 matched surfaces are tangent continuous at their shared edge. The graphics on the left below are the two original surfaces. The graphics on the right below are the same surfaces using the *G1 Slope* option.

Tech Note:

G1 stands for geometric continuity matched to the first derivative, tangency. G1 matched surfaces are tangent continuous.



Using the G1 Slope Surface Option

1. Select the **Match Surface** tool.

2. From the pull-down menu in the Message Line, select *Match G1 Slope*. The Message Line reads: *Match Surface: Pick surface edge to modify tangency*.

3. Click on the edge of the surface you want to modify.

   Be sure to click on the edge and not somewhere on the body of the surface. The entire surface highlights. The Message Line now reads, *Match Surface: Pick surface edge to match*.

4. Click on the surface edge to which you are matching the first surface.

   The slope of the first surface edge now matches the second surface edge with a G1 continuity. You can verify this continuity by using the surface evaluation feature. In a surface evaluation plot, stripes represent the slope of the surface.

These stripes line up from the first surface to the second surface for G1 continuous surfaces.

See the Surface Evaluation section at the end of this chapter for information.

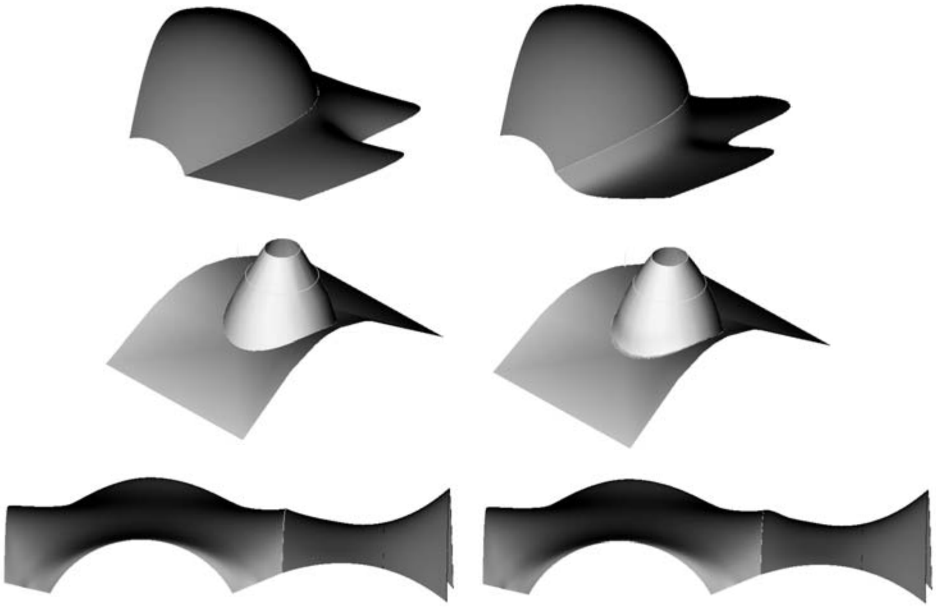
There are no entries in the Status Line.

Tech Note:

G2 stands for geometric continuity matched to the first and second derivative tangency and curvature continuous. G1 matched surfaces are tangent and curvature continuous.

Match G2 Slope Option

Use this option to match two untrimmed surfaces with G2 continuity. G2 matched surfaces are tangent and curvature continuous at their shared edge. The graphics on the left below are the two original surfaces. The graphics on the right below are the same surfaces using the *G2 Slope* option.



Using the G2 Slope Surface Option

1. Select the **Match Surface** tool.
2. From the pull-down menu in the Message Line, select *Match G2 Slope*. The Message Line reads: *Match Surface: Pick surface edge to modify tangency.*
3. Click on the edge of the surface you want to modify.

   Be sure to click on the edge and not somewhere on the body of the surface. The entire surface highlights. The Message Line now reads, *Match Surface: Pick* *surface edge to match.*

4. Click on the surface edge to which you are matching the first surface.

   The slope of the first surface edge now matches the second surface edge with a G2 continuity. You can verify this continuity by using the surface evaluation feature. In a surface evaluation plot, stripes represent the slope of the surface. These stripes line up from the first surface to the second surface for G2 continuous surfaces.

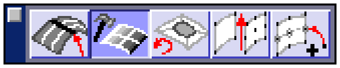
   See the Surface Evaluation section at the end of this chapter for information.

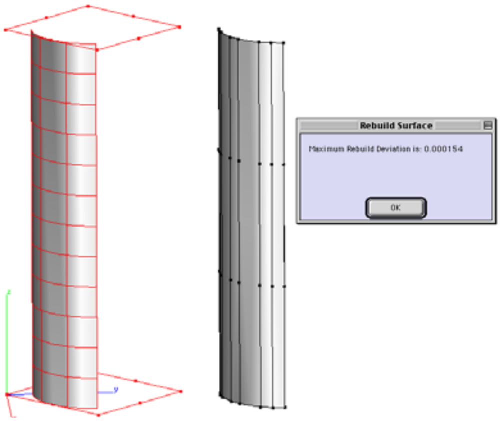
There are no entries in the Status Line.

Rebuild Surface Tool

The Rebuild Surface tool reconstructs an approximating surface to the surface to rebuild. The tightest tolerance achieved is displayed in a dialog box. If the tolerance achieved is not tight enough use undo to put the surface back in its orginal condition.

The Rebuild Surface tool is useful for converting analytics to cubic NURBs, trimmed surfaces to untrimmed surfaces, and repairing surfaces. The rebuild tool is limited to surfaces with 3 or 4 sides.



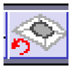


Using the Rebuild Surface Tool

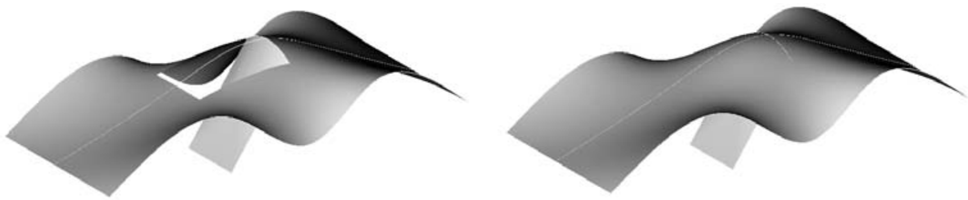
1. Select the Rebuild surface tool. The Message Line reads: Rebulid Surface:Pick surface to rebuild.
2. Click the surface to rebuild.

There are no status line entries for this command.

Untrim Surface Tool

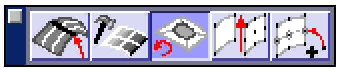


Using this option removes all trim boundaries on a surface face type surface. The left graphic below shows a trimmed surface. The right graphic shows the surface after this option is used.



Using the Untrim Surface Option

1. Select the **Untrim Surface** tool.

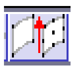


2. From the pull-down menu in the Message Line, select *Untrim Surface*. The Message Line reads: *Untrim Surface: Pick surface to untrim*.

   The surface untrims.

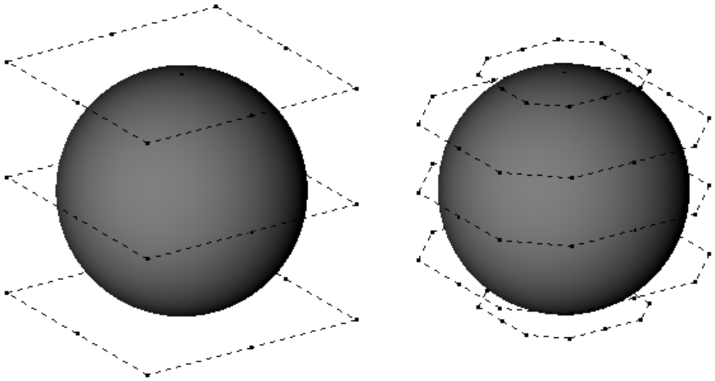
There are no entries in the Status Line.

Elevate Surface Tool

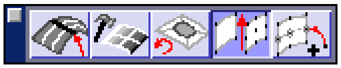


This option elevates the degree of a surface introducing more control points. These control points can be used to modify the surface.

NURB surfaces are defined by polynomial equations. The most basic equation possible is used to define the surface. Where cylindrical and spherical shapes are defined by second degree polynomials, skin, cover and net surfaces are defined by third degree polynomials due to their complexity. Shapes defined by higher degree polynomicals have more control points than those defined by lesser degree polynomials. This option raises the degree level for a shape providing more control points for surface manipulation.



Surfaces can be elevated up to the 22nd degree. However, it is recommended that you never elevate a curve higher than 9 degrees. Degree elevation is useful when trying to match surfaces and minimize the effect of the match on the surface.



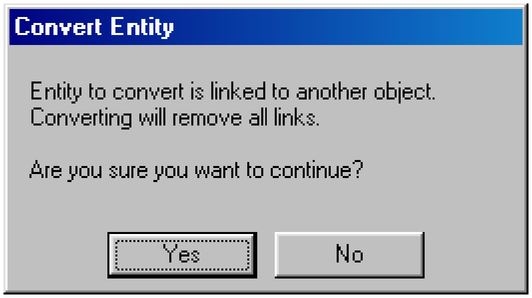
Using the Elevate Surface Tool

1. Select the surface to which you want to add control points.
2. Choose ***Window>Edit Objects*** to verify that the surface is a surface face.

   If your surface is a surface face continue with step 5.
3. Choose ***Edit>Change Object Type***.
4. Select the Surface option and click OK.

   If the surface you selected is linked to another object, the following warning box appears.

   Click Yes to continue conversion.



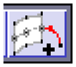
5. Choose ***Edit>Show Points*** to display the surface control points.
6. Select the **Elevate Surface** tool.

7. From the pull-down menu in the Message Line select: *Elevate Degree*. The Message Line reads: *Elevate Surface: Pick surface to elevate.*
8. Select the surface.

   The surface elevates to the next degree.

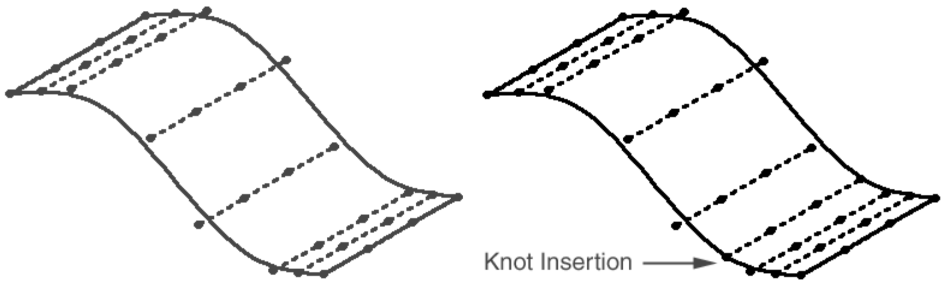
There are no entries in the Status Line.

Insert Knot Tool



This tool allows you to add a row or column of control points to a surface. The surface must be untrimmed and the *surface face* type. Net, skin and cover surfaces must be converted into the surface face type using the *Change Object Type* command in the Edit menu. You can verify the surface type by double-clicking on the surface to display the Edit Objects dialog box.

Knot insertion is valuable when you want to adjust a surface shape at a location where there are no control points. Knot insertion is also useful when trying to match surfaces and minimize the effect of the match on the surface. The left graphic below displays the control points of the original surface. The right graphic shows an additional row of control points at the right end of the surface.



After inserting control points using this option, you may notice that other control points may have shifted. This is automatically done to preserve the surface shape.

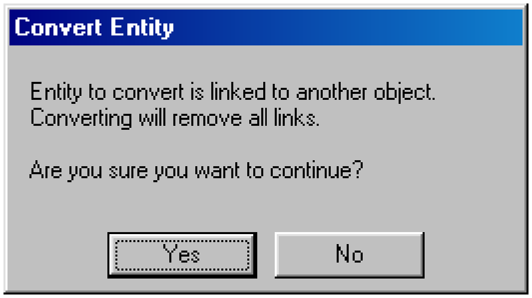
Using the Insert Knot Tool

1. Select the surface to which you want to add control points.
2. Choose ***Window>Edit Objects*** to verify that the surface is a surface face.

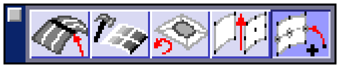
   If it is continue on with step 5.

3. Choose ***Edit>Change Object Type***.
4. Select the Surface option and click OK.

   If the surface you selected is linked to another object, the following warning box appears.

   Click Yes to continue conversion.
5. Choose ***Edit>Show Points*** to display the surface control points.
6. Select the Insert Knot tool.



7. From the pull-down menu in the Message Line select *Insert Knot*. The Message Line reads: *Insert Knot: Pick surface to insert a new knot.*
8. Select the surface. The Message Line now reads: *Insert Knot: Specify location for new knot.*
9. Select a location along an edge of the surface for the new row or column of knots.

   A new series of knots are added. If you want to add another row or column of knots you must select the surface again.

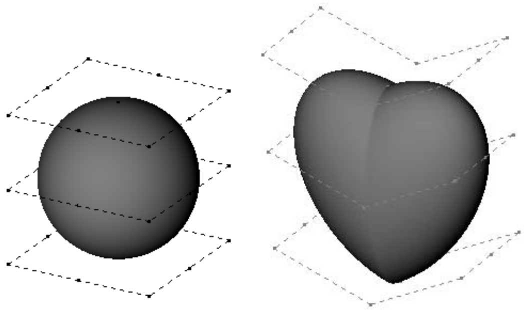
There are no entries in the Status Line.

Surface Control Vertex Modification

Our Designer Elements program supports the ability to manipulate the control points for all surface face type NURB surfaces. All other surfaces like net, skin and cover, must be converted into the surface face type using the *Change Object Type* command. Remember that using the *Change Object Type* command removes all links from the surface.

After showing points on the selected surface you can select any point and move it to a new location. With this ability you can create such things as creases and other surface variations. If surface smoothness is a concern, be sure to use the surface evaluation plot available through the Verify menu or the advanced rendering features. See the next section.

The left graphic below shows a sphere converted into a surface and displaying its control points. The right graphic shows the surface after control point modification creating a heart shape.

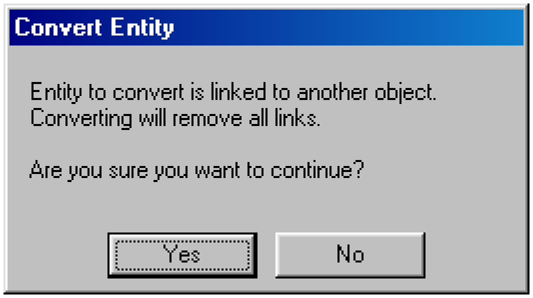


Performing a Control Vertex Modification

1. Select the surface you want to modify.
2. Choose ***Window>Edit Objects*** to verify that the surface is a surface face.

   If it is continue on with step 5.
3. Choose ***Edit>Change Object Type***.
4. Select the *Surface* option and click OK.

   If the surface you selected is linked to another object the following warning box appears:

   Click Yes to continue conversion.
5. Choose ***Edit>Show Points*** to display the surface control points.
6. Using the **Selection** tool, drag the control points as desired.

Surface Evaluation

Our Designer Elements program gives you the ability to evaluate the smoothness of your surfaces. This is useful for identifying surface irregularities. The curvature evaluation is accomplished by placing your surface inside a brightly lit imaginary cylinder with longitudinal stripes. The cylinder stripes are reflected onto the surface to convey the surface smoothness. This can especially important when you match surfaces using this **Match Surface** tool. The graphics here show the surface evaluation for two matched surfaces using no surface matching (G0), G1 and G2 surface matching.

G0 Matched Surfaces

G1 Matched Surfaces

G2 Matched Surfaces

Notice how the bands of black differ from graphic to graphic. In the G0 Matched Surfaces graphic (no surface match), the right surface bands are parallel. In the G1 Matched Surfaces graphic, the right surface bands are not as parallel showing that the surface was adjusted to match the left surface. In the G2 Matched Surface graphic, the right surface bands show how the surface has even further adjusted to match the left surface.

You can evaluate your surfaces either through the Verify menu using the *Surface Analysis* commands or through the advanced rendering feature.

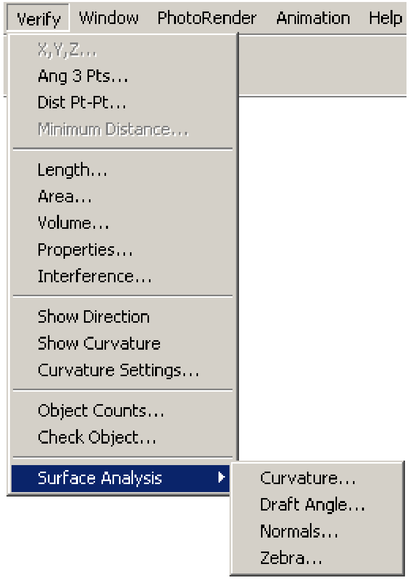
Evaluation Through the Verify Menu

Refferal Note:

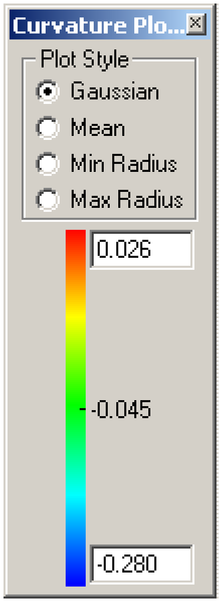
This section is duplicated in the Verify Menu section of Chapter 24.

The *Surface Analysis* commands allow you to quickly evaluate the smoothness of one or more surfaces. All analysis colors are preset with these commands. If you would like to define your own settings use the surface evaluation shader through the Advanced Rendering feature. See “Evaluation Through the Advanced Rendering Feature” on page 34.

You have four surface analysis commands, *Curvature*, *Draft Angle*, *Normals* and *Zebra*. When you choose ***Verify>Surface Analysis***, a submenu appears containing the commands.



Curvature Command



This command creates a curvature plot of your surface. When you select the command the Curvature Plot dialog box appears containing: Analysis Styles, Histogram, Color Spectrum and Histogram Data Fields.

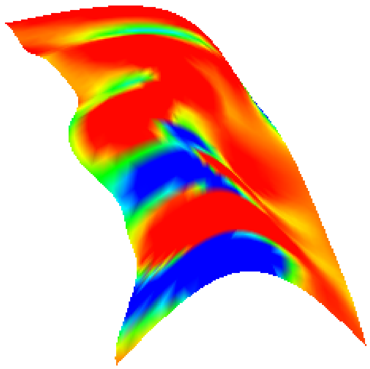
The dialog box contains the following elements:

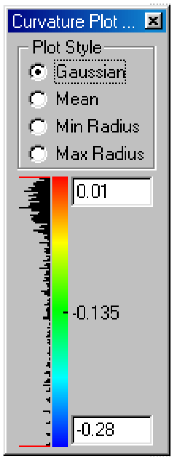
Plot Style — This section contains the styles for the analysis and include: *Gaussian*, *Mean*, *Min Radius* and *Max Radius*. A description and illustration of each style is provided in the next section.

Histogram — Located to the left of the color spectrum in the dialog box these horizontal bars represent the frequency of a curvature smoothness (change in a curve over the change in curvature) using the color spectrum. The length of the bar represents the frequency. Our Designer Elements program calculates this histogram and displays it so that the entire graph fits into the dialog box area.

Color spectrum bar The section displays the color spectrum used to create the histogram.

Histogram data fields The data fields display the maximum and minimum values used to calculate the histogram. When you initially select the command, our Designer Elements program scans the surface(s) and sets the min and max fields for curvature. If you enter different values in the fields such that a large number of curvature values fall outside of the specified range, a red line appears at the end(s) where the values fall. All values are still calculated even though they are not displayed due to the specified range. In the graphic here the large number of values fall outside the maximum value of 0.006.





If you make changes to the values our Designer Elements program waits for two seconds of non-action before recalculating the histogram, giving you time to change the values before the image is rendered again.

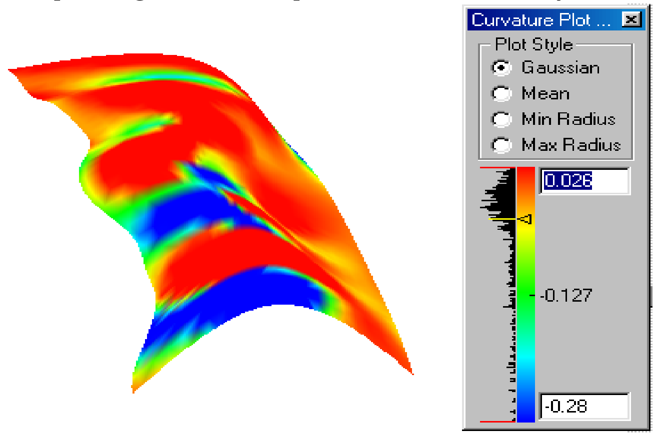
You can return to the range initially displayed by clicking on the selected option again. (The option does not deselect when you do this.) Windows users can also reset the maximum and minimum values by right mouse clicking near the respective end of the spectrum.

Plot Styles

You have four curvature plot styles to choose from when you select the *Curvature* command: *Gaussian*, *Mean*, *Min Radius* and *Max Radius*.

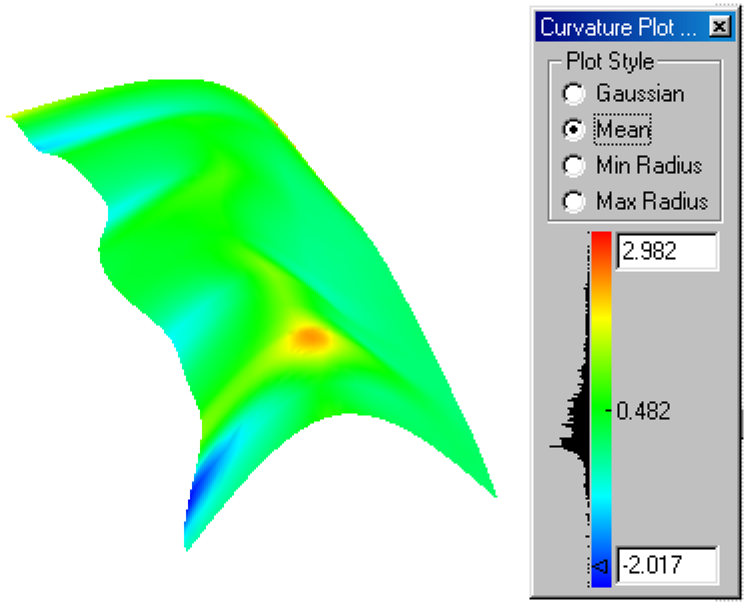
Gaussian

Selecting this option creates a Gaussian curvature plot on your selected surface. The plot registers the change in a curve over the change in curvature. Mathematically, the Gaussian value is the product of the kmin (minimum radius curvature) and kmax (maximum radius curvature) of each vertex. (The letter "k" refers to the curvature.) Any sharp change in color represents a discontinuity.



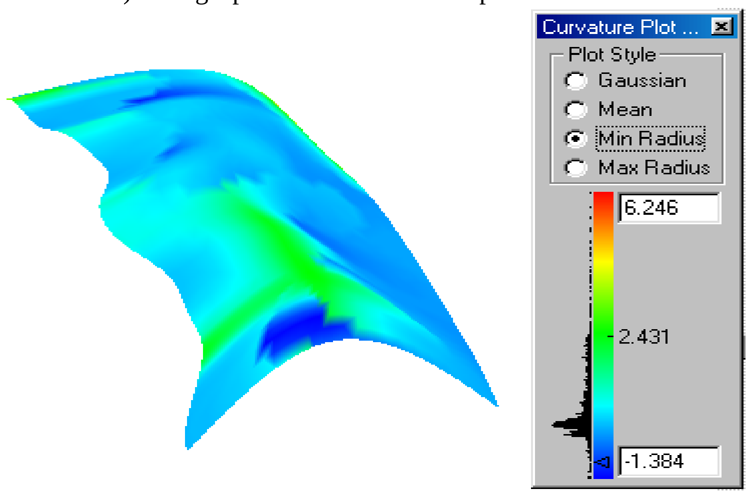
Mean

Selecting this option creates a mean curvature plot. Mathematically, the mean value is the average of the kmin (minimum radius curvature) and kmax (maximum radius curvature) of each vertex. (The letter “k” refers to the curvature.) The graphic here is an example.



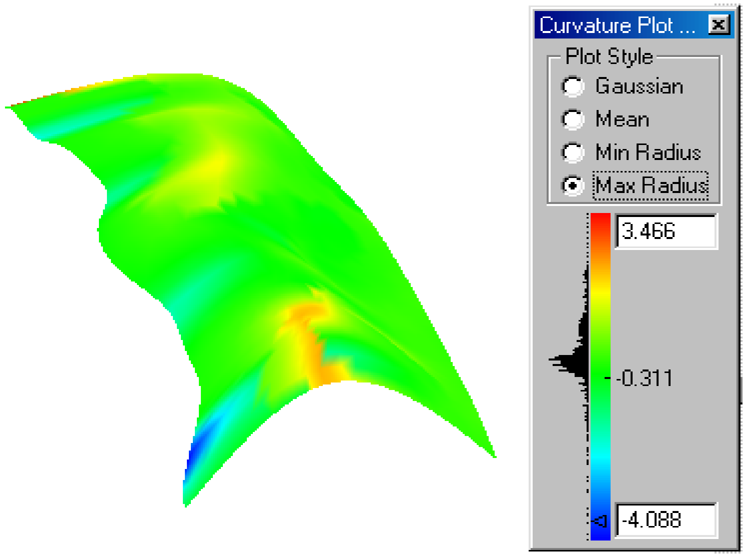
Min Radius

Selecting this option creates minimum radius curvature plot. Mathematically, the plot is the kmin values (minimum radius curvature) of each vertex. (The letter “k” refers to the curvature.) The graphic here is an example.



Max Radius

Selecting this option creates maximum radius curvature plot. Mathematically, the plot is the kmax value (maximum radius curvature) of each vertex. (The letter “k” refers to the curvature.) The graphic here is an example.



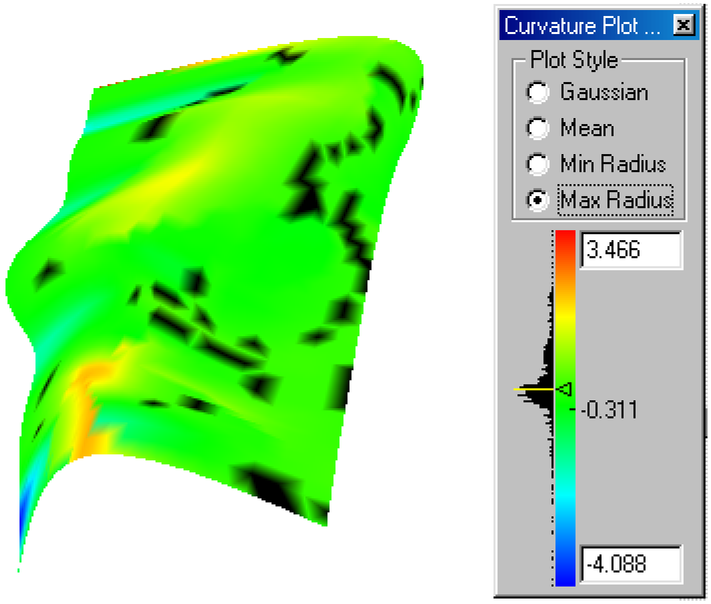
Using a Curvature Surface Analysis Command

1. Select the surface.
2. Choose ***Verify>Surface Analysis>Curvature***.

   The dialog box displays with the analysis. Your geometry also displays the analysis.
3. Select a Plot style, if the desired style is not already selected.

   A new curvature analysis appears.

4. To display a certain analysis/color area, place the pointer at the location over the color spectrum and click the mouse.

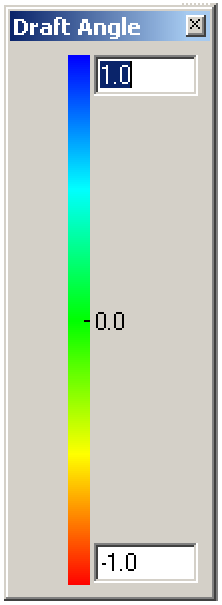


Notice that a triangular indicator appears at the selected location and the related color highlights in the histogram. The same color highlights in black on your surface.

5. To remove the triangular indicator click in the dialog box outside of the color spectrum bar.

6. Change the histogram values in the data fields and the histogram and analysis automatically recalculates.

Draft Angle



This command evaluates the drafts of an object for molding purposes. When you select this command the Draft Angle dialog box appears with the draft angle analysis containing the histogram with the analysis, the color spectrum and histogram data fields.

The dialog box contains the following elements:

Histogram — Located to the left of the color spectrum in the dialog box these horizontal bars represent the frequency of a draft angle using the color spectrum. The length of the bar represents the frequency.

Our Designer Elements program calculates this histogram and displays it so that the entire graph fits into the dialog box area.

Color spectrum bar — The section displays the color spectrum used to create the histogram.

Histogram data fields — The data fields display the maximum and minimum values used to calculate the histogram. When you initially select the command our Designer Elements program scans the surface(s) and sets the min and max fields for the draft angle analysis.

If you enter different values in the fields such that a large number of values fall outside of the specified range a red line appears at the end(s) where the values fall. All values are still calculated even though they are not displayed due to the specified range.

If you make changes to the values our Designer Elements program waits for two seconds of non-action before recalculating the histogram, giving you time to change the values before the image is rendered again.

You can return to the range initially displayed by clicking on the selected option again. (The option does not deselect when you do this.) Windows users can also reset the maximum and minimum values by right mouse clicking near the respective end of the spectrum.

The graphic here is an example of a draft angle analysis.

Using the Draft Angle Command

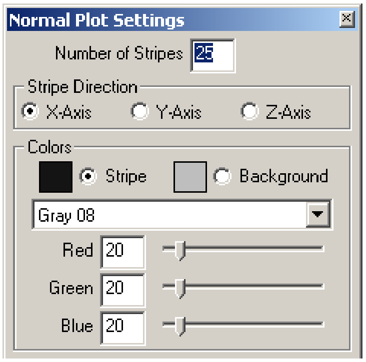
1. Select the object.
2. Choose ***Verify>Surface Analysis>Draft Angle***.

   The dialog box displays with the analysis. Your geometry also displays the analysis.
3. To display a certain analysis/color area place the pointer at the location over the color spectrum and click the mouse.

   A triangular indicator appears at the selected location and the related color highlights in the histogram. The same color highlights in black on your surface.
4. To remove the triangular indicator click in the dialog box outside of the color spectrum bar.
5. Change the histogram values in the data fields and the histogram and analysis automatically recalculates.

Normals

This command creates a normals plot of your surface. When you select the command the Normal Plot Settings dialog box appears.



The dialog box contains the following elements:

Number of Stripes This data field contains the number stripes that appear on your surface.

Stripe Direction This section provides three direction options: X-Axis, Y-Axis and Z-Axis.

Colors This sections contains the *Stripe Color* option and swatch, *Background Color* option and swatch; the color pull-down menu and RGB data fields and sliders.

Stripe Color option: When selected you can set the stripe color from the color pull-down menu or the RGB fields.

Background Color option: When selected you can set the surface background color from the color pull-down menu or the RGB fields.

This graphic shows an example of a normal surface analysis.

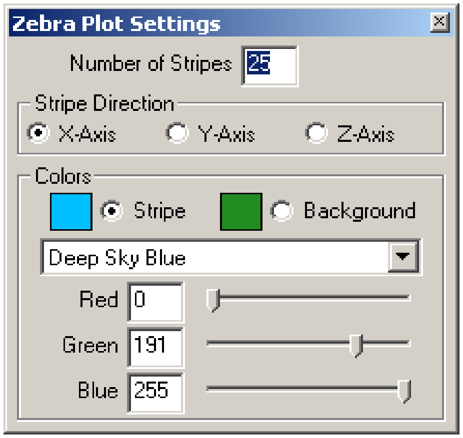
Using the Normal Command

1. Select the surface.
2. Choose ***Verify>Surface Analysis>Normals***.

   The dialog box displays with the analysis. Your geometry also displays the analysis.

Zebra

This command creates a zebra plot of your surface. When you select the command the Zebra Plot Settings dialog box appears.



The dialog box contains the following elements:

Number of Stripes This data field contains the number stripes that appear on your surface.

Stripe Direction This section provides three direction options: X-Axis, Y-Axis and Z-Axis.

Colors This sections contains the *Stripe Color* option and swatch, *Background Color* option and swatch, the color pull-down menu and RGB data fields and sliders.

Stripe Color option: When selected you can set the stripe color from the color pull-down menu or the RGB fields.

Background Color option: When selected you can set the surface background color from the color pull-down menu or the RGB fields.

This graphic shows an example of a zebra surface analysis.

Using the Zebra Command

1. Select the surface.
2. Choose ***Verify>Surface Analysis>Zebra***.

   The dialog box displays with the analysis. Your geometry also displays the analysis.

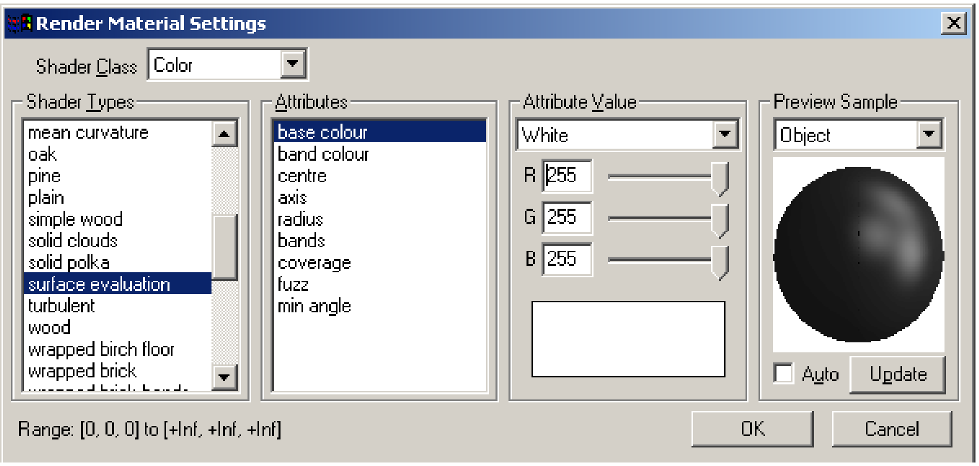
Evaluation Through the Advanced Rendering Feature

Through the advanced rendering feature you can evaluate your surfaces using user-defined attributes for the evaluation.

1. Select the surface.

   The surface must have a material applied to it.
2. Double-click on the surface to display the Edit Objects dialog box.
3. In the *Material* page of Edit Objects, click the Advanced button.

4. From the Color class choose the surface evaluation type. The following window displays:



base color	This attribute sets the base color of the cylinder used for the evaluation. This color is reflected onto your surface. If your surface color is dark and your base color is white, you will not notice much change to the surface color when rendered.
band color	This attribute sets the band or stripe color for the cylinder. This color is also reflected onto your surface. Use a color that is clearly visible to create the surface evaluation.
center	This attribute sets the center of the evaluation cylinder. Click the desired location in your drawing and the numbers automatically enter into the X, Y and Z data fields.
axis	This attribute specifies the cylinder axis (0 = X axis, 1 = Y axis, 2 = Z axis).
radius	This attribute sets the cylinder radius.
bands	This attribute sets the total number of bands around the evaluation cylinder.
coverage	This attribute sets the ratio of the area covered by the bands to the area not covered (displaying the

	base color). Entering a zero displays only the base color. Entering a one displays only the band color.
fuzz	This attribute controls the fuzziness of the band (0 = sharp band edges, 1 = fuzzy band edges).
minimum angle	This attributes controls the angle of the normal along the cylinder axis that defines the length of the cylinder and thus the reflection on the surface. Smaller values create longer cylinders.

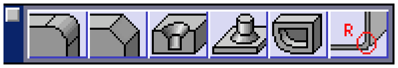
5. Specify the value for each attribute.
6. Click the Update button to preview your surface evaluation in the preview window.

   If you select the *Auto* option the preview window automatically updates each time you make an attribute change.
7. Click OK to save the changes and close the dialog box.

   Close the Edit Objects dialog box if you want a clean screen when you render your scene.
8. Render your scene using one of the photorealistic commands in the Render menu.

   The rendered scene shows your surface evaluation.

Solids Editing Tools - Features



After you've created a solid model from a primitive or a profile, you may use any of the feature tools to modify it. You cannot use these tools on grouped objects.

For each tool you can enter values in the Status Line either before or after you create the feature. If you enter the values after you select the tool but before you create the feature, your first click in the drawing area automatically registers all Status Line values. If you enter values after creating the feature and while it is still selected, pressing ENTER (Windows) or RETURN (Macintosh) updates the feature to reflect the new values.

Many of the tools have a two step process to use them. The first step after choosing the tool may be to select one or more objects. The second step is to perform the editing operation. Once you the complete the first step (which may involve holding down the SHIFT key to select multiple objects), you cannot select more objects to include in the same operation. To select additional objects reset the tool and start again.

After adding features to your solid you can modify them by selecting them in the Design Explorer and choosing ***Window>Edit Objects*** or double-click on their names in the Design Explorer **(Cobalt™ and Xenon™ only)**.

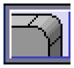
All features are associative. Any modifications made to the parent will automatically alter all child objects that reference it **(Cobalt™ and Xenon™ only)**.

The topics explained here include:

- **Blend** tools

- **Chamfer Edge** tools
- **Hole** tools
- **Boss** Feature
- **Shell Object** tool
- **Bend Solid** tool
- **Feature Editing** tools

Blend Tools

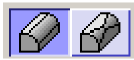


The Blend tools create fillets along the edge of a solid. When you select the Blend tool, a subpalette appears in the Message Line containing two tools for creating blends: Constant Blend and Variable Blend.

Tech Note:

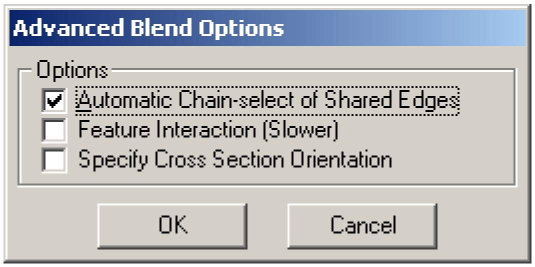
The **Blend** tools performs a similar operation on solids to what the fillet tool performs on a wireframe.

In the solids world, blend refers to both filleting and rounding. A filleting operation adds material to a concave intersection. A rounding operation removes material from a convex intersection.



Each of these tools includes a pull-down menu containing options for creating that type of blend. The graphic shown here is the Constant Length Blend tool menu.

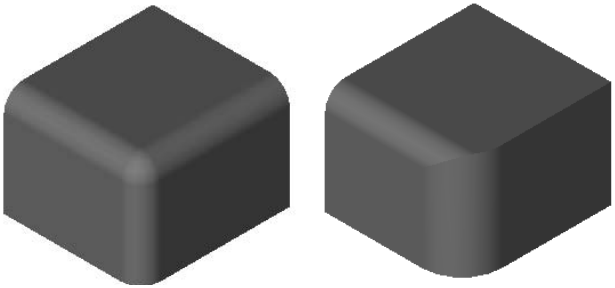
Many of these tool options also have advanced capabilities. These include: Constant Radial Blend, Constant Elliptical Blend, Variable Linear Blend, Variable By Position Blend, Variable Fixed Width Blend and Variable Hold Line Blend. These capabilities are indicated in the Message Line by the notation, [Ctrl = Advanced] (Windows) or [Option = Advanced] (Macintosh). When you press the CTRL (Windows) or OPTION (Macintosh) key, the Advanced Blend Options dialog box appears.



The dialog box contains the following options:

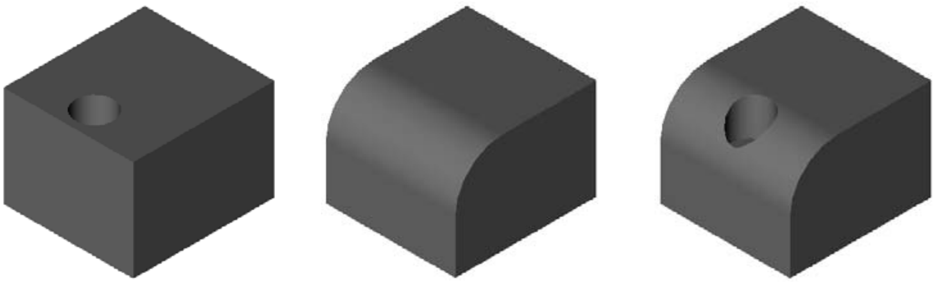
Automatic Chain-select of Shared Edges

Checking this box enables you to blend all edges that share a tangent with the selected edges. This is the default setting. Without this checked only the selected edges blend. The left graphic shows three edges blended with the Constant Radial Blend tool with this option selected. The right graphic shows two edges blended with the Constant Radial Blend tool without the option selected.



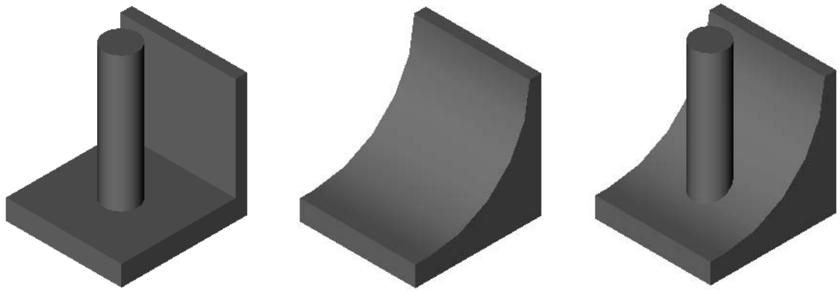
Feature Interaction (Slower)

Checking this box allows you to apply a blend that intersects a cutout or protrusion. Without this checked the feature may be deleted. With this checked the cutout is taken into account. The graphics here illustrate this. The left graphic is the original part with a hole cutout to half the part's depth. The middle right graphic is the blended part without this option selected. The right graphic is the blended part with the option selected.



The same thing occurs with a protrusion as shown below. As before, the left graphic is the original part;

the middle graphic is the part without the option selected and the right graphic is the part with the option selected.

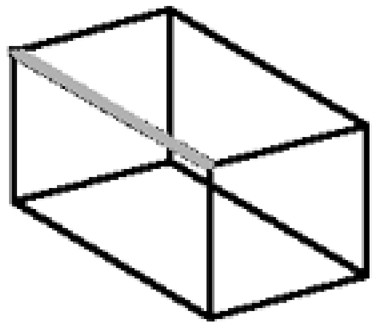


Using this option results in a longer blending time.

Specify Cross Section Orientation

Checking this box allows you to specify the orientation of the cross-section for the blend. This is especially valuable if you are blending an edge between faces containing a draft and you want to maintain the arc. See the next section for instruction on using this option.

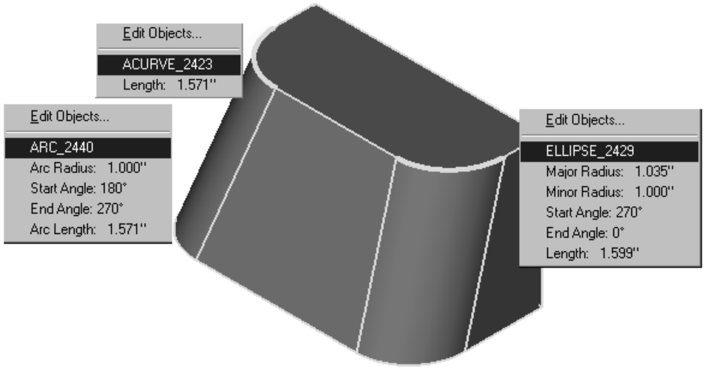
If your model contains a problem that prevents a successful blending operation, an ACIS Error dialog box appears highlighting the location of the problem.



Specifying a Cross Section Orientation for a Blend

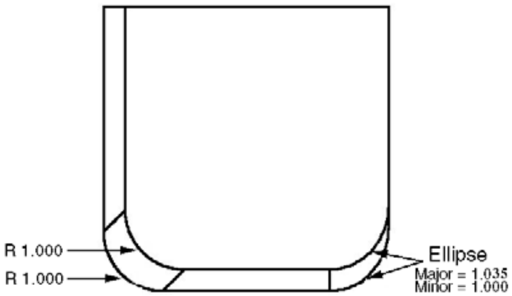
By default, this Designer Elements program uses the selected edge to define the orientation of the cross-section resulting in an arc shape for the blend. However, when you apply a blend to an object with a draft or taper, the cross-section of the resulting blend is elliptical because the blended edge is not perpendicular to the intersecting faces. If you want to maintain the arc shape for the cross-section, use the Specify Cross Section Orientation option in the Advanced Blend Options dialog box.

The graphic below shows a tapered object (tapered on the left and front sides) with a cross-section blend on the left and a standard blend on the right. If you hold down the right mouse button (Windows) or CONTROL and the mouse button (Macintosh), you can verify the object shape. (In the case of the cross-section blend an acurve object is listed. Use one of the Arc tools to trace the shape and verify that the blend shape is an arc.)



The graphic to the right is a top view of a similar shape and provides the measurements for the blends.

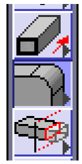
Before using this option you must create a curve along the z axis for specifying the cross-section orientation.



Using the Cross Section Orientation Option

Although the example here uses the Radial option of the Constant Blend tool, this option is also available for Constant Elliptical Blends, Variable Linear Blends, Variable By Position Blends, Variable Fixed Width Blends and Variable Hold Line Blends.

1. Create a curve along the z-axis.
2. Select the Blend tool.

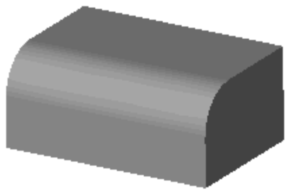


3. In the Message Line select the Constant Blend tool and the Radial option for the pull-down menu. The Message Line reads: *Radial Blend: Pick edges AND/OR faces to blend [Ctrl=Advanced (Windows) or Option=Advanced (Macintosh), Shift=Extend].*
4. Press CTRL (Windows) or OPTION (Macintosh) to display the Advanced Blend Options dialog box.
5. Select the Specify Cross Section Orientation option and click OK to close the dialog box.
6. Select the edge(s) for the blend.

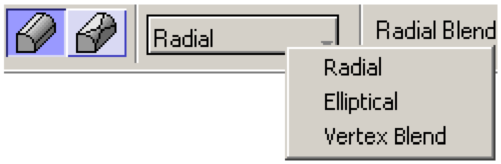
   The Message Line now reads, *Blend Solid: Pick curve to define cross section orientation*.
7. Select the curve.

   The blend is created.

Constant Blend Tool



When you select this tool its option pull-down menu appears.



You have three constant blend options: Radial, Elliptical and Vertex Blend.

Important: (Windows users) When a constant blend option uses the CTRL and SHIFT keys to perform specific operations for the tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when blending.

Using the Radial Option

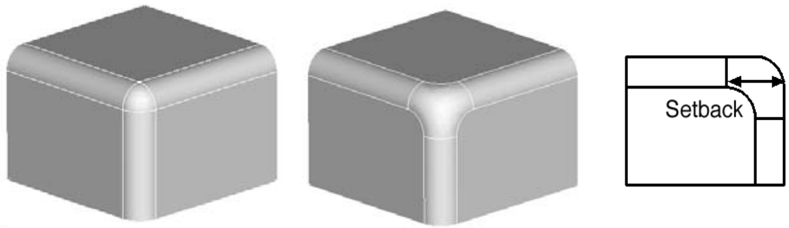
Tip:

If you are having difficulty creating a radial blend, try creating a variable linear blend using the same radius for the beginning and the end. ACIS uses a different algorithm for linear blends.

This option adds a blend with a constant radius along the edges you select and with a radius you specify in the Status Line.

If you select three or more edges you can create a smooth vertex blend or a setback blend. A setback is the distance that a blend extends back along the object edge from the vertex of the intersecting edges. A spherical corner results at the intersection of the three edges.

The left graphic below shows a three edge blend with no setback; the middle graphic shows the same object with a setback and the right graphic is a setback illustration.



1. Select the Blend tool.
2. In the Message Line select the Constant Blend tool and the Radial option for the pull-down menu. The Message Line reads: *Radial Blend: Pick edges AND/OR faces to blend [Ctrl=Advanced (Windows) or Option=Advanced (Macintosh), Shift=Extend]*.

   Set the advanced blend options by pressing CTRL (Windows) or OPTION (Macintosh), if desired.
3. If you want a setback enter the value in the Status Line data field.
4. Select the edge(s) and/or face(s) for the blend.

While the blend is still selected you can change the radius of the blend in the Status Line. Type a new value and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the Radius and Setback for the blend.

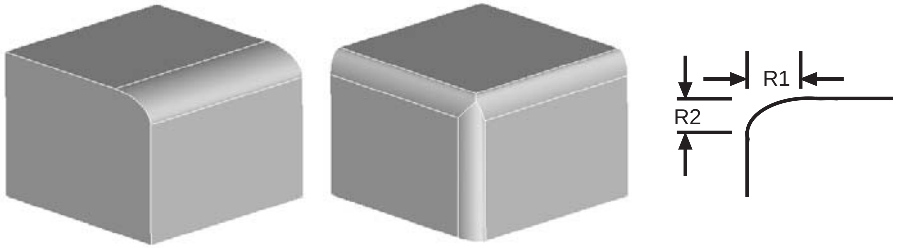
Geometric Characteristics

A constant radial blend is created by choosing a solid edge and specifying a radius and setback, if desired. It is made up of these characteristics according to Edit Objects: Setback, Radius and a list of the edges blended with their radii.

You can select the blend using the Design Explorer. Double-clicking on the item in the Design Explorer or choosing Edit Parameters command from the Design Explorer menu displays the Edit Objects dialog box. You can make changes to the settings on the Display and Attribute pages. See Chapter 24 for more information on the Design Explorer.

Using the Elliptical Option

This option adds a blend with a constant elliptical radius along the edges, based on the radii you specify in the Status Line. When using this option the automatic chain select of Shared Edges feature does not function. You must either select all intersecting edges before blending or blend each edge separately. The left graphic below shows a basic elliptical blend. The middle graphic shows an elliptical blend with a mitered edge. The right graphic illustrates the radius variables.



1. Select the Blend tool.
2. In the Message Line select the Constant Blend tool and the Elliptical option from the pull-down menu. The Message Line reads: *Elliptical Blend: Pick edges to blend [Ctrl=Advanced (Windows) or Option=Advanced (Macintosh), Shift=Extend*].

   Set the advanced blend options by pressing CTRL (Windows) or OPTION (Macintosh), if desired.
3. Enter a radius values for the ellipse, R1 and R2, in the Status Line data field. R1 is the major axis radius and R2 is the minor axis radius.
4. Select the edge(s) for the blend.

While the blend is still selected you can change the radius and elliptical ratio of the blend in the Status Line. Type a new value and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the Radius and Ratio for the blend.

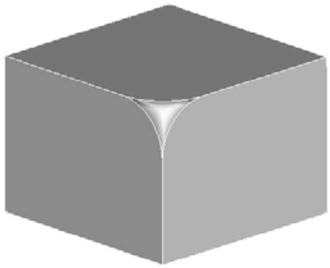
Geometric Characteristics

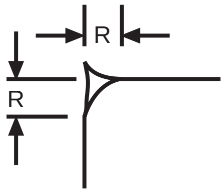
A constant elliptical blend is created by choosing a solid edge and specifying a radii for the major and minor axis of the ellipse. It is made up of these characteristics according to Edit Objects: R1, R2 and a list of the edges blended with their radii.

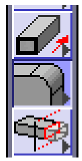
You can select the blend using the Design Explorer. Double-clicking on the item in the Design Explorer or choosing Edit Parameters command from the Design Explorer menu displays the Edit Objects dialog box. You can make changes to the settings on the Display and Attribute pages. See Chapter 24 for more information on the Design Explorer.

Using the Vertex Blend Option

This option adds a vertex blend at the intersection of three or more edges based on the radius you specify in the Status Line. The left graphic shows an example of a vertex blend. The right graphic illustrates the radius variables.







1. Select the Blend tool.
2. In the Message Line select the Constant Blend tool and the Vertex Blend option from the pull-down menu. The Message Line reads: *Vertex Fillet: Select vertex to blend.*
3. Enter a radius in the Status Line data field.
4. Select the vertex for the blend.

While the blend is still selected you can change the radius of the blend in the Status Line. Type a new value and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the radius (R) for the blend.

Geometric Characteristics

A constant vertex blend is created by choosing three or more solid edges and specifying a radius. It is made up of one characteristic according to Edit Objects: Radius.

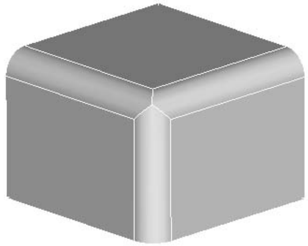
You can select the blend using the Design Explorer. Double-clicking on the item in the Design Explorer or choosing Edit Parameters command from the Design Explorer menu displays the Edit Objects dialog box. You can make changes to the settings on the Display and Attribute pages. See Chapter 24 for more information on the Design Explorer.

Additional Examples

The Constant Blend tool enables you to create a variety of blends. Here are some examples of advanced blends and the steps to create them.

Mitered Blend

A mitered blend can be created by turning off the Automatic Chain-select of Shared Edges option in the Advanced Blend Options dialog box and selecting each edge individually. A mitered corner results at the intersection of the three edges.



Creating a Mitered Blend

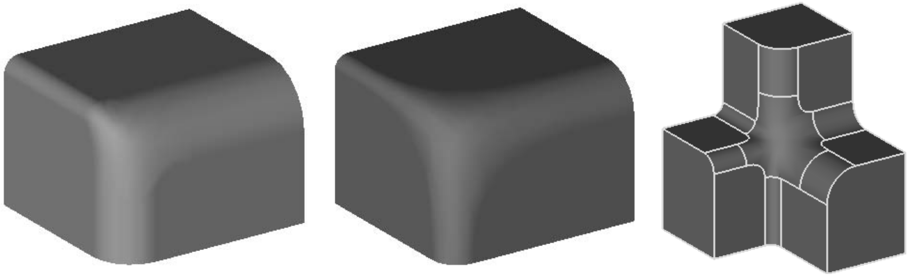
1. Select the Blend tool.
2. In the Message Line, select the Constant Blend tool and the Radial option from the pull-down menu. The Message Line reads: *Radial Blend: Pick edges AND/OR faces to blend [Ctrl=Advanced (Windows) or Option=Advanced (Macintosh), Shift=Extend].*
3. Press CTRL (Windows) or Option (Macintosh) to display the Advanced Blend Options dialog box.
4. Turn off the Automatic Chain-select of Shared Edges option and click OK.
5. Enter the desired radius in the Status Line.
6. Select each of the edges individually to create a mitered intersection. (If you hold down the SHIFT key a smooth vertex blend will be created rather than a mitered blend.)

   If you want a different radius for each edge blend enter the new value in the Radius data field before selecting the edge.

   A mitered blend is created.

Multi-radius Blend without and with a Setback

A multi-radius blend is created after applying a constant radial blend to an object and then changing the radius of each edge through the Edit Objects dialog box. You can also add a setback to this blend through Edit Objects or when creating the radial blend initially. The left graphic shows a multi-radius blend. The middle graphic shows the same object with a setback. The right graphic shows a multi-radius, multi-edge blend with a setback.



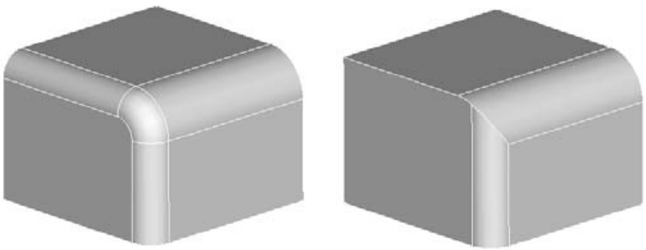
1. Select the object.
2. Choose ***Window>Design Explorer***.
3. Expand the history tree to display the fillet blend.(**Cobalt™** and **Xenon™** only)
4. Double-click on the fillet blend to display the Edit Objects dialog box.
5. Select the edge with the radius you want to change. The radius value appears in the data field. Enter a new radius.

   Change the other radii, if desired.
6. Click Apply to accept the changes and your geometry updates.

Chain Select Shared Edges On/Off Blend

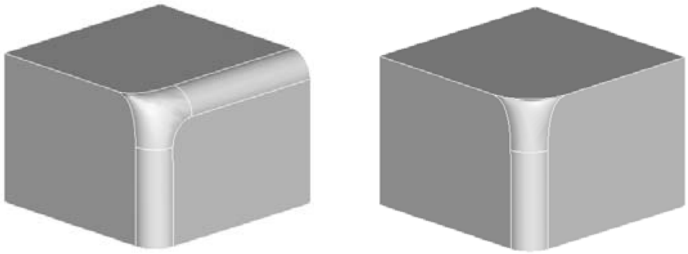
Two different kinds of blends are created depending on whether the Automatic Chain-select of Shared Edges option in the Advanced Blend Options dialog box is selected.

When this option is selected on edges with different radii, an object blends to create tangent continuous edges. In the left graphic the two edges have the same radius. When this option is not selected and different radii are used, the edges are capped, as shown in the right graphic.



Two Edge or One Edge Vertex Blend

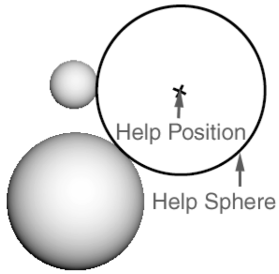
You can create two and one edge vertex blends by first creating a constant radial blend. Using the Design Explorer double-click on the blend to display the Edit Objects dialog box. For a two edge vertex blend change the radius of one edge to zero. For a one edge vertex blend, change two radii to zero. The left graphic below shows the two edge vertex blend and the right graphic a one edge vertex blend.



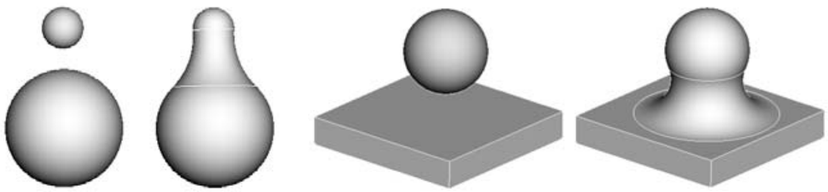
Disjoint Blend

A disjoint blend is a blend between faces that do not touch or are on two separate bodies. To create this blend in the past you had to perform a boolean operation. A disjoint blend is created by specifying a radius for the blend, selecting the two faces or objects and specifying a blend help position.

The help position tells this Designer Elements program where to begin calculating the blend. Visualize a sphere with this help position at its center. This sphere rolls along the virtual blend between the two bodies as this Designer Elements program calculates the actual blend. The sphere diameter equals twice the blend radius. The illustration here shows the help sphere and the help position for two disjoint bodies and applies to all disjoint blend operations.



The graphics below are two examples of objects before and after a disjoint blend is created.



Tips:

- If a help position does not create the blend try another location in the same quadrant.
- When creating a blend between objects of different colors, the color of the first object selected becomes the color of the new blended object.

Creating a Disjoint Blend

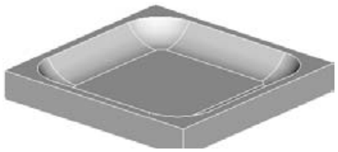
1. Select the Blend tool.
2. In the Message Line, select the Constant Blend tool and the Radial option from the pull-down menu. The Message Line reads: *Radial Blend: Pick edges AND/OR faces to blend [Ctrl=Advanced (Windows) or Option=Advanced (Macintosh), Shift=Extend]*.
3. Hold the SHIFT key down and select the objects. The Message Line now reads, *Constant Blend: Specify help position*.
4. Click the help position location. (If the help position does not create the blend try another location in the same quadrant.)

A disjoint blend is created.

Disjoint Edge to Face Blend

A disjoint edge to face blend is created by selecting the edge and face of an object. What makes this blend unique is that our Designer Elements program allows you to specify a radius for the blend that is greater than the distance from the edge to the face.

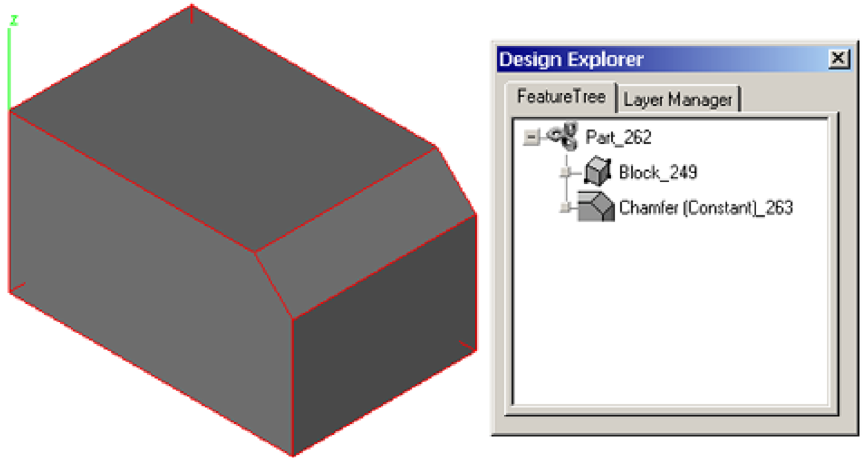
In the graphic here a one inch blend is applied using the inside top edge of the object and its inside bottom face. The user supplied help position is important in this blend. If you choose the wrong location the blend will not work. The one inch radius is larger than the distance from the edge to the face.



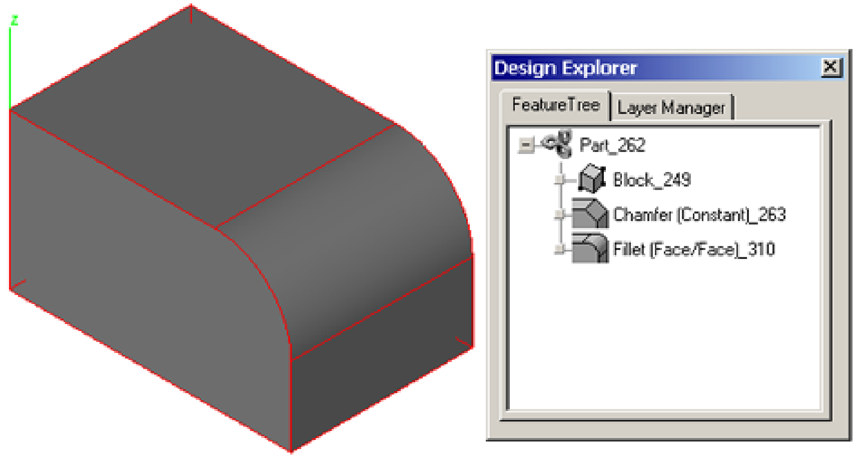
Disjoint Face to Face Remote Blend

A disjoint face to face remote blend is created when applying a blend to two faces that do not share the same edge. In graphic below, a face to face remote blend is applied to the top and right face separated by a chamfer. Below are the before (chamfered block) and after pictures (Blended top and right face seperated by the chamfer).

Before.

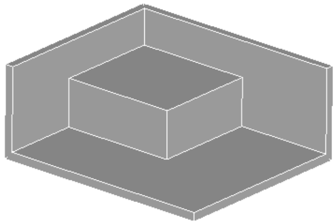


After



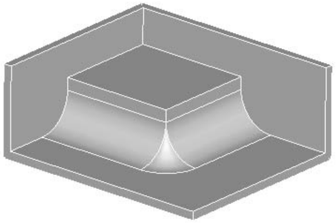
Notice in the Design Explorer, shown above, that the fillet blend does not replace the chamfer but follows the chamfer operation in the history tree.

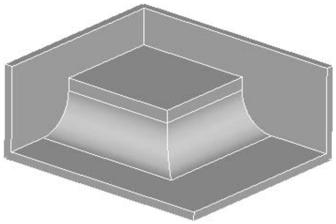
Edge to Face Roll Blend



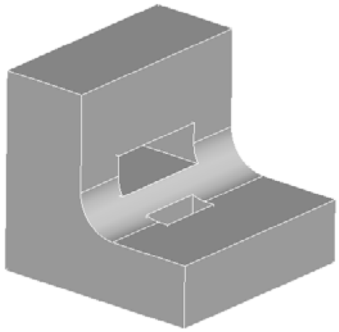
An edge to face roll blend is create by specifying a radius, selecting an edge and face to blend and choosing a help position. The graphic to the right shows the original object used for this blend.

The left graphic below shows the edge to face roll blend in which the vertical edge and the inside bottom face are selected. Using the Drafting Assistant, the help position is located out from the shared edge/face corner vertex along the z axis. The right graphic shows the default blend if the vertical faces and inside bottom faces were selected.





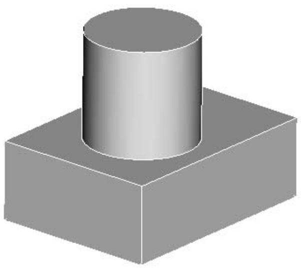
Blend with Cutouts

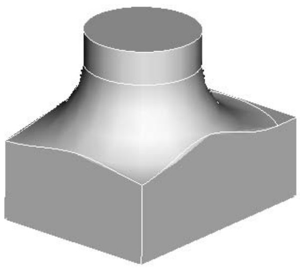


A blend with cutouts is created by applying a blend to faces that contain cutouts using the Constant Blend tool. There are no special steps to create this blend.

Extrapolation Blend

An extrapolation blend is created when you apply a constant radial blend to an edge with a radius that is larger than what the intersecting face will accommodate. The intersecting face extends to accept the blend. The left graphic below is the original object. The right graphic shows that the faces were extended to accommodate the blend.



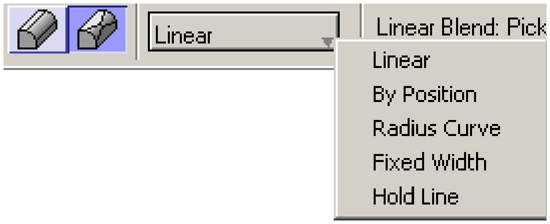


Creating an Extrapolation Blend

1. Select the Blend tool.
2. In the Message Line, select the Constant Blend tool and the Radial option from the pull-down menu. The Message Line reads: *Radial Blend: Pick edges AND/OR faces to blend [Ctrl=Advanced (Windows) or Option=Advanced (Macintosh), Shift=Extend]*.
3. Enter the radius in the Status Line.
4. Select the edge to blend.

The blend is created and the intersecting side extends as necessary.

Variable Blend Tool



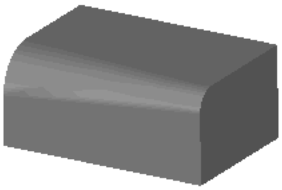
When you select this tool its option pull-down menu appears.

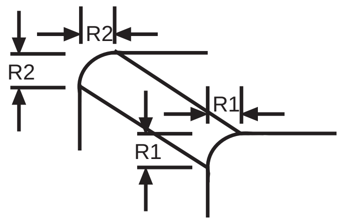
You have five variable blend options: Linear, By Position, Radius Curve, Fixed Width and Hold Line.

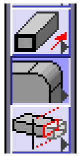
Important: (Windows users) When a variable blend option uses the CTRL and SHIFT keys to perform specific operations for the tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when blending.

Using the Linear Option

This option adds a blend with a tapered radius along the edges you select. You specify the beginning radius (R1) and the ending radius (R2) in the Status Line.







1. Select the Blend tool.
2. In the Message Line, select the Variable Blend tool and the Linear option from the pull-down menu. The Message Line reads: *Linear Blend: Pick edge to linearly blend. [Pick edge near R1][Ctrl=Advanced (Windows) or Option=Advanced (Macintosh)]*.
3. Select the edge along which to make the blend.

   The solid edge closest to your point of selection becomes R1.

While the blend is still selected you can change the radii (R1 and R2) in the Message Line. Type the new values and press ENTER (Windows) or RETURN (Macintosh).

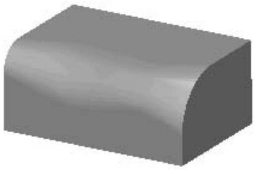
The Status Line contains R1 (beginning radius) and R2 (ending radius) for the linear blend.

Geometric Characteristics

A variable linear blend is created by choosing a solid edge and specifying a radius for the beginning and end of the blend. It is made up of these characteristics according to Edit Objects: Radius 1 and Radius 2.

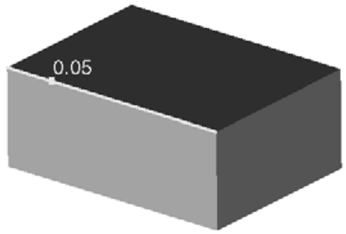
You can select the blend using the Design Explorer. Double-clicking on the item in the Design Explorer or choosing Edit Parameters command from the Design Explorer menu displays the Edit Objects dialog box. You can make changes to the settings on the Display and Attribute pages. See Chapter 24 for more information on the Design Explorer.

Using the By Position Option



This option adds a blend with a varied radius along the edges you select. You specify the various radii in the Status Line.

1. Select the Blend tool.
2. In the Message Line, select the Variable Blend tool and the By Position option from the pull-down menu. The Message Line reads: *Position Blend: Pick edge(s) for variable radius blend. [End=ESC,Dbl Click] [Ctrl=Advanced (Windows) or Option=Advanced (Macintosh)] [Shift=Extend]*.
3. Select the edge along which to make the blend. The Message Line reads: *Position Blend: Specify location for radius value [End=ESC, Dbl Click]*.
4. Enter a radius in the R field of the Status Line.
5. Specify the location(s) along the selected edge for the radius.



As you apply a radius to a location the value appears on the edge.

6. Enter a new radius in the R field.
7. Specify the location for this radius along the edge.

   Continue following this procedure until you have entered all the desired radii. Double-click the last place radius point and the variable by position blend is created.

The Status Line contains the Radius for the blend.

Geometric Characteristics

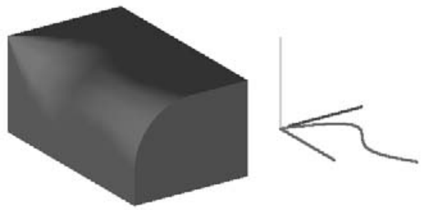
A variable position blend is created by choosing a solid edge and various radii at different locations along the edge for the blend. It is made up of these characteristics according to Edit Objects: a list of the edges blended with their radii and Param points, Radius and Param value. Param is the percentage point location along the edge for the selected item in the list. When you select one of the items in the list its Radius displays in the Radius data field and its percentage point location displays in the Param field.

You can select the blend using the Design Explorer. Double-clicking on the item in the Design Explorer or choosing Edit Parameters command from the Design Explorer menu displays the Edit Objects dialog box. You can make changes to the settings on the Display and Attribute pages. See Chapter 24 for more information on the Design Explorer.

Using the Radius Curve Option

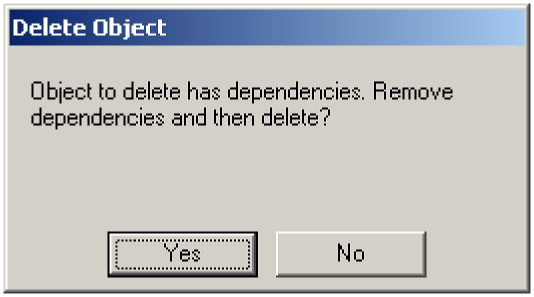
This option adds a blend based on a specified radius curve. Before using this option you must create an open curve in the positive **xy** quadrant. This curve cannot cross over itself or backtrack. A circle or a loop would not be acceptable curves. The graphic here is an example of an acceptable curve.

This curve defines the radius distribution. The start point of the curve represents the start point of the edge. The **y** value of the curve represents the radius. The **x** length of the curve is divided into percentages. The **y** value at a particular per-



centage point on the curve is mapped as the radius to the same percentage point along the selected solid edge.

One advantage of this tool is the ability to modify the blend by changing the curve. You can also add or remove control points to the radius curve to change the blend. The blend is associative to the curve. Changing the curve, automatically changes the blend. If you attempt to delete the curve, the following dialog box appears:



Deleting the curve changes the object from a solid block to an ACIS solid.

1. Create the curve that will be referenced for the blend.

   To ensure that the curve is in the positive **xy** quadrant, change the view to Top and draw the curve using the Axis as a reference.

1. Select the Blend tool.
2. In the Message Line, select the Variable Blend tool and the Radius Curve option from the pull-down menu. The Message Line reads: *Radius Curve Blend: Pick edge for variable radius blend*.
3. Select the solid edge. The Message Line now reads, *Radius Curve Blend: Pick radius curve (in positive xy plane)*.
4. Select the radius curve.

   The blend is applied to the edge.

There are no Status Line entries.

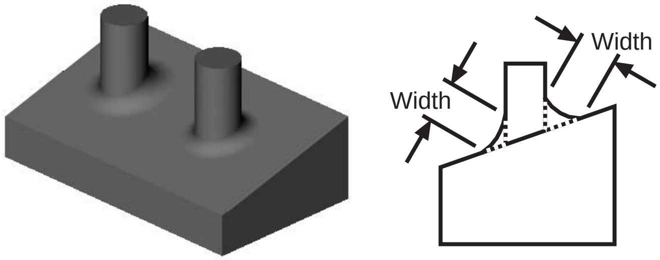
Geometric Characteristics

A variable radius curve blend is created by choosing a solid edge and a curve in the positive **xy** quadrant to specify the radii along the edge. According to Edit Objects, a radius curve blend contains no unique geometric characteristics. You can make changes to the Display and Attribute settings through the Edit Objects dialog box. You cannot change the blend radius through the Edit Objects dialog box. It must be done using the original curve.

Select the curve and change the **x** or **y** location in Edit Objects or drag the control point on the curve to the new positive **x**, or **y** location (The **z** location must remain at zero).

Using the Fixed Width Option

This option adds a blend with a fixed width along the edges you select adjusting the blend radius to maintain the width. This is important when placing a blend along the edge of two faces where the angle changes. You specify the width in the Status Line. In the graphic, the left boss has a constant radial blend with a .5 inch radius and the right graphic a fixed width blend with a .5 inch width.



1. Select the Blend tool.
2. In the Message Line, select the Variable Blend tool and the Fixed Width option from the pull-down menu. The Message Line reads: *Fixed Width Blend: Pick edges AND/ OR faces to blend [Ctrl=Advanced (Windows) or Option=Advanced (Macintosh), Shift=Extend].*
3. Enter the fixed width for the blend in the Status Line.
4. Select the edge for the blend.

   A fixed width blend is created.

The Status Line contains the width (W) of the blend.

Geometric Characteristics

A variable fixed width blend is created by choosing a solid edge and specifying the blend width. It is made up of these characteristics according to Edit Objects: Radius and a list of edges blended with their associated width.

You can select the blend using the Design Explorer. Double-clicking on the item in the Design Explorer or choosing Edit Objects command from the Design Explorer menu displays the Edit Objects dialog box. You can make changes to the settings on the Display and Attribute pages. See Chapter 24 for more information on the Design Explorer.

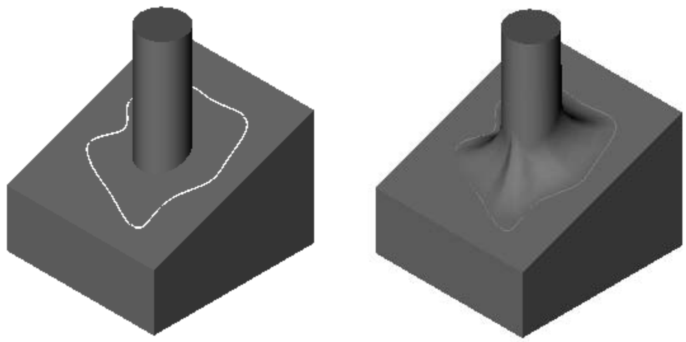
Using the Hold Line Option

This option adds a blend to the hold line curve specified adjusting the blend radius as needed to fit the line.

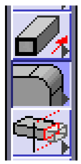
Hold Line Rules:

- The hold line curve should be one curve (circle, ellipse or spline). If the curve is not closed, a linear blend is applied where there is no hold line using the open ends of the curve for the radius values.
- Although the hold line curve does not have to be on the plane of the surface, it should be close to it and in the same plane as the surface. Curves not on the surface are projected to the closest surface to calculate the blend.
- Grouped curves are not supported.

The left graphic shows a part with a spline on the slanted face. The right graphic shows the same part using the spline as the hold line for the blend.



1. Create a hold line curve.
2. Select the Blend tool.
3. In the Message Line, select the Variable Blend tool and the Hold Line option from the pull-down menu. The Message Line reads: *Hold Line Blend: Pick edges AND/OR faces to blend [Ctrl=Advanced (Windows) or Option=Advanced (Macintosh), Shift=Extend]*.



4. Select the edge for the blend. The Message Line now reads, *Hold Line Blend: Pick curve for hold line [Shift = Extend]*.

5. Select the hold line curve.

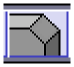
   A hold line blend is created.

There are no Status Line entries.

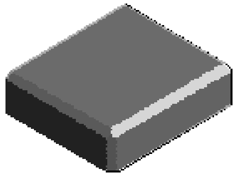
Geometric Characteristics

A variable hold line blend is created by choosing a solid edge and a curve for the hold line. According to the Edit Objects, a hold line blend contains no unique geometric characteristics. You can make changes to the Display and Attribute settings through the Edit Objects dialog box.

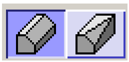
Chamfer Edge Tools



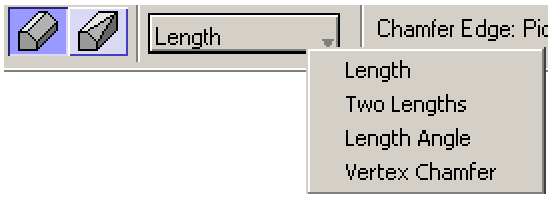
The Chamfer Edge tools create flat surfaces along a specified edge. The graphic here shows a chamfered solid along all four edges.



When you select the Chamfer Edge tool a subpalette appears in the Message Line containing two chamfer edge tools: Constant Length and Variable Length.

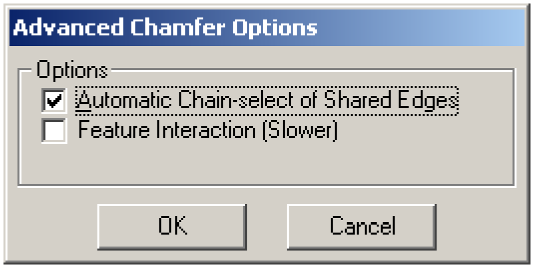


Each of these tools includes a pull-down menu containing options for creating that type of chamfer. The graphic shown here is the Constant Length Chamfer tool menu.



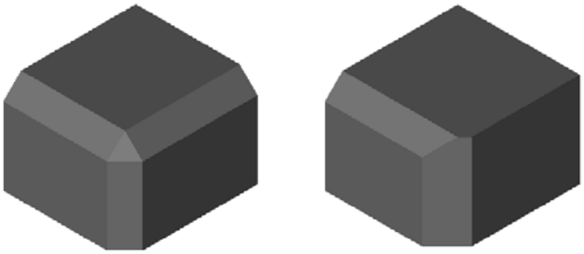
Many of these tool options also have advanced capabilities. These include: Constant Length Chamfer, Two Lengths Chamfer, Length Angle Chamfer, Variable Lengths Chamfer, Variable Four Lengths Chamfer, Variable Lengths Angle Chamfer and Variable By Position Chamfer. These capabilities are indicated in the Message Line by the notation, [Ctrl = Advanced] (Windows) or [Option =

Advanced] (Macintosh). When you press the CTRL (Windows) or OPTION (Macintosh) key, the Advanced Chamfer Options dialog box appears.

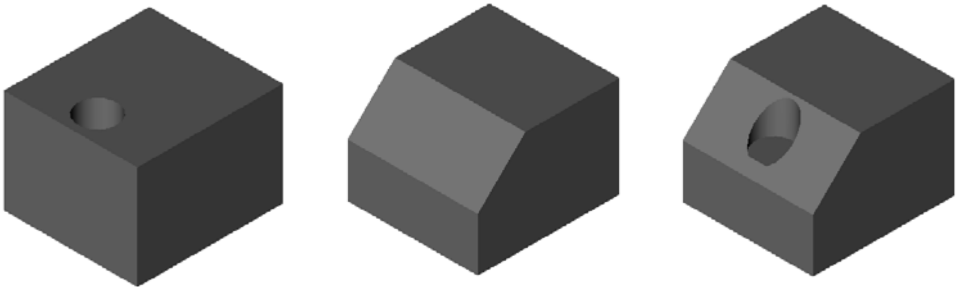


The dialog box contains the following options:

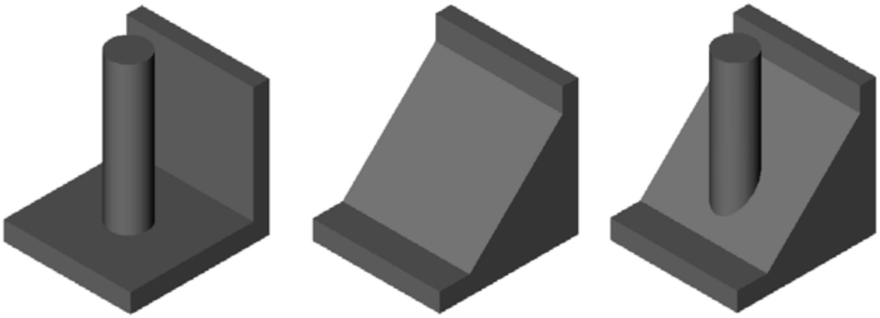
Automatic Chain-select of Shared Edges Checking this box enables you to chamfer all edges that share a tangent with the selected edges. This is the default setting. Without this checked only the selected edges chamfer. The left graphic shows three edges chamfered with the Constant Length Chamfer tool with the option selected. The right graphic shows two edges chamfered with the Constant Length Chamfer tool without the option selected.



Feature Interaction (Slower) Checking this box allows you to apply a chamfer that intersects a cutout or protrusion. Without this checked the feature may be deleted. With it checked the cutout is taken into account. The graphics illustrate this. The left graphic is the original part with hole cutout to half the part’s depth. The middle graphic is the chamfered part without this option selected. The right graphic is the chamfered part with the option selected.

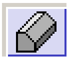


Without this checked for a protrusion the chamfer takes precedence over the protrusion. The left graphic below is the original part. The middle graphic is the part without the option selected. The right graphic is the part with the option selected.

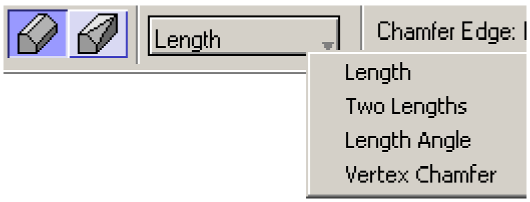


Using this option results in a longer chamfering time.

Constant Length Chamfer Tool



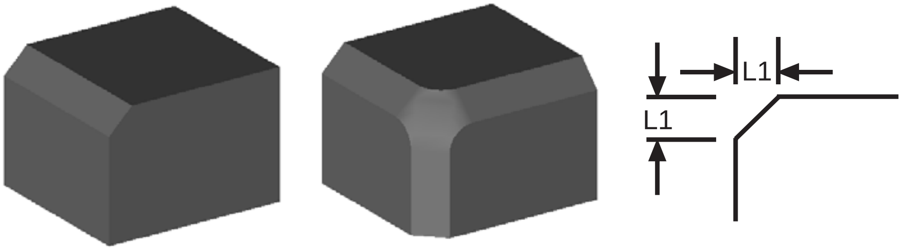
When you select this tool its option pull-down menu appears.



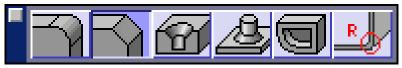
You have four constant length options: Length, Two Lengths, Length Angle and Vertex Chamfer.

Using the Length Option

This option adds a 45° chamfer based on the specified length to edge distance along the selected edge. The left graphic is an example of an object using this option. The middle graphic here shows a .5 inch chamfer with a vertex setback of 1.0. (You create this by selecting three or more edges with a setback at the corner vertex.) The right graphic illustrates the variables.



1. Select the Chamfer Edge tool.



2. In the Message Line, select the Constant Length Chamfer tool and the Length option from the pull-down menu. The Message Line reads: *Chamfer Edge: Pick solid edges to chamfer [Ctrl = Advanced (Windows) or Option = Advanced (Macintosh)] [Shift=Extend]*.
3. Enter a setback value in the Setback data field of the Status Line, if desired, for three or more intersecting edges.
4. Select the edge(s) to chamfer.

   The chamfer is created.

While the chamfer is still selected you can change L1 and the setback length in the Status Line. Type the desired values in the data field and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains L1 and the Setback.

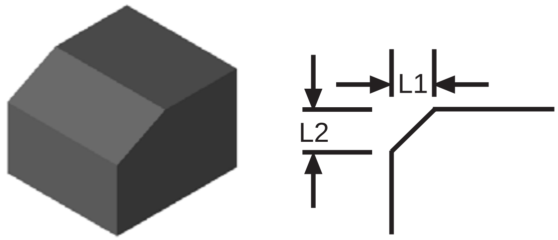
Geometric Characteristics

A constant length chamfer is created by choosing a solid edge and specifying the length to the edge. It is made up of two characteristic according to Edit Objects: Setback and Length. The setback is the distance from the shared vertex of three or more edges that the chamfer extends.

It also includes an edge list displaying the edge name and its associate length. To modify the chamfer characteristics, select the chamfer using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the chamfer name. See Chapter 24 for more information on the Design Explorer.

Using the Two Lengths Option

Using this option adds a chamfer based on the specified length to edge distances along the selected edge to create a constant length chamfer.



1. Select the Chamfer Edge tool.
2. In the Message Line, select the Constant Length Chamfer tool and the Two Lengths option from the pull-down menu. The Message Line reads: *Chamfer Edge: Pick solid edges to chamfer [Ctrl = Advanced (Windows) or Option = Advanced (Macintosh)] [Shift=Extend]*.
3. Enter L1 and L2 values in the Status Line.
4. Select the edge(s) to chamfer.

   The chamfer is created.

While the chamfer is still selected you can change the L1 and L2 values in the Status Line. Type the desired values in the data field and press ENTER (Windows) or RETURN (Macintosh).

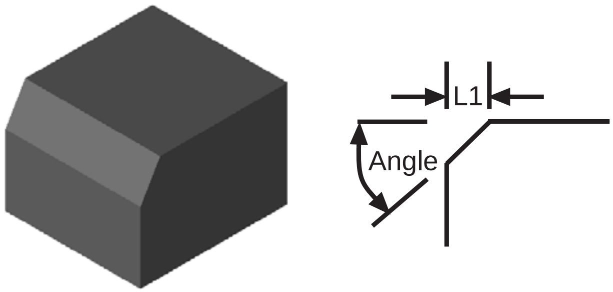
The Status Line contains L1 and L2.

Geometric Characteristics

A constant two length chamfer is created by choosing a solid edge and specifying the lengths to the edge. It is made up of two characteristics according to Edit Objects: Length 1 and Length 2. To modify the chamfer characteristics, select the chamfer using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the chamfer name. See Chapter 24 for more information on the Design Explorer.

Using the Length Angle Option

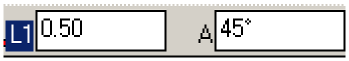
This option adds a chamfer at an angle and a constant length to the edge that you specify in the Status Line. The angle is measured from the tangent of the selected edge to the chamfered edge (see the right graphic).



1. Select the Chamfer Edge tool.
2. In the Message Line, select the Constant Length Chamfer tool and the Length Angle option from the pull-down menu. The Message Line reads: *Chamfer Angle Edge: Pick solid edges to chamfer [Ctrl = Advanced (Windows) or Option = Advanced (Macintosh)] [Shift=Extend]*.
3. Enter an L1 and angle in the Status Line data fields.
4. Select the edge(s) to chamfer.

While the chamfer is still selected you can change the length and angle in the Status Line. Type the desired values in the data fields and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains L1 and the A chamfer angle) data fields.



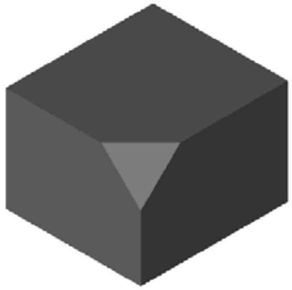
Geometric Characteristics

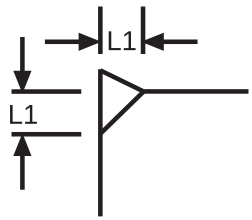
An constant length angle chamfer is created by choosing a solid edge and specifying an angle and length. It is made up of these characteristics according to Edit Objects: Setback and Angle. The setback is the distance from the shared vertex of three or more edges that the chamfer extends.

To modify the chamfer characteristics, select the chamfer using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the chamfer name. See Chapter 24 for more information on the Design Explorer.

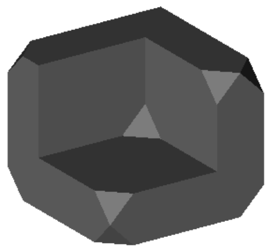
Using the Vertex Chamfer Option

This option adds a constant vertex chamfer to the intersection of three edges. The length is measured from the vertex to the edge as indicated in the right graphic below.





The faces next to the intersecting edges must be planar and have the same convexity or concavity. The graphic here shows convex and concave vertex chamfers.



1. Select the Chamfer Edge tool.
2. In the Message Line select the Constant Length Chamfer tool and the Vertex Chamfer option from the pull-down menu. The Message Line reads: *Vertex Corner: Select vertex to corner*.
3. Enter an L1 value in the Status Line data field.
4. Select the vertex to chamfer.

While the chamfer is still selected you can change the length in the Status Line. Type the desired value in the data field and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains L1 chamfer length.

Geometric Characteristics

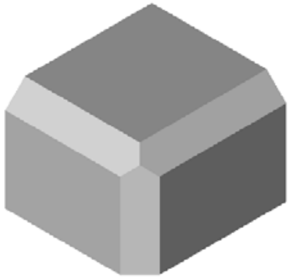
An vertex corner chamfer is created by choosing a vertex and specifying a length. It is made up of one characteristic according to Edit Objects: Radius 1. To modify the chamfer radius select the chamfer using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the chamfer name. See Chapter 24 for more information on the Design Explorer.

Additional Examples

Here are a few chamfer variations using the constant chamfer tools.

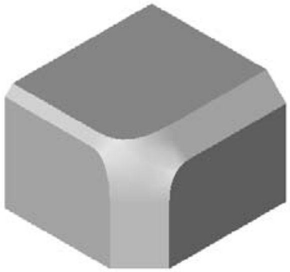
Multi-edge Chamfer with Mitered Corner

This chamfer is created using the Length option of the Constant Length Chamfer tool. In the Advanced Chamfer options dialog box turn off the Automatic Chain-select of Shared Edges option and then chamfer each edge separately.

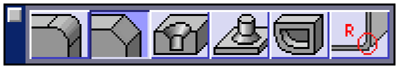


Multi-edge Chamfer with Varied Lengths and Setback

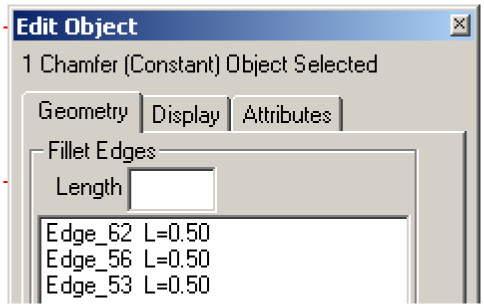
This chamfer is created using the Length option of the Constant Length Chamfer tool.



1. Select the Chamfer Edge tool.



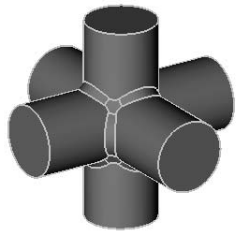
2. In the Message Line, select the Constant Length Chamfer tool and the Length option from the pull-down menu. The Message Line reads: *Chamfer Edge: Pick solid edges to chamfer [Ctrl = Advanced (Windows) or Option = Advanced (Macintosh)] [Shift=Extend]*.
3. Enter a setback value in the Setback data field.
4. Hold down the SHIFT key and select the edges to chamfer them.
5. Using the Design Explorer, double-click on the chamfer to display the Edit Object dialog box.



6. In the edge list select an edge. It’s associated L1 value appears in the length data field.
7. Enter a new value in the field.

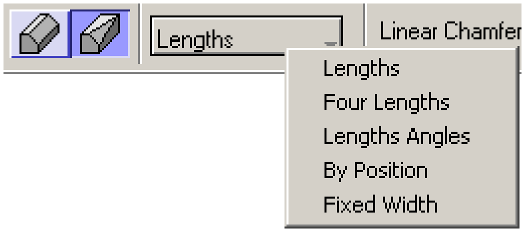
   Change other values as desired.
8. Click apply and the chamfer updates.

Complex Chamfer



This Designer Elements program supports complex chamfering for intersecting objects with or without setbacks. The example here includes setbacks.

Variable Chamfer Edge Tool

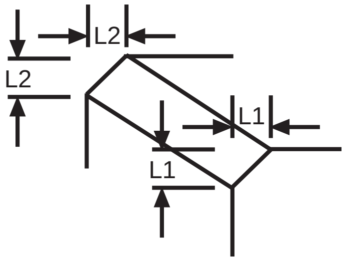


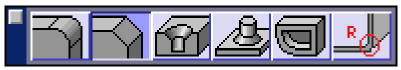
When you select this tool its option pull-down menu appears.

You have five variable length options: Lengths, Four Lengths, Lengths Angles, By Position and Fixed Width.

Using the Lengths Option

This option adds a chamfer tapered linearly along the edge from two specified setback lengths, L1 and L2.

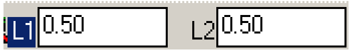




1. Select the Chamfer Edge tool.
2. In the Message Line, select the Variable Length Chamfer tool and the Lengths option from the pull-down menu. The Message Line reads: *Linear Chamfer: Pick edge to linearly chamfer. (Pick edge near L1) [Ctrl = Advanced (Windows) or Option = Advanced (Macintosh)]*.
3. Enter the L1 (the beginning setback length of the chamfer) and L2 (the ending setback length of the chamfer) values in the Status Line data fields.
4. Select the edge to chamfer.

While the chamfer is still selected you can change the setback lengths (L1 and L2) in the Status Line. Type the desired values in the data fields and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the chamfer setback lengths, L1 and L2.



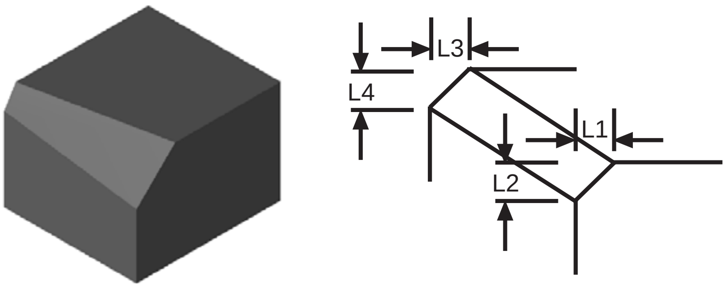
Geometric Characteristics

A variable lengths or linear chamfer is created by choosing a solid edge and specifying a length for the beginning and end of the chamfer. It is made up of these characteristics according to Edit Objects: L1, L2 and a list of the edges chamfered with their corresponding lengths.

To modify the chamfer characteristics, select the chamfer using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the chamfer name. See Chapter 24 for more information on the Design Explorer.

Using the Four Lengths Option

This option adds a chamfer tapered linearly along the edge using four specified setback lengths, L1, L2, L3 and L4.



1. Select the Chamfer Edge tool.
2. In the Message Line select the Variable Length Chamfer tool and the Four Lengths option from the pull-down menu. In the Message Line select the Linear Chamfer tool. The Message Line reads: *Chamfer Edge: Pick solid edges to chamfer. [Ctrl = Advanced (Windows) or Option = Advanced (Macintosh)] [Shift=Extend]*.
3. Enter the L1 and L2 values for the beginning setback lengths of the chamfer and L3 and L4 values for the ending setback lengths of the chamfer in the Status Line data fields.
4. Select the edge(s) to chamfer.

While the chamfer is still selected you can change the setback lengths in the Status Line. Type the desired values in the data fields and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the chamfer setback lengths, L1, L2, L3 and L4.

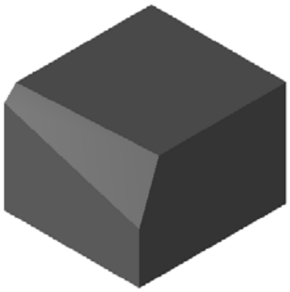
Geometric Characteristics

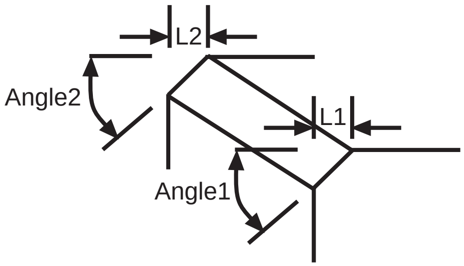
A variable four lengths chamfer is created by choosing a solid edge and specifying a lengths for the beginning and end of the chamfer. It is made up of these characteristics according to Edit Objects: L1, L2, L3, L4 and a list of the edges chamfered with their corresponding lengths.

To modify the chamfer characteristics select the chamfer using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the chamfer name. See Chapter 24 for more information on the Design Explorer.

Using the Variable Lengths Angles Option

This option adds a chamfer with the specified lengths and angles for the beginning and end of the chamfer along the selected edge. The angles are measured from the tangent of the selected edge to the chamfered edge (see the right graphic).





1. Select the Chamfer Edge tool.
2. In the Message Line, select the Variable Length Chamfer tool and the Lengths Angles option from the pull-down menu. The Message Line reads: *Chamfer Angle Edge: Pick solid edges to chamfer [Ctrl = Advanced (Windows) or Option = Advanced (Macintosh)] [Shift=Extend].*
3. Enter an L1 and A1 (angle) for the beginning of the chamfer and L2 and A2 (angle) for the end of the chamfer in the Status Line data fields.

4. Select the edge(s) to chamfer.

While the chamfer is still selected, you can change the lengths and angles in the Status Line. Type the desired values in the data fields and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains L1, A1, L2 and A2 data fields.

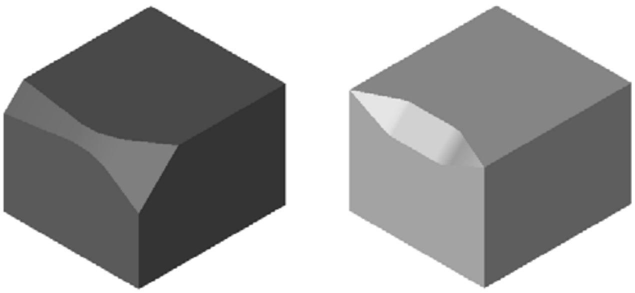
Geometric Characteristics

A variable length angle chamfer is created by choosing a solid edge and specifying the beginning and end chamfer angles and lengths. It is made up of these characteristics according to Edit Objects: L1, Angle 1, L2, Angle 2 and a list of the edges chamfered with their corresponding lengths and angles.

To modify the chamfer characteristics, select the chamfer using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the chamfer name. See Chapter 24 for more information on the Design Explorer.

Using the Variable By Position Option

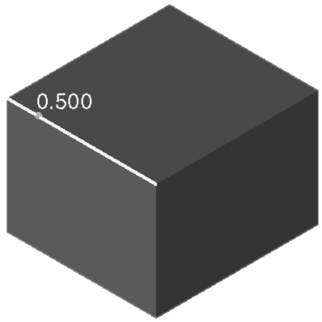
This option adds a chamfer with a varied lengths along the edges you select. You specify the lengths in the Status Line. You can also create a degenerative chamfer with this option by specifying a zero length at both ends of the edge. The left graphic shows a varied length chamfer. The right graphic shows a degenerative chamfer.



1. Select the Chamfer Edge tool.
2. In the Message Line, select the Variable Length Chamfer tool and the By Position option from the pull-down menu. The Message Line reads: *Position Chamfer: Pick edges for variable position chamfer [End = ESC, Dbl Click] [Ctrl = Advanced (Windows) or Option = Advanced (Macintosh)] [Shift=Extend]*.

3. Select the edge along which to make the chamfer. The Message Line reads: *Position Chamfer: Specify location for length value [End=ESC, Dbl Click]*.
4. Enter a setback length in the Length field of the Status Line.
5. Specify the location(s) along the selected edge for the chamfer.

   As you specify the location the length appears in red at that point and all additional locations. The values remain until you double-click to specify the last chamfer point and the chamfer is created. The graphic illustrates this.



6. Enter a new length in the Length field.
7. Specify the location for this length along the edge.

   Continue following this procedure until you have entered all the desired lengths. Double-click the last place chamfer point and the variable by position chamfer is created.

The Status Line contains the Length for the chamfer setback.

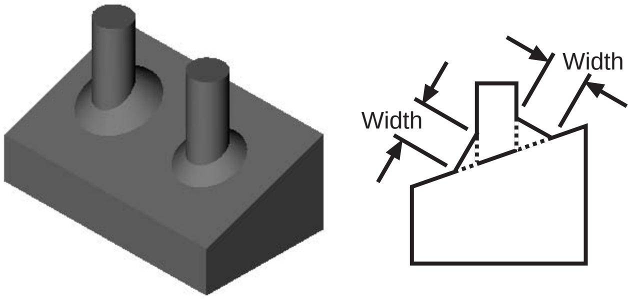
Geometric Characteristics

A variable by position chamfer is created by choosing a solid edge and various setback lengths at different locations along the edge for the chamfer. It is made up of these characteristics according to Edit Objects: a list of the edges chamfered with their widths and Param points, Width (which corresponds to the Status Line length) and Param value. Param is the percentage point location along the edge for the selected item in the list. When you select one of the items in the list its Length displays in the Width data field and its percentage point location displays in the Param field.

To modify the chamfer characteristics select the chamfer using the Design Explorer and choose ***Window>Edit Objects***. See Chapter 24 for more information on the Design Explorer.

Using the Fixed Width Option

This option adds a chamfer with a fixed width along the edges you select adjusting the chamfer length to maintain the width. This is important when placing a chamfer along of the edge of two faces where the angle changes. You specify the width in the Status Line. The left graphic shows a sloped block with two bosses where the left boss has at constant length chamfer and the right boss has a variable fixed width chamfer. The right graphic illustrates the fixed width characteristic.



1. Select the Chamfer Edge tool.
2. In the Message Line select the Variable Length Chamfer tool and the Fixed Width option from the pull-down menu. The Message Line reads: *Fixed Width Chamfer: Pick edges AND/OR faces to chamfer [Ctrl = Advanced (Windows) or Option = Advanced (Macintosh)] [Shift=Extend]*.
3. Enter a width in the W data field of the Status Line.
4. Pick the edge(s) for the chamfer.

While the chamfer is still selected you can change the width in the Status Line. Type the desired values in the data fields and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the width for the chamfer.

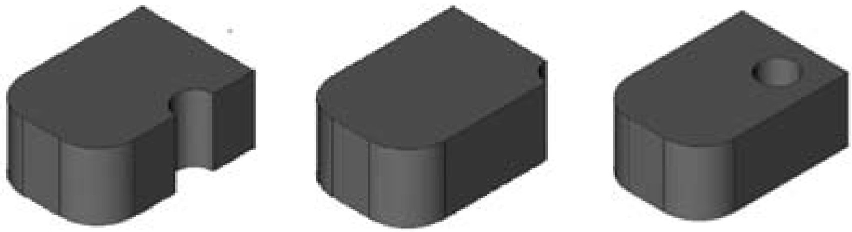
Geometric Characteristics

A variable fixed width chamfer is created by setting a chamfer width and selecting the edge for the chamfer. It is made up of these characteristics according to Edit Objects: Width and a list of the edges chamfered with their width. When you select one of the items in the list its width displays in the Width data field.

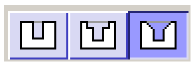
To modify the chamfer characteristics select the chamfer using the Design Explorer and choose ***Window>Edit Objects***. See Chapter 24 for more information on the Design Explorer.

Hole Feature Tools

With the Hole Feature tools you can model bolt holes found frequently in mechanical design parts. This Designer Elements program holes are predefined features that remove cylindrical shapes from objects. Holes can be placed through models, along face edges at corner vertices and on planar and non-planar faces. The graphic below is an example of a simple hole.



Holes, like other features in this Designer Elements program, are associative. You can modify them at any point in your design process. When you select the Hole Feature tools a subpalette appears containing three tools for creating holes: Simple, Counter Bore and Counter Sink.

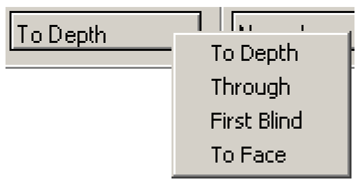


Tech Note:

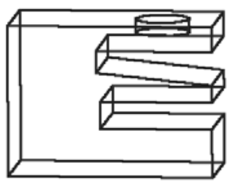
If you want to place a hole through an object that will intersect a blend, use the *Depth* option rather than the *To Face* option. If *To Face* is used, when the hole intersects the blend, a face change occurs and the hole ends.

Through Types

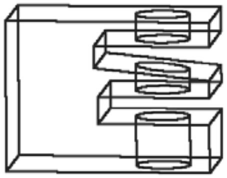
The Message Line also contains a pull-down menu allowing you to choose a Through Type for your hole. You have four options: Depth, Through, First Blind and To Face.



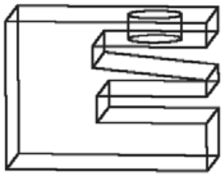
Depth Extends the depth of the hole to the value specified in the Status Line.



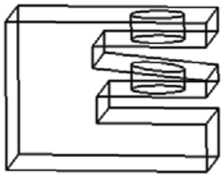
Through Extends a hole through the selected solid.



First Blind Extends a hole to the first open face.

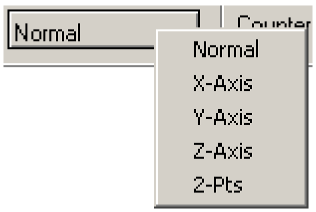


To Face Extends a hole to the selected face.



Hole Direction

The Message Line contains a pull-down menu allowing you to specify a boss direction. You have five options: Normal, X-Axis, Y-Axis, Z-Axis and 2-Pts.



Normal Creates a hole perpendicular to the solid face.

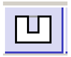
X-Axis Creates a hole along the X-axis.

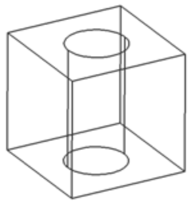
Y-Axis Creates a hole along the Y-axis.

Z-Axis Creates a hole along the Z-axis.

2-Pts Creates a hole in the direction specified by two user defined points.

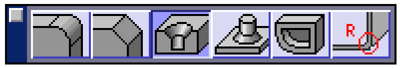
Simple Hole Tool





This tool creates straight, cylindrical holes through the selected solid. You can specify the X, Y and Z center location, depth, diameter and draft angle of the hole in the Status Line.

Using the Simple Hole Tool



1. Select the Hole tool.
2. In the Message Line select the Simple Hole tool. The Message Line reads: *Simple Hole: Pick solid face to place hole*.
3. Select a Through Type and Hole Direction from their respective pull-down menus in the Message line.
4. Select the face where you want to locate the hole.
5. Select the location for the center of the hole.
6. If you selected the 2-Pts direction click two points to specify the direction.

   The hole is created.

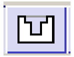
While the solid is still selected you can change the hole depth in the Status Line. Type the desired value in the data field and press ENTER (Windows) or RETURN (Macintosh).

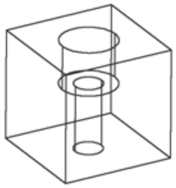
The Status Line contains the hole's X, Y and Z center location, Diameter and Draft Angle.

Geometric Characteristics

A simple hole is created by choosing the solid face and the location of the hole. It is made up of these characteristics according to Edit Objects: Hole Type, Thru Type, Hole Depth, Diameter and Draft Angle. To modify the hole characteristics, select the hole using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the hole name. See Chapter 24 for more information on the Design Explorer.

Counter Bore Tool





This tool creates a counter bore hole based on the values you specify in the Status Line. A counter bore consists of the bore and the hole. The bore is a straight-sided cylinder with a diameter larger than the hole. The hole extends from the end of the bore and completes the counter bore. See the graphic here.

Using the Counter Bore Hole Tool

1. Select the Hole tool.
2. In the Message Line select the Counter Bore Hole tool. The Message Line reads: *Counter Bore: Pick solid face to place counterbore*.
3. Select a Through Type and Hole Direction from their respective pull-down menus in the Message Line.
4. Select the face to place the counter bore.
5. Specify the location for the center of the counterbore hole.
6. If you selected the 2-Pts direction click two points to specify the direction.

   The bore is created.

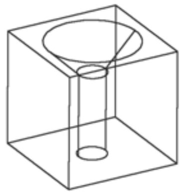
While the solid is still selected you can change the depth, diameter and draft angle of the hole and the bore depth and radius in the Status Line. Type the desired values in the data fields and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the counter bore's Depth, Diameter, Draft Angle, Bore Depth and Bore Diameter.

Geometric Characteristics

A counter bore is created by choosing the solid face and the location of the hole. It is made up of these characteristics according to Edit Objects: Hole Type, Thru Type, Hole Depth, Diameter, Draft Angle, Counter Bore Diameter and Depth. To modify the hole characteristics, select the hole using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the hole name. See Chapter 24 for more information on the Design Explorer.

Counter Sink Tool



This tool creates a hole with angled sides, called the sink and a hole that extends from the sink. A counter sink hole can be contrasted with a counter bore hole where the bore has straight sides.

Using the Counter Sink Hole Tool

1. Select the Hole tool.
2. In the Message Line select the Counter Sink Hole tool. The Message Line reads: *Counter Sink Hole: Pick solid face to place countersink.*
3. Select a Through Type and Hole Direction from their respective pull-down menus in the Message line.
4. Select the face where you want to locate the hole.
5. Select the location for the center of the hole.
6. If you selected the 2-Pts direction click two points to specify the direction.

   The hole is created.

While the solid is still selected you can change the depth, diameter and draft angle, sink radius and sink angle in the Status Line. Type the desired values in the data fields and press ENTER (Windows) or RETURN (Macintosh).

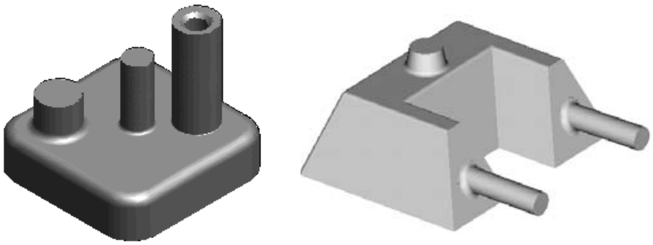
The Status Line contains the countersink's Depth, Diameter, Draft Angle, Sink Diameter and Sink Angle.

Geometric Characteristics

A counter sink hole is created by choosing the solid face and the location of the hole. It is made up of these characteristics according to Edit Objects: Hole Type, Thru Type, Hole Depth, Diameter, Draft Angle, Counter Sink Diameter and Angle. To modify the hole characteristics, select the hole using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the hole name. See Chapter 24 for more information on the Design Explorer.

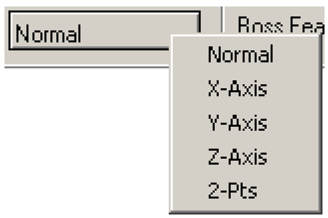
Boss Feature Tool

This tool creates a boss by adding a cylinder of a specified size to another solid and filleting the intersection of the two. A boss can be placed on planar and non-planar surfaces. You specify the height, diameter and fillet radius in the Status Line.



Boss Direction

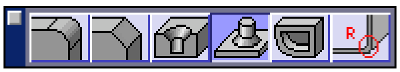
The Message Line contains a pull-down menu allowing you to specify a boss direction. You have five options: Normal, X-Axis, Y-Axis, Z-Axis and 2-Pts.



Normal	Creates a boss perpendicular to the solid face.
X-Axis	Creates a boss along the X-axis.
Y-Axis	Creates a boss along the Y-axis.
Z-Axis	Creates a boss along the Z-axis.
2-Pts	Creates a boss in the direction specified by two user defined points.

Using the Boss Feature Tool

1. Select the Boss Feature tool. The Message Line reads: *Boss Feature: Pick solid face to place boss*.



2. Select the boss direction from the pull-down menu.

3. Select the face to locate the boss.
4. Specify a location for the boss center.
5. If you selected the 2-Pts direction click two points to specify the direction.

   The boss is created.

While the solid is still selected you can change the height, diameter and fillet radius of the boss in the Status Line. Type the desired values in the data fields and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the boss' Height, Radius and Fillet Radius.

If the solid face selected in incompatible with the axis direction the following ACIS Error appears.

Geometric Characteristics

A boss is created by choosing the solid face and specifying the values for the cylinder. It is made up of these characteristics according to Edit Objects: Height, Diameter and Fillet Radius. To modify the boss characteristics, select the boss using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the boss name. See Chapter 24 for more information on the Design Explorer.

Feature Editing

Cobalt and Xenon Only

One of this Designer Elements program' more powerful features is the transparent parent/child associativity it creates for solid objects. As you add features to your part this Designer Elements program automatically establishes relationships between them. When you modify or change a parent, this Designer Elements program updates and regenerates all the children. For example, consider the following procedure:

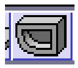
1. Create a block primitive.
2. Fillet the block's edges.
3. Place a boss at the block's center.

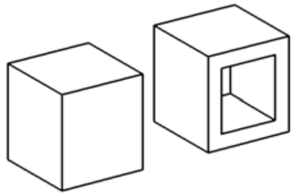
4. Place a countersink hole at the boss' top center.

Now try these modifications:

5. Select a face or edge from the original block. Choose ***Window>Edit Objects*** and change the block's length, width, and height. The fillets, boss and countersink hole automatically regenerate based on the new length, width, and height.
6. Select the block. Choose ***Window>Design Explorer***. Click on the + sign to display the history tree of the block. Double-click on the fillet name in the Design Explorer. The Edit Objects dialog box appears. Change the fillet's radius. Note that the boss and hole automatically regenerate.
7. In the Design Explorer select one of the boss' face names. The Edit Object dialog box now displays the boss information. Change the boss to 100000 (or any outrageous value). In this case this Designer Elements program cannot regenerate the solid and returns it to the previously valid state.

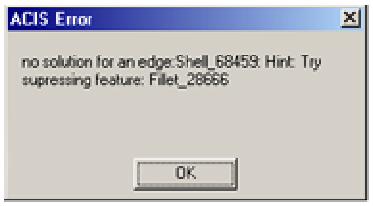
Shell Solid Tool

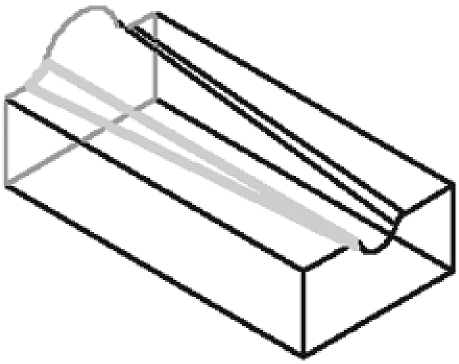




The Shell Solid tool hollows out a solid with a thickness you set.

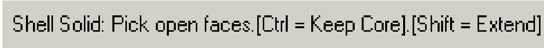
This tool also includes an error detection routine. If your model contains a problem that prevents the completion of the local face modeling operation, an ACIS error dialog box appears with the problem location highlighted.





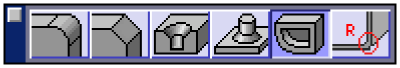
After shelling you can edit the face thickness in the Edit Objects dialog box.

This tool also gives you the option of keeping the core of the shelled object. Notice the Messages Line:



Using the Shell Solid Tool

1. Select the Shell Solid tool. The Message Line reads: *Shell Solid: Pick solid to shell*.



2. In the Status Line, enter the desired offset or thickness of the shell.
3. Select the solid. The Message Line reads: *Shell Solid: Pick open faces [Shift=Extend]*.
4. Select the face or faces that will be open. The object shells.

   If you click in an area with no faces, the object will be hollowed out with no open faces.

Tech Note:

Like all tools, you can set the thickness of the shell after the operation using the Status Line. When working with small objects, you may need to change the default thickness first to shell successfully.

While the solid is still selected you can enter a new value in the offset field. Press ENTER (Windows) or RETURN (Macintosh) to accept the change.

The Status Line contains the offset or shell thickness.

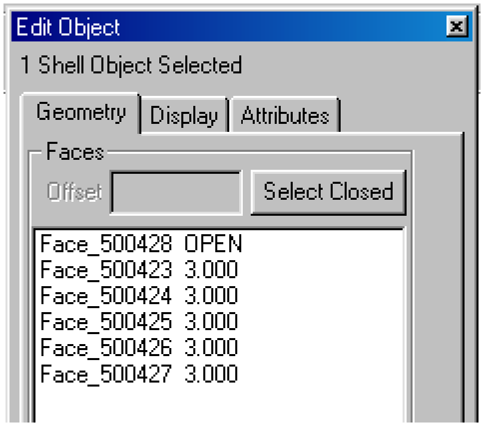
Modifying a Shelled Solid

After you shell a solid you can modify the shelling operation through the Edit Objects dialog box.

1. Select the shelled solid.
2. Choose ***Window>Design Explorer***.
3. Open the history tree for the solid by clicking on the + (plus sign).
4. Select the shell operation.

5. Choose ***Window>Edit Objects***.

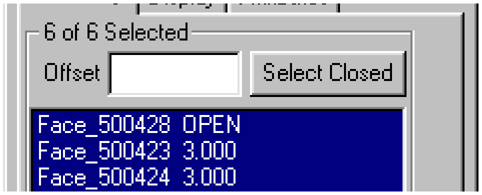
   Edit Objects includes the Offset data field, Face list and a Selection button which toggles between three operations, Select Open, Select Closed and Select All.



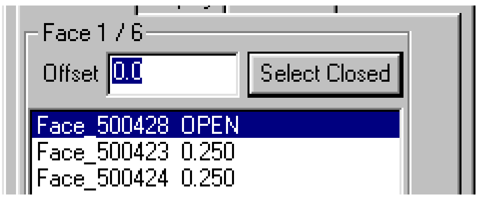
Offset — This field displays the offset when a shell face is selected from the face list.

Face list — This list specifies the faces contained in the solid, a Designer Elements identifier number and the offset for the face. If the face was removed in place of an offset value appears the word open. If you suppress a face, suppressed is displayed instead of an offset value.

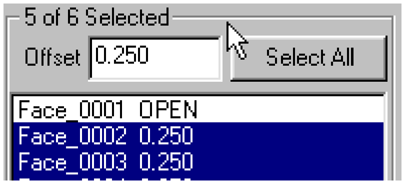
Select Open — This selection operation displays when all faces are selected. Click this button to select all open faces.



Select Closed — This selection operation displays when an open face or no faces are selected. Click this button to select all closed faces.



Select All — This selection operation displays when all closed faces are selected. Click this button to select all faces.



If you select one or more faces manually the buttons will not toggle automatically. You can also change the more than one face by holding down the SHIFT key when selecting or dragging to select multiple faces.

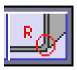
6. Select the desired face. (As you move through the list of faces, the selected face is highlighted on your object.)
7. Make any desired modifications.

   You can suppress a face or change the offset for each face to another value. An offset of 0 removes the face.
8. Click Apply to accept the changes.

Geometric Characteristics

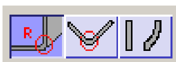
A shelled solid is created by choosing a solid, specifying an offset and the face(s) you want open. It is made up of these characteristics according to Edit Objects: Offset thickness, Suppress Face check box and a list of the solid faces with the respective offset distance. This list is only available if you select the operation in the Design Explorer first and then display the Edit Object dialog box. See Chapter 24 for more information on the Design Explorer.

Bend Solid Tool



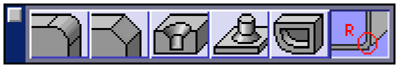
The Bend Solid tool creates a solid by bending a solid about an axis through a given radius.

There are three options in the Message Line; Bend, Center Bend and Bend Along Curve.

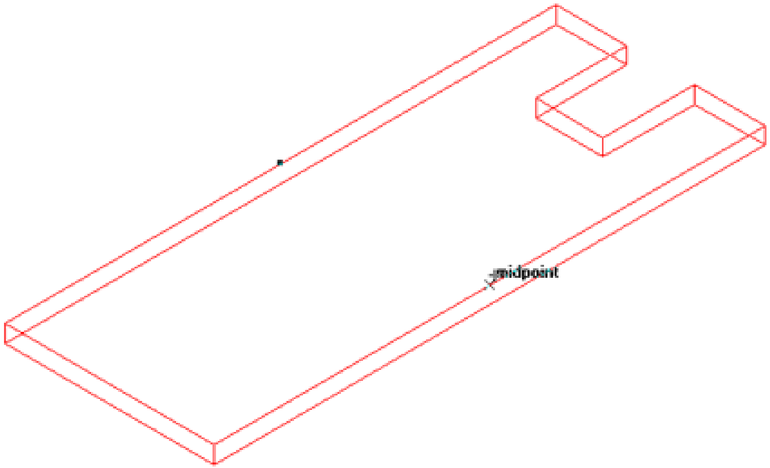


Using the Bend Axis Option

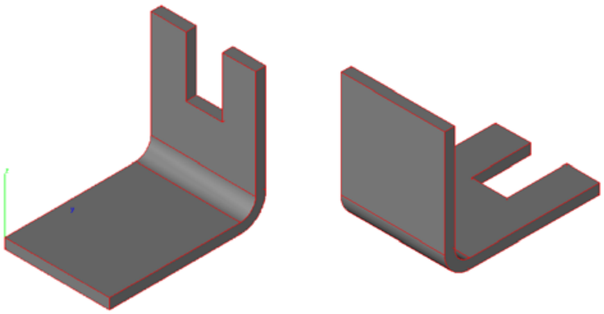
1. Click the Bend Solid tool. Select the first tool in the Message Line. The Message Line reads: *Bend Solid: Pick Solid to Bend*.



2. The Status Line contains the Bend Radius and Bend Angle fields. Enter the desired values for your bend solid. Tab between data fields.
3. Pick the solid that you wish to bend
4. The message line now reads, *Bend Solid: Enter two points for bend axis*. Specify two points for the bend axis. The two points represent the inner tangent line for the bend solid. After you click the two points, the solid is bent about the axis.



5. The Message Line now reads, *Bend Solid: Pick Solid to Bend [Option = Flip Material]*. Hit the option key to flip the side of the material that was bent.



While the object is still selected, you can edit the bend radius and bend angle. Enter the new values and press ENTER (Windows) or RETURN (Macintosh) and the solid updates

Using the Center Bend Option

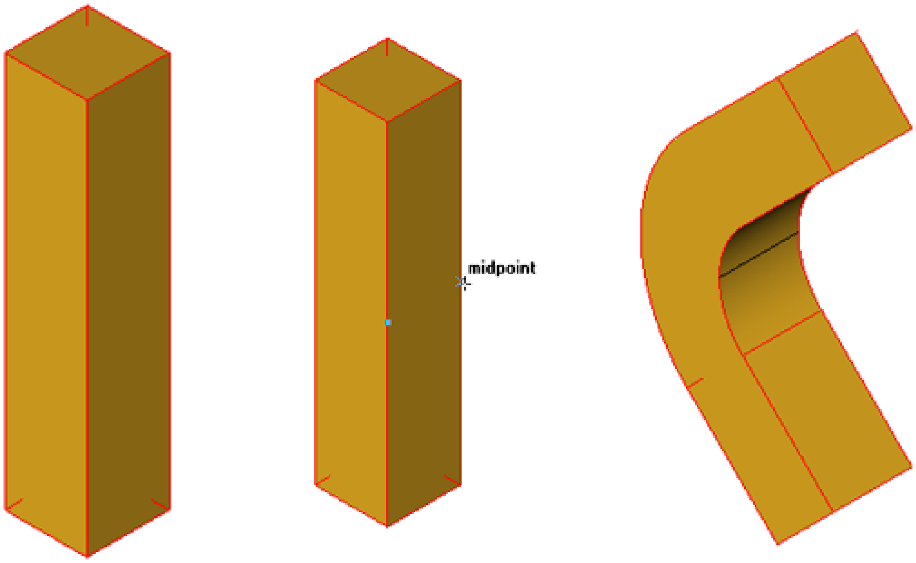
1. Click the Bend Solid tool. Select the first tool in the Message Line. The Message Line reads: *Center Bend Solid: Pick Solid to Bend.*

2. The Status Line contains the Bend Radius and Bend Angle fields. Enter the desired values for your bend solid. Tab between data fields.

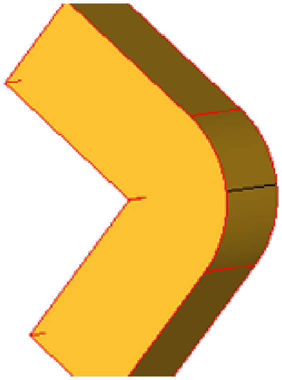
3. Pick the solid that you wish to bend

4. The message line now reads, *Bend Solid: Enter two points for bend axis.* Specify two points for the bend axis. The two points represent the inner tangent line for the bend. After you click the two points, the solid is bent about the axis.

5. The Message Line now reads, *Center Bend Solid: Pick Solid to Bend [Ctrl or Option = Flip Material]*. Hit the option key to flip the side of the material that was bent.



Solid to Bend Bend Axis Resulting Bend

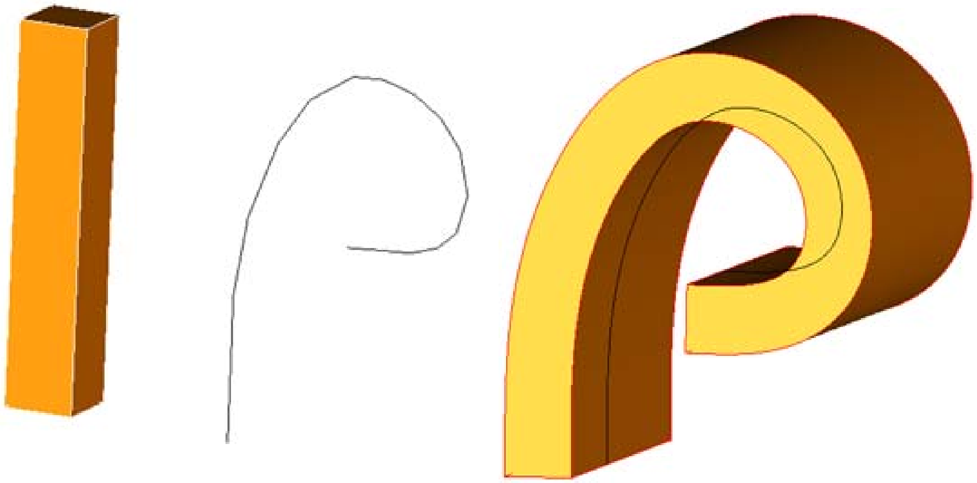


Flipped Material

Using the Bend Along Curve Option

1. Click the Bend Solid tool. Select the first tool in the Message Line. The Message Line reads: *Bend Solid Along Path: Pick Solid to Bend*..
2. Pick the solid that you wish to bend
3. The message line now reads, *Bend Solid Along Path: Pick curve to bend solid to.* Choose your curve.
4. The Message Line now reads, *Bend Solid: Enter two points for bend height axis*. Specify two points for the bend axis. The two points represent thepoint at which the bend will start.

5. The Message Line now reads, *Bend Solid: Enter two points for alignment axis*. Choose the points on the solid you want aligned with the curve.



Solid Utilities and Features

After you've created a solid model you may need to modify it. This Designer Elements program provides several advanced editing tools to accomplish this. Use Boolean routines to add solids to each other, subtract them or create a new solid from their intersections. With the other tools you can split a solid in two, stitch several surfaces together to form a solid, hollow out a solid to create a shell, change a solid face's draft angle or collectively change the draft angles of a set of upper and lower surfaces. You can also evaluate drafts placed on solid objects using the Draft Evaluation feature.

This Designer Elements program also provides you with assembly modeling tools for connecting, mating, aligning and inserts solids.

For each tool you can enter values in the Status Line to perform the editing operation either before or after the operation is complete. If you enter the values after you select the tool but before you perform the operation, your first click in the drawing area automatically registers all Status Line values. If you enter values after performing the operation and while the solid is still selected, pressing ENTER (Windows) or RETURN (Macintosh) updates the solid to reflect the new values.

After using one of these tools to modify your solid you can make further changes to them by selecting the tool operation in the Design Explorer and choosing ***Window>Edit Objects***.

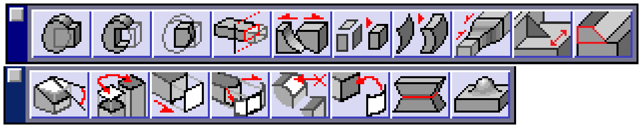
All editing operations conducted on a solid are associative. Any modifications made to the parent will automatically alter all child objects that reference it (**Cobalt™ and Xenon™ only**).

The topics covered in this chapter include:

- **Solid Editing** tools
- Draft Evaluation feature
- **Assembly Modeling** tools

Solid Utility Tools

This Designer Elements program provides you with advanced editing or utilities tools located in the last subpalette of the Solids tool palette. The tools include: Boolean, Split Solid, Stitched Solid, Shell Solid, Parting Line Solid, Local Face Modeling, Thicken Solid and Deform Face.



Boolean Tools

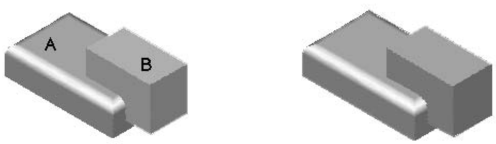
Your Designer Elements programs Boolean tools allow you to perform boolean operations on solid objects. You can add solids together, subtract one solid from another or intersect a solid with others.

Tech Note:

The add, subtract and intersect tool replace the two original solids with a new composite solid. The only way to recover the original solids after this boolean operation is to undo the command. A technique for retaining the originals is to copy them to another layer before using a Boolean tool.

Union Solid Tool

The Union Solid tool combines two or more solids into one. In the left graphic below Solid A and Solid B are individual solids. The right graphic is a single solid.



Using the Union Solid Tool

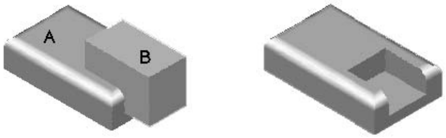
1. Select the Boolean tool.

2. In the Message Line select the Union Solid tool. The Message Line reads: *Union Solid: Pick solid to add to and solid(s) to add [Shift=Extend]*.
3. Select the solid to which you want to add.
4. Select the solid(s) that you want to add to the first solid. Hold down the SHIFT key before you select the first solid to select multiple solids.

There are no Status Line entries.

Subtract Solid Tool

The Subtract Solid tool subtracts one or more solids from another. In the right graphic Solid B is subtracted from Solid A.

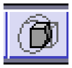


Using the Subtract Solid Tool

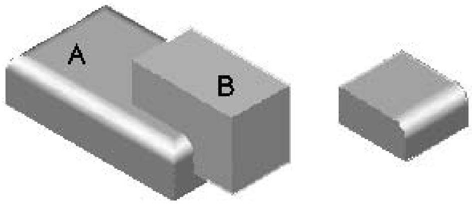
1. Select the Boolean tool.
2. In the Message Line select the Subtract Solid tool. The Message Line reads: *Subtract Solid: Pick solid to subtract from and solid(s) to subtract out [Shift=Extend]*.
3. Select the solid from which you want to subtract.
4. Select the solid(s) that you want to subtract from the first solid. Hold down the SHIFT key before you select the first solid to select multiple solids.

There are no Status Line entries.

Intersect Solid Tool



The Intersect Solid tool creates a new solid from the common volume of two intersecting solids. The right graphic below is the intersection of Solid A and Solid B.



Using the Intersect Solid Tool

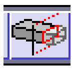
1. Select the Boolean tool.
2. In the Message Line select the Intersect Solid tool. The Message Line reads: *Intersect Solid: Pick two solids to intersect [Shift=Extend].*
3. Select the two solids to form the intersect solid.

There are no Status Line entries.

Geometric Characteristics

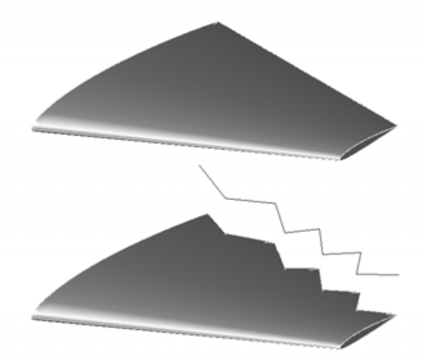
There are no characteristics listed in Edit Objects specific to a boolean solid.

Trim Solid Tool

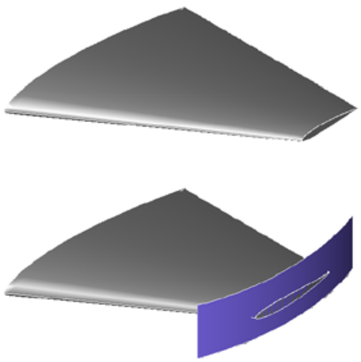


The Trim Solid tool trims a solid with a curve, surface, or solid. The trim tool is useful for removing material from a base solid. In addition to trimming to surfaces or another solid, the trim solid tool has the unique ability to allow trimming a solid to a curve. In the case of a curve, an internal surface is built that sweeps into the screen or sweeps by user defined vector. Use the pull down menu to specify your preference.

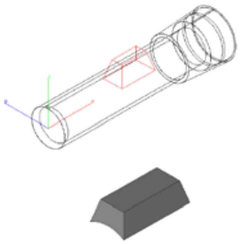
In the case of trimming a solid to a surface or other solid, the pull down option has no bearing on the results.



Trim Solid to Curve.



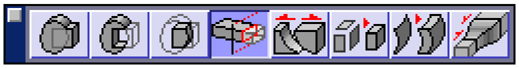
Trim Solid to Surface



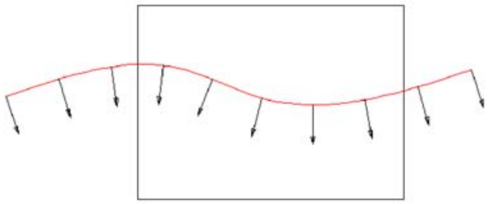
Trim Solid to Solid.

Using the Trim Solid Tool

1. Click the Trim Solid tool. The Message Line reads: *Trim Solid: Pick curve, surface or solid to trim to.*



2. The Message Line now reads, *Trim Solid: Pick solid to trim [Crtl (Windows) or Option (Macintosh) = Flip Direction to Remove]*



3. The solid is trimmed to the curve, removing the solid in the direction of the arrows.

4. Press the Control or Option key if you want to change the material that is kept.

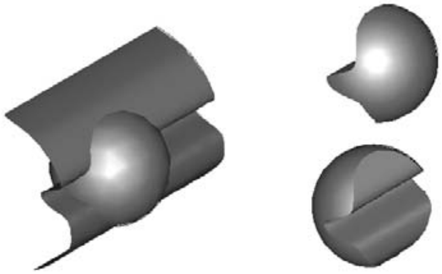
Note: Press the Control or Option key to change the direction of the material to be removed **after** the trim is completed. Do not hold the Control or Option keys during the trimming process.

When trimming a solid to another solid the portion that is kept is the portion that was selected by the user.

There is no Status Line for the trim solid tool.

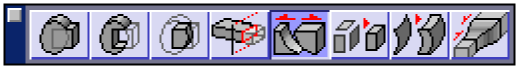
Split Solid Tool

The Split Solid tool divides a solid into two parts. This Designer Elements program uses an infinite plane, surface, or solid as the splitting entity so you must construct the splitting object before you can make the split.



Using the Split Solid Tool

1. Select the Split Solid tool. The Message Line reads: *Split Solid: Pick solid to split.*



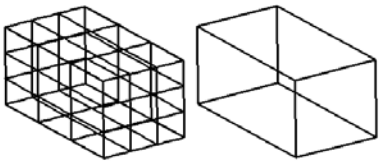
2. Select the solid. The Message Line reads: *Split Solid: Pick surface for splitting solid*.
3. Select the infinite plane or surface to split the solid.

There are no Status Line entries.

Geometric Characteristics

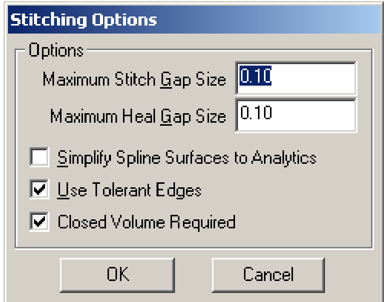
There are no characteristics listed in Edit Objects specific to a split solid.

Stitched Solid Tool



The Stitched Solid tool creates a closed solid from a collection of surfaces. The color and resolution of the first surface selected are used for the stitched solid.

The tool uses healing technology to repair any small gaps in the model. and creates a solid only if the selected surfaces create a closed body. This tool also provides you with three stitching options. Press the CTRL key (Windows) or OPTION key (Macintosh) to display the Stitching Options dialog box.



The dialog box contains the following options:

Maximum Heal Gap Size

This field set the maximum size that Vellum will attempt to stitch to objects. Enter a value between 0 and 1.

Example: If the gap size is .1 but the object has a gap of .2, the surfaces won't stitch.

Simplify Spline Surfaces to Analytics

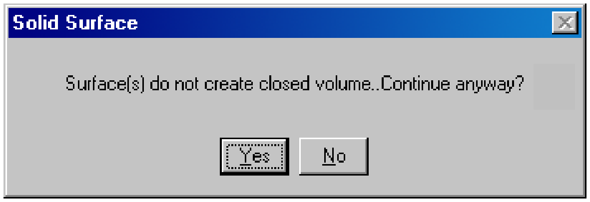
This option sets whether spline surfaces are simplified when they are stitched. Solids use surfaces as their underlying geometry. These surfaces can be either Analytic surfaces or B-Spline Nurb surfaces. Analytic surfaces are used to create cylinders, cones and spheres. B-Splines surfaces are used to create a variety of shapes. ACIS is “turbocharged” for analytic surfaces.

When selected, this Designer Elements program attempts to simplify your spline surfaces. If your surfaces create a standard shape such as a cylinder, they convert into analytic surfaces. This conversion speeds up the calculation process for the solid and for your file. By default, this option is not selected.

Use Tolerant Edges

This option determines whether tolerant edges will be used if a model can't stitch because of a gap. When selected and an edge can't stitch, it is marked as tolerant. Only operations that involve the entire edge can be conducted.

If this option is not selected and this Designer Elements program discovers a gap larger than specified in the Maximum Heal Gap Size data field the following dialog box appears.



Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when stitching solids.

Using the Stitched Solid Tool

1. Select the Stitched Solid tool. The Message Line reads: *Stitched Solid: Pick surfaces to create solid from*. *[Ctrl=Advanced]* (Windows) or *Option=Advanced* (Macintosh) *[Shift=Extend]*.
2. Press the CTRL key (Windows) or OPTION key (Macintosh) to display the Stitching Options dialog box.
3. Specify the settings and click OK. Click Cancel to close the dialog box.
4. Hold down the SHIFT key and select the surfaces or drag a selection fence around them.

   The surfaces are stitched into a solid.

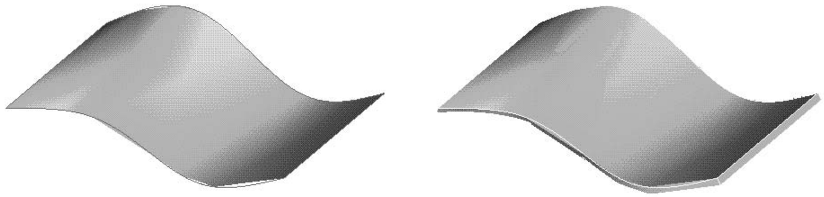
There are no Status Line entries.

Geometric Characteristics

There are no characteristics listed in Edit Objects specific to a stitched solid.

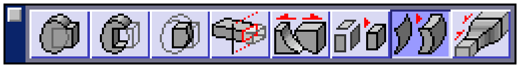
Thicken Solid Tool

The Thicken Solid tool allows you to thicken surfaces and solids. You specify the desired thickness. A positive or negative value determines the direction of the thickening. The left graphic shows a surface and the right graphic shows the thickened surface, now a solid.



Using the Thicken Solid Tool

1. Select the Thicken Solid tool. The Message Line reads: *Thicken Solid: Pick surface or solid to thicken*. *[Ctrl or Option =Flip thickness side]*.



2. Type the desired thickness in the Thickness data field in the Status Line.
3. Select the surface or solid object to be thickened. This Designer Elements program thickens the selected object.

By pressing the CONTROL or OPTION key you can toggle which side of the surface is thickened

While the object is still selected you can enter a new value in the Thickness data field and press ENTER (Windows) or RETURN (Macintosh) to change the thickness.

The Status Lines contains the Thickness value for the thickened object.

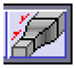
Geometric Characteristics

A thickened solid object is created by specifying a thickness and selecting the object. A thickened object is made up of the Thickness characteristic according to the Edit Objects dialog box. To modify the characteristics of the thickened object select the object using the Design Explorer (**Cobalt™ and Xenon™ only**) and choose ***Window>Edit Objects*** or double-click on the face name.

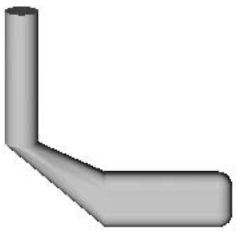
Display contains settings for Iso Lines and Silhouette. You can display Iso Lines by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on the object. You have three Silhouette settings, Smart, Off and On.

See Chapter 24 for more information on the Design Explorer.

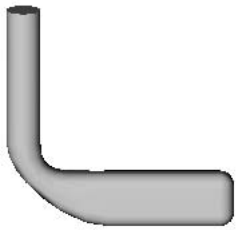
Lofted Solid Tool



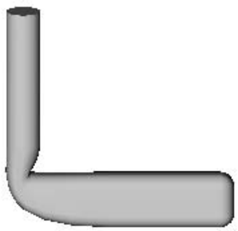
This tool creates a tangent solid between two neighboring solids using a specified bulge factor. The higher the bugle factor the more influential the slope of the tangent for the selected solid. A loft solid with a bulge factor of zero (0) creates a discontinuous loft. The graphics below illustrate the effects of the bulge factor.



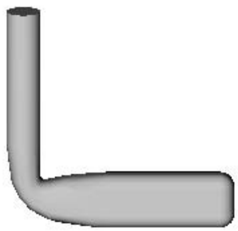
Bulge Factor = 0



Bulge Factor = 5



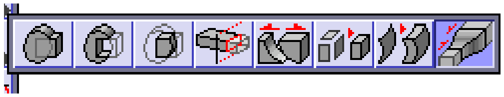
Bulge Factor 1 = 5; Bulge Factor 2 = 15
(Set in Edit Objects)



Bulge Factor 1 = 8; Bulge Factor 2 = 10
(Set in Edit Objects)

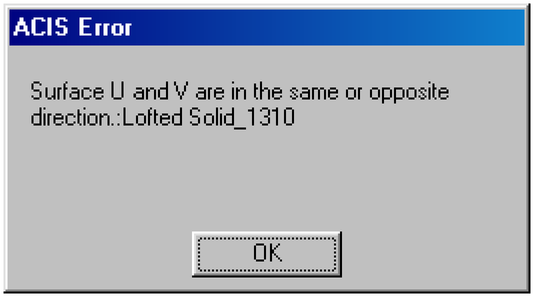
This tool does not support lofting if one of the solids is a sphere. If a sphere is selected no loft is created.

Using the Lofted Solid Tool



1. Select the Lofted Solid tool. The Message Line reads: *Lofted Solid: Pick two faces for lofted solid*.
2. Enter a value in the Bulge data field.

   Be careful not to apply a bulge so large that it results in a self-intersecting body. If it does you will receive the following error message.



   Also be aware that bulge factors that are too small with respect to the solids may prevent later feature operations such as shelling and blending.

3. Select a solid face.
4. Select a face on the other solid.

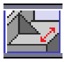
   One loft solid is created from the two solids using the shared faces for tangent conditions.

The Status Line contains the Bulge data field.

Geometric Characteristics

A lofted solid is created by selecting a face on two solids. It is made up of the following characteristics according to the Edit Objects dialog box: Bulge 1 and Bulge 2. Bulge 1 applies to the bulge between the first face selected and the lofted solid. Bulge 2 applies to the bulge between the second face selected and the lofted solid. To display the Edit Objects dialog box for the lofted solid only, double-click the lofted solid item in the Design Explorer.

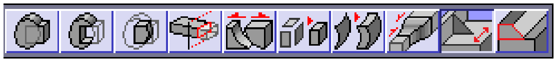
Rib Solid Tool



The rib tool extrudes and thickens a closed or open profile into a base solid. The profile is terminated up to the first face it reaches in the direction of the extrude.

Using the Rib Solid Tool

1. Select the Rib Solid tool. The Message Line reads: *Rib Feature: Pick solid for rib.*



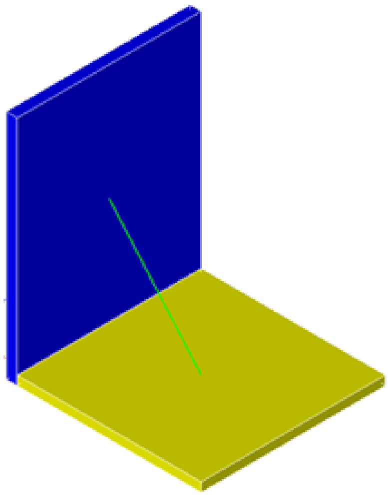
2. Select the solid you want to add the rib to. The Message Line reads: *Rib Feature: Pick rib profile. [Shift = Extend]*

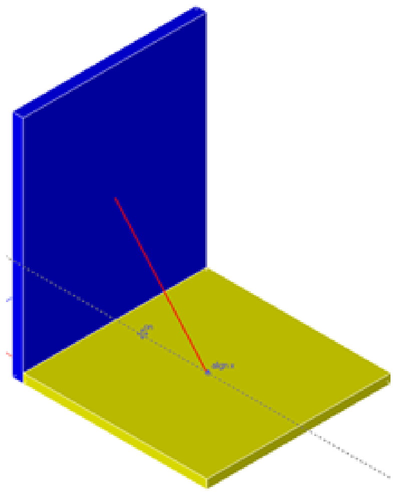
   After you select the rib profile you need to specify the thickness of the rib.This is done in the Status Line entry box pictured here.

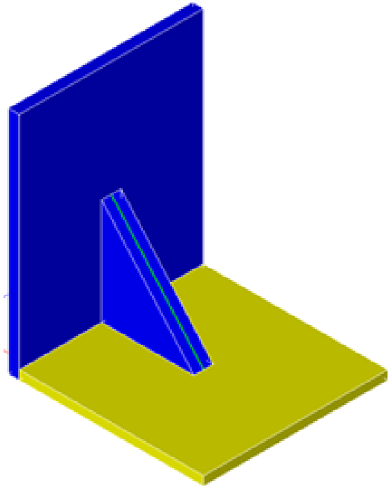
3. Once you select the rib profile The Message Line reads: *Rib Feature: Specify 2 points for rib direction*.

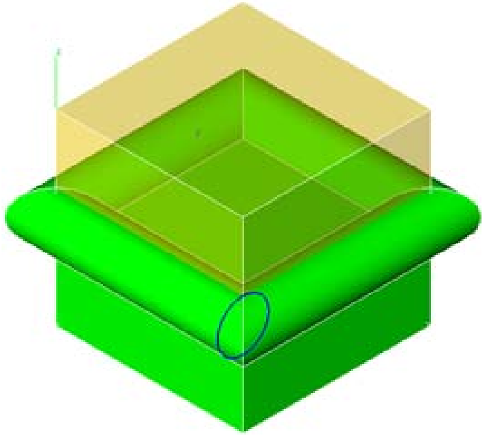
   For the tool to function correctly your direction should run into the solid you are adding the rib feature to.

If you want to edit the Thickness of the rib after you have placed the rib, you must go through the Design Explore and access the Edit Objects box associated with the Rib. The examples below show the tool use progression.



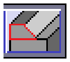






Lip Solid Add Feature Tool example.

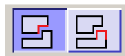
Lip Solid Tool



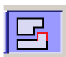
The Lip Solid tool sweeps a profile about a collection of edges and either adds or subtracts the resultant sweep to or from the base solid. The given profile is automatically aligned perpendicular to the selected edge. In addition, a reference point is specified such that the profile is translated from the reference point to the start position along the edge selected. When the base solid is regenerated the profile is always translated from the reference position to that on the edge and then realigned.

Note: If you pick a face for the lip solid tool all edges of the selected face are used for the lip tool.

The Lip Solid tool has two options in the Message Line, Add Feature and Remove Feature.



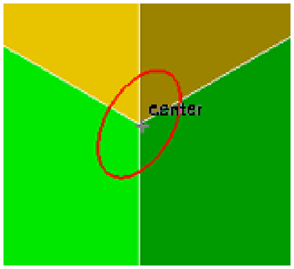
Using the Lip Solid Add Feature Tool



1. Select the Lip Solid tool. The Message Line reads: *Lip Feature Add: Pick curves for lip profile. [Shift = Extend]*

2. Select the curves for the lip profile. The Message Line reads: *Lip Feature: Specify profile origin*.

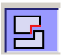
   The center of the circle is selected to be the origin in this example.



3. The Message Line reads: *Lip Feature Add: Pick edges for profile to sweep. [Shift = Extend]*

4. Pick the edges of the solid you want to add the lip feature to. In the example above, the edges of the green block were chosen.

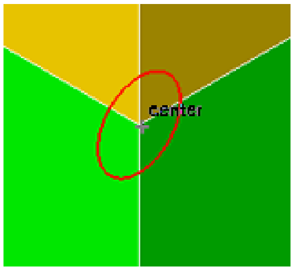
Using the Lip Solid Remove Feature Tool.



1. Select the Lip Solid tool. The Message Line reads: *Lip Feature Remove: Pick curves for lip profile. [Shift = Extend]*

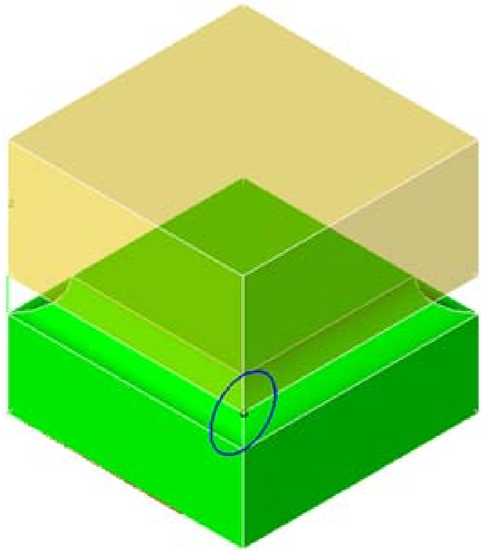
2. Select the curves for the lip profile. The Message Line reads: *Lip Feature: Specify profile origin*.

   The center of the circle is selected to be the origin in this example.



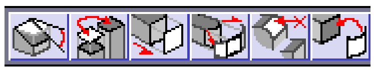
3. The Message Line reads: *Lip Feature Remove: Pick edges for profile to sweep. [Shift = Extend]*

4. Pick the edges of the solid you want the lip feature profile to be removed from. In the example below, the edges of the green block were chosen.



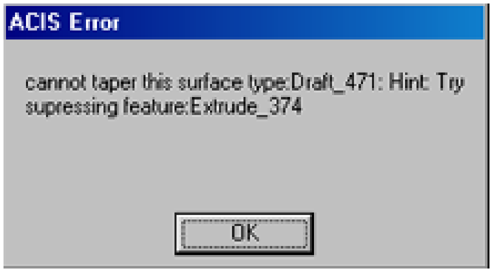
Note: You cannot use the remove feature on both block edges at the same time.

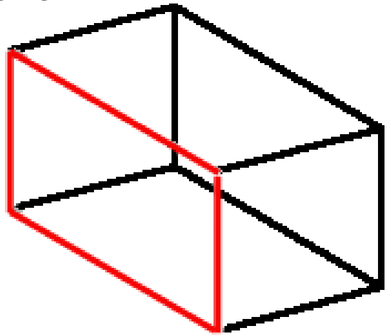
Local Face Modeling Tools



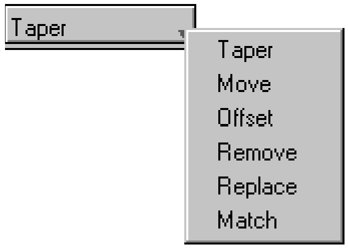
The Local Face Modeling tool allows you to perform operations on faces independently of how the part was created. This means you can bring in parts from other systems and still perform edits to the geometry.

This tool also includes an error detection routine. If your model contains a problem that prevents the completion of the local face modeling operation, an ACIS error dialog box appears with the problem location highlighted.





The Local Face Modeling tool offers six modeling options, as listed in the Message Line: *Taper*, *Move*, *Offset*, *Remove*, *Replace* and *Match*.



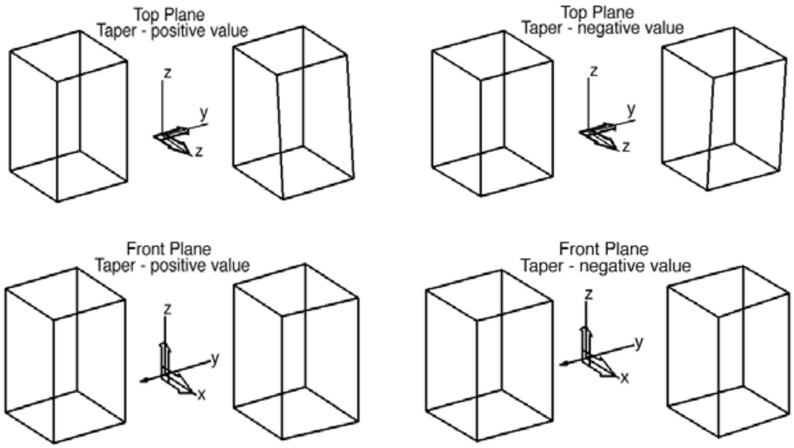
The Draft Face Tool

Choosing the Draft Face tool applies a draft to a group of selected faces. Positive tapers (draft) add material. Negative tapers (draft) remove material. The taper direction is relative to the pull-direction defined by the current work plane and a neutral point.

Rules

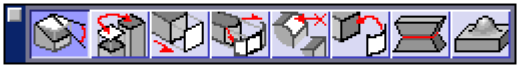
- The work plane cannot be parallel to the face or faces that you taper. The work plane defines the pull-direction or direction of draw, the direction the part would eject from a mold.
- The neutral point is the position on the face or edge that remains fixed while the face tapers.
- The taper is the angle that the face rotates relative to the neutral point. The value can be negative or positive and the rotation occurs relative to the work plane. Typically positive values add material and negative values remove material. The

graphics here illustrate the relationship between that work plane and the taper angles.



Using the Draft Tool

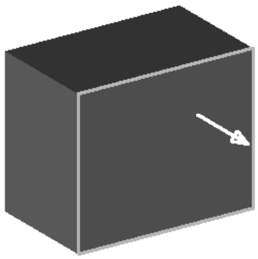
1. Select the Draft Solid tool. The Message Line reads: *Draft Solid: Pick reference face or edge for draft (click nothing for work plane)*.



2. Select the reference face, edge or work plane. This tells your Designer Elements program the pull direction of the normal for the taper.

   When you select the reference an arrow displays indicating the normal direction. An example is shown here.

   The Message Line now reads, *Draft Solid: Pick faces of a solid to draft [Shift=Extend]*.



Tech Note:

Make sure the face you choose is not normal to the z-axis. If it is, nothing will happen. Change the work plane and try again.

3. Select a face or hold down the SHIFT key to select more than one face. The Message Line now reads, *Draft Solid: Specify taper neutral position*.
4. Click a point on the screen to serve as the neutral position. The neutral position will remain constant.

   Your Designer Elements program applies the taper or draft relative to a pull direction defined by the normal.

While the solid is selected you can change the Draft angle in the Status Line. Press ENTER (Windows) or RETURN (Macintosh) and the taper updates.

The Status Line contains the Draft angle.

Modifying Drafted Solids

You can modify the draft angle of your solid by choosing the draft in the Design Explorer for the solid and displaying the Edit Objects dialog box. If you have more than one face drafted, you can specify a different draft for each one. Click Apply to accept the changes.

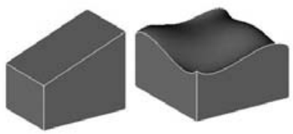
Geometric Characteristics

A draft solid is created by choosing a solid, specifying a draft, the face for the draft and the neutral point position. It is made up of these characteristics according to Edit Objects: Draft angle and the solid face list with the drafts. This list is only available if you select the operation in the Design Explorer first and then display the Edit Object dialog box. See Chapter 24 for more information on the Design Explorer.

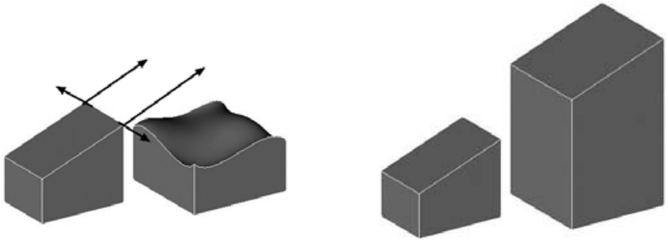
The Match Face Tool

Choosing the **Match Face** tool matches the selected surface to a referenced surface. This tool only works if the referenced surface is analytical (planar or circular). A nurb surface is not an analytical surface.

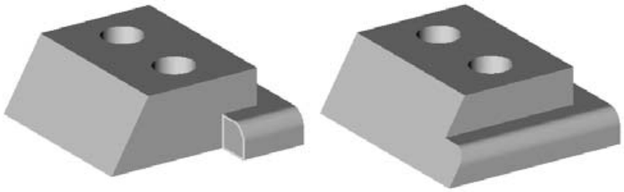
Planar Example



This is an example of matching the top face of the right model to the top face of the left model, which acts as a reference. Imagine the top planar face of the left solid extended beyond its bounds, as shown in the left graphic here. When the operation is complete the nurb face of the right solid is replaced with the planar face, as shown in the right graphic. The solid is extended to the imaginary intersection with the top planar face of the left solid.

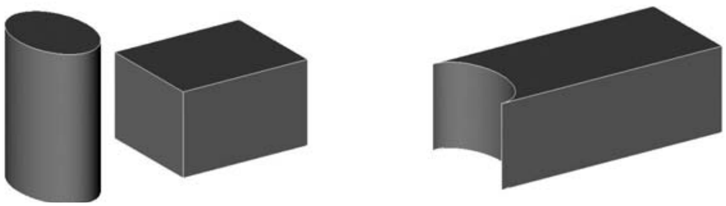


Here’s another example of the match face option. The left graphic shows a select face on a solid object. The right graphic shows the face matched to the sloped face.



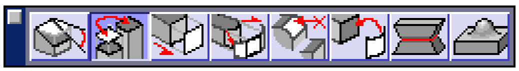
Elliptical Example

This left graphic below shows an example of matching a side face of the rectangular solid to the elliptical solid, which acts as a reference. When the operation is complete the solid is extended to the elliptical solid and the planar face becomes an elliptical face, shown in the right graphic.



Using the Match Face Tool

1. Select the **Match Face** tool.The Message Line reads: *Match Face: Pick face to*



modify.

2. Select the face of the solid you want to modify or match to another face. To select more than one face be sure to hold down the SHIFT key before selecting the first face.

   The Message line now reads, *Match Face: Pick face to match to*.

3. Select the face of the solid that you want the selected face to match.

   The face is matched.

The Status Line contains no entries.

Geometric Characteristics

A matched face has no specific characteristics. To modify the Display and Attribute characteristics of the matched face, select the face using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the face name.

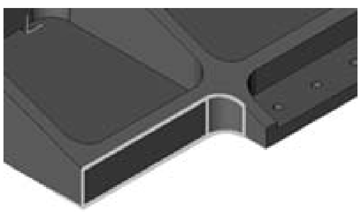
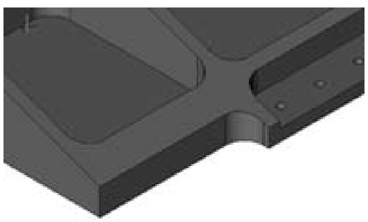
Display contains settings for Iso Lines and Silhouette. You can display Iso Lines by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on the object. You have three Silhouette settings, Smart, Off and On.

See Chapter 24 for more information on the Design Explorer.

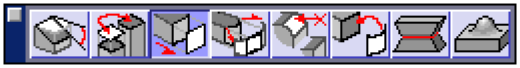
The Move Face Tool

Choosing Move Face tool moves the selected group of faces by a vector defined with the Drafting Assistant.

If moving more than one face the selected faces cannot be perpendicular to others. This tool is valuable for moving a flat face and fillet or a face and a hole.The graphics here show just such an operation. The left graphic shows the original geometry with the selected faces and the right graphic shows the geometry with two faces moved.

Using the Move Face Tool



1. Select the Move Face tool.The Message Line reads: *Move Face: Pick faces of a solid to move [Shift=Extend]*.

2. Select the face(s) of the solid you want to move. To select more than one face hold down the SHIFT key before selecting the first face.

   The Message Line now reads, *Move Face: Pick two points for move*. The points define a vector with DX, DY, DZ values. You can either click two points or enter the desired values in the Status Line.

3. Select the two points to specify dX, dY, dZ values for the move. The order you select your points determines whether the value is positive or negative.

   The selected face moves to the new location.

While the face is still selected you can enter new values in the dX, dY, or dZ data fields in the Status Line. Press ENTER (Windows) or RETURN (Macintosh) to update the move.

The Status Line contains the dX, dY and dZ values of the move.

Geometric Characteristics

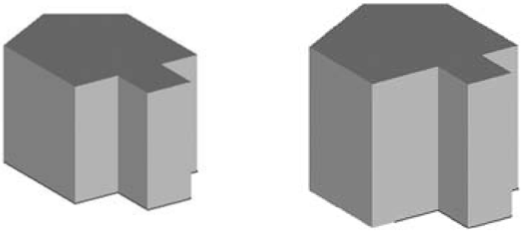
A move face has no specific characteristics. To modify the Display and Attribute characteristics of the move face, select the face using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the face name.

Display contains settings for Iso Lines and Silhouette. You can display Iso Lines by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on the object. You have three Silhouette settings, Smart, Off and On.

See Chapter 24 for more information on the Design Explorer.

The Offset Face Tool

Choosing the Offset Face tool offsets the selected faces. Adjacent faces and edges are adjusted as necessary for the offset. The left graphic below shows the original object. The right graphic shows a positive offset of the left and top face.

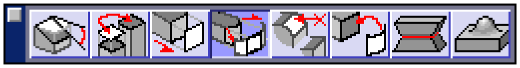


Tech Note:

When using the *Move Face* option, you will receive an error if you try to move the whole face off a solid.

Using the Offset Face Tool

1. Select the Offset Face Modeling tool.



2. The Message Line reads: *Offset Face: Pick faces of a solid to offset [Shift=Extend]*.
3. In the Status Line enter a value in the data field. A positive value enlarges the object and a negative value reduces the object.
4. Select the face of the solid you want to offset. To select more than one face be sure to hold down the SHIFT key before selecting the first face.

   The selected face moves the offset specified.

While the object is still selected you can enter new values in the data field. Press ENTER (Windows) or RETURN (Macintosh) to update the offset.

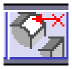
The Status Line contains the Offset value.

Geometric Characteristics

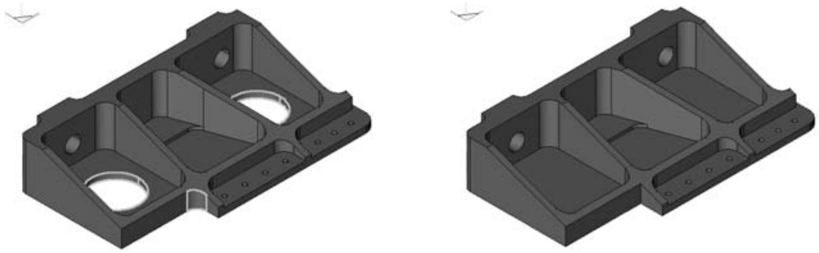
An offset face is defined by its offset value according to the Edit Objects dialog box. To modify the offset or the Display and Attribute characteristics of the offset face, select the face using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the face name.

Display contains settings for Iso Lines and Silhouette. You can display Iso Lines by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on the object. You have three Silhouette settings, Smart, Off and On. See Chapter 24 for more information on the Design Explorer.

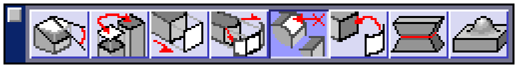
The Remove Face Tool



Choosing the Remove Face tool removes a face from a solid, extends the adjacent faces and relimits the solids as necessary. This is useful for removing holes and fillets from geometry imported without a history tree. The left graphic shows the original object. The right graphic shows the object with the holes and a fillet removed.



Using the Remove Face Tool

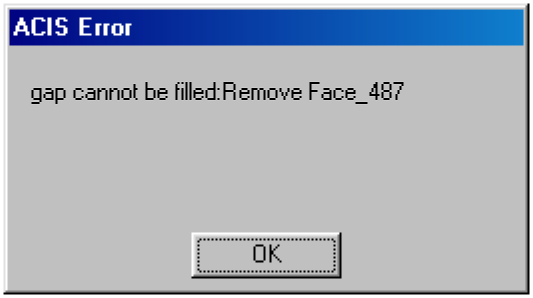


1. Select the Remove Face tool.
2. The Message Line reads: *Remove Face: Pick faces of a solid to remove [Shift=Extend].*
3. Select the face of the solid you want to remove. To select more than one face be sure to hold down the SHIFT key before selecting the first face.

   The selected face is removed.

There are no Status Line entries.

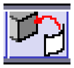
Warning: A face can only be removed from a solid as long as the integrity of a solid is retained after the face removal. You cannot remove a face of a cube if the cube has a blend, the blend can be removed. If you attempt to remove a face that will challenge the solid's integrity, the following error message appears.



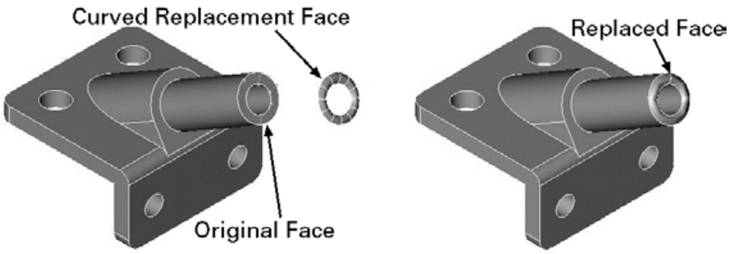
Geometric Characteristics

A removed face has no specific characteristics.

The Replace Face Tool

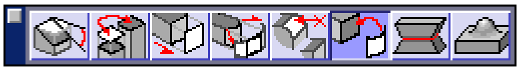


Choosing the Replace Face tool replaces the selected face with the new face specified. The new face must have the same boundaries or edges as the original face.



Using the Replace Face Tool

1. Select the Replace Face tool.



2. The Message Line reads: *Replace Face: Pick face to replace*.
3. Select the face of the solid you want to replace. To select more than one face be sure to hold down the SHIFT key before selecting the first face.

   The Message line now reads, *Replace Face: Pick new surface.*

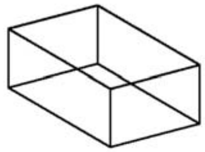
4. Select the replacement face.

   The original face is replaced.

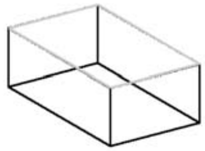
The Status Line contains no entries.

Replace Face Example

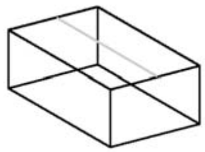
You have a block (below) and you want to replace the top face with a warped surface.



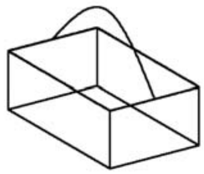
1. Select the Rectangle tool and trace over the top face of the block.



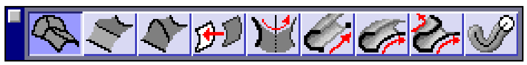
2. Select the Through Point B-Spline tool and create a three point spline from the midpoint of the of one edge to the midpoint of the opposite edge.



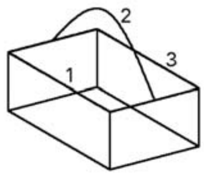
3. Using the Selection tool, drag select the center control point of the spline and drag it up along the z axis some distance.



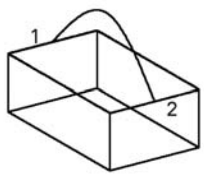
4. Select the Net Surface tool.



5. Select the M curves consisting of the left edge, spline and right edge curves, (curves 1, 2 and 3 in the graphic).



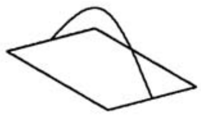
6. Select the N curves consisting of the top and bottom curves (curves 1 and 2 in the graphic) two complete the net surface.

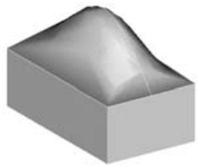


7. Select the *Replace Face* tool.

8. Select the top face of the block to replace.
9. Select the net surface you just created to replace the face.

   The top face of the solids is replaced.
10. Now render the block.



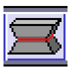


Geometric Characteristics

An replace face has no specific characteristics. To modify the Display and Attribute characteristics, select the face using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the face name.

Display contains settings for Iso Lines and Silhouette. You can display Iso Lines by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on the object. You have three Silhouette settings, Smart, Off and On. See Chapter 24 for more information on the Design Explorer.

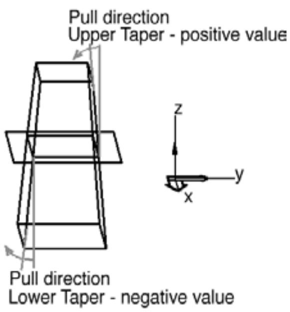
Parting Line Solid Tool

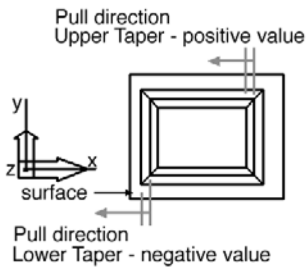


The Parting Line tool automatically applies draft angles to a collection of upper and lower surfaces. These faces are determined by the location of the parting line curve. This Designer Elements program uses curves as the parting entity so you must create the curves before using the tool.

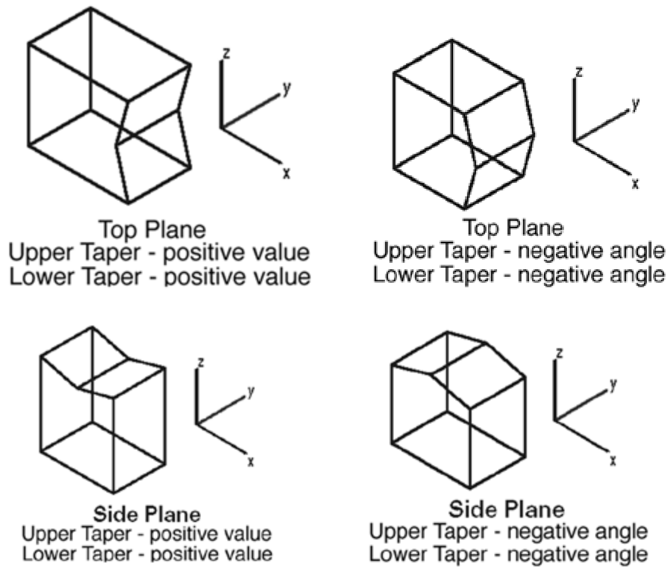
Rules

- The work plane cannot be parallel to the face or faces that you taper. The work plane defines the pull-direction or direction of draw, the direction the part ejects from a mold.



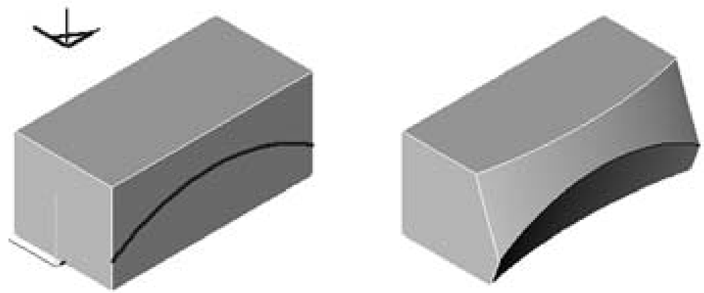


- The taper is an angle that the face rotates relative to the neutral point. The value can be negative or positive and the rotation occurs relative to the work plane. Typically, positive values add material and negative values remove material. The graphics here illustrate the relationship between that work plane and the taper angles.



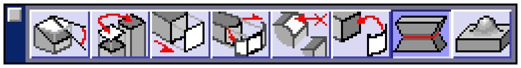
- If you have difficulty placing a draft on a solid try changing the work plane and attempting the draft again.

The left graphic shows an object with a parting line curve. The right graphic shows the object after using the tool.



Using the Parting Line Solid Tool

1. Select the Parting Line tool. The Message Line reads: *Parting Line Solid: Pick solid for parting line.*



2. Select the solid that you want to part. The Message Line reads: *Parting Line Solid: Pick curves for parting line.*
3. Select the parting line curves. To select more than one curve be sure to hold down the SHIFT key before selecting the first curve.

While the solid is still selected you can change the upper and lower taper values. Press ENTER (Windows) or RETURN (Macintosh) and the part updates.

The Status Line contains the Upper Taper and the Lower Taper angle for the solid.

Modifying Parting Line Solids

You can modify the taper angles of your solid by choosing the taper in the Design Explorer for the solid and displaying the Edit Objects dialog box. You can modify the taper angle for the Upper and Lower Tapers. Click Apply to accept the changes.

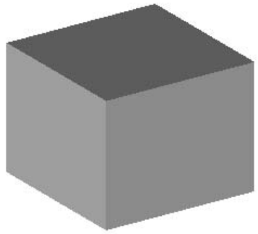
Geometric Characteristics

A parting line solid is created by choosing a solid, intersecting curve and the upper and lower taper. It is made up of these characteristics according to Edit Objects: Draft angle and the solid face list with the tapers. This list is only available if you select the operation in the Design Explorer first and then display the Edit Object dialog box. See Chapter 24 for more information on the Design Explorer.

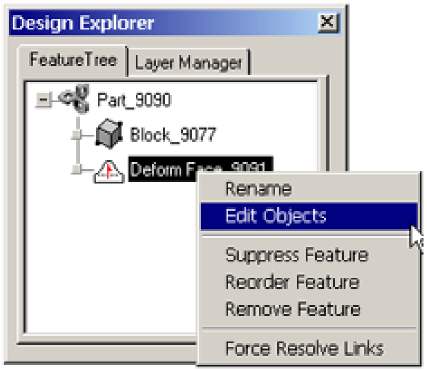
Deform Face Tool

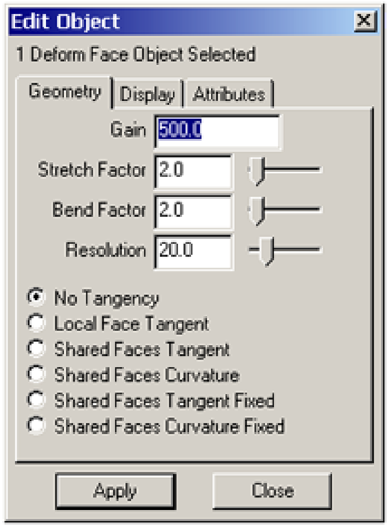
The Deform Face tool allows you to deform surfaces and solid faces. When you select the Deform Face tool, a subpalette appears in the Message Line containing three tools, Deform with Gain, Deform to Point and Deform to Curve.

All deform face examples will use the solid block here to illustrate each Deform Face tool.



After you have used the Deform Face tool and continue building you model and then you decide you want to edit the deformed face you must access the Edit Objects dialog box through the Design Explorer.





The Deform Face Edit Objects box contains the following options:

Gain

The gain value is similar to a constraint pressure applied to the surface. Use positive gain to inflate the surface and negative to deflate. Depending on the stiffness and resolution of the surface practical gains can range anywhere from 0 to 1000000.

Stretch Factor

The stretch factor controls the deformable surface's resistance to stretching. A surface with a large stretch value is said to be stiff. Deformable surfaces with large stretch values act like soap bubbles seeking to always minimize their area. This results in flatter looking surfaces that allow regions of rapid bending.

Bend Factor

The bend factor controls the deformable surface's resistance to bending. Deformable models with large bend values act like elastic beams attempting to distribute regions of bending over large areas and typically generate very fair shapes.

Resolution

The resolution slider controls the precision of the resultant deformed shape by inserting additional control points to the surface. Higher resolution values will show more detail for the given deformation values. Lower resolution values calculate faster but with less detail. When using the tangent and curvature options, start with a resolution factor of 80 for best results.

The following options apply to only to pressure deformations, the first tool icon. It does not apply to deformations to a point or curve. Be sure to use higher resolutions when using the below options to insure tangency and curvature precision with the results.

No Tangency

The No Tangency option deforms the face and allows the shape at the edges to deform.

Local Face Tangent

The Local Face Tangent option deforms the shape but preserves the existing tangencies of the face.

Shared Faces Tangent

Shared Face Tangent will modify all faces that share edges with the face being modified. The modification will impose tangencies at all shared edges.

Shared Faces Curvature

The Shared Face Curvature option will modify all faces that share edges with the face being modified. The modification will impose curvature continuity at all shared edges.

Shared Faces Tangent Fixed

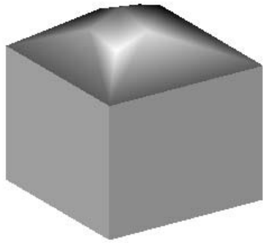
The Shared Face Tangent Fixed option will only modify the selected face to be tangent to all faces that share an edge with the selected face.

Shared Faces Curvature Fixed

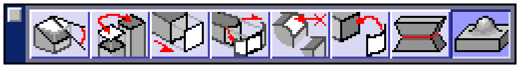
The Shared Face Curvature Fixed option will only modify the selected face to be curvature continuous to all faces that share an edge with the selected face.

Deform Face with Gain Tool

This tool applies a pressure gain (negative or positive) to the selected surface or face. The gain displays in the Status Line. Large gains distort the object while small gains barely move the object. The pressure applies to the entire surface or face. You cannot specify a location for the deformation. Use the Deform Face to Point tool for that ability. The graphic shows the top face deformed using this tool.



Using the Deform Face with Gain Tool



1. Select the **Deform Face** tool.
2. Select the Deform Face with Gain tool in the Message Line. The Message Line reads: *Deform Face: Pick face to deform with pressure.*
3. Type the desired pressure in the Gain data field of the Status Line.

   The number can be a positive or negative value.
4. Select the face or surface that you want to deform.

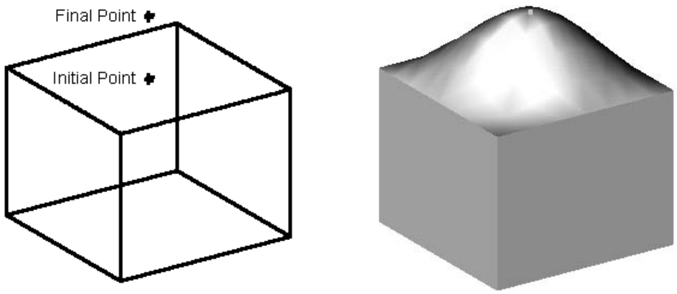
   The object is deformed. Render your object using any render option other than wireframe to view the deformation.

While the object is still selected you can enter a new value in the Gain data field and press ENTER (Windows) or RETURN (Macintosh) to change the gain.

The Status Line contains the Gain value for the deformation.

Deform Face to Point Tool

This tool applies a pressure gain (negative or positive) to the selected surface or face passing through a specific point in space. You choose a point on the object to begin the deformation and the point's final location. The left graphic shows the initial point on a solid face and its final location. The right graphic shows the rendered object and the point.



The pressure gain displays in the Status Line.

Using the Deform Face to Point Tool

If you want the deformation to pass through a point not already in your drawing, create the point before using this tool.

1. Select the Deform Face tool.
2. Select the Deform Face to Point tool in the Message Line. The Message Line reads: *Deform Face: Pick face to deform*.
3. Type the desired pressure in the Gain data field of the Status Line.
4. Select the surface or face that you want to deform.

   The Message Line reads: *Deform Face: Pick location to deform from*.
5. Click a location on the selected surface/face to set the point from which the face deforms.

   The Message Line reads: *Deform Face: Pick location to deform to.*
6. Click the final location for the face point where the deformation begins.

   The object deforms. Render your solid using any render option other than wire-frame to view the deformation.

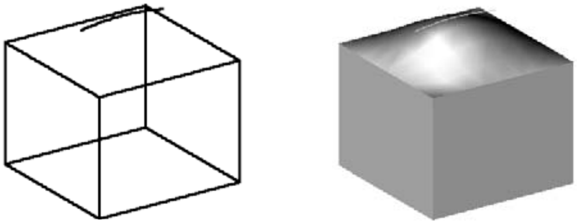
While the surface or face is still selected you can enter a new value in the Gain data field and press ENTER (Windows) or RETURN (Macintosh) to change the gain.

You can move the point later by dragging it to a new location or using the Edit Objects dialog box when the point is selected. The deformed object automatically updates.

The Status Line contains the Gain value for the deformation.

Deform Face to Curve Tool

This tool applies a pressure gain (negative or positive) to the selected surface or face attached to a specified curve. You create the curve before using this tool. The left graphic shows the object with the curve. The right graphic shows the rendered image and the curve.



The pressure gain displays in the Status Line. A high enough gain forces the surface or face to pass through the curve. The curve is associative to the deformation. Moving the curve changes the associated deformation.

Using the Deform Face to Curve Tool

If you want the deformation to use a curve not already in your drawing, create the curve before using this tool.

1. Select the Deform Face tool.
2. Select the Deform Face to Curve tool in the Message Line. The Message Line reads: *Deform Face: Pick face to deform to curve.*
3. Type the desired pressure in the Gain data field of the Status Line.
4. Select the surface or face that you want to deform.

   The Message Line reads: *Deform Face: Pick curve constraint.*
5. Select the curve.

   The object is deformed. Render your object using any render option other than wireframe to view the deformation.

While the object is still selected you can enter a new value in the Gain data field and press ENTER (Windows) or RETURN (Macintosh) to change the gain.

The Status Line contains the Gain value for the deformation.

Geometric Characteristics

A deformable surface or face is created by selecting object and using pressure, a point or curve. This deformed surface or face is made up of the following characteristics according to the Edit Objects dialog box: Gain, Stretch Factor, Bend Factor and Resolution. The dialog box also contains the Keep Boundary Tangents option.

Gain

This field displays the pressure applied to the surface or face. You can enter a different gain and click Apply.

Stretch Factor

This field sets the resistance of the surface or face to the stretching. Higher values result in a flatter surface. Lower values result in a smoother surface. Enter a value yourself or use the slide to specify a factor.

Bend Factor

This field sets the surface or face resistance to bending. Higher values prevent sharp radical changes to the surface. Enter a value yourself or use the slide to specify a factor.

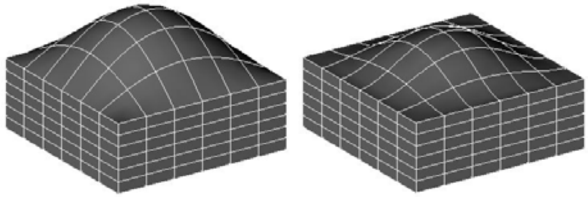
Resolution

This field sets the number of internal grids used to calculate the deformation. Higher values increase the influence of the bend and stretch factors to pressure gains. Lower values decrease the influence, providing quick results but few discriminating features to your surface. Enter a value or use the slide to specify a resolution.

Keep Boundary Tangents

This option determines whether the edges of the surface or face remain tangent. When selected, the edges are tangent. This left graphic here shows the top deformed solid face without tangent edges.

The right graphics shows the deformed face with tangent edges.



To modify the Display and Attribute characteristics of the deformed surface or face select it using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the item's name.

Display contains settings for Iso Lines and Silhouette. You can display Iso Lines by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on the object. You have three Silhouette settings, Smart, Off and On.

See Chapter 24 for more information on the Design Explorer.

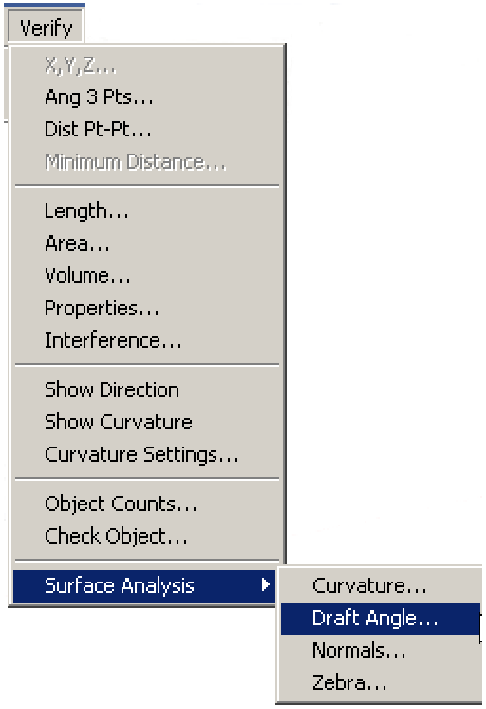
Draft Evaluation Feature

This Designer Elements program gives you the ability to evaluate the drafts of an object for molding purposes. This feature is available through the Verify menu and the advanced rendering capabilities.

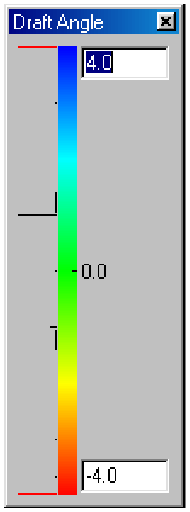
Draft Evaluation through Verify Menu

The *Draft Angle* in the Verify menu allows you to quickly evaluate the draft angles of a model.

All analysis colors are preset with this commands. If you would like to define your own settings, use the draft evaluation shader through the Advanced Rendering feature. See “Draft Evaluation through Advanced Rendering” on page 40.



This command evaluates the drafts of an object. When you select this command, the Draft Angle dialog box appears with the draft angle analysis, containing the histogram with the analysis, the color spectrum and histogram data fields.



The dialog box contains the following elements:

Histogram

Located to the left of the color spectrum in the dialog box, these horizontal bars represent the frequency of a draft angle using the color spectrum. The length of the bar represents the frequency.

This Designer Elements program calculates this histogram and displays it so that the entire graph fits into the dialog box area.

Color spectrum bar

The section displays the color septum used to create the histogram.

Histogram data fields

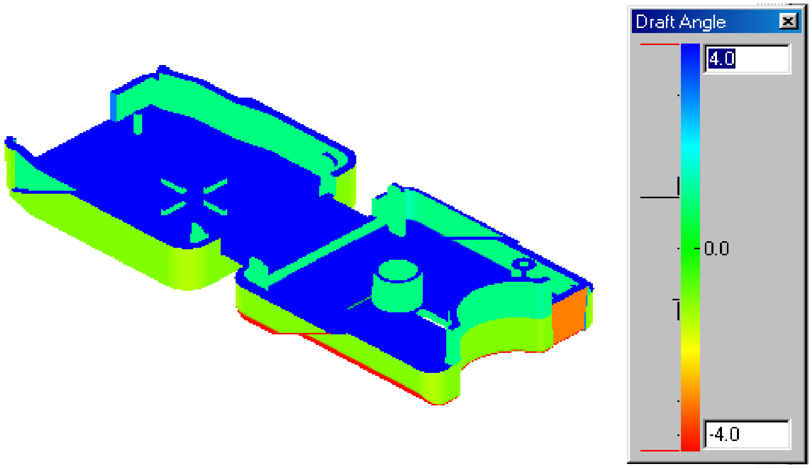
The data fields display the maximum and minimum values used to calculate the histogram. When you initially select the command, this Designer Elements program scans the surface(s) and sets the min and max fields for the draft angle analysis.

If you enter different values in the fields such that a large number of values fall outside of the specified range a red line appears at the end(s) where the values fall. All values are still calculated, even though they are not displayed due to the specified range.

If you make changes to the values, this Designer Elements program waits for two seconds of non-action before recalculating the histogram, giving you time to change the values before the image is rendered again.

You can return the range initially displayed by clicking on the selected option again. (The option does not deselect when you do this.) Windows users can also reset the maximum and minimum values by right mouse clicking near the respective end of the spectrum.

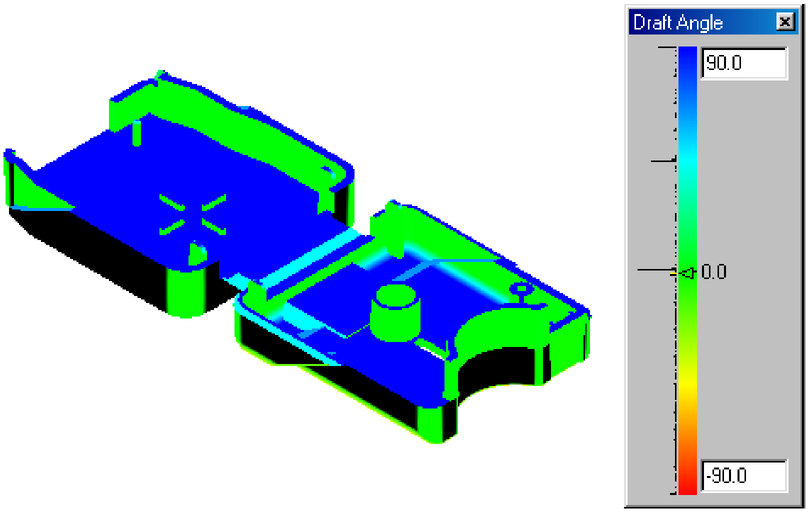
The graphic here is an example of a draft angle analysis.



Using the Draft Angle Command

1. Select the object.
2. Choose ***Verify>Surface Analysis>Draft Angle***.

   The dialog box displays with the analysis. Your geometry also displays the analysis.
3. To display a certain analysis/color area, place the pointer at the location over the color spectrum and click the mouse.

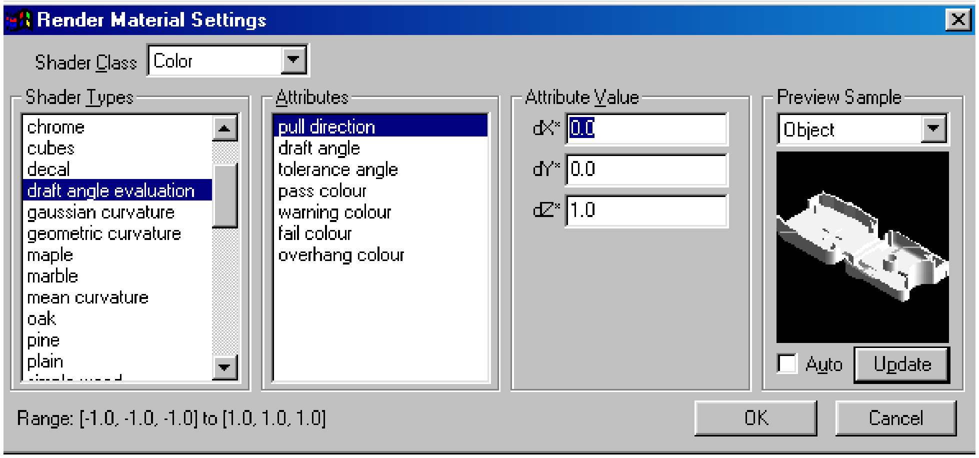


Notice that a triangular indicator appears at the selected location and the related color highlights in the histogram. The same color highlights in black on your object.

4. To remove the triangular indicator click in the dialog box outside of the color spectrum bar.
5. Change the histogram values in the data fields and the histogram and analysis automatically recalculates.

Draft Evaluation through Advanced Rendering

Using the Advanced Rendering feature you can specify your own draft angle settings. When you display the Render Material Settings dialog box and choose the draft angle evaluation type, the dialog box appears.



The draft evaluation type contains the following attributes:

pull direction This specifies the direction the object is pulled from the mold. Enter the values for the pull direction or drag in the drawing area to have the values automatically entered. The asterisk next to the dX, dY and dZ names indicate this ability.

draft angle This specifies the draft angle required to pull the object out. This angle is usually 1°. Entering a 0 results in the fail color zone disappearing when the scene is rendered.

tolerance angle — This specifies the degree tolerance added to the draft angle that still allows the object to be pulled from a mold but with difficulty. Entering a 0 results in the warning color zone disappearing.

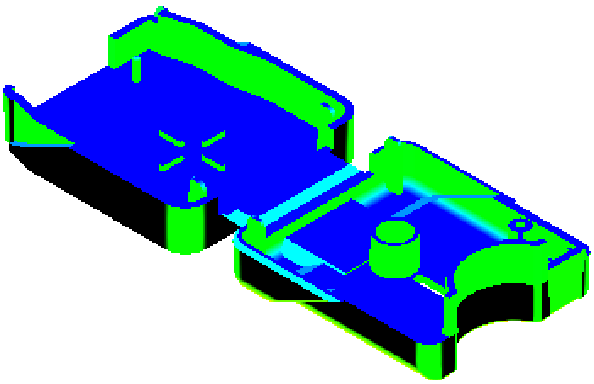
pass color — This specifies the color of the object for those areas where the draft angles are satisfactory. The surface normals fall between 90°+ draft angle + tolerance angle and 180° with the pull direction.

warning color — This specifies the color that appears on the object when the surface normals fall between 90°+ draft angle and 90°+ draft angle + tolerance angle.

fail color — This specifies the color of the object in those areas where the object cannot be pulled from the mold. The surface normals fall between 90° and 90°+ draft angle.

overhang color — This specifies the color that appears for any part of the mold overhanging the object. An overhang on the mold prevents the object from being pulled from the mold.

In the graphic here the pass color areas are medium gray, the overhang color areas are black and the warning color areas are light gray.



Using the Draft Evaluation Feature

1. Double click on the object you want to evaluate to display the Edit Objects dialog box.

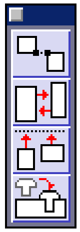
A material must already have been placed on the object.

2. Click the Advanced button to display the Render Material Settings dialog box.
3. Select draft angle evaluation from the Shader Types list.
4. Specify the settings for the attributes.
5. Click OK to close the dialog box and save the settings.
6. Render your object using one of the photorealistic commands.

The object displays using your specified draft evaluation colors and values.

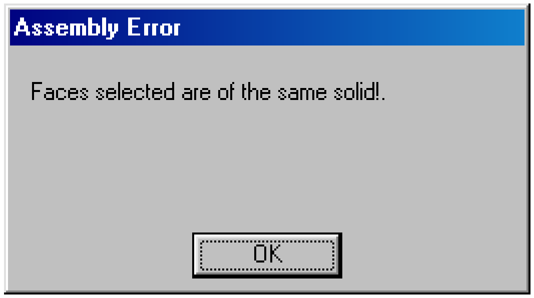
Assembly Modeling Tools

This Designer Elements program provides you with tools for doing assembly modeling. These tools are located in their own tool palette. Choose ***Window>Assembly*** to display the tool palette.

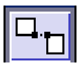


The Assembly Modeling tools include: Assembly Connect, Assembly Mate, Assembly Align and Assembly Insert.

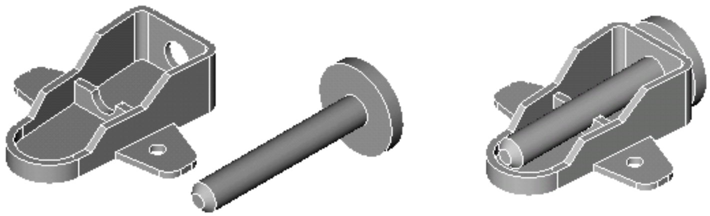
Each tool performs a particular operation involving two solids. The following error appears if you attempt to perform an operation on a single solid.



Assembly Connect Tool

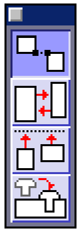


The Assembly Connect tool to connects one solid to another at a specified connection point. The resulting part is associative. When you move one solid the other moves as well to maintain the connection. The left graphic shows two solid objects. The right graphic shows the solids after connecting them.



This tool supports connection points along object edges including start, end and middle and on locations recognized by the Drafting Assistant (*center*, *midpoint*, *endpoint*, etc.). You cannot select more than one solid to connect at a time.

Using the Assembly Connect Tool



1. Select the Assembly Connect tool. The Message Line reads: *Assembly Connect: Pick solid to move.*
2. Select the solid.

   The Message Line now reads, *Assembly Connect: Pick the connection point location on selected solid to move.*
3. Using the Drafting Assistant select the connection point on the selected solid model.

   The Message Line reads: *Assembly Connect: Pick solid to move to.*
4. Select the solid to which you are moving the first solid.

   The Message Line reads: *Assembly Connect: Pick new location on the selected solid for first solid selected*.
5. Using the Drafting Assistant select the connection point on the selected solid to which you want to move the first solid selected.

   The two solids are connected.

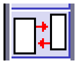
The Status Line contains no entries.

Geometric Characteristics

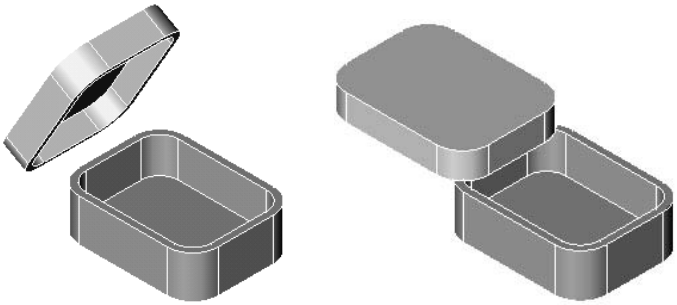
A connected assembly is made up of two solids. It does not include an characteristics that are unique. To modify Display and Attribute characteristics of an assembly, select the object using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the face name. See Chapter 24 for more information on the Design Explorer.

Display contains settings for Iso Lines and Silhouette. You can display Iso Lines by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on the object. You have three Silhouette settings, Smart, Off and On.

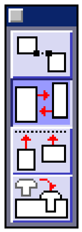
Assembly Mate Tool



The Assembly Mate tool mates planar faces. You can specify an offset distance between mated faces. The normals of the mated faces point in opposite directions. This tool only ensures that the selected faces are co-planar. If you want to align the faces, use the Assembly Align tool. The left graphic shows the two solids. The right graphic shows the solids after mating them.



Using the Assembly Mate Tool



1. Select the Assembly Mate tool. The Message Line reads: *Assembly Mate: Pick solid face to mate.*
2. Select the solid face you want to mate.

   The Message Line now reads, *Pick solid face to mate with.*
3. Select the solid face you want to mate with.

   The two solids are now mated.
4. If you want the first solid offset a certain distance from the second, enter a value in the Status Line data field and press ENTER (Windows) or RETURN (Macintosh).

The objects are offset the specified distance but are still co-planar.

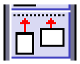
The Status Line contains the Offset data field.

Geometric Characteristics

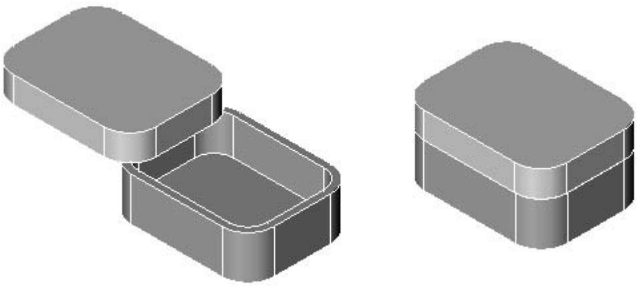
A mate assembly is made up of two mated solid objects and includes an Offset value according to the Edit Object dialog box. To modify the characteristics of a mate assembly, select the assembly using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the face name. See Chapter 24 for more information on the Design Explorer.

Display contains settings for Iso Lines and Silhouette. You can display Iso Lines by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on the object. You have three Silhouette settings, Smart, Off and On.

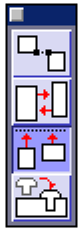
Assembly Align Tool



The Assembly Align tool aligns a face of two solid objects and supports cylindrical, spherical and planar faces. This tool reorients the parts so that the faces are aligned in the same direction. The left graphic shows the two solid. The right graphic shows the solids after aligning them.



Using the Assembly Align Tool



1. Select the Assembly Align tool. The Message Line reads: *Assembly Align: Pick solid face to align*.
2. Select the solid face you want to align.

   The Message Line now reads, *Assembly Align: Pick solid face to align with*.
3. Select the solid face to align with.

   The two solids are now aligned.

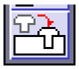
The Status Line contains no entries.

Geometric Characteristics

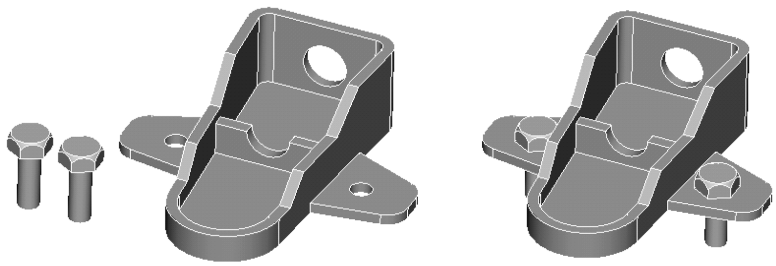
An aligned object is made up of two aligned solids and includes an Offset value according to the Edit Object dialog box. To modify the characteristics of an aligned object, select the object using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the face name. See Chapter 24 for more information on the Design Explorer.

Display contains settings for Iso Lines and Silhouette. You can display Iso Lines by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on the object. You have three Silhouette settings, Smart, Off and On.

Assembly Insert Tool

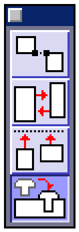


The Assembly Insert tool aligns and mates the faces of two solid objects. The align faces must be cylindrical and the mate faces must be planar. This tool is very useful for inserting bolts into holes. The left graphic shows two bolts and another solid. The right graphic shows the bolts inserted into the solid.



In the example, the bolt cylinder is aligned with the circular hole and its planar face mated with the planar face of the other solid though which the hole passes.

Using the Assembly Insert Tool



1. Select the Assembly Insert tool. The Message Line reads: *Assembly Insert: Pick solid face to align*.
2. Select the solid face you want to align.

   The Message Line now reads, *Assembly Insert: Pick solid face to align with*.
3. Select the solid face to align with.

   The Message Line reads: *Assembly Insert: Pick solid face to mate.*
4. Select the solid face you want to mate.

   The Message Line now reads, *Assembly Insert: Pick solid face to mate with*.
5. Select the solid face you want to mate with

   The first solid is now inserted into the second solid.
6. If you want the first solid offset a certain distance from the second, enter a value in the Status Line data field and press ENTER (Windows) or RETURN (Macintosh).

   The objects are offset the specified distance but are still co-planar.

The Status Line contains the Offset data field.

Geometric Characteristics

An inserted object is composed of two aligned and mated solids and includes an Offset value according to the Edit Object dialog box. To modify the characteristics of an inserted object, select the object using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the face name. See Chapter 24 for more information on the Design Explorer.

Display contains settings for Iso Lines and Silhouette. Display Iso Lines by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on the object. You have three Silhouette settings, Smart, Off and On.

Editing Commands

This Designer Elements program provides a variety of editing commands to enable you to edit your geometry quickly an easily. Some commands deal with the placement or your geometry like *Cut* and *Align*. Some commands change the appearance of your object like *Change Resolution*. Still others change your geometry like *Change Object Type* and *Edit Objects*.

The chapter includes these sections:

- Basic Editing Commands
- Advanced Editing Commands
- Object Commands
- Verify Menu

Basic Editing Commands

These commands provide some standard editing features like: *Cut*, *Paste*, *Undo* and *Redo*.

Cut - CTRL+X (Windows); z +X (Macintosh)

This command in the Edit menu removes the selected object(s) and places them on the Clipboard. Each selection you cut or copy to the Clipboard replaces the previous Clipboard contents.

Cutting Objects

1. Select the object to be cut.
2. Choose ***Edit>Cut***.

Once you cut a selection you can paste it. You can use *Cut* and *Paste* to move geometry or text around the document or from one document to another. You can even paste the cut selection onto a document in a different application.

Moving Geometry with the Cut Command

1. Select the object(s) you want to move.
2. Choose ***Edit>Cut***.
3. Display the location where you want the selection to appear in the drawing area, scrolling if necessary.
4. Choose ***Edit>Paste***.

The object appears in the center of the drawing area on the original layer on which it was created or onto the work layer if it's from a different application. The object is selected so you can move it, if you want.

Copy - CTRL+C (Windows); z +C (Macintosh)

This command in the Edit menu places a copy of the selection onto the Clipboard without deleting the original selection. You can paste the copy elsewhere in the current document or into a different document. You can even paste the copied selection into a document created with a different application.

Tech Note:

Do not use *Copy* and *Paste* to create an instance of a solid. By doing so you will break the parent/child relationship. (**Cobalt™ and Xenon™ only**)Use the **Move** tool instead. See Chapter 25 for more information.

Copying Objects

1. Select the object(s) to be copied.
2. Choose ***Edit>Copy***.

   The selection goes on the Clipboard and it remains in the current document.

Copy Pict/Copy Bitmap

By using the *Copy Bitmap* (Windows) or *Copy Pict* (Macintosh) command you can specify a region to copy as a Raster image. You can then paste the image back into the same document, another document or a different application.

Paste - CTRL+V (Windows); z +V (Macintosh)

This command in the Edit menu pastes a copy of the Clipboard contents onto the center of the drawing area. The Clipboard contents are not changed when you use the *Paste* command.

You can also paste the selection into another document or into a document created with another application.

If you want to create an even distribution of geometry, use the Linear Duplicate tool or the Polar Duplicate tool both are located in the Transformation tools palette.

Tech Note:

You cannot use the *Copy* and *Paste* commands to copy entries in data fields like in the Status Line and dialog boxes. For Windows, press the right mouse button and use the commands in the menu. For Macintosh, use z +C to copy and z +V to paste.

Rules:

1. If the object was cut or copied from a Designer Elements program document, when it is pasted into another Designer Elements program document it goes on the work layer regardless of the layer from which it was cut or copied. A pasted selection retains its original attributes.
2. Objects cut or copied from non-Ashlar-Vellum documents are pasted onto the work layer.

The Clipboard

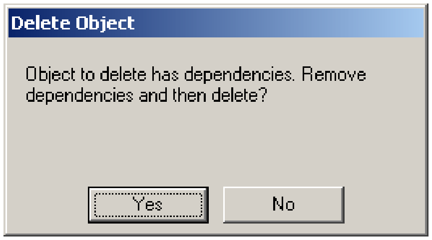
The *Cut* and *Copy* commands place a copy of the selected object on the Clipboard. The Clipboard is a buffer—a temporary storage place that holds the last cut or copied selection. The contents of the Clipboard are objects, not bitmaps or picts (unless you chose *Copy Bitmap/Pict*).

Erasing/Deleting Geometry

This Designer Elements program provides two methods for erasing objects:

- You can select the object(s) then press the BACKSPACE (Windows) or DELETE (Macintosh) key. (You can retrieve what you deleted with the *Undo* command.)
- You can select the object(s) then choose ***Edit>Cut***. (You can retrieve what you cut by using the *Paste* command, as long as you have not cut or copied anything else.)

If you try to delete an object that is parent in a parent/child relationship, this warning message will appear (**Cobalt™ and Xenon™ only**):



When you delete objects in this Designer Elements program the data is still retained in the file until you delete again. If you delete objects before closing a file, perform the operation again to ensure that the data is removed.

Undo - CTRL+Z (Windows); z +Z (Macintosh)

This command in the Edit menu reverses the last action. You can undo an infinite number of times in this Designer Elements program. When you choose *Undo* the last action taken is displayed next to the command.

You can *Undo* actions that create and edit geometry and text but not actions that do not change the contents of the drawing such as resizing the window or quitting.

If you are using a tool which involves a multi-step process, such as creating a 3-Entity Fillet, choosing *Undo* reverts to the beginning of the process. Some activities can be terminated by pressing the ESC key.

Redo

This command in the Edit menu reverses the *Undo* command. You can redo an infinite number of times in this Designer Elements program. When you choose *Redo*, the last action taken is displayed next to the command.

You can *Redo* actions that create and edit geometry and text but not actions that do not change the contents of the drawing such as resizing the window or quitting.

Advanced Editing Commands

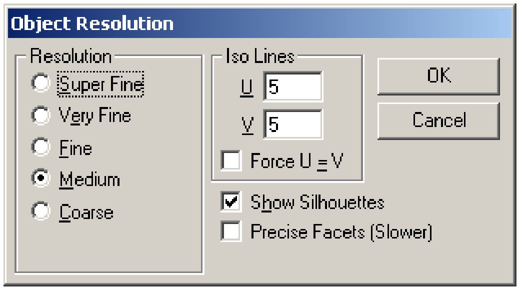
These commands provide more specific editing features/functions involving location, selection and display.

Change Resolution

This command, located in the Edit menu, changes the display resolution of curves, surfaces and solids.

A higher resolution means that this Designer Elements program uses more line segments to display curves on the screen so they look smoother but take longer to draw.

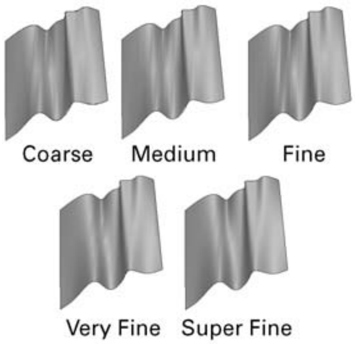
For surfaces and solids a higher resolution also means that this Designer Elements program uses more facets to display the objects. You can control the number of isolines used to display surfaces with this command.



The dialog box contains the following options:

Resolution

This section contains radio buttons for five resolution types *Course*, *Medium*, *Fine*, *Very Fine* and *Super Fine*.



Iso Lines

This section contains the data fields for *U* and *V* isopram (Iso) lines and the *Force U=V* check box.

The U and V data fields allow you to set the number of iso lines drawn for a surface. Iso lines are constant parameter curves that lie on a surface typically defined in parameter space. The parameter space coordinate system uses U and V coordinates where U = horizontal and V = vertical. A zero (0) in both fields turns off Iso lines. The appropriate U/V values may enhance the visual appearance of the surface at the expense of drawing speed. The letters, U and V are industry standard space coordinates.

The Force U=V check box automatically sets equal number of U and V lines. Entering a new value in one automatically changes the other isoline value.

Show Silhouettes This check box controls whether this Designer Elements program draws the silhouette edges of objects. Silhouette edges are view dependent and can cause a significant reduction in drawing speed. A check in the box turns on the silhouette.

OK Click this button to close the dialog box and save the new settings.

Cancel Click this button to close the dialog box without saving the changes.

Using the Change Resolution Command

1. Select the object whose resolution you want to change.
2. Choose ***Edit>Change Resolution***. The Object Resolution dialog box appears.
3. Select the desired resolution.
4. Click OK to save the resolution and close the dialog box.

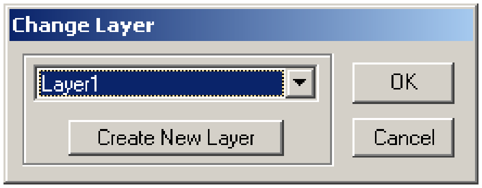
Change Layer

This command in the Edit menu moves selected objects to a different layer and creates new layers.

Using the Change Layer Command

1. Select the object that you want to move to a different layer.

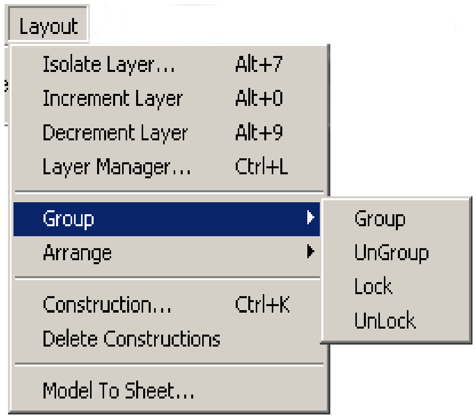
2. Choose ***Edit>Change Layer***. The following Change Layer dialog box appears:



Click on the pull-down menu and select the layer to which you want to move the selected object. If the layer doesn't exist click the Create New Layer button. A new layer appears in the layer field. Enter a new name if you wish.

Click OK to close the dialog box. The selected object is now placed on the selected layer.

Grouping Objects



A group is a collection of objects or short cuts that this Designer Elements program treats as one unit. When you move a group all members of the group move as a single unit. If you delete a group all members are deleted.

Group Command

This command located in the Layout menu includes a submenu with these commands: *Group*, *Ungroup*, *Lock* and *UnLock*.

Using the Group Command

1. Select the objects to be grouped.
2. Select ***Layout>Group>Group***.

Creating temporary groups is useful when moving multiple objects. You can drag a selection fence around several objects to treat them as a single unit while they are selected.

This Designer Elements program supports nested groups. You can create a group and then include it in a second group with other objects or groups.

If you want to add new objects to an existing group you have two options. If you select the group and the new object and choose the *Group* command, this Designer

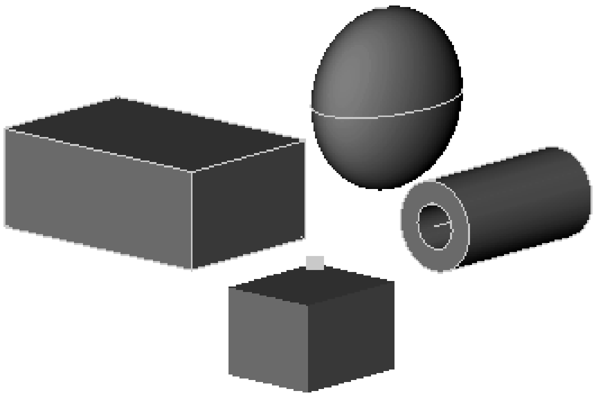
Elements program creates a group within a group. If you want all of the objects to be in a single group, follow these directions:

1. Select the group.
2. Choose ***Layout>Group>Ungroup***. The geometry is ungrouped and the individual objects remain selected.
3. Hold down the SHIFT key and select the geometry you want to add to the group.
4. Choose ***Layout>Group>Group***.

You can follow a similar procedure to remove members from a group.

Grouping Rules

- Grouped objects cannot be extruded, revolved or modified.
- If you change the color of a group and then ungroup the objects, all individual objects retain the new color.
- When grouping objects the *Group* command now ignores objects that only have a point selected. In the graphic below, the three top objects are completely selected and only one control point for the bottom block. When you use the command the control point is ignored. This is helpful when you have many objects located so closely to each other that it might be difficult to select the desired objects without selecting portions of others.



- If you are grouping objects on different layers, the group layer is the current work layer. However, if you ungroup the objects, the objects return to their original layer.

Ungroup Command

The *Ungroup* command separates selected grouped objects or short cuts into their individual components. If the group contains objects that were originally on different layers than the group layer. When ungrouped, the objects return to their original layer.

Using the Ungroup Command

1. Select the group.
2. Choose ***Layout>Group>Ungroup***.

Lock Command

You can lock any object or short cut. Once it is locked you cannot modify, select, translate, rotate or in any way change the position or shape of a locked object.

Tip:

The lock command not only prevents any changes to the locked object, it prevents them from being selected.

Using the Lock Command

1. Select the objects to be locked.
2. Choose ***Layout>Group>Lock***.

If you want to protect an entire document from modification:

1. Choose ***Edit>Select All***.
2. Choose ***Layout>Group>Lock***.

Selected objects can also be locked by clicking the *locked* box in the Edit Objects dialog box.

Unlock Command

You can unlock any Designer Elements program object or short cut that has previously been locked.

1. Choose ***Layout>Group>Unlock***.
2. Select the locked object by clicking on the object or dragging a selection fence around it. The locked object is unlocked and selected.

Arrange Command

The *Arrange* command in the Layout menu allows you reposition overlapping objects. This is a common command for graphics programs and is helpful in editing complicated drawings.

This command displays a submenu of the following commands: *Move Forward*, *Move to Front*, *Move to Back* and *Move Backward*.

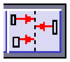
Move Forward Moves the selected object up one position in the display.

Move to Front Moves the selected object to the top (or in front) of all other objects in the display.

Move to Back Moves the select object to the back of the display.

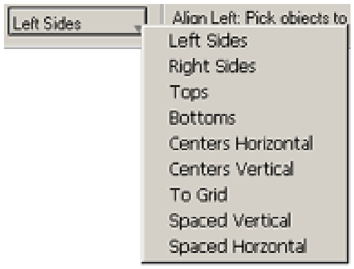
Move Backward Moves the selected object down one position in the display.

Align Command

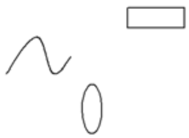


This command in the Transformation tool palette moves selected objects, including text, relative to other objects.

The *Align* command includes a submenu with these options, *Left Sides*, *Right Sides*, *Tops*, *Bottoms*, *Centers Horizontal*, *Centers Vertical*, *To Grid*, *Spaced vertical and Spaced Horizontal*.



The following figure illustrates each of these options.



Left Sides Aligns the left sides of selected objects.

Right Sides Aligns the right sides of selected objects.

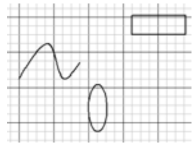
Tops Aligns the tops of selected objects.

Bottoms Aligns the bottoms of selected objects.

Centers Horizontal Aligns the center points of selected objects to the same x location.

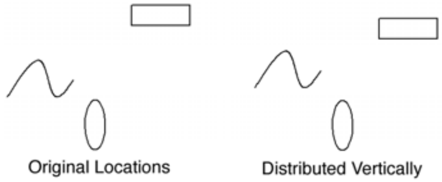
Centers Vertical Aligns the center points of selected objects to the same y location.

To Grid Aligns the left and bottom portion of an object to the closest grid point. The grid does not have to be displayed to use this option.



Spaced Vertical Equally distributes the selected objects relative to each other.

Spaced Horizontal Equally distributes the selected objects relative to each other.



Using the Align Command

1. Select the objects to align.
2. Then choose the alignment tool from the Transformation tool palette. The Message Line reads: *Enter Alignment Point*.
3. Choose the point at which you want the objects to align.

Object Commands

These commands allow you to modify various aspects of your geometry including direction, type and size.

Change Direction

This command in the Edit menu reverses the orientation of curves and surfaces.

You may sometimes find it necessary to change the direction of curves to straighten twisted surfaces. Changing the direction of surfaces reverses their normals which can change the way rendered surfaces respond to lighting.

The graphic below shows the direction of a surfaces using the ***Edit>Change Direction*** command. The right graphic shows the changes made by using *Change Direction*.

Using the Change Direction Command

1. Select the object whose direction you want to change.
2. Click ***Edit>Change Direction***.

This Designer Elements program displays temporary arrows along the object to show its direction. You can pick ***View>Redraw Screen*** to make disappear.

Referral:

This command can impact the rendering of your object. See Rendering Options and Flip normals in Chapter 33 for more information.

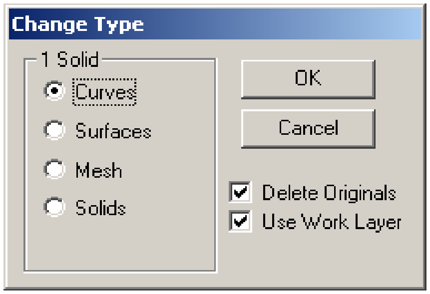
Change Object Type

This command in the Edit menu enables you to change an object's type. This is helpful for editing as well as exporting geometry to software that may not support a particular type of entity. After using this command for a particular operation, this Designer Elements program remembers the selected conversion type (Curves, Surface, Mesh or Solids for a Solid object) until you change the option type.

With this command you can convert:

- Solids to meshes, surfaces or curves
- Surfaces to meshes, surfaces or curves
- Curves to lines, polylines or splines

When you select this command a dialog box similar to the one below appears:



The dialog box contains the following standard options:

Selected Object section — This section contains the conversion options. The options differ depending on whether you select a wireframe, surface or solid object.

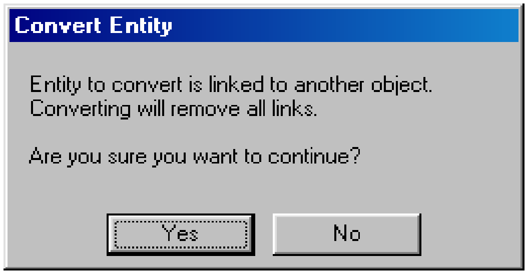
Delete Originals — Selecting this box deletes the original objects following the object conversion.

Use Work Layer Selecting this box places converted objects on the work layer.

OK Click this button to convert the object.

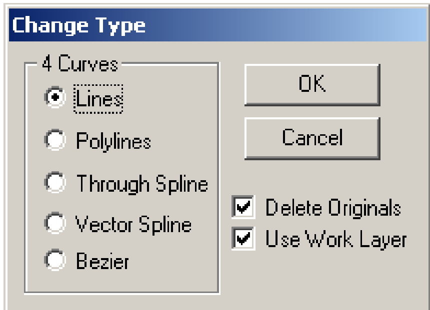
Cancel Click this button to Cancel the command.

If you are converting a model containing links before the conversion proceeds, the following dialog box displays asking you to confirm your awareness that the links will be removed by completing the conversion.



Changing Curves

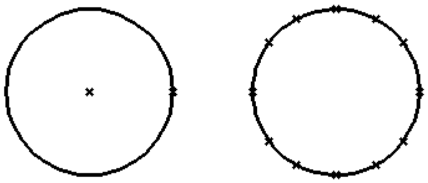
When you choose the *Change Object Type* command after selecting a curve this dialog box appears:



You have the option to convert your curve to Lines Splines. If you want to delete the original curve click the *Delete Originals* check box.

Curves to Lines Option

When the *Lines* option is selected you can convert a curve or set of curves to lines. You can use this command to change a spline into a collection of lines or smart polygons into individual lines.



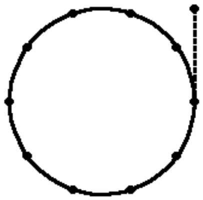
If the *Delete Originals* option is checked this Designer Elements program removes the parent curve after exploding it, leaving only the curves.

Curves to Polylines Option

When the *Polylines* option is selected, you convert curves or a set of curves into polylines. The graphic here is an example of a circle converted into a polyline. Unlike the line option which would convert a circle into multiple line segments, this option converts the circle into one polyline.

Curves to Through Spline Option

When the *Through Spline* option is selected you convert a line, conic, arc, circle, or ellipse into a Through spline. A Through spline's shape, slope and control points can be edited. This Designer Elements program fits the resulting spline to within a 0.001 drawing unit tolerance of the original curve. If the *Delete Originals* option is checked, this Designer Elements program removes the parent curve after exploding it, leaving only the splines.The graphic here is an example of a circle converted into a Through spline.



Curves to Vector Spline Option

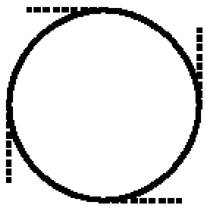
When the *Vector Spline* option is selected you convert a line, conic, arc, circle, or ellipse into a vector spline. A vector spline's shape, slope and control points can be edited. This Designer Elements program fits the resulting spline to within a 0.001 drawing unit tolerance of the original curve. If the *Delete Originals* option is checked, this Designer Elements program removes the parent curve after exploding it, leaving only the splines.

The convert curve command is also useful for removing excessive numbers of control points from constrained splines. The left graphic below shows a spline containing 50 points. The *Change Object Type* command reduces that number to five while maintaining a tolerance of 0.001 inches shown in the right graphic.

After selecting a type click OK and the object will convert.

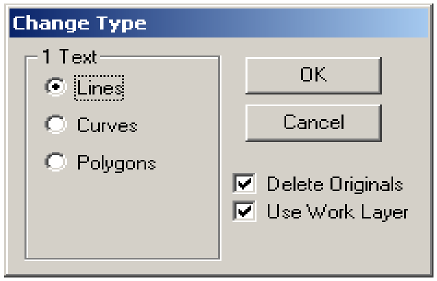
Curves to Bezier Option

When the *Bezier* spline option is selected you convert a line, conic, arc, circle or ellipse into a bezier spline. The bezier splines shape can be edited by moving the control points and the slope of the spline at those points. If the *Delete Originals* option is checked, this Designer Elements program removes the parent curve after exploding it leaving only the splines.



Converting Text to Curves

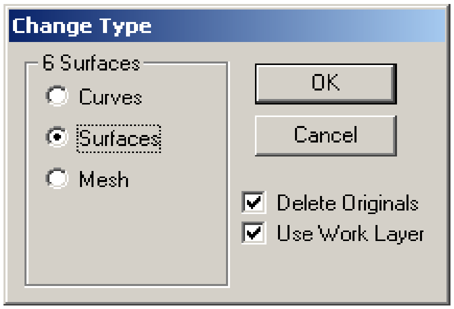
1. Create text using one of the Text tools.
2. Choose ***Edit>Change Object Type***.
3. In the Change Type dialog box choose curves.
4. Click OK.



Each letter of the text converts into a closed grouped curve. You can now perform any curve operation on the text curve.

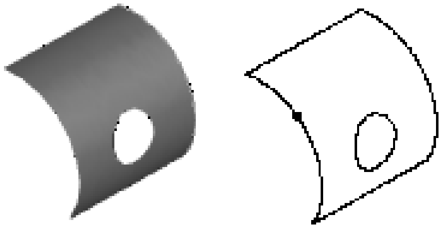
Changing Surfaces

When you choose the *Change Object Type* command after selecting a surface, this dialog box appears:



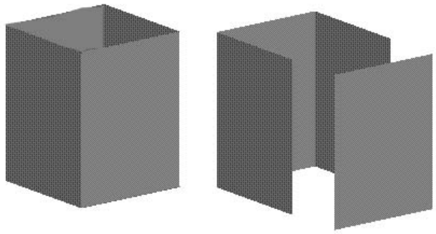
Surface To Curves Option

If you select the *Curves* option this Designer Elements program converts the surface into a collection of editable vector splines. You can modify the shape, slope and location of the spline control points. If the *Delete Originals* option is checked, this Designer Elements program removes the parent surface after exploding it leaving only the curves. The resulting curves may consist of lines, arcs, circles, ellipses or splines.



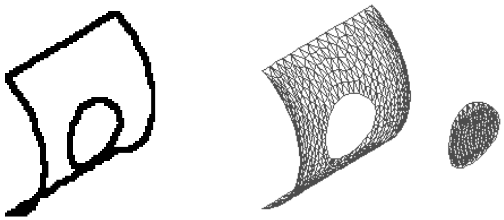
Surface To Surfaces Option

Selecting the *Surfaces* option explodes a surface composed of multiple faces into discrete surfaces. Once the surface is exploded you can edit individual surfaces; move them to a different layer, transform or remove them.



Surface To Mesh Option

If you check the *Mesh* option the surface is converted to a triangular mesh. Meshes are useful for sharing this Designer Elements program data with applications that cannot import precise surfaces. Exploding surfaces to mesh allows you to edit individual vertices.



The precise mathematical representation of a solid or surface must often be converted into a collection of imprecise planar facets. These facets, for example, may be used to export a model to the STL format, and when changing a solid or surface

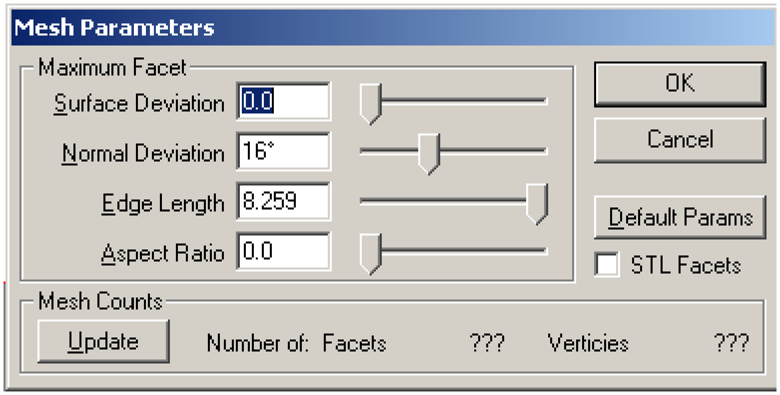
(**Edit>Change Object Type**) to a mesh. The amount error that results from this conversion is controlled by the settings in the mesh parameters dialog box.

During the conversion vertex points are distributed on the surface or solid. These vertices are then grouped into 3-sided and 4-sided facets. The conversion is deemed acceptable when the generated vertices and facets satisfy the settings. The five available settings are: Surface Deviation, Normal Deviation, Edge Length, Aspect Ratio and STL Facets. These settings are defined in the sections below.

Change the facet settings as needed in the dialog and then click the Update button to see the number of facets and vertices generated. Determining the combination of settings that will work for a given situation can be a little bit of an art. If one setting becomes too tight the other settings will have no effect. If one setting becomes too loose, it will have no effect.

Keep in mind that under “real” circumstances the settings are used by the faceting algorithms if possible. It is often not possible to satisfy all settings simultaneously. In this situation the algorithm decides which settings to “loosen”.

When you explode a surface into a mesh, the mesh dialog box appears with characteristics for specifying the resolution of the mesh.



The dialog box includes:

Aspect Ratio Sets the maximum ratio between triangle edges. It prevents forming long narrow triangles. Enter a value in the field or use the slide to set the value.

Surface Deviation Sets the maximum acceptable distance between the facet and the surface represented by the mesh.

Enter a value in the data field or use the slide to set the value.

Normal Deviation

Sets the maximum angular deviation between adjacent facets. 20 degrees is the default. Use 10 for extremely dense meshes. Enter a value in the data field or use the slide to set the value.

Edge Length

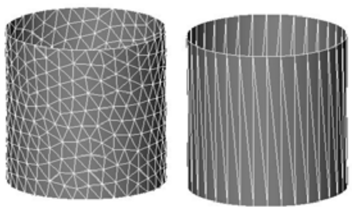
Sets the maximum acceptable edge length for facets. Enter a value in the data field or use the slide to set the value.

Mesh Count

This section includes the Update button and information on the Number of Facets and Vertices. Click the Update button to calculate the approximate number of facets based on mesh characteristics. The Number of Facets and Vertices information display questions marks until you click the Update button. The calculated values replace the question marks.

Smoothing

Selecting this option creates facets that are regularly spaced. The left graphic here shows an object without smoothing. The right shows the object with mesh smoothing.



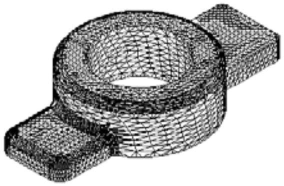
Default Params

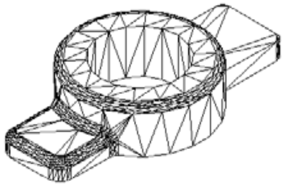
Click this button to revert the mesh parameters to this Designer Elements program default values.

STL Facets

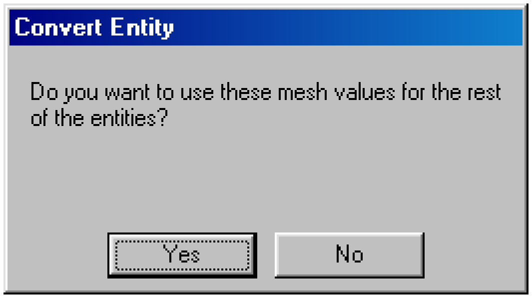
This setting will force the facets generated to be suitable for stereolithography usage. This setting is usually used when exporting STL files.

The mesh in the left graphic below was created with a *Normal Deviation* of 10 and a *Max Edge Length* of 0.1. The right graphic was created with a *Normal Deviation* of 20 and a *Max Edge Length* of 0.





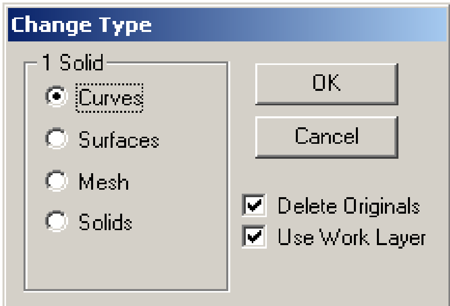
After setting the desired values, click OK and the object will convert. If you select more than one surface to convert the following dialog box appears:



Click Yes to use the same mesh conversion values for all selected surfaces.

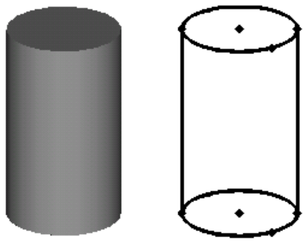
Changing Solids

When you choose the *Change Object Type* command after selecting a solid this dialog box appears:



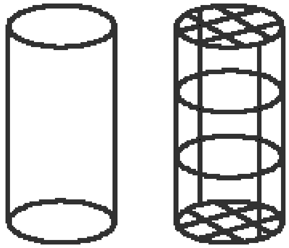
Solid To Curves Option

If you select the Curves option, this Designer Elements program converts the solid into a collection of editable vector splines. You can modify the shape, slope and location of the spline control points. If the *Delete Originals* option is checked, this Designer Elements program removes the parent solid after exploding it leaving only the curves. The resulting curves may be lines, arcs, circles, ellipses or splines.



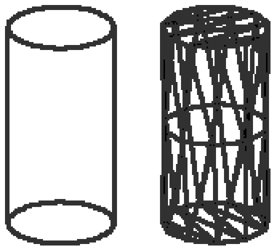
Solid to Surfaces Option

Selecting the Surfaces option explodes a solid into discrete surfaces. Once the solid is exploded you can edit individual surfaces, move them to a different layer, transform or remove them.

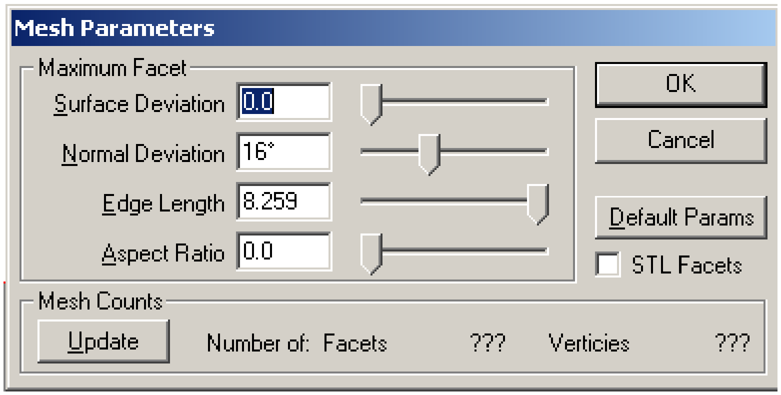


Solid To Mesh Option

If you check the *Mesh* option the solid converts to a triangular mesh. Exploding solids to meshes allows you to edit individual vertices.



When you explode a solid into a mesh, the mesh dialog box appears with characteristics for specifying the resolution of the mesh:

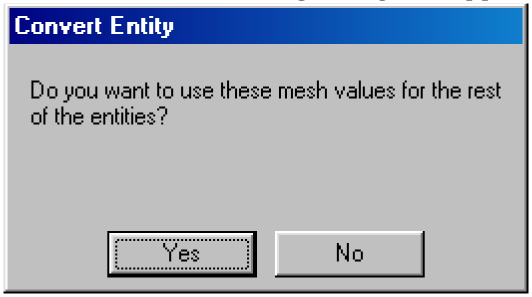


The dialog box includes:

Aspect Ratio	Sets the maximum ratio between triangle edges. It prevents forming long narrow triangles. Enter a value in the data field or use the slide to set the value.

Surface Deviation	Sets the maximum acceptable distance between the facet and the surface represented by the mesh. Enter a value in the field or use the slide to set the value.
Normal Deviation	Sets the maximum angular deviation between adjacent facets. 20 degrees is the default. Use 10 for extremely dense meshes. Enter a value in the data field or use the slide to set the value.
Edge Length	Sets the maximum acceptable edge length for facets. Enter a value in the data field or use the slide to set the value.
Mesh Count	This section includes the Update button and information on the Number of Facets and Vertices. Click the Update button to calculate the approximate number of facets based on mesh characteristics. The Number of Facets and Vertices information display questions marks until you click the Update button. The calculated values replace the question marks.
Smoothing	Selecting this option creates facets that are regularly spaced. (See the graphic included with the Surface to Mesh conversion in an earlier section for a smoothing effect example.)
Default Params	Click this button to revert the mesh parameters to this Designer Elements program default values.
STL Facets	This setting will force the facets generated to be suitable for stereolithography usage. This setting is usually used when exporting STL files.

After entering the desired values, click OK and the object will convert. If you select more than one solid to convert, the following dialog box appears:



Click Yes to use the same mesh conversion values for all selected solids.

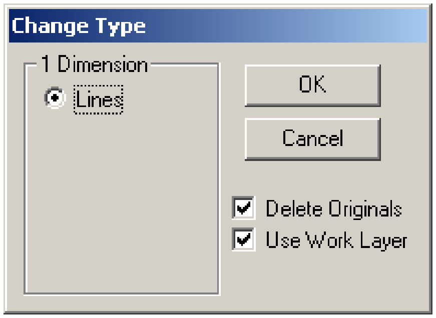
Solids To Solids Option

You can use the Solids option to break one solid up into several smaller units. This option comes in handy if you want to edit only one part of a larger solid.

If you want to find out the mass properties of several solids as if they are combined into one, you can first unite them and analyze their properties. After your measurement is completed use the *Change Object Type Solid* option to break the part back into individual pieces.

Change Dimensions

When you choose the *Change Object Type* command after selecting a dimension this dialog box appears.



Use this to change dimensions into lines and text. This is helpful when exporting files with dimensions that may not be supported by the other software.

Simplify Object

This command in the Edit menu will examine the selected objects and if the entity falls within tolerances, will do the following:

- Convert zero length curves to points
- Convert circular splines to arcs or circles
- Convert straight splines to lines

• Convert NURB surface or solid faces to analytics

Potential NURB shapes converted to analytic shapes include cylinders, cones, speres, tori and planes. Candidate shapes for simplification include data imported via IGES, Step, Rhino and Alias. Also some surfaces created in this Designer Elements program are initially created as NURBS and may simplify to an analytic shape.

Some advantages of analytic representation of shapes over NURBS are analytic operations are considered exact whereas NURBS are precise. Analytic operations execure much faster internally and take up less memory in RAM and less space when the file is saved. For example, an IGES file can be reduced by half after using the Simplify Object command.

Note: Using the Simplify Objects command may cause loss of part history on complex objects.

Edit Objects

This command in the Window menu edits selected objects by changing individual characteristics such as: weight, layer, resolution and other specifications.

You can display the Edit Objects dialog box in two ways:

• Select the object and choosing ***Window>Edit Objects***

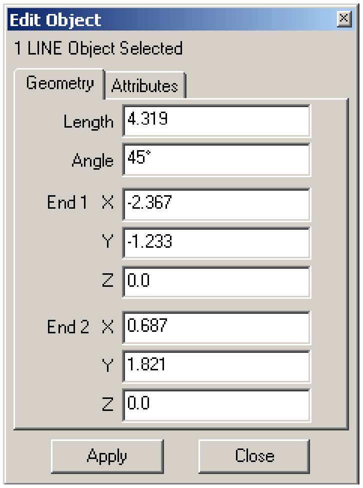
• Double-click on the object.

Note: You cannot double-click on editing operations performed on a solid such as blending, placing holes, shelling etc. except from within the Design Explorer (**Cobalt™ and Xenon™ only**). (See a later section for information on the Design Explorer.)

Changes made through this dialog box can be reversed with the *Undo* command.

When you choose Edit Objects a dialog box similar to the one here displays containing between two and four possible tabs: *Geometry*, *Display*, *Attributes* and *Material*.

Geometry Tab



All tools that create geometry will include a geometry tab in the Edit Objects dialog box.

The specific characteristics listed depend on the object selected. The object selected in this case was a line and includes the following characteristics: *Length*, *Angle*, *End 1* (X, Y and Z values) and *End 2* (X, Y and Z values). All values appear in this dialog box as whole numbers and decimals, depending on the decimal setting in Preferences.

Entering Values

You can enter values in this dialog box in any units desired. These will automatically be converted into the units you set in Preferences. See Chapter 6.

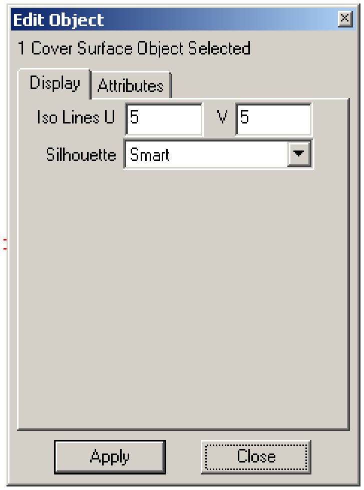
When units are set to feet and inches (in the Prefernces dialog box) it's important to be aware of the following rules:

- All numbers are assumed to be feet unless accompanied by the unit symbol like " for inches. Entering a 1.5 in the field is read as 1.5 feet. If you want 1.5 inches, enter 1.5", 1.5i, 1.5in, 1.5 inch, etc.
- If you want to enter fractions of inches, each entry must include the unit symbol. For example, 5 feet 6 5/8 inches must be entered 5 6" 5"/8. Internally this is converted as 5' + 6" + 5"/8. If the inch symbol is not included with the fraction, 5/8 will be interpreted as a fraction of a foot.

Drawing views created for Layouts have their own Geometry page. Some editing operations like *Split Solid* contain no specific geometry characteristics and therefore do not have a Geometry page.

Every chapter in this manual that deals with this Designer Elements program objects includes a *Geometric Characteristics* section listing all characteristics specific to the object. Those features that do not have specific characteristics are noted accordingly.

Display Tab



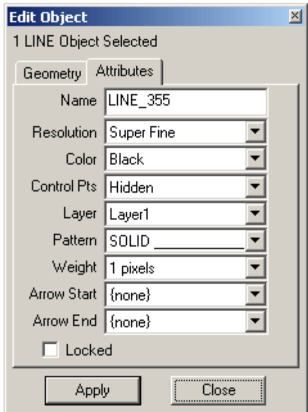
This page in Edit Objects appears for surface and solid objects only.

The graphic shows the characteristics included in the Display page for a surface object and includes: *Iso Lines* and *Silhouette*.

The number of Iso Lines allows you to control the isopram lines drawn for a surface. These Iso (isopram) lines are constant parameter curves that lie on a surface, typically defined in parameter space. The parameter space coordinate system uses *U* and *V* coordinates, where U = horizontal and V = vertical. A zero (0) in both fields turns off Iso lines. The appropriate *U/V* values may enhance the visual appearance of the surface at the expense of drawing speed. The letters *U* and *V* are industry standard space coordinate identifiers.

The *Silhouette* option controls whether this Designer Elements program displays silhouette edges of objects. Silhouette edges are view dependent and can cause a significant reduction in drawing speed. The pull-down menu displays the options: On, Off and Smart. Smart Silhouettes display a silhouette only if it does not degrade the display performance.

Attributes Tab



This tab displays a standard list of characteristics for the selected object.

The attributes include:

Name — Displays the name of the object. You can enter a new name if you wish.

Resolution — Displays the resolution setting for the object. You can select any of the options listed in the pull-down menu. See the Object Resolution section for more information.

Color Displays the color of the selected object. You can choose any color for your object.

If you object is on a layer with an override color, the object will display in the override color but the Edit Objects color field will display the actual color of the object.

Control Pts Displays the status of the control points, hidden or visible. You can choose either setting.

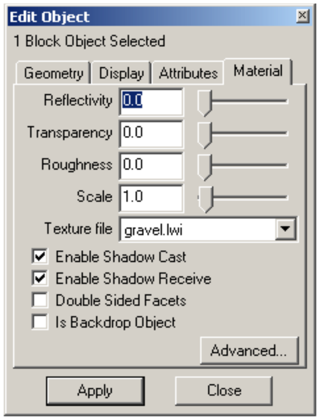
Layer Displays the layer where the object is located. You can place the object on any available layer.

Locked Choosing this option locks the object preventing it from being selected or modified. This is same operation that occurs when choosing ***Layout>Group>Lock***.

For wireframe objects this tab also includes: Pattern, Weight, Arrow Start and Arrow End.

Material Tab

This tab appears for objects that have rendering materials applied to them.



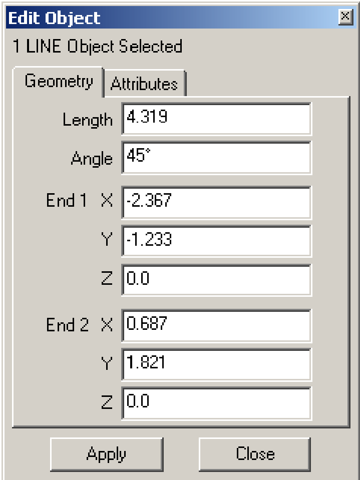
The material characteristics include the following:

Reflectivity This field sets the material's reflectivity. Values can be entered between zero (0) and one (1). Entering a zero in the field renders a flat finish. Entering a one in the field renders a mirrored finish.

Transparency	This field sets the transparency of the material. Values can be entered between zero (0=transparent) and one (1=opaque). This option only works correctly when objects are rendered with ray trace rendering commands.
Roughness	This field sets the roughness of the material. Values can be entered between zero (0) and one (1).
Scale	This field sets the scale of the material. Typically scale increases the size of the detail. Values can be entered equal to or greater than zero (0).
Texture File	This field displays the selected material. The pull-down menu lists all images in the texture folder. For those materials that do not support textures this menu is not available.
Enable Shadow Cast	This check box allows you to specify if an object casts shadows. This is valuable for reducing the shadows present in complex drawings. When checked the selected object casts shadows.
Enable Shadow Receive	This check box allows you to specify whether an object will receive shadows from other objects. This ability is valuable for reducing the number of shadows present in complex drawings. When checked the selected object receives shadows.
Double Sided Facets	This check box provides you with additional rendering control for surfaces. Objects with normals facing away from the line of sight are not rendered. When checked all objects are rendered regardless of the normal direction. Select this option for objects that cause light refraction like glass. When left unchecked facets on the back side of the glass are ignored resulting in an inaccurate rendering.
Is Backdrop Object	This check box allows you to optimize rendering calculation time for an object like a wall, that functions only as a backdrop for other objects. Since

	the object automatically receives light due to its large size, this Designer Elements program does not need to spend much time performing light ray calculations.
Advanced button	Clicking this button displays the Render Material Settings dialog box. Use this dialog to change the advanced rendering settings used on the selected object. See Chapter 33 for a detailed description of these settings

Using the Editing Objects Command



1. Select an object to edit.
2. Choose ***Window>Edit Objects*** and the dialog box appears.

   The dialog box contains between two or four possible tabs depending on the object you select, *Geometry*, *Display*, *Attributes* or *Materials*.
3. Click on the tab containing the desired information.
4. Change the information in an entry box by double-clicking the entry data field and typing a new entry. Do not press ENTER (Windows) or RETURN (Macintosh). Many data fields have pop-up menus for selection. Press the down arrow and then drag to the selection you want.
5. Make changes in other entry fields as required.
6. Click Apply and the changes automatically occur on your selected geometry.
7. Click Close to close the Edit Objects dialog box.

You can use *Undo* or *Redo* to reverse changes made through this dialog box.

Coordinates are measured in the units set in the *Units* page of Preferences.

If you are doing a lot of editing you may want to leave the Edit Objects dialog box displayed. That way you can select the object, make the changes in the dialog box, click Apply and then go on to the next object.

If more than one object is selected when you choose *Edit Objects*, only the common information is displayed. Entries are blank when the information isn't common. For example, if you select two concentric circles, the center will be displayed in the Edit Objects box but the diameter box will be blank.

If you select several objects and then choose *Edit Objects*, all of the objects will reflect the changes you make in the dialog box. For example, if you select all dimensions and change the text entry in the Edit Objects dialog box to 2, all the dimensions will display a 2.

Moving an Object to a Different Layer

1. Select the object(s) to be moved.
2. Choose ***Window>Edit Objects***.
3. Click the arrow for the Layer entry field.
4. Pick the layer from the list.
5. Choose Apply.

Links

Cobalt and Xenon Only

Many of the objects in this Designer Elements program are associated with other objects in parent-child relationships. Associative objects retain a history of how they were created and a set of rules that define their geometric description. If you modify a parent object (an object that defines another object), this Designer Elements program automatically regenerates the dependent or child object. For example, consider a skinned surface that is defined by two lines. If you modify one of the lines, the skinned surface will automatically regenerate.

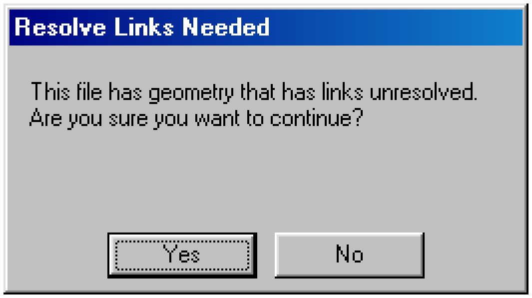
This Designer Elements program provides two commands for dealing with Links: *Resolve* and *Remove*.

Resolve Links Command

This command is in the Edit menu. As you modify an entity that is part of a parent-child relationship the word, *Edit* in the menu bar and the command, *Resolve Links* highlight in red.

In most cases, this Designer Elements program will resolve the links in a parent/child relationship automatically and the words will un-highlight when finished. If the words stay red, select ***Edit>Resolve Links*** to force the program to resolve the relationship.

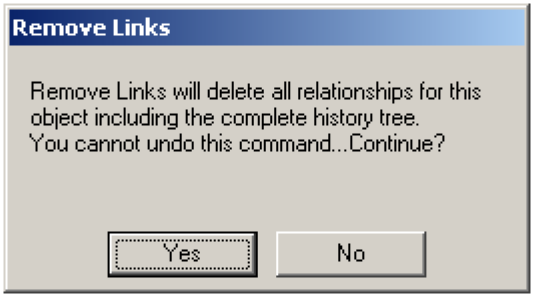
If you attempt to save a file containing links that are not resolved, the following warning appears:



Remove Links Command

The *Remove Links* command in the Edit menu deletes all associative links between a parent and child.

When you select the command the following warning appears:



This warning reminds you that the selected object has a parent/child relationship to another object and that deleting this object not only deletes the history tree but cannot be undone.

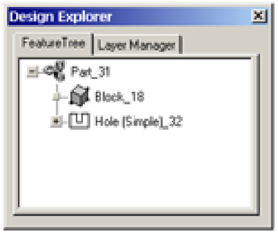
Press the Yes button to continue the *Remove Links* command or No to discontinue.

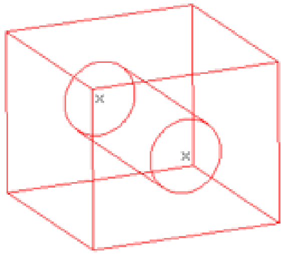
Referral/Tech Note:

Do not use the *Remove Links* command to remove a feature from the history tree. This removes all parent/child associativity for your geometry. Use the Design Explorer instead. See a later section for information on the Design Explorer.

Design Explorer

This Designer Elements programs Design Explorer is a valuable feature for traversing your drawing's design history. It offers a Windows-style tree structure with expandable branches that display, in a hierarchical fashion, the associative geometry used in the creation of all of the entities in the drawing.





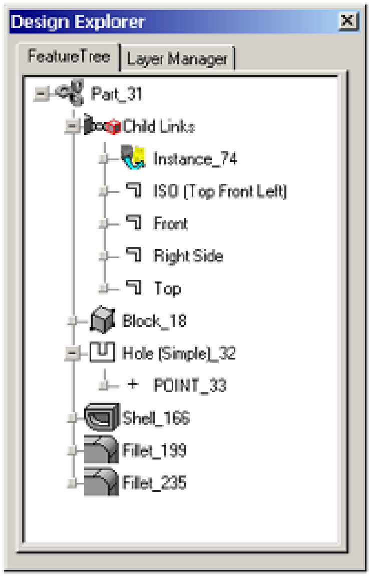
The figure shows a solid cube with a hole in one face. The Design Explorer window identifies the entity as a part and shows that it consists of a block, the cube and a hole feature. As you click on items in the tree the corresponding part in the drawing will highlight.

If you select an item in the Design Explorer when the Edit Objects dialog box is displayed, the dialog box automatically updates to reflect the selected item.

Child Links

The child links feature organizes all persistent parts of your model in the Design Explorer. If you place material or decals on your model or create drawing views they group separately from any geometry operations under the *Child Links* category.

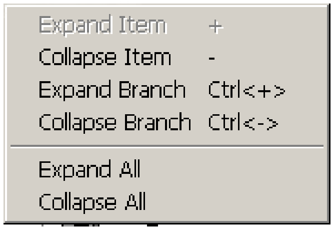
This frees you up to modify your part without having to replace material or recreate views.

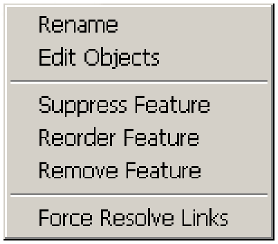


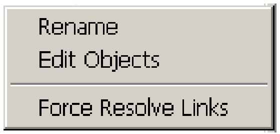
Cobalt and Xenon Only

Design Explorer Commands

The Design Explorer also includes several of its own menu of commands. For Windows users, (and Macintosh users with a two button mouse) click the right mouse button on the feature name to display the menu. For Macintosh users, hold down the CONTROL key and click on the feature. Depending on where you right click and when, one of the following menus appear:

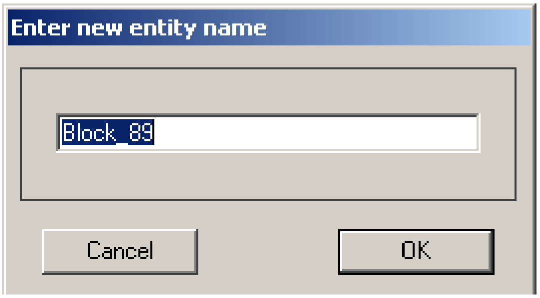






Rename

The *Rename* command lets you rename any of the entities. It displays this dialog box.



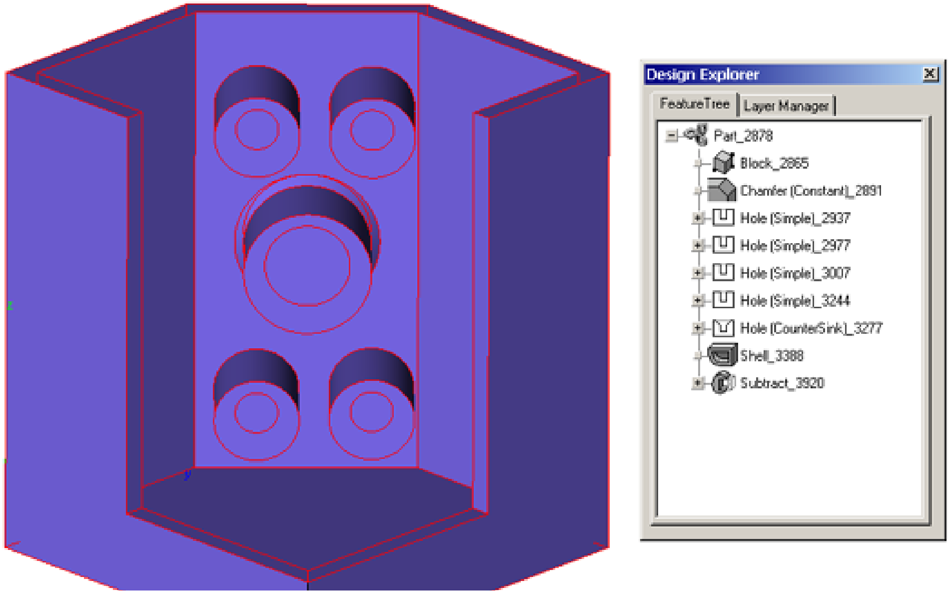
Enter the new name and click OK.

Suppress Feature/Unsuppress Feature

These commands allow you to turn off or on a particular feature in the solid history tree. They toggle between each other depending on whether an operation is suppressed.

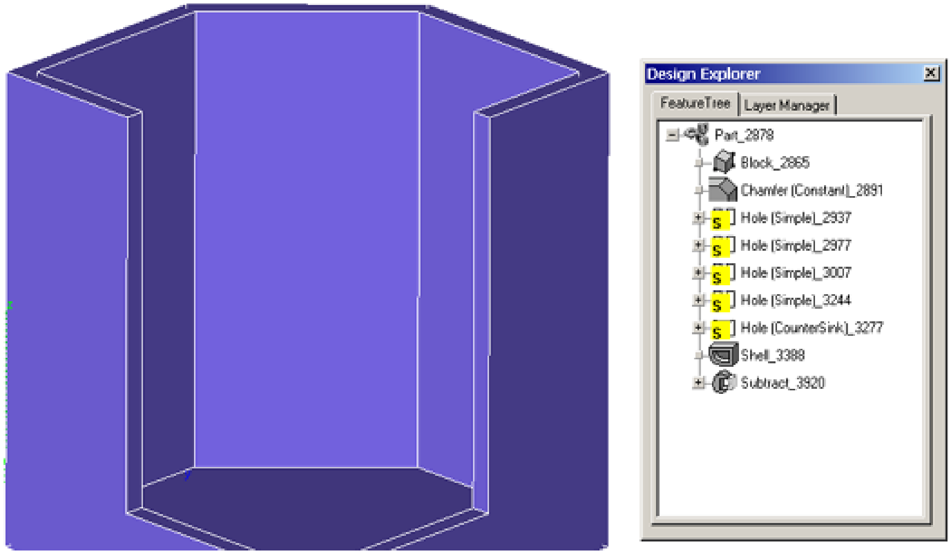
Cobalt and Xenon Only

When a feature is suppressed, this Designer Elements program rebuilds the solid as if the operation never occurred. This feature can be unsuppressed and reintegrated into the part at a later time. The graphic below shows a part and the Design Explorer open.



Cobalt and Xenon Only

This next graphic shows the same part with all of the holes suppressed. Notice the “S” displayed over the operation icon in the Design Explorer.



Some uses for feature suppression include the following:

- FEA Models: when creating FEA (Finite Element Analysis) models you may not want certain features included in the analysis.
- Part Regeneration: removing certain features may speed up rebuilding the part, including display time.
- Shelling: if a part fails to shell you can turn off blends and other features that may prevent a successful shelling operation. Then, using the *Reorder Feature* command, you can move the shelling operation before the suppressed feature in the history tree so that shelling succeeds. Finally, you can unsuppress the selected feature using the *Unsuppress Feature* command.

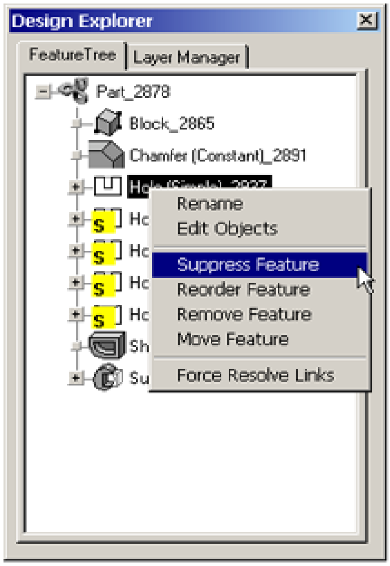
Using the Suppress Feature Command

Cobalt and Xenon Only

1. Select the model which contains the feature you want to suppress.
2. Choose ***Window>Design Explorer*** to display the history tree.
3. Expand the tree to display the desired feature by clicking on the plus (+) sign to the left of the part name.
4. Select the desired feature.

   The feature highlights.
5. Click the right mouse button (Windows) or press the CONTROL key (Macintosh) to display the Design Explorer menu.
6. Choose the *Suppress Feature* command.

   The feature suppresses and your model regenerates without the feature. The Design Explorer now displays an “S” over the operation icon.

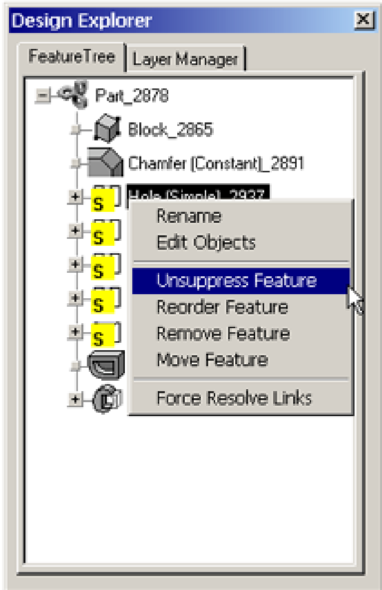


Using the Unsuppress Feature Command

The *Unsuppress Feature* command replaces the *Suppress Feature* command in the Design Explorer menu for suppressed features only.

1. In the Design Explorer select the suppressed feature.
2. Click the right mouse button (Windows) or press the CONTROL key (Macintosh) to display the Design Explorer menu.
3. Choose the *Unsuppress Feature* command.

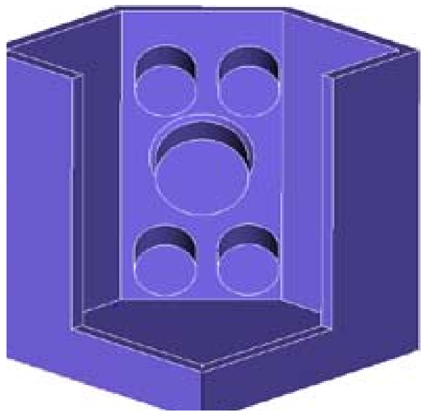
   The feature reappears and the model regenerates. The feature icon no longer displays an “S.”



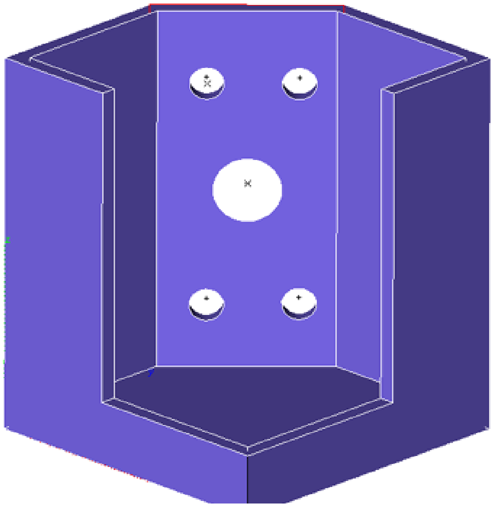
Cobalt and Xenon Only

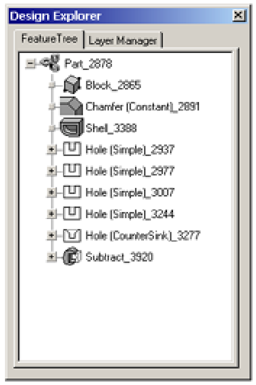
Reorder Feature

Sometimes you may need to modify your part by rearranging its features. This command allows you to move features up or down in the history tree. In the graphic here, the shell operation occurred after the holes were placed.



Using the *Reorder* command, the shelling operation was moved before the hole operation, resulting in the model shown here.





Cobalt and Xenon Only

Using the Reorder Feature Command

1. Select the part.
2. Open the Design Explorer and display the tree to expose the feature you want to reorder.
3. In the Design Explorer menu select the *Reorder Feature* command.

   The Message Line reads: *Feature Reorder: Click on new location in Design Explorer.*
4. Click on the operation below which you want to move the selected operation.

   The operation moves and the part regenerates.

Move Feature

The *Move Feature* command allows you to reposition holes and bosses within the assembly.

Using Move Feature Command

1. Select the feature in the Design Explorer that you want to move. The feature is selected in your drawing area.
2. Display the Design Explorer menu (Right mouse button - Windows, CONTROL+mouse button - Macintosh).

Tech Note/Tip:

You cannot move or reorient a feature to any side of a solid that is not visible. The operation automatically believes you are choosing one of the visible faces. Rotate the solid so that the face is visible before choosing the Move Feature command.

3. Select the *Move* command. The Message Line tells you to select a new center for the feature.
4. Click a new center on your solid. The feature is repositioned.

Remove Feature

This command deletes a feature from the assembly.

Edit Parameters

This command displays the Edit Objects dialog box. You can also double-click any item in the tree to open Edit Objects.

Expand Item

This commands expands the part of the tree under the selected item.

Collapse Item

This command collapses the part of the tree under the selected item.

Cobalt and Xenon Only

Expand Branch

This command expands the entire branch that includes the selected item.

Collapse Branch

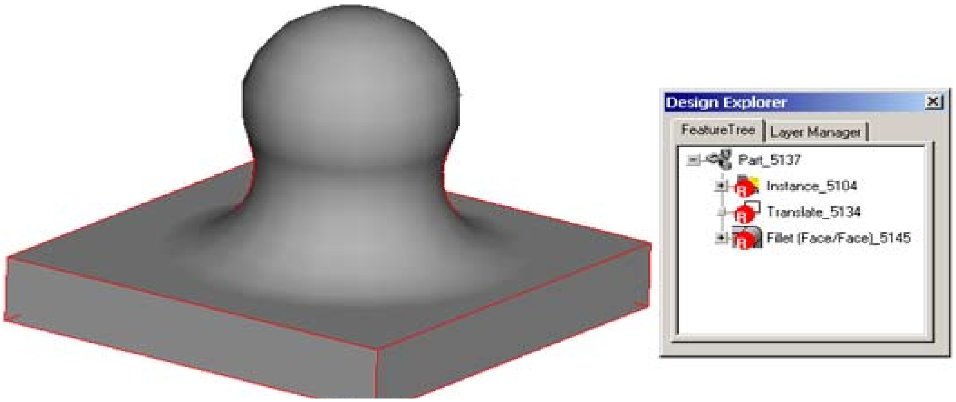
This command collapses the entire branch that includes the selected item.

Close Window

This command closes the Design Explorer.

Force Resolve Links

This command resolves all unresolved child geometry. If you select the part and display the Design Explorer, a red "R" appears over any child operation icons that is not resolved.

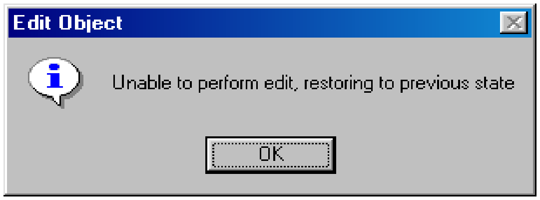


This will occur if you had turned off the *Enable Auto Resolve* option in the General page of Preferences. When activated, this Designer Elements program automatically resolves all child geometry when a parent is modified. When deactivated use the *Force Resolve Links* command to resolve child geometry.

Design Explorer Rules

Cobalt and Xenon Only

- Within the Design Explorer you cannot select a curve within a profile and edit its length through the Edit Objects dialog box. This attempt will make a later operation invalid. The message here appears when you choose Apply in the Edit Objects dialog box.



  Click OK to restore the original length.

  For example, if you want to lengthen a line that has been extruded to a solid, the change in length does not affect the lines connected to the edited line. The closed profile would be destroyed. To lengthen a line use the Move tool and select the endpoint of the line and move it. By default, it will select and move the endpoints of all lines connected at that endpoint.

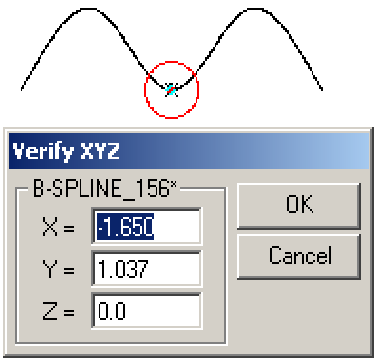
- The Design Explorer will only show one part at a time.

Verify Menu

The Verify menu gives you access to a variety of commands that provides information about your file or about specific objects.

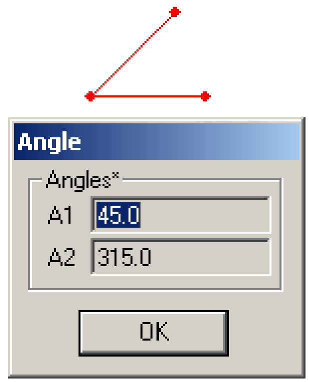
X,Y, Z Command

The *XYZ* command displays the model coordinate values of the selected point. You may edit and change a selected value.



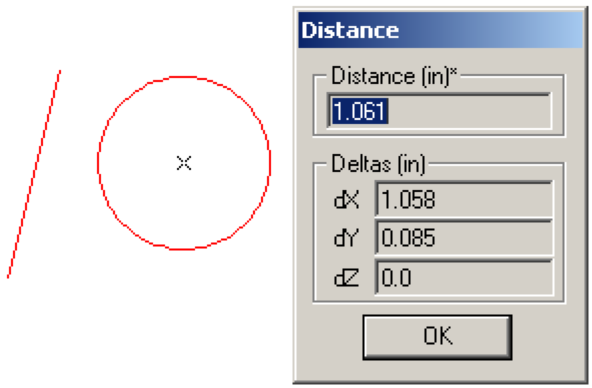
Angle 3 Pts Command

The *Angle 3 Pts* command calculates the angle formed by three points that you choose.



Distance Pt-Pt Command

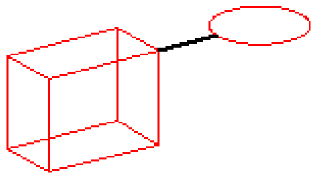
The *Distant Pt-Pt* command calculates the distance between two points taken from the drafting assistant.

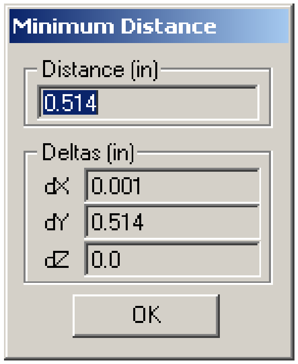


Minimum Distance Command

The *Minimum Distance* command calculates the smallest distance between two selected objects.

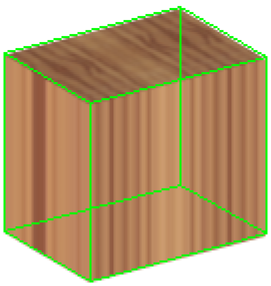
This command is useful for checking if two objects interfere with each other. It will calculate the minimum distance between any combination of curves, surfaces or solids.

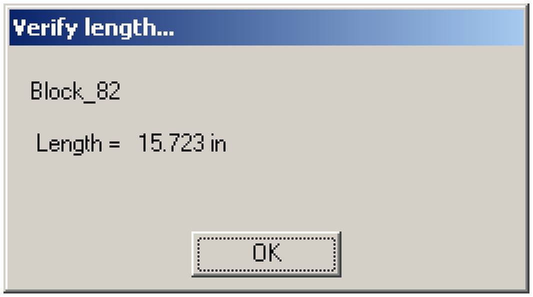




Length Command

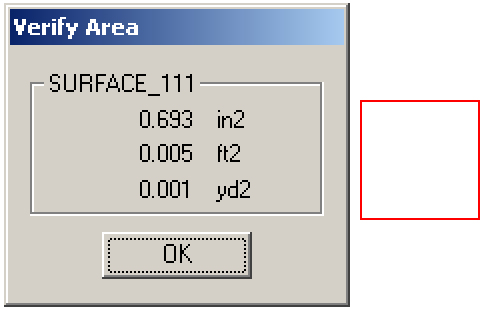
This *Length* command calculates the length of a line, spline, arc, circle, conic, surface and solids. In the case of surfaces and solids the command adds all edge lengths, providing a perimeter length value as shown in the graphic here.





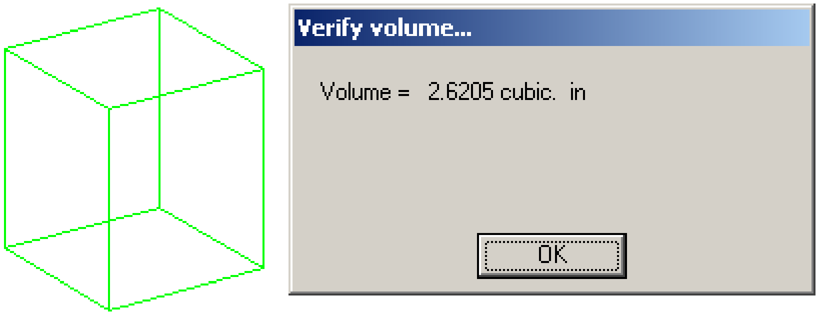
Area Command

The *Area* command calculates the area of closed curves or a surface. These curves include basic objects such as circles and ellipses but also splines connected to create an enclosed area.



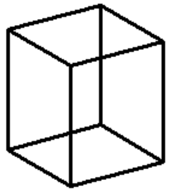
Volume Command

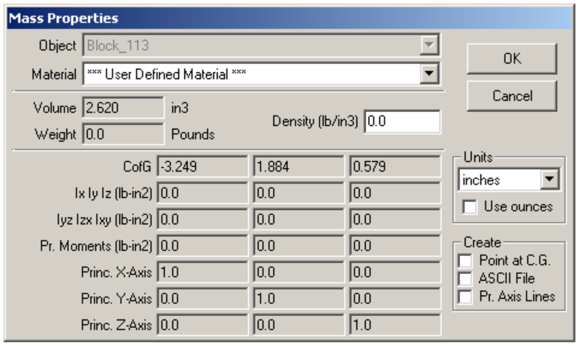
The *Volume* command calculates the volume of a closed object.



Properties Command

The *Properties* command calculates a solid's volume, center of gravity, principle moments, or moments of inertia.





The Properties dialog box includes the following sections:

Material

This pull-down menu allows you to specify a material type.

The menu includes User Defined, (Misc) Cast Iron (typical), Steel (SAE 1020, SAE 4130 normalized, SAE 4140, 18-8 stainless, Aluminum (2024-T3, 2014-T6, 7075-T-6), Magnesium (M-1/Am35, AZ61A/AMC575), (Titanium) Titanium (typical), Wood (Birch grain W, Birch grain X, White Oak grain W, White Oak grain X, Douglas Fir grain W, Douglas Fir grain X, Souther Pine grain W, Southern Pine grain X), Ceramic (Brick Soft, Brick Hard, Sewer Pipe, Glass), Aggregate (Concrete low strength, Concrete medium strength, Concrete high strength).

Tech Note:

This material selected here is set separately from material hatching used for geometry in a section view.

See Chapter 28 for more information on crosshatching and Chapter 36 for more information on section views.

Select one of these materials and the associated density displays in the Density field. For the User-defined material you can enter your own value.

Basic

This section includes the volume, weight and density of the selected object. The density is based on the selected material.

Advanced

This section includes the Center of Gravity, Moments of Inertia and axis information.

Create

This section includes three check boxes: ASCII file, Point at C.G and Pr. Axis Lines.

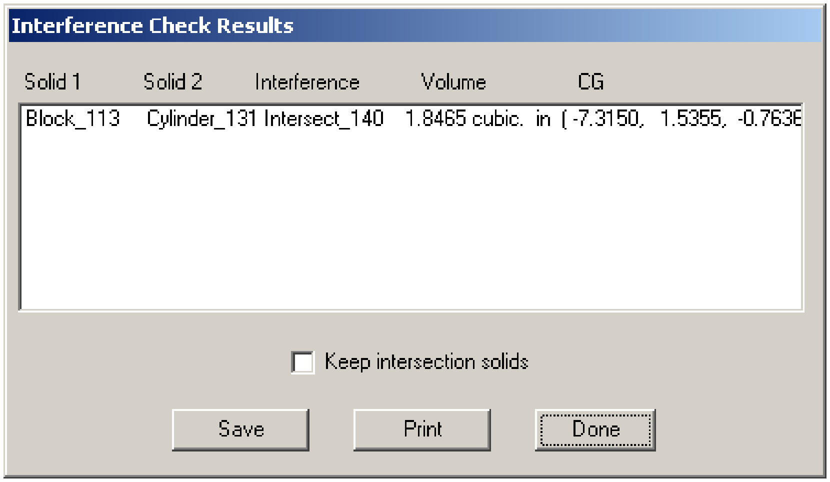
ASCII File - Clicking in the check box allows you to export the information contained in this dialog box to an ASCII file. When you click OK to close the Properties dialog box, the *Save document as* window appears asking for a name and location for the ASCII file.

Point at C.G. - Clicking in this check box, places a point at the Center of Gravity for the selected object. Click OK to close the dialog box and a C. G. point appears. The style of the point is based on the selected style for the Point tool. See Chapter 9 for more information.

Pr. Axis Lines - Clicking in this check box, gives you the axis direction vectors for the coordinate system where the products of inertia vanish.

Interference Command

This command in the Verify menu checks the interface volume of two or more solids to determine an intersection or interference. If an interference is found the Interference Check Results dialog box appears:



The dialog box contains the interference list window: *Keep intersection solids* check box, and the Save, Print and Done buttons.

Interference window This window lists the interference between the selected solids and includes Solid 1 and Solid 2 (the solids names), Interference (the interference solid name), Volume (the volume of the interference solid), and the CG (the center of gravity for the interference solid - x, y and z location).

Keep intersection solids When checked this option retains the intersection solids in your drawing.

Save Click this button to save the interference data to a log file which can be opened in any text editor.

Print Click this button to print the interference data.

Done Click this button to close the dialog box.

Using the Interference Command

1. Display your model as a wireframe.

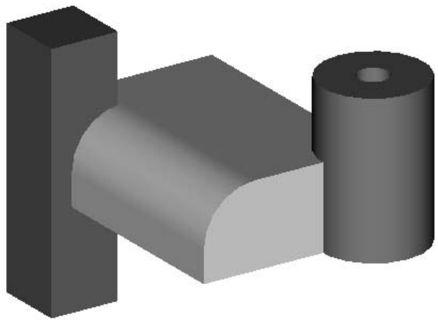
2. Select the solids on which you want to perform the check.
3. Choose ***Verify>Interference***.

   If an interference is encountered the Interference Check Results dialog box appears listing the interference and displaying the interference solids in black on your model.
4. If you want to retain these interference solids in your model check the *Keep intersection solids* box.
5. Click Save or Print if you would like to save or print the log of the interference solids.
6. Click Done to close the dialog box.

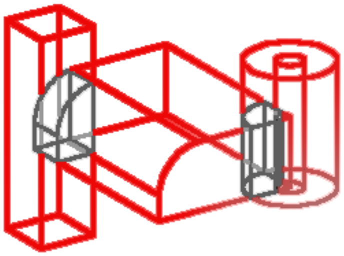
   If *Keep intersection solids* box is checked your model displays the intersection solids.

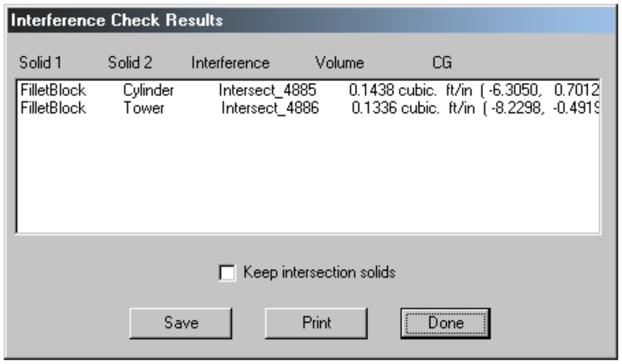
   Example:

   The model here displays three intersecting solids.

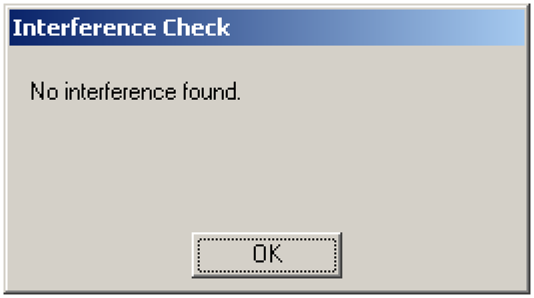


By choosing ***Verify>Interference*** on these selected models (shown below in wireframe), the Interference Check Results dialog box appears and the interference solids are shown in black.

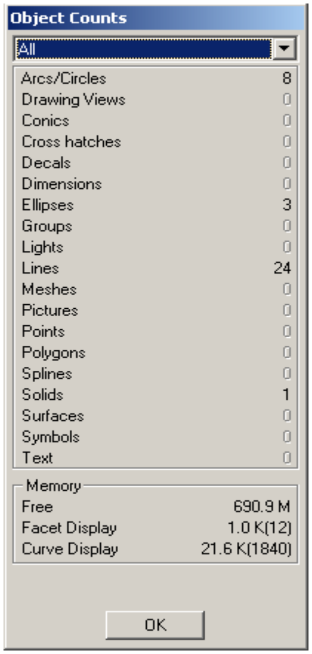




If no interference is found the following dialog box appears.



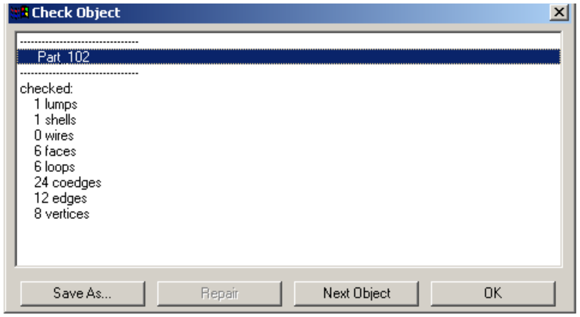
Object Counts Command



This command counts the number of objects in the current drawing.

For grouped objects, each individual object is counted as well as the group itself.

Check Object Command



The Check Object command examines an object for proper data structure, topology and issues geometric warnings associated with ACIS data. This includes checks for curves, surfaces, and solids. The tool is frequently used in conjunction with imported data created from non-ACIS kernels. Some of the many items checked include:

Data Structure Checks

- Entities have has appropriate child-level entity; e.g.: body has lump. Face has edges, etc
- Presence (non-NULL) and closure of back pointer from child to parent; e.g.: body's lump points to body.
- The coedge on spline surface has pcurve.

- Pcurve indexing (0/+-1/+-2) is appropriate.
- The pcurve has non-NULL 2D B-spline curve.
- If edge has non-NULL curve, then curve must have equation.

Topological Checks

- Loops must be closed in both the next and prev directions.
- Apex edge loops are correct.
- Coedge has a partner, except apex coedge
- All coedge partners point to same edge
- Sequential coedges share a vertex
- Edge is in exactly one of start and end vertex edge groups. For example, edge can be reached for 1 value of i using start()->edge(i)->coedge() and partner and next (or previous) pointers.

Geometric Checks

- Face gaps along shared edges and vertices
- Entities with geometry must have non-NULL geometry. For example, a face points to a surface.
- Analyzes a curve or surface for C1/G1 discontinuties
- Self intersecting surfaces
- Analytic surfaces have valid definitions
- Pcurve surface matches face surface (warning only if not equal since surface could be trimmed).
- Pcurve form is agrees with curve form, e.g. closed, open, periodic.
- Pcurve parameter period agrees with curve period.
- Pcurve at points 0, 1/3, 2/3, and 1 way along curve must lie on the edge and tangent directions at these points must roughly agree, i.e., have positive dot product. This also tests the following:
- Spline surface form is set correctly, e.g. surfaces closed in u report this. Checks the underlying 3D B-spline surface at 10 points along seam to verify form.
- Checks that coedge vertices do not lie on spline surface singularities.

- Face normal is consistent with coedge direction.
- Start and end vertices of coedge lie on face.
- Edge lies on face. Checks at 10 points along edge.
- Start and end vertices lie on edge geometry.
- Faces are ordered correctly around edge, according to sidedness.
- Coedges are ordered correctly around edge, according to face curvature.
- Edge has same sense as curve (taking reverse bit into account).
- Checks curve has correct form.
- Edge parameter range is good and agrees with start and end points.
- Check edge for bad approximation direction.
- D3 checks on intcurve (option that can be turned on).
- No two vertices have the same location.
- Optional face/face intersection checking (option check_ff_int).

You may select one or more objects with the Check Object command. A report is presented in the above dialog box. The dialog box has the following options:

- Save As: Saves the reported as an ASCII file.
- Repair: Attempts to repair any errors that were reported.
- Next Object: Step to the next item in the list.
- OK: Exit back to the program.

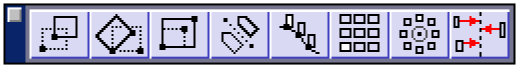
Transforming Geometry

This Designer Elements program provides tools and commands to transform your geometry with respect to scale, location and orientation.

This chapter covers the following topics:

- **Transformation** tools
- Transformation techniques

Transformation Tools



The tools on the Transformation tool palette include: move, rotate, expand or shrink and mirror objects, polar duplicate, linear duplicate and align. You can select the object you want to transform before or after you select a Transformation tool.

You can copy at the same time you transform objects by holding down the CTRL (Windows) or the OPTION (Macintosh) key while you specify the transformation.

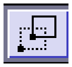
When you press the SHIFT key you can select additional objects after you have selected a transformation tool (see the Message Line). As soon as you release the SHIFT key the Transformation tool is active again.

For each tool you can enter values in the Status Line to perform the operation either before or after the operation is complete. If you enter the values after you select the tool but before you perform the operation, your first click in the drawing area automatically registers all Status Line values. If you enter values after performing the operation and while the object is still selected, pressing ENTER (Windows) or RETURN (Macintosh) updates the object to reflect the new values.

Geometric Characteristics

There are no geometric characteristics specific to these tools. The geometric characteristics are based on the objects. For information on the geometric characteristics of an object, see its related chapter.

Move Tool



Tech Note:

Use this tool to create an instance of a solid rather than cutting/copying and pasting which breaks the parent/child relationship. (**Cobalt™ and Xenon™ only**)

An instance is created when you perform an operation on a solid or surface and then move it to another location. This instance object displays its own unique *Geometry* page

The Move tool moves selected objects, including control points to a new location. You can copy the selection by holding down the CTRL (Windows) or OPTION (Macintosh) key. If you select more than one object, they remain in the same position relative to each other.

Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when moving objects with this tool.

Using the Move Tool

You can select the object before or after selecting this tool. The Message Line adjusts to accommodate this ability.

1. Select the Move tool.
2. Select the Move tool icon in the Message Line. The Message Line reads: *Move: Select objects to move [Shift = Extend]*.

   If your objects are already selected skip to step 3. The Message Line you see reflects the previous selection.
3. Select the object(s) to be moved.

   The Message Line reads: *Move: Select beginning reference point [Shift=Select, Ctrl=Copy* (Windows) or *Option=Copy* (Macintosh)*]*.

Tech Note:

Be aware that if you use this tool on an object that has had a subtraction boolean operation, you must also move the subtracted object. Otherwise, you may alter or lose the subtraction operation.

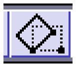
4. If necessary, use SHIFT-click to select more objects, and then pick a beginning point. The Message Line now reads, *Move: Select ending reference point [Shift=Select, Ctrl=Copy* (Windows) or *Option=Copy* (Macintosh)*]*.
5. Click an ending reference point or drag the selected object(s) to a new location.

The reference and destination points need not be on the object(s) you are moving. The move is performed relative to the specified points.

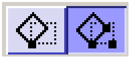
The Status Line contains the dX, dY and dZ values of the move distance and direction.

After the move and while the object is still selected you can change a value in a data field and press ENTER (Windows) or RETURN (Macintosh) to accept the value.

Rotate Tools



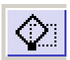
When you select the Rotate tools icon a subpalette appears in the Message Line containing three tools, Rotate by Angle (or by one point) and Rotate by Three Points.



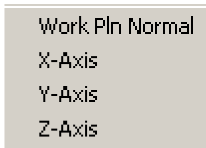
You can use them to rotate a selected set of objects around a point in any axis, or to create a circular array.

When you rotate by one point or by three points, you can copy the selection by holding down the CTRL (Windows) or OPTION (Macintosh) key while you select the objects. If you select more than one object, they remain in the same position relative to each other.

Rotate 1 Pt Tool

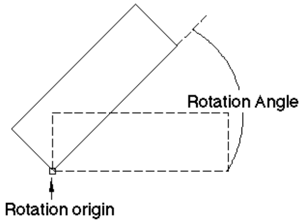


This tool rotates one or more objects around a specified point. The Message Line contains a pull-down menu for specifying the rotation axis. The options include:



Work Pln Normal	Rotates the objects about the normal vector of the work plane.
X-Axis	Rotates the objects about the X axis.
Y-Axis	Rotates the objects about the Y axis.
Z-Axis	Rotates the objects about the Z axis.

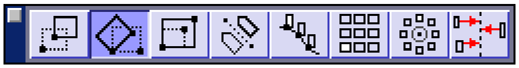
Arb. Axis Rotates the objects about an axis you specify by picking two points on screen.



Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when rotating objects with this tool.

Using the Rotate 1 Pt Tool

You can select the object before or after selecting this tool. The Message Line adjusts to accommodate this ability.



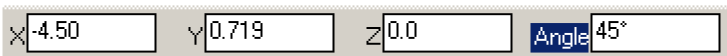
1. Select the Rotate tool.
2. Select the Rotate 1 Pt tool in the Message Line. The Message Line reads: *Rotate: Select items to rotate [Shift = Extend]*.

   If your objects are already selected skip to step 3. The Message Line you see reflects the previous selection.
3. Select the objects that you want to rotate.

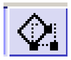
   The Message Line reads: *Rotate 1 Pt: Enter location to rotate about [Shift=Select, Ctrl=Copy* (Windows) or *Option=Copy* (Macintosh)*]*.
4. Use SHIFT-click to select more objects. The Message Line also includes a pull-down menu for choosing the rotation axis.
5. Select the rotation axis from the pull-down menu.
6. Specify the center of rotation.

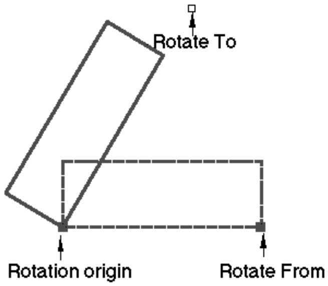
   The object is rotated.

The Status Line contains the X, Y and Z values of the rotation point and the rotation angle. After you complete the rotation click in any data field and enter a new value.



Rotate 3 Pts Tool





The Rotate 3 Pts tool rotates an object based on three defining points, the center of rotation, a beginning reference point, and an ending reference point. This Designer Elements program determines the rotation angle from the two reference points. The two reference points need not be on the object.

Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when rotating objects with this tool.

Using the Rotate 3Pts Tool

You can select the object before or after selecting this tool. The Message Line adjusts to accommodate this ability.

1. Select the Rotate tool.
2. Select the Rotate 3 Pts tool in the Message Line. The Message Line reads: *Rotate: Select items to rotate [Shift = Extend]*.

   If your objects are already selected, skip to step 3. The Message Line you see reflects the previous selection.
3. Select the objects that you want to rotate.

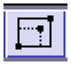
   The Message Line reads: *Rotate: Select center of rotation [Shift=Select, Ctrl=Copy* (Windows) or *Option=Copy* (Macintosh)*]*. If necessary, use SHIFT-click to select more objects.
4. Click the rotation point.
5. Click the first reference point.
6. Click the second reference point.

   The object is rotated.

The Status Line contains the rotation angle.

After you complete the rotation and while the object is still selected you can enter a new value if necessary. Press ENTER (Windows) or RETURN (Macintosh) to accept the value.

Expand/Shrink and Scale Tool



The Expand/Shrink and Scale tool enlarges or reduces objects. The Message Line contains a pull-down menu with two options:

Uniform When selected, the tool is the Expand/Shrink tool. This tool maintains the proportions of the stretched objects. This tool can be used for curves, surfaces and solids.

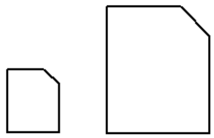
Differential When selected this tool is the Scale tool and allows you to specify separate scales in the X, Y and Z axes. This tool can be used for curves, surfaces and solids.

You can copy the selection as it is scaled by holding down the CTRL (Windows) or OPTION (Macintosh) key. If you select more than one object, they remain in the same position relative to each other.

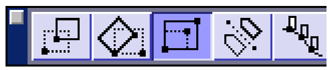
Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when stretching objects.

Using the Uniform Option or the Expand/Shrink Tool

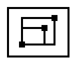
This option/tool lets you expand or shrink your curves, surfaces and solids uniformly. You can select the object before or after selecting this option. The Message Line adjusts to accommodate this ability. The graphic here shows uniformly scaled curves.



1. Select the Expand/Shrink tool. In The Message Line reads: *Expand/Shrink: Select items to scale [Shift = Extend]*.



   If your objects are already selected skip to step 3. The Message Line you see reflects the previous selection.
2. Select the object(s) to scale. The option pull-down menu appears.
3. Select the *Uniform* option from the menu. When you move the pointer into the drawing area the pointer becomes the expand/shrink icon, shown here.



The Message line now reads, *Expand/Shrink: Select anchor point [Shift = Select, Ctrl = Copy* (Windows) or *Option = Copy* (Macintosh)*]*. If you wish to select more objects, do so.

4. Click the anchor point on your geometry. This point remains fixed in the stretch. Use the SHIFT key to select more objects to stretch for this operation.

   The Message Line now reads, *Expand/Shrink: Select beginning reference point [Ctrl = Copy* (Windows) or *Option = Copy* (Macintosh)*]*.

   The beginning reference point is a point on your geometry used in conjunction with the ending reference point to set the scale of the operation. Rather than specifying reference points you can also just enter a value in the Scale data field and press ENTER (Windows) or RETURN (Macintosh) and the object will scale.

5. Click the beginning reference point.

   The Message Line reads: *Expand/Shrink: Select ending reference point [Shift=Select, Ctrl = Copy* (Windows) or *Option = Copy* (Macintosh)*]*.

6. Click the ending reference point. The scale factor is calculated based on the beginning reference point and this point.

   The selected object is resized. The Scale is displayed in the Status Line data field.

After you complete the scale and while the object is still selected you can enter a new value scale. Press ENTER (Windows) or RETURN (Macintosh) to accept the new value.

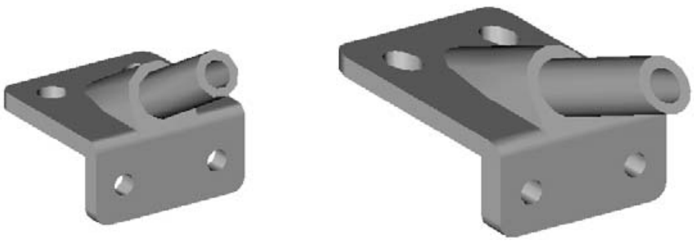
The Status Line contains the Scale factor. Entering a number between zero (0) and one (1) reduces the object. Entering a value above 1 enlarges the object.

Using the Differential Option or the Scale Tool

This option/tool lets you scale your curves, surfaces and solids differentially. You can select the object before or after selecting this option. The Message Line adjusts to accommodate this ability. The graphic here shows a differentially scaled solid.

Tech Note:

When you use this *Differential* option of the **Expand/ Shrink** tool for surfaces and solids, the stretched object displays its own *Geometry* page in Edit Objects.



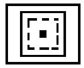
1. Select the Expand/Shrink tool. In The Message Line reads: *Expand/Shrink: Select items to scale [Shift = Extend]*.

   If your objects are already selected skip to step 3. The Message Line you see reflects the previous selection.

2. Select the object(s) to scale.

   The option pull-down menu appears.

3. Select the *Differential* option from the menu. When you move the pointer into the drawing area the pointer becomes the scale icon, shown here

   The Message Line reads: *Scale: Enter point to scale about [Shift = Select, Ctrl = Copy* (Windows) or *Option=Copy* (Macintosh)*]*. If necessary, use SHIFT-click to select more objects.

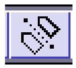
4. Enter the X, Y and Z Scale values in the Status Line.

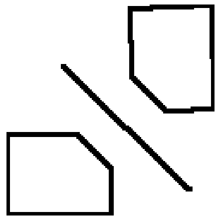
5. Click a point about which your object will be scaled.

The Status Line contains the X-Scale, Y-Scale and Z-Scale data fields.

After you complete the scale, while the object is still selected, you can enter new values. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

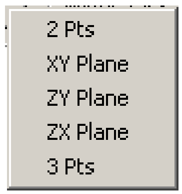
Mirror Tool





The Mirror tool creates a mirror image of an object or objects on the opposite side of a reference line. You can copy the selection by holding down the CTRL (Windows) or OPTION (Macintosh) key. If you select more than one object they remain in the same position relative to each other.

The Message Line contains a pull-down menu with five options for specifying the mirror reference line or plane.



2 Pts	Mirrors the object around the reference created by the two specified points.

XY Plane	Mirrors the object around the XY plane.
ZY Plane	Mirrors the object around the ZY plane.
ZX Plane	Mirrors the object around the ZX plane.
3 Pts	Mirrors the object around the mirror plane specified by three points.

Tip:

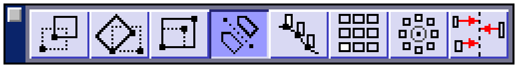
When using the plane options, it may be helpful to display the Axis for reference. Choose ***Layout>Show Axis***.

Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when mirroring objects.

Using the Mirror Tool - 2 Pts Option

You can select the object before or after selecting this option. The Message Line adjusts to accommodate this ability.

1. Select the Mirror tool. The Message Line reads: *Mirror: Select items to mirror [Shift = Extend].*



   If your objects are already selected, skip to step 3. The Message Line you see reflects the previous selection.

2. Select the object(s) to be mirrored.

   An option pull-down menu appears.

3. Select the *2 Pts* option in the pull-down menu. The Message Line reads: *Mirror: Select beginning of reference line [Shift=Select, Ctrl=Copy* (Windows) or *Option=Copy* (Macintosh)*]*.

   If necessary, use SHIFT-click to select more objects.

4. Click a location to set the beginning of the reference line around which the object mirrors. The Message Line now reads, *Mirror: Select end of reference line [Ctrl = Copy* (Windows) or *Option = Copy* (Macintosh)*]*.

   If you drag you specify the beginning and endpoints and the object mirrors.

5. Click to locate the end of the reference line.

   The reference line need not be parallel to the object.

Using the Mirror Tool - XY Plane Option

You can select the object before or after selecting this option. The Message Line adjusts to accommodate this ability.

1. Select the Mirror tool. The Message Line reads: *Mirror: Select items to mirror [Shift = Extend]*.

   If your objects are already selected skip to step 3. The Message Line you see reflects the previous selection.

2. Select the object(s) to be mirrored.

   An option pull-down menu appears.

3. Select the *XY Plane* option in the pull-down menu. The Message Line reads: *Mirror 1 Pt: Enter 1 point for mirror origin [Shift=Select, Ctrl=Copy* (Windows) or *Option=Copy* (Macintosh)*]*.

   Use SHIFT-click to select more objects.

4. Click a point in the XY plane to specify a reference point for the mirror.

Using the Mirror Tool - ZY Plane Option

You can select the object before or after selecting this option. The Message Line adjusts to accommodate this ability.

1. Select the Mirror tool. The Message Line reads: *Mirror: Select items to mirror [Shift = Extend]*.

   If your objects are already selected skip to step 3. The Message Line you see reflects the previous selection.

2. Select the object(s) to be mirrored. An option pull-down menu appears.
3. Select the *ZY Plane* option in the pull-down menu. The Message Line reads: *Mirror 1 Pt: Enter 1 point for mirror origin [Shift= Select, Ctrl=Copy* (Windows) or *Option=Copy* (Macintosh)*]*.

   Use SHIFT-click to select more objects.

4. Click a point in the ZY plane to specify a reference point for the mirror.

Using the Mirror Tool - ZX Plane Option

You can select the object before or after selecting this option. The Message Line adjusts to accommodate this ability.

1. Select the Mirror tool. The Message Line reads: *Mirror: Select items to mirror [Shift = Extend]*.

   If your objects are already selected skip to step 3. The Message Line you see reflects the previous selection.

2. Select the object(s) to be mirrored. An option pull-down menu appears.

3. Select the *ZX Plane* option in the pull-down menu. The Message Line reads: *Mirror 1 Pt: Enter 1 point for mirror origin [Shift=Select, Ctrl=Copy* (Windows) or *Option=Copy* (Macintosh)*]*.

   Use SHIFT-click to select more objects.

4. Click a point in the ZX plane to specify a reference point for the mirror.

Using the Mirror Tool - 3 Pts Option

You can select the object before or after selecting this option. The Message Line adjusts to accommodate this ability.

1. Select the Mirror tool. The Message Line reads: *Mirror: Select items to mirror [Shift = Extend]*.

   If your objects are already selected skip to step 3. The Message Line you see reflects the previous selection.

2. Select the object(s) to be mirrored.

   An option pull-down menu appears.

3. Select the *3 Pts* option in the pull-down menu. The Message Line reads: *Mirror 3 Pts: Enter 3 points for mirror plane [Shift=Select, Ctrl=Copy* (Windows) or *Option=Copy* (Macintosh)*]*.

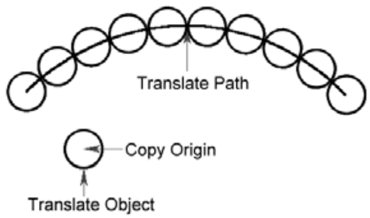
   Use SHIFT-click to select more objects.

4. Click three points to specify a reference plane for the mirror.

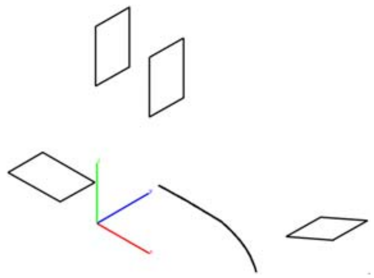
There are no Status Line entries for the Mirror tools.

Copy Along Path Tool

The Copy Along Path tool copies an object along a path by the number of times you set. The objects will be equally spaced along the curve.

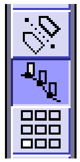


You can align your copies along the X, Y and Z axis (the alignment axis). The graphic here shows a polygon created in the Top plane with a Z alignment axis.



Using the Copy Along Path Tool

You can select the object before or after selecting this tool. The Message Line adjusts to accommodate this ability.



1. Select the Move tool.
2. Select the Copy Along Path tool in the Message Line. The Message Line reads: *Copy Along Path: Select items to translate along path [Shift = Extend]*.

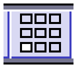
   If your objects are already selected skip to step 3. The Message Line you see reflects the previous selection.
3. Select the objects to copy.

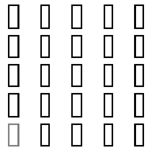
   The Message Line now reads, *Copy Along Path: Select the translation path for objects*.
4. Enter the number of objects to copy in the Qty (Quantity) field of the Status Line.
5. Click the path curve. The Message Line reads: *Copy Along Path: Enter copy origin*.
6. Click the point where the copy should begin. The Message Line reads: *Copy: Along Path: Enter two points for alignment axis.*

   You can align the copies along the X, Y or Z axis. Use the Drafting Assistant to choose the desired axis.
7. Pick two points on the screen to show your Designer Elements program where to align the object(s).

The Status Line contains the Qty (Quantity) field displaying the number of objects this Designer Elements program will create. You must set this value before copying the objects.

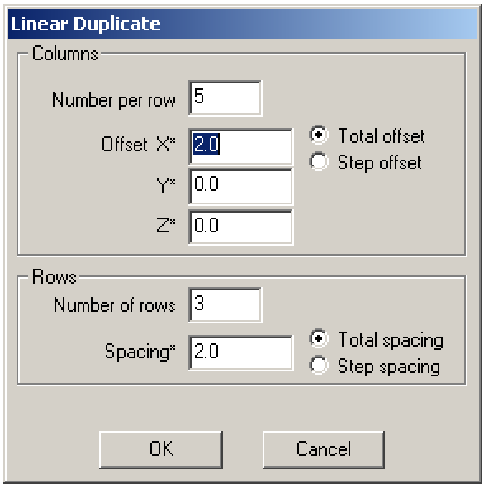
Linear Duplicate Tool





The Linear Duplicate tool creates copies of an object in a rectangular array.

To complete the array, you set the number of horizontal rows and vertical columns and the amount of space between each object in the array. This is done through the Linear Duplicate dialog box.



The dialog box contains the following options:

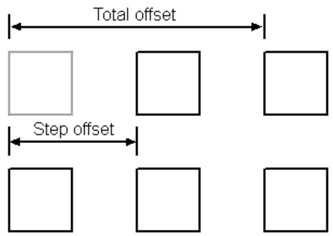
Number per row (Columns section) This field sets the total number of objects per row. Be sure to include the selected object in this total count.

If you don't know the number, you can use math operators to determine the number of copies. For example, if you want to place studs 16 inches apart on a 17 foot wall, you would enter (17*12)/16. This Designer Elements program will truncate the result to 12 copies.

Offset X*, Y*, Z* (Columns section) These fields set the offset length in the X, Y or Z direction for the duplicated objects. These values work with the Total offset and Step offset options. Enter these values yourself or drag in the drawing area to enter the values.

Total offset

(Columns section) When selected, this option specifies that the offset values in the X, Y and Z fields are the total offset values from the selected object to the last copy.



Step offset

(Columns section) When selected, this specifies that the values in the X, Y and Z fields are step offset values from the selected object to the first copy.

Number of rows

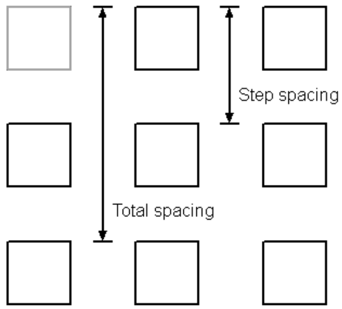
(Rows section) This field sets the total number of rows.

Spacing*

(Rows section) This field sets the perpendicular spacing for rows. This field operates with the Total spacing and Step spacing options.

Total spacing

(Rows section) When selected, this option specifies that the Spacing value is the total spacing from the selected object to the last copy. This option works with the Spacing field.



Step spacing

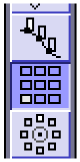
(Rows section) When selected, this option specifies that the Spacing value is the distance from the first

	selected object to the first copy. This option works with the Spacing field.
OK/Cancel	Click OK to complete the operation. Click Cancel to discontinue the operation.

The asterisk (*) indicates that the value automatically fills in when you drag the pointer in the drawing area to indicate the offset or spacing.

Using the Linear Duplicate (Rectangular Arrays) Tool

You can select the object before or after selecting this tool. The Message Line adjusts to accommodate this ability.



1. Select the Move tool.
2. Select the Linear Duplicate tool in the Message Line. The Message Line reads: *Linear Duplicate: Select objects for linear duplicate [Shift = Extend].*

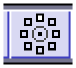
   If your objects are already selected the Linear Duplicate dialog box appears. Skip to step 3.
3. Select the object(s) to be duplicated. The dialog box appears.

   The Message Line now reads, *Linear Duplicate: Specify parameters for linear duplicate.*
4. Enter the number of columns in the *Number per row* data field.
5. Select offset option, either Total or Step.
6. If the *Offset X** field is not selected, click in the field.
7. Move the pointer to the drawing area and drag to indicate the offset for the row of copies. Values automatically appear in the X*, Y* and Z* data fields.
8. Select the spacing option, either Total or Step.
9. If the *Spacing** field is not selected, click in the field.
10. Move the pointer to the drawing area and drag to indicate the spacing for the spacing for the rows.
11. Click OK to close the dialog box and create the duplicates.

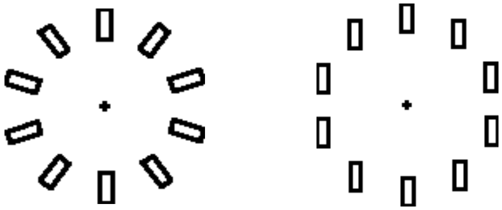
There are no Status Line entries.

Note: This Designer Elements program does not support using this tool for features.

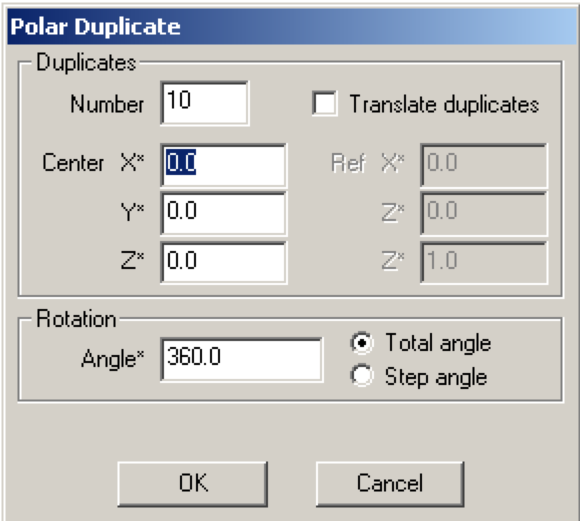
Polar Duplicate Tool



The Polar Duplicate tool copies a selection around a central point. The entities in the selection may be rotated as they are copied or remain upright.



The number and location of duplicates are controlled through the Polar Duplicate dialog box.



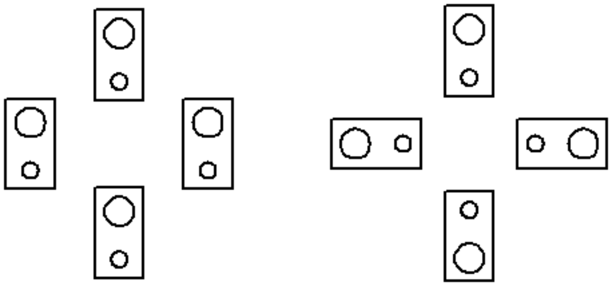
The dialog box contains the following options:

Number — (Duplicates section) This field sets the total number of objects. Be sure to include the selected object in this total count.

Center X*, Y*, Z* — (Duplicates section) These fields set the X, Y and Z location for the center of the circular array. You can enter these values yourself or click the place in the drawing area for the center.

Translate duplicates

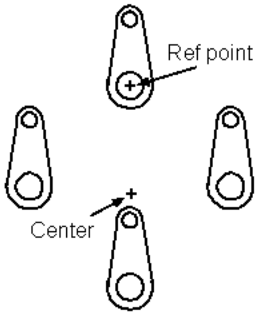
(Duplicates section) When selected, this option creates copies that are upright with respect to the original object. When this option is not selected, copies are rotated around the center you specified. The left graphic here shows an duplicated object when the option is selected. The right graphic shows a duplicated object when the option is not selected.



Ref X*, Y*, Z*

(Duplicates section) These fields represent the reference point for the Translate duplicates option.

Ref X, Y and Z determine the imaginary point duplicated around the center you specified. The selected objects are reproduced in the same position relative to each imaginary point duplicated. The imaginary Ref point and center point are indicated in the example here.

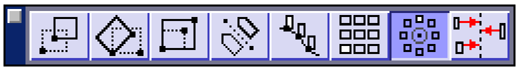


	You can specify the Ref point by entering values or clicking at the desired location in the drawing area.
Angle	(Rotation section) This field specifies the angle of rotation. This field operates with the Total angle and Step angle options.
Total angle	(Rotation section) When selected, this option specifies that the angle value is the total number of degrees between the center of the selected object and the center of the last object. (If 360° is the specified angle, the last object is the selected object.)
Step angle	(Rotation section) When selected this option specifies that the angle value is the angle between the center of two adjacent objects.
OK/Cancel	Click OK to complete the operation. Click Cancel to discontinue the operation.

The asterisk (*) indicates that the value automatically fills in when you drag the pointer in the drawing area to indicate the offset or spacing.

Using the Polar Duplicate Tool

You can select the object before or after selecting this tool. The Message Line adjusts to accommodate this ability.



1. Select the Polar Duplicate tool
2. The Message Line reads: *Polar Duplicate: Select items to polar duplicate [Shift = Extend].*

   If your objects are already selected the Polar Duplicate dialog box appears. Skip to step 3.
3. Select the object to be duplicated. The dialog box appears.

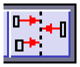
   The Message Lines reads, *Polar Duplicate: Specify parameters for polar duplicate*.
4. Enter the number of objects in the circular array.
5. If the *Center X** data field is not selected, click in the field.
6. Move the pointer to the drawing area and click the center point of the array in the drawing area. Values automatically appear in the X*, Y* and Z* data fields.

7. If you want objects to be upright with respect to the selected objects, select the *Translate duplicates* option.
8. If you specified Translate duplicates, click a location for the reference point in the drawing area.
9. Select the angle option, Total or Step.
10. Enter the angle of rotation in the *Angle* field or drag in your drawing area along the desired angle using the Draft Assistant.
11. Click OK.

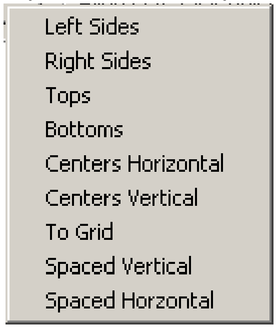
There are no Status Line entries.

Note: This Designer Elements program does not support using this tool for features.

Align Objects Tool



With this Designer Elements program you have the ability to align any object created using the *Align* command. This includes aligning text objects with other text objects and text objects with geometry.



Using the Align Command

1. Choose the Selection tool.
2. Hold down the SHIFT key and select the text and geometry you want to align.
3. Choose the align tool from the Transformation tool palette.

The objects align.

For more detailed information regarding the Align Tool refer to chapeter 24, The Align Command.

Transforming Techniques

There are many ways to accomplish the same transformation task in this Designer Elements program. This section includes some of those ways.

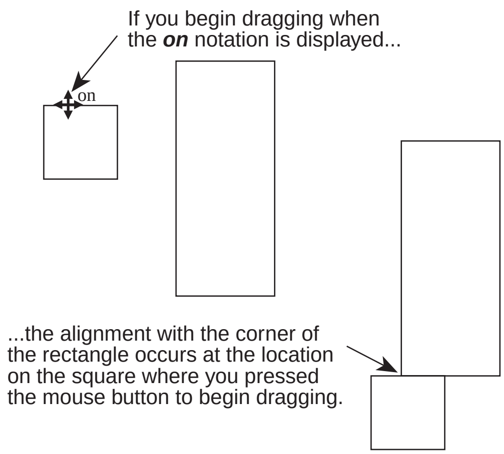
Moving Objects with Tools

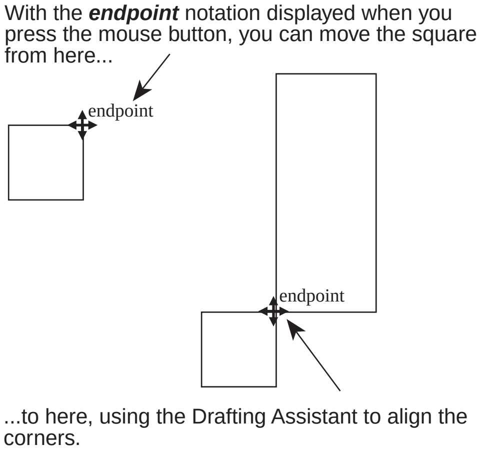
When an object is selected, move the pointer near the object until the *4-way Move* symbol appears as shown below and drag the object to a new location.

Note: If the 4-way Move symbol doesn't appear, make sure you're in the Selection tool.

Using the Drafting Assistant for Moving

When the pointer becomes the 4-way Move symbol, you can drag the object around and see the Drafting Assistant's notations relative to the object's location when you press the mouse button. This way, if you move the pointer over a control point so the Drafting Assistant activates it, you can align the object you are moving with that point.

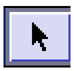




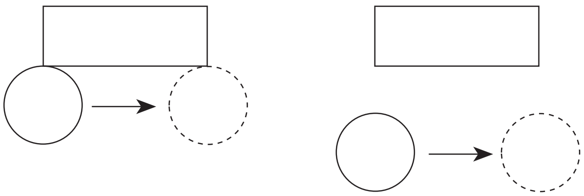
Selected Move versus Move Tool

You can move objects either with the Selection tool or with the Move tool.

Moving with the Selection Tool

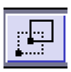


The move function of the **Selection** tool allows free movement of the selected object.



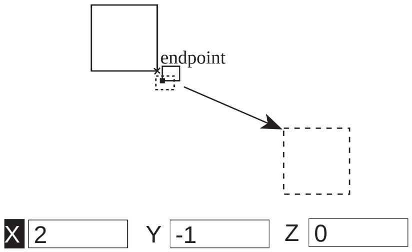
To move a circle from one location to another as shown above, use the Selection tool and the Drafting Assistant to align the 90° *quadrant* with the corner of the rectangle. The circles do not need to touch the rectangle to be aligned with it.

Moving with the Move Tool



The Move tool (in the Transformation subpalette) allows you to move the selection with reference to other geometry in the X, Y and Z direction.

Move a 1-inch square 2 inches in the X direction and 1 inch in the Y direction. See the following graphic.



1. Select the Move tool.
2. Select the square to be moved.
3. Click one corner of the square when the *endpoint* notation is displayed.
4. Enter **2** in the dX entry field on the Status Line and **-1** in the dY entry field. A positive or negative value entered in the Status Line determines the direction along the X or Y axis. A negative value moves the object to the left or down on the screen and a positive value moves the object to the right or up.
5. Press the ENTER (Windows) or RETURN (Macintosh) key. The square is moved.

Moving Objects to Another Layer

1. Choose ***Window>Edit Objects***.
2. In the Layer field, scroll down to the layer on which you want the object.
3. Click OK. The object is now located on the new layer.

Copying Objects with Tools

You can copy selections with the *Copy* command, as discussed earlier, or by holding down the CTRL (Windows) or OPTION (Macintosh) key using these tools:

- **Selection** tool
- **Single Line** tool
- **Center-Point** and **Opposite-Point Circle** tools
- **Ellipse** tools
- **Conic** tools
- **Polygon** tools

- **Project Curve** tool
- **Transformation** tools

Copying with the Selection Tool

You can hold down the CTRL (Windows) or OPTION (Macintosh) key and drag a copy of the selection to a new location.

When you use the Selection tool to copy a surface or solid using this technique you create an instance of the original object. All changes made to the original object are automatically reflected in the instance (**Cobalt™ and Xenon™ only**). If you do not want to create an instance, you can select the original object, choose ***Edit>Copy*** and then ***Edit>Paste***.

1. Choose the Selection tool.
2. Select the object(s) you want copied.
3. Hold down the CTRL (Windows) or OPTION (Macintosh) key.
4. Drag a copy of the selection to a new location. The copy is placed on the current work layer.

Copying with the Geometry Tools

You can make copies with the following drawing tools:

- **Single Line** tool
- **Center-Point** and **Opposite-Point Circle** tools
- **Ellipse** tools
- **Conic** tools
- **Polygon** tools
- **Project Curve** tool

To create a copy with the drawing tools proceed as follows:

1. Construct the geometry you want to copy.
2. Hold down the CTRL (Windows) or OPTION (Macintosh) key.
3. Click a new location. The click determines the location of the first point specified during the construction of the original geometry (the *center* of a Center-Point Circle, for example).

Copying with the Transformation Tools

You can hold down the CTRL (Windows) or OPTION (Macintosh) key while using a transformation tool to make a copy of the selected geometry.

1. Select one of the transformation tools.
2. Select the object you want to copy and transform.
3. Hold down the CTRL (Windows) or OPTION (Macintosh) key.
4. Perform the transformation according to the directions in the Message Line.

Copy CTRL (Windows) or OPTION (Macintosh) versus Copy Command

Normally, using the copy option of a geometry tool is faster than using the *Copy* and *Paste* commands in the Edit menu. The *Copy* and *Paste* commands are useful for copying to a different document or application.

Sizing Objects with Tools

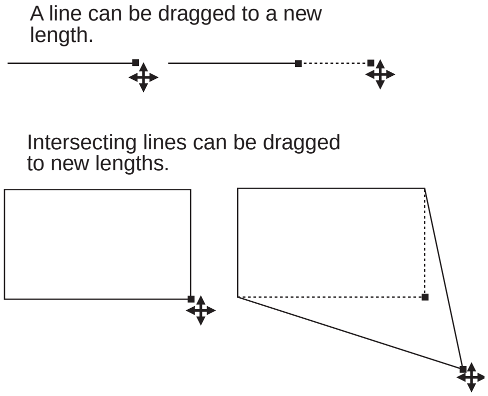
Normally you size an object with the Selection tool. In some cases it is useful to use the Move tool for sizing objects.

Sizing an Object with the Selection Tool

You can stretch objects by selecting a point and dragging it to a new location.

1. In the Edit menu, be certain that *Selectable Points* is set.
2. Click the Selection tool.
3. Drag a selection fence around the control points that represent the area you want to stretch.

4. Drag the points to a new location.

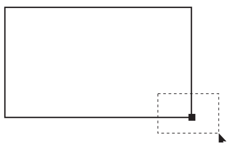


Sizing an Object with the Move Tool

You can also size an object with the Move tool. Using the Move tool allows you to specify the distance the selected point(s) should be moved by specific values along the dX, dY and dZ direction in the Status Line.

To move the corner of the single line rectangle in the next graphic with the Move tool, proceed as follows:

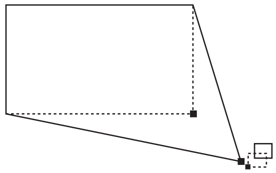
1. In the Edit menu, be sure that *Selectable Points* is set.
2. Click the Selection tool.
3. Drag a selection fence around the lower right corner of the rectangle.



4. Select the Move tool.
5. Enter **+1** in the dX data field, a **-1** in the dY field and a **0** in the dZ field.

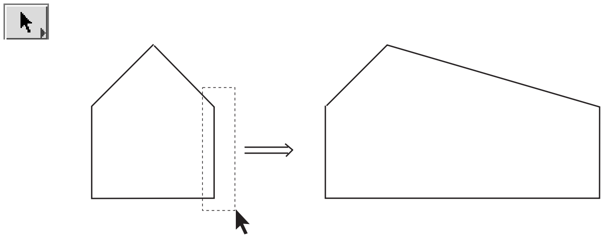
   Press the ENTER (Windows) key or the RETURN (Macintosh) key.

The corner of the rectangle is moved the distance you specified in the Status Line.

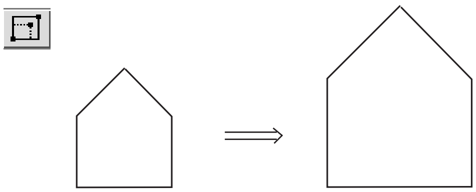


Selected Sizing versus Expand/Shrink Tool

Dragging a control point of a selected object not only changes the size of the object, but distorts the object by changing the proportion between height and width.



The Expand/Shrink tool on the Transformation subpalette enables you to resize geometry while maintaining its proportions.



Use the Scale tool to enlarge or shrink a shape proportionally

In addition, you can specify proportions by clicking points on other geometry. For example, you can resize an object to fit within another object by clicking the boundary into which the resized object must fit.

Text

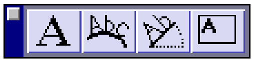
When you want to add some basic annotations to your drawing use one of the text tools. You can create and edit text. You can use the Text menu to set the characteristics such as font, text size and case. The topics discussed in this chapter include:

- **Text** tools
- Text Commands

Tech Note:

This Designer Elements program does not support the importation of text documents.

Text Tools

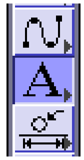


This Designer Elements program has three tools for creating text: Horizontal Text, Text Along a Curve and Text at an Angle. The text tools in this Designer Elements program do not use text boxes. Simply click the cursor at the location you want the text to begin and start typing. Press ENTER (Windows) or RETURN (Macintosh) to end one text object and begin another. These text objects move independently.

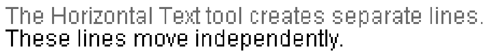
Horizontal Text Tool

The Horizontal Text tool creates horizontal text at the location you specify.

Using the Horizontal Text Tool

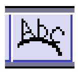


1. Select the Horizontal Text tool from the main tool palette. The Message Line reads: *Enter location for text [Press Mouse to End].*
2. Place the cursor at the beginning location for the text and click. The cursor becomes a text cursor.
3. Enter the desired text. If you press ENTER (Windows) or RETURN (Macintosh), you begin a new and separate line of text that is aligned left with the previous line.



Since this is a separate text line it can be moved independently.

Text Along a Curve Tool



The Text Along a Curve tool creates text along the curve you specify.

Tech Note:

This tool may not support all of your system's fonts. True Type fonts that don't form closed polygons are not supported.

Using the Text Along a Curve Tool

1. Create the curve along which you want the text, placing the points in the order that you want the text placed. For a horizontal line created left to right, the text appears left to right. For a vertical line created top to bottom, the text appears top to bottom.
2. Select the Text Along a Curve tool from the main tool palette. The Message Line reads: *Select path for text [Shift=Extend].*
3. Select the curve you just created. The cursor becomes a text cursor placed at the start point of the curve.
4. Enter the desired text. It displays horizontally.

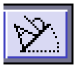
5. Press ENTER (Windows) or RETURN (Macintosh) and the text line molds to the curve.

A parent/child relationship exists between the curve and the text. Any change that you make to the curve affects the text. Select any curve control point and drag it to a new location and the text updates.

If the curve is not long enough for the text, you can extend the curve by selecting a curve endpoint and dragging it to a new location.

This tool only supports one line of text per curve. You cannot move the text independently of the curve. If you do not want to display the curve with the text, change the curve color to blend with your background or place the curve on the construction layer. Then you still have the ability to edit the text appearance on screen by editing the curve.

Text at an Angle Tool



The Text at an Angle tool creates text at the angle and location you specify.

Tip:

If you enter the angle in the Status Line after you type the text, you will have to hit ENTER (Windows) or RETURN (Macintosh) twice. The first time registers the values and the second changes the text angle to the new angle.

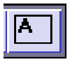
1. Select the Text at an Angle tool from the main tool palette The Message Line reads: *Enter location for text.*
2. In the Status Line, enter the angle for the text in the Angle data field. Press ENTER (Windows) or RETURN (Macintosh) to register the value.

3. Place the cursor at the beginning location for the text and click. The cursor becomes a text cursor.
4. Enter the desired text. The text displays horizontally.
5. Press ENTER (Windows) or RETURN (Macintosh) and the text angle changes.

Geometric Characteristics

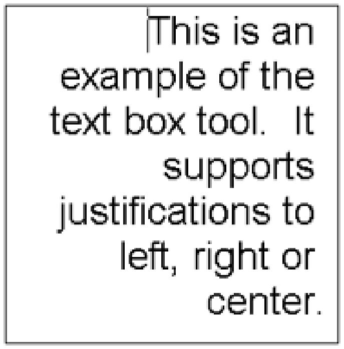
A text object is created by clicking the location of the start point of the text and typing the desired text. A text object is made up of the following characteristics according to the Edit Objects dialog box: Text and Angle. To display the dialog box, select the text and choose ***Window>Edit Objects*** or double-click on the text.

BoundingBox Text Tool



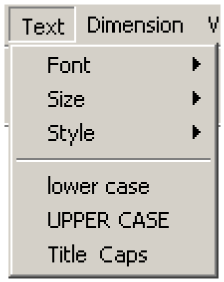
Bounded Box Text is defined by two diagonal points used to specify the width and height of a text box. Text is automatically wrapped to the width of the box. The text box also supports left, right and center justifications.

In addition, you can double click on the text to make edits directly on the screen. On screen editing supports cutting, pasting and copying into the text box. For example, you can copy of paragraph from Microsoft Word and paste it into a text box.

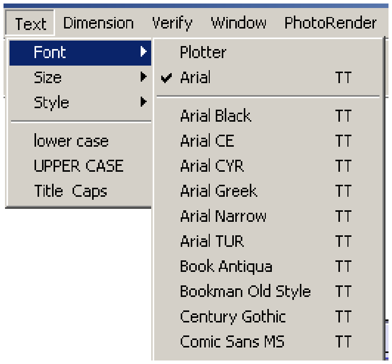


Text Commands

All Text commands are contained in the Text menu and include commands for changing the text font, size, style and case.



Font



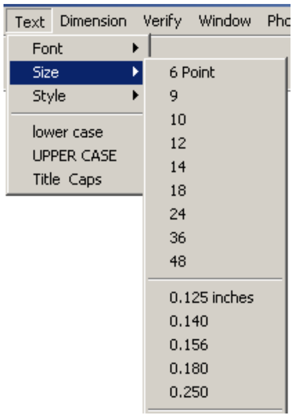
This command in the Text menu changes the font for selected text or future text entries.

The menu lists fonts installed on your computer and one Plotter font. Use the Plotter font whenever you are creating text for a drawing that you intend to send to a plotter, since plotting Postscript or TrueType fonts takes much more time and fonts may be substituted.

If you select text and change the font only the selected text is affected. If no text is selected and you change the font, you are setting the default font for future entries until you choose another font.

Special characters and accents are available as described in Appendix B.

Size



This command in the Text menu sets the font size for the selected text or for future text entries in the current document.

If you select text and change the size only the selected text is affected. If no text is selected and you change the text size, you are setting the default size for future entries.

The size can be specified as either points (12 pt) or as units (.156 inches) as defined in the *Units* page of Preferences. If no unit is specified this Designer Elements program picks the default unit.

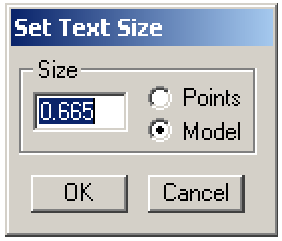
Specifying a Non-standard Text Size as the Default Size

Choosing *User* from the Size submenu brings up a dialog box allowing you to specify the exact text size. You can choose a point size or the units you've set for your drawing.

1. Choose ***Text>Size>User***.

2. Select the size option, either *Points* or *Units* by clicking in the appropriate radio button.
3. Enter the size you want.

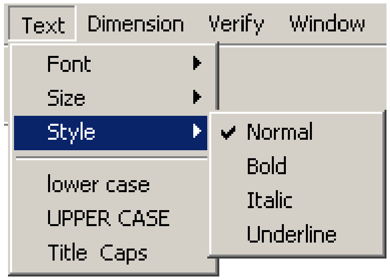
   If you choose *Model*, the size is measured in the current units as set in *Units* page of Preferences.
4. Click OK to accept the change and close the dialog box. Click Cancel to close the dialog box without accepting the changes.



The size you specify stays in effect until you choose another size.

Style

This command in the Text menu sets the style (such as **Bold** or <u>Underline</u>) for selected text or future text entries.

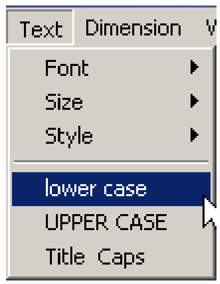


The style you specify stays in effect until you choose another default style. A check (✔) indicates the current style.

If you only want to change the style of one text line, select the text and choose ***Text>Style*** and the option you want. This does not affect the default style setting.

Case

This Designer Elements program supports three text cases: lower case, UPPER CASE and Title Caps.



lower case	Displays text all in lower case letters.
UPPER CASE	Displays text in upper case letters.

Title Caps Displays the first letter of each word in upper case and the remaining letters in lower case.

You cannot set case as a default setting like font, size and style. It simply changes the case of any selected text.

Using the Case Options

1. Select a text line.
2. Choose ***Text>lower case***, ***UPPER CASE*** or ***Title Caps***.

   Your text line changes to reflect your choice.

Tip:

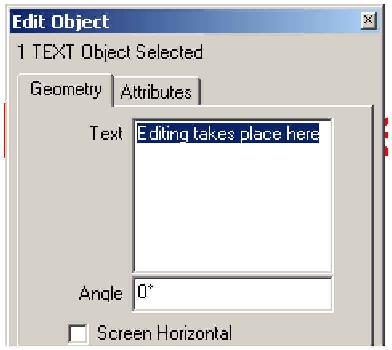
If you want to change from *UPPER CASE* to *Title Caps*, you must first changes your text to *lower case* and then change the case to *Title Caps*.

Text Handling

You may want to perform various operations on your text like editing, moving etc.

Editing Text

To edit text you've created, select the text line and choose ***Window>Edit Objects***.



In the Text entry field, make any desired changes. Click Apply to accept the changes and then Close to exit the dialog box. Click Close without hitting Apply if want to close the box without making any changes.

Moving Text

To move text, choose the Selection tool, select the desired text and drag the text to a new location.

Screen Horizontal Text

As seen in the picture above, the Edit Objects dialog box contains a check box for keeping text horizontal to your screen. With the box checked your text will stay horizontal to your screen when you use the Trackball to rotate your drawing. With the box unchecked your text will stay horizontal to the plane it was created on. For example, if you create text in the top view and switch to the trimetric view you text will appear as if it were lying flat.

Dimensions

This chapter explains how to use the dimension tools in this Designer Elements program. The following topics are covered:

- Dimension Menu
- Associative Dimensions
- Dimension tools
- Dimension Appearance
- Dimensions and Drawing Views

Dimension Menu

The Dimension menu contains commands for displaying the palette of dimensioning tools and for setting the format for dimensions. When you are ready to start dimensioning, choose the Dimension tool you want from the Dimension tool palette.

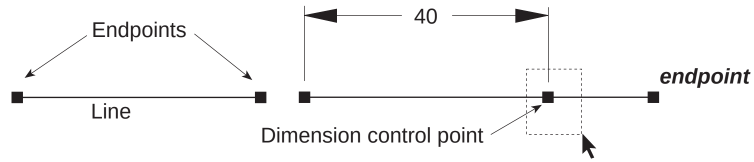
Like the other tool palettes, the Dimension palette is a floating tool palette. You select dimension tools in the same way you select tools from the main tool palette. The first two tools and the last tool have subpalettes which also work like the subpalettes of the main tool palette.

Associative Dimensions

This Designer Elements program' geometric dimensions are associative—when you make a change to the geometry, the dimension changes also. This is not true if you overwrite the # symbol in the data field or Edit Objects with a set value/text string.

This associativity is a tremendous time saver because dimensions automatically update whenever you make a change in the geometry (**Cobalt™** and **Xenon™** only). You can even change the units from English to metric (in the Units page of Preferences) and every dimension on your drawing will reflect the change.

When you extend a line by selecting the endpoint of the line and dragging it to a new position the dimension changes also because the dimension has a control point at the same position of the line's endpoint. So when you select the endpoint of the line you also select the dimension control point.



Dimensions are associative relative to the points they measure. If you change the length of a line using Edit Objects, the dimension will not update because the point was not changed. To correct the dimension, select the dimension's vertex point and drag it to the new endpoint of the line.

Dimension Tools

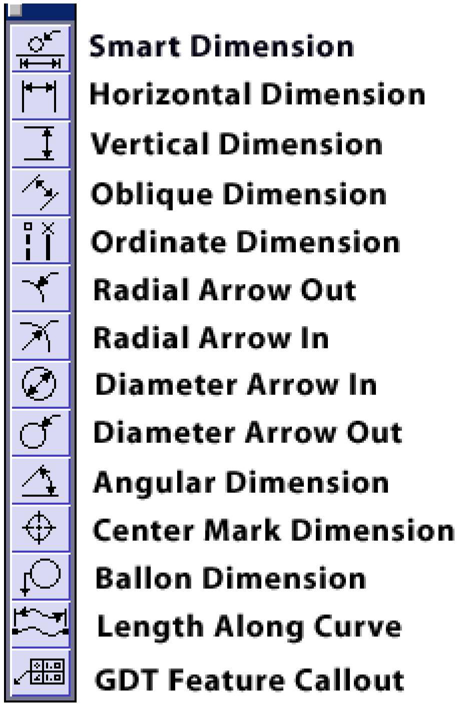
This Designer Elements programs dimensioning tools are on a floating palette which you can drag around the drawing area. These tools allow add basic dimensions to your drawing.

Show Palette

The Dimensions palette is located in the standard tools palette.

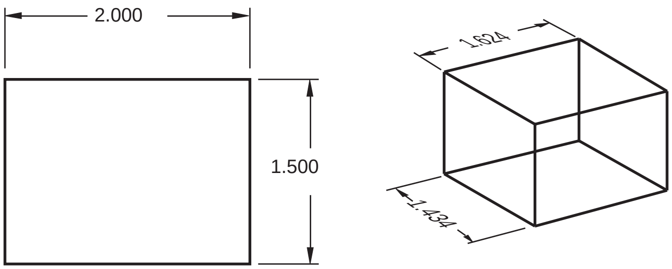
These tools can be used to measure either an object or the area between objects. Once you have dimensioned an object (or space), the dimensions update when you make changes to the geometry **(Cobalt™ and Xenon™ only)**.

Some of the dimensioning tools, such as the Horizontal and Vertical tools, require that two points be selected; others, such as the Radial and Diameter, require only one. The location and the status (on/off) of the Dimension palette is automatically saved when you exit the program.



Dimensions and the Work Plane

Dimensions appear in the current work plane of the active view. (They are defined by the coordinate system of the work plane at the time they are created.) Set the work plane to correspond to the view before you begin adding text or dimensions in a view.

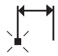


For example, if the current work plane is Top, dimensions appear parallel to the work plane regardless of the view. See the graphic here.

All point to point dimensions are also placed parallel to the work plane. If you were working in the front plane and you were using the horizontal dimension tool, the dimension would be placed parallel to the front plane.

Dimension Smart Pointer

The dimension pointer is a smart pointer with a hot spot—the dot shows which side of the object to select first.



If you select as indicated by the position of the dot and cross hair, the text appears above or to the right of the leader. If you select in the opposite order, the text appears below or to the left of the leader.

Dimensioning Objects and Placement

1. Select the appropriate dimension tool.
2. Click the points you want to measure.

   Dimensions automatically use the dimension pen style and current dimension text characteristics. If you want to change the dimension pen color or weight, you can do so in the Dimension menu.
3. Move the dimension to a new location if its placement is not where you want it.

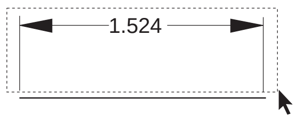
Moving the Dimension

After you place a dimension, you may want to reposition it relative to your geometry. The steps differ depending on the tool. You can reposition your dimension after it has been placed with the Selection tool.

1. Choose the Selection tool.
2. Place the Selection tool over the dimension text and it becomes the Move symbol.

3. Drag the dimension to the desired location.

   If you want to select more than one dimension drag a selection fence around them.



You can also reposition the text along the dimension line by dragging it along the line.

For radial and diameter dimensions, you can also reposition the dimension's control point by dragging a selection fence around the tip of the dimension line arrow.

Dimension Status Line Fields

The Horizontal, Vertical, Parallel, Angular, Center Mark and Balloon Dimension tools each have Status Line fields associated with them.

The Status Line field, similar to the one below, appears when one of these dimensioning tools is selected. Those Status Lines that differ are noted with the tools.

The # symbol in the text field indicates that the dimension is the actual value of the object's measurement. If you delete this symbol the dimension will no longer be associative (that is, it won't update if you change the geometry it measures).

You can also add text before or after the # symbol which will display with the dimension text.

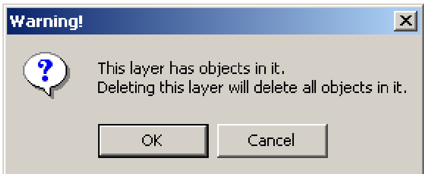
Geometric Characteristics

Dimension text is created automatically when you dimension an object and is made up of one characteristic according to Edit Objects, Text.

Automatically Placing Dimensions on a Separate Layer

This Designer Elements program automatically creates a Dimension layer. Dimensions automatically go on that layer except in the case of dimensions placed in drawing tools. These drawing view dimensions go on the Sheet View layer. See Chapter 36 for information about drawing views.

If you accidently delete the Dimension layer, This Designer Elements program automatically recreates the layer if you close and open the file. If you try to delete the Dimension layer and there are objects on it, this Designer Elements program displays the following warning.



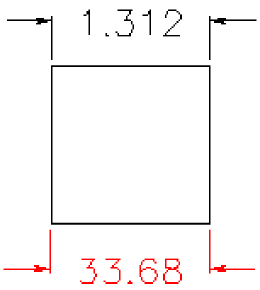
Dimensions and Edit Objects

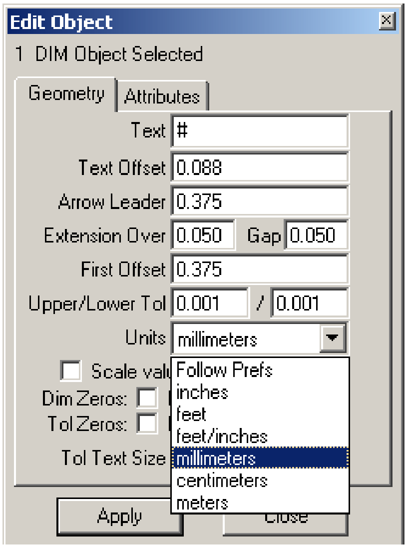
In the Edit Objects dialog box the text characteristic field includes a # symbol for all dimensions, which indicates the dimension is the actual value of the object's measurement. If you delete the # symbol and enter another value, the dimension is fixed and will not update if you change the units or the size of the geometry.

Some of the dimensioning tools, such as the Radial tools, add a letter in the text status box which appears in the dimension itself.

You can add text before or after the # symbol. You can enter parentheses to enclose the resulting dimension in parentheses (#), or enter # 2 Plcs to display the dimension followed by the text 2 Plcs. When you include the # symbol, this Designer Elements program uses the measurement of the geometry as well as the text you add.

Edit Objects also includes dimension characteristics including a Units data field. This field allows you to set the units for a selected dimension to those in Preferences or another settings. This allows you to dimension a part with mixed units. The graphic below shows an example of mixed units with the Edit Objects dialog box.

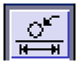




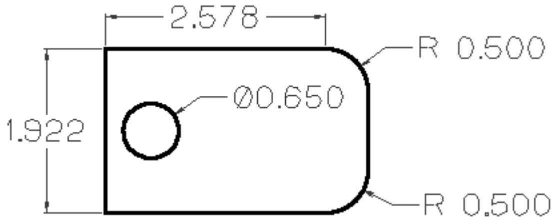
Dimension Surfaces and Solids

You can dimension surfaces and solids using the Horizontal, Vertical and Parallel dimension tools only. The Radial, Diameter and Angular Dimension tools can only be used for wireframe objects.

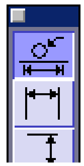
Smart Dimension Tool



This tool in the Dimension tool palette allows you to quickly place horizontal, vertical, radial and diameter dimensions with a single click. These dimensions are associative to the curves. As you change the curve, the dimension automatically updates (**Cobalt™** and **Xenon™** only). This tool is especially useful for drawing views.



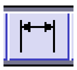
Using the Smart Dimension Tool



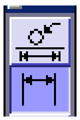
1. Select the tool from the Dimension tool palette. The Message Line reads: *Smart Dimension: Pick curve to dimension*.
2. Select the curve.

The dimension appears. You can drag the dimension to a new location. Dimension and extension lines automatically redraw.

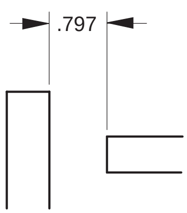
Horizontal Dimension Tools



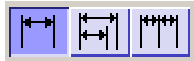
These tools measure horizontal spaces or the distance between linear objects.



This tool dimensions objects or spaces horizontally. When you select the Horizontal Dimension tools, a subpalette appears in the Message Line containing three tools, Horizontal Dimension, Horizontal Base Line Dimension and Horizontal Chain Dimension.



Using the Horizontal Dimension Tool



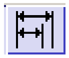
1. Select the Horizontal Dimension tools. Choose the Horizontal Dimension tool from the Message Line. The Message Line reads: *Horizontal: Pick first dimension point*.
2. Click the left point of the geometry.

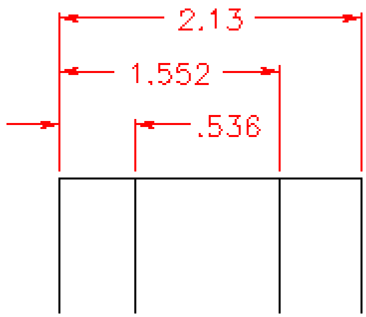
   The Message Line now reads, *Horizontal: Pick next dimension point*.
3. Click the right point.

   The dimension appears. You can drag it to a new location. Dimension and extension lines automatically redraw.

   Click the points in the opposite order if you want the dimension to display below the objects.

Horizontal Base Line Dimension Tool





This tool dimensions objects or spaces from a base point.

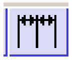
Using the Horizontal Base Line Dimension Tool

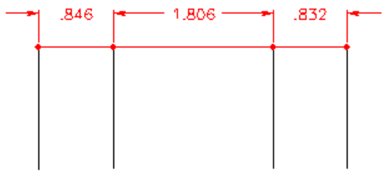
1. Select the Horizontal Dimension tools. Choose the Horizontal Base Line Dimension tool. The Message Line reads: *Horizontal Base Line: Pick first dimension point*.
2. Click the base point of the geometry.

   The Message Line now reads, *Horizontal Base Line: Pick next dimension point*.
3. Click the point. The dimension appears.
4. Click the next place for the dimension. This dimension appears above the first measured from the base point. Continue clicking the points you want dimensioned. You can drag each dimension to a new location. Dimension and extension lines automatically redraw.

   Click the points in the opposite order if you want the dimension to display below the objects.

Horizontal Chain Dimension Tool





This tool dimensions objects or spaces from end to end, horizontally.

Using the Horizontal Chain Dimension Tool

1. Select the Horizontal Dimension tools. Choose the Horizontal Chanin Dimension tool. The Message Line reads: *Horizontal Chain: Pick first dimension point*.

2. Click the first point on the geometry.

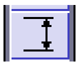
   The Message Line now reads, *Horizontal Chain: Pick next dimension point*.

3. Click the point. The dimension appears.

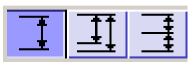
4. Click the next place for the dimension. This dimension appears measured from the last point clicked. Continue clicking all the points you want dimensioned. You can drag each dimension to a new location. Dimension and extension lines automatically redraw.

   Click the points in the opposite order if you want the dimension to display below the objects.

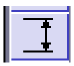
Vertical Dimension Tools

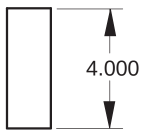


These tools measure vertical space or the distance between vertical objects. When you select the Vertical Dimension tools, a subpalette appears in the Message Line containing three tools: Vertical Dimension, Vertical Base Line Dimension and Vertical Chain Dimension.



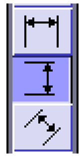
Vertical Dimension Tool





This tool dimensions an object or space vertically.

Using the Vertical Dimension Tool



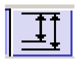
1. Select the Vertical Dimension tool. Choose the Vertical Dimension tool. The Message Line reads: *Vertical: Pick first dimension point*.
2. Click the top point of the geometry first.

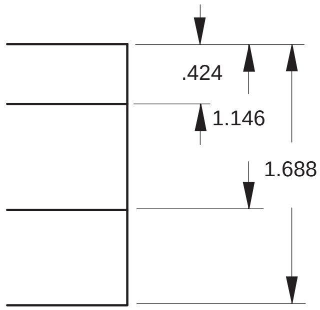
   The Message Line now reads, *Vertical: Pick next dimension point*.
3. Click the bottom point.

   The dimension appears. You can drag it to a new location. Dimension and extension lines automatically redraw.

   Click the points in the opposite order if you want the dimension to display below the objects.

Vertical Base Line Dimension Tool





This tool dimensions objects or spaces from a base point.

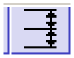
Using the Vertical Base Line Dimension Tool

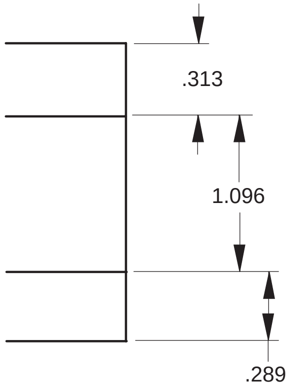
1. Select the Vertical Dimension tools. Choose the Vertical Base Line Dimension tool. The Message Line reads: *Vertical Base Line: Pick first dimension point*.
2. Click the base point of the geometry.

   The Message Line now reads, *Vertical Base Line: Pick next dimension point*.
3. Click the point. The dimension appears.
4. Click the next place for the dimension. This dimension appears to the right of the first measured from the base point. Continue clicking all the points you want dimensioned. You can drag each dimension to a new location. Dimension and extension lines automatically redraw.

   Click the points in the opposite order if you want the dimension to display below the objects.

Vertical Chain Dimension Tool





This tool dimensions objects or spaces from end to end, vertically.

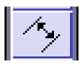
Using Vertical Chain Dimension Tool

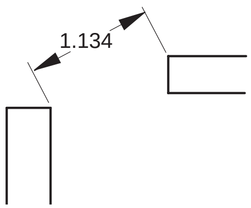
1. Select the Vertical Dimension tools. Choose the Vertical Chain Dimension tool. The Message Line reads: *Vertical Chain: Pick first dimension point.*
2. Click the first point on the geometry.

   The Message Line now reads, *Vertical Chain: Pick next dimension point*.
3. Click the point. The dimension appears.
4. Click the next place for the dimension. This dimension appears measured from the last point clicked. Continue clicking all the points you want dimensioned. You can drag each dimension to a new location. Dimension and extension lines automatically redraw.

   Click the points in the opposite order if you want the dimension to display below the objects.

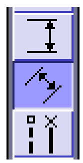
Oblique Dimension Tool





This tool measures space or objects parallel or point to point.

Using the Oblique Dimension Tool



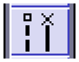
1. Select the Oblique Dimension tool. The Message Line reads: *Oblique: Pick first dimension point*.
2. Click the left point of the geometry first.

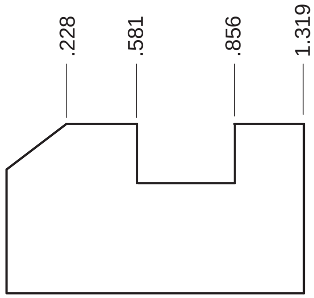
   The Message Line now reads, *Oblique: Pick next dimension point*.
3. Click the right point.

   The dimension appears. You can drag it to a new location. Dimension and extension lines automatically redraw.

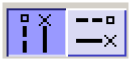
Click the points in the opposite order if you want the dimension to display below the objects.

Ordinate Dimension Tools





This tool dimensions objects or spaces from a base point. Use the Selection tool to move, position or create an elbow ordinate.

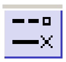


Using the Horizontal Ordinate Dimension Tool

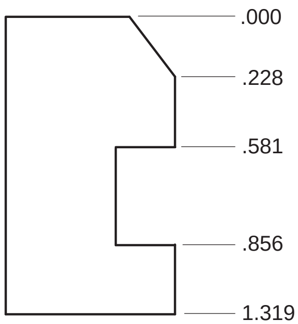
1. Select the Horizontal Ordinate Dimension tool from the Message Line. The Message Line reads: *Horizontal Ordinate: Pick first dimension point*.
2. Click the base point of the geometry.

   The Message Line now reads, *Horizontal Ordinate: Pick second dimension point*.
3. Click the next point. The dimension appears, measured from the base point.
4. Click the next point for the dimension. This dimension appears measured from the base point. Continue clicking all the points you want dimensioned. If you want to display a dimension at the base point, click the base point after you have dimensioned all other points.

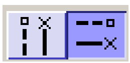
   You can drag each dimension to a new location. Dimension and extension lines automatically redraw.



Using the Vertical Ordinate Dimension Tool



1. Select the Horizontal Ordinate Dimension tool. The Message Line reads: *Vertical Ordinate: Pick first dimension point*.

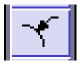
   
2. Click the base point of the geometry.

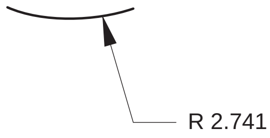
   The Message Line now reads, *Vertical Ordinate: Pick* *second dimension point*.

3. Click the next point. The dimension appears, measured from the base point.
4. Click the next point for the dimension. This dimension appears measured from the base point. Continue clicking all the points you want dimensioned. If you want to display a dimension at the base point, click the base point after you have dimensioned all other points.

You can drag each dimension to a new location. Dimension and extension lines automatically redraw.

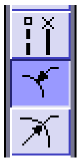
Radial Arrow Out Dimension Tool





This tool measures the radius of a circle, arc, or fillet with the arrow on the side of the arc indicated when the arc is picked.

Using the Radial Arrow Out Dimension Tool



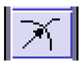
1.Select the Radial Arrow Out Dimension tool. The Message Line reads: *Radial Arrow Out: Select arc/circle*.

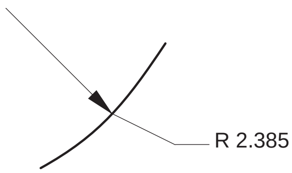
2.Click near the circle, arc or fillet you want to dimension.

The dimension is placed on the outside of the selected object. When the dimension appears the leader line is placed at the nearest 15° increment from the location you clicked. You can move the dimension by choosing the Selection tool, selecting the dimension, placing the pointer over the end of the leader line and when the pointer becomes the Move symbol, dragging the text to a new location.

For 3D geometry, radial dimensions are created in the plane of the arc or fillet. This tool can only be used on curves, not surfaces or solids.

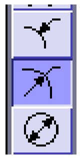
Radial Arrow In Dimension Tool





This tool measures the radius of a circle, arc, or fillet with the arrow inside the geometry.

Using the Radial Arrow In Dimension Tool



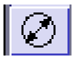
1. Select the Radial Arrow In Dimension tool. The Message Line reads: *Radial Arrow In: Select arc/circle*.

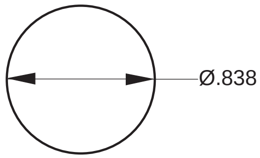
2. Click near the circle, arc or fillet you want to dimension.

   The dimension text appears inside the object. The arrow line starts from the arc center. When the dimension appears the leader line is placed at the nearest 15° increment from the location you clicked. You can move the dimension by choosing the Selection tool, selecting the dimension, placing the pointer over the end of the leader line and when the pointer becomes the Move symbol dragging the text to a new location.

For 3D geometry, radial dimensions are created in the plane of the arc or fillet. This tool can only be used on curves, not surfaces or solids.

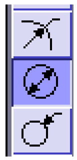
Diameter Arrow In Dimension Tool





This tool measures the diameter of a circle.

Using the Diameter Arrow In Dimension Tool



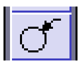
1. Select the Diameter Arrow In Dimension tool. The Message Line reads: *Diameter Arrow In: Select arc/circle*.

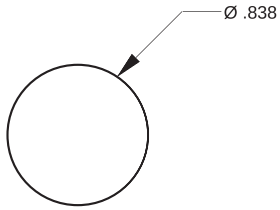
2. Click near the circle or arc you want to dimension.

   The dimension appears inside the object (depending on the circle and font size). When the dimension leader line is placed at the nearest 15° increment from the location you clicked. Move the dimension by choosing the Selection tool, selecting the dimension, placing the pointer over the end of the leader line and when the pointer becomes the Move symbol, dragging the text to a new location.

For 3D geometry, diametric dimensions are created in the plane of the circle. This tool can only be used on curves, not surfaces or solids.

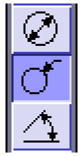
Diameter Arrow Out Dimension Tool





This tool measures the diameter of a circle.

Using the Diameter Arrow Out Dimension Tool

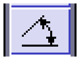


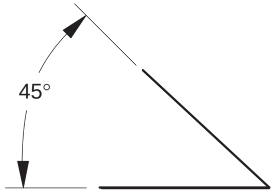
1. Select the Diameter Arrow Out Dimension tool. The Message Line reads: *Diameter Arrow Out: Select arc/circle*.
2. Click near the circle or arc you want to dimension.

   The dimension appears outside the object. When the dimension appears the leader line is placed at the nearest 15° increment from the location you clicked. You can move the dimension by choosing the Selection tool, selecting the dimension, placing the pointer over the end of the leader line and when the pointer becomes the Move symbol dragging the text to a new location.

For 3D geometry, diametric dimensions are created in the plane of the circle. This tool can only be used on curves, not surfaces or solids.

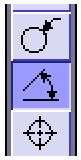
Angular Dimension Tool





This tool measures the angle between two lines.

Using the Angular Dimension Tool



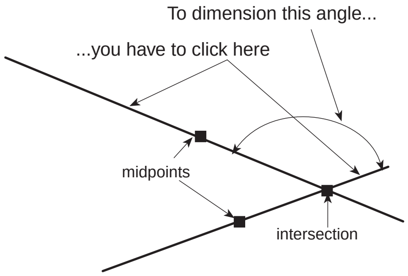
1. Select the Angular Dimension tool. The Message Line reads: *Angular: Pick first line*.
2. Click on the first line near the endpoint from which you want the angle measured.

   The Message Line now reads, *Angular: Pick second line*.
3. Click on the second line.

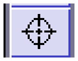
   The inside angle is measured between the lines. This Designer Elements program always measures the smaller angle between the selected lines.

If you dimension intersecting lines don't confuse the intersection with the midpoint of the lines.

The Angular Dimension tool will not dimension angles over 180°. For 3D geometry, Angular dimensions are created in the plane of the two lines. This tool can only be used on curves not surfaces or solids.

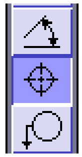


Center Mark Dimension Tool



This tool creates a center line mark for circles and arcs. The center mark overlap can be defined in the Status Line or the Edit Objects dialog box. The overlap units are based on those in the Units page of Preferences.

Using the Center Mark Dimension Tool



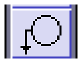
1. Select the Center Mark Dimension tool. The Message Line reads: *Center Mark: Select circle for center mark [Shift = Extend]*.

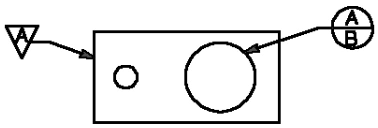
2. Select the circle or arc you want to dimension. Hold down the SHIFT key if you want to dimension more than one object at the same time.

   You can change the Axis overlap value in the Status Line. Type the value and press ENTER (Windows) or RETURN (Macintosh) and the overlap updates.

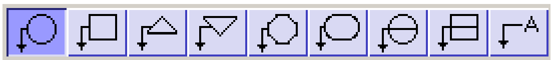
The Status Line contains the Axis overlap data field.

Balloon Dimension Tools





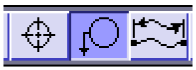
These tools dimension objects with a callout balloons. You have nine balloon tools available in the Message Line: Circle, Rectangle, Triangle, Inverted Triangle, Octagon, Ob-round, Split Circle, Split Rectangle and Callout.



Use the Status Line or the Edit Objects dialog box to change the text or width of the balloon dimension. The width units are based on those set in the Units page of Preferences.

Using a Balloon Dimension Tool

1. Select the Balloon Dimension tool. The Message Line reads: *Balloon: Enter 2 points for balloon dimension*.



2. Click the point on the geometry for the dimension arrow.
3. Click the second point to specify the location of the symbol.

   The balloon symbol appears.
4. Enter the text and frame width in the Status Line data fields. Press ENTER (Windows) or RETURN (Macintosh) to update the dimension.

   You can move the dimension by choosing the Selection tool, selecting the dimension, placing the pointer over the end of the leader line and when the pointer becomes the Move symbol, dragging the text to a new location.

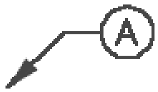
Balloon dimension text does not increment as you place additional balloons. If you want specific text in the balloon dimension, you must enter it yourself.

The Status Line contains Text and Width data fields.

Geometric Characteristics

According to the Edit Objects dialog box a balloon dimension includes the following characteristics: Frame, Width, Text 1, Text 2 (when a split balloon is used) and Extension. The Frame pull-down menu lists all available balloon symbols. Width defines the symbol width. Text 1 sets the upper text in a split balloon. Text 2 sets the lower text in a split balloon.

The extension is the line distance from the symbol to the leader line. In the graphic here the horizontal line is the extension.



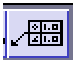
Length Along Curve Dimension Tool

This tool dimensions curved objects such as splines.

Using the Length Along Curve Dimension Tool

1. Select the Lingth Along Curve Dimension tool. The Message Line reads: *Length Dimension: Specify two points along curve*.
2. Click the first point along the curve.
3. Click the second point along the curve.

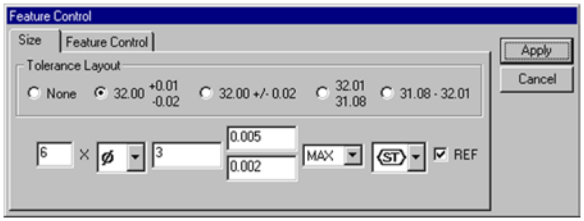
GDT Feature Control Tool



Selecting this icon from the dimensioning tool palette activates the new GD&T tool. The prompt message requests the input of 2 leader points. The first point defines the arrow location. The second point defines the shoulder location. After the leader points have been specified, the Feature Control dialog box is displayed. This dialog box contains the Size and Feature Control tab pages.

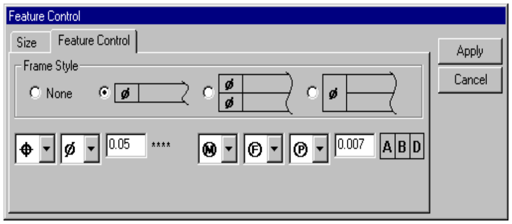
Size Tab Page

The Size tab page is used to define the basic dimension and tolerance values along with other optional modifiers. This size specification is also known as the Limits of Size. Selecting one of the five Tolerance Layout radio buttons controls the tolerance style. The dialog controls are re-positioned to match the current Tolerance Layout selection.



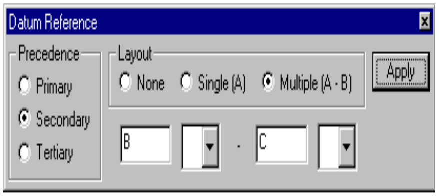
Feature Control Page

The Feature Control tab page is used to define the geometric relationships for the dimensioned feature. Selecting one of the four Frame Style radio buttons controls the layout for the feature control frames. The supported frame styles are: None, Single, Dual, and Composite. The dialog controls are re-positioned to match the current Tolerance Layout selection. The Feature Control tab also uses 2 additional support dialogs; Unit Basis and Datum Reference. The Unit Basis dialog is displayed by clicking on the region of the dialog box. The Datum Reference dialog is displayed by clicking on the region of the dialog box.



Unit Basis Dialog

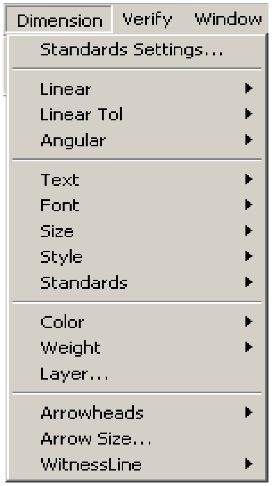
When appropriate for a feature, the Unit Basis Dialog defines the unit basis interval (subregion) over which the tolerance value is applied. The unit basis interval can be one of "None", "Linear", and "Area". A "None" interval (the default) applies the allowed tolerance over the entire feature. A "Linear" interval applies the allowed tolerance per any linear sub-length along the feature. An "Area" interval applies the allowed tolerance per any rectangular sub-area of the feature. If a linear interval is defined, the Feature Control tab page will display a /L indicator. If an area interval is defined, the Feature Control tab page will display a /LxW indicator.



Datum Reference Dialong

When appropriate for a feature, the Datum Reference Dialog associates existing datum planes (local coordinate origin) with the dimension. A maximum of 3 datums (Primary, Secondary, Tertiary) can be associated with a feature control frame. A datum must be fully defined in order to add another datum reference. A datum consists of a datum letter (any combination of the letters A-H,J-N,P,R-Z. I,O, and Q are not allowed) and an optional modifier symbol. For a multiple datum, a second datum letter and optional modifier symbol follow a dash.

Dimension Appearance



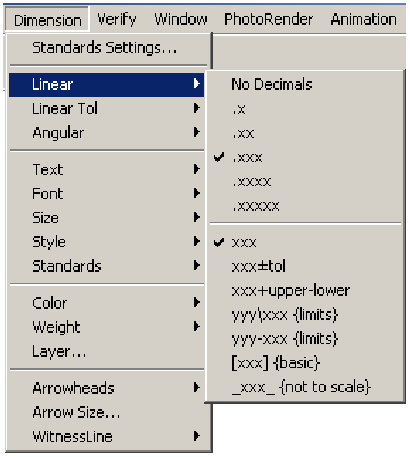
This Designer Elements program automatically creates dimensions according to ANSI Y14.5, DIN, ISO or JIS standards. However, many companies and individuals have developed their own standards. The commands in the Dimension menu allow you to control virtually every aspect of the dimension appearance without having to construct dimensions manually and while retaining the associativity of Vellum's smart dimensions.

Settings made in the Dimension menu affect the currently selected dimension and all future dimensions.

Linear

Tech Note:

The number of decimals displayed with your dimensions is not affected by the number of decimals specified for units in Preferences.



This command in the Dimension menu allows you to set the decimal precision.

All dimensions in this Designer Elements program are displayed in decimals regardless of the units set in Preferences.

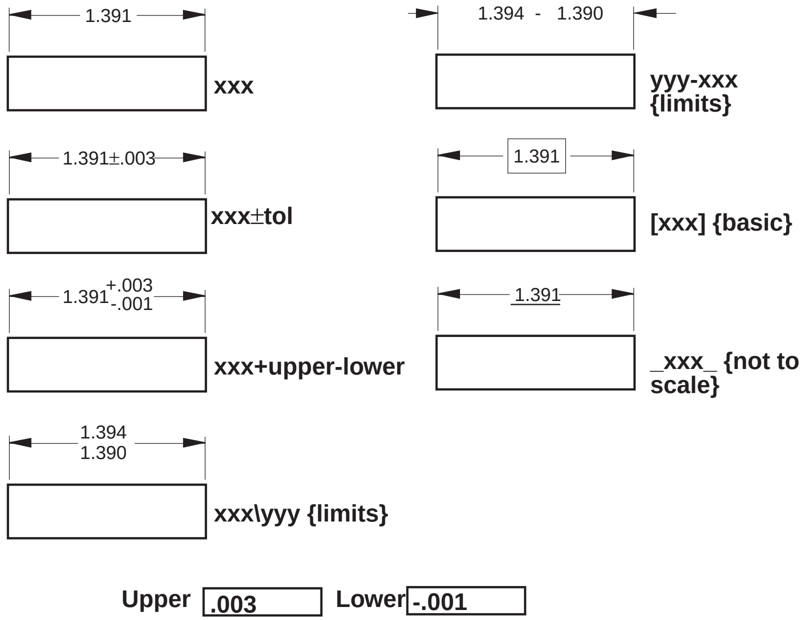
If Feet/inches units is set in Preferences, objects over one foot in length will display both feet and inches.

This Designer Elements program does not support dimensioning in fractions. The default is three decimal places.

Linear Tolerance

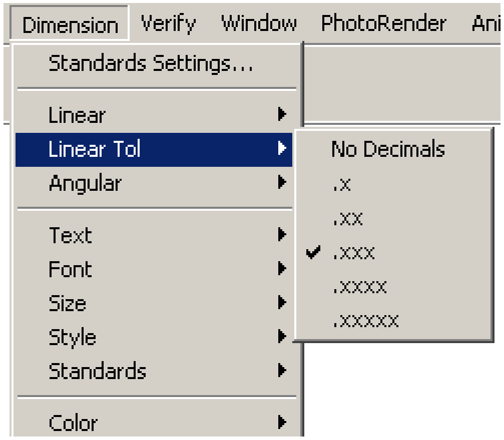
The Linear command in the Dimension menu allows you to set the tolerance for the following Linear Dimension tools: Horizontal Dimension tool, Vertical Dimension tool, Oblique Dimension tool, Angular Dimension tool and Smart Dimension tool. Choose ***Dimension>Linear*** to select one of the seven tolerance options.

The following graphic shows the appearance of each tolerance option if you entered the limits shown in the Status Line.



When you select a Dimension tool, such as the Horizontal Dimension tool, the Status Line includes data fields for the upper and lower tolerances, as shown here.

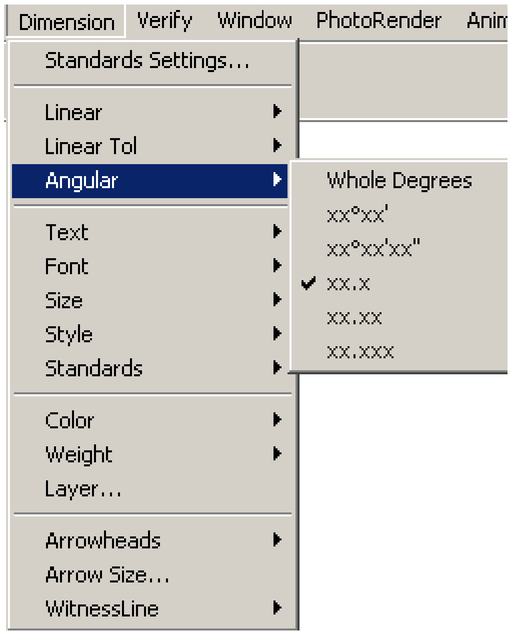
To set the number of decimal places for the linear tolerance, choose ***Dimension>Linear Tol***.



Angular

This command in the Dimension menu allows you to set the format of Angular dimensions.

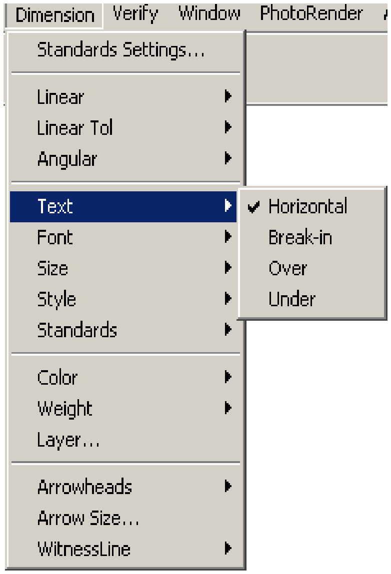
You can set degrees, minutes and seconds for angular dimensions. The default format is degrees and minutes.

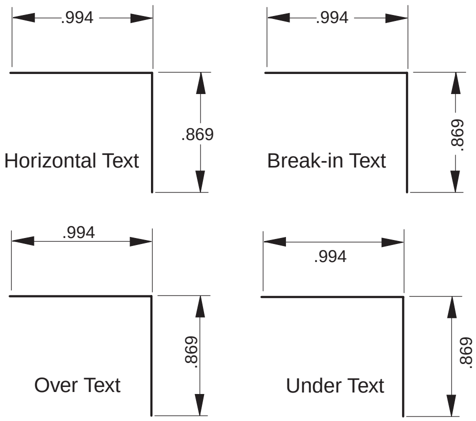


Text

This command in the Dimension menu allows you to specify the position and orientation of dimension text.

The default is Horizontal text. The Break-in, Over and Under options produce text that is aligned with dimension leader lines. Over and Under display text above or below leader lines while Break-in places the text between leader lines.





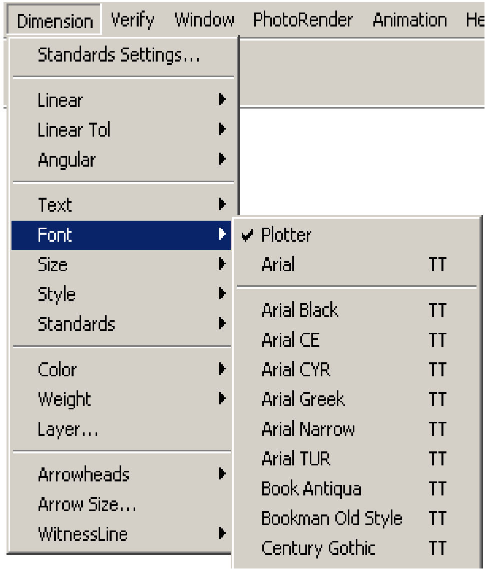
Tip:

This command only affects dimensions. To change the font of other text objects, choose **Text>Font**.

The Font submenu in the Text menu has no effect on dimensions.

Font

This command in the Dimension menu lets you pick the font for dimension text. The fonts in this list are the same as those available in the Font submenu from the Text menu.



Tip:

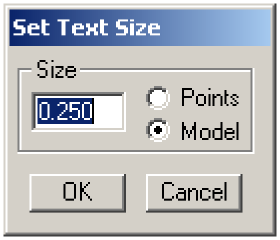
This command only affects dimensions. To change the size of other text objects, use Edit Objects or the Size sub-menu from the Text menu.

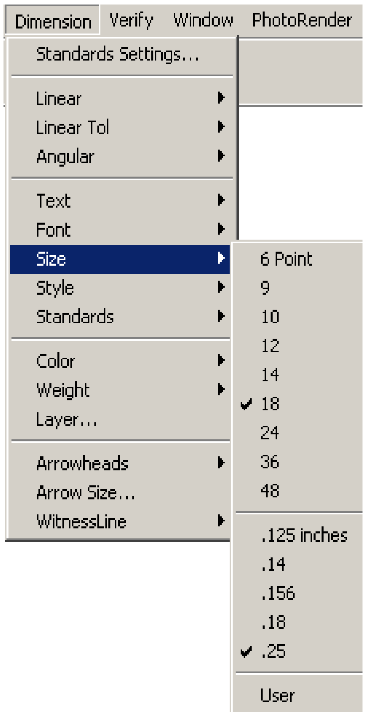
The Size submenu from the Text menu has no effect on dimensions.

Size

This command in the Dimension menu lets you pick the size of dimension text only. The sizes in this list are the same as those available in the Size submenu from the Text menu.

Selecting the User option allows you to specify a non-standard font size for dimension text.

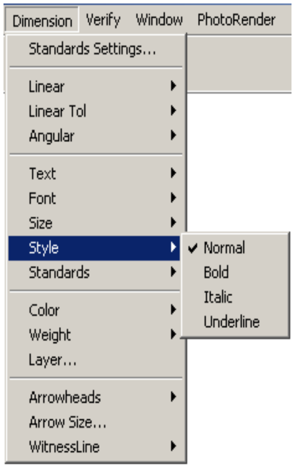




Selecting the Model option displays the text size in the Units you specified in Preferences. Selecting the Points option, displays the point size of the text. Select the desired radio button, enter a value in the data field and click OK.

Style

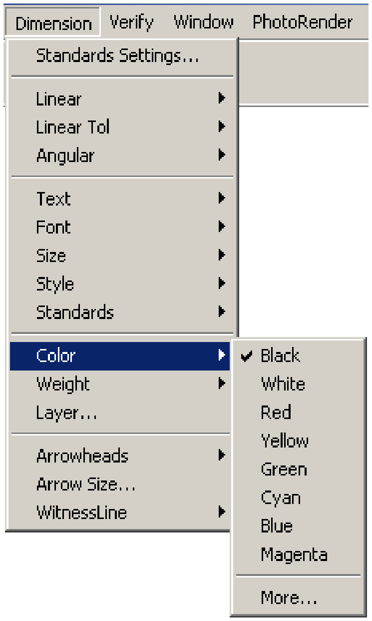
This command in the Dimension menu lets you pick the style of dimension text, Normal, Bold, Italic and Underline. The styles in this list are the same as those available in the Style submenu from the Text menu. However, these settings do not affect those in the Text menu.



Tip:

This command only affects dimensions. To change the style of other text objects, use the Style submenu from the Text menu.

The Style submenu from the Text menu has no effect on dimensions.



Color

This command in the Dimension menu allows you to specify the color of the dimension. The default color is blue.

Tip:

This command only affects dimensions. To change the color of other objects, use Edit Objects or the Color submenu from the Pen menu.

The Color submenu from the Pen menu has no effect on dimensions.

Tip:

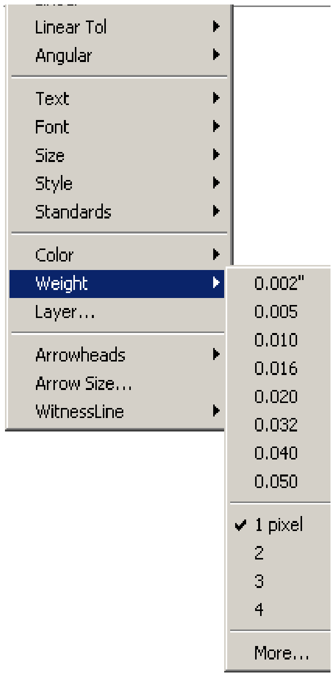
This command only affects dimensions. To change the weight of other objects, use Edit Objects or the Weight submenu from the Pen menu.

The Weight submenu from the Pen menu has no effect on dimensions.

Weight

This command in the Dimension menu allows you to specify the weight of dimension lines.

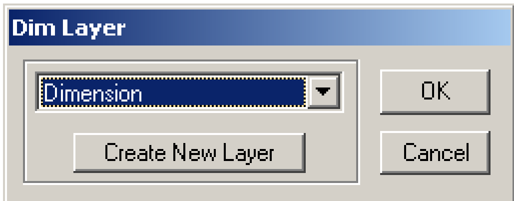
Dimension lines are typically drawn in the thinnest weight available. For this Designer Elements program, that is 0.002" or 0.05mm. The default weight is 0.002" or 0.05mm.



Layer

This command in the Dimension menu allows you to specify the layer on which dimensions are placed. This command does not apply to dimensions placed within drawing views. These are automatically placed on the Sheet View layer. See Chapter 36 for information on drawing views.

When you select this command the following dialog box appears:



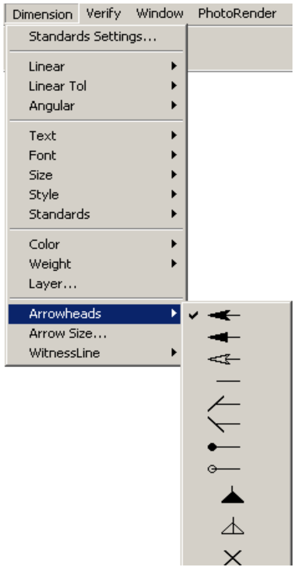
Dimensions can be placed on any visible layer. In the pull-down menu you can select a layer for your dimensions. The default layer is the Dimension layer.

You can place the dimensions on a new layer not yet created by clicking the Create New Layer button. The data field displays the new layer name. Give the layer a new name if desired. Click OK to save the change.

If the layer on which dimensions are to be placed is hidden or deleted, future dimensions will be placed on the current work layer.

Arrowheads

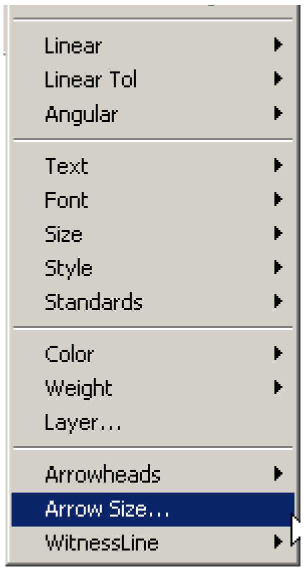
This command in the Dimension menu allows you to specify the type of arrowhead used for dimensions.



Tip:

This command only affects dimensions. To select the arrowhead type for lines or arcs, use the Arrowheads submenu from the Pen menu.

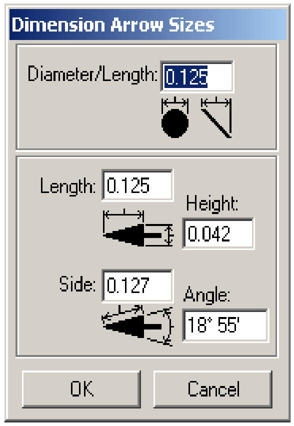
The Arrowheads submenu from the Pen menu has no effect on dimensions.



Arrow Size Command

This command in the Dimension menu lets you specify the size of the arrowhead that you have selected.

When you select this command the following dialog box appears: This dialog box includes:



Diameter/Length This value is the diameter or length of dots, slash and standard arrow styles. The value here affects all Length, Side and Angle fields. The value displayed is in the current units specified in the Units page of the Preferences dialog box.

Length This value is the length of the arrowhead as the horizontal distance from its tip to the furthest extension of its base.

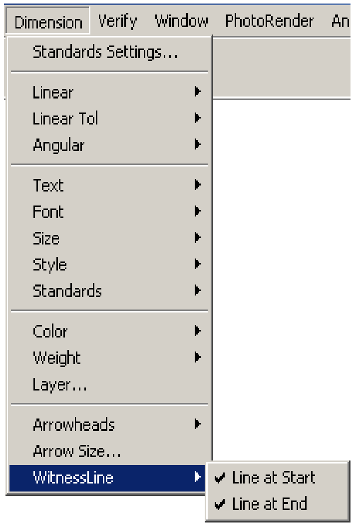
Height This value is the height of the arrowhead as the vertical distance of its base.

Side This value is the edge length of the arrowhead.

Angle This value is the angle of the tip of the arrowhead.

If you change any value in the Length, Height, Side, or Angle entry fields, this Designer Elements program will change the values in the other entry fields accordingly.

Witness Lines



This command in the Dimension menu lets you specify which sides of the linear dimension should have witness lines.

A check mark indicates that a witness line will be placed at the specified location. This option is useful to plot a drawing that contains baseline or chain dimensions. By turning off one or more of the overlapping witness lines, you can prevent the plotter from drawing multiple witness lines when only one is needed.

The Start of a dimension is the first point clicked. The End of a dimension is the last point clicked. The default settings have witness lines at both the start and end of a dimension.

This command affects only linear dimensions. Radial and diametric dimensions are not affected by the settings made in the Witness Lines command.

Dimension Standards

The dimension standards feature has been modified to allow you to manage and define your own standards and specify which standard is used from the Standards submenu in the Dimension menu.

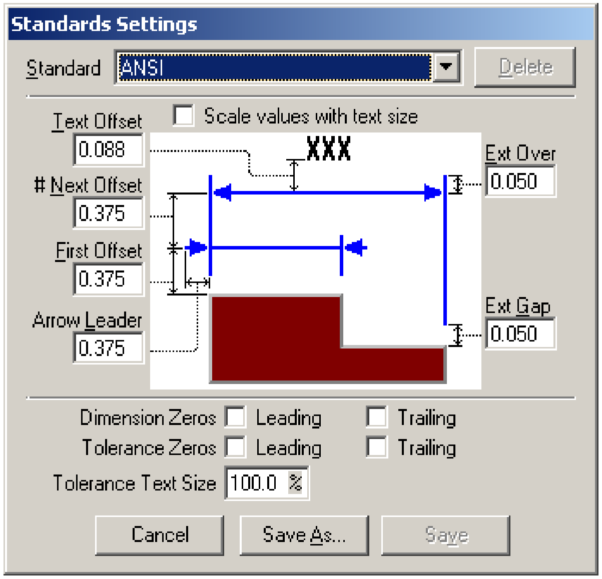
A user-defined standard contains all the current Dimension menu settings at the time the Standard Settings dialog box appears. This includes dimension text font, style and size. It does not include the Witness Line setting.

Tech Note:

The default dimension standard is ANSI. The actual setting does not save with files.

Standards Settings Manager

Dimension standards are set through the Standards Settings dialog box displayed by choosing ***Dimension>Standards Settings***.



Standard This pull-down lets you set the dimension standard. This Designer Elements program contains settings for ANSI, DIN, ISO, JIS and the new Architecture, Dual Side, Dual Stacked and Stacked Fraction default standards. These default standards cannot be altered. However you can create user-defined standards and save them under their own name. Examples of each default standard follow this section.

Text Offset This entry field controls the distance from the base of the dimension text to the leader lines. A positive entry places the text above the leader lines, and a negative value places it below the leader lines. A zero value places the text on the leader lines.

 # Next Offset This entry field controls the perpendicular distance between subsequent leader lines.

First Offset

This entry field controls the perpendicular distance between the object being dimensioned and the leader line and dimension text.

Changes to this value only affect new dimensions.

Arrow Leader

This entry field controls the distance the leader lines extend beyond the witness lines when the arrows appear outside of the witness lines.

Ext Over

This entry field controls the distance the witness lines extend beyond the leader lines.

Ext Gap

This entry field controls the distance between the dimension point and the witness lines.

Delete

Click this button to delete a user-defined standard. (This button is unavailable for the ANSI, DIN, ISO, JIS, Architecture, Dual Side, Dual Stacked and Stacked Fraction default standards.)

Cancel

Click this button to close the dialog box without accepting any changes.

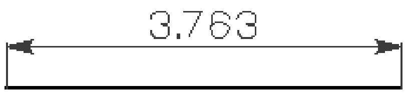
Save As

Click this button to save the settings for a user-defined standard under a new name.

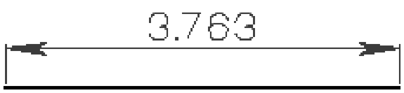
Save

Click this button to save changes to a user-defined standard. (This button is unavailable for the ANSI, DIN, ISO, JIS, Architecture, Dual Side, Dual Stacked and Stacked Fraction default standards.)

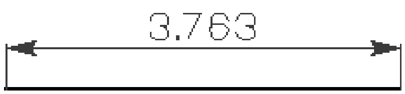
ANSI Standard



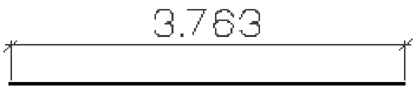
DIN Standard



ISO Standard

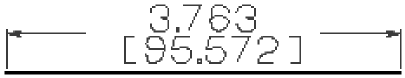


JIS Standard

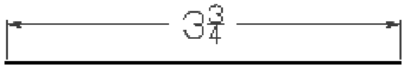


Architectural Standard

Dual Side Standard



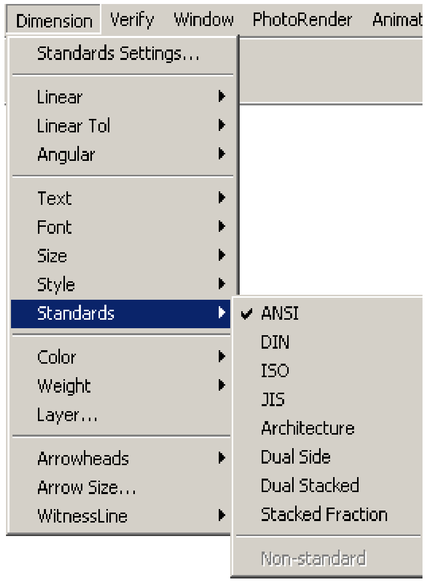
Dual Stacked Standard



Stacked Fraction Standard

Setting the Standard

A Standards menu allows you to set the standard as the default or change the standard for a selected dimension. If you have created user-defined standards, they are also listed in the Standards menu.



Choosing a Standard

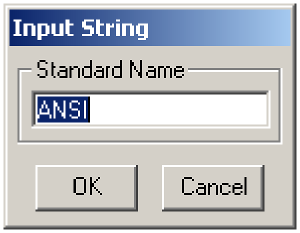
You can choose one of the predefined standards or a user-defined standard. Changing the standard will also change the Dimension menu settings. If no dimension is selected, you are setting the default standard. If one or more dimensions are selected, you are only changing the standard for the selected dimensions.

1. Choose ***Dimension>Standards***.
2. In the Standards submenu, choose one of the listed standards.
3. Click OK to accept the standard as the default and close the dialog box.

Creating a User-defined Standard

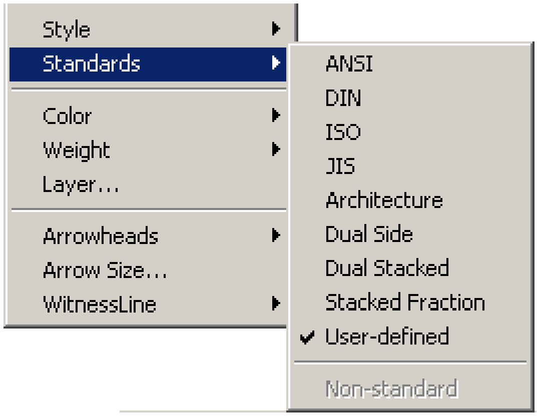
1. Choose ***Dimension>Standards Settings***.
2. Enter values in the desired fields.
3. Click Save As.

An Input String dialog box appears with a Standard Name data field.



4. Enter a new name in the data field.
5. Click OK to close the dialog box and save the new standard.

You can later delete this user-defined standard through the Standards Settings dialog box. This standard also adds to the Standards menu.



Dimensions and Drawing Views

If dimensions in the drawing view go outside the bounds of the view so that you can't see them, choose the Frame to Extent command in the Drawing View menu. You can also drag the view edge manually to completely display the dimension. See Chapter 36 for information on drawing views.

Dimensions created in drawing views are placed on the Sheet View layer rather than the Dimension layer. This allows you to turn off all other layers including the Dimension layer when printing the sheet. This is helpful when you have dimensioned the original geometry and do not wish to print those dimensions.

Fill and Crosshatching

This Designer Elements program provides you with the ability to apply fill to smart polygons and to apply crosshatching to your geometry. This is especially useful for setting certain objects apart visually.

This chapter covers the following topics:

- Fill Pattern and Fill Color
- Crosshatching

Fill Pattern and Fill Color

This Designer Elements program provides Fill Patterns and Fill Color for *Smart Polygons* created with the Polygon tools: Rectangle, Inscribed Polygon, Circumscribed Polygon, Arbitrary Polygon and Polygon from Curves. These patterns and colors enable you to highlight the polygons or separate overlapping polygons visually in your drawing. After creating the polygon, you place a fill pattern and fill color in the polygon. If you make changes to the polygon the pattern and color automatically update.

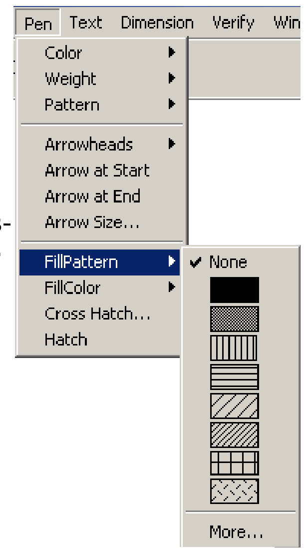
Tech Note:

Fill Pattern and Fill Color are not the same as Crosshatching or Solid Fills. Fill Pattern and Fill Color are predefined bitmaps and cannot be edited.

In the Pen menu you will find the two fill commands—*Fill Pattern* and *Fill Color.*

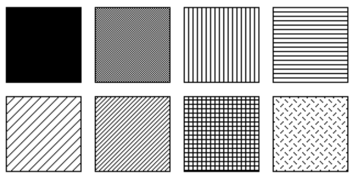
These commands work with each other. Using the *Fill Pattern* command, choose the pattern for your polygon. Using the *Fill Color* command, choose the pattern color.

Fill Pattern

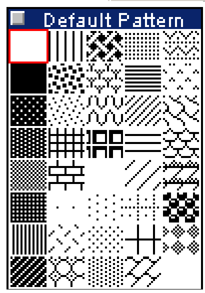


A wide variety fill patterns are available. Patterns appear parallel to the work plane and only in the view in which they were added. In other words, patterns are drawn correctly when viewed along the z-axis of the work plane.

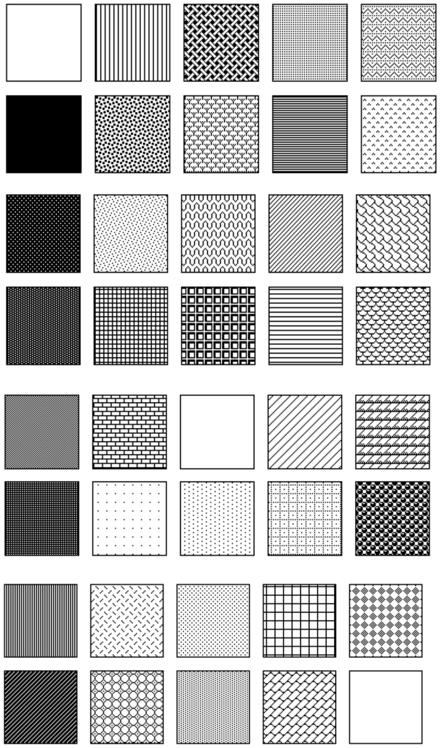
When you select the *Fill Pattern* command a submenu displays. You have the option to choose None (for no fill) or one of eight patterns (including solid fill) as shown here.



If you select ***Pen>Fill Pattern>More***, the Patterns option box appears providing you more patterns.



The patterns are enlarged below for clarity.

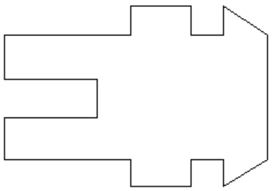


You cannot edit these patterns. They are predefined bitmaps. The default pattern is None.

Choosing a Fill Pattern

1. Choose the Selection tool.
2. Select the polygon that you want to display with a fill pattern. Hold down the SHIFT key to select more than one polygon.

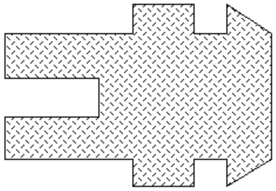
   One example is displayed here.



3. Choose ***Pen>Fill Pattern*** and one of the fill patterns.

   (If you choose a fill pattern from the Patterns option box, a selection box will appear around the pattern.)

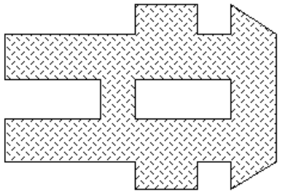
The polygon is filled with the selected pattern.



Fill Patterns and Holes

This Designer Elements program does not support the placement of “holes” in polygons. However, you can place a polygon within another polygon and choose a solid white fill to give the illusion of a hole.

Using the example above you would get the following result.

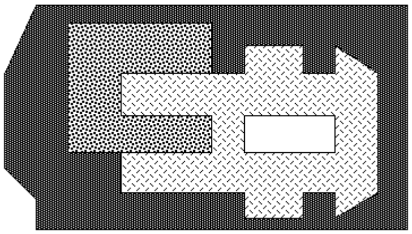


Overlapping Objects and Patterns

Since you can have multiple objects with multiple patterns displayed in your drawing, there may be times when some overlapping occurs.

You can specify the order of the overlapping in two ways:

- Select the objects with the Selection tool in the reverse order that they are displayed on the screen
- Adjust the placement of the objects using the *Arrange* command in the Layout menu (see Chapter 24 for more information on using this command).



You can use the *Group* command in the Layout menu to keep the patterned objects in the specified ordered.

Changing the Existing Pattern of a Polygon

1. Select the existing polygon with the Selection tool.

2. In the Pen menu, choose a new pattern from the Fill Pattern submenu.

   The fill pattern changes as you specified.

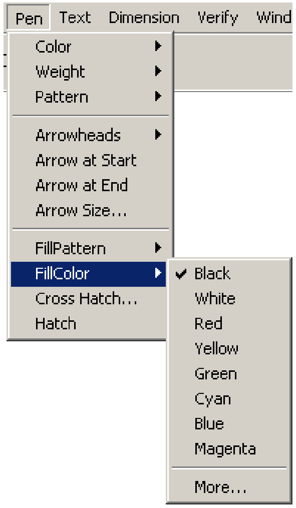
Removing the Fill Pattern

1. Select the existing polygon with the Selection tool.
2. In the Pen menu, choose None from the Fill Pattern submenu.

   The fill pattern is removed.

You can also double-click the object to display the Edit Objects dialog box and change the Fill characteristic from Yes to No.

Fill Color



This command in the Pen menu works only with the *Fill Pattern* command.

The submenu displays the same colors available for creating geometry (see Chapter 5, “Pen Settings”).

You can choose a specific color for all fill patterns. The default color is black.

Choosing a Fill Color

1. Choose the Selection tool.
2. Select the polygon you want to fill.

   Hold down the SHIFT key to select more than one polygon.
3. Select the fill color. The color displays.
4. Choose ***Pen>Fill Color*** and select the desired color.

   The new fill pattern color is displayed in your polygon.

You cannot choose a fill color for a polygon that does not contain a pattern.

Changing the Color of an Existing Fill

1. Select the existing polygon by clicking it with the Selection tool.
2. In the Pen menu, choose a new color from the Fill Color submenu.

   The fill pattern color changes as you specified.

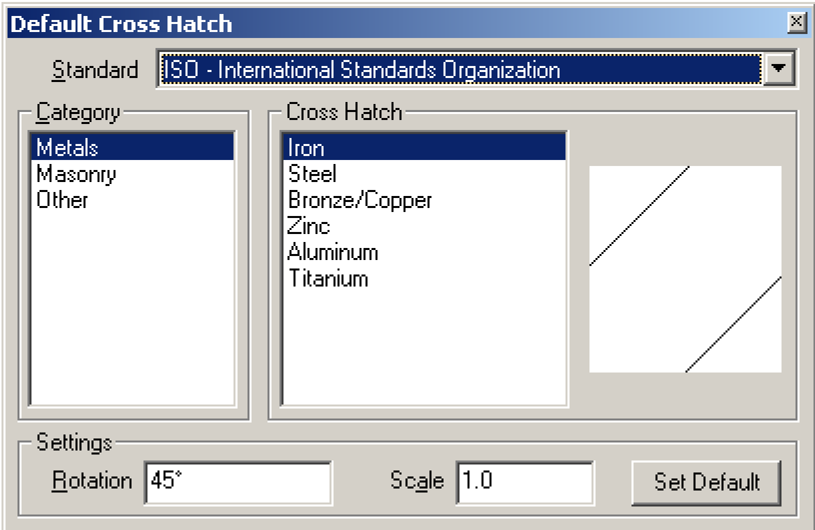
Crosshatching

This Designer Elements program can crosshatch any enclosed area in your drawing and then automatically update the crosshatching when you change the geometry. You can select a closed area containing a hole or cutout and this Designer Elements program accurately excludes the cutout area from the crosshatching. Crosshatching can also be applied to section cuts made through your geometry when using the drawing composition tools.

The Pen menu contains two crosshatching commands: *Cross Hatch* and *Hatch*. Use these to apply crosshatch patterns to your geometry.

Cross Hatch Patterns

You have a wide variety of crosshatch patterns available to you in both ISO and DIN styles. Choosing ***Pen>Cross Hatch*** brings up a dialog box. If no objects are selected the dialog box is called, *Default Cross Hatch*. If objects are selected, the dialog box is called *Object Cross Hatch*.



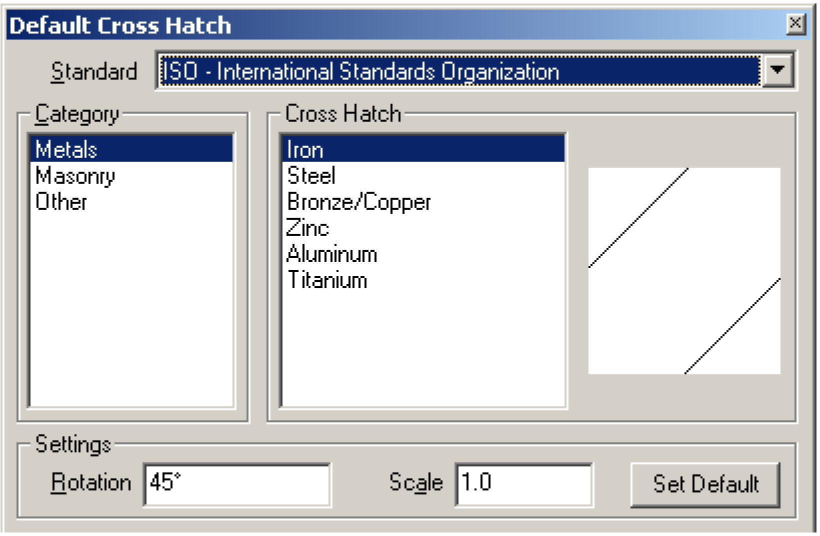
The dialog box contains the following options:

Standard This option sets the crosshatching standard. Select either the ISO (International Standards Organization) standard or the DIN (German Standards Institute) standard from the pull-down menu.

Category	This section lists the hatch categories available for the selected standard.
Cross Hatch	This section lists the crosshatch patterns available for the selected category.
Pattern Window	This window, to the right of the Cross Hatch list, displays the selected crosshatch pattern.
Rotation	(Settings) This field sets the rotation angle of the crosshatch pattern.
Scale	(Settings) This field sets the scale of the crosshatch pattern.
Set Default	(Default Cross Hatch dialog box) Click this button to set the default crosshatch pattern for your drawing.
Apply	(Object Cross Hatch dialog box) Click this button to apply the crosshatch pattern to the selected object.

ISO Patterns

If you choose the ISO standard the ISO categories and crosshatch patterns display in the dialog box.



The categories for the ISO standard are:

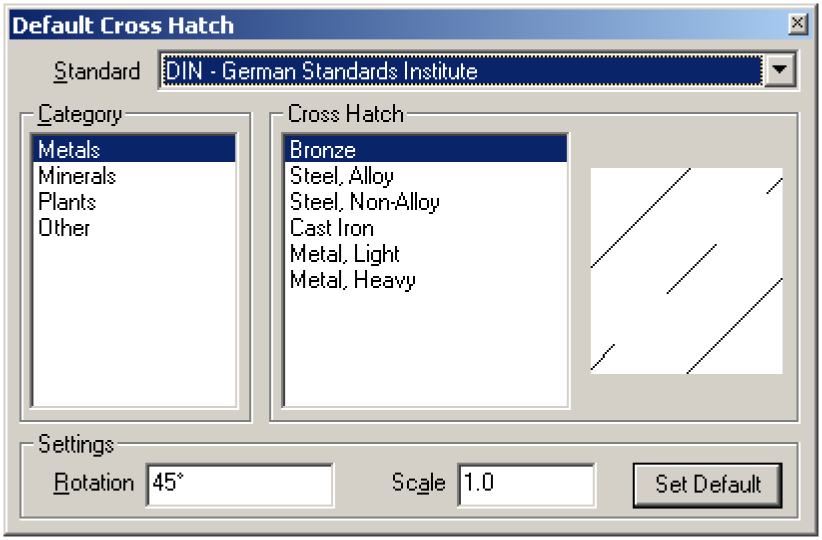
Metals The patterns in this category include: Iron, Steel, Bronze/Copper, Zink, Aluminum and Titanium.

Masonry The patterns in this category include: Brick and Concrete.

Other The patterns in this category include: Rubber, Electric, Marble/Glass, Thread, Grass, Earth, Fabric, Insulation and Mud.

DIN Patterns

If you choose the DIN standard the DIN categories and crosshatch patterns display in the dialog box.



The categories for the DIN standard are:

Metals The patterns in this category include: Bronze, Steel (Alloy), Steel (Non-Alloy), Cast Iron, Metal (Light) and Metal (Heavy).

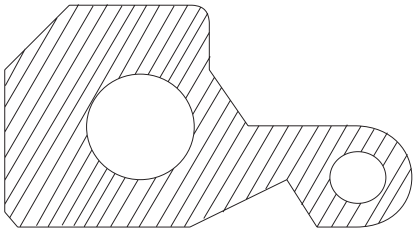
Minerals The patterns in this category include: Rock, Sand, Clay, Peat/Humus Soil, Coal, Salt and Sandstone.

Plants The patterns in this category include: Wood (Horizontal Grain), Wood (Vertical Grain), Wood (Materials), Wood (Cut) and Tar.

Other The patterns in this category include: Masonry (Brick Work), Masonry (Increased Strength), Masonry (Light Brick), Masonry (Pumice) Plaster Plate, Plaster (Mortar I), Plaster (Mortar II), Concrete (Reinforced), Concrete (Non-reinforced), Concrete (Pumice) and Concrete (Waterproof).

Crosshatch Command

This command in the Pen menu enables you to apply crosshatching to your geometry.



Setting the Default Hatch Patterns

When no objects are selected, you can set the default crosshatch pattern for the current session of this Designer Elements program.

1. Choose ***Pen>Cross Hatch***. The Default Cross Hatch dialog box appears.
2. Choose the crosshatch standard from the pull-down menu.
3. Choose the category and the crosshatch pattern. The pattern appears in the pattern window.
4. Specify a rotation angle and scale.
5. Click Set Default. The default crosshatch pattern is set.
6. Click the Close button (Windows) or the Close box (Macintosh) to close the dialog box.

   If you choose the *Hatch* command in the Pen menu this default pattern is applied to your selected geometry.

Applying a Crosshatch Pattern for an Object

If one or more objects are selected, you can choose a specific crosshatching for them without changing the default.

1. Select the object you want to crosshatch.
2. If you want to apply the default crosshatch, choose ***Pen>Hatch*** and the object is crosshatched.

   If you want to apply a different crosshatch, choose ***Pen>Cross Hatch***. The Object Cross Hatch dialog box appears.
3. Choose the crosshatch standard from the pull-down menu.
4. Choose the category and the crosshatch pattern. The pattern appears in the pattern window.
5. Specify a rotation angle and scale.
6. Click Apply. The pattern is applied to the selected object.
7. Click the Close button (Windows) or the Close box (Macintosh) to close the dialog box.

Geometric Characteristics

According to the Geometry page of the Edit Objects dialog box, crosshatching has the following characteristics: Rotation and Scale. You can change the values and click ENTER (Windows) or RETURN (Macintosh) to alter the hatching.

Hatch Command

This command in the Pen menu crosshatches the selected objects using the default crosshatch pattern.

1. Select the object you want to crosshatch.
2. Choose ***Pen>Hatch.***

   The selected object crosshatches with the default pattern.

Editing Hatching

You can edit a crosshatch pattern for a selected object by changing the pattern, rotation or scale.

To change the hatch pattern select the hatching within the object and choose ***Pen>Cross Hatch***. Change the hatching and click Apply.

To change the hatch pattern rotation angle and scale you can do so in the Object Cross Hatch dialog box or in the Edit Objects dialog box.

Crosshatching and Section Cuts

Crosshatching is especially valuable for sections cuts created using the drawing composition tools.

When hatching is placed in a section cut the default hatch pattern is used. Like any other crosshatch pattern, you can change the pattern, rotation and scale. You can also choose different hatch patterns for objects cut in the same section. See Chapter 36, “Drawing Composition,” for information on sections and the drawing composition tools.

Symbols

In this Designer Elements program, symbols are used to define 2D or 3D shapes that frequently occur throughout a drawing. They can be wireframe, surface or solid objects. When used properly, symbols and instances dramatically speed the design and drafting process while reducing your file size.

An instance is a copy of the original or master geometry. It is created when you place a symbol in your drawing or modify that symbol by moving, rotating or scaling it.

A symbol consists of a master symbol composed of geometry (text and dimensions are not supported) which is instanced into a drawing by clicking the placement location. Once a symbol is placed you can use the Transformation tools to move, rotate, scale or mirror it.

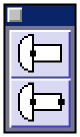
Symbols are created and organized through the Symbol Manager. Once the symbols are created use the Symbol tools to place them into your drawing.

The following topics are covered in this chapter:

- Symbol Palette
- Symbol Manager
- Creating Symbols
- Symbol Tools
- Symbol Example
- Transforming Symbols

- Editing Symbols
- Symbols and Rendering
- Symbols and Files

Symbol Palette

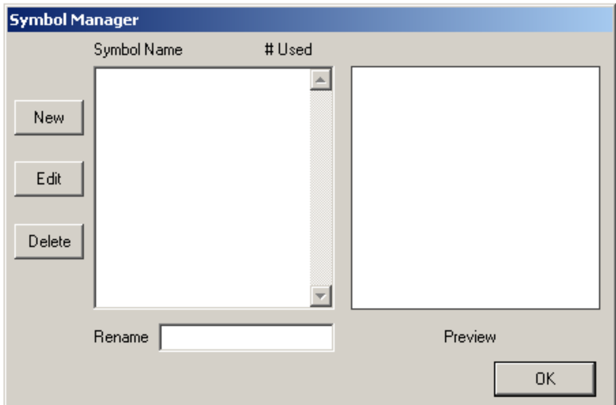


The symbol tools are contained in the Symbol tool palette. To display the palette choose ***Window>Symbols***.

The default status of the Symbol palette is closed and the default location is below the main tool palette. Like the other palettes, you can save the status and location of the palette anywhere in your drawing area.

There are two tools for placing symbols into your drawing, Symbol 1 Point and Symbol 2 Point. Before you are able to use these tools, you must create the symbol through the Symbol Manager. Once you create your symbol, use one of these tools to place it into your drawing.

Symbol Manager



Use the Symbol Manager to create and organize symbols in your drawing file. To display the Symbol Manager select one of the symbol tools and press the CTRL key (Windows) or the OPTION key (Macintosh).

The Symbol Manager contains the following elements:

Symbol Name	This list displays the names of all master symbols contained in the file whether or not they are instanced into the drawing.
# Used	Lists the number of times a master symbol is instanced into your drawing.

Preview Displays a 2D preview of the selected symbol in window.

Rename Allows you to rename any of the symbols in your drawing. When you initially create a symbol this Designer Elements program automatically provides a name.

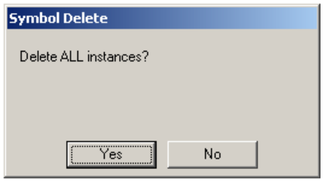
To change the symbol name, select the symbol name from the list and enter a new name in the *Rename* data field. The name updates as you type.

New Opens the Edit Symbol drawing screen for creating a new symbol.

Edit Opens the Edit Symbol drawing screen to edit a selected symbol in the *Symbol Name* list.

Delete Deletes the selected symbol and all instances (copies) of the symbol is your drawing.

If you have instances of the symbol in your drawing when you click this button, the following dialog box appears.



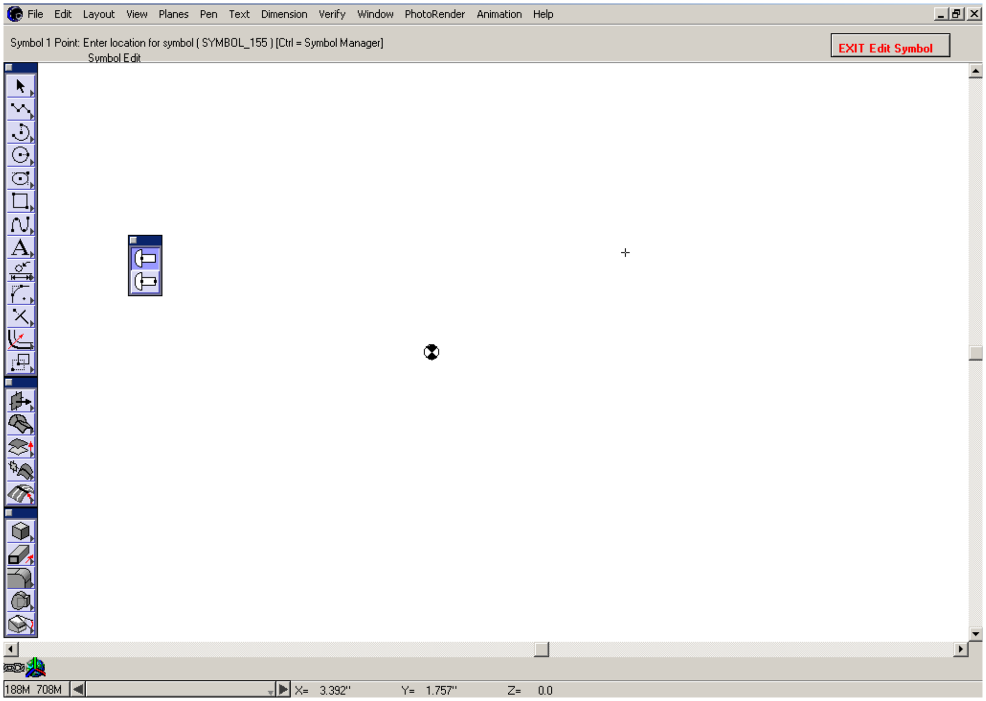
Click Yes to delete the symbol and all instances or No to close the box and return to the Symbol Manager.

Creating Symbols

Creating symbols is identical to creating any geometry in this Designer Elements program. Symbols cannot contain text or dimensions. When you create a symbol, you are creating a master. You place copies or instances of this symbol into your drawing using one of the symbol tools.

1. Select a symbol tool.
2. Press CTRL (Windows) or OPTION (Macintosh) to display the Symbol Manager.

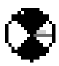
3. Click the New button. The Symbol Edit screen displays.



The screen is almost identical to the standard drawing area except for the EXIT Edit Symbol button and the Symbol Edit name in the Work Layer Indicator.

EXIT Edit Symbol Click this button in the upper right corner of the drawing area when you finish creating your symbol geometry to return to the drawing area.

Symbol Edit This name in the Work Layer Indicator indicates you are in the Symbol Edit screen. This layer is not accessible in the Layer Manager.



The Axis is located at the origin for a master symbol. The origin is represented by this marker.

4. Create your symbol geometry.

Use all of the tools and commands of this Designer Elements program.

5. When you complete your symbol geometry, click the EXIT Edit Symbol button to return to the standard drawing area.

Follow these steps to create more symbols for your file.

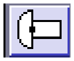
Like all other geometry, symbols are saved with your file. They do not become a part of a library for use with other files.

This Designer Elements program does not support text or dimensions with symbols.

Symbol Tools

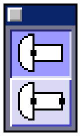
This Designer Elements program include two symbol tools for instancing symbols into your drawing, Symbol 1 Point and Symbol 2 Points.

Symbol 1 Point Tool



The Symbol 1 Point tool places a copy of the selected symbol at the location you specify. The symbol x-axis is aligned with the x-axis of the work plane.

Using the Symbol 1 Point Tool

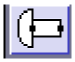


1. Select the Symbol 1 Point tool. The Message Line reads: *Symbol 1 Point: Enter location for symbol [Ctrl* (Windows) or *Option* (Macintosh) = *Symbol Manager]*.
2. Press the CTRL key (Windows) or OPTION key (Macintosh) to display the Symbol Manager.
3. Select the symbol you want to place in your drawing.
4. Click OK to close the Symbol Manager dialog box.

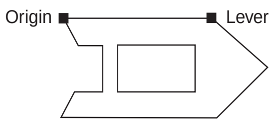
   The Message Line displays the name of the selected symbol.
5. Click in the drawing area to set the symbol's origin (as defined by the master).

   The symbol is placed in your drawing. You can place as many instances of the symbol as you desire.

Symbol 2 Points Tool

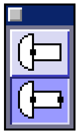


The Symbol 2 Points tool places a copy of the selected symbol at the origin and lever point you specify. The



lever point defines a rotation for the inserted angle and represents the top right corner of the symbol. See the graphic here.

Using the Symbol 2 Points Tool



1. Select the Symbol 2 Points tool. The Message Line reads: *Symbol 2 Points: Enter symbol origin and lever point [Ctrl* (Windows) or *Option* (Macintosh) = *Symbol Manager]*.
2. Press the CTRL key (Windows) or OPTION key (Macintosh) to display the Symbol Manager.
3. Select the symbol you want to place in your drawing.
4. Click OK to close the Symbol Manager dialog box.

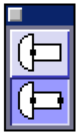
   The Message Line displays the name of the selected symbol.
5. Click the location for the symbol origin. The location is the symbol origin defined by the master symbol.
6. Click the lever point. The symbol is placed in your drawing.

   You can place as many copies of the symbol is your drawing as you desire.

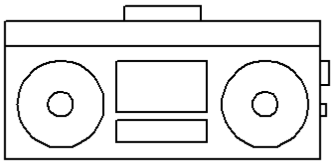
Symbol Example

Try this simple example of creating and placing a symbol.

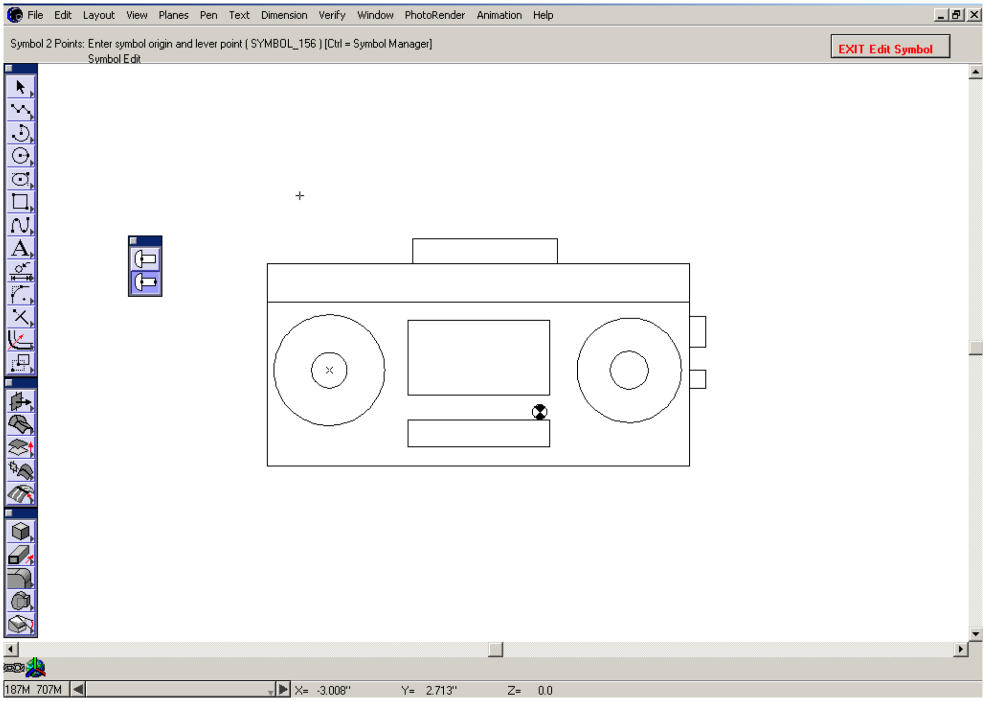
1. Open a new file.
2. Choose ***Window>Symbols*** to display the Symbol tools palette, if it is not already displayed.
3. Select the Symbol 2 Points tool.



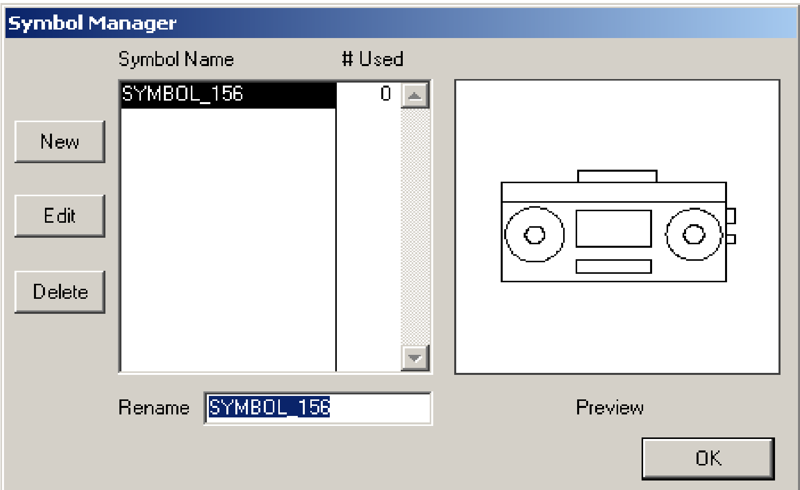
4. Press the CTRL key (Windows) or the OPTION key (Macintosh) to display the Symbol Manager.
5. Click on the New button. The Symbol Edit screen appears.
6. Create an object similar to the graphic here.



7. Once you've completed your symbol, select the Symbol 2 Point tool and then click the EXIT Edit Symbol button in the upper right corner.



8. Press the CTRL key (Windows) or the OPTION key (Macintosh) to display the Symbol Manager.



Your symbol geometry is displayed in the *Preview* window with a default name in the *Symbol Name* list.

9. In the *Rename* field, already highlighted, type *BoomBox*. As you type, the name is replaced in the *Symbol Name* list.
10. Click OK to accept the name and close the Symbol Manager.

The Message Line now includes the name, *BoomBox*. (If it does not click the Symbol 2 Points tool again.)

11. In your drawing click to place the origin point.
12. Move the cursor to the right horizontally using the Drafting Assistant and click to place the lever point.

The *BoomBox* symbol is placed in your drawing. If you had clicked a level point at a 45° angle, the BoomBox Symbol would have been placed at that angle.

13. Press the CTRL key (Windows) or the OPTION key (Macintosh) to display the Symbol Manager. The *# Used* field shows that the symbols was instanced once in your drawing.

Using the Transformation tools you can now perform numerous operations on the symbol.

Transforming a Symbol

Once you place a symbol you can use the Transformation tools to do some manipulation. You can move, rotate, scale or mirror the symbols. Follow the steps for using these tools in Chapter 25, “Transforming Geometry.”

You can also move a symbol with the arrow keys and the Selection tool. Select a symbol by dragging a selection fence around it.

Editing a Symbol

Symbols cannot be edited in your drawing area, only in the Symbol Edit screen.

1. Select a symbol tool.
2. Press the CTRL key (Windows) or OPTION key (Macintosh) to display the Symbol Manager.

3. Select the symbol you want to edit.
4. Click on the Edit button.

   The Symbol Edit screen displays.
5. Make your changes.
6. Click the EXIT Edit Symbol button.

   All symbol instances of the edited master symbol are updated.

Symbols and Rendering

This Designer Elements program does not support the rendering of symbols. If you want a symbol to render you must open the symbol through the Symbol Manager, copy the symbol and paste it into your drawing. Then the object, which is no longer a symbol, can be rendered.

Symbols and Files

Sometimes you may have symbols you want to use in more than one file or that were created elsewhere.

Using Symbols in Other Files

Symbols are file specific Symbols created in one file are not automatically available as a symbol for other files. However, if open the file in the Symbol Manager, you can copy and paste the geometry into a new Symbol Edit screen in another file. This way you won't have to recreate the geometry.

This Designer Elements program does not support Symbol libraries.

Importing

You can import drawing files into this Designer Elements program and then place that geometry within the Symbol Edit screen by copying and pasting. You can make additions to the symbol in the Symbol Edit screen.

This Designer Elements program does not automatically support importing symbols from Vellum 3D. Geometry from a symbol file can be imported into this Designer Elements program copied and then pasted into the Symbol Edit screen as mentioned above.

Viewing Geometry

In this Designer Elements program you create your geometry in one model space and view it from many different angles and view scales as needed. A view describes the orientation of your eye position and direction toward your geometry. You can use a number of tools and commands to help you to view your geometry. The following topics are covered:

- Zooming
- Panning
- View Displays
- View Rotation
- View Commands
- Views and Planes
- Views and Zoom Scale

Zooming

This Designer Elements program gives you several ways to change the magnification of your drawing by zooming in and out using commands, zoom tools, and strokes.

Zoom Commands

Zoom In, *Zoom Out*, *Zoom All*, *Zoom Previous*, *Zoom Window*, *Zoom Home* and *Zoom Ratio* from the View menu change the view magnification of your geometry depending upon your selection and input. If you want to zoom a particular area, use the *Stroke* feature or the Zoom tool, described later in this chapter.

Zoom All - CTRL+F (Windows); z +(Macintosh)

This command in the View menu zooms in or out to make all objects on your drawing fill the screen, regardless of the size of the objects.

Zoom In - CTRL+] (Windows); z +] (Macintosh)

This command in the View menu zooms in to the screen center by a factor of two.

Zoom Out - CTRL+[(Windows); z +[(Macintosh)

This command in the View menu zooms out from the screen center by a factor of two.

Zoom Previous

This command in the View menu zooms to the previous magnification.

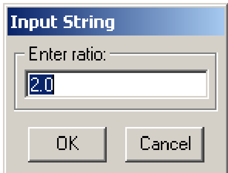
Zoom Window - ALT+4 (Windows); z +4 (Macintosh)

This command in the View menu allows you to drag a selection fence around the desired view window (from the upper left to the lower right).

Zoom Home - ALT+6 (Windows); z +6(Macintosh)

This command in the View menu adjusts the view scale so that the origin of the drawing (0, 0, 0) is centered on the screen.

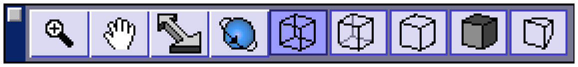
Zoom Ratio Command



This command in the View menu displays an Input String dialog box.

Enter a ratio in the data field. A value of .5 zooms out by a factor of two. A value of 2 zooms in by a factor of two. Click OK to close the dialog box and save the value. Your drawing scales to the value.

Zoom Tools



The Zoom tools are located in the View tool palette.

Using the Magnifying Glass Tool

With the Zoom tools you drag a selection fence around an area on the screen, so only that area is displayed.

1. Select the Zoom tool from the tool palette.
2. Drag a box around the area on the screen you want to magnify or reduce.
3. Release the mouse button. The content of the dragged box is displayed.

Zoom Tool

This tool zooms in by the specified factor. The default factor is two. You can also specify a zoom scale (1 = full scale) in the Scale field of the Status Line. Zooming causes a visual rather than a physical change.

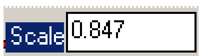
Tip:

Using the CTRL (Windows) or the OPTION (Macintosh) key, you can toggle between the **Zoom In** and **Zoom Out** tool.

1. Choose the Zoom In tool. The Message Line reads: *Zoom In: Pick area to enlarge [Ctrl* (Windows) or *Option* (Macintosh) = *Zoom Out]*.

   If you want a specified zoom scale, enter the value in the Scale data field.
2. Click in the drawing area; that position is displayed in the center of the screen and the drawing is enlarged by a factor of two. You can also drag a selection fence around an area, so only that area is displayed.

The Status Line contains the Scale data field.

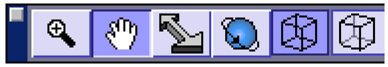


Pressing the CTRL (Windows) or the OPTION (Macintosh) key while using this tool causes it to change to the Zoom Out tool.

Using the Dynamic Pan Tool

This tool is located in the **View** tool palette at the bottom of the main tool palette.

1. Select the Dynamic Pan tool. The Message Line reads: *Dynamic Pan: Drag mouse to translate view.*

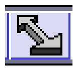


The pointer becomes a hand icon.

2. Place your pointer over the section of the screen you want to move and drag.

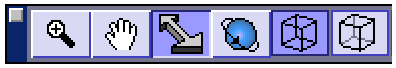
When you release the mouse, the view has been repositioned. Notice that the scroll bars have adjusted accordingly.

Dynamic Zoom Tool

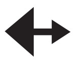


This tool zooms in or out by the amount specified by your stroke in the drawing area.

1. Choose the Dynamic Zoom tool. The Message Line reads: *Dynamic Zoom: Drag mouse to zoom view. Move right zooms out.*



2. Move the cursor to the drawing area. The cursor becomes the dynamic zoom icon.



3. Drag the mouse to the right to zoom out or to the left to zoom in. The view scale changes according to your stroke.

Stroke Zoom

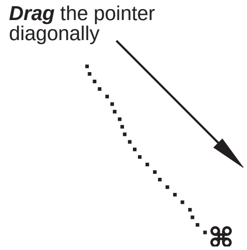
Tech Note:

Windows users: The Stroke Zoom function is not accessible for tools that already use the SHIFT and CTRL keys. This note is repeated in the chapters containing the tools to which this applies.

You can use stroke commands to zoom—magnify or reduce your drawing. Stroke commands are useful because they don't require you to get out of the tool you're using in order to zoom. Hold down the SHIFT+CTRL keys (Windows) or the z key (Macintosh) and drag diagonally across the screen as described below. The pointer takes on the z shape when you hold down the SHIFT+CTRL (Windows) or the z key (Macintosh) keys.

This Designer Elements program remembers up to eight zoom strokes, enabling you to return to previous zoom magnifications.

Using Stroke Zoom



***Drag* Diagonally**	**Result**
Upper left to lower right	Zoom-in enlargement: centered over the stroked area.
Lower right to upper left	Zoom Previous: Reverses *Zoom In* stroke to the previous magnification.
Upper right to lower left	Zoom-out reduction: the current screen reduces to the size of the area defined by the stroke.
Lower left to upper right	Zoom Previous: Reverses *Zoom Out* stroke to previous magnification.

Note: For *Zoom In* and *Zoom Out*, the size and location of the stroke rectangle is important for determining the result of the Zoom operation. For *Zoom Previous*, the size and location of the stroke rectangle is irrelevant. All cases just give the previous magnification.

Zooming and Views

This Designer Elements program retains the last zoom scale that you set in a particular view. When you zoom in a particular view, change views and zoom in the new view, the view scale in the previous view is not affected.

Panning a View

You may want to move your view around to better see your geometry. This Designer Elements program provides the Dynamic Pan tool that enables you to do this without using the scroll bars. Simply hold down the space bar, click on your geometry and drag to the desired location.

View Displays

You have two view types in this Designer Elements program: pre-defined and user-defined. Use these to set the view orientation of your geometry.

Pre-defined Views

This Designer Elements program provides five standard views: Side, Front, Top, Isometric and Trimetric. Each view is defined by Eye Pt locations on the X, Y and Z axes or the Azimuth and Elevation. Changing the values in either Eye Pt or Azimuth/Elevation automatically changes the other fields.

You can change these values as desired. In ***File>Preferences>General***, you can set the view definitions to Default or Aerospace. Included with each view is a description based on the view definition and its associated values.

Tech Note:

An azimuth is an angle measured clockwise from the selected point to the vertical.

Side

Default: The view of the y, z plane. The Eye Pt values are: X = 500, Y = 0.0, Z = 0.0. The Azimuth value is 0.0. The Elevation is 90.0.

Aerospace: The view of the x, z plane. The Eye Pt values are: X = 0.0, Y = -500.0, Z = 0.0. The Azimuth value is 0.0. The Elevation is 90.0.

Front

Default: The view of the x, z plane. The Eye Pt values are: X = 0.0, Y = -500.0, Z = 0.0. The Azimuth value is 0.0. The Elevation is 90.0.

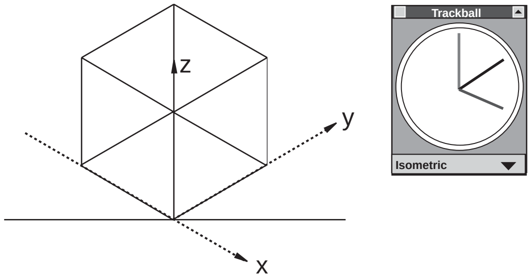
Aerospace: The view of the y, z plane. The Eye Pt values are: X = -500.0, Y = 0.0, Z = 0.0. The Azimuth value is 0.0. The Elevation is 90.0.

Top

Default &Aerospace: The view of the x, y plane. The Eye Pt values are: X = 0.0, Y = 0.0, Z = 500.0. The Azimuth value is 0.0. The Elevation is 0.0.

Isometric

The view of the axes is rotated as shown.

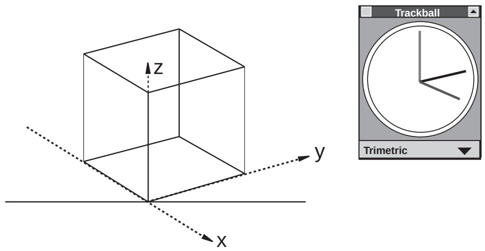


Default: The Eye Pt values are: X = 500.0, Y = -500.0, Z = 500.0. The Azimuth value is -45.0. The Elevation is 54.736.

Aerospace: The Eye Pt values are: X = -500.0, Y = -500.0, Z = 500.0. The Azimuth value is -45.0. The Elevation is 54.736.

Trimetric

The view of the axes rotated as shown.



Default: The Eye Pt values are: X = 382.176, Y = -256.20, Z = 195.712. The Azimuth value is -33.837. The Elevation is 66.957.

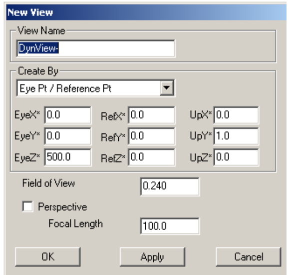
Aerospace: The Eye Pt values are: X = -382.176, Y = -256.20, Z = 195.712. The Azimuth value is 33.837. The Elevation is 66.957.

User-defined Views

You have the ability to define any new view using the *New View* command or the Trackball. You also have the ability to modify these views using the *Modify View* command.

User-defined Views are saved with the current file. They are not saved as defaults for the program.

New View Command



The *New View* command, located in the View menu, allows you to specify a new view based on a current view or entirely independent of any of the available views.

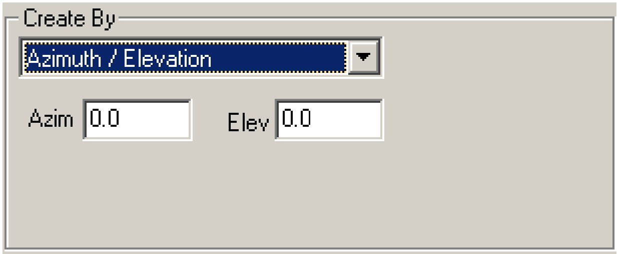
Choosing the *New View* command brings up the dialog box and contains the following options:

View Name Contains the name of the current view.

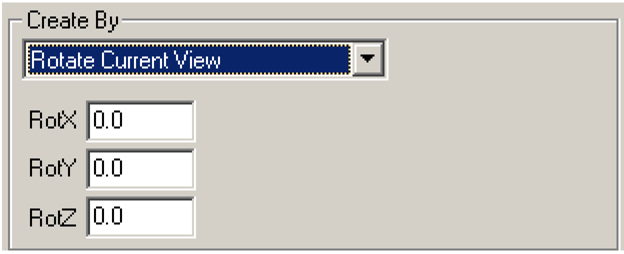
Create By This field includes a pull-down menu allowing you to choose how to specify your new view; *Eye Pt/Reference Pt*, *Azimuth/ Elevation* and *Rotate Current View*.

Eye Pt/Reference Pt - Allows you to create a view based two sets of values. The *Eye Pt* refers to the location of the viewer's eye. The *Reference Pt* refers to an existing point on a model. An asterisk next to the fields denotes your ability to click the location in the drawing area and have the values automatically entered in the fields.

Azimuth/Elevation - Allows you to specify angle of the azimuth and the elevation of your eye with respect to the view.



Rotate Current View - Allows you to rotate the current view by a specified amount. The values entered in these fields affect the related fields in the other *Create By* options.



OK Click this button to save the new view and close the dialog box.

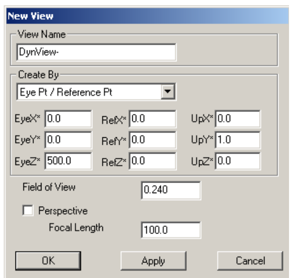
Cancel Click this button to close the dialog box without saving the view.

Specifying a View with the New View Command

1. Choose ***View>New View***.

   The New View dialog box displays the name and location of the current view with its corresponding values. The View Name is highlighted.

2. Enter the new name for the view.
3. Select one of the *Create By* methods for defining your view.



4. Enter the appropriate values according to the *Create By* option you choose.

   For the *Eye Pt/Reference Pt* method, you can also click the appropriate points in your drawing and the values will be entered automatically into the data fields.

5. Click OK. The dialog box closes and the new view is defined. (Click Cancel to close the dialog box without saving the view.)

Once you create a new view, you can select it by choosing ***View>User View*** and the view name or choosing the view in the Trackball pull-down menu.

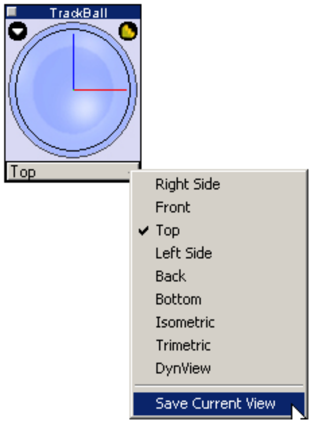
Tip:

If you change a view of your geometry but make no changes to your geometry, the *Save* command is unavailable. To save the view, create an object and then delete it. The *Save* command is now available and can be used to save your view with the file.

Be aware that simply rotating the view does not alter the orientation of the work plane in 3D space (except for the Side, Front and Top views).

Trackball - Save Current View

You can also define a view using the Trackball command, *Save Current View*, at the bottom of the Trackball pull-down menu.

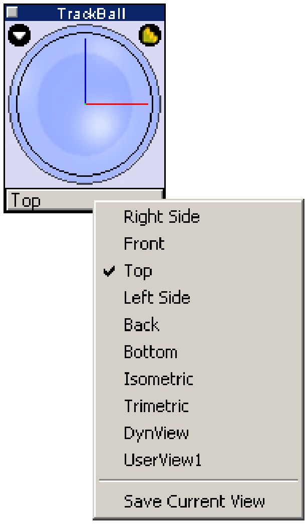


1. Display the Trackball.
2. Rotate your view as desired.
3. Click the view name on the Trackball to display the pull-down menu.

4. Select *Save Current View* and release the mouse. The new view saves.

   When you display the pull-down menu again, your new view is listed as *User View1*.

   You will also see the addition of a *DynView*, which is the current non-standard view.



All views defined in this way are numbered sequentially. You can rename these views by choosing ***View>Modify View***.

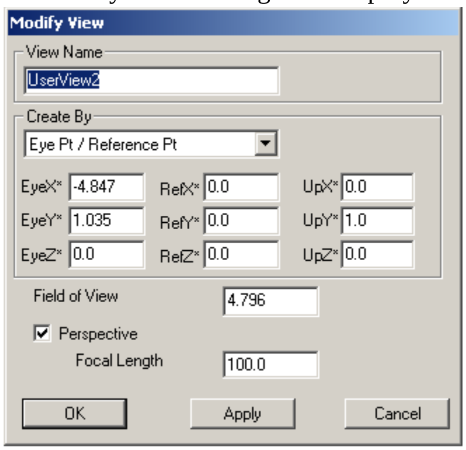
Modifying a View

You can only modify User-defined views. You cannot modify a standard view. An attempt to do so creates a duplicate of the view except for the changes you made. This new view becomes a User-defined view which you can rename as desired. If you choose ***View>User View***, the modified view will be displayed.

To change the name or any coordinate locations for user-defined views, use the *View Properties* command in the View menu

Using the Modify View Command

1. Choose ***View>View Properties*** and select the view you want to change. The Modify View dialog box displays.

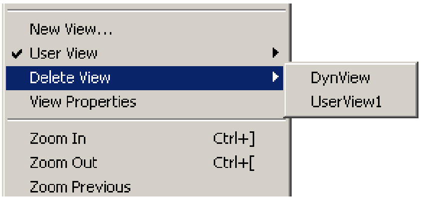


   This dialog box is identical to the New View dialog box except for its title.

2. Make all of the desired changes in the appropriate fields. (See the New View section earlier in this chapter for an explanation of the data fields.)
3. Click OK. The dialog box closes and the new view is defined. (Click Cancel to close the dialog box without saving the view.)

Deleting a View

You can delete any DynViews or user-defined views using the *Delete* command in the View menu. Choose ***View>Delete View*** and the view you want to remove.



You cannot delete the current view or the standard views. This command is unavailable if there are no user-defined views.

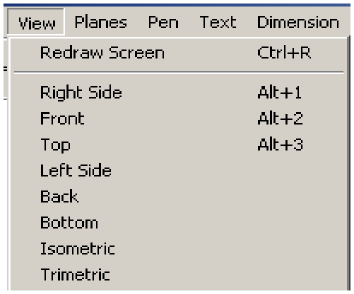
View Rotation

You can choose views in a number of ways:

- Choosing a standard view from the View menu or the Trackball
- Choosing a user-defined view from the View menu or the Trackball
- Choosing an undefined view by rotating the Trackball
- Using the Dynamic Rotation tool
- Using the SHIFT and Arrow keys

Choosing a Predefined View

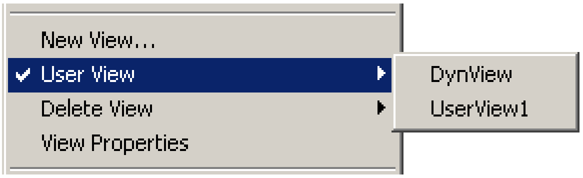
To use any of the five standard views mentioned earlier choose one in the View menu or in the Trackball pull-down menu.



See Chapter 7 for more information on using the Trackball.

Choosing a User-defined View

To use any of the user-defined views, choose one in the *User View* submenu of the View menu or in the Trackball pull-down menu.



The *User View* command in the View menu is not available when no user views have been defined.

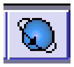
Choosing an Undefined View

You can choose undefined views by using the Trackball to rotate the view to a new orientation.

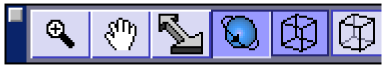
Notice that when you rotate the view with this method the view name changes to DynView. This view has been added to your Trackball pull-down menu and is a temporary view that will change as your rotate your view using the Trackball. One advantage of this feature is that you don't have to define this view but it will still be available to you until the next Trackball rotation.

Dynamic Rotate Tool

The Dynamic Rotate tool, located in the View tool palette, allows you to rotate your view dynamically around any axis.



1. Select the Dynamic Rotate tool. The Message Line reads: *Dynamic Rotate: Drag mouse to rotate view*.



   The cursor becomes a plus (+) sign.

2. Drag your cursor to change the view.

   or

   Enter values in the Status Line if you want the view to rotate a specified angle.

   Press ENTER (Windows) or RETURN (Macintosh) and the view changes.

Using the Shift and Arrow keys

In any view you can hold down the SHIFT key and use the arrow keys to rotate your view.

View Commands

Redraw Screen - CTRL+R (Windows); z +R (Macintosh)

This command in the View menu refreshes the screen. When you make changes to your constructions the geometry may not be redrawn cleanly.

Redrawing the Screen

To redraw all of the geometry and remove extraneous geometry choose the *Redraw Screen* command from the View menu.

Stopping a Screen Refresh

Windows: You can press the ESC or BREAK key to stop the redrawing of the screen. For interrupting long operations such as redraw or linear and polar duplicate, use the BREAK key. If the operation was initiated by a Control key command (such as

CTRL+R for redraw) the ESC key is read by MS-Windows and it brings up a task list at the end of the operation.

Macintosh: You can press ESC or the z (Command) key to stop the redrawing of the screen.

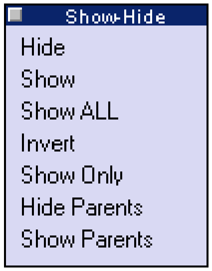
View the Plane

This command located in the View menu, changes the view to your current work plane.

Show/Hide

This command in the Window menu helps you manage the display of objects in your drawing.

You can choose which objects display at a given time. When you choose ***Windows>Show/Hide***, the following list of commands appears.



Using the Hide Command

1. Select the *Hide* command. The Message Line reads: *Select entities to hide [Shift = Extend]*.
2. Select an object. The object hides from view.

Using the Show Command

1. Select the *Show* command. The Message Line reads: *Select entities to show [Shift = Extend]*.
2. Select an object by clicking on the approximate location of the object or using a selection fence. The object shows on the screen.

Using the Show ALL Command

Select the *Show ALL* command to display all objects on active layers in your drawing.

Using the Invert Command

Select on the *Invert* command to display all object currently hidden and hide all objects currently displayed.

Using the Show Only Command

1. Select the *Show Only* command. The Message Line reads: *Select entities to show only [Shift = Extend]*.
2. Select the object that you want displayed. Hold down the SHIFT key to select more than one object.

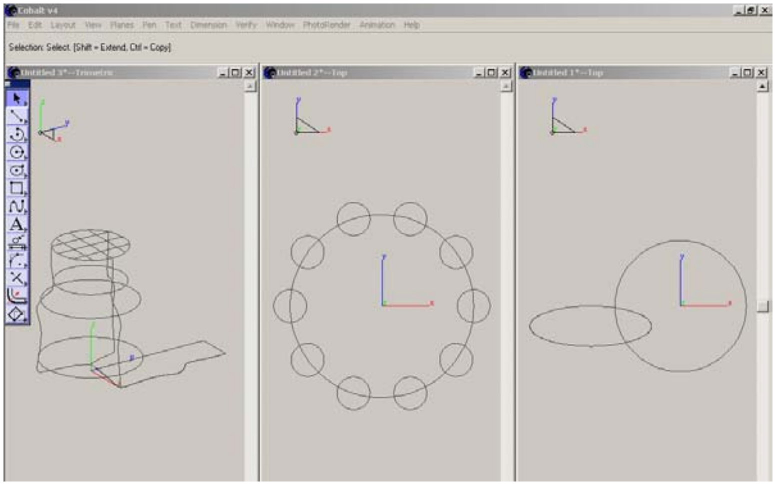
   All other objects in your drawing are hidden.

Tile

In some cases you may find you have multiple files open at the same time. Rather than having to switch back and forth between them, you can choose to display them all at once. In the Windows menu you have two commands to accomplish this: *Tile Vertically* and *Tile Horizontally*. (Windows Only)

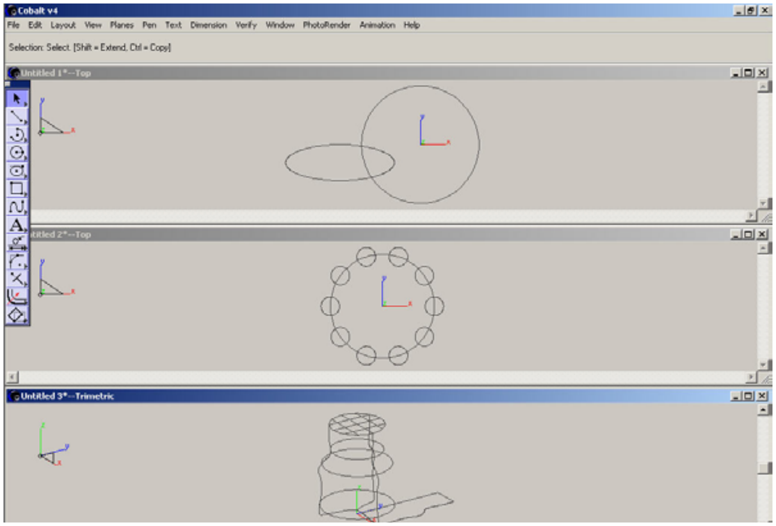
Tile Vertically

This command resizes the window for each file equally and arranges them vertically across your screen. The files are arranged left to right starting with the file most recently active. The view scale is changed to display all geometry in each file.



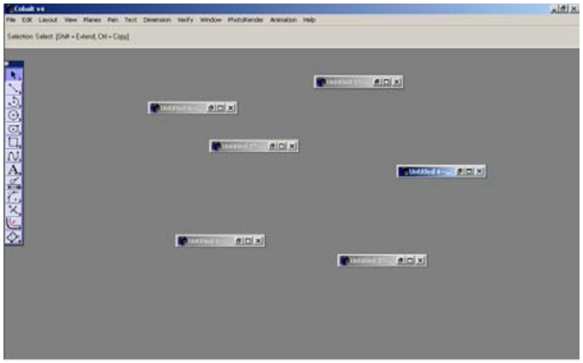
Tile Horizontally

This command resizes the window for each file equally and arranges them horizontally down your screen. The files are arranged top to bottom starting with the file most recently active. The view scale is changed to display all geometry in each file.



Clicking on the Maximize button at the top right of your screen, resizes the selected file to full screen.

Arrange Icons (Windows only)



On occasion you may have many open files reduced to their title bars but arranged haphazardly around your screen.

Choose ***Window>Arrange Icons*** and your files will be neatly arranged in the lower left corner of your screen.

Open File Windows

The bottom of the Window menu shows the names of all the open this Designer Elements program files. To bring a different document to the top choose it from the list.

Views and Planes

When you want to view geometry in this Designer Elements program, you choose a particular view orientation as discussed in this chapter. When you want to create geometry you choose a particular plane on which to draw.

To eliminate the potential confusion between work planes and view, this Designer Elements program has tied the Front, Side and Top views to their respective planes. For example, if you choose the Top view, the Top plane is selected in the Plane menu and all other planes are grayed out. See Chapter 32 for more information about Planes.

Views and Zoom Scale

this Designer Elements program has linked the view to the zoom scale. Each view remembers the last zoom scale set in that view. You can change the view and alter the zoom scale and it won't affect the zoom scale of any other view.

Example: In the Top view set the zoom scale to 1:2. Choose the Isometric view and change the scale to 1:4. Return to the Top view and the scale returns to 1:2.

Layers

You can think of visible layers as transparent pages and hidden layers as invisible pages. You can use layers to show and hide various components of your drawing. They are particularly useful in helping you view and print complex drawings. For example, when you dimension a part, the dimensions can be placed on a separate layer which you can display or not, as your needs require. Layers allow you to print different versions of the same document.

- You can hide the dimension layer to present a design to a planning team and show the dimensions when presenting the drawing to engineers.
- You can hide some drawing components when you print or plot. For example, you can hide the construction layer so that construction lines and geometry don't print but remain in the drawing ready for use when you want to alter them.
- You can construct different layouts using one layer as the basis. For example, you can use one layer to define the basic shape of an object and then use other layers to try different methods of detailing.

In this Designer Elements program you can have up to *65,000* layers in your drawing. Layers must be visible to select objects on them. Layers do not have an orientation or origin in this Designer Elements program.

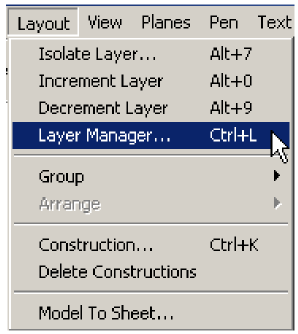
The topics explained in this chapter include:

- Layer Manager
- Creating New Layers
- Creating New Sub Layers

- Renaming Layers
- Deleting Layers
- Hiding Layers
- Displaying Layers
- Locking Layers
- Unlocking Layers
- Making a Layer the Active Work Layer
- Layers and Colors
- Layer and Copying/Pasting Objects

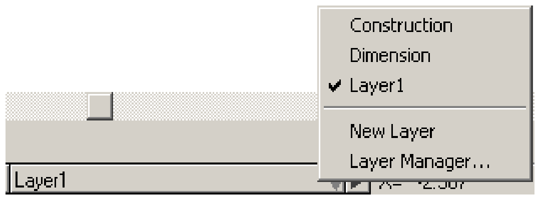
Layer Manager-CTRL+L(Windows);z +L(Macintosh)

The Layer Manager allows you to create, delete, hide, display and rename layers, as well as set layer specifications. The work layer is the active layer—on which geometry is created. There are Three ways to open the Layer Manager.

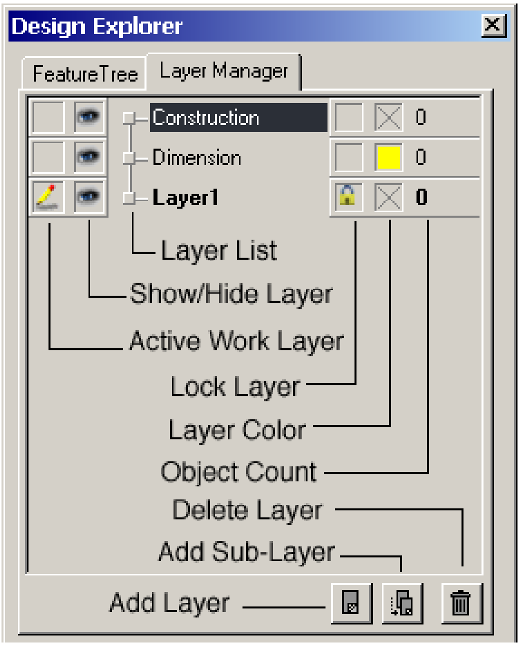


The Layer Manager dialog box is integrated with the Design Explorer dialog box.

1. Open the Design Explorer (***Window>Design Explorer***)
2. Choose ***Layout>Layer Manager***:
3. Click on the *Work Layer Indicator* to display the pop-up menu and choose the *Layer Manager* command



The Layer Manager dialog box appears.



The Layer Manager contains the following elements

Active Work Layer

This column sets your active work layer. To change your active work layer move the pencil icon to the layer you wish to add geometry to.

Layer list

The Layer list displays all layers in the file.

Layer-Show/Hide

This column shows whether a layer is visible or hidden. You cannot hide the active work layer.

If the Layer icon (see below) displays, a layer is visible (on).

Layer Lock

If this column contains a lock icon (see below) next to a layer, the layer is locked.

Objects on this layer cannot be selected or modified.

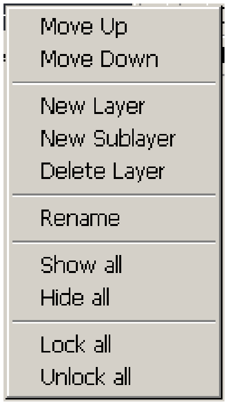
Layer Color

This column sets the color for all objects on a layer.
The original colors of objects are retained but they are displayed in the layer color as it takes precedent.

Object Count

This column displays the number of objects on the layer.

By right clicking on a layer the follwoing menu box pops up:



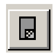
Move Up

Choosing this option moves the selected layer up one position in the dialog box.

Move Down

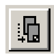
Choosing this option moves the selected layer down one position in the dialog box.

New Layer



Choosing this option adds a new layer to the layer list. Layers are numbered sequentially, Layer 1, Layer 2, etc. This option also has a button at the bottom of the Layer Manager dialog box.

New Sublayer



Choosing this option adds a sublayer to the selected layer. These layers are controled by their parent layer. If you show/hide the parent the sublayer responds in kind. This option also has a button at the bottom of the Layer Manager dialog box.

Delete Layer

Choosing this option deletes the selected layer. You cannot delete the active work layer.This option also has a button at the bottom of the Layer Manager dialog box.

Rename

Choosing this option lets you rename the highlighted layer. You may also click twice on a layer name, once to select the layer and the second time to rename it.

Show All

Choosing this option shows all layers.

Hide All

Choosing this option hides all layers except the active work layer.

Lock All

Choosing this option locks all layers including the active work layer.

Unlock All

Choosing this option unlocks all layers.

Default Layers

When you open a new drawing the default layers include: *Construction*, *Dimension* and *Layer 1*.

Construction

This layer automatically accepts all Construction lines created with stroke commands or with the Construction dialog box. If you accidentally delete this layer, it automatically recreates when you create a construction line.

Dimension

Dimensions are normally placed on the layer you select from the Layer list of the Dimension menu. The default layer is the Dimension layer. Generally, this layer should be reserved for dimensions.

Tech Note:

Dimensions placed in drawing views go on the Sheet View layer. See Chapter 36 for more information.

Layer 1

This layer is the current work layer for new files. If the file only contains the default layers all geometry will normally be placed on Layer 1.

You can edit any geometry or text that is visible regardless of its layer. If you want to make some geometry unselectable but still visible use the *Select Mask* in the Window menu or lock the layer containing the geometry. You can also specify the selectability of layers with the *Select Mask*.

Creating New Layers

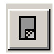
There are three ways you can create a new layer: using the Layer Manager, the Work Layer Indicator or the *Isolate Layer* command.

Using the Layer Manager

Tip:

If the Selection Mask is open and you add a new layer, the Selection Mask automatically updates.

1. Display the Layer Manager dialog box.
2. Click the new layer button. A new layer is added to the layer list.

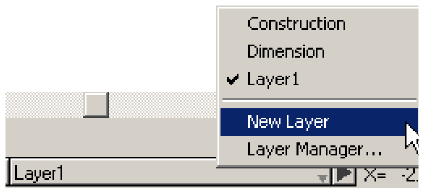


   You may name the layer by clicking twice on the layer name, once to select the layer and the second time to rename it. You can have as many as 32,700 layers.
3. The new layer is added.

Using the Work Layer Indicator

1. Click on the Work Layer Indicator to display the pop-up menu.
2. Choose the *New Layer* command.

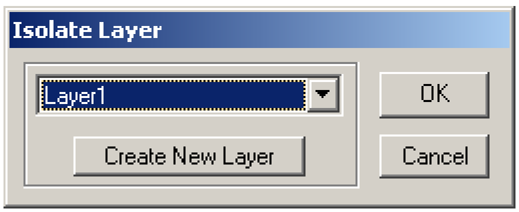
   A new layer is created and automatically set as the active work layer.



Using the Isolate Layer Command - Alt+7 (Windows); z +7(Macintosh)

1. Choose ***Layout>Isolate Layer***.

   The Isolate Layer dialog box appears.



2. Click Create New Layer.

   A new layer is created and the name is placed in the entry field.

3. Click OK to close the dialog box and save the new layer. The layer is now the active work layer and all other layers are hidden.

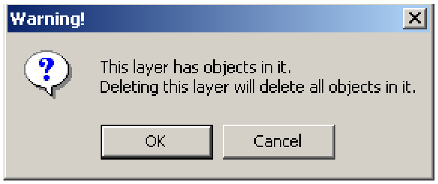
Renaming Layers

1. Display the Layer Manager dialog box.
2. Click twice on the layer you wish to rename, once to select the layer and the second time to rename it.

Deleting Layers

1.Display the Layer Manager dialog box.

2.Select the name of the layer from the list box.



3. Click on the trash can in the lower right corner of the dialog box. The layer is deleted.
4. If the layer contains any geometry on it a warning appears.

Click OK to delete the layer or Cancel to close the warning box without deleting.

Hiding Layers

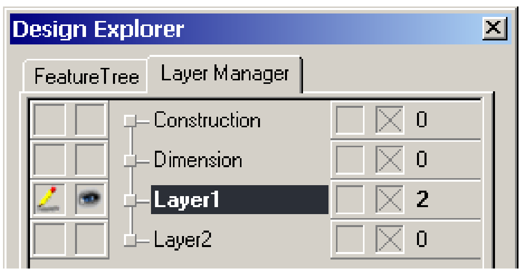
You can hide one layer at a time or all layers but the active work layer.

Hiding One Layer

1. Display the Layer Manager dialog box.

2. Click on the “eye” icon to the right of the layer name that you want to turn hide.

The eye icon disappears and the layer is now hidden.



Hiding all Layers

1. Display the Layer Manager dialog box.
2. Right click on a layer name and choose Hide All.
3. Cloaw the Layer Manager dialog box.

Notes:

- You can't hide the active work layer
- If a layer is hidden and you choose *Select All* and delete, the objects on the hidden layer are not deleted.

Showing Layers

You can turn on one or all layers in your drawing.

Showing One Layer

1. Display the Layer Manager dialog box.
2. Click the box to the left of the layer name that you want to turn show.

   A eye icon appears next to the layer’s name in the list.

Showing all Layers

1. Display the Layer Manager dialog box.
2. Right click on a layer name and choose Show All.

Locking Layers

1. Display the Layer Manager dialog box.
2. Click in the *Lock Layer* column next to the desired layer and the lock icon appears.

   The layer locks.

Unlocking Layers

1. Display the Layer Manager dialog box.

2. Click on the lock icon in *Lock Layer* column next of the desired layer. The lock icon disappears and the layer unlocks.

Making a Layer the Active Work Layer

You have number of ways to make a layer the active work layer. These include: using the Layer Manager, the Work Layer Indicator, the *Isolate Layer* command, the *Increment Layer* command or the *Decrement Layer* command.

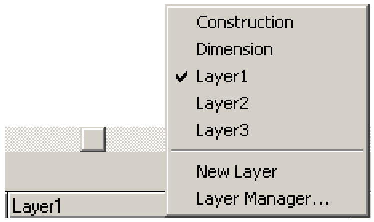
Using the Layer Manager

1. Display the Layer Manager dialog box if it is not already displayed.
2. Click in the *Work Layer* column to the left of the desired layer.

   The selected layer becomes the work layer as shown by the pencil icon.

3. Close the dialog box and save the change.

Using the Work Layer Indicator



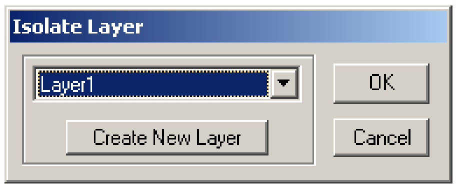
1. Click on the Work Layer Indicator to display the menu.
2. Select the layer you want to make the active work layer

   The layer you selected is now the active work layer.

Using the Isolate Layer Command - Alt+7 (Windows)z +7 (Macintosh)

1. Choose ***Layout>Isolate Layer***.

   The Isolate Layer dialog box appears.



2. Click on the arrow in the Layer entry field to display all available layers.
3. Select the desired layer.
4. Click OK to save this layer as the work layer and close the dialog box. All other layers are hidden.

Increment Layer - ALT+0 (Windows); z +0 (Macintosh)

Choosing this command in the Layout menu makes the next layer in the Layer Manager dialog box the active work layer and hides all other layers. You cannot use this command if your active work layer is the last layer in the list.

Decrement Layer - ALT+9 (Windows); z +9 (Macintosh

Choosing this command in the Layout menu makes the previous layer in the Layer Manager dialog box the active work layer and hides all other layers. You cannot use this command if your current work layer is the first layer in the list.

Layers and Color

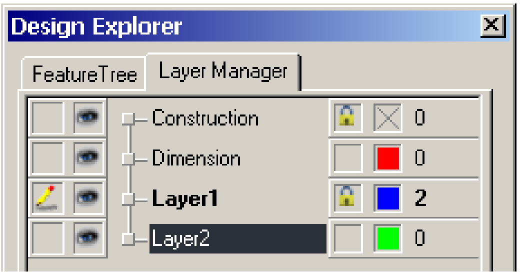
This Designer Elements program provides the ability to choose a color for all objects on a layer. You have nine color options: white, red, blue, green, cyan, gray1, purple, yellow and gray2.

Objects created on this layer use the current pen color but are displayed in the layer color.

Objects created on another layer and in another color placed on this layer have their color display overridden. They retain their original color but display in the layer color. If the object is moved to another layer with no color override, it displays in its own color.

Using the Layer Color Override

1. Display the Layer Manager dialog box if it is not already displayed.
2. Click in the *Layer Color* column next to the desired layer. A color patch appears displaying the first color available, white.



3. Click on the color patch to advance to the next patch color.
4. Select the desired color for the layer.

5. Click OK to close the dialog box and save the change.

Layer Color Warning

This Designer Elements program provides you with a Layer Color warning if you attempt to change the color of an object on a layer with a specified color. The following Change Color box appears:

As the box indicates, you cannot change the color unless you remove the color override for that layer.

Layers and Copying/Pasting Objects

When you copy objects on a specific layer in one file into another file containing those same layers, the objects are not automatically placed on the specific layer. The copied objects are placed on the work layer.

Planes

A plane is an infinite surface on which you can create geometry. This Designer Elements program provides predefined planes, Front, Side and Top and the ability to create user-defined planes.

If a plane is chosen as the *work plane*, all geometry created from that point on is placed on that plane. The work plane is an x, y plane with an origin of 0, 0, 0 for all data input. You can move the work plane as desired by creating your own or choosing one of the predefined planes.

Referral:

Planes can also be created by using the **Infinite Plane** tool in the **Surfaces** tool palette. See Chapter 16 for more information.

As you learn to use this Designer Elements program, you should attempt to use the Z-Drafting Assistant without moving the work plane. Even though the Z-Drafting Assistant does a lot of the work for you the work plane is still an essential element of 3D modeling.

The Z-Drafting Assistant assumes that geometry is being created in the current work plane unless it snaps to an *align:z* or to a logical snap point that is not in the current work plane.

You will need to use a work plane to properly position geometry in the following cases:

- To create geometry that is not parallel to the work plane and does not snap to one of the Drafting Assistant's constraints like endpoint or midpoint.
- To create geometry that requires less than three points for their specification such as the Center-Point Circle, Rectangle or 2-Point Ellipse.

Those tools which use three points for specification can be drawn non-parallel to the work plane.

The Plane menu and the Work Plane Manager, accessed through the coordinate system axis, contain all of the commands for creating and setting the work plane.

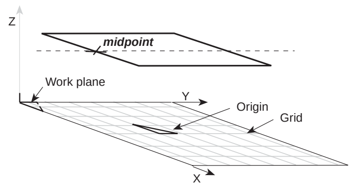
The topics explained in this chapter include:

- Work Plane
- Work Plane and Views
- Work Plane and World Coordinates

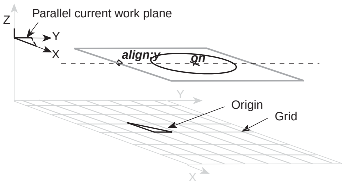
Work Plane

The work plane is an important feature of any CAD program. In conventional CAD programs you have to specify both the orientation of the work plane and its exact location along the z-axis.

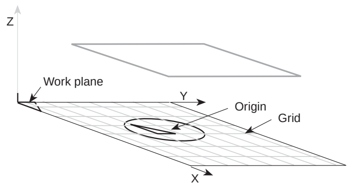
In this Designer Elements program you need only specify the orientation of the work plane. Once that is done, all parallel planes act equally as the current work plane (e.g. the Drafting Assistant identifies the location of the work plane automatically).



When you identify one point of an object with the Drafting Assistant...



...the new object geometry is created on the plane of the identified object.



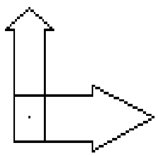
If you do not identify an object the new object geometry is placed onto the work plane at the origin.

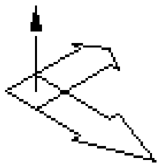
Another way to explain the relationship presented in the graphics above is to remember the role of the Drafting Assistant. For new geometry to be placed on the same plane as another piece of geometry either create a user-defined work plane at that location or brush over that “old” geometry to wake up one of its control points like midpoint or endpoint. This Designer Elements program will then retain this plane information as you create the new geometry.

If no object exists or no geometry is referenced for plane information this Designer Elements program places the new object geometry onto the work plane.

Show Work Plane Command

Choosing this command in the Plane menu displays the work plane icon in your drawing. The left graphic below shows the work plane icon in the Top plane. The right graphic shows the icon in the Isometric view.





The work plane icon is a helpful reference for creating geometry on multiple planes.

Work Plane Manager

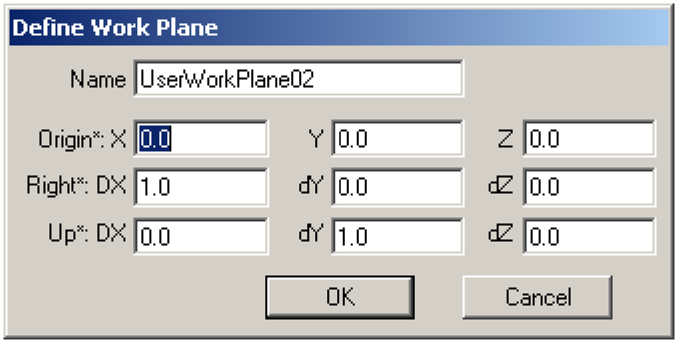
The work plane manager, accessed through the coordinate system icon in the Status Line, is a menu of commands that allow you to define, delete, display the properties of and save the current work plane. It also allows you to show or hide the work plane.

Display this menu by clicking on the coordinate system axis icon to the left of the Status Line.

The work plane manager menu displays.

New Command

This command, found in the Planes Menu (***Planes>New Work Plane***) allows you to define a new work plane. You can define an unlimited number of planes. When you choose this command the Define Work Plane dialog box appears.



The dialog box contains the following items.

Name Displays the work plane name. If you have not entered a unique name, the name field displays UserWorkPlane01 or some other numbered increment.

Origin These fields display the X, Y and Z coordinates for the origin of the work plane.

Right These fields display the DX, DY and DZ coordinates for the right arm of the work plane.

Up These fields display the DX, DY and DZ coordinates for the up arm of the work plane.

The asterisk (*) next to the field indicates your ability to specify locations by clicking in the drawing area. You can also enter the values manually.

Using the New Command to Define a Work Plane

1. From the work plane manager menu, choose *New*.

   The Define Work Plane dialog box appears.

2. Enter the values for the origin, right and up arms of the work plane. You can also specify the locations by clicking in the drawing area.
3. Enter in name for the work plane in the Name data field.

If you don't enter a name, the default name displayed in the field will save with the work plane coordinates.

4. Click OK to save the new plane.
5. Display the work plane manager menu again.

Referral:

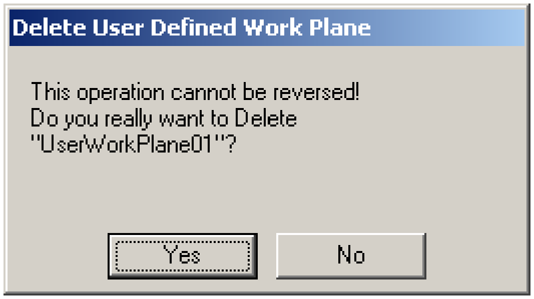
See Chapter 7 for information on coordinates systems.

Notice that the new work plane is listed and is the current work plane. You are now operating in the user-defined coordinate system.

Delete Command

Use this command, located in the Planes Menu (***Planes>Delete Work Plane***) to delete a user-defined work plane. You cannot delete the GlobalWorkPlane or DynWorkPlane items.

1. Select the user-defined work plane you want to delete.
2. Choose the *Delete* command. A warning box appears explaining that the operation cannot be reversed and asking for confirmation.



3. Click Yes to delete the plane.

Properties Command

This command displays the properties of the selected work plane.

1. Choose a user-defined work plane.
2. Choose the *Properties* command. The Define Work Plane dialog box appears displaying the name and the X, Y, Z coordinates of the work plane.
3. Change any value and click OK to close the dialog box and save the changes.

Save Current Command

This command saves the current work plane and adds it to the list of work planes in the Work Plane manager. It is automatically titled with the next available default label. If there are no other user-defined planes using a default label, the plane is titled, *UserWorkPlane01*.

You can rename this plane by selecting the name and choosing the *Properties* command in the Work Plane manager.

Show Work Pln Command

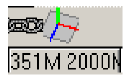
This is the same command available though the Planes menu. Choosing the command displays the work plane icon in your drawing.

Work Plane Identification

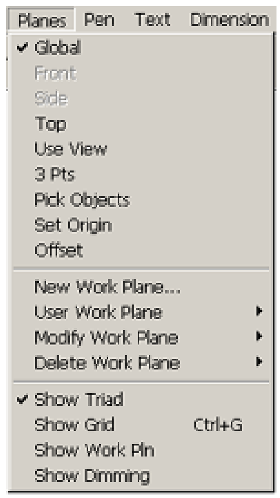
You can determine if your current work plane is global or user-defined through the coordinate system/work plane icon displayed to the left side of the Status Line.

The Global Work plane icon is a miniature version of the axis icon.

When a user-defined work plane or dynwork plane is chosen, an altered axis icon displays in the Status Line, as shown here.



Setting the Work Plane



In the Planes menu you have commands for setting the work plane. You may want to define a work plane other than the standard planes. For example, if you want to work on an angled face, you can reorient the work plane.

Tech Note:

If you are in the Front, Side or Top view, the work plane is already set to the respective plane and the other two pre-defined planes in the Plane menu are unavailable.

Temporary planes created with the *User View*, *3 Pts*, *Pick Objects* and *Define* commands are not saved for future use.

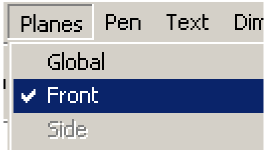
You can set the work plane six ways. From the Planes menu, choose a predefined plane, Front, Side or Top or use one of these commands, *Use View*, *3 Pts*, *Pick Objects* and *Define*. As explained earlier, you can also define and set the work plane using the work plane manager menu.

Work planes created using the *Use View*, *3 Pts*, *Pick Objects* and *Define* commands do not save for use later. They are only available until you set another work plane. Use the *New* command in the work plane manager to define and save a work plane with your file. If you have created objects on a temporary work plane, you can use the *Pick Objects* command and select the objects on that plane to set the work plane to them. (See a later section on using the *Pick Objects* command.

Choosing a Predefined Plane

1. Click or drag on the Plane menu to display the submenu.

2. Select one of the predefined planes.

   All three predefined planes are only available when your view is set to Isometric, Trimetric or a User-defined view.

   The work plane has changed to the selected plane.

Choosing the Front plane sets the x-axis to 0, 1, 0 and the y-axis to 0, 0, 1. Choosing the Side plane sets the x-axis to 1, 0, 0 and the y-axis to 0, 0, 1. Choosing the Top plane sets the x-axis to 1, 0, 0 and the y-axis to 0, 1, 0.

Choosing the Use View Command

This command sets the work plane to be coincident with the screen; the origin is in the center of the screen; the x-axis is coincident with the width of the screen; the y-axis is coincident with the height of the screen.

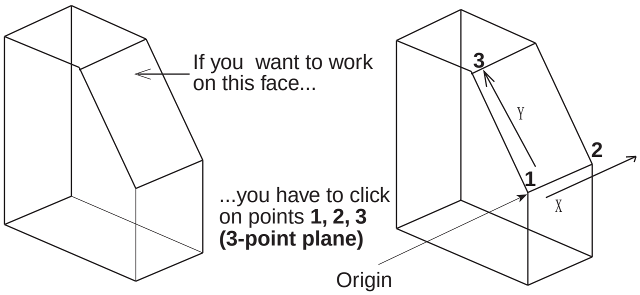
This sets the plane to match the screen in all views which is particularly useful when you are moving from view to view while drafting.

Choosing the 3 Pts Command

This command sets a temporary work plane from the three points you specify. This command is especially helpful if you want to create non-parallel planes.

Referral:

See Chapter 15 for information on using the **Infinite Plane** tool.



1. Choose ***Plane>3 Pts***. The Message Line reads: *Work Plane: Enter three points for work plane (1 origin, 2 x-axis, 3 y-axis).*

2. Click in the drawing area to indicate the origin point for the new work plane.
3. Click a point to define the positive x-axis.
4. Click a point to define the positive y-axis.

   A temporary work plane is created.

Choosing the Pick Objects Command

This command allows you to set the work plane based on selected objects. This includes selecting an infinite plane icon.

Tech Note:

This tool does not create a plane between curves on different planes. It only sets the plane to objects already in a plane.

1. Choose ***Planes>Pick Objects***. The Message Line reads: *Work Plane: Select objects to set Work Plane [Shift = Select].*
2. Select a curve or curves located in the plane that you want to set as the work plane.

   The work plane changes to the plane of the objects. Choose ***Plane>Show Work Pln*** to verify that the new plane is selected.

You can also use this tool to set the work plane to the face of a solid object.

Using the Work Plane Manager to set the Work Plane

1. Click on the work plane icon at the left of the Status Line.
2. From the menu select a work plane. (This menu displays the global work plane and any user-defined work planes.)

   The selected work plane is now set.

Work Plane Snapping

A short cut was added to allow rapid work plane positioning and alignment. This has been accomplished by further integration of the work plane with the Drafting Assistant. By default the short cut is set to the “c” key. When you hit the short cut key associated with work plane snapping you will see the following behavior:

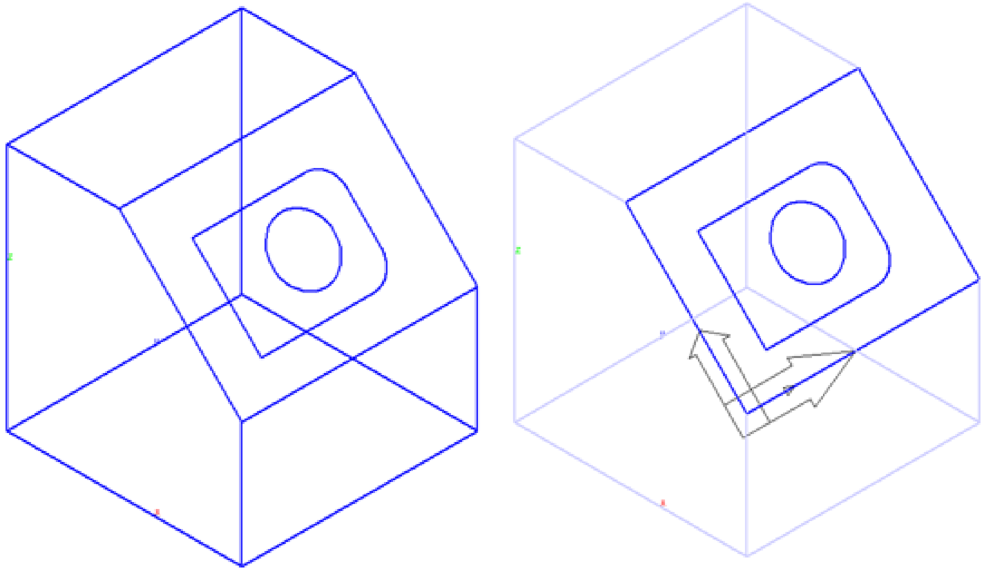
- Work plane origin moves to drafting assistant snap location. Aligns to top plane.
- Second “c” hit changes from top to side work plane orientation.
- Third “c” hit changes from side to front work plane orientation.
- Fourth “c” hit changes from front to use view work plane orientation.
- Fifth “c” hit changes from use veiw back to top plane.

If you have the face snap options on, hitting the short cut key will automatically align with respect to the face normal.

Note: you must have an object selected in order for the work plance snapping short cut to work.

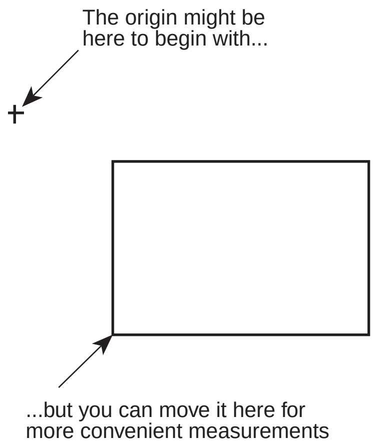
Work Plane Dimming

This menu command provides a means to dim objects that do not lie in the work plane.



Moving the Origin

Occasionally, you may simply want to move the origin of the work plane. This is especially useful for measuring distances. For example, you can open a document and start drawing without regard to the location of the origin, then move the origin to a convenient location for future reference.



Set Origin

This command sets a new origin in the current work plane.

Specifying a New Origin

1. Choose ***Planes>Set Origin***.
2. Click in the drawing area to indicate the location for the new origin.

   The origin of the current work plane moves but the orientation of x, y, z remains the same.

If you want to move the origin and change the orientation of the work plane, use either the *3 Pts* command or the *Define Plane* command.

Offsetting the Work Plane

Tip:

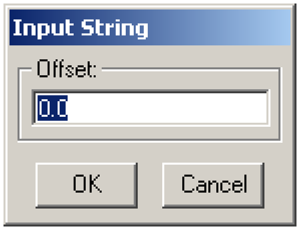
Use the *3Pts* command in the Planes menu if the new plane is not parallel.

As you're using this Designer Elements program, you may want to move the work plane a certain distance normal to the current work plane so that new geometry will snap to it. The *Offset* command in the Planes menu gives you this ability.

Using the Offset Command

1. Display the work plane icon. (This is not required but it will help you visualize the offset.)
2. Choose ***Planes>Offset***.

The following dialog box displays.



3. Enter the distance to offset (negative or positive) the work plane in the data field. The units for the distance are based on the *Units* page of Preferences.
4. Click OK to accept the value and close the dialog box.

   The work plane has changed. Click Cancel to close the dialog box without accepting the value.

Work Plane and Views

Standard Views

When you want to create geometry you choose a particular plane on which to draw. When you want to view geometry in this Designer Elements program, you choose a particular view orientation.

To eliminate the potential confusion between work planes and views this Designer Elements program has tied the Front, Side and Top views to their respective planes. For example, if you choose the Top view, the Top plane is selected in the Plane menu as the work plane and all other planes are unavailable.

See Chapter 30 for more information about Views.

View the Plane

This command in the View menu changes the view to the current work plane.

Work Plane and World Coordinates

This Designer Elements program uses the world coordinate system (as opposed to the User Coordinate System) for defining planes.

Rendering

After you create your geometry you can render it to display it more realistically. This Designer Elements program includes functionality for basic and advanced rendering. Basic rendering allows you to visualize and construct your model in various display modes. Advanced rendering allows you to create photorealistic images.

Tip:

When rendering geometry increasing the resolution on solid objects can improve the final image. This is especially helpful for round parts.

The topics covered in this chapter include the following:

- Lighting
- Basic Rendering
- Advanced Rendering
- Editing the Rendered Scene
- Photorealistic Rendering
- Animation
- Perspective Rendering

Lighting

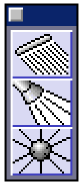
When you render your model this Designer Elements program automatically provides default lights to display your objects. This Designer Elements program also allows you to design your own lighting using four types of light sources: three directional sources (distant, spot and point) and the ambient light source. Use these lights to define the lighting characteristics of the model.

Directional Lighting

The distant light source, spot light source and point light source are directional sources. For each source you specify its location and direction. You place these lights in your drawing by using the Light tool palette.

These light sources also support shadows when working within the advanced rendering environment.

Light Palette



The Light tool palette does not automatically display when you launch this Designer Elements program. To display the palette choose ***Window>Lights***.

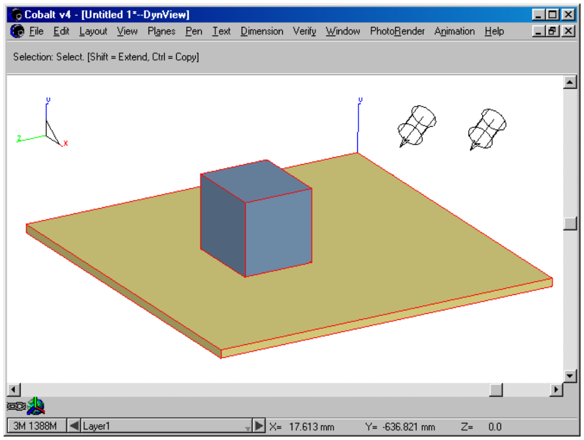
The Light tool palette includes the Distant Light (), Spot Light () and Point light (). Light sources placed using this palette contribute light color and intensity to all objects that lie within their influence and are used to define the light.

Light sources appear in your drawing as symbols in the wireframe and basic render modes for construction purposes. The light source symbols do not appear when using the photorealistic rendering commands.

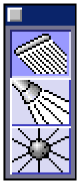
Light sources can be hidden using the *Show/Hide* command or moved to a separate layer and hidden. However both actions turn off the light.

Distant Light Source

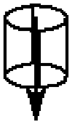
A distant light source illuminates a scene with parallel rays of light as if they emanate from a very distant light source. Like the sun, this source illuminates all parallel surfaces equally.



Using the Distant Light Tool



1. Select the Distant Light tool. The Message Line reads: *Distant Light: Enter location and direction positions.*
2. Click a point to set the location of the light.
3. Click the next point to establish the direction of the light.



The exact distance is not important. A distant light source symbol appears in your drawing composed of a cylinder with an arrow pointing along the source center-line.

If *Show Points* in the Edit menu is checked the center-line extends from the source location to the direction point, as in the graphic here.

Place more lights as desired. As you add lights you may want to render the model again to verify the lighting effect.

Geometric Characteristics

According to the Edit Objects Geometry page a distant light source is made up of the following characteristics: Type and Intensity. It also includes the Enable Light check box, Cast Shadows check box and its associated pull-down menu and the Cast Volumetric Shadows check box. The Attenuation menu is not available since a distant light source illuminates all surfaces equally.

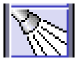
The Enable Light check box gives you the ability to turn off a light while retaining the intensity, direction and location settings. When checked, the light is on. The default setting is on.

The Cast Shadows check box controls whether a shadow is cast by an object when a particular light is directed towards it. When checked, shadows are cast. When unchecked, no shadows are cast as a result of that light. With its associated pull-down menu, you also have the option of setting the type of shadow casting. You have four shadow type options: hard, medium, soft and blurry.

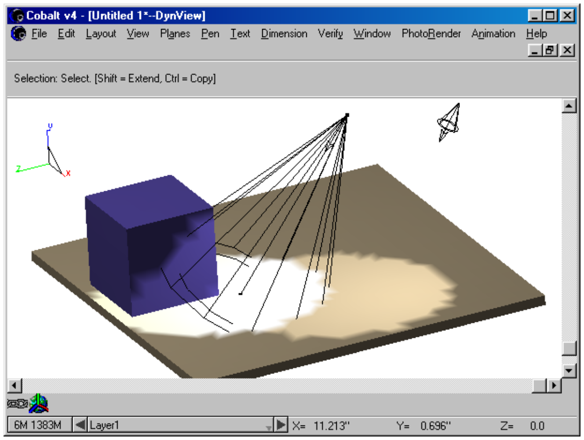
The ability to control shadow casting is especially valuable in a scene containing multiple light sources. Too many shadows can render a scene confusing. This check box setting only applies when rendering your scene with Preview Render (shadows on) or Raytrace Render (shadows on, Anti-Alias). See the “Photorealistic Rendering Commands” section later in this chapter.

The light source color is accessible through the Attributes page.

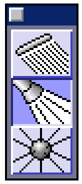
Spot Light Source



A spot light source illuminates a scene with a cone of light emanating from a local source. This source functions similarly to a flashlight.



Using the Spot Light Tool



1. Select the Spot Light tool.
   The Message Line reads: *Spot Light: Enter location and direction positions.*
2. Click the first point to set the location of the light.
3. Click the next point to establish the direction of the light.

Distance is only important if you want to set the light with an attenuation (see the Geometric Characteristics section for information on attenuation). The default setting does not include attenuation. A spot light source symbol appears in your drawing, composed of an inner and outer cone and an arrow pointing along the source center-line.

If *Show Points* in the Edit menu is checked, the center-line extends from the source location to the direction point. The sides of the cone extend to a plane normal to the direction, and end at the direction point, as in the graphic here.

Place more lights as desired. As you add lights, you may want to render the model to verify the lighting effect.

Geometric Characteristics

According to the Edit Objects Geometry page, a spot light source is made up of the following characteristics: Type, Intensity, Attenuation, Cast Volumetric Shadows, the Cone Angle data field, Falloff Angle data field, Falloff Rate data field and the Slide check box. It also includes the Enable Light check box, Cast Shadows check box and its associated pull-down menu.

The Enable Light check box gives you the ability to turn off a light while retaining the intensity, direction and location settings.

The Cast Volumetric Shadows check box enables simulation of a full range of effects occurring in a participating medium, i.e. attenuation within the medium, light filtration through a colored medium and first order light scattering inside the medium with volumetric shadows.

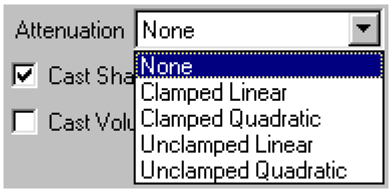
The light scattering effects are modeled for all light shaders except "ambient", "eye" and "sky" (though are available in the "area sky" shader). The "first order scattering" mentioned above means that the shader visualizes direct the effect of scattering through the medium, scattered light coming to the observer but does not consider the secondary effect, multiple bounces of light within the medium or illumination of surfaces by scattered light.

The shader performs attenuation of the original surface color according to given medium attenuation coefficient and medium color. As one could expect the longer the distance from the shaded point, the dimmer and more colored by the medium it becomes. In addition to attenuation it is also possible to specify the medium ambient light scattering, which gives the overall veiling effect. Finally the shader models scattered light from all the supported light sources if the "scattering" parameter of the source is set to TRUE.

The Cast Shadows check box controls whether a shadow is cast by an object when a particular light is directed towards it. When checked, shadows are cast. When unchecked, no shadows are cast as a result of that light. With its associated pull-down menu, you also have the option of setting the type of shadow casting. You have four shadow type options: hard, medium, soft and blurry. This check box setting only applies when rendering your scene with Preview Render (shadows on) or Raytrace Render (shadows on, Anti-Alias). See the “Photorealistic Rendering Commands” section later in this chapter.

Attenuation Settings

Attenuation controls how quickly the light intensity diminishes with the distance from the light. You have the following attenuation options:



None Light intensity does not change with distance.

Clamped Linear Light intensity diminishes according to the following formula, *Intensity /(distance+1)*.

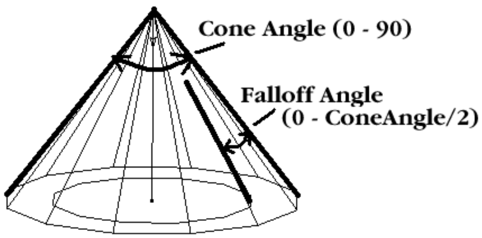
Clamped Quadratic Light intensity diminishes according to the following formula, *Intensity* $/(distance^2+1)$.

Unclamped Linear Light intensity diminishes according to the following formula, *Intensity /distance*.

Unclamped Quadratic Light intensity diminishes according to the following formula, *Intensity* $/distance^2$.

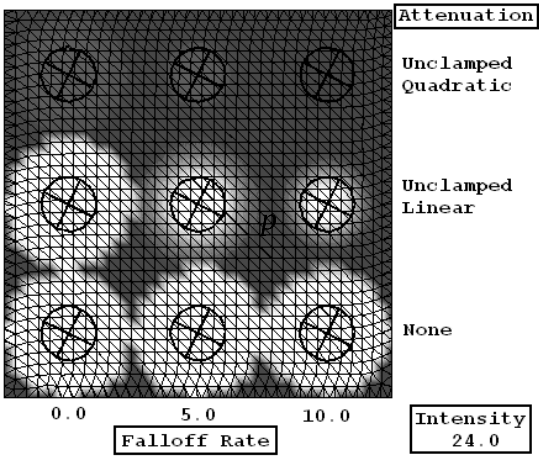
Use clamped attenuations in situations where the distance from a light source to an object is less than one 1” (25.4 mm). Clamped attenuations do not increase intensity when the distance is less than 1” (25.4 mm). Unclamped situations are more realistic and apply to most situations.

Cone Angle controls the maximum spread of the spot light. Objects that lay outside the area defined by the cone does not receive light. Objects that lay within the cone receives light according to the specified attenuation, falloff angle and falloff rate.



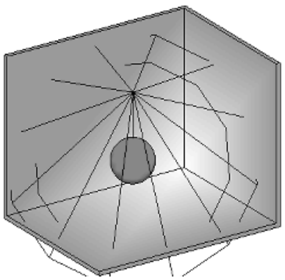
Falloff Angle controls the sharpness of the spot light’s edge. The smaller the angle is the sharper the edge. The light intensity diminishes from full intensity at the inner cone to zero at the outer cone. The value for the falloff angle ranges from zero (0) degrees to *Cone Angle/2* degrees.

Falloff Rate controls how light is distributed within the spot light's cone. The intensity diminishes from the center line of the cone outward with the cosine of the angle raised to the power of the falloff rate. A falloff rate of zero (0) results in uniform light within the cone. Higher values produce more pronounced falloff. The falloff rate can be any value from zero (0) to ten (10).

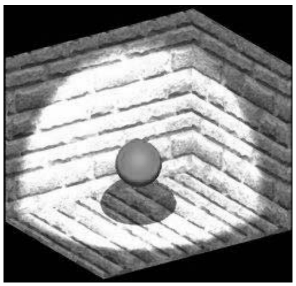


The graphic here shows the light attenuation and falloff rate for a spot light with an intensity of 24.

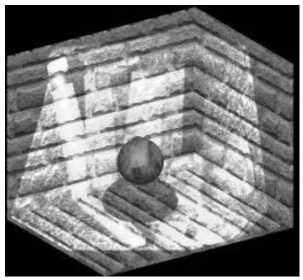
The Slide check box controls whether a spot light functions as a slide projector and projects an image on the scene. The example below shows the lighting scene with the spot light location and the lighting environment with and without the slide image.



The scene



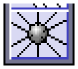
Without slide image

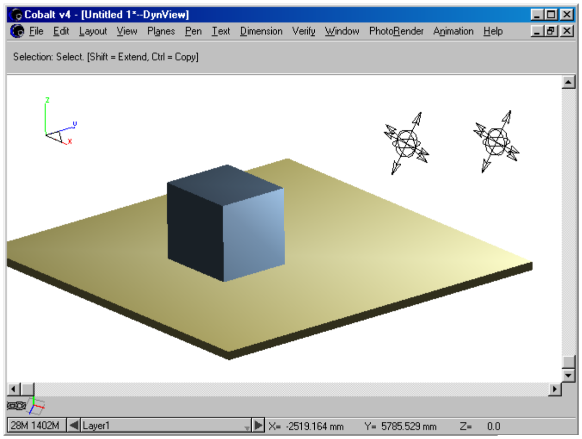


With slide image

The light source color is accessible through the Attributes page.

Point Light Source

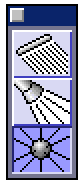




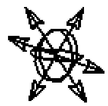
A point light illuminates a scene with light emanating in all directions. A candle or a table lamp is a type of point light.

Using the Point Light Tool

1. Select the Point Light tool.
   The Message Line reads: *Point Light: Enter location*.



2. Click to set the source location of the light.



Distance is only important if you want to specify an attenuation (see the Geometric Characteristics section for information on attenuation). The default setting does not include attenuation. A point light source symbol appears in your drawing with arrows pointing outward If *Show Points* in the Edit menu is checked, the source location point displays.

Place more lights as desired. As you add lights, you may want to render the model again to verify the lighting effect.

Geometric Characteristics

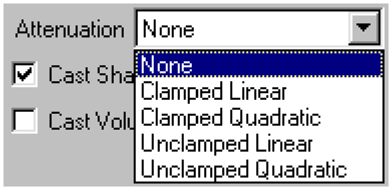
According to the Edit Objects Geometry page, a point light source is made up of the following characteristics: Type, Intensity, Attenuation. It also includes the check boxes, Enable Light and Cast Shadows and its associated pull-down menu and the Cast Volumetric Shadows check box.

The Enable Light check box gives you the ability to turn off a light while retaining the intensity, direction and location settings.

The Cast Shadows check box controls whether a shadow is cast by an object when a particular light is directed towards it. When checked, shadows are cast. When unchecked, no shadows are cast as a result of that light. With its associated pull-down menu, you also have the option of setting the type of shadow casting. You have four shadow type options: hard, medium, soft and blurry. This check box setting only applies when rendering your scene with Preview Render (shadows on) or Raytrace Render (shadows on, Anti-Alias). See the “Photorealistic Rendering Commands” section later in this chapter.

Attenuation Settings

Attenuation controls how quickly the light intensity diminishes with the distance from the light. You have the following attenuation options:



None	Light intensity does not change with distance.
Clamped Linear	Light intensity diminishes according to the following formula, *Intensity /(distance+1)*.
Clamped Quadratic	Light intensity diminishes according to the following formula, $Intensity /(distance^2+1)$.
Unclamped Linear	Light intensity diminishes according to the following formula, *Intensity /distance*.
Unclamped Quadratic	Light intensity diminishes according to the following formula, $Intensity /distance^2$.

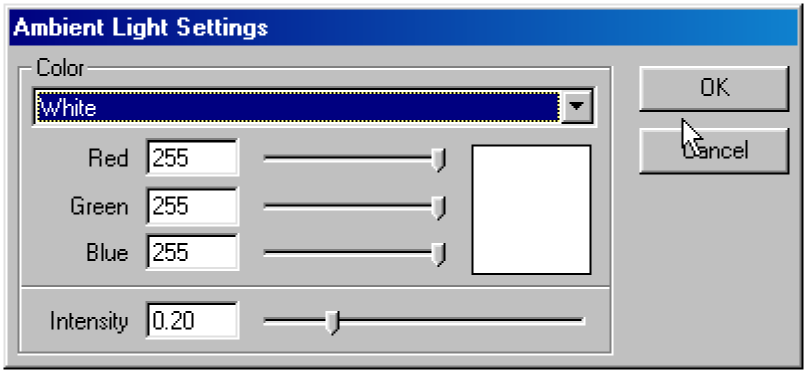
Use clamped attenuations for situations were the distance from a light source to an object is less than 1” (25.4 mm). Clamped attenuations do not increase intensity when the distance is less than 1” (25.4 mm). Unclamped situations are more realistic and apply to most situations.

The light source color is accessible through the Attributes page.

Ambient Lighting

Ambient light contributes light color and intensity to all objects in the scene. Ambient light penetrates all holes, indentations and cutouts of an object, illuminating all surfaces equally. Ambient light does not cast shadows.

Choose ***View> Ambient Light Settings*** to display the dialog box.



The dialog box contains the following options:

Color — This area sets the ambient light color. You can either choose a color from the menu, enter a Red, Green, or Blue value in the respective data field or drag the RGB slides to the desired value. A color preview appears to the right of the sliders. The default ambient light color is white.

Intensity — This option sets the ambient level. You can either enter the value in the data field or use the slide to set the intensity. Zero (0) is off.

After choosing your ambient settings, click OK. When you save your file, the ambient light setting is also saved.

Modifying the Lights

As you design your geometry and render it you may want to view it under various lighting conditions, either by changing the layout or editing the light source.

Modifying Ambient Light

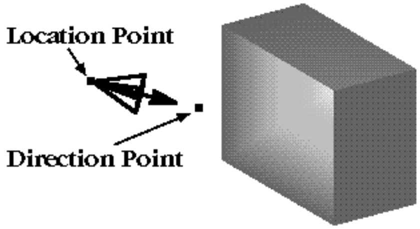
Ambient light can modified through the Ambient Light Setting dialog box in the View menu. Adjust the color and intensity level and click OK to save the changes. Choose ***PhotoRender>Render*** to display your geometry with the new settings.

Modifying Directional Lights

You can modify directional lights (Distant, Spot and Point) in a number of ways including moving, deleting and changing the intensity and color.

Moving a Light Source

Light sources are objects and can be moved like any other object. The Distant and Spot light sources have two control points, the source location and the source direction. You can move one or both points The Point light source has one control point, the source location.



Moving the Entire Light Source

Select the source. Place the cursor over the source location point and drag it to the new location. Choose ***PhotoRender>Render*** to display the image with the new light setting.

Moving the Control Points

Select the source and choose ***Edit>Show Points*** to display the control points. Drag a selection fence around the desired point and move it to the new location. Choose ***PhotoRender>Render*** to display the model with the new light setting.

Editing a Directional Light Source

You can perform comprehensive editing on directional light sources through the Edit Objects dialog box. The options available depend on the light source.

Geometry Tab

Type	You choose from the three lights available: Distant, Spot and Point.
Intensity	You can change the light intensity. Any value equal to or greater than zero is valid. Zero (0) turns the light source off.
Attenuation	(Spot and Point lights only) Controls how quickly the light intensity diminishes with the distance from the light.
Enable Light	This check box controls the activation of the light source. When checked, the light is on. This check box allows you to turn off a light while retaining the intensity, direction and location settings. The default setting is on.

Cast Shadows This check box controls the display of shadows when a light shines on an object. A pull-down menu offers four shadow types: hard, medium, soft and blurry. This command works with Preview Render (shadows on) and Raytrace Render (shadows on, Anti-Alias) rendering commands.

Cone Angle (Spot light only) Controls the maximum spread of the spot light.

Falloff Angle (Spot light only) Controls the sharpness of the spot light's edge.

Falloff Rate (Spot light only) Controls how light is distributed within the spot light's cone.

Slide (Spot light only) This check box controls whether a spot light functions as a slide projector and projects an image on the scene.

Attributes Tab

Name You can enter a specific name for the source.

Resolution This characteristic does not apply to light sources.

Color You can specify any color for your light source. The pull-down menu offers the standard colors and the *More* option, which displays the color palette when selected.

Control Pts You can display or hide the control points of the light source.

Layer You can place a light source on any layer. The menu displays all available layers.

Locked You can locked the light source to prevent modification.

Click Apply to accept your changes and then Close to exit the dialog box. Choose ***PhotoRender>Render*** to display your geometry with the new light settings.

Deleting a Light Source

To delete a light source select the source and press the BACKSPACE key (Windows) or the DELETE key (Windows and Macintosh).

Default Lighting

This Designer Elements program provides default lighting for your geometry when you use the *Render* command. The default lighting is based on the viewer's eye location and direction to provide a high quality rendering of your geometry. The default light source cannot be modified. However, you can specify your own ambient light with the default lighting. When you place your own light sources, the default light source deactivates.

User-Defined Lighting Layouts

If you create your own lighting layout, you can use that layout for other drawings by exporting the lights to a stand-alone Designer Elements program file. This lighting file can then be imported into other files. To export a lighting set, do the following:

1. Select all the light sources by selecting each individually or using the Selection Mask.
2. Choose ***File>Export***.
3. Select the Designer Elements program format.
4. Choose the Selected Only option.
5. Click OK and save the file as desired.

You can now import this lighting into other Designer Elements program files.

Basic Rendering

In this Designer Elements program you are able to design, display and edit your models in a wide range of visualization modes. These include the static and dynamic options, wireframe, Flat shading, Gouraud shading, Phong shading and Hidden Line modes (see the Render Options section for shading definitions). Basic rendering is implemented under Open GL available through your video card. You do not need to select the objects to render them.

Tech Note:

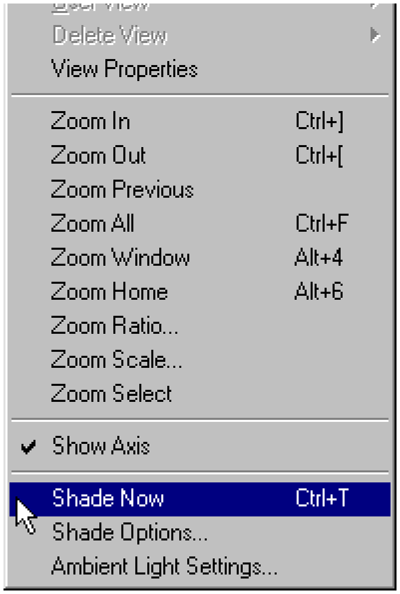
This Hidden Line mode is different from the hidden line images that appear in Drawing Views when using the *Model to Sheet* command. See Chapter 36 for more information.

Shade Now - CTRL+T (Windows); z +T (Macintosh)

This *Shade Now* command in the View menu, displays your geometry using the *Render Now* setting selected in the Render Options dialog box (see the next section).

To use this command, choose ***View>Shade Now***.

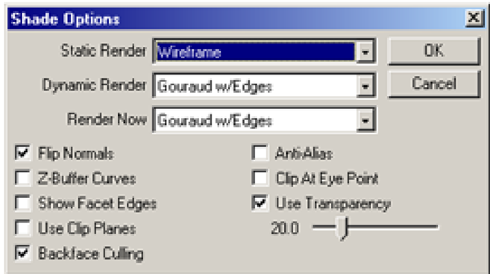
All surface and solid geometry in your drawing renders.



Shade Options

This Designer Elements program provides a number of options for rendering your geometry.

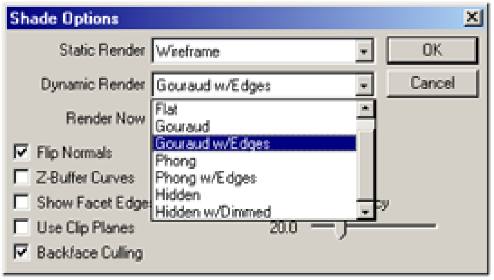
When you choose ***View>Shade Options***, the following dialog box displays.



The dialog box includes the following rendering categories:

Static Render	This category defines how your geometry displays when the view is stationary. This Designer Elements program automatically repaints the object if this setting is changed and saved.
Dynamic Render	This category defines how your geometry displays as you rotate the view.
Render Now	This category defines how your geometry displays when you choose ***View>Shade Now***.

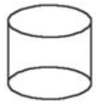
When you click the arrow in a category a pull-down menu appears.



You have the following rendering options:

Wireframe

This option displays only the edges of the 3D model. This is the normal repaint display mode.



Flat

This option displays your geometry with a painter's algorithm using constant shading techniques.

Gouraud

This option displays your geometry based on calculated light intensities at each vertex. This rendering method uses Open GL (Windows) or QuickDraw 3D (Macintosh).

Gouraud w/Edges

This option displays your model with face edge boundaries, silhouettes and isolines of a surface or solid on top of a rendered model. You can specify the edge color in the *Display* page of Preferences (see Chapter 6, "Preference Settings").

Phong

This option displays your geometry based on calculated light intensities at each pixel location.

Phong w/Edges

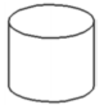
This option displays your model with face edge boundaries, silhouettes and isolines of a surface or solid on top of a rendered model. You can specify the edge color in the *Display* page of Preferences (see Chapter 6, "Preference Settings").

Tech Note:

If you set your Dynamic Render mode to Phong with Edges and notice that at time your model vanishes, that's the result of light reflection. Adjust your light levels or location and render again.

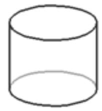
Hidden

This option displays only the visible edges of your geometry. Since this a rendering mode, any text present in wireframe does not display. Text and wireframe objects do not display in this-mode.



Hidden w/Dimmed

This option displays the visible edges with the hidden edges of your geometry dimmed. Since this is a rendering mode, any text present in wireframe does not display. Text and wireframe objects do not display in this mode.



The dialog box also includes another render option setting:

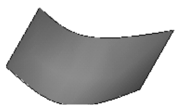
Flip Normals

Placing a check mark in this box flips the light normal on objects. The Flip Normals box is checked by default. This Designer Elements program automatically repaints the object if this setting is changed and saved.

When you display a rendered object, its appearance is determined by the light locations and the object orientation. If light normals are pointed away from the viewer, the object will appear dark. This can happen when geometry is imported in from another program.

Flip Normal Example

If you import the following object and render it without the *Flip Normal* option checked parts of the object appear dark.



Check the *Flip Normal* option and render the object again. The normals pointing away from you are flipped and the darkened areas of the object render more acceptably.

You can also choose ***Edit>Change Direction*** to flip the direction of the normals.

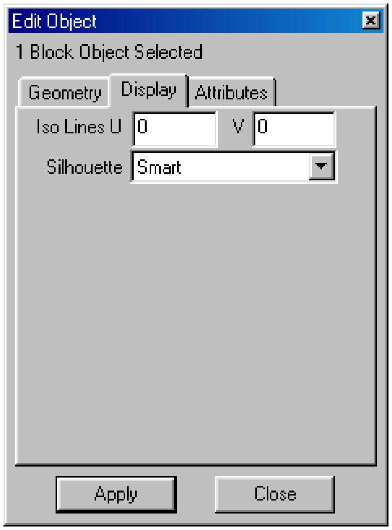
Setting the Shade Options

1. Choose ***View>Shade Options***. The dialog box displays.
2. Select the render options you desire.
3. Click OK and the options are accepted. (Click Cancel to close the dialog box without saving the settings.)

   If you changed the *Flip Normal* or the *Static Render* settings this Designer Elements program automatically repaints.

These settings save as the default settings when you exit the program.

If you've chosen Gouraud w/Edges or Phong w/Edges and you want to display your model with isolines or silhouette, double-click on your model to open the Edit Objects dialog box. In the *Display* page, specify your desired settings and click Apply.



Z-Buffer Curves — This option turns on or off OpenGL depth buffering of curves as they are displayed in context with surfaces and solids. With this option on, curves behind surfaces or solids are hidden and visible when turned off.

Show Facet Edges — The Show Facet Edges option turns on or off the edges of the display facets. The facet density is determined by the entity resolution.

Use Clip Planes — This option indicates whether to process clip planes for OpenGL rendering. When turned on, all planes entities that are marked as clip planes will clip the current view. To mark a plane as a clip plane, right click over the entity and select the appropriate option in the menu.

Backface Culling

Back face culling is an option that greatly enhances OpenGL performance. When this option on, OpenGL ignores all facets from solids that have normals facing away from the viewing direction. Turn this feature off if you have hybrid solids that mix open surfaces with solids, otherwise the surface may appear invisible in OpenGL display modes.

Anti-Alias

This option tells the OpenGL drivers to enable anti aliasing of edges.

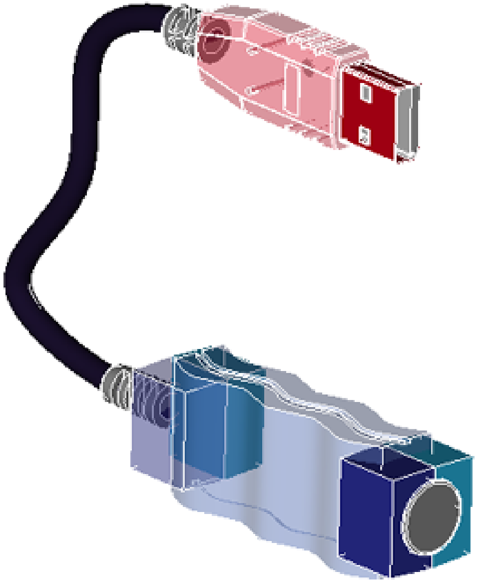
Clip At Eye Point

The clip at eye point turns on or off clipping at the eye point. If off, clipping is automatically determined based on the extents of the given model.

Use Transparency

This option indicates whether to process objects marked with a transparency flag for OpenGL transparency. When turned on all surface and solid entities that are marked as transparent will clip the current view. To mark an object as transparent, right click over the entity and select the transparent option from the menu.

The slider value indicates a global transparency setting. A setting of 100 implies no transparency and a value of 0 implies fully transparent. Below is an example of an object containg transparent parts.



Render Option Combinations and View Rotation

Because the rendering options deal with the display of your geometry, the rendering categories operate with each other and are affected by the view.

Shade Options Example

1. Create some geometry.
2. In the Shade Options dialog box, set *Render Now* to Hidden w/Dimmed and *Static Render* to Wireframe.
3. Click OK to save the setting and close the dialog box.
4. Choose ***View>Shade Now*** and your geometry is displayed in Hidden w/ Dimmed view mode.
5. Rotate the view. The geometry is now displayed as wireframe because of the *Static Render* setting.

Other Shade Combinations

Here are a few render combinations.

Static Render - Wireframe, Dynamic Render - Gouraud

As you rotate the view your geometry displays with Gouraud shading. When you complete the rotation the geometry displays as a wireframe.

Static Render - Gouraud, Dynamic Render - Gouraud

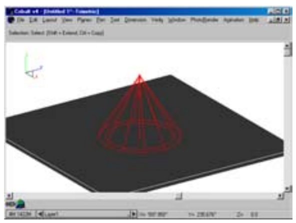
As you rotate the view your geometry displays with Gouraud shading. When you complete the rotation the geometry displays in Gouraud shading.

Render Options and Preferences

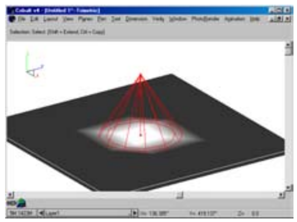
Render Options are saved automatically when you exit the program. They are written out to the preferences file.

Basic Rendering and Lighting

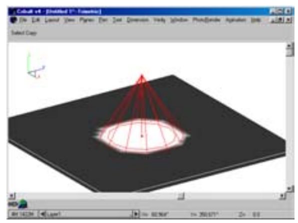
All facets are illuminated according to how much light falls on their vertices (the intersection of perpendicular isolines). If a light source is positioned over a large flat surface (such as a floor), it will appear that the source is not casting light. Set the object resolution to *Very Fine* or *Super Fine* to increase the facet density and enhance the lighting result. See Chapter 24 for more information on resolution.



Fine Resolution



Very Fine Resolution



Super Fine Resolution

Advanced Rendering

This Designer Elements program provides you with advanced rendering tools that control the definition and generation of photorealistic images. Unlike the basic rendering functionality, advanced rendering is not used during geometry construction and editing. Advanced rendering is a back-end tool that interacts with the geometry already created. Advanced rendering is implemented using the LightWorks rendering engine for both Windows and Macintosh.

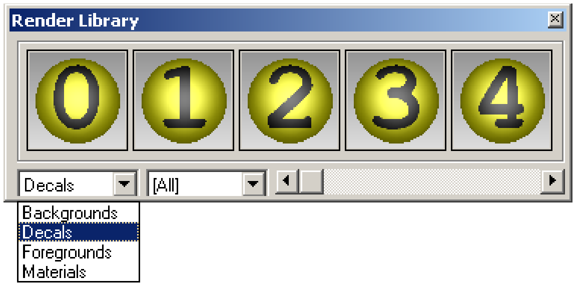
The advanced rendering tools include the render library, advanced settings and rendering modes. You do not need to select objects to render them.

Render Library

This Designer Elements program comes with an extensive render library containing materials and decals that you can apply to objects and background and foreground properties that you can apply to the scene. This library is contained in the Render Library tool palette. Choose ***Window>Render Library*** to display the tool palette.

You can resize the palette horizontally or vertically by dragging its edges or corners.

The palette contains the preview area, library type menu, category menu and scroll bar. The type menu controls which library displays on the palette. The Decals library type is shown in the graphic here.



Render Library Items

The render library menu contains four render libraries: Backgrounds, Decals, Foregrounds and Materials. You can select any library by clicking on it or using the up or down arrow keys on your keyboard.

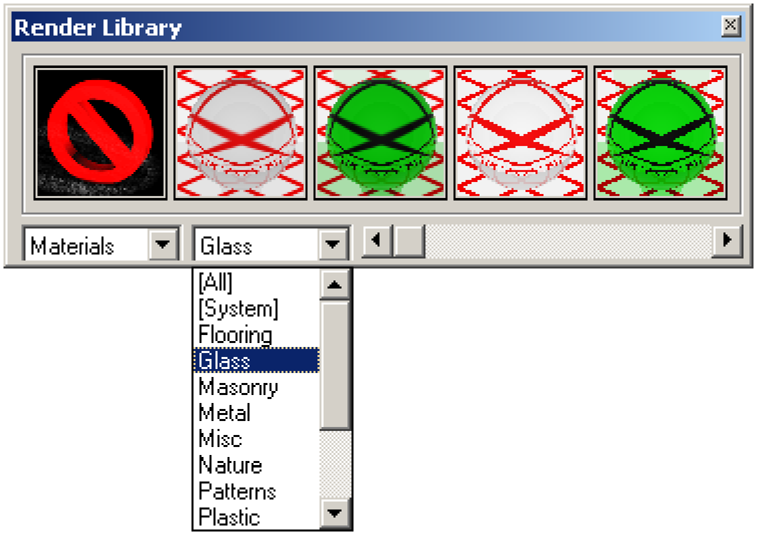
Backgrounds — Contains pre-defined background effects that can be applied to a scene.

Decals — Contains pre-defined decals that can be applied to renderable objects.

Foregrounds — Contains pre-defined foreground effects that can be applied to a scene.

Materials — Contains pre-defined material properties that can be applied to renderable objects.

Within each library render images are divided into categories and displayed in the category menu. You can select any category within the category menu by clicking on it or using the up or down arrow keys on your keyboard. The Glass category for Materials is shown in the graphic here.



In addition to the render image categories each library also includes two standard categories, [All] and [System].

[All] — Selecting this category displays every image in the chosen library.

[System] — Selecting this category displays images originally shipped with this Designer Elements program.

Most of the items in the Render Library define the color of your objects regardless of their pen color. However, to provide you with greater variety some materials use the object's color. These include the following: Glass (Colored), Masonry (Brick-

Colored), Metal (Polished), Misc. (Plain, Plain Rough, Plain Texture, Screen), Nature (Clouds), Patterns (Check Board, Cubes, Grid, Polka Dot Solid, Polka Dot Wrapped), Plastic (Clear Rough, Clear Texture, Opaque Rough, Opaque Texture, Stone (Marble), Wood (Simple) and Decals.

When you place one of these materials on an object, the material displays using the object's color. For example: Brick-colored masonry placed on a blue object renders as blue masonry; Clear Rough Plastic placed on the yellow object renders with a yellow tint; Clouds placed on a red object renders as white clouds on a red object; Polka Dots placed on a green object renders as green dots on a white object.

Using the Materials Library

The Materials library contains materials you can apply to your model. A material defines the surface color, transparency, reflectivity and roughness properties of a model. You can produce a wide variety of visual appearances with different combinations of these attributes.

The categories in the Materials library include Flooring, Glass, Masonry, Metal, Misc, Nature, Patterns, Plastic, Stone, Tiled Textures, Walls, Wood and Woven Textures.

Flooring	Render images include: Tile 1, Tile 2, Tile 3, Tile 4, Tile 5, Tile 6, Tile 7, Wood Slats 1 and Wood Slats 2.
Glass	Render images include: Clear, Colored and Mirror. The Clear and Colored options only display correctly when using one of the ray trace rendering commands.
Masonry	Render images include: Block Stone, Brick (Colored), Brick (White), Brick (Peach), Cobble Stone, English 1, English 2, Flemish, Pavement (Gray), Pavement (Red), Rustic, Stone Wall.

Tech Note:

If you're using highly reflective material for an object in an empty space, the rendered image may disappear since there is nothing to reflect. Add some surrounding environment and render again.

Metal	Render images include: Aluminum, Aluminum (Circular Brushed), Aluminum (Linear Brushed), Chromium, Cobalt, Copper, Gold, Graphite, Mercury, Nickel, Palladium, Platinum, Polished, Silver and Tungsten.
Misc	Render images include: Bubble Wrap, Chisel, Plain Rough, Plain Texture and Screen.
Nature	Render images include: Clouds, Clover, Grass, Leather, Skin, Sky 1, Sky 2 and Sponge.
Patterns	Render images include: Checker Board, Cubes, Grid, Polka Dot Solid and Polka Dot Wrapped.
Plastic	Render images include: Clear Rough, Clear Texture, Opaque Rough, Opaque Texture and Translucent. The Clear Rough, Clear Texture and Translucent options only display correctly when using one of the ray trace rendering commands.
Stone	Render images include: Conglomeration, Granite, Gravel, Limestone, Marble, Marble (Gray), Marble (Pink), Sand and Silt.
Tiled Textures	Render images include: Knurl, Knurl 2, Mesh 1, Mesh 2, Pitted, Splatter, Swarf, VS Logo and Waffle.
Walls	Render images include: Diamond Paper, Tile 1, Tile 2, Tile 3, Tile 4 and Tile 5.
Wood	Render images include: Bark, Beech, Black Limb, Black Palm, Black Palm 2, Bocotec, Brazilian Ebony, Brazilian Rosewood, Bubingac, Canary, ChakteKok, Cocobola, Cork 1, Cork 2, CurlyKoac, Flamewood, Gabon Ebony, Goncalo Alvesc, Holly, Honduran Rosewood, Indonesian Rosewood, Kingwood, Lacewood, Lignum Vitae, Macassar Ebony, Madagascar Rosewood, Mahogany, Maple, Narrac, Pau Ferr, Pear, Pernambuco, Peroba Rosa 2, Pine, Pink Ivory, Primavera, Satinwood, Sawdust, Sheduac, Simple, Spalted, Tulipwood, Verawood, Walnut and Zebrawood.

Woven Textures Render images include: Basket 1, Basket 2, Denim 1, Denim 2, Dog Tooth, Hessian, Loop and Webbing.

The graphic shown is an example of Cobble Stone masonry applied to a block.

Applying Materials

1. Choose ***Window>Render Library*** to display the dialog box.
2. Select the materials library and desired category in their respective menus.
3. Place the pointer over the desired render image and drag. As the image is dragged over the drawing, the pointer becomes the material application symbol.

Application Sym-

4. Continue dragging to the object to which you want to apply the image. The object highlights. Release the mouse button.

   You can apply a material to multiple objects simultaneously by selecting one or more objects while holding down the SHIFT key, position the cursor over the preview image (while still holding down the SHIFT key), then click the mouse button.

   A dialog box appears asking you to confirm the operation.

5. Apply more images to your objects as desired. Then render your drawing using one of the commands in the Render menu.

Tip:

If you want to apply different materials to different faces of a solid, convert the faces to surfaces using the *Change Object Type* command and apply the materials to each individual surface.

Geometric Characteristics

When you apply materials to an object, a Material page is added to the Edit Objects dialog box. Each material potentially has a large set of characteristics determined by the internal material definition. These characteristics have been mapped to a basic set or common characteristics and include the following items: Reflectivity, Transparency, Roughness, Scale, Texture File, Enable Shadow Cast check box, Enable Shadow Receive check box, Double Side Facets check box and Is Backdrop Object check box. These characteristics are explained in the Material Editing section on page 38.

Removing or Replacing Material

To remove material from an object, drag the None symbol from the Render Library to the object and the material is removed. The symbol is shown here.

You can replace one material with another by simply dragging the new material image to the desired object. You can also replace a material by displaying the Edit Objects dialog box, selecting the Material page and from the Texture file pull-down menu, choose another texture.

See page 38 for more information about editing materials.

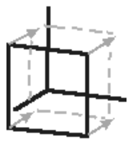
Using the Decals Library

The Decals library contains pre-defined decals that you can apply to your model. A decal overrides the local surface color properties of an object to which is it applied.

When applied to an object, the decal snaps to its surface and adjusts its shape (planar rectangle, cylindrical patch or spherical patch) according to the selected surface. The decal shape is automatically determined by the surface curvature using one of this Designer Elements program internal wrap modes, Planar, Cylindrical or Spherical.

Planar Wrap

This mode projects artwork along the decal's normal vector and appears on the surface at the location you specify. The graphic shows a planar decal symbol. The gray arrows indicate the nor-

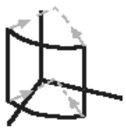


mal direction. The area defined by the artwork and the normals (the dotted lines) is the planar wedge.

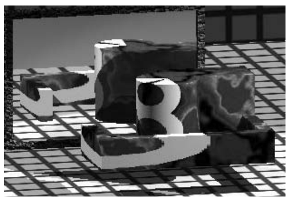
The graphic shows planar wrapping for one object using two decals. One decal extends beyond the surface edge and wraps along the other adjacent surface as defined by the planar wedge. The object's reflection appears in the mirror behind it.

Cylindrical Wrap

This mode projects artwork along the surface normals of an imaginary cylinder that passes through the decal's location point. The decal molds itself to the selected object adjusting to the object's radius. The area defined by the artwork and the normals are the decal wedge area.



The graphic here shows a cylindrical decal symbol with the normal direction indicated by the arrows. Any portion of the object that lies within the area bounded by cylindrical wedge (the dotted area in the graphic) and its normal displays the artwork. If the decal extends past the object edge, the decal will drag towards the center of the object.



The graphic here shows cylindrical wrapping for one object composed of a cylinder, bracket and block. The decal extends beyond the edge of the cylinder and bends toward the center of the cylindrical wedge, wrapping around the object. The object's reflection appears in the mirror behind it.

Spherical Wrap

This mode projects artwork along the surface normals of an imaginary sphere that passes through the decal's location point. The decal molds to the selected object adjusting to the object's radius. The graphic shows a spherical symbol with the normal direction indicated by the arrows. Any portion of the object that lies within the spherical wedge (the dotted area in the graphic) and its normal displays the artwork.

The graphic shows spherically wrapping for one object composed of a sphere and a bracket.

The categories in the Decals library include Misc and Digits.

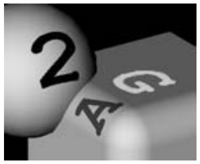
Misc

Render images include: Eroded. When an object is rendered with this decal, the object material appears to be wearing away.

Digits

Render images include the numbers 0 through 9.

This graphic is another example of the decal wrap modes.

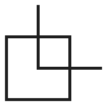


Tip:

To prevent a decal from showing through an object, select the decal and display the Edit Objects dialog box. Deselect the Auto Wrap option and select either the cylindrical or spherical wrap mode.

Applying Decals

1. Choose ***Window>Render Library*** to display the dialog box.
2. Select the desired library and category in their respective menus.
3. Place the pointer over the desired render image and drag. As the image is dragged over the drawing, the pointer becomes the decal symbol.



Decal Symbol

4. Continue dragging the decal to the object to which you want it applied. Use the axes included with the symbol for alignment. Release the mouse button.

Apply more decals to your objects as desired. Then render your drawing using one of the commands in the Render menu.

Removing or Replacing Decals

To remove a decal from an object select the decal and delete it like any other object. You can replace one decal with another using the Edit Object dialog box.

Geometric Characteristics

A decal is considered an object in this Designer Elements program. When you apply a decal, the Edit Objects dialog box contains characteristics specific to a decal object. The characteristics include: Rotation, Width, Height, Wrap Mode, Auto Wrap check box, Lock Normal check box, Masking, Stencil and Logo.

Rotation This field sets the rotation angle of the decal. The default value is 0. Entering a new value (between 0 and 360) and clicking Apply rotates the decal around the decal normal by the specified angle.

Width/Height These fields set the decal's coverage area.The values are linear or arc lengths depending on the wrap mode.

Wrap Mode This field displays the decal wrap mode. It also includes a pull-down menu listing the three wrap modes, planar, cylindrical and spherical.

When Auto Wrap is not selected, use the wrap mode list to set the decal wrap mode. If the decal is moved to another object, the wrap mode does not change.

Auto Wrap This check box allows you to control the wrap operation. When this box is checked, the wrap mode is matched to the surface curvature of the underlying object. The computed wrap mode is displayed in the Wrap Mode field. If the decal is moved to another face of the object, the wrap mode is re-calculated.

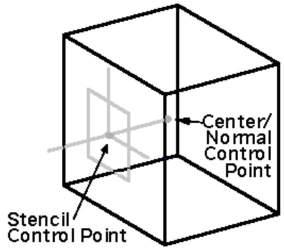
When this box is left unchecked, the wrap mode is set using the Wrap Mode pull-down menu.

One possible application of this is when you want to apply a decal to a fillet. Uncheck auto wrap and lock normal and place the decal over the fillet.

Lock Normal

This check box locks the normal of the decal. When this box is checked, the decal's normal is always perpendicular to the object at the selected surface. When left unchecked, the decal can be placed at any angle with respect to the surface.

By selecting the decal and choosing ***Edit>Show Points***, two control points display, the stencil control point and the normal point.



Select the stencil control point and drag it to a new angle. If you select Lock Normal again and click Apply, the decal snaps back to the normal location.

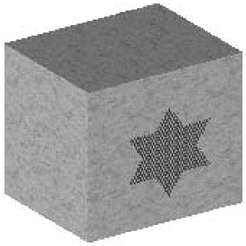
Radius

(Cylindrical and Spherical wrap only) Specify a radius for the center point or the stencil control of the decal, different from the default value listed.

When placing a decal on a face it automatically projects through to the back face. Changing this value such that the radius falls within the body results in the decal appearing only on the face to which it was applied.

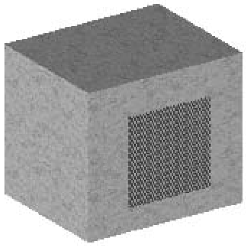
Masking

This menu controls how a decal is applied on an object's surface. You have two options, *Stencil* and *Factor*.



Stencil masking uses an image file to define the regions of the decal artwork that will show on an object's surface. The graphic here uses a star stencil and mesh artwork.

Factor masking is used to put a full image on an object surface. Where stencil masking only places artwork on the object as defined by the stencil, factor masking places the entire artwork image on the object with no cropping. This can be used for placing logos on objects or artwork on walls.



The graphic here shows the same artwork with factor masking instead of stencil masking.

Stencil

(Appears when Stencil Masking is selected) This field displays the name of the image file used. These stencil files are located in the Textures folder within the PhotoRender folder. If you want to use your own stencils in this Designer Elements program, place them in this folder.

When creating your own stencils it is important to know that a stencil is composed of pixels using the RGB color system. The red component (per RGB) of each pixel in the stencil image file is used as the color mix factor. A red value of 0.0 indicates that 0% of the artwork color is used at the pixel location. Thus, that pixel color is determined by the color of the object's material. A red value of 1.0 indicates that 100% of the artwork color is used at the pixel location in place of the color of the object's material. An intermediate value results in a mixture of the artwork color and the underlying material color.

Example: If you create a stencil that is 100% red (RGB), only the artwork color projected onto the object is used. If you lower the red percentage, some of the underlying material color mixes in. If you use black and red in your stencil, those areas that are black (or a 0% red value) use the underlying material color when the artwork is projected. Any color other than black or red in the stencil is ignored.

Typically, a stencil file is composed of fully red and fully black pixels. This provides a clean cropping of decal artwork. However, aged effects can be created using a stencil image file that consists of a mottled red patch on a black background.

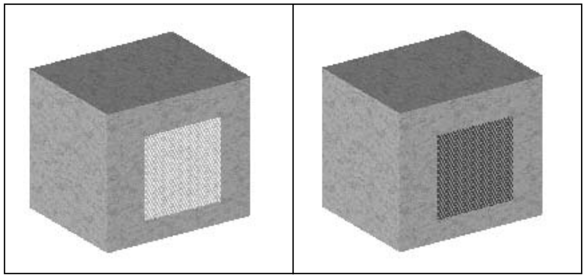
In equation form, this combination translates into: *SurfaceColor=ArtWorkColor*RedVal+MaterialColor*(1.0-RedVal)*.

This formula shows how this Designer Elements program references the stencil file.

Factor slide

(Appears when Factor Masking is selected) This slide allows you to set the value for Factor masking. Factor masking uses a single mix factor to compute the artwork material color combination. This masking is similar to Stencil masking except that there is only one value used for all pixels.

When the slider is set to the far left, representing 0.0, the decal color is set by the object's material. When the slider is set to the far right, representing 1.0, the decal color is used. The left graphic below has a 0.0 factor setting. The right graphic has a 1.0 factor setting.



In equation form the combination translates into: *Surface=ArtWorkColor*MixFactor+Material-Color*(1.0-MixFactor)*.

Logo

This menu sets the source for the decal's artwork. You have two options, Color and Image. In the example here, the DigitCourier2 is used as the decal stencil and a pebble image is used as the artwork.

Color logos use a solid color for the decal artwork. By default the color of the decal stencil is used when placed on an object. You can change the color of the decal by selecting it and choosing a different color in the Pen menu or on the Attributes page of the Edit Object dialog box.

Image logos use an image file for the decal artwork.

Image

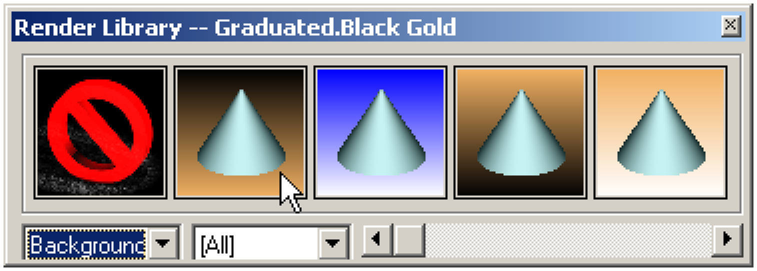
(Appears when Image is selected from the Logo menu.) This pull-down menu lists the images available for decal artwork. These images are located in the Textures folder within the Photo Render folder. If you want to use your own images place them in this folder. See page 36 for more information.

Using the Backgrounds and Foregrounds Libraries

Backgrounds and foregrounds can be used to set the overall scene. You can only use one foreground and one background per rendered drawing.

Backgrounds Library

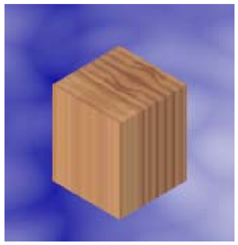
The Backgrounds library contains pre-defined background effects you can apply to your model. Backgrounds control the appearance of those regions of a scene that do not contain objects.



The categories in the Backgrounds library include Images, Misc and Nature.

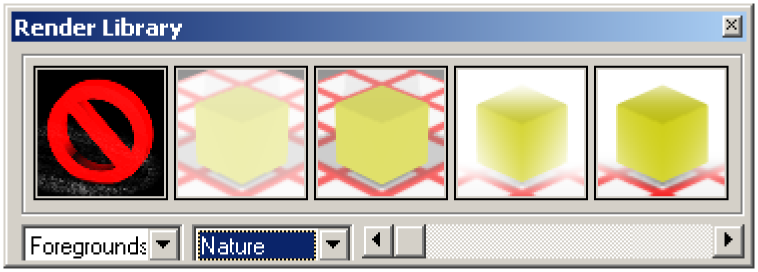
Images	Render images include: VS Logo.
Misc	Render images include: Graduated (Gray Black, Purple White, Red White, White Black, White Gray, White Purple, White Red) and Plain (Black, Gray, White).
Nature	Render images include: Clouds.

The graphic shown is an example of a cloud background.



Foregrounds Library

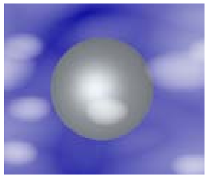
The Foregrounds library contains pre-defined foreground effects you can apply to your model. Foregrounds effect the way the space between the eye point and the scene objects alter the rendering results.



The categories in the Foregrounds library include Misc and Nature.

Misc	Render images include: Depth Cue (Black, Gray and White).
Nature	Render images include: Fog (Heavy, Light), Ground Fog (Deep, Shallow) and Snow (Light, Heavy).

The graphic shown is an example of a snow foreground.



Applying a Background or Foreground

1. Choose ***Window>Render Library*** to display the dialog box.
2. Select the desired foreground or background library.
3. Place the pointer over the desired render image and drag. As the image is dragged over the drawing, the pointer becomes the application symbol.
4. Drag to the drawing area and release the mouse button.

Removing or Replacing Backgrounds and Foregrounds

To remove a background or foreground from a scene drag the None symbol from the Render Library to the object and the material is removed. The No Material symbol is shown here.

You can replace one background or foreground with another by dragging the new image to the scene.

Geometric Characteristics

Foregrounds and Backgrounds have no geometric characteristics therefore are not accessible through the Edit Objects dialog box. However, they can be edited using the Edit Foreground or Edit Background commands in the Render menu. For information, see “Background and Foreground Editing” on page 47.

User-defined Images

You can use your own material and decal images in this Designer Elements program when used with the Edit Objects dialog box. This ability is useful if you have specific materials or decals unique to your company and industry. On possible use is the application of company logos to objects in your drawing.

To prevent distortion all texture images should be square. LightWorks, the rendering engine, maps textures to a 1” square when the scale is set to 1.0.

Creating your own Decals

If you want to apply your own decals you need create your own decal stencils and artwork. Since the artwork projects through the stencil, any bitmap image is acceptable. Create your artwork image using any graphic program that supports bitmaps.

To create a stencil do the following:

1. Create the shape of your stencil.
2. Color your stencil using a percentage of red depending on the desired stencil affect.

   Set the Red value (RGB color) to 255 or 100% red if you only want to project the artwork color onto the object without using the material’s color. Lower the Red value to mix some of the underlying material’s color with the artwork color. Use black in those areas where you only want to use the material’s color when the artwork is projected.

Applying User-defined Material Images

1. Create your own bitmaps images.
2. Place them in the Textures folder located within the PhotoRender folder stored with the program.

   These textures will not appear as an image in the Render Library.
3. Apply a Designer Elements program supplied texture to your object.
4. Double-click on the object to display the Edit Objects dialog box.

5. From the Texture file menu select the image you created.
6. Click Apply and render your scene.

   Your own material is displayed on your object.

Applying User-defined Decal Images

1. Create your own bitmap images, both the decal stencil and the decal artwork.

   Make sure you follow the stencil and artwork formulas as directed earlier in this chapter.
2. Place the stencils and image in the Textures folder located within the PhotoRender folder stored with the program.
3. Double-click on the decal to display the Edit Objects dialog box.
4. From the Stencil menu in Edit Objects select the stencil image you created.
5. In the Logo menu choose the Image option.
6. From the Image menu, select the artwork image you created.
7. Click Apply and render your scene.

   Your own decal is displayed in your scene.

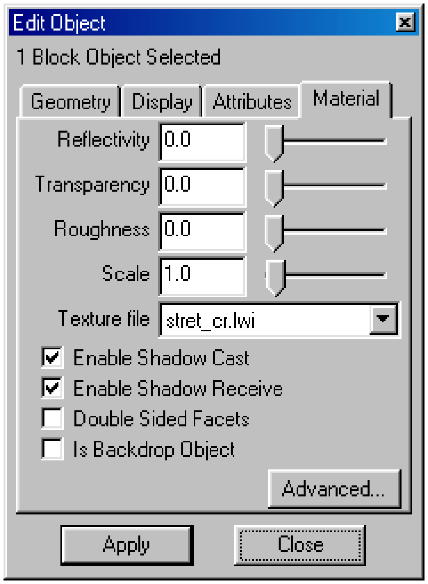
Editing the Rendered Scene

This Designer Elements program now gives you access to Lightworks shader technology. Instead of one shader, you now have five different shaders to render your scene. You can change your object materials or the background and foreground images.

Once you have applied materials to your objects and applied a background or foreground if desired, you can edit them.

Material Editing

There are two levels of editing materials of an object; the first is through the Material page in the Edit Objects dialog box; the second is through the Render Material Settings dialog box accessed through the Material page.



The Material page of the Edit Objects dialog box include the following characteristics or options:

Tip:

When using materials such as polished metal setting the reflectivity to a value less than one may produce a better rendering.

Reflectivity This field sets the material's reflectivity. Values can be entered between zero (0) and one (1). Entering a zero in the field renders a flat finish. Entering a one in the field renders a mirrored finish.

Transparency This field sets the transparency of the material. Values can be entered between zero (0=transparent) and one (1=opaque). This option only works correctly when objects are rendered with ray trace rendering commands.

Roughness This field sets the roughness of the material. Values can be entered between zero (0) and one (1).

Scale

This field sets the scale of the of the material. Typically scale increases the size of the detail. Values can be entered equal to or greater than zero (0).

Texture File

This field displays the selected material. The pull-down menu lists all images in the texture folder. For those materials that do not support textures this menu is not available.

Enable Shadow Cast

This check box allows you to specify if an object casts shadows. This is valuable for reducing the shadows present in complex drawings. When checked the selected object casts shadows.

Enable Shadow Receive

This check box allows you to specify whether an object will receive shadows from other objects. This ability is valuable for reducing the number of shadows present in complex drawings. When checked the selected object receives shadows.

Double Sided Facets

This check box provides you with additional rendering control for surfaces. Objects with normals facing away from the line of sight are not rendered. When checked all objects are rendered, regardless of the normal direction.

Select this option for objects that cause light refraction, like glass. When left unchecked facets on the back side of the glass are ignored resulting in an inaccurate rendering.

Is Backdrop Object

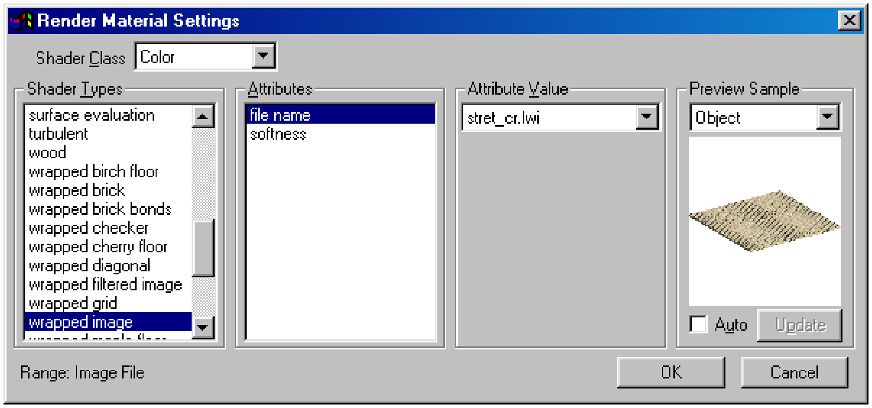
This check box allows you to optimize rendering calculation time for an object, like a wall, that functions only as a backdrop for other objects. Since the object automatically receives light due to its large size, this Designer Elements program does not need to spend much time performing light ray calculations.

Advanced button

Clicking this button brings up the Render Material Settings dialog box. Use this dialog to change the advanced rendering settings used on the selected object. See the next section for a detail description of these settings.

Render Material Settings

When you click the Advanced button on the Material page of the Edit Objects dialog box, the following dialog box appears.



The dialog box includes the following sections and options:

Shader Class This section displays the shader class. Select the class from the pull-down menu. You have five shader classes: Color, Displacement, Reflectance, Transparency and Texture Space. The shader types, their associated attributes and values vary with the shader class. Each shader class works independently of the others. The setting for one shader does not affect another. However, the settings in each class work together to create the final object appearance when it's rendered.

Shader Types This section lists the shader types for the selected class. Select the type from the list. Each set of types is unique to the class. Each type has its own set of attributes.

Attributes This section displays the attributes for the selected type. Select an attribute for the type from those listed. Each attribute has its own set of values.

Attribute Value

This section usually displays the values for the selected attribute. Choose or enter the attribute value. The value range appears at the lower left corner of the dialog box. The values can be numerical, an image file or a color setting. An Edit button displays in this section when a shader attribute is selected. Clicking this button displays a copy the dialog box allowing you to select the shader from the same shader type list as the original dialog box.

Preview Sample

This section displays the *preview window* shape list, *preview window*, the *Auto* option and an Update button.

The *preview window* shape list allows you to set the object shape shown in the window. There are five shapes available from the pull-down menu: cone, cube, cylinder, object and sphere. This feature allows you to see how the same shading characteristics appear on different shapes. Object, which displays the actual shape, is the default shape.

The *preview window* displays the shape based on the shader class, type, attributes and values currently set.

The *Auto* option controls whether the preview image is automatically refreshed when a material setting changes. Checking this box results in an automatic refresh. When checked, the Update button is unavailable.

The Update button controls the manual refresh of the preview window. Click the button to refresh the preview window image after making a material setting change.

Range

This area, at the lower left edge of the dialog box, displays the value range for the selected attribute.

OK	Click this button to close the dialog box and save the settings. The selected object automatically updates to reflect the changes.
Cancel	Click this button to close the dialog box without saving the changes.

After you specify the settings for an object from the desired classes and types, this Designer Elements program combines the settings from all five shaders (Color, Displacement, Reflectance, Transparency, Texture Space) to render the object with one of the photorealistic commands. The settings work together to create the final effect.

Tech Note:

If you enter values in the attribute value field and change to another shade type, those values are not retained. If you want to experiment with various settings, record the values before changing shader types.

Color Class

Use this class to define the object color. The color shader types allow you to specify a color from one color to more complex patterns of color. The shader types are listed alphabetically. For the purposes of this explanation, the types are grouped into following categories:

Curvature	These types use color to show the object's curvature and include: absolute, gaussian, geometric and mean.
Texture	These types are 3D color shaders that give the object a specific appearance. The texture shaders include: birch, blue marble, cherry, chrome, maple, marble, oak, pine, plain, simple wood, solid clouds, solid polka and wood. These shaders are different from image shaders in that they calculate a particular pixel color in 3D space for the object based on the type attributes. Imagine the object carved out of a block of the selected texture. In contrast, an image shader uses the associated 2D bitmap to determine the object color rather than individual pixels.
Evaluation	These types use color to evaluate the selected object and include: draft angle and surface. See the Verify Menu section of Chapter 24 for information of surface and draft evaluation. You can also see

Chapter 21 for surface evaluation and Chapter 23 for draft evaluation.

Wrapped Image These types use an associated 2D bitmap image to determine the object color. The image is wrapped around and mapped to the entire object. These types include: wrapped image and wrapped filtered image. The available images are located in the Textures folder inside the PhotoRender folder.

Wrapped Textures These types are 2D color shaders that are wrapped around the object. Unlike wrapped image shaders, these shaders calculate the shader color based on the attribute values and are not associated with a bitmap image. These shaders include: birch floor, wrapped brick, wrapped checker, wrapped cherry floor, wrapped diagonal, wrapped grid, wrapped maple floor, wrapped oak floor, wrapped pine floor, wrapped polka, wrapped s stripe, wrapped t stripe, wrapped textured brick and wrapped wood floor.

Decal The decal shader type allows you to define the decal texture space, transparency and color.

Turbulent The turbulent shader type creates an agitated or turbulent effect using color and contrast.

All shader attributes are defined in Appendix G.

Displacement Class

Use this class to define the roughness of an object. The displacement shader types allow you specify a variety of roughness patterns. The shader types are listed alphabetically. For the purposes of this explanation, the types are grouped into following categories:

Displacement These types create a displacement calculated in 3D space using the pattern defined by the shader's attributes and include: casting, flat, leather and rough. Like the Color texture shaders, imagine that the object is carved out of a displacement shader block.

Wrapped Displacement These types create a displacement by wrapping the 2D pattern defined by the type attributes around the object. The shaders include: wrapped dimple, wrapped knurl, wrapped leather, wrapped rough and wrapped tread plate.

Wrapped Image These types use an associated 2D bitmap image to determine the displacement and include: wrapped bump map and wrapped old bump map. When using an image for displacement, the shader examines the % color change from pixel to pixel to determine the displacement. For example, an image with black and white stripes might appear as a groove displacement. Gradual color changes in an image result in a more general displacement from pixel to pixel. The available images are located in the Textures folder inside the PhotoRender folder.

None This type creates no displacement.

All shader attributes are defined in Appendix G.

Tech Note:

Remember that by definition applying a reflectance to an object means that the object will have something to reflect. If there is no background on other object in the scene to reflect, when rendered the object may disappear.

Reflectance Class

Use this class to define the object's reflectance. The reflectance shader types allow you specify a variety of reflectance values and patterns defining a surface's finish. If no reflectance is specified for a material, the default Gouraud type finish is applied.

The shader types are listed alphabetically. For the purposes of this explanation, the types are grouped into following categories:

Standard These types are reflectance modelers that provide a particular appearance according to the shader selected. These shaders include: chrome 2D, constant (color), matte finish, metal finish, multilayer paint, phong, plastic, translucency and translucent plastic.

Ray Trace These type modelers are reflectance simulations using ray tracing. They range in accuracy from approximations to physically accurate. These shaders include: conductor (metallic), dielectric (glass), glass (approximation) and mirror,

Wrapped These type reflectance shaders wrap the reflectance image around the object creating the effect specified by the shader. These shaders include: wrapped anisotropic (parallel ridges), wrapped circular anisotropic (circular ridges), wrapped mirror map and wrapped woven anisotropic (woven threads).

Decal This reflectance shader allows you to define the decal and base color reflectance.

All shader attributes are defined in Appendix G.

Transparency Class

Use this class to define the transparency of the object. This shader types allow you to set transparent or opaque nature of an object. These shaders can create transparency effects which range from simple to complex and irregular. The shader types are listed alphabetically. For the purposes of this explanation, the types are grouped into following categories:

Standard These types create a transparent effect based on the shader selected and include: eroded, glow, plain (based on the red component of the color selected where red is transparent and black is opaque) and plain coverage (based on a value).

Wrapped These types create a transparent effect by wrapping the effect around the object and include: wrapped checker, wrapped grid, wrapped mask and wrapped square.

Wrapped Image These types create a transparent effect by wrapping a 2D image around the object and include: wrapped image and wrapped mask. The available images are located in the Textures folder inside the PhotoRender folder.

None This type creates no transparent effect.

All shader attributes are defined in Appendix G.

Texture Space Class

Use this class to define the plane of the texture space or how textures are projected to the object. If no texture space is selected, the texture is projected down the z axis and uses the z plane. The types include the following:

arbitrary plane	This type allows you to specify the vector along which the texture is projected.
auto axis	This type automatically chooses either the x, y or z axis, depending on the normal of the object face.
cylindrical	This type projects the texture image to a cylindrical space using the user-defined origin to set the texture's starting point.
spherical	This type projects the texture image to a spherical space around the radials.
uv	This type projects the texture onto the object using the uv coordinate system.
x plane	This type projects the texture to a constant x plane where the positive z axis is up.
y plane	This type projects the texture to a constant y plane where the positive z axis is up.
z plane	This type projects the texture to a constant z plane where the positive y axis is up.

All shader attributes are defined in Appendix G.

Using the Render Material Settings Dialog Box to Edit Your Object

1. Double click on the object you want to modify to open the Edit Objects dialog box.

   Remember that a material must already have been applied to the object.
2. Select the Material page.
3. Click the Advanced button to display the Render Material Settings dialog box.
4. Select the desired settings for each shader class, as desired.
5. Preview your object using the Update button or the *Auto* option.

6. Click OK when you are satisfied with the settings to close the dialog box and save the settings.

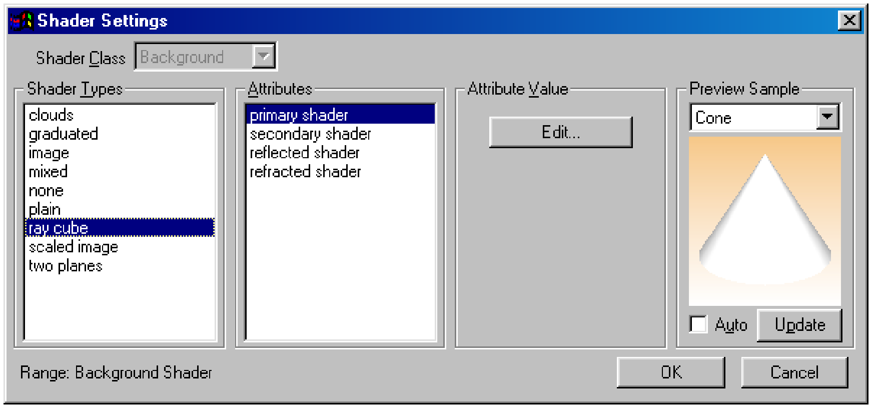
   You can now select another object to edit or use the Eye Dropper tool to apply the same setting from this object to another object. The Eye Dropper tool copies object characteristics such as line font, color, pattern, arrow at start and end and render materials from one object to another. See the Chapter 4 for information on the Eye Dropper tool.

7. Render your scene with one of the photorealistic commands. You objects now appear based on the settings you specified for each object.

   See page 52 for information on using the photorealistic commands.

Background and Foreground Editing

To edit the background or foreground appearance already applied to your scene, choose ***PhotoRender>Edit Background*** or ***Foreground***. (If you did not apply a background or foreground to your scene, no dialog box appears.) The Shader Settings dialog box appears.



This dialog box is almost identical to the one that appears when editing object materials and is divided into the same sections, Shader Class, Shader Types, Attributes, Attribute Value and Preview Sample. See page 40 for a description of each section.

For certain shader types the Attribute value is the Edit button. Clicking the button opens another Shader Settings dialog box that is identical to the first. Select the shader type, attributes and attribute values and click OK. You are returned to the first Shader Settings dialog box.

Background Settings

The Shader Settings dialog box contains only the Background class and its related types, attributes and values. The shader types are listed alphabetically and include:

clouds	This shader type creates a cloudy background.
graduated	This shader type displays a graduated background in the color you specify.
image	This shader type applies a background from an image file. The available images are located in the Textures folder inside the PhotoRender folder.
mixed	This shader type allows you to mix background shaders according to the mixing ratio. Ratios between 0 and .5 favor the base shader. Clicking on the Edit button in the Attribute Value section opens a second Shader Settings dialog box for choosing the shader.
none	No background appears in your scene.
plain	This shader type displays one color background.
ray cube	This shader type displays a background using primary and secondary shaders to deal with refractions and reflections. One shader is used for background areas that are directly visible. The other shader is used for the refraction component. For example, if this Designer Elements program casts a ray that does not intersect any geometry, the primary shader is used. If the ray does intersect geometry the ray is reflected and uses the secondary shader.
scaled image	This shader type applies the background from an image file which is scaled to fill the drawing window. The available images are located in the Textures folder inside the PhotoRender folder.
two planes	This shader type applies a background using a back and front shader to deal with refractions and reflections. One shader is used for background

areas that are directly visible. The other shader is used for the refraction component. For example, if you have a mirrored object and set the front shader to one type and the back shader to another, the shaders are reflected on the object.

Foreground Settings

The Shader Settings dialog box contains only the Foreground class and its related types, attributes and values. The shader types are listed alphabetically and include:

depth cue This shader applies a background color with a linear attenuation of the color between a specified near and far value.

fog The shader applies a gradual fog to the scene based of the color, density and distance specified by the user.

fog light This shader creates a light scattering effect when used with point and spot light sources.

ground fog This shader simulates fog that decreases gradually with altitude.

light scattering This shader creates an atmospheric scattering of light effect more general than that created by the fog light shader.

none No foreground appears in your scene.

scattering medium This shader simulates a dense scattering medium based on various settings including color, shadows, attenuation and density.

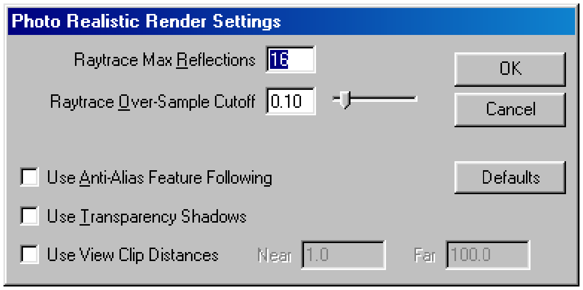
snow This shader simulates the effect of falling snow.

Photorealistic Rendering

This Designer Elements program gives you the ability to photorealistically render your geometry. You can specify rendering settings and choose among various commands before rendering your scene.

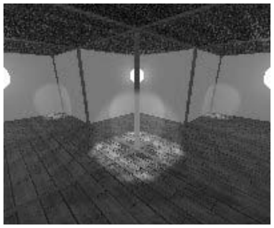
Photorealistic Rendering Settings

You can change the low-level behavior of the rendering engine through the Photo Realistic Render Settings dialog box. Choose ***PhotoRender>Advanced Settings*** to display the dialog box.

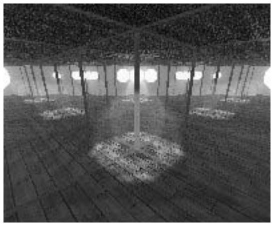


The dialog box contains the following options:

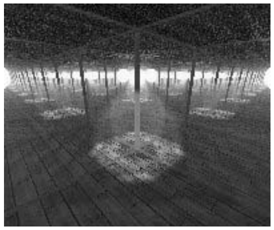
Raytrace Max Reflections This setting controls the maximum number of bounces a ray travels. Once a ray has reached the specified limit no further color calculations occur for that ray. The images of a mirror-walled room, shown here, demonstrate the effect of changes in the value.



Max Reflections = 2



Max Reflections = 6



Max Reflections = 16

Raytrace Over-Sample Cutoff This setting controls the threshold for adaptive image over-sampling. When the rendering operation encounters a significant color change in your image, it will be sampled until the largest of the red, green and blue components for the adjacent color samples do not differ by more than the specified value. The value can be between 0.0 and 1.0.

Enter a value or use the slider to set the sampling limit. The default value is 0.1.

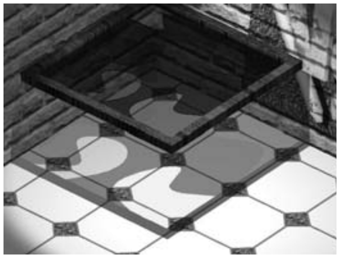
Use Anti-Alias Feature Following

This check box controls the application of Feature Following. When checked, a second pass is made over an image bringing out small geometric feature details that may have been lost due to ray sampling aliasing.

Use Transparency Shadows

This check box controls the behavior of shadow generation for transparent objects. When checked, the transparent object casts a shadow as determined by its shape and color. When left unchecked, the transparent object casts an opaque shadow.

One example of this is a stained glass window. When checked, the window projects color shadows. Left unchecked, the window projects an opaque shadow.



Use View Clip Distances

This check box controls the near and far clipping plane behavior. Near and far planes are normal to the view vector and are at a specified distance from the eye point (see Chapter 30 for information on eye points). Any objects or portions of objects that lie before the near clipping plane and after the far clipping plane are ignored. As implied, an object can be sliced if intersected by a clipping plane.

When checked, the Near and Far data fields become available. Enter the Near and Far distances. The units are based on those set in *Units* page of Preferences.

The graphic here shows an example of a clipped image.

When left unchecked, clipping planes are automatically set to the near and far view extents of the model and the objects in view are rendered. This box is left unchecked by default.

OK Click this button to save the new settings and close the dialog box.

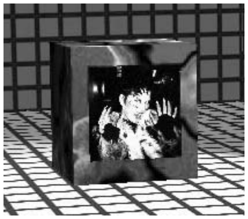
Cancel Click this button to exit the dialog box without saving the settings.

Defaults Click this button to return the settings to the factory state.

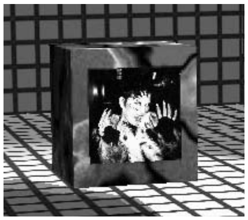
Photorealistic Rendering Commands

There are four render to window commands in this Designer Elements program: *Preview Render (shadows off)*, *Preview Render (shadows on)*, *Raytrace Render (shadows off)* and *Raytrace Render (shadows on, Anti-Alias)*.

Preview Render (shadows off) This command creates a quality rendering of your scene. It renders most rapidly of all the commands since there are no ray tracing and shadow calculations.

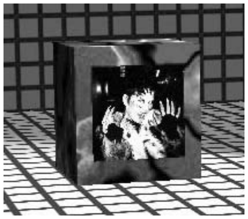


Preview Render (shadows on) This command creates a more realistic rendering than Preview Render (shadows off) with shadows. It renders quickly since there is no ray tracing.



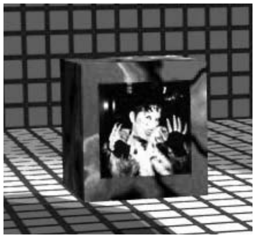
Raytrace Render (shadows off)

This command creates a high quality rendering of your scene. The processing time is extended with addition of the ray tracing operation.



Raytrace Render (shadows on, Anti-Alias)

This command creates the highest quality rendering of your scene, eliminating jagged edges. It uses the accuracy of ray tracing with anti-alias over-sampling. The computational time is significantly longer. The Advanced Setting dialog box controls the sampling calculations.



When using a rendering command any objects with no specified material have a default material/shader applied to them the first time the scene is rendered. The default shader settings are as follows: Color shader - plain, Reflectance shader - Phong, Transparency shader - none and Displacement shader - none.

The *DefaultVSMFile* entry in the *[RenderOptions]* section of the *Render.ini* file controls the default material. If the *DefaultVSMFile* entry is not found, the material specified by the *MiscPlain* material is used.

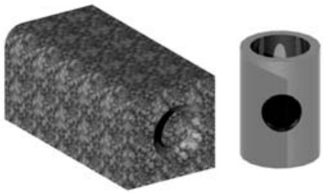
For more information on shader types, see the “Render Material Settings” section on page 40.

Rendering your Geometry

This Designer Elements program allows you to render your entire scene or a specified area.

Rendering your Entire Scene

This Designer Elements program enables you to render your entire scene with one command, showing you how the applied materials appear on your objects. The graphic is rendered using the *Raytrace Render (shadows on, Anti-Alias)*.



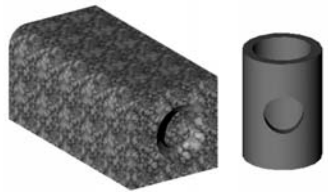
1. Apply materials to the objects in your scene.

2. Choose ***PhotoRender>*** and the command.

   Your scene renders.

Rendering an Area

When applying materials to your objects, you may want to see the results from one specific area of your scene. The area render feature gives you that ability. Using the CTRL (Windows) or OPTION (Macintosh) key, you can specify an area to render. In the graphic only the area around the left object is rendered.



1. Apply materials to objects in your scene.
2. Hold down the CTRL (Windows) or OPTION (Macintosh) key and from the PhotoRender menu, select one of the photorealistic commands.

   The Message Line reads: *Advanced Render: Box area to render.*
3. While still holding down the CTRL (Windows) or OPTION (Macintosh) key, drag a selection fence to define the rendering area.
4. Release the CTRL (Windows) or OPTION (Macintosh) key and the mouse button.

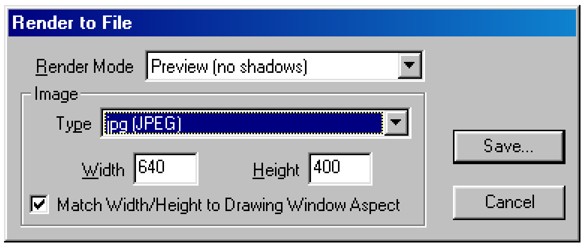
   The area renders.

Tech Note:

This Designer Elements program does not support directly printing a photorealistic image. Use the Render to File command on your image, open the file in a graphics application and print from there.

Render to File

In addition to the commands for rendering your scene to a window, you can render your image to a file. Choose ***PhotoRender>Render to File*** to display the dialog box.



The dialog box contains the following options:

Render Mode — The pull-down list allows you to specify the render mode used in creating the file.

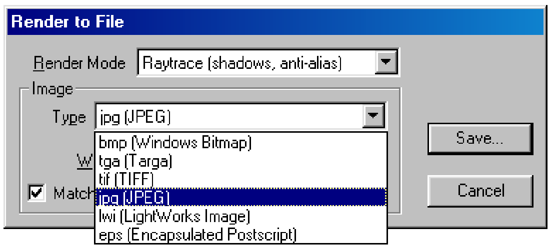
Image Type — This section provides you with output file types. These include eight image format types: Windows bitmap (bmp), Targa (tga), TIFF (tif), JPEG (jpg),

	Lightworks (lwi) and Encapsulated Postscript (eps).
Image Width/Height	These data fields allow you to specify the image size in pixels. If you want a six inch wide image with 300 dpi, enter 6”*300 in the Width data field. These fields function with the Match Width/Height to Drawing Window Aspect check box.
Match Width/Height to (Drawing Window Aspect)	When checked, the image’s width to height ratio equals the drawing window’s width to height ratio. Entering a value in either the Width or Height field automatically enters a corresponding value in the other field. With the box unchecked, you can enter any value
Save	Click this button to save the file and close the dialog box.
Cancel	Click this button to exit the dialog box without creating a file.

Creating Image Files

This Designer Elements program allows you to create publication quality images with higher resolution images than produced on a typical computer monitor.

1. Create your drawing and apply the render materials to your objects.
2. Choose ***PhotoRender>Render to File***.
3. Choose the render mode from the pull-down menu.
4. Choose the image file format you want to create.
5. Enter the desired width and height values. If the Match Width/Height to Drawing Window Aspect box is checked, you only have to enter a value in one field.
6. Click Save to create the file.



Display Last Image Command

Use this command in the Render menu to display your previously rendered scene without having to render your scene again. This command is helpful if a dialog box appears or you inadvertently click your mouse and lose the rendered image. This command saves time if you've rendered your scene with one of the photorealistic commands.

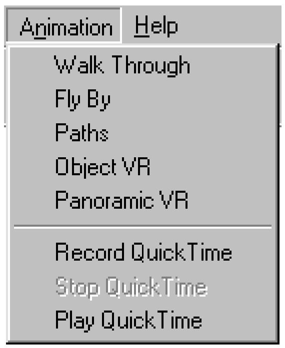
Note: If you rotate your geometry after rendering your scene and then use this command, the image that appears reflects your last rendering and not your current drawing orientation.

Rendering and Multi-processors

If your computer uses multi-processors and you are rendering an image with one of the photorealistic commands, this Designer Elements program will make use of those processors to separate the image into pieces and thus improve the rendering time.

Animation

This Designer Elements program provides the ability to create five types of movies. The animation features are available through the Render menu.



You can create five types of movies. The first three, *Walk Through*, *Fly By* and *Paths*, allow you to specify a path for the camera movement through a scene and require that QuickTime be installed on your Windows or Macintosh computer. The last two, *Object VR* and *Panoramic VR*, allow you to specify the pan or tilt of the camera but the camera path is predefined.

Note: Background images can be used for *Walk Through*, *Fly By* and *Paths* animations. They cannot be used for *Object VR* and *Panoramic VR* animations.

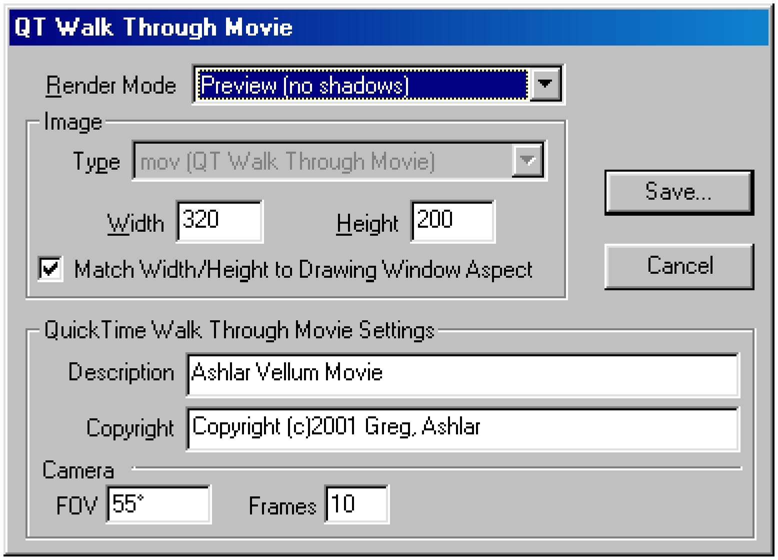
QuickTime Movies with Camera Movement

The *Walk Through*, *Fly By* and *Paths* movies allow you to define the movement of the camera using a curve. As you proceed through the steps to create a movie two dialog boxes appear, the QT Movie dialog box and the Compressor dialog box. These are almost identical for all three of these movie types and allow you to specify the movie settings.

Curve Rules:

- One or more curves can be selected for a path.
- Although path curves should normally be connected, they do not need to be. Disconnected curves will result in a non-continuous jump in the movie.
- Curves cannot be grouped.
- Check the curve direction using ***Verify>Show Curvature*** to ensure that the direction is correct for the desired animation.
- Select the curves in the desired order.

The QT Walk Through Movie dialog box is shown here.

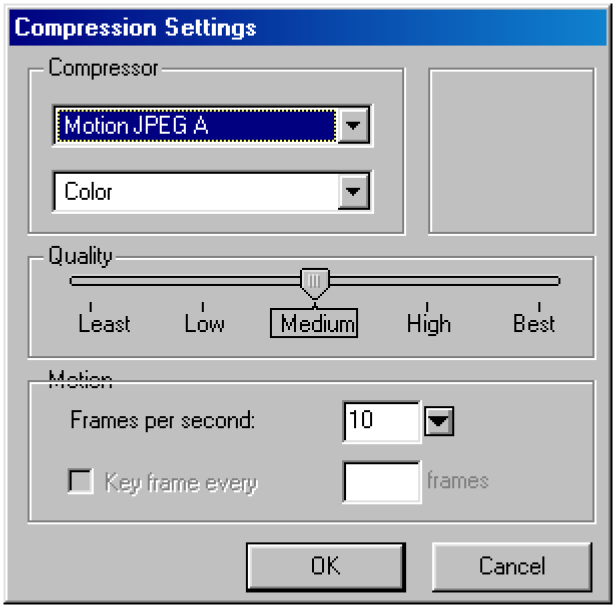


The dialog box includes the following options:

Render Mode The pull-down menu allows you to set the render mode for the movie. You have six options: Preview

	(shadows off), Preview (shadows on), Raytrace (shadows off) and Raytrace (shadows on), Raytrace (Shadows off, Anti-Alias) and Raytrace (Shadows on, Anti-Alias).
Image Type	This field displays the image type as mov (QT Walk Through Movie) and cannot be changed in this dialog box. It is set when you choose the animation type.
Image Width/Height	These data fields allow you to specify the image size in pixels. If you want a six inch wide image with 300 dpi, enter 6”*300 in the Width data field. These fields function with the Match Width/Height to Drawing Window Aspect check box.
Match Width/Height to (Drawing Window Aspect)	When checked, the image’s width to height ratio equals the drawing window’s width to height ratio. Entering a value in either the Width or Height field automatically enters a corresponding value in the other field. With the box unchecked, you can enter any value
Description	This field contains the movie description which displays when the movie is viewed. Enter your movie description.
Copyright	This field contains the movie copyright which displays when the movie is viewed. Enter your copyright information.
Camera	This section contains the FOV (Field of View) angle and the Frames for the movie. The FOV field sets the view angle for the camera. The default angle is 55°. The Frames field sets the number images you want generated for the movie. The default number is 10.
Save	Click this button to save the file and close the dialog box.
Cancel	Click this button to exit the dialog box without creating a file.

The Compressor Settings dialog box, shown here, includes the following options:



Compressor Type — The pull-down menu allows you to set the compressor type. The types include Animation, BMP (Windows only), Cinepak, Component Video, DV - NTSC, DV - PAL, Graphics, H.261, H.263, Intel Indeo Video® 4.4 (Windows only), Intel Video® R3.2 (Macintosh only), Intel Raw (Macintosh only), Motion JPEG A, Motion JPEG B, None, Photo - JPEG, Planar RGB (Windows only), PNG, Sorenson Video, TGA (Windows only), TIFF (Windows only) and Video. The movie quality is affected by the compression setting.

Compressor Color — From the pull-down menu you can choose a color option. The options vary according to the compressor type chosen.

Quality — This slider allows you to set the movie quality.

Motion — This section includes the *Frames per second* data field and the *key frame every x frames* check box/ data field.

Frames per second: Enter the number of frames per second or use the pull-down menu to specify the number.

Key frames every x *frames*: Enter a number in the data field to guarantee that a specific or key frame will be played back during a certain time frame. Playback speed is affected by the computer speed on which the movie is run. If you later want to sync sound to the movie track, a key frame provides some control. For example: if you create a movie set to 20 frames per second and a key frame every 100 frames, a key frame will be marked every 5 seconds. When the movie is played, the movie will skip frames so that every 5 seconds, the key frame is viewed. This allows a sound track to be synced to the movie using the key frame setting.

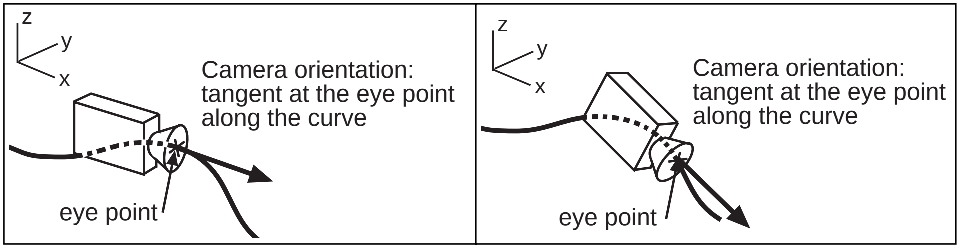
This field only activates for the Animation, Cinepak, Graphics, H.261, H.263, Intel Indeo Video® 4.4 (Windows only), Intel Video® R3.2 (Macintosh only), Intel Raw (Macintosh only), Sorenson Video, TGA (Windows only) and Video compressor types.

OK Click this button to generate the movie

Cancel Click this button to exit the dialog box without creating a movie.

Walk Through Animation

This animation command creates a walk through movie using a curve to define the camera path. The eye point is on the curve. As the camera moves along the curve, its orientation is tangential to and along the curve at the eye point. The graphics illustrate the eye point and camera orientation at two locations along a curve.



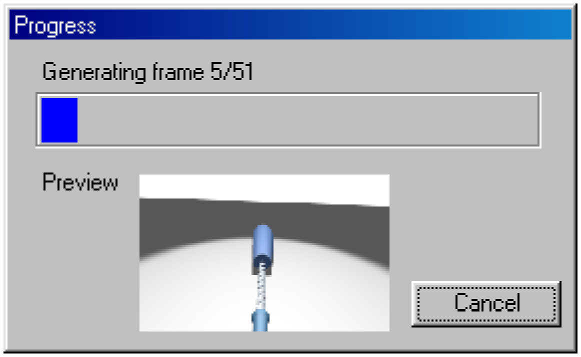
Each movie frame is taken at a particular location on the curve based on the number of frames. If a curve slopes down, the camera follows the tangent and slopes down as well, resulting in the eye looking down at the angle specified by the tangent. The up vector of the camera is always the z axis.

Creating a Walk Through Animation

1. Create a curve along which the camera will travel for the movie.
2. Choose ***Animation>Walk Through***. The Message Line reads: *Walk Through Animation: Pick curve for camera eye path [Shift=Extend].*
3. Select the curve. (Hold down the SHIFT key to select more than one curve.)

   The QT Walk Through Movie dialog box appears.
4. Choose your settings.
5. Click Save. The Save document As dialog box appears.
6. Type in the movie name and navigate to the location where you want to save the file.
7. Click Save. The Compression Settings dialog box appears.
8. Specify your settings for the compressor.

9. Click OK. A progress dialog box appears providing you with a preview movie window and information on the number of frames generated.



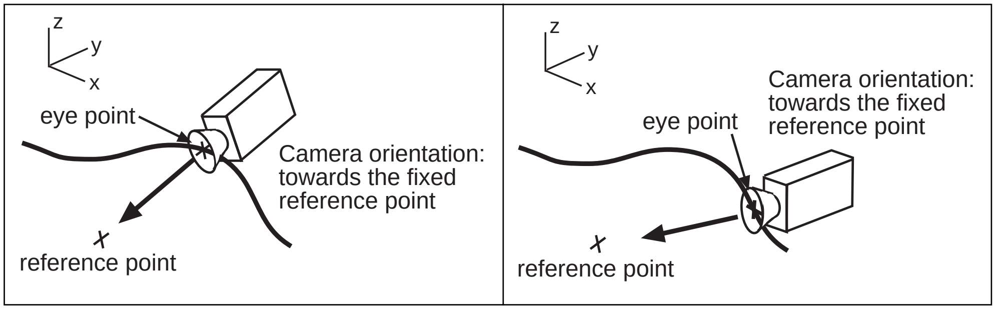
The Message Line provides you with information on the pass, time elapsed and time remaining to generate the movie.

You can click to end the operation before it's complete. This Designer Elements program will finish generating the current frame and write out the movie at that frame. Once the movie is generated you cannot add more frames to it. The dialog box disappears when the movie generation is complete.

10. You can now view the movie using QuickTime player.

Fly By Animation

This animation command creates a fly by movie using a curve to define the camera path. The eye point is on the curve. The camera orientation is directed towards the reference point you specify. The up vector of the camera is always the z axis. The reference point never changes. The point can be in space or on an object. The graphics illustrate the eye point, camera orientation and reference point at two locations.



Creating a Fly By Animation

1. Create a curve along which the camera will travel for the movie.

2. Choose ***Animation>Fly By***. The Message Line reads: *Fly By Animation: Pick curve for camera eye path [Shift=Extend].*
3. Select the curve. (Hold down the SHIFT key to select more than one curve.)

   The Message Line now reads, *Fly By: Pick location for camera reference point.*
4. Pick a reference point location towards which the camera will always be directed.

   The QT Fly By Movie dialog box appears.
5. Choose your settings.
6. Click Save. The Save document As dialog box appears.
7. Type in the movie name and navigate to the location where you want to save the file.
8. Click Save. The Compression Setting dialog box appears.
9. Click OK. A progress dialog box appears providing you with a preview movie window and information on the number of frames generated.

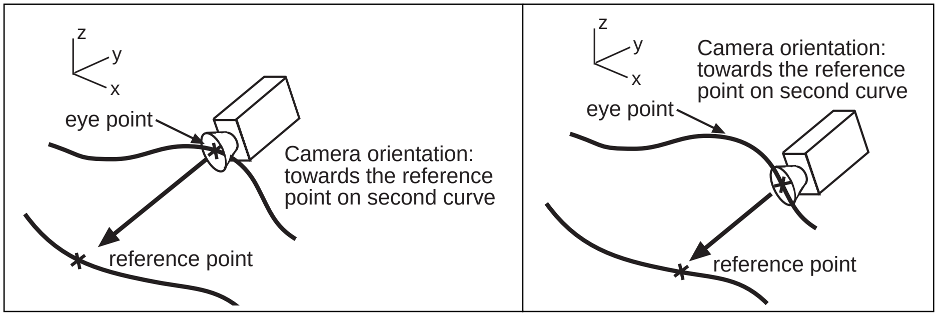
   The Message Line provides you with information on the pass, time elapsed and time remaining to generate the movie.

   You can click to end the operation before it's complete. This Designer Elements program will finish generating the current frame and write out the movie at that frame. Once the movie is generated you cannot add more frames to it. The dialog box disappears when the movie generation is complete.
10. You can now view the movie using QuickTime player.

Paths Animation

This animation command creates a paths movie using a curve to define the camera path and another curve to define the location of the reference point towards which the camera is directed. The eye point is on the first curve. The up vector of the camera is always the z axis.

Each of the curves are divided according to the number of frames specified for the movie. The first frame eye point on the camera path curve corresponds to the first frame reference point on the reference path curve. The graphics illustrate the eye point, camera orientation and reference point at two locations.



Creating a Paths Animation

1. Create two curves, one to define the camera path and the other to define the reference point towards which the camera is directed.
2. Choose ***Animation>Path***. The Message Line reads: *Paths Animation: Pick curve for camera eye path [Shift=Extend].*
3. Select the camera path curve. (Hold down the SHIFT key to select more than one curve.)

   The Message Line now reads, *Paths: Pick curve for camera reference path [Shift=Extend].*
4. Select the reference point curve.

   The QT Paths Movie dialog box appears.
5. Choose your settings.
6. Click Save. The Save document As dialog box appears.
7. Type in the movie name and navigate to the location where you want to save the file.
8. Click Save. The Compression Setting dialog box appears.
9. Click OK. A progress dialog box appears providing you with a preview movie window and information on the number of frames generated.

   The Message Line provides you with information on the pass, time elapsed and time remaining to generate the movie.

   You can click to end the operation before it’s complete. This Designer Elements program will finish generating the current frame and write out the movie at that frame. Once the movie is generated you cannot add more frames to it. The dialog box disappears when the movie generation is complete.

10. You can now view the movie using QuickTime player.

Backgrounds and Movies

You can put in a background for *Walk Through*, *Fly By* and *Paths* animations.

1. Create your background image in a graphic software.
2. Place the image file in the Textures folder within the PhotoRender folder.
3. Display the Render Library.
4. Select the Backgrounds library and the Images category.
5. Apply the VS Logo image to your background.
6. Choose ***PhotoRender>Edit Background***.
7. Select image in the Shader Types list.
8. From the Attribute Value pull-down menu, choose your image.
9. Click OK and render your scene.

QuickTime VR Movies

This Designer Elements program allows you to create to types of VR movies, Object and Panoramic.

QuickTime Object Movie

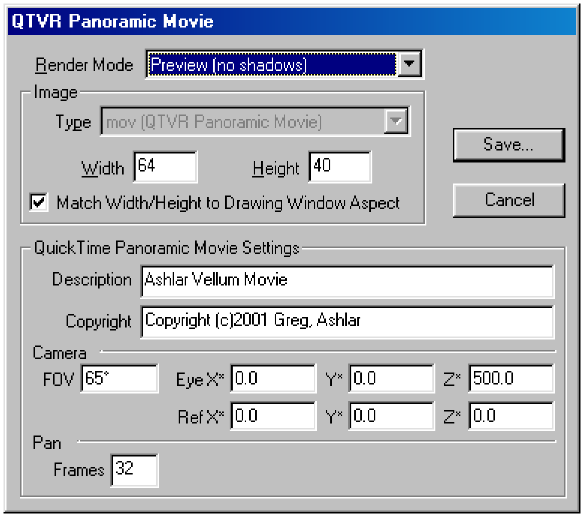
An object movie keeps the observation point fixed as the eye point is moved at a fixed distance about the observation point. This gives the visual effect of moving completely around an object on the surface of an invisible sphere.

The QuickTime movie options produce an interactive movie that can be viewed with a QuickTime player. (A player and browser Plug In can be obtained from the Apple web site.) Each movie is composed of many individual scene images rendered from a slightly different point of view.

Creating an Object Movie

1. Create your drawing and apply the render materials to your objects.
2. Choose ***Animation>Object VR***.

The QTVR Object Movie dialog box appears.



The dialog box includes these additional options:

Description This field contains the movie description which displays when the movie is viewed. Enter your movie description.

Copyright This field contains the movie copyright which displays when the movie is viewed. Enter your copyright information.

Pan This section contains the fields for setting the pan frames and angle, where a zero angle represents the viewer's eye normal to the model as currently displayed on the screen. Since the pan angle is based on the viewer's eye, the current view of the model is irrelevant.

The *Frames* field sets the number of images generated around the sphere equator (latitude).

The *Angle Min* field sets the location of the minimum pan angle. A zero in this field means the pan will include the 0° location in the pan.

The *Max* field sets the largest angle to include in the pan with 360° as the maximum.

The *Start* field sets the starting angle location for the pan. The Start angle must be a value within the range set by the Angle Min and Max fields.

Tilt

This section contains the fields for setting the tilt frames and angle, where zero represents the viewer's eye normal to the model as currently displayed on the screen.

The *Frames* field sets the number of images generated from pole to pole for the movie.

The *Angle Min* field sets the location of the minimum tilt angle. A zero in this field means the pan will include the 0° location. The minimum angle must be between -90° (looking straight up) and 90° (looking straight down).

The *Max* field sets the highest tilt angle with a maximum angle of 90°.

The *Start* field sets the starting angle location for the tilt. The Start angle must be within the range set by the Angle Min and Max fields.

Camera

This section contains the *FOV* (Field of View) angle, the eye (*Eye X*, *Y* and *Z*) and reference (*Ref X*, *Y* and *Z)* point coordinates for the movie.

The *FOV* field sets the view angle for the perspective. The standard angle is 60°. A greater angle produces greater distortion, especially when the eye point is within the scene (like a room).

The *Eye X*, *Y* and *Z* fields set the location for the eye point.

The *Ref X*, *Y* and *Z* fields set the location for the reference point.

The asterisk (*) next to the field indicates your ability to specify locations by clicking in the drawing area. You can also enter the values manually.

3. Enter the desired values.

4. Click Save to create the movie.

QuickTime VR Panoramic Movie

A panoramic movie keeps the eye point fixed as the observation point is rotated 360°. This gives the visual effect of turning 360° in place.

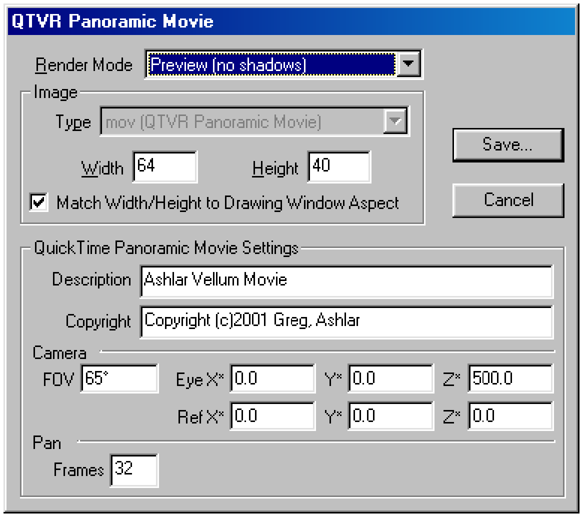
The QuickTime movie options produce an interactive movie that can be viewed with a QuickTime 3.0 player. (A player and browser Plug In can be obtained from the Apple web site.) Each movie is composed of many individual scene images rendered from a slightly different point of view.

Creating a Panoramic Movie

1. Create your drawing and apply the render materials to your objects.

2. Choose ***Animation>Panoramic VR***.

   The QTVR Panoramic Movie dialog box appears.



The dialog box includes these additional options:

Description	This field contains the movie description which displays when the movie is viewed. Enter your movie description.
Copyright	This field contains the movie copyright which displays when the movie is viewed. Enter your copyright information.

Pan The Frames field sets the number images you want generated for the movie. The default number is 32.

Camera This section contains the *FOV* (Field of View) angle, the eye (*Eye X*, *Y* and *Z*) and reference (*Ref X*, *Y* and *Z)* point coordinates for the movie.

The *FOV* field sets the view angle for the perspective. The standard angle is 60°. A greater angle produces greater distortion, especially when the eye point is within the scene (like a room).

The *Eye X*, *Y* and *Z* fields set the location for the eye point.

The *Ref X*, *Y* and *Z* fields set the location for the reference point.

The asterisk (*) next to the field indicates your ability to specify locations by clicking in the drawing area. You can also enter the values manually.

3. Enter the desired description and values.
4. Click Save to create the movie.

Perspective Rendering

This Designer Elements program provides you with the ability to render your model in perspective using the advanced rendering commands. This perspective capability is not available for the basic render modes.

The perspective commands include the *Perspective ON/OFF* check box that is in the modify view dialog box.

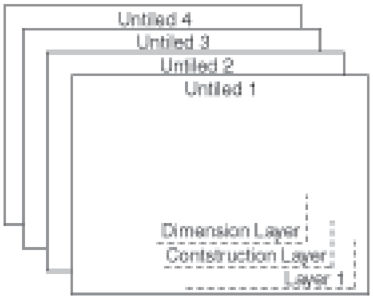
Tech Note:

Perspective settings do not save with a file. Be careful not to have two files open with different settings. If you do the last file overrides the first.

File Management

Whenever you open this Designer Elements program or choose *New* from the File menu, a new document appears. You can open multiple files to work with as desired.

Each file supports the layers feature. This feature adds a great deal of flexibility to your documents. Layers allow you to see various parts of a document individually or as part of the whole. Each document is made up of layers which can be hidden or displayed as needed.



Desinger Elements Files

Consider file organization early. This is particularly important if you are sharing files with other people.

The following topics are covered in this chapter.

- Using Documents, including starting this Designer Elements program, opening files, recent file list and saving

- Files and Preferences
- Files and Locking
- File Size & Deleting
- Uninstalling this Designer Elements program on Windows

Using Documents

A drawing can be a simple part or a complex assembly. A new document opens as *Untitled* and remains untitled until you save it. Saving a drawing is explained later in this chapter.

File Use and Operation System Short Cuts

This Designer Elements program takes advantage of some features of your operating systems when working with files.

Windows

This Designer Elements program allows you to open files and start up the program within the *Windows Explorer* or other folders. It also supports the *Drag and Drop* functionality.

Windows Explorer and other Folders

You can open Designer Elements program files by double-clicking on the file icon in Windows Explorer, the Desktop or another folder. If this Designer Elements program is not running, the program launches first and then opens the file.

If you want to open multiple files, hold down the CTRL key to select the files. Press the right mouse button and select the *Open* command. Each file opens in a separate drawing. If this Designer Elements program is not running, the program launches first, before the files open.

Drag and Drop

This Designer Elements program gives you the ability to manipulate files by dragging and dropping them.

You can drag and drop files into this Designer Elements program to open the files.

- If you drag and drop one or more files into an open drawing, they are merged into the drawing.

- If you drag and drop one or more files into this Designer Elements program when no files are open, each opens individually.
- If you hold down the SHIFT key before dropping multiple files into this Designer Elements program when no files are open, files are merged into a single new drawing.
- You can drag and drop one or more files onto the Designer Elements program desktop icon and the files open individually with the program open.

Macintosh

You have the ability to double-click on a Designer Elements program file to launch the program and open the file.

- If you drag and drop one or more files into an open drawing, they are merged into the drawing.
- If you hold down the SHIFT key before dropping multiple files into this Designer Elements program when no files are open, files are merged into a single new drawing.
- You can drag and drop one or more files onto the Designer Elements program desktop icon and the files open individually with the program open.

File Menu Commands

The File menu contains commands to manipulate documents, including, *New*, *Open*, *Close*, *Save* and *Revert*.

Referral:

Use the *File>Import* command if you want to use drawings of other format types, such as DXF, IGES, or SAT. See Chapter 35 for more information.

New - CTRL+N (Windows); z *+N (Macintosh)*

This command in the File menu creates a new Designer Elements program document. The new document has no name (the title bar shows *Untitled 1*), and is set with the default options, such as pen style or grid display.

If you open more than one new document, the subsequent documents are numbered sequentially until you name them by saving.

Open - CTRL+O (Windows); z *+O (Macintosh)*

This command in the File menu opens an existing Designer Elements program document. Using this command you can also open Graphite files. A progress bar appears as the file is opening. This is especially helpful for large files.

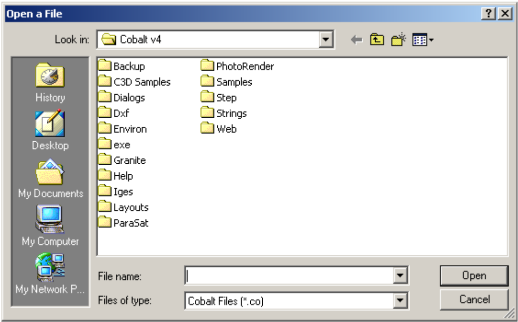
Tech Note:

If you want to open a Designer Elements program file in prior versions of Ashlar-Vellum software, export the file using the appropriate export version. This Designer Elements program embeds ACIS data in its binary files.Vellum Solids 99 and 98 use an earlier ACIS version and ACIS is not backwards compatible.

The document appears in the drawing area maintaining the same settings as the last time you saved it.

The dialog box allows you to specify the document you want and lets you change folders, if necessary.

Windows - If the *Open as read only* box is checked, the file can be opened and printed but not altered. Macintosh - If a file is locked, it can be opened and printed but not altered. You can unlock a file by selecting the file and choosing Get Info (Macintosh standard).



Opening a Document

1.Choose ***File>Open***.

The dialog box appears.

The current folder displays with the files and/or folders it contains.

2.Choose the appropriate folder containing the document you want to open.

3.Click the File name you want to open in the list box.

4. Click OK.

Recent File List

Another way to access files opened recently is through the *Recent File List* that appears in the File menu after the *Exit* command (Windows) or *Quit* command (Macintosh). This list contains the names and paths of the most recent files opened in this Designer Elements program.

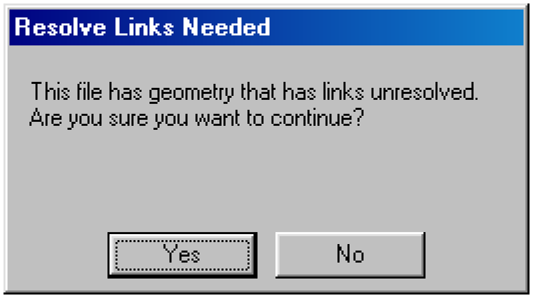
The number of files displayed depends on the number selected in the Filing page of the Preferences dialog box. You can display as many as 20 files in this list.

To open a file from the *Recent File List*, select the file name from the File menu. If the file has been moved since it was last used and the path is no longer accurate, This Designer Elements program will ask you to locate the file by providing the standard Open dialog box.

Saving a Drawing

You can save a drawing by choosing ***File>Save*** or ***Save As***. The file is stored on your computer in the folder you specify.

If you attempt to save a file containing links that are not resolved, the following warning appears:



Save - CTRL+S (Windows); z +S (Macintosh)

This command in the File menu saves the current Designer Elements program document to its original folder. If you want to save it to a different folder or with a different name, choose *Save As*. If you choose *Save* and the document has not been saved previously, the Save As dialog box appears automatically, allowing you to name the document and specify the folder in which to save it.

If you don't make a geometrical change to any object in your file, the *Save* command is not available. Changing an object's layer is not recognized as a geometrical change. If you have named and saved the document before, a brief message appears when you choose *Save*. The program pauses while it updates the information.

Note: You should save frequently. It is important to save before performing any intricate, multistep procedure. That way, if the result is not exactly what you had hoped, you can abandon the file by closing it without saving.

Tech Note:

If you try to save a file with the name of a locked file, a warning message will appear.

If you delete objects before saving a file, the data for that information may still be retained with the file. Perform the delete operation again to guarantee the data is removed.

Save As

This command in the File menu saves the current document. A dialog box appears so you can name the current document, give it a different name, or save it to a different directory. You can use the *Save As* command to make a backup of a document.

(Windows) You also have the ability of saving a file as a Read only file. Click the check box for that option.

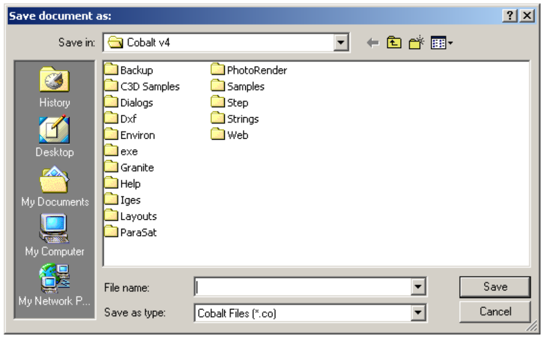
Saving in the Current Directory

1. Choose ***File>Save As***.

   The Save document as dialog box appears.

2. If necessary, display a different folder.
3. Type the name you want to use in the File name box.
4. Click OK.

   The filename extension *.vs* is automatically appended to the filename.



Tech Note:

Macintosh users: Although the .co, .xe, ar, .ne and *.vs* extension is not required for the file name, it is included to ease file transfer between Macintosh and Windows.

Making a Backup

You should make a backup of your work in case you make many changes and want to go back to the original version. You can choose ***File>Save As*** and save the file with another name.

Auto Save

This command is found under ***File>Preferences>Filing*** and directs this Designer Elements program to save a backup copy of your work periodically. If your computer hangs up for any reason, the work you did up to the last Auto Save will be recoverable.

You have the option to save after a certain number of commands or minutes. You can also specify the number of backup files created before reusing a backup file name. (See Chapter 6 for more information on setting up or using the Auto Save feature.) For Windows, files opened with the READ ONLY attribute are also autosaved.

Files saved using this feature are placed in the Backup folder within the Designer Elements program folder.

If *Auto Save* is set to on in Preferences but you have not modified your geometry since the last autosave, the function does not activate.

Revert

This command in the File menu deletes all changes made in the drawing since the file was last saved.

1. Choose ***File>Revert***. The following dialog box appears.
2. Click OK to return to the original file opened and close the dialog box. (Click Cancel to close the dialog box without enabling the command.)

Close - CTRL+F4 (Windows);z +F4 (Macintosh)

This command in the File menu closes the current Designer Elements program document. If other Designer Elements program documents are open, they remain open when you close the current document. If you have made any changes since you last saved the current document, a dialog box allows you to save the changes. You can close the document with or without saving the changes.

For Windows, you can also close the document by double-clicking the Control menu at the upper left corner of the title bar. For Macintosh, click the Close button in the upper left corner.

Exit - CTRL+Q (Windows); Quit - z +Q (Macintosh)

This command in the File menu closes this Designer Elements program. If you have made changes since you last saved, a dialog box allows you to save the changes. If you have more than one document open, an alert message allows you to save unsaved documents.

Files and Preferences

In the *Filing* page of the Preferences dialog box you can choose settings for your this Designer Elements program files. These include options for saving native picture formats, clearing undo, compacting files, read-only network file sharing, recent files and auto save.

Files and Locking

Both Windows and Macintosh operating systems provide you with the ability to lock files. Once locked these files can be opened and printed but not altered until unlocked. This feature is valuable when working on a network. When one person is working on the file, another person can open the file but is unable to make changes. See the User Guide for your system for more information.

Tip:

If your system crashes with an open file, the file will lock. Unlock the file according to your system instructions. You can also save the file under the same name. As you save you will be prompted to confirm the overwriting of the locked file.

File Size & Deleting

When you delete objects in this Designer Elements program, the object data is still retained until a second delete operation is performed. If your notice that your file size is large in comparison to the amount of geometry in the file, this may be the reason. Perform another delete operation (create a line and delete it), save the file and reopen it.

Uninstalling This Program on Windows

When you uninstall this Designer Elements program, the *Prefs.ini* file does not automatically uninstall since it was created after installation. This file is located within the Designer Elements program folder in the Environ folder.

Importing & Exporting

There are various reasons to use the import and export functions. You might want to bring a document in from another application to work on in this Designer Elements program. You might need to save a document in a format other than the this Designer Elements program format for use with another application. This Designer Elements program offers you a wide variety of options to accomplish this.

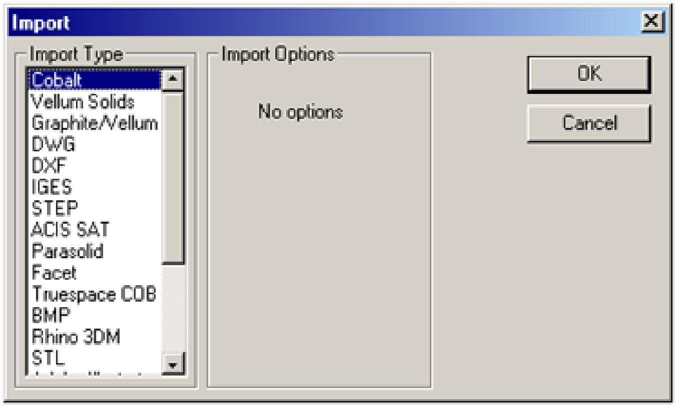
The following topics are covered in this chapter:

- Importing
- Exporting

Importing

This Designer Elements program allows you to import many different kinds of files. Some types include import options specific to the format.

When you choose the *Import* command, the following dialog box appears.



The dialog box contains these options:

Import Type	Lists the file formats that this Designer Elements program imports.
Import Options	Includes the options that are available for the selected format.
OK	Click this button to close the import dialog box and display the standard Open dialog box for locating the file.
Cancel	Click this button to close the dialog box and ends the operation.

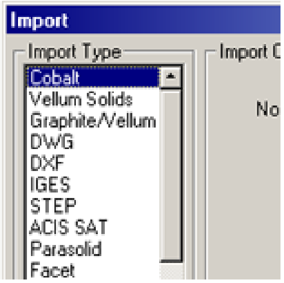
Supported Import Formats

This Designer Elements program imports these file formats: Cobalt, Vellum Solids, Graphite/Vellum 3D, DWG, DXF, IGES, STEP, ACIS SAT, Parasolid (Windows only), Facet, Truespace COB, BMP, Rhino 3DM, STL, Adobe Illustrator, 3D Studio, Catia v4, Spline, Text, Grid Surface, ProE/Granite (Windows only) and PICT (Macintosh only).

Vellum Solids

Selecting this type imports files created in Vellum Solids. There are no options for this import type. In order to read previous versions of Vellum Solids into this Designer Elements program, you must import the file using this format.

Graphite/Vellum



Selecting this type imports files created in Graphite/Vellum 3D. Graphite/Vellum 3D includes three options: *Vellum Layers*, *Auto Heal Bodies* and *Feature Recognition*.

This translator imports horizontal, vertical, diameter and radial dimensions in addition to geometry.

Vellum Layers

This section includes check boxes, *Create*, *Create Empty* and *Display All*.

Create - When checked, This Designer Elements program creates/imports the layers in the Vellum 3D file and places geometry on their respective layers. If this box is not checked, all geometry is placed on the work layer.

Create Empty - When checked, this Designer Elements program creates layers that contain no geometry. If this box is not checked, layers that contain no geometry are deleted.

Display All - When checked, this Designer Elements program displays all objects on all layers. If this box is not checked, the import file determines which layers display.

Auto Heal Bodies

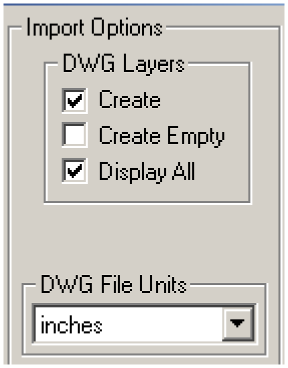
When checked, this Designer Elements program applies auto-healing to imported surfaces. Auto-healing attempts to find collections of surfaces that define closed volumes and convert them to solids. With this box checked, the *Feature Recognition* check box is enabled.

If this box is not checked, auto-healing is not be performed and the *Feature Recognition* check box is disabled.

Graphite/Vellum 3D Notations

- The Vellum 3D import translator does not support the following entities: work plane, fill, bitmap, balloon, hatch, sheet and view.

- ACIS does not support skewed ellipses. Only ellipses created with the 2-Point Center Ellipse tool and the Opposite-Corner Ellipse tool in Vellum 3D can be imported in this Designer Elements program.
- This Designer Elements program does not support models (model space). When you import a file with geometry in different models, each model is placed on its own layer.
- When you import a file into this Designer Elements program and you want to surface them to intersect with other objects or project curves onto them, use the Cover, Skin or Net Surface tools.



DWG

Selecting this type imports DWG files created in AutoCAD or other programs that support the DWG file format. The translator will read DWG files up to and including version R2000.

This translator imports horizontal, vertical, diameter and radial dimensions in addition to geometry.

DWG includes three options: *DWG Layers*, *Feature Recognition* and *DWG Units*.

DWG Layers

This section includes check boxes, *Create*, *Create Empty* and *Display All*.

Create - When checked, this Designer Elements program creates/imports the layers in the DWG file and places geometry on their respective layers. If this box is not checked, all geometry are placed on the work layer.

Create Empty - When checked, this Designer Elements program creates layers that contain no geometry. If this box is not checked, layers that contain no geometry are deleted.

Display All - When checked, this Designer Elements program displays all objects on all layers. If left unchecked, the import file determines which layers display.

DWG File Units This list sets the units for the incoming file. The units include inches, feet, mm, cm and meters.

DWG Notations

The DXF/DWG translator does not support the following entities: Shape, Ole2frame, MLine, Leader, MText, ViewPort, Tolerance, Proxy, Hatch and Image.

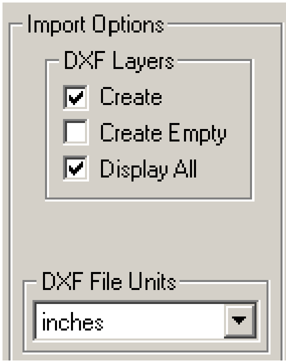
If you import a DWG file that contains entities that are not supported, a warning box will appear telling you that the entities were not read.

See Appendix C for supported DWG entities.

DXF

Selecting this type imports DXF (AutoCAD's Data eXchange Format) files. The translator will read DXF files up to and including version R2000.

This translator imports horizontal, vertical, diameter and radial dimensions in addition to geometry.



DXF includes three options: *DXF Layers*, *Feature Recognition* and *DXF Units*.

DXF Layers This section includes check boxes, *Create*, *Create Empty* and *Display All*.

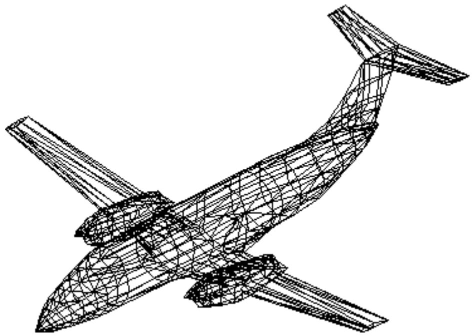
Create - When checked, this Designer Elements program creates/imports the layers in the DXF file and place geometry on their respective layers. If this box is not checked, all geometry is placed on the work layer.

Create Empty - When checked, this Designer Elements program creates layers that contain no geometry. If this box is not checked, layers that contain no geometry are deleted.

Display All - When checked, this Designer Elements program displays all objects on all layers. If this box is not checked, the import file determines which layers display.

DXF File Units

This pull-down list sets the units for the incoming file. The units include inches, feet, mm, cm and meters.



DXF Facet Files

In DXF and DWG files created prior to AutoCAD Release 13, surfaces were represented as a collection of facets. This Designer Elements program converts these surfaces to 3-Point mesh elements. This graphic shows an example of a DXF facet file that was imported into this Designer Elements program.

DXF Notations

The DXF translator does not support the following entities: Shape, Ole2frame, MLine, Leader, MText, ViewPort, Tolerance, Proxy, Hatch and Image.

If you import a DXF file that contains entities that are not supported, a warning box will appear telling you that the entities were not read.

See Appendix C for supported DXF entities.

IGES

Selecting this type imports IGES files created by various CAD programs. IGES includes three options: *Flavor, Auto Heal Bodies* and *Feature Recognition*.

This translator also supports importing horizontal, vertical, diameter and radial dimensions.

Flavor

This section includes a pull-down menu allowing you to choose to import three different IGES flavors: *Generic*, *AutoCAD* or *SolidWorks*. Use *Generic* if your specific IGES flavor is not listed.

Auto Heal Bodies

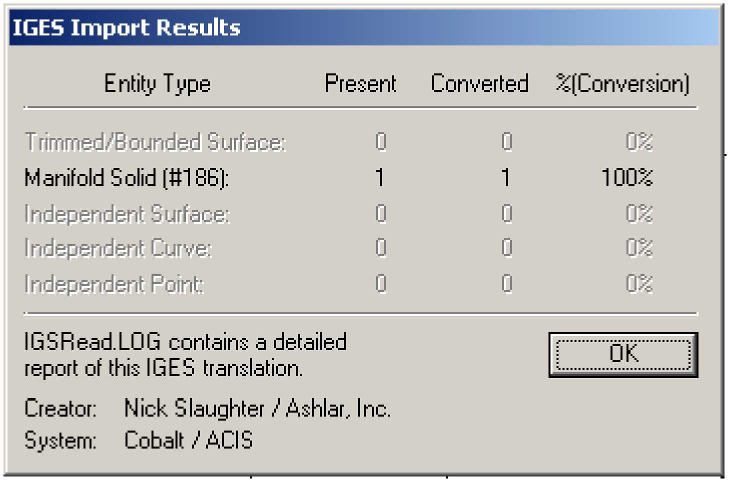
When checked, this Designer Elements program box applies auto-healing to imported surfaces.

Auto-healing attempts to find collections of surfaces that define closed volumes and convert them to solids. With this box checked, the *Feature Recognition* check box is enabled.

If this box is not checked, auto-healing is not performed and the *Feature Recognition* check box is disabled.

See Appendix D for supported IGES entities.

When you import an IGES file, an IGES Import box appears with the result of your import.



The box contains the following items:

Entity Type — This column lists five entity types: Trimmed/Bounded Surface, Manifold Solid (#186), Independent Surface, Independent Curve and Independent Point.

Present — This section displays the number of a particular entity in the file.

Converted — This section displays the number of entities for the type that converted.

%(Conversion) — This column displays the percentage conversion of the particular entity.

Creator Information This section contains the name of the person who created the IGES file and system information.

ACIS SAT

Selecting this type imports SAT files created by various ACIS-based CAD programs. There are no options for this type.

Facet

Selecting this type imports ASCII Facet files created by various CAD programs. There are no options for this import type.

The facet file format was created by the United States Electromagnetic Code Consortium and Lockheed Fort Worth. This file format provides an efficient means for transferring geometry models to a variety of government-sponsored signature predication codes. The format includes geometry definitions of 3- and 4-sided facets and material IDs for each facet.

F-15 Model Courtesy Dr. Shung-Wu Lee, DEMACO

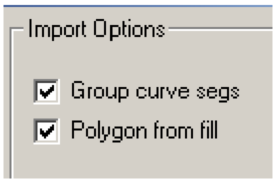
The graphic shows a facet file after it was imported into this Designer Elements program. Each shade represents a material to be analyzed by the signature predication code. The canopy and raydome are transparent, so they are not included in the signature model.

PICT (Macintosh only)

Selecting this type imports Pict files created by various programs. Pict is the Macintosh native file format. There are no options for this import type.

Adobe Illustrator

Selecting this type imports Adobe Illustrator version 5.0 through 8.0 files. Adobe Illustrator includes two options: *Group curve segs* and *Polygon from fill*.



Group curve segs This option sets how curves are imported. Illustrator creates curve segments. When selected, separate segments are grouped into one curve. When this option is not selected, curves, like a circle are imported into this Designer Elements program as four arcs.

Polygon from fill — This option sets whether fill is imported with curves if the objects contain fill. When selected, the fill and curves are imported. When this option is not selected, fill is not imported.

This translator also supports importing text.

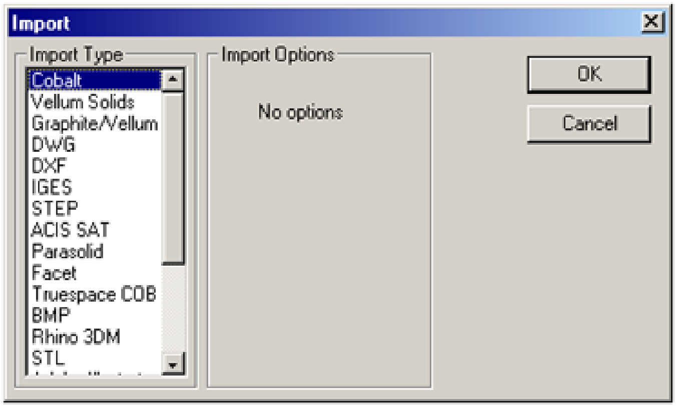
Spline

Selecting this type imports ASCII Spline files created by various CAD programs. There are no options for this import type. See Appendix F for information on how to create spline file for import using a text editor.

Import Command

This command in the File menu imports a document and places it in the current file.

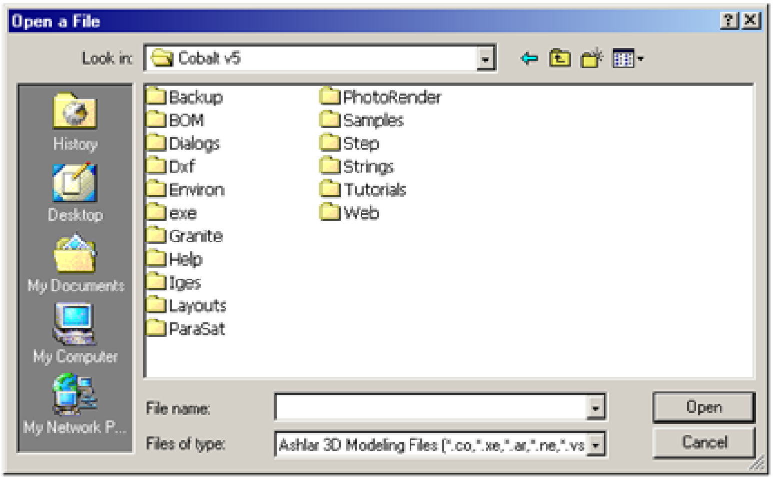
1. Choose ***File>Import***.



The Import dialog box appears with all the import options.

2. Select the Import type from the list and the options for the type as desired.

3. Click OK. The standard Open dialog box appears.



4. Navigate to the file you want to import.

   (Windows users: If the *Open as read only* box is checked, the original file cannot be edited but the imported geometry can be.)

5. Click Open to import the file.

The file appears in the drawing area. If you want to save the file in its original format after editing, you must choose the *Export* command from the File menu.

Importing Splines

When you import a text file that contains the coordinates of a spline, Vellum will create the spline according to the imported coordinates.

1. Select ***File>Import***.

   The Import dialog box appears.

2. Specify the import option Spline.
3. Click OK. This Designer Elements program displays the standard Open dialog box.
4. Select a text file that contains the coordinates for the spline.
5. Click Open to import the file.

   This Designer Elements program begins creating the spline.

See Appendix F for information on how to create a text file for importing a spline.

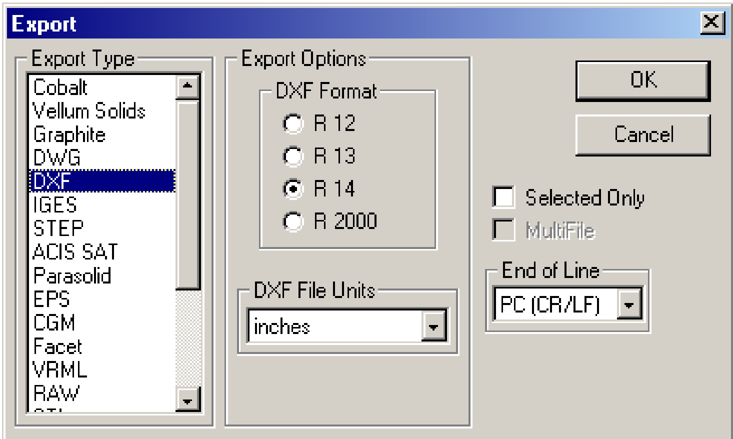
Importing Tips and Notations

These tips and notations will help you successfully import files.

- **Groups** - This Designer Elements program can import grouped geometry and groups within groups.
- **Layers and Color** - Be careful importing geometry into a layer with a color override. Objects placed on that layer will be displayed in the color of the layer.
- **ACIS Data** - Surfaces and solids data is written out in DXF only.
- **Smart Walls** - This Designer Elements program does not support the Smart Walls feature of Graphite and prior versions of Vellum 3D. Smart walls imported into this Designer Elements program are converted into individual lines without attributes such as height and thickness.
- If this Designer Elements program displays geometry that was not in the original AutoCAD DXF file, go back to the original file in AutoCAD. Choose the PURGE command and purge any unnecessary blocks in the file. Then export the DXF file and import it into this Designer Elements program.
- If an error occurs when you try to import a file, this Designer Elements program creates a file log in the Designer Elements program folder. For example, if you try to import a file through the IGES translator and an error occurs, the file IGSRead is created.
- Importing an IGES, Vellum 3D, SAT, DXF or DWG file with the *Feature Recognition* checked - There may be times when the Feature recognition operation returns a successful code but no geometry displays. This gives the impression that no geometry was imported when in fact *Feature Recognition* failed and discarded the geometry. Import the file with *Feature Recognition* off and the geometry will successfully import.

Exporting

This Designer Elements program allows you to export to many different kinds of files formats. Some types include export options specific to the format. When you choose the *Export* command, the dialog box appears.



The dialog box contains these elements:

Export Type — Lists the file formats that this Designer Elements program exports.

Export Options — Includes the options that are available for the selected format.

Selected Only — With this option check marked, only selected geometry is exported.

MultiFile — This option is useful when exporting files to products that do not have the ability to handle multiple objects within one file such as Pro/E and SolidWorks. This option will create a file for each object in your drawing, its name will be the same as in the Design Explorer.

End of Line — This menu allows you to choose an end of line structure for your exported file. You have three options: Mac (LF), PC (CR/LF) and Unix (CR).

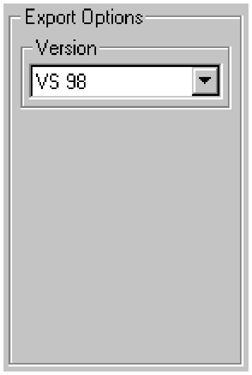
OK Click this button to close the Export dialog box and displays the standard Save as dialog box.

Cancel Click this button to close the dialog box and end the operation.

Supported Export Formats

This Designer Elements program exports these file formats: Cobalt,Vellum Solids, Graphite, DWG, DXF, IGES, STEP, ACIS SAT, Parasolid, EPS, CGM, Facet, VRML, RAW, STL, Adobe Illustrator, Catia v4, Text, ProE/Granite, Macrmedia, Viewpoint Media and PICT (Macintosh only).

Vellum Solids



Selecting this type exports files as a Vellum Solids file. You have four options: VS 2000, VS 2000 SP1, VS 99 SP1, VS99 and VS 98.

VS 2000 This format exports Vellum Solids 2000 files. Use this format when exporting from a Vellum Solids 2000 service pack to be read into Vellum Solids 2000.

VS2000 SP1 This format exports Vellum Solids 2000 Service Pack 1 files.

VS 98 This exports the Vellum Solids 2000 file as a Vellum Solids 98 file.

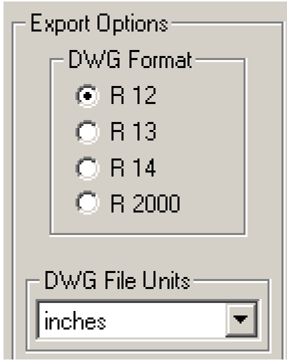
VS 99 This format exports Vellum Solids 99 files.

VS 99 SP1 This format exports Vellum Solids 99 Service Pack 1 files.

Graphite/Vellum 3D

Selecting this type exports files as a Vellum 3D file. This translator exports text and horizontal, vertical, diameter and radial dimensions in addition to geometry. Angle center mark, ordinate and balloon dimensions are exploded into lines and text. There are no options for this type.

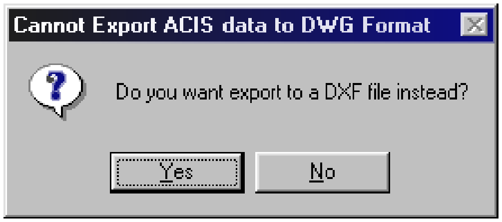
DWG



Selecting this type exports DWG files compatible with AutoCAD and other programs that support the DWG file format. (DWG is the binary version of DXF.) DWG includes four format options, *Version 12*, *Version 13*, *Version 14* and *Ver. 2000*.

This translator exports all dimension types: horizontal, vertical, diameter, radial, ordinate, angled, center mark and balloon dimensions. It also exports groups and bezier and vector splines. Hatching is converted into lines.

Warning: Layer names are limited to the following character set: ‘a’ through ‘z’, ‘A’ through ‘Z’, ‘0’ through ‘9’, ‘-’ and ‘_’. All other characters in a layer name convert into an underscore (_).



ACIS data cannot be exported using the DWG translator. If you attempt to use this translator on a file containing ACIS data, a warning dialog box appears asking if you want to export the file using the DXF translator.

Click Yes to create a DXF file or No to end the operation.

Version 12 This option exports the file as an R12 file. This does not support ACIS data. Ellipses, conics, splines are converted into polylines. ACIS curves are converted into b-splines. Surfaces and solids are converted into facets (Face3D).

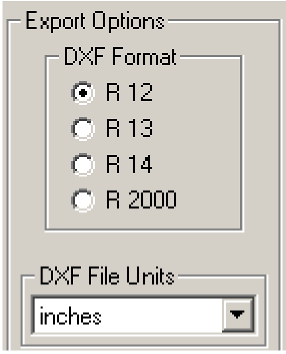
Version 13 This option exports the file as an R13 file. Ellipses, splines and ACIS curves are supported. Conics are converted into polylines.

Version 14 This option exports the file as an R14 file. Ellipses, splines and ACIS curves are supported. Conics are converted into polylines.

Version 2000 This option exports the file as an R2000 file.

Choose the DWG option based on the translator version supported by the program into which you want to import your file.

DXF



Selecting this type exports DXF files compatible with AutoCAD. DXF includes four format options: *Version 12*, *Version 13*, *Version 14* and *Ver. 2000*.

Choose the end of line structure (Mac (LF), PC (CR/LF) or Unix (CR)) from the End of Line pull-down menu.

This translator exports all dimension types: horizontal, vertical, diameter, radial, ordinate, angled, center mark and balloon dimensions. It also exports groups and bezier and vector splines. Hatching is converted into lines.

Warning: Layer names are limited to the following character set: ‘a’ through ‘z’, ‘A’ through ‘Z’, ‘0’ through ‘9’, ‘-’ and ‘_’. All other characters in a layer name convert into an underscore (_).

Version 12 This option exports the file as an R12 file. This does not support ACIS data. Ellipses, conics, splines are converted into polylines. ACIS curves are converted into b-splines. Surfaces and solids are converted into facets (Face3D).

Version 13 This option exports the file as an R13 file. Ellipses, splines and ACIS curves are supported. Conics are converted into polylines.

ACIS data for surfaces and solids are written out as SAT data (Spatial Technologies format). A program that supports these versions does not automatically support SAT data. Check your program manual or with the manufacturer to determine whether it can read SAT data. Geometry exported using these versions is considered more accurate than facet representation.

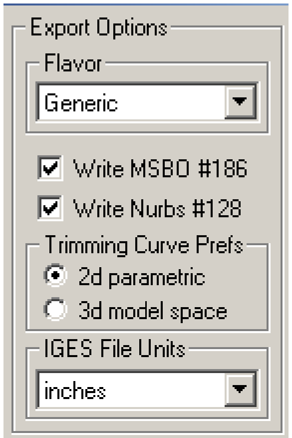
Version 14 — This option exports the file as an R14 file. Ellipses, splines and ACIS curves are supported. Conics are converted into polylines.

ACIS data for surfaces and solids are written out as SAT data (Spatial Technologies format). A program that supports these versions does not automatically support SAT data. Check your program manual or with the manufacturer to determine whether it can read SAT data. Geometry exported using these versions is considered more accurate than facet representation.

Version 2000 — This option exports the file as an R2000 file.

Choose the DXF option based on the translator version supported by the program into which you want to import your file and the end of line setting.

IGES



Selecting this type, exports various versions of IGES files. IGES includes four format options: *Flavor, Write MSBO #186*, *Write Nurbs #128* and *Trimming Curve Prefs*.

Choose the end of line structure (Mac (LF), PC (CR/LF) or Unix (CR)) from the End of Line pull-down menu.

This format also exports groups.

Flavor — This section includes a pull-down menu allowing you to export different IGES flavors: *Generic*, *AutoCAD*, *SolidWorks*, *Vellum v3.0*, *Vellum v2.7*, *Pro/E* or *Alias*.

AutoCAD, R13: Certain MSBOs are not supported by AutoCAD. This flavor converts the MSBOs so they can be read in AutoCAD.

SolidWorks: This flavor does not support IGES Conic Arc (#104) which this Designer Elements program uses to write an ellipse. Ellipses convert into nurb splines.

Vellum v3.0: All solids convert into surfaces.

Vellum v2.7: All solids and surfaces convert into curves.

Pro/E: Does not include the Trimming Curve Prefs options.

Alias: Includes all the options listed for the Generic flavor.

Use Generic if your specific IGES flavor is not listed.

Write MSBO #186 Check this box to export solids using this IGES5 solid object type. (MSBO #186 is a Manifold Solid B-Rep entity.) If this box is not checked, only 3D parametric trimming curves for analytic surfaces are exported.

Write Nurbs #128 Check this box to export solids using this nurb surface type.

Trimming Curve Prefs This section includes two check boxes *2D Parametric* and *3D Model Space*. Choose one option.

2D Parametric - Check this option to export 2D parametric trimming curves for analytic surfaces. This option is valuable to programs that can read only 2D data and need the 3D data mapped to a 2D parametric.

3D Model Space - Check this option to export the actual 3D trim curve in the model space.

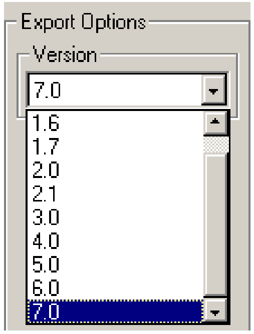
See Appendix D for the supported IGES entities.

STEP

Selecting this type exports a STEP file. STEP (STandard for the Exchange of Product model data) is a neutral file format used to export models among CAD, CAM and CAE applications.

You can also choose the end of line structure (Mac (LF), PC (CR/LF) or Unix (CR)) for your file from the End of Line pull-down menu. There are no other options for this export type.

ACIS SAT



Selecting this type exports SAT files compatible with various versions of ACIS. This format includes one option: *Version*.

Version — The option sets the ACIS export version. The pull-down menu includes: 1.5, 1.6, 1.7, 2.0, 2.1, 3.0, 4.0, 5.0, 6.0 and 7.0.

This translator supports exporting curves, surfaces, solids and grouped objects. It does not support exporting layers.

Tip:

This Designer Elements program EPS exports an embedded post script file. This type is great for printing and embedding in such applications as Microsoft Word and PowerPoint.

This EPS export is different from the EPS export used in Vellum 3D which exports an Adobe Illustrator version of post script.

EPS

Selecting this type exports an EPS (encapsulated postscript) file. There are no options for this type.

CGM

Selecting this type exports a computer graphics metafile. This format is a 2D data exchange format which allows graphical data to be stored and exchanged among graphics devices, applications and computer systems. This metafile is not a picture but a description of a picture. There are no options for this translator.

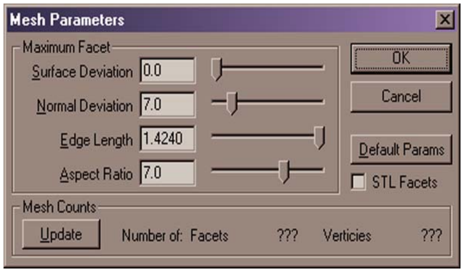
Facet

Selecting this type exports an ASCII Facet file. There are no options for this type.

STL

The precise, mathematical representation of a solid or surface must often be converted into a collection of imprecise planar facets. These facets may be used to export a model to the STL format and when changing a solid or surface

(**Edit>Change Object Type...**) to a mesh. The amount of error that results from this conversion is controlled by the settings in the mesh parameters dialog box.



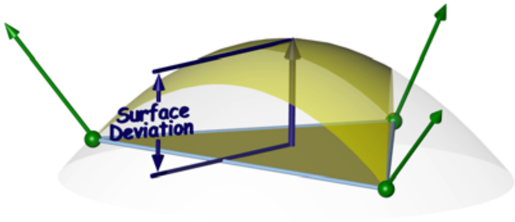
During the conversion, vertex points are distributed on the surface or solid. These vertices are then grouped into 3-sided and 4-sided facets. The conversion is deemed acceptable when the generated vertices and facets satisfy the settings. The 5 available settings are Surface Deviation, Normal Deviation, Edge Length, Aspect Ratio, and STL Facets. These settings are defined in the sections below.

Change the facet settings as needed in the dialog and then click the Update button to see the number of facets and vertices generated. Determining the combination of settings that will work for a given situation can be a little bit of an art. If one setting becomes too tight, the other settings will have no effect. If one setting becomes too loose, it will have no effect.

Keep in mind that the settings are used by the faceting algorithms if possible. It is often not possible to satisfy all settings simultaneously. In this situation, the algorithm decides which settings to “loosen”.

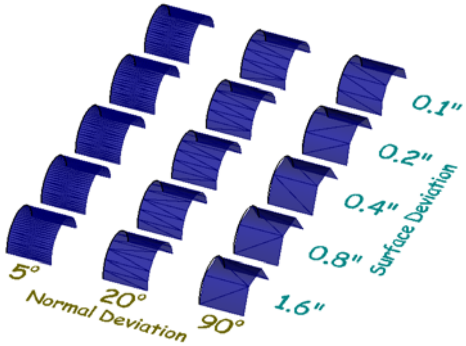
The Mesh Parameters dialog box contains the following options:

Surface Deviation This setting controls the maximum allowed distance between any point on the actual surface or solid and the facet representing that point. The exaggerated figure below shows the largest distance between a patch on the actual surface (yellow) and the corresponding planar facet (brown).



Normal Deviation

This setting controls the maximum allowed angular difference between any normal on the actual surface or solid and the corresponding interpolated normal on the facet.



Edge Length

This setting controls the maximum allowed edge length of any given facet.

Aspect Ratio

This setting controls the maximum allowed aspect ratio of any given facet.

STL Facets

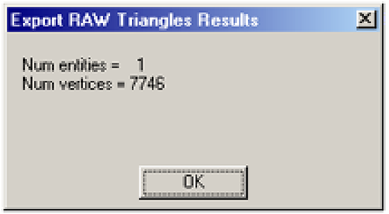
This setting will force the facets generated to be suitable for stereolithography usage. This setting is usually used when exporting STL files.

VRML

Selecting this type exports virtual reality modeling language files. There are two options for VRML, Version 1.0 and Version 2.0.

RAW

Selecting this type exports a raw file consisting of triangular vertices. These vertices define the x, y and z locations of the 3D faces which make up your model. There are no options for this type.



After exporting, a dialog box appears displaying the number of entities and vertices in the exported model.

You can view the raw file by opening it in any text editor.

Adobe Illustrator

Selecting this type exports an Illustrator file. This Illustrator file is compatible with Adobe Illustrator versions 5.0 through 8.0. This translator supports exporting the Hidden and Hidden w/dimmed render options. All dimensions are converted into lines and text. There are no options for this type.

Macintosh only: When you export an Illustrator file, this Designer Elements program automatically displays it with an Illustrator icon. If you double-click the file, Adobe Illustrator launches.

Text

Selecting this option will export the text contained in the file along with a list of objects contained in the file.

Catia v4

Selecting this type exports a CATIA v4 file. You can choose to export only the selected items and have each item export into its own file with the MultiFile option checked.

ProE/Granite

Selecting this type exports a file that is based upon the Granite kernal that PTC uses in the Pro/E program. This provide a kernal to kernal export for more accurate translations. There are five options for this export option: Granite, Neutral, IGES, Step and VDA.

Macromedia

Selecting this type exports a Macromedia file. You can choose the Geometry Quality and you have the option of creation a log file if you check the box.

ViewPoint Media

Selecting this export type exports a Viewpoint Media file with an .mts or .mtx file extension. The .mtx file contains the xml representations of each object as a mesh object, the .mts file contains a list of all objects that should be displayed along with their display status. For futher information please refer to the Viewpoint developers guide or examine the sample .html code generated by our 3D Web Publish command. To view the .mtx/.mts files you will need a viewpoint compatible 3D web creations product such as Adobe Atmosphere.

To skip the export and view this file type immediately choose **File>3D Web Publish**, your geometry will be displayed in your web browser.

PICT (Macintosh only)

Selecting this type exports Pict files, the Macintosh native file format. There are no options for this type.

Export Command

This command in the File menu saves a document in the format you specify.

1. Choose ***File>Export***.

   The Export dialog box appears.

2. Select the export type and its options. If you wish to export only selected objects on the screen, rather than the entire document, click *Selected Only*.

3. Click OK. The standard Save document as dialog box appears prompting you for a name and location for the exported file.

   Windows: The type of file you are exporting is indicated with the appropriate filename extension.

4. Enter the file name and click Save. The exported file is saved with the name and location entered.

Exporting Tips and Notations

These tips and notations will help you successfully export files.

- Windows - this Designer Elements program does not support exporting bitmaps.

- When exporting files from this Designer Elements program for import into another program, determine what version of the translator is used by the other program. Choose the appropriate translator in this Designer Elements program. Be sure to check whether there are any tips or notations for a successful translation in this Designer Elements program or the other application.
- In general, it is best show all layers before you export from this Designer Elements program or from another program that you will import into this Designer Elements program, so that you know what you are exporting.
- AutoCAD does not like the & symbol in the layers table of the DXF file. Remove the symbol from your layer name in this Designer Elements program or AutoCAD will generate an error reading in the file.
- If line patterns do not import into AutoCAD accurately from a Designer Elements program DXF or DWG file, change the AutoCAD variable LTSCALE to display the patterns at an appropriate scale.

Drawing Composition

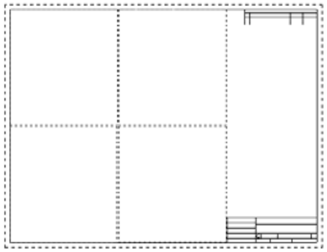
This Designer Elements program allows you to quickly create 2D drawings from one or more surface or solid objects. The 2D drawing views are bi-directionally associative. Changes made to the objects automatically appear in the 2D drawings. Likewise, you can change objects in the 2D drawing by using the Edit Objects dialog box. These also update on your objects. The topics covered in this chapter include:

- Model to Sheet Command
- Drawing Views
- Undfolding Views
- Editing a Drawing View
- Layout Templates
- Printing the Layout

Tip:

If you turn on all layers and zoom out, your original geometry is visible in the Top view.

Model to Sheet Command



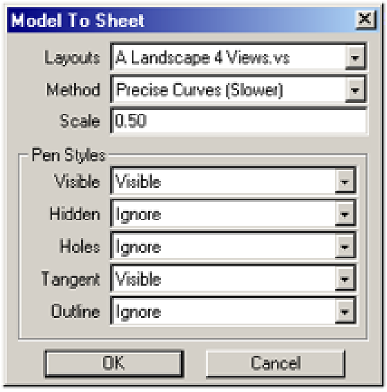
The Model to Sheet command, located in the Layout menu, automates the process of creating drawings through the use of templates. Templates are empty drawings with predefined and pre-arranged drawing views set within drawings borders. With this feature, this Designer Elements program creates 2D geometry from the 3D surface and solid models you selected and places them in drawing views on a new layer, called Sheet View. Your original 3D geometry remains.

Warning: You cannot use this command with curves or groups.

If your geometry contains objects or drawing border lines using the background color, this Designer Elements program automatically changes it to the foreground color. This capability is especially helpful when your background color is black. See Chapter 6 for information on setting the background and foreground colors.

The new sheet view displays in the Top view. Drawing views placed on this sheet can only be selected when the view orientation is Top. The sheet is positioned at the origin using the lower left corner as the alignment reference.

Choosing the Model to Sheet command displays the following dialog box.



The dialog box contains these options:

Layouts This option sets the layout for displaying your geometry. These layouts are predefined and located in the Layout folder within the same folder as the Designer

Elements application. The layouts contain drawing views and may contain title blocks and drawing size boundaries depending on the layout selected.

You can create or modify any of the templates based on your drawing layout preferences. See the “Layout Templates” section at the end of this chapter for more information.

Method This option sets how hidden edges display in the final drawing. You have five settings, Precise Curves (Slower), Polylines (Faster), Flat, Gouraud and Phong.

Tech Note:

The hidden line settings for Edge display do not affect the Hidden Line render mode available in the Render Options dialog box. See Chapter 33 for information on the render modes.

Pen Styles These option are covered in the Draw View Properties section of this chapter.

Scale This option sets the scale of the drawing.

OK, Cancel Clicking OK completes the Model to Sheet command and displays your geometry based on the specified settings. Clicking Cancel, cancels the command and closes the dialog box.

Using the Model to Sheet Command

1. Select the models you want to display on the sheet.
2. Choose ***Layout>Model To Sheet***.

   The Model to Sheet dialog box displays.
3. Choose the Layout and Edge Display from their respective pull-down lists.
4. Enter a scale in the Scale field.
5. Click OK.

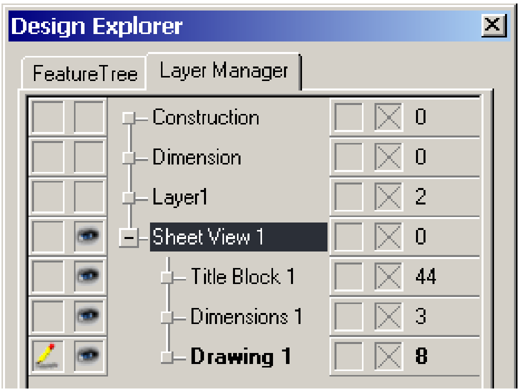
   2D geometry is created from your 3D model, placed in the drawing views of the selected layout, and displayed in the Top view. A Sheet View layer is also created on which the drawing views containing your geometry are placed.

   If you selected a layout with a format, A, B, C, D or E, the format is also placed on the Sheet View layer. Choose ***Layout>Layer Manager*** to see the new layer and the num-

Referral:

Drawing views are explained later in this chapter.

ber of new 2D objects on that layer.



You can also see that the Sheet View layer is now the work layer and all other layers are turned off.

Undoing the Model to Sheet Command

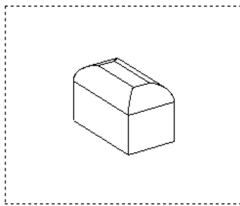
If you inadvertently selected the Model to Sheet command or later want to remove the sheet, you can use the Undo command or the Layer Manager.

- Using Undo: Choose ***Edit>Undo*** (CTRL+Z (Windows) or z+Z (Macintosh). You may have to use the Undo command more than once depending on the number of operations conducted since the Model to Sheet command was chosen.
- Using the Layer Manager: Display the Layer Manager. Choose another layer as the work layer. Then select the Sheet View layer and delete it. Click OK to close the dialog box. Your view is still set to Top. Change your view and zoom scale to view your geometry.

Tech Note:

Drawing views operate differently from those in Vellum 3D. You are not looking at a 3D model in a drawing view but rather 2D wireframe object created from the 3D model. You cannot use the Trackball to rotate the view orientation of a specific drawing view or zoom in the drawing view. However, you can use the *Change View* command and *Properties* command in the Drawing View menu to make adjustments. See the “Drawing View Menu” section later in this chapter.

Drawing Views

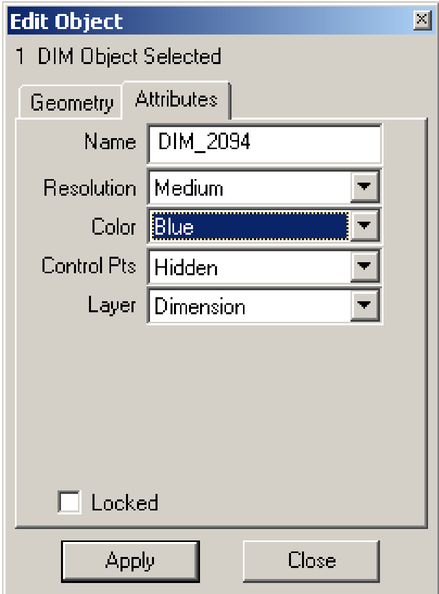


(**Cobalt™** and **Xenon™** Only)A drawing view is a defined area into which the 2D geometry is placed when using the Model to Sheet command. The drawing view displays the geometry in a particular view orientation, like Top, Trimetric, etc. The example here shows a drawing view with a Trimetric view orientation.

Drawing views are created automatically using the Model to Sheet command or by using one of the Sheet tools. Drawing views are placed on top of the Sheet view layer rather than on the layer. Each drawing view has its own work plane. Any additional objects, text or dimensions placed in the view appear only in that view on that work plane, set to Top.

The drawing view frame uses the foreground color set in Preferences. If the Show Triad command is activated, the Triad displays in each drawing view.

Spacing for crosshatching, dimensions, text, arrow size and line font in drawing views can now be set relative to the sheet through the Edit Objects dialog box, independent of the scale set in the drawing view. This allows you to set any scale for the view without concerning yourself with how the item will appear.

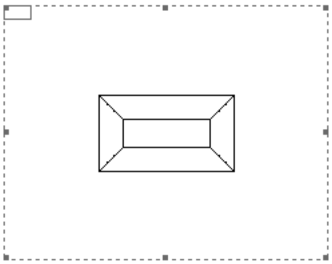


Just double-click the item to display the Edit Objects dialog box. Select the Attributes page and the Ignore Scale option. Click Close or Apply to update the selected item.

Cobalt and Xenon Only

Activating Views

Once you've created a drawing, you activate individual views by clicking in the rectangular region surrounding the view. When activated, the view boundaries highlight in red.



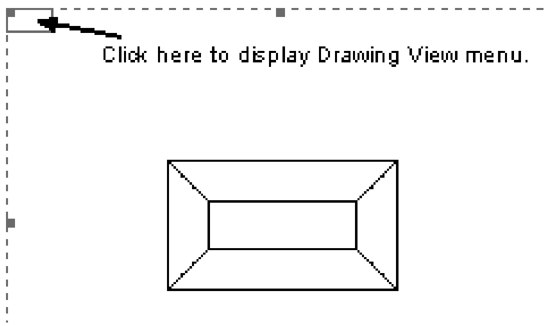
You cannot activate drawing views if the sheet is displayed in any view other than Top. All geometry in the view becomes available for selection when the view is activated. The Drafting Assistant recognizes all snap points in the view. Click outside the area to deactivate the view.

Tech Note:

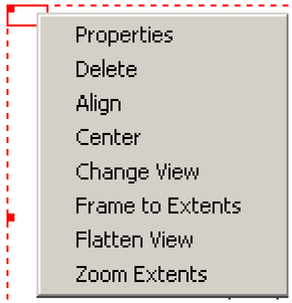
The Drafting Assistant only recognizes geometry within an active view.

Drawing View Menu

Drawing views have specific commands associated with them. These are accessible through a pull-down menu in the drawing view. Click the mouse in the upper left region of the view.



The view menu displays.



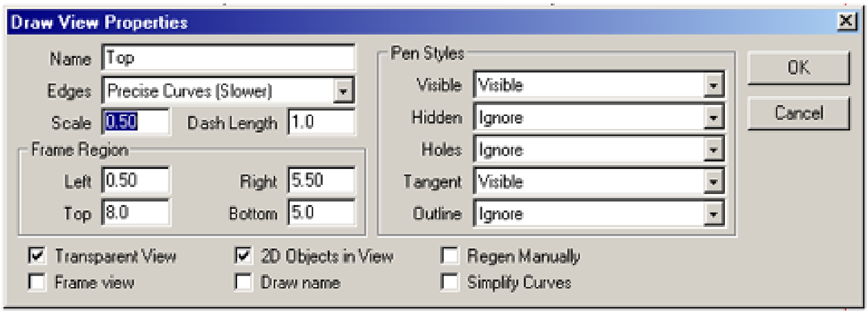
You can also display the view menu by clicking the right mouse button (Windows) or by holding down the CONTROL key and clicking the mouse button (Macintosh).

The view menu contains the following commands: Properties, Delete, Align, Center, Change View, Frame to Extents and Flatten View.

Properties Command

The Properties command gives you access to a set of properties that affect a drawing view's display on the screen. When you choose the command, the following dialog box displays.

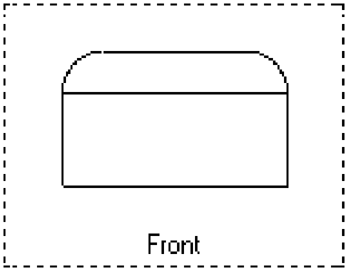
Cobalt and Xenon Only



The dialog box contains the following options:

Name

This field allows you to specify the name that appears in the drawing view when you activate the Draw name check box in the dialog box.



Edges

This option sets how hidden edges display in the final drawing. You have five settings, Precise Curves (Slower), Polylines (Faster), Flat, Gouraud and Phong.

Tech Note:

The hidden line settings for Edge display do not affect the Hidden Line render mode available in the Render Options dialog box. See Chapter 33 for information on the render modes.

Scale

This field sets the scale of the drawing.

Dash Length

This field sets the length of dashes when using either the Dash All Hidden (precise) or Dash Only Holes (precise) edge display options.

Frame Region

This region defines the rectangular area of the selected drawing view and its location. The field values represent the view edge's distance from the origin (where the sheet is automatically placed).

The Left field sets the distance from the left view edge to the origin (0,0,0). The Right field sets the distance from the right view edge to the origin (0,0,0). The Top field sets the distance from the top view edge to the origin (0,0,0). The Bottom field sets the distance from the bottom view edge to the origin (0,0,0).

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Units are based on those set in Preferences.

Pen Styles

Pen styles are used to define edge attributes options within a draw view. Edge options include visible, hidden, holes, tangent and outline. A pen style defines the pen weight, pattern and color.

One advantage to using a pen style to define an edge attribute is the ease at which you can then explore different pen styles for all your draw views. Any change to the master pen style will automatically update all edges that use that pen style.

For example, suppose you want your visible lines printed in a E size drawing at a heavier weight than an A size drawing. In this case just change the Visible pen weight to your desired weight and print.

Note: You can also select "Use Object Color" which does not use a pen style but always follows the object color used to create the draw view.

Visible

The Visible edge format is applied to edges that are not hidden. However this excludes edges that are classified as tangent or outline.

Hidden

The hidden edge format is applied to edges that are occluded.

Holes

The holes edge format is applied to edges that are hidden and cylindrical.

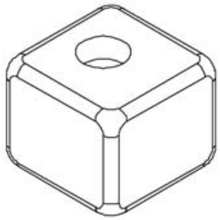
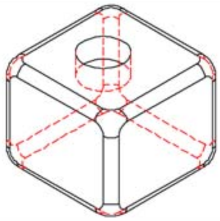
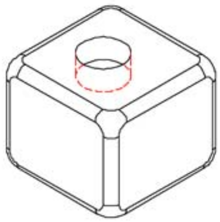
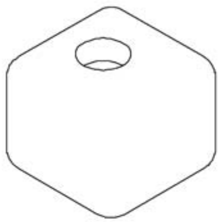
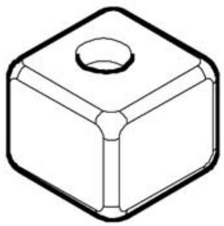
Tangent

The tangent edge format is applied to edges that are shared by two faces that are tangent. Fillet edges are examples of tangent curves.

Outline

An edge is classified as an outline if the normal to the face on one side of the edge points towards the eye and the normal to the face on the other side of edge points away from the eye.

The following picture gives examples of how changing these parameters affect your model.

Visible = Visible Pen Style Hidden = Ignore Holes = Ignore Tangent = Visible Pen Style Outline = Ignore	
Visible = Visible Pen Style Hidden = Hidden Pen Style Holes = Hidden Pen Style Tangent = Visible Pen Style Outline = Ignore	
Visible = Visible Pen Style Hidden = Ignore Holes = Hidden Pen Style Tangent = Visible Pen Style Outline = Ignore	
Visible = Visible Pen Style Hidden = Ignore Holes = Ignore Tangent = Ignore Outline = Ignore	
Visible = Visible Pen Style Hidden = Ignore Holes = Ignore Tangent = Visible Pen Style Outline = My Thick Pen Style	

Cobalt and Xenon Only

Transparent View This check box controls the whether the drawing view background is clear or opaque. When selected, the background is clear and objects located underneath the view are visible through the view.

Remember that drawing views are placed on top of the Sheet view layer rather than on the layer. Geometry placed on the layer may fall underneath a view.

Frame view

This check box controls the display of the drawing view frame. When selected, the view frame is visible.

Regen Manually

This check box controls whether the view regenerates or updates automatically after making changes to your geometry. This control is helpful if you have a file that requires a long regeneration time. With this option selected, you can make a series of changes without having to deal with the delay of regeneration. If you want to regenerate one particular drawing view, toggle this option between on and off.

Cobalt and Xenon Only

This option relates to the Manual View Regeneration tool in the Sheet tools palette which regenerates all views on the sheet.

2D Objects in View

This check box controls the 3D nature of geometry within the drawing view. When checked, all geometry is flattened to 2D geometry. For example, an arc in a Trimetric drawing view becomes an ellipse in 2D when this option is checked. When left unchecked, no objects are flattened.

Draw name

This check box controls the display of the text in the Name field. When selected, the text displays at the lower border of the frame, centered between the left and right edges.

Simplify Curves

The simplify curve option will attempt to simplify splines into lines, arcs, or circles when appropriate. This is primarily useful if the objects used in the model to sheet consisted of splines instead of the typical analytics.

Auto Hatch

(Only appears for section views) This check box controls hatching for a section view. When selected, hatch automatically appears within section geometry. This option uses the default hatch set in the Cross Hatch dialog box. See Chapter 28.

Area Hatch Only (Only appears for section views) This check box controls the geometry that appears in the section view. When selected, only the geometry cut by the section displays. When not selected, the section view shows the section geometry and any portion of the geometry lying behind the section cut.

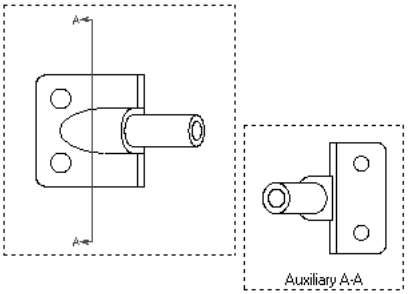
Delete Command

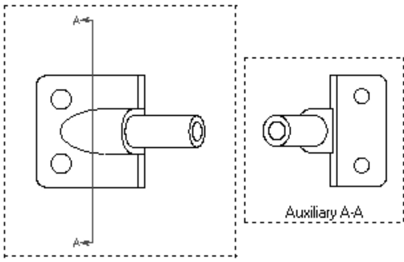
This Delete command removes the drawing view and its contents from the drawing. The original model remains.

Align Command

Cobalt and Xenon Only

The Align command functions like a tool in this Designer Elements program in that the Message Line contains steps for its use. This command aligns an active view with another selected view. You can use the command on section views, general drawing views and auxiliary views. The left graphic here shows two views, the Top view of a CAM part with its associated auxiliary view. The right graphic shows the same two views aligned.





1. Select the view you want to align.
2. In the Drawing View menu, choose the Align command.

   The Message Line reads: *Align View: Pick parent view to align with*.
3. Select the parent view.

   The two views align.

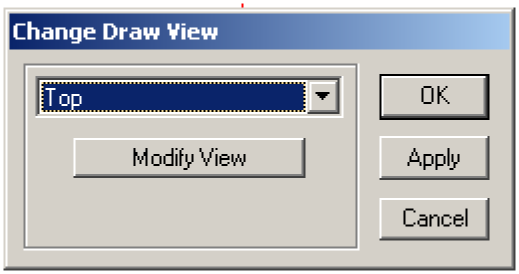
Center Command

The Center command centers the objects within the frame boundaries. This is especially helpful if you've moved the drawing view or changed the view orientation of the geometry.

Change View Command

The Change View command allows you to change the view orientation of the selected drawing view. You can change the view to one of the pre-defined or userdefined views or modify the view by specifying the Eye/Reference point, Azimuth/Elevation or rotation values.

When you choose this command, the Change Aux View dialog box appears.



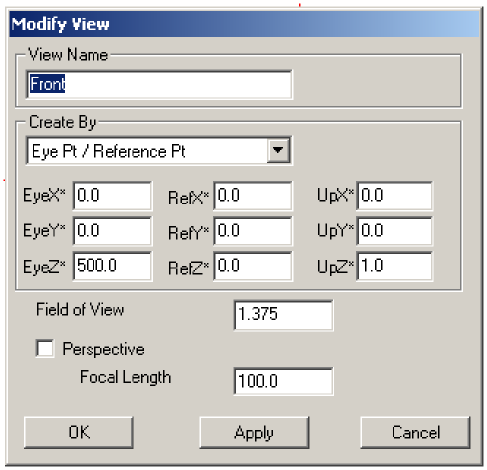
Cobalt and Xenon Only

The dialog box contains the following options:

View setting — This setting displays the name of the current view and a pull-down list for selecting a different view. The views include Right Side, Left Side, Front, Back, Top, Bottom, Iso (Top Front Left), Iso (Top Front Right), Iso (Top Back Left), Iso (Top, Back Right), Iso (Bottom, Front Left), Iso (Bottom Front Right), Iso (Bottom Back Left), Iso (Bottom Back Right), Trimetric and any user-defined views.

Modify View

Choosing this button brings up the Modify View dialog box.



Change the Eye/Reference point, Azimuth/Elevation or rotation values for the view. You can also name the view. If you enter a name and want to display it in your drawing view, you must also enter the new name in the Draw View Properties dialog box. This view is specific to the drawing view and not available through the Trackball.

Cobalt and Xenon Only

OK

Click this button to close the dialog box and the view changes.

Cancel

Click this button to close the dialog box without making any changes.

Apply

Click this button to change the view but keep the dialog box open.

Frame to Extents Command

The Frame to Extents command shrinks the frame to the size of the geometry in the view.

Flatten View Command

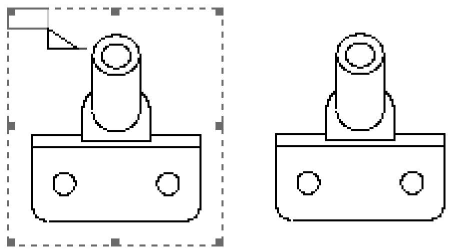
The Flatten View command deletes the drawing view, projects all objects into 2D entities and places them on the sheet creating complete 2D geometry. This command destroys any associativity between the flattened objects and the 3D model that created them.

The objects are scaled according to the value set in the Properties dialog box for the view. If your view contains dimensions, they are flattened also. The dimension value from the original view is placed in the flattened view. If the view scale was set to anything other than 1.0, the actual measurement of the flattened object will be different than the dimension value. For example, if you dimensioned an object in a view that was 1.25 inches and shown at a scale of 2, the flattened dimension would still read 1.25. However, the actual length is 2.50. I

Important: Be sure to mark your drawings, “Not to Scale,” if flattening a view with any scale other than 1 so when people read the dimension value rather than measuring the object.

Cobalt and Xenon Only

Since the geometry in the view is scaled, its size on the screen after being flattened, is the same as it was in the view. The left graphic here shows a selected view. The right graphic shows the same geometry flattened.

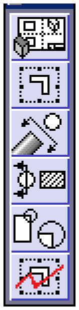


If you flatten a detail view, the detail view boundary is converted into a circle with a phantom pen pattern and flattened onto the sheet with the geometry.

Zoom Extents Command

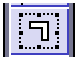
The Zoom Extents command changes the scale of the drawing view so the objects within it fill the drawing view. This command operates similarly to the Zoom All command except that it applies to a drawing view.

Sheet Tools



After you've created the drawing, you may want to make some changes to the drawing to better illustrate certain aspects of your model. This Designer Elements program provides you with tools to create new views, modify a selected view and add auxiliary views, section views and details views to your drawing. These tools are available through the Sheet tools palette. The tools include Drawing View, Auxiliary View, Section View, Detail View and Manual View Regeneration. Choose ***Window>Sheet Tools*** to display the palette.

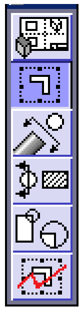
Drawing View Tool



This tool allows you to create empty drawing views. Use this tool to add drawing views to your current layout already containing views or create customized layouts.

Cobalt and Xenon Only

Using the Drawing View Tool



1. Select the Drawing View tool. The Message Line reads: *Drawing View: Pick start position view frame*.
2. Click the starting point position for the frame. The Message Line now reads: *Drawing View: Pick end position view frame*.

   Notice that as you move the pointer to select the ending position an outline of the frame appears.
3. Click the ending point position for the frame.

   The new view appears with the frame edges. The view orientation is Front. Continue placing new views as desired. To activate the view, choose the Selection tool and select the frame

Adding Associative Geometry to Empty Drawing Views

This Designer Elements program allows you to add geometry (associative to the original 3D geometry) to empty views. When changes are made to the geometry all associative geometry updates.

1. Turn on the layers containing the 3D geometry you want to place in the new view.

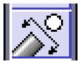
2. Select the 3D geometry.

   If you select geometry from an existing drawing view, you would only be copying 2D wireframe geometry.

3. Choose ***Edit>Copy***.
4. Select the empty drawing view.
5. Choose ***Edit>Paste***.

   The geometry pastes into the new view. Use the Drawing View menu to center the geometry, change the view, etc.

Auxiliary View Tool



Cobalt and Xenon Only

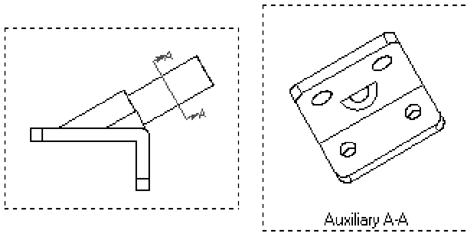
This tool allows you to create auxiliary views from a referenced drawing view. Auxiliary views dynamically align to that view and are associative to both the original view and the auxiliary dimension. Change the geometry and auxiliary view automatically updates. Move the dimension location in the original view and again the auxiliary view reflects the changes. Auxiliary dimension lines appear in the original view. These dimensions are placed on the Sheet View layer. The Auxiliary tool can only be used when a drawing view is present.

Tech Note:

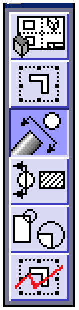
The font type, size or style of the dimension label cannot be changed. Use the Text tool to create to create your own labels.

By default, auxiliary labels are alphabetical. If you place more than one auxiliary view in your drawing, the auxiliary view label automatically increments to the next letter. If you close the file containing auxiliary views and later reopen it and add another auxiliary view, the label increments to the next letter based on the last auxiliary label in the file. You can also enter any label text desired. However, user-defined label text does not automatically increment.

The left graphic here shows the referenced view with auxiliary view dimension lines. The right graphic shows the auxiliary view.



Using the Auxiliary View Tool



1. Select the Auxiliary View tool. The Message Line reads: *Auxiliary View: Pick drawing view for auxiliary view.*
2. Select the drawing view from which you want to create the auxiliary view.

   The Message Line now reads: *Auxiliary View: Pick start and end of folding line*.
3. Click the start and endpoint locations that specify the folding line for the view. The points do not need to be located on the geometry.

   The Message Line reads: *Drag window to final position*.
4. Drag the view to a new location and click the mouse button. Notice that as you move the pointer an outline of the frame appears.

   The auxiliary view displays with a view label at the specified location. The auxiliary view dimension appears in the original view.

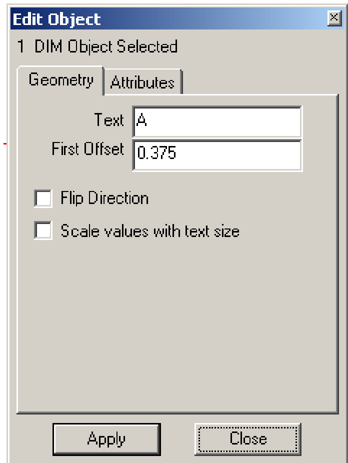
   If you want to delete the view later, you must delete both the view and the dimension line in the original view. You may have to adjust the auxiliary labels to accommodate the deletion.

Cobalt and Xenon Only

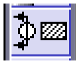
Changing the Direction or Label of the Auxiliary View

If you want to change the auxiliary view direction opposite to that indicated by the view dimension, select the dimension and choose ***Window>Edit Objects***. In the Geometry page, check the Flip Direction option and click Apply. The direction of the view changes.

You can also change the view label in this dialog box.



Section View Tools



These tools create horizontal, vertical and 2 point section views. Section view geometry is associative to the geometry in the view and the section dimension. Change the geometry and the section view automatically updates. Move the dimension location in the original view and again the section view reflects the changes. Section view dimensions are placed on the Sheet View layer.

Cobalt and Xenon Only

The Section tools allow you to specify whether you only want to see the section cut or both the section cut and the geometry lying behind it. This ability is controlled in the Draw View Properties dialog box, accessed through the Drawing View menu. See the “Drawing View Menu” section earlier in this chapter.

Tech Note:

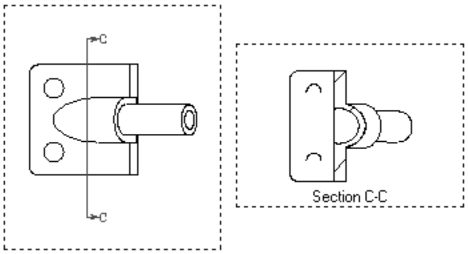
The font type, size or style of the dimension label cannot be changed. Use the Text tool to create to create your own labels.

Tech Note:

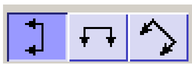
You cannot paste solids into section views. If you want to add a solid, you must add it to the original view.

By default, section labels are alphabetical. If you place more than one section view in your drawing, the section view label automatically increments to the next letter. If you close the file containing section views and later reopen it and add another section view, the label increments to the next letter, based on the last section label in the file. You can also enter any label text desired. However, user-defined label text does not automatically increment.

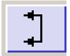
Section views also support crosshatching. These tools can only be used when a drawing view is present. The graphics here show a vertical section view.



When you select the Section View tool, a subpalette appears containing three tools: Vertical, Horizontal and 2 Pt.

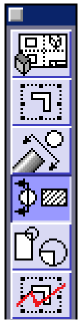


Vertical Section View Tool



This tool creates a vertical section view.

Using the Vertical Section View Tool



1. Select the Section View tool.
2. Select the Vertical Section View tool in the Message Line. The Message Line reads: *Section View: Pick drawing view for section*.

   If a view is already selected, skip to step 3.
3. Select the drawing view.

   The Message Line now reads: *Section View: Pick location for vertical section*.
4. Click a point on your geometry for the section.

   The Message Line reads: *Drag window to final position*.
5. Drag the view to a new location and click the mouse button. Notice that as you move the pointer an outline of the frame appears.

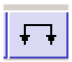
   The section view displays with the view label shown in the Status Line. If you want a different label, enter it in the Section data field and press ENTER (Windows) or RETURN (Macintosh). The section view dimension appears in the original view.

   If you want to delete the view later, you must delete both the view and the dimension line in the original view. You may have to adjust the section labels to accommodate the deletion.

Cobalt and Xenon Only

The Status Line contains the section view label.

Horizontal Section View Tool



This tool creates a horizontal section view.

Using the Horizontal Section View Tool

1. Select the Section View tool.
2. Select the Horizontal Section View tool in the Message Line. The Message Line reads: *Section View: Pick drawing view for section*.

If a view is already selected, skip to step 3.

3. Select the drawing view.

   The Message Line now reads: *Section View: Pick location for horizontal section*.

4. Click a point on your geometry for the section.

   The Message Line reads: *Drag window to final position*.

5. Drag the view to a new location and click the mouse button. Notice that as you move the pointer an outline of the frame appears.

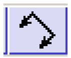
   The section view displays with the view label shown in the Status Line. If you want a different label, enter it in the Section data field and press ENTER (Windows) or RETURN (Macintosh). The section view dimension appears in the original view.

   If you want to delete the view later, you must delete both the view and the dimension line in the original view. You may have to adjust the section labels to accommodate the deletion.

The Status Line contains the Section view label.

Cobalt and Xenon Only

2 Pt Section View Tool



This tool creates a section view based on the orientation of two user-defined points.

Using the 2 Pt Section View Tool

1. Select the Section View tool.

1. Select the 2 Pt Section View tool. The Message Line reads: *Section View: Pick drawing view for section*.

   If a view is already selected, skip to step 3.

2. Select the drawing view.

   The Message Line now reads: *Section View: Pick start and end of section orientation*.

3. Click two points on your geometry to indicate start and end points for the section cutting line.

   The Message Line reads: *Drag window to final position*.

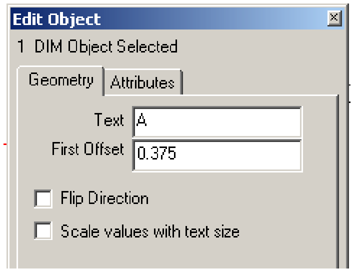
4. Drag the view to a new location and click the mouse button. Notice that as you move the pointer an outline of the frame appears.

The section view displays with the view label shown in the Status Line. If you want a different label, enter it in the Section data field and press ENTER (Windows) or RETURN (Macintosh). The section view dimension appears in the original view. You may have to adjust the section labels to accommodate the deletion.

If you want to delete the view later, you must delete both the view and the dimension line in the original view.

The Status Line contains the Section view label.

Changing the Direction or Label of a Section View



Cobalt and Xenon Only

If you want to change the section view direction opposite to that indicated by the view dimension, select the dimension and choose ***Window>Edit Objects***. In the Geometry page, check the Flip Direction option and click Apply. The direction of the view changes.

You can also change the view label in the dialog box.

Sections and Crosshatching

By default, section views automatically contain crosshatching. You can remove the crosshatching or change the hatch pattern for a selected view through the Cross Hatch dialog box.

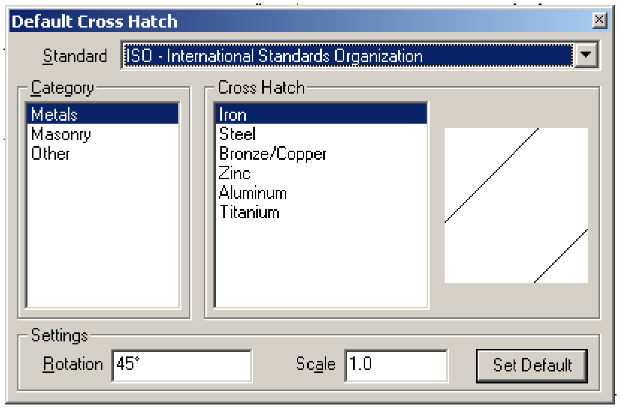
You can also set the default hatch pattern through this dialog box. When nothing is selected, choose ***Pen>Cross Hatch***. See Chapter 28 for more information on the dialog box and the hatches available.

If you have more than one object cut in the section, you can define separate hatch patterns.

Tech Note:

The crosshatching used for geometry in a section view must be set separately from the material set in the Mass Properties dialog box. See Chapter 24 for more information on Mass Properties.

1. In the section view, select the hatch pattern.
2. Choose ***Pen>Cross Hatch***. The following dialog box displays.



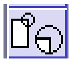
3. Select a category and Cross Hatch pattern from their respective lists.
4. Set the hatch rotation angle and scale.
5. Click Apply to change the selected hatch.

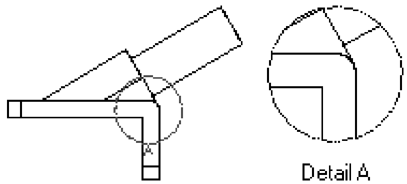
   If you are not satisfied with the hatch pattern, choose another. You can also change the hatch patterns of any other section geometry in the view or in other views.

6. Click the Close button (Windows) or the Close box (Macintosh) to exit the dialog box when you're finished.

Cobalt and Xenon Only

Detail View Tool





This tool creates a detail view from a drawing view. Detail view geometry is associative to the geometry in original view and the detail dimension. Change the geometry within the area defined by the detail and the detail view automatically updates. Move the dimension location or the size of the detail in the original view and again the detail view reflects the changes. Detail view dimensions are placed on the Sheet View layer.

Tech Note:

The font type, size or style of the dimension label cannot be changed. Use the Text tool to create your own labels.

By default, details dimensions are alphabetical. If you place more than one detail view in your drawing, the detail view dimension and label automatically increment to the next letter. If you close the file containing detail views and later reopen it and add another detail view, the dimension and label increment to the next letter, based on the last detail label in the file. You can also enter any label text desired. However, user-defined label text does not automatically increment. The graphic here shows a drawing view and an associated detail view.

Once a detail view is created, you can move the detail view dimension in the original view from its default location, shown as A in the left graphic above. Select the label, choose

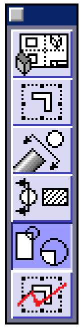
Edit>Show Points and drag the control point/detail dimension to another location. The detail view label, shown as Detail A above, cannot be moved.

This tool can only be used when a drawing view is present.

Tech Note:

You cannot paste solids into section views. If you want to add a solid, you must add it to the original view.

Using the Detail View Tool



1. Select the Detail View tool. The Message Line reads: *Detail View: Pick drawing view for detail view.*

   If a view is already selected, skip to step 3.

2. Select the drawing view.

   The Message Line now reads: *Detail View: Pick detail center.*

3. Click the center point on your geometry for which you want a detail view.

   The Message Line reads: Detail View: Pick detail edge pt.

4. Click the detail edge point.

   The Message Line reads: *Drag window to final position*.

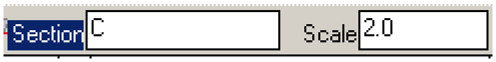
5. Drag the view to a new location and click the mouse button. Notice that as you move the pointer an outline of the frame appears.

   The detail view displays with the view label and the scale shown in the Status Line If you want a different label or scale, enter the data in the appropriate data field and press ENTER (Windows) or RETURN (Macintosh). The detail view dimension appears in the original view.

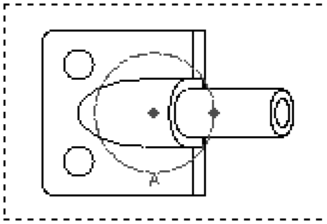
   If you want to delete the view later, you must delete both the view and the dimension line in the original view. You may have to adjust the detail labels to accommodate the deletion.

Cobalt and Xenon Only

The Status Line contains the detail view label and Scale.



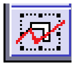
Resizing the Detail View Dimension



You can change the diameter of the detail view dimension. Select the detail dimension and choose ***Edit>Show Points***. Two controls point appear defining the center and edge of the dimension.

Select one of the control points and drag it to a new location.

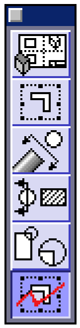
Manual View Regeneration Tool



This tool regenerates all drawing views on the sheet. This is useful if you have selected the Manually Regen option in a Draw View Properties dialog box for one or more views and made changes to them.

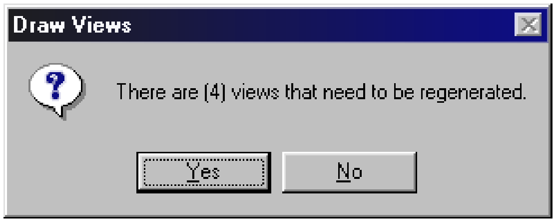
Using the Manual View Regeneration Tool

Cobalt and Xenon Only



1. Select the Manual View Regeneration tool.

   The Draw Views dialog box appears telling you the number of views that need to be regenerated.

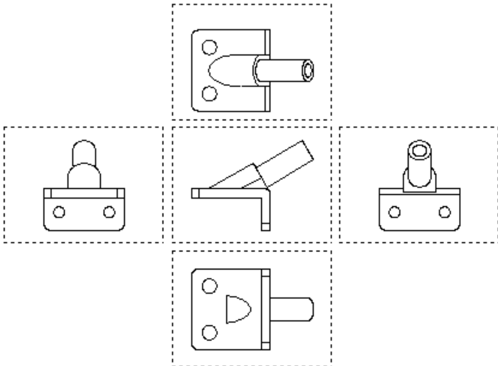


2. Click Yes to regenerate all views.

   Click No to close the dialog box without regenerating the views.

Unfolding Views

You can quickly create new views from an existing drawing view by using an unfolding operation. The graphic here shows a center view unfolded in four directions.



1. Choose the Selection tool and activate you want to unfold.
2. Hold down the CTRL key (Windows) or the OPTION key (Macintosh) and drag the view to the left, right, top or bottom of the view.

   A new view is created with the geometry rotated 90° from the selected view.

Editing a Drawing View

Adding Objects to a View

You can add objects in an existing or new view by either creating them using a drawing tool or by copying and pasting them into a view.

Cobalt and Xenon Only

Creating Objects in a View

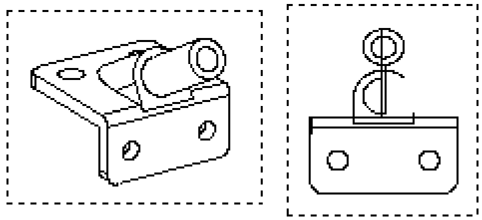
Once a view is activated, you can create additional wireframe and solid objects in the view. Since the Drafting Assistant functions inside the view, you can place the objects relative to the other objects already present.

You can also place dimensions and text in the view using the Dimension tools and the Text tools. All dimensions are placed on the Sheet View layer rather than the Dimension layer for drawing views. Since the drawing view work plane is set to top, all text and dimensions appear correctly.

All entities created in an active view, display only in that view.

Copying and Pasting Objects into a View

You can also paste objects copied from another location into a selected view. When you paste the objects, this Designer Elements program examines the contents of the paste buffer and inserts it in the active view. The inserted object is associative to the original object. This feature is valuable when creating a new drawing view.



If the objects were copied from a 3D model, the 2D objects created from them are associative to the model. You can change the view later and the geometry will display correctly. If you copy 2D objects from a view (regardless of whether they were created from a 3D model), these objects do not reference the 3D model. Since this 2D geometry is not associative to a 3D model, changing the view may not create a complete view. The graphic here shows an example of this. The 2D geometry in the Trimetric view was pasted into the Right Side view.

Editing Objects in the View

Editing 2D Geometry created from 3D Geometry

You can edit your 2D geometry in a number of ways: changing the line characteristics, layers or modifying the geometry.

Changing Line Patterns and Layers

Cobalt and Xenon Only

Since an object placed in the drawing views are true 2D wireframe, you can assign a different line pattern, color, weight or layer to the entities making up the object. To change a line characteristic, select the line in the view. In the Pen menu, choose a new pattern, color or weight. You can also change line characteristics through the Edit Objects dialog box.

To change the layer for the line, choose Edit>Change Layer and select the new layer. You can also change the layer through the Edit Objects dialog box.

Modifying 2D Geometry

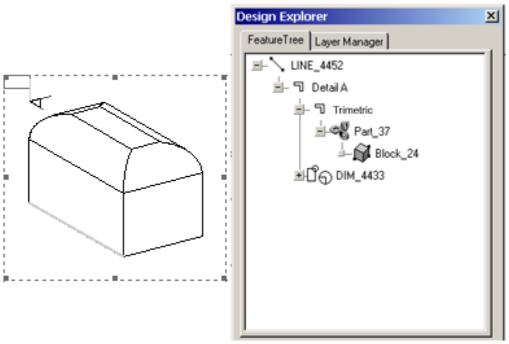
Because the 2D geometry created from the 3D model is composed of individual curves, you cannot modify the length or the location of the curve by dragging a control point like you would in a normal model. If this were possible, the associativity of the model would be destroyed. For example, a line in a model may represent the visible edge of a cylinder which is not actually present in the 3D model. If this was changed, you would no longer have a cylinder.

If you attempt to modify the geometry, this Designer Elements program provides the following warning.

Given this, you have two ways to edit geometry, removing the links of the selected geometry or editing the 2D geometry by changing the parent 3D geometry using the Design Explorer and the Edit Objects dialog box.

- Select the object and choose ***Edit>Remove Links***. A warning appears reminding you that this command deletes all associative relationships and that you cannot undo this operation. Click OK and the object can now be changed.
- Select the object. Display the Design Explorer and open the history tree to show the parent geometry for the selected object. Double-click on that object to display the Edit Objects dialog box. Change the desired value and click OK. The 3D model and the 2D geometry updates. The graphic here shows a selected edge and the Design Explorer with the ACIS Solid parent.

Cobalt and Xenon Only

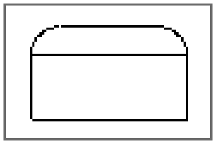


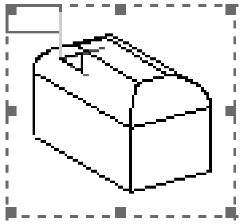
Editing View Characteristics

Changing the Scale of a View

You have two methods for changing the scale within a view:

- Choose the Properties command in the Drawing View menu and change the scale value within the dialog box.
- Select the drawing view boundaries by dragging a selection fence around the view rather than clicking on the view. Choose ***Window>Edit Objects*** and in the Geometry page, enter a new scale. Clicking the view activates the view but does not select just the view boundaries. The left graphic here shows the selected view boundary. The right graphic shows an activated view.



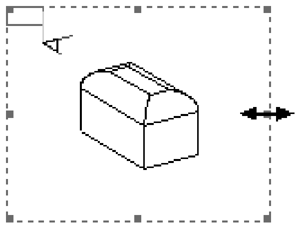


Select all drawing view boundaries in a layout by using the Selection Mask.

Resizing a Drawing View

You have three methods for resizing a drawing view:

- Dragging the view edge: Select the view and place the pointer on a control point of the edge you want to move. The pointer becomes a two directional arrow. Drag the edge to a new location. Place the pointer at a corner control point to resize two adjacent edges at the same time. The graphic shows the view control points and the directional arrow.



- Using the Properties command in the Drawing View menu
- Using the Frame to Extents command in the Drawing View menu

Dimensions and Drawing Views

If dimensions in the drawing view go outside the bounds of the view such that you can't see them, choose the Frame to Extent command in the Drawing View menu. You can also drag the view edge manually to completely display the dimension. See Chapter 27 for information on the standard dimension tools.

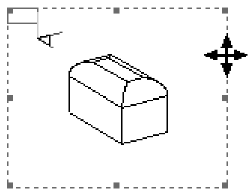
All dimensions placed in drawing views, go on the Sheet View layer rather than the Dimension layer. This enables you to turn off all other layers and still print the sheet with views containing dimensions.

Changing or Deleting View

The view orientation, scale and view properties can be changed through the Drawing View menu for each view. If you change the view orientation of a drawing view and cannot see all of the geometry in the view, choose the Center command or Frame to Extents command in the Drawing View menu. See the “Drawing View menu” section earlier in this chapter for more information.

You can delete a view through the Drawing View menu. You can delete all drawing views at once by selecting the Drawing view type in the Selection Mask, double-clicking on the Selection tool and pressing the BACKSPACE key (Windows) or the DELETE key (Macintosh).

Moving a Drawing View



You have two methods for moving a drawing view:

- Dragging the View: Select the view. Place the pointer over a view edge, not a control point. The pointer becomes the move symbol. Select the top or bottom view edge to drag the view vertically. Select the left or right view edge to drag the view horizontally.
- Using the Properties command in the Drawing View menu

Drawing Views and the Edit Menu

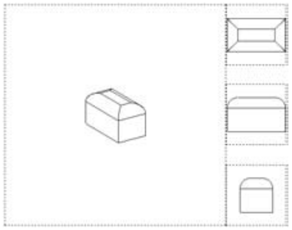
If you notice that the Edit menu name is red as you work in a Sheet view, your drawing contains some unresolved links. This occurs when you move geometry or make some other change. Choose Edit>Resolve Links. The Edit menu name becomes black again.

Layout Templates

This Designer Elements program provides you with 14 layout templates. These include the following:

1ViewA (B, C, D or E).vs	Displays Top view on the specified size sheet.
4ViewA (B, C, D or E).vs	Displays four views, Top, Front, Right Side and Trimetric on the specified size sheet.

Design4.vs	Displays four views, Trimetric, Top, Front and Right Side in drawing views and arranged in a design view layout (as shown here).
Draft4.vs	Displays four views, Top, Front, Right Side and Trimetric in drawing views.
FrontRight.vs	Displays two views, Front and Right in drawing views.
FrontTop.vs	Displays two views, Front and Top in drawing views
Top.vs	Displays the Top view in a drawing view.
Trimetric.vs	Displays the Trimetric view in a drawing view.



You create change the views used in particular layout by using the Properties command in the Drawing View menu (see an earlier section) or create a layout with your own views. See the next section.

Creating your own Layout Templates

You can create your own templates from scratch or by modifying an existing template.

Modifying an Existing Template

All templates provided are Designer Elements program files that can be opened like any other file.

1. Open one of the files in the Layouts folder.
2. Make whatever changes you desire. You can adjust the format, add text to the title block, and add or remove drawing views.
3. Save the file under another name in the Layouts folder.

   You can now select this template from the Layout pull-down menu in the Model to Sheet dialog box.

Creating a New Template

You can create a new template from scratch using a standard Designer Elements program file.

1. Open a new file.
2. Set the view and plane to Top.

3. Create the title block and border for the layout and add text if desired.
4. Using the Drawing View tool, add drawing views.
5. Save the file in the Layouts folder.

   You can now select this template from the Layout pull-down menu in the Model to Sheet dialog box.

Printing the Layout

If you want to print your layout with the views, turn off all layers other than the Sheet View layer. Make sure your Page Setup (Windows) or Print Setup (Macintosh) matches the layout.

Page Setup and Printing

You've created all your geometry using the wireframe, surface and solids tools, conducted various operations on it such as extruding, sweeping and blending and added some basic annotation to your drawing.

All during your design process, you did not have to worry about scaling or your paper size. Now that you're ready to print a hard copy, you'll need to set up the page according to a scale, paper format, orientation and your printer specifications.

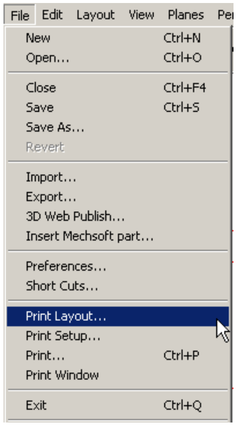
This Designer Elements program prints and plots on most printers and plotters supported by your computer. After you have followed the manufacturer's instructions for installing and setting up the printer or plotter, you can set up your page size as needed.

The following topics are covered:

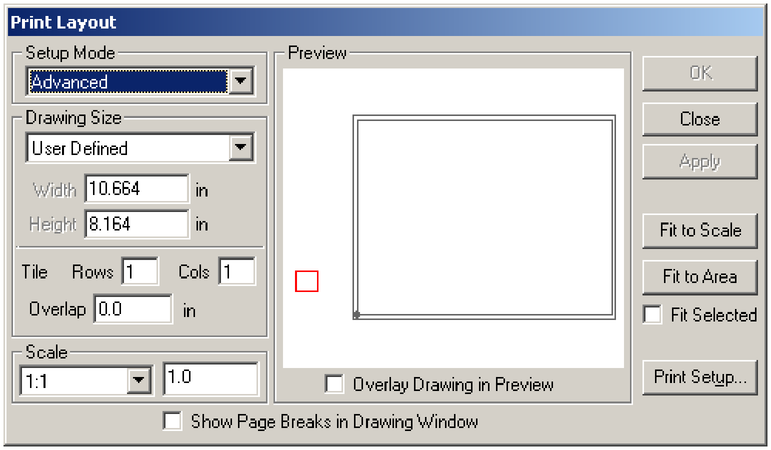
- Drawing Size
- Printing a Drawing
- Print Window

Print Layout Command

The *Print Layout* command, located in the File menu, displays the dialog box which allows you to set the page size, scale and other options.



When you select the command the Drawing Size dialog box appears. The graphic here shows the *Advanced* Setup mode.



The Drawing Size dialog box includes the standard buttons: OK, Close and Apply.

OK	Click this button to accept all changes and close the dialog box
Close	Click this button to ignore any changes made since you last clicked Apply and close the dialog box.
Apply	Click this button to accept and register all entries made in the dialog box without closing it.

The Drawing Size dialog box includes the following sections: *Setup Mode*, *Drawing Size*, *Scale*, *Preview* and *Utility Controls*.

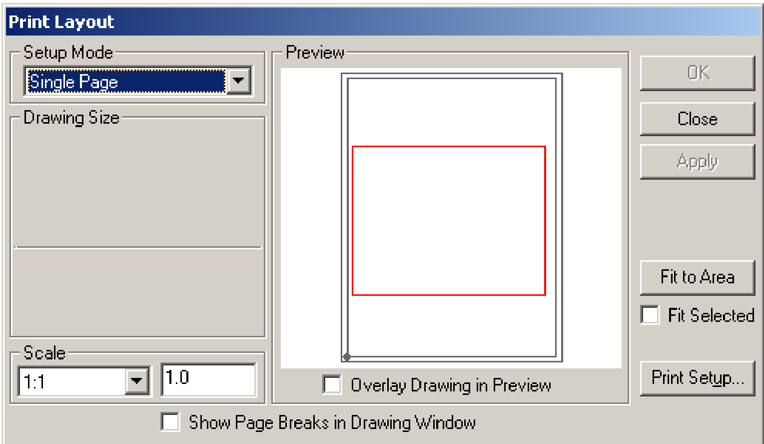
Setup Mode Section

The Drawing Size dialog box supports four setup modes; *Single Page*, *Height* and *Width*, *Rows and Columns* and *Advanced*.

For all setup modes, a drawing frame displays in the Preview window. The outer drawing frame represents the physical page size. The inner drawing frame represents the printable page area. The page settings are obtained from the current printer settings. Changed the printer settings by choosing ***File>Print Setup*** (Windows) or ***Page Setup*** (Macintosh).

Single Page Mode

The *Single Page* mode is the simplest printing mode. It is the best mode to use when printing to a large format plotter or when you need a quick single page plot. As the printer settings are changed, the drawing frame updates to conform to the new settings. Selecting this mode displays the most basic Drawing Size dialog box.

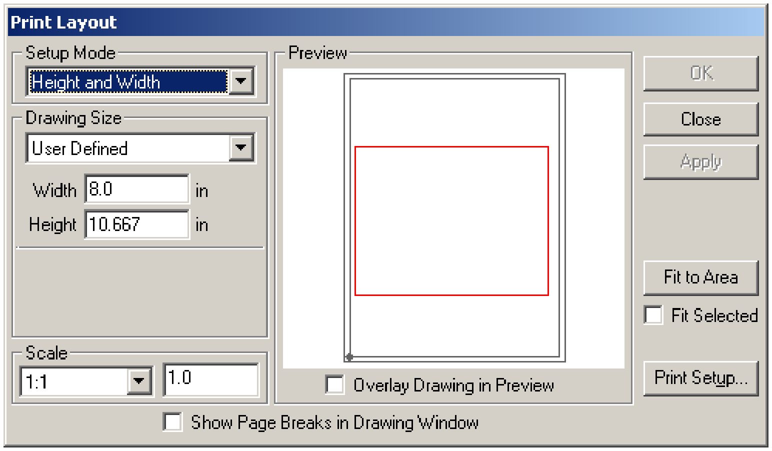


You can set the plot scale by making a selection from the Scale drop down list or by changing the value in the Scale data field.

Use the Fit to Area button to automatically compute the scale that will fit the drawing objects to the printable area.

Height and Width Mode

The *Height and Width* mode is used to generate large standard or user defined plot sizes when using small format print devices (e.g. 8.5 x 11 laser or inkjet printer). Selecting this mode displays the format area of the Drawing Size section.

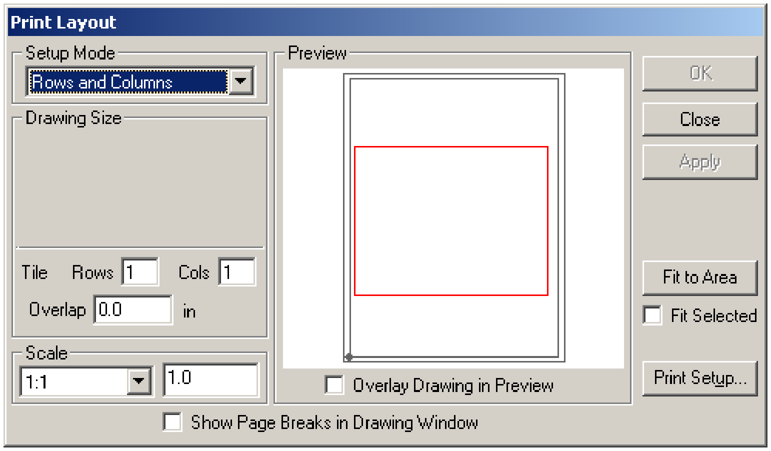


For drawing sizes larger than what your printing device allows, the drawing is tiled and can later be assembled into the large format plot. The *Preview* window displays the page tile edges within the drawing frame automatically.

You can specify a standard drawing size or a custom drawing size. You can set the scale in the *Scale* section or automatically compute the scale using the Fit to Area button.

Rows and Columns Mode

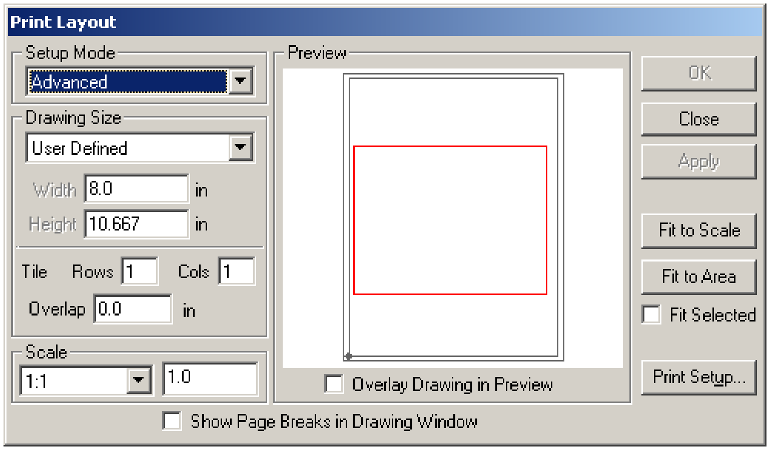
The *Rows and Columns* mode is used to force whole pages to be used for tiled plots. Unlike the *Height and Width* mode, this mode will use all the printable area available for the plot. However, the plot will not be a standard size. Selecting this mode displays the *Tile* area of the *Drawing Size* section.



You can specify the tile rows and columns by changing the values in the *Rows* and *Cols* data fields. The Overlap data field controls the how tile pages will overlap. The overlap region helps align the pages when assembling the final plot.

Advanced Mode

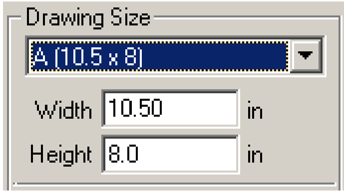
The *Advanced* mode provides access to all height, width, rows and columns plot settings. This mode permits complete control over all aspects of tiling. Selecting this mode displays the entire Drawing Size section.



You can specify any of the listed elements.

Drawing Size Section

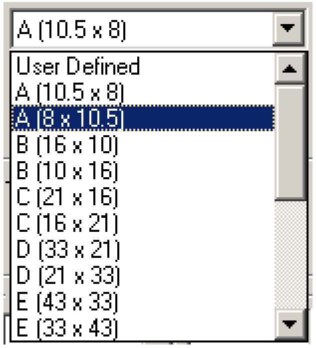
This section contains the drawing format sizes available and the height and width of the selected format size.



Tech Note:

When you choose a drawing format size, be sure that you set the page orientation for your printer to the same orientation selected in the Drawing Size dialog box. Choose ***File>Print Setup*** or ***Page Setup*** to display the Printer dialog box to check your current page orientation.

The format size field includes a pull-down menu listing all of the formats and their sizes.



You can choose any of the predefined sizes, A, B, C, D, E, F, G, H, J and K. Each format size includes a listing for portrait and landscape orientation.

The units (inches or mm) are determined by your preference setting. When you select one of these predefined formats, the size is displayed in the *Width* and *Height* fields.

You also have the ability to set your own drawing size by selecting the *User Defined* option in the list. When you choose this option, you enter the size in the *Width* and *Height* fields.

The drawing sizes displayed in the pull-down list are contained in the *DrawSize.ini* file in the Environ folder within the Designer Elements program folder. This file can be edited but keep it mind it may change or be overwritten by future Designer Elements program installations. If you do choose to edit the file, save the original version under another name before doing so. Then you will have a copy in case you want to return to the default sizes.

If the page size is larger than the size supported by your printer, values are automatically entered in the *Tile* area to accommodate the drawing and appears as such in the *Preview* window. (Choosing the *Advanced* mode shows the tiling specifics.) See the next section for more information on tiling.

Before choosing your size, you must determine what size format your printer or plotter can support.

Drawing size and page tiling are synchronized based on the scale. Values entered in the drawing size fields affect those in the page tiling and vice versa. *Height* affects

Rows and *Width* affects *Cols*. The last field in which you enter values controls the drawing dimension, represented by the activated field name. The associated field name is unavailable.

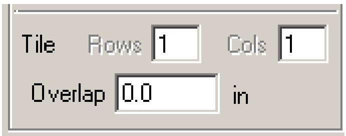
Setting the Format Size

1. Choose ***File>Print Layout***.
2. Choose the *Height and Width* mode from the pull-down menu.
3. Display the pull-down menu for the drawing format size.
4. Select the desired size.

   The size is displayed in the *Width* and *Height* fields.
   If you selected the *User Defined* format size, enter the size in the *Width* and *Height* fields.

Page Tiling

This *Tile* area of the Drawing Size section allows you to set up your file so you can print a larger drawing in tiled sheets.



This area includes these elements:

Rows Represent the number used to print your file, based on the size supported by your printer driver and the drawing scale. A value automatically appears in this field when you select a drawing size larger than that which your printer supports.

If the exact drawing size is unimportant, just specify the number of rows. Since the number is synchronized with the *Height*, entering a different value changes the height of your drawing.

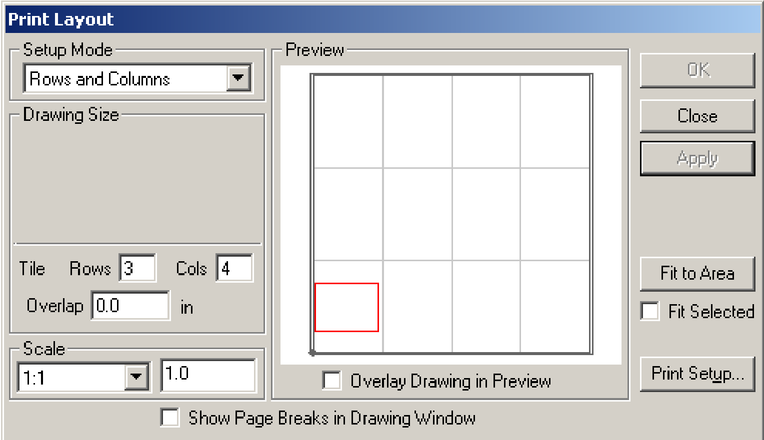
Cols Represent the number of columns used to print your file, based on the size supported by your printer driver and the height of your drawing. A value automatically enters in this field when you select a drawing size.

If the exact drawing size is unimportant, just specify a number of columns. Since the number of columns is synchronized with the *Width*, entering a different value changes the width of your drawing.

Overlap

When tiling, you can specify a page overlap (between 0 and .75 inch or 20 mm). The overlap determines how much of the geometry repeats on the right and top area of each tile page. The overlap region can be used to align the tiles when joining the pages.

This graphic shows an example of tiling with three rows and four columns.



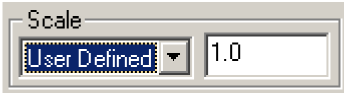
Tiling operates independently of the page orientation.

Setting the Page Tiling

1. Choose ***File>Print Layout***.
2. Choose the *Rows and Columns* mode from the pull-down menu.
3. Enter the desired values in the *Rows* and *Cols* fields. The drawing size fields adjust accordingly.
4. Specify the overlap for the tiled pages. The units are determined by your preferences setting.

Scale

This section allows you to specify the scale of your drawing. You can select a standard scale from the pull-down list or set your own scale in the data field.



Scale Options

The pull-down list provides you with these scaling options: User Defined, 5:1, 4:1, 3:1, 2:1, 1:1, 1:2, 1:3, 1:4 and 1:5.

Choosing one of the standard scales enters a value in the data field. A 5:1 scale, enters 5.0 in the field. A 1:5 scale enters a 0.20 in the field.

If you want to specify your own scale, enter the value in the data field. The scale name changes to User Defined, regardless of the scale entered.

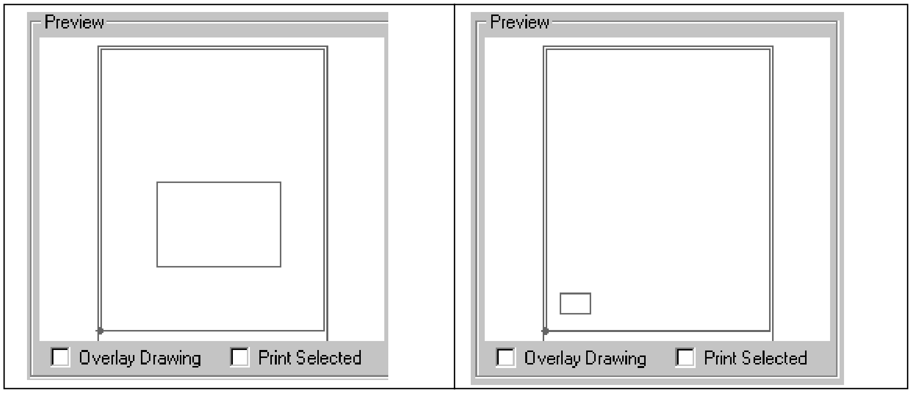
The drawing scales displayed in the pull-down list are contained in the *Draw-Size.ini* file in the Environ folder within the Designer Elements program folder. This file can be edited but keep it mind it may change or be overwritten by future Designer Elements program installations. If you do choose to edit the file, save the original version under another name before doing so. Then you will have a copy in case you want to return to the default scales.

Setting the Scale

1. Choose ***File>Print Layout***.
2. Display the pull-down list for the scale.
3. Select the desired scale. The scale appear in the edit field.

   If you selected the User Defined scale option, enter the scale in the data field. The left graphic below shows the *Preview* window of a rectangle at a scale of 2:1. The right graphic shows the *Preview* window of an rectangle at a scale of

1:2.



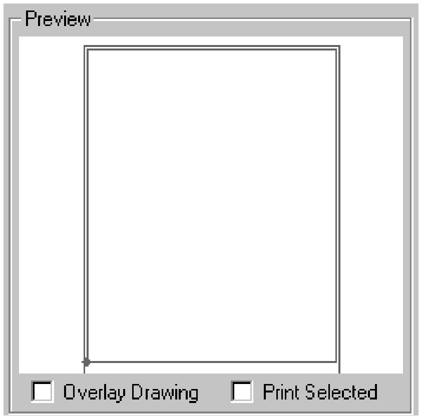
Scaling does not change the actual dimensions of the part. Verify this by selecting an object and choosing ***Window>Edit Objects***.

Preview Section

To assist you in choosing the correct format size for your drawing, This Designer Elements program includes the *Preview* section containing the Preview window and two check boxes, O*verlay Drawing* and *Print Selected*.

Preview Window

A preview of the drawing appears in this section of the Drawing Size dialog box.



The outer drawing frame represents the physical page size. The inner drawing frame represents the printable page area. The page settings are obtained from the current printer settings. If you change the drawing size, the *Preview* window still displays the last drawing size selected until you click Apply.

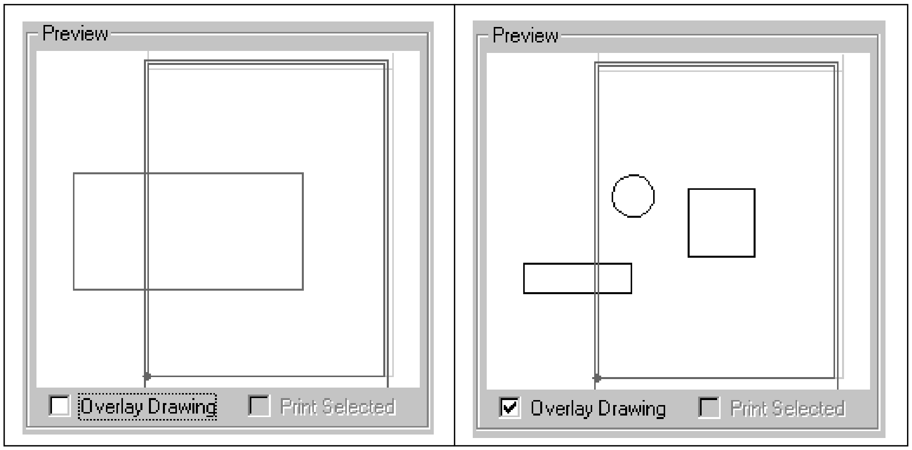
Only objects or part of objects that lie within the page bounds are printed.

The *Preview* window displays a rectangle, representing the drawing or the actual geometry. This display is determined by the *Overlay Drawing* setting. See the next section for more details.

The Axis displays in the *Preview* window only if it's displayed in your drawing area. The Axis does not print.

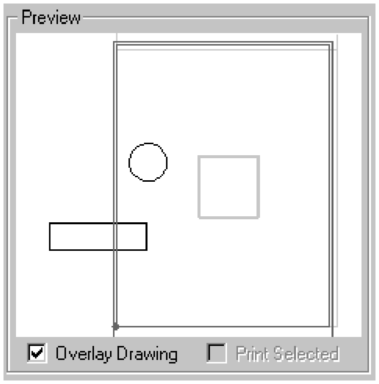
Overlay Drawing

This check box determines how your geometry appears in the *Preview* window. When the box is not checked, a red rectangle displays, representing the drawing area used by the objects (the graphic on the left below). When the box is checked, the actual geometry displays (the right graphic).



Print Selected

This check box allows you to specify which objects within the drawing bounds get printed. When the check box is empty, all geometry within the page bounds print. When the box is checked, only selected geometry within the page bounds prints and the view window zooms in on that area. In the graphic here, only the square prints.



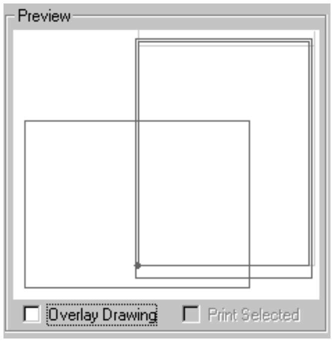
In the graphic, all of the geometry displays. If you deactivate the *Overlay Drawing* check box, only the selected rectangle displays in the *Preview* window.

This check box only becomes available when you select geometry.

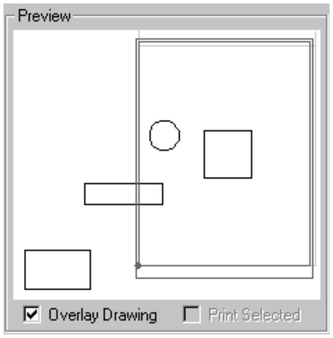
Preview Section Example

An example of using the option in this area might clarify how the *Preview* window and check boxes interrelate.

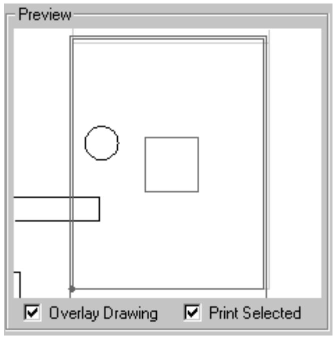
1. Draw some geometry.
2. Choose ***File>Print Layout***. The Drawing Size dialog box displays. The *Preview* window shows a red rectangle representing the object area.



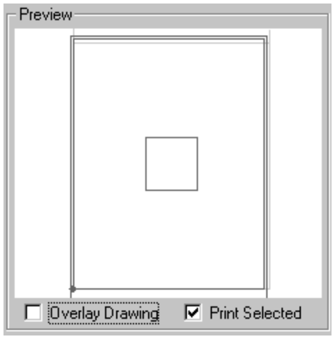
3. Check the *Overlay Drawing* box to display the actual geometry.



4. Click OK in the dialog box to close it.
5. Select an object within the page bounds.
6. Choose ***File>Print Layout*** to display the dialog box again.
7. Check the *Print Selected* box. The *Preview* window zooms in on selected object.



8. Click the *Overlay Drawing* check box to remove the check mark. Only the selected geometry displays.

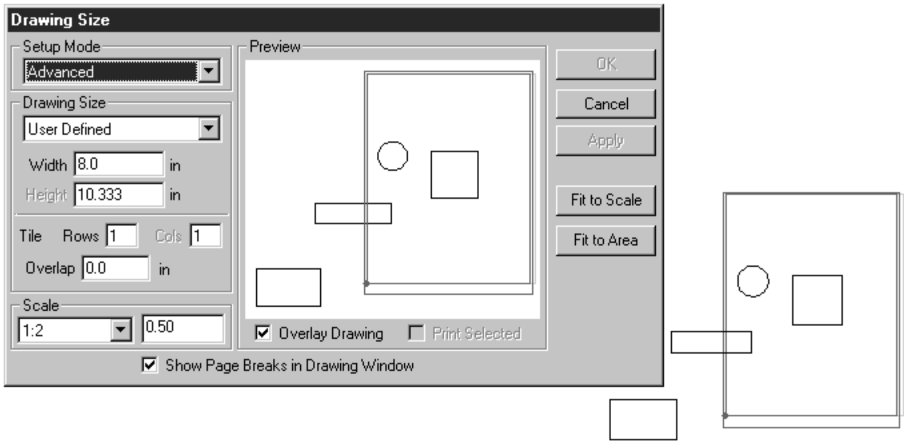


Utility Controls

The Drawing Size dialog box contains utility controls for setting up your drawing. These include the *Show Page Breaks in Drawing Window* check box, the Fit to Scale button and the Fit to Area button.

Show Page Breaks in Drawing Window

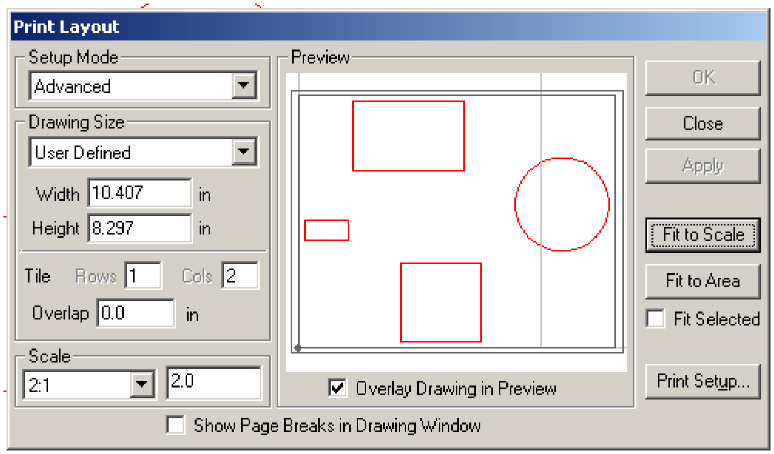
This check box allows you to specify whether to display the page breaks/boundaries in your drawing area. When this box is checked, the page bounds display.



When you click OK and close the dialog box, you can move the page boundaries. Place the cursor over the marker at the lower left corner of the page boundaries. The cursor becomes the move symbol (shown to the right). Drag the page boundaries to the new location. See the “Move the Print Boundaries” section at the end of this chapter for more information.

Fit to Scale

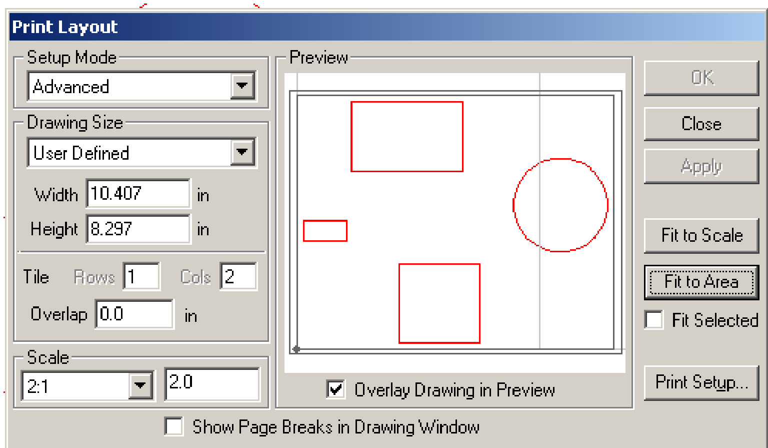
Clicking this button uses the current scale value and automatically changes the height, width and page boundaries to fit the geometry.



This button only displays with the *Advanced* mode.

Fit to Area

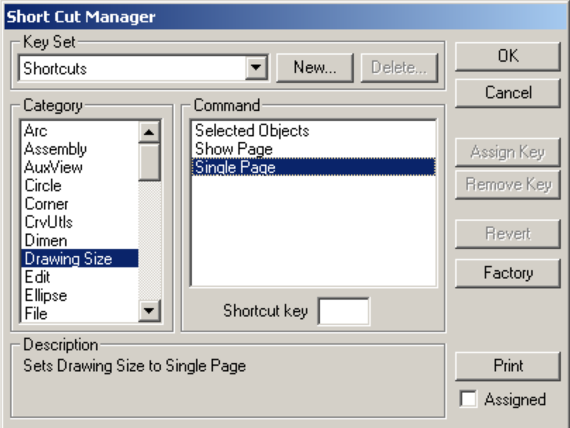
Clicking this button automatically computes the scale and page boundaries to fit the geometry to the entire tiled plot's printable area.



For a single page to print, set both *Rows* and *Cols* to 1 and press the Fit to Area button.

Drawing Size and Short Cut Key

A new short cut key, Single Page, was added to Drawing Size category in the Short Cut Manager.



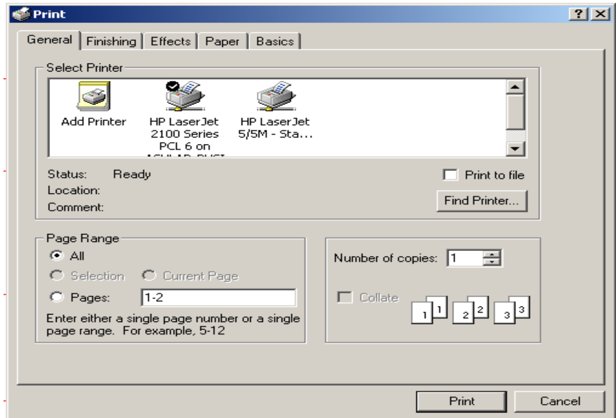
Using this key automatically chooses the *Single Page* mode, activates the *Fit to Area* function and *Shows Page Breaks*. You can assign whatever short cut key you desire.

Printing a Drawing

After you have set up your page you are ready to print. Choose ***File>Print Setup*** (Windows) or ***Page Setup*** (Macintosh).

Print Setup (Windows); Page Setup (Macintosh)

Choosing this command in the File menu displays your printer setup window.



Choose the necessary settings for paper size and page orientation to agree with your settings in the Drawing Size dialog box. Click OK to save settings. See your printer manual for information about setting your printer options.

Print Command - CTRL+P (Windows); z +P (Macintosh)

This command in the File menu prints or plots the current document as specified in the Drawing Size dialog box.

The area printed or plotted is the portion that fits on the page size specified in the Drawing Size dialog box. Choose ***File>Print Layout*** to scale your drawing to the appropriate size and reposition the print/plot region.

You can specify tiling (printing on several pages to be pasted together) by choosing ***File>Print Layout>Advanced***.

Print to a File

You can print to a file rather than to a plotter or printer. In that way, you don't have to have a plotter attached to your computer. Someone else can plot the drawing without having this Designer Elements program on the plotter's computer. The type of plotter you choose when setting up the page determines the format of the plot file.

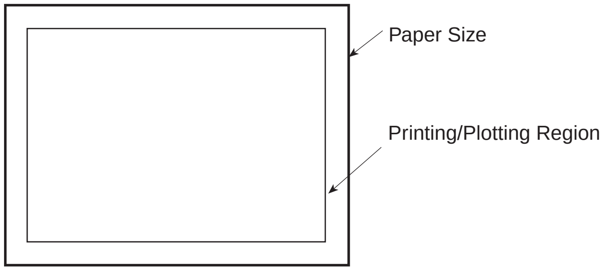
If you choose a PostScript printer, the file format is Encapsulated PostScript; use the HPGL language when you select Hewlett Packard plotters. The computer that finally plots the file must have an application compatible with the file format of your printer or plotter.

Plotter Font

When you are using a plotter, you should specify the Plotter font for the text and dimensions on your drawing. You can also generate special characters and accents as described in Appendix B.

Printing/Plotting Region

When you choose ***File>Print*** only the geometry within the page boundaries prints. You can view those boundaries by choosing ***File>Print Layout***. If your printer does not support the size, gray boundary lines are displayed in the window, representing the boundaries and the tiling feature activates. For all printers and plotters, the plotting region is smaller than the actual page size because most printers and plotters cannot plot to the edge of the paper, allowing room for the margins.



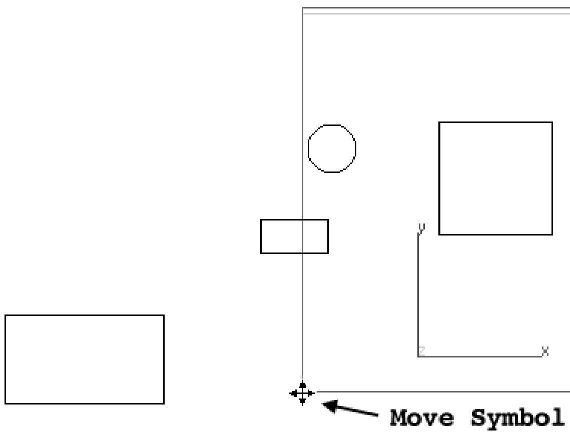
Vellum displays the Printing/Plotting region

The size of this region is based on the paper size and the printer or plotter driver currently selected.

Moving the Print Boundaries

If the geometry you want to print is not contained within the page boundaries you can move the page boundaries.

1. Choose ***File>Print Layout***.
2. Select the *Show Page Breaks in Drawing Window* check box.
3. Click OK. The dialog box closes and the page boundaries are displayed in your drawing.
4. Place the cursor over the marker at the lower left corner. It becomes the *Move* symbol.
5. Drag the boundaries to the new location.



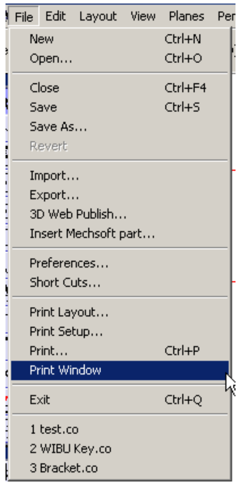
Printing and Rendering

This Designer Elements program prints both wireframe and rendered geometry. When you want to print rendered geometry, set the *Static Render* option in the Render Options dialog box to the desired mode.

If your printer supports color, *Static Render* must be set to Flat or Gouraud.

Print Window

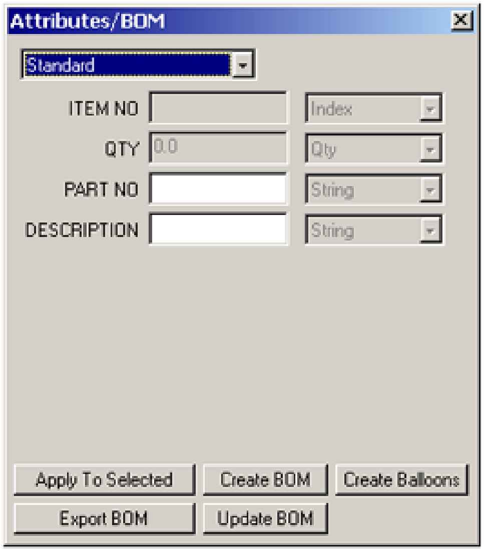
The *Print Window* command in the File menu copies the image within the drawing screen and sends it to the printer.



Attributes and Bill of Materials

User Attributes and Bill of Materials provide a means to apply custom attributes to entities and display that data in a table or export it to an external application such as Microsoft Excel.

This tool is located in the Window menu (***Window>Attributes and BOM***). After selecting this tool the following dialog box appears:



The Bill of Materials dialog box contains a drop down menu of predefined attribute templetes and five button options.

The predefined templets are:

Material	Assigns a material to an entity.
Price	Assigns a price to an entity.
Standard	Assigns a part number and description to an entity.
Stock Size	Assigns part number, stock size and description to an entity.
Vendor No.	Assigns a part number, a vendor number and a description to an entity.
Perimeter	Calculates the 2D perimeter for curves.
Volume	Calculates the volume of a solid.
Weight	Calculates the weight of a solid using the material assigned in the ***Verify>Mass Properties*** command.
Area	Calculates the surface area of any polygon, surface or solid.
Area 2D	Calculates the 2D area and centroid properties for curves.
Mass Properties	Calculates the Mass Properties for a solid using the ***Verify>Mass Properties*** command.

The first five attribute templates require the user to supply all of the information associated with the attribute. The latter six automatically extract attribute information from the entity.

The five buttons on the Attributes/BOM dialog box have the following functions:

Apply To Selected	This options applies the current BOM attribute to the selected entities.
Create BOM	This option creates a Bill of Material Table using the attributes defined by the pull down menu. The user is

prompted to enter text height, colun width and item order.

ITEM NO	PART NO	DESCRIPTION	CG-X	CG-Y	AREA
1		POLYGON_33	-5.187963	1.490709	2.082140
2		ELLIPSE_34	-5.379836	-0.656798	1.245559
3		POLYGON_35	-5.379836	-2.472215	1.530388

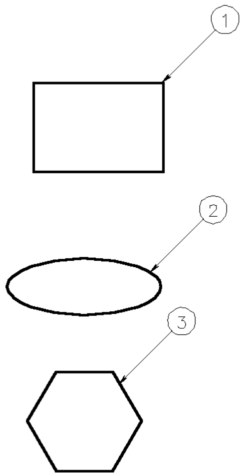
ITEM NO	PART NO	DESCRIPTION	CG-X	CG-Y	AREA
3		POLYGON_35	-5.379836	-2.472215	1.530388
2		ELLIPSE_34	-5.379836	-0.656798	1.245559
1		POLYGON_33	-5.187963	1.490709	2.082140

Graphic 2 equals Item Numbers Up

Tip:
The Bill of Materials function does not incorporate the use of Copy/Cut and Paste.

Create Balloons

This option adds balloon item dimensions to geometry. This option requires an item number type be used with the attribute. All of the supplied attribute files have item numbers.



ITEM NO	PART NO	DESCRIPTION	CG-X	CG-Y	AREA
1		POLYGON_33	-5.187963	1.490709	2.082140
2		ELLIPSE_34	-5.379836	-0.656798	1.245559
3		POLYGON_35	-5.379836	-2.472215	1.530388

Export BOM

This options creates a comma-delimited file that can be read into Microsoft Excell or other similar spreadsheet programs.

Update BOM

This option updates the BOM table and the corresponding balloons.

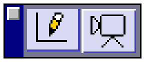
BOM User Defined Templates

You can create user-defined templates by creating an attribute definition file. Attribute definition files are located in the BOM folder of the install directory. The first line of the file contains the attribute name. This is the name that will appear in the attribute pull-down menu. The next lines contains two columns, the first column contains the attribute definition string, and the second column the attribute type. Commas separates each column

The following attribute types are supported:

Float	User-assigned signed decimal value.
Integer	User-assigned signed integer value
String	User-assigned character string
Area	Calculates the entity area and assign.
Perimeter	Calculates the entity perimeter and assign.
Volume	Calculates the entity volume and assign.
Weight	Calculates the entity weight and assign.
Qty	Counts the number of occurrences this attribute is used.
Index	Index used to refer to the attribute in the BOM table or balloon callout.
Name	Extracts the name from the entity.
CG-X	Calculates the CG-X value from an entity.
CG-Y	Calculates the CG-Y value from an entity.
CG-Z	Calculates the CG-Z value from an entity.
Sum	Sums the value the previous attribute.
Material	Extracts the material type from the entity.

Parametric Constraints (Cobalt Only)



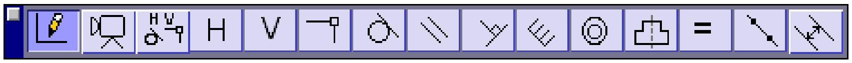
Cobalt support 2D profile dimensional constraints. This feature provides a mechanism for dimensioning entities to define distances and angles in order to establish geometric constraints between curves.

To access the Constraint tools and enter a 2D Sketch Mode go to the Window menu and choose Constraints.

Sketch Mode

To enter the 2D Sketch Mode in the Parametric Constraint tool you must first choose the sketch tool.

Clicking on this tool expands the tool palette, showing the tools available while in sketch mode.



By entering the sketch mode the application knows to adjust the user interface in such a manner that creating 2D sketches is intuitive. The sketch mode does the following automatically when creating a new sketch or modifying an existing sketch:

1. Hides tool palettes such as surface and solid modeling which have no meaning in the 2D sketch environment.
2. Adjusts the drafting assistant to always snap into the sketch plane and see only snaps in the sketch plane. This is accomplished by turning on the "Work Plane" and "Plane Only" options in the Snaps dialog box.
3. Turns on Auto Constraints. Auto constraints will automatically create geometric relationships (coincident, tangent, concentric, perpendicular) as you create and modify geometry. You can turn this off in sketch mode through the popup menu activated with right-click (Control + click on Mac).
4. Creates a set of layers for the sketch that includes the container layer, "Sketch 1" and sub layers that include construction, profile, constraints, and dimensions. Note the geometry in the construction layer is ignored if the sketch is used in profile-based operations such as skinning, sweeping, lathing, or extruding.
5. Show Dimming is turned on. This feature is useful when working on faces of solids where edges not in the sketch plane are dimmed. In addition, if you modify a sketch

on the face of a solid, the part is rolled back to the point where the sketch was original created.

Exiting Sketch Mode

Exiting the sketch will restore the above settings back to the previous settings before the sketch was entered. One other advantage of the sketch mode is that it allows the ability to temporarily suspend the regeneration of a dependent feature. This means that while in sketch mode you can perform a series of operations such as adding and removing curves to the sketch that would otherwise invalidate downstream operations. However, when exiting the sketch mode, you must resolve a valid profile such that dependent features can then be updated.

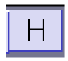
The tools available in Sketch Mode are listed below.

Sketch Tool Creates or Modifies an existing sketch.

Auto Constraints Tool The auto constraint tool automatically applies constraints to curves and dimensions selected by the user.

This tool is particularly useful when working with data created outside the sketch tool. The auto constraints tool will add the following constraints to the selected geometry: Horizontal, Vertical, Tangent, Concentric and Coincident.

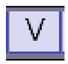
Horizontal Constraint Tool Adds a horizontal constraint to a line. You can select one or more curves to apply a horizontal constraint. Horizontal is defined by the work plane x-axis. The two chosen points will have the same x value when completed.



In this tool, note the message line:

When applying a horizontal constraint, pressing the CONTOL key (OPTION on Mac) will give the user the option to align the geometry horizontally to a specific point on another line. The two chosen points will, when completed, have the same x value.

Vertical Constraint Tool Adds a vertical constraint to a line.



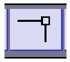
You can select one or more curves to apply a vertical constraint. Vertical is defined by the work plane y-axis.

In this tool, note the message line:

Vertical Constraint: Pick line for vertical constraint.[Ctrl = Share Y position][Shift = Extend]

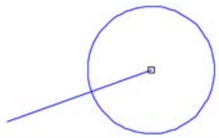
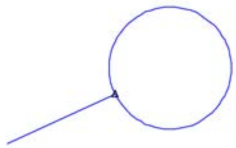
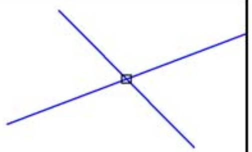
When applying a vertical constraint, pressing the CONTROL key (OPTION on Mac) will give the user the option to align the geometry vertically to a specific point on another line. The two chosen points will have the same y value when completed.

Coincident Constraint Tool This tool adds a coincident constraint between two object positions.



Objects that are recognized for coincident constraints include lines, arcs, circles, ellipses, splines, points, and edges of solids. The point of coincidence to the object is automatically determined by use of the drafting assistant. Referencing end points, mid-points, vertex, centers, and point on are preserved. In the case of point on (point along curve) the coincidence constraint may be anywhere along the curve. All others are fixed to specific locations. The floating coincident constraint is represented by a small triangle symbol whereas a fixed constraint is a small rectangle about the two shared points.

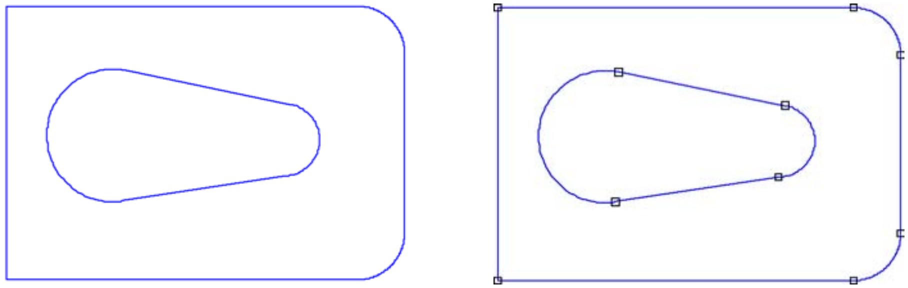
Some examples of coincident constraints:

End Point/End point	
End point/Center Point	
Point On/End Point	
Midpoint/Midpoint	

The coincident constraint tool allows for several means of attaching relationships. The first method is to follow the user prompts:

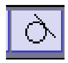
1. Pick the first curve for coincident constraint.
2. Pick the position along the first curve using the drafting assistant.
3. Pick the second curve for the coincident constraint.
4. Pick the position along the second curve using the drafting assistant.

The second method for creating coincident constraints is to box select two or more curves at the first prompt. In this case all curve end points that lie within 0.001 inches will be applied a coincident constraint.

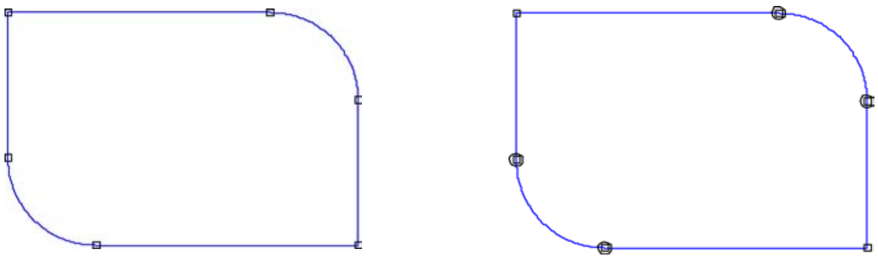


In addition, the coincident constraint tool allows for pre selections. If any curves are pre-selected before you hit the tool palette icon, coincident relations are automatically applied to the selected objects.

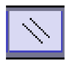
Tangent Constraint Tool This tool creates a tangent constraint between two or more curves.



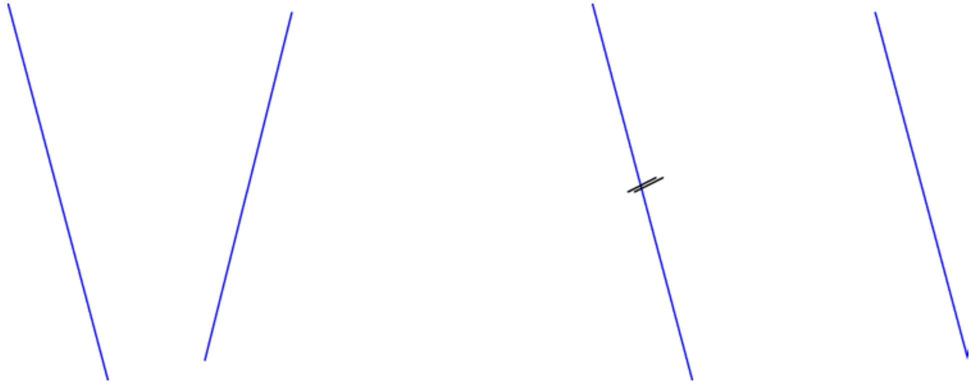
To use the tangent constraint tool simply select the curves you would like to apply a tangent constraint. When selecting more than 2 curves, the tangent constraint tool only applies a constraint between curves that are already tangent within 1 degree. This constraint draws a circle to represent the existence of a tangent constraint.



Parallel Constraint Tool The parallel constraint tool adds a parallel constraint between two lines.

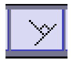


Only lines can be used in this tool, arcs, circles, ellipses, splines are ignored. The constraint symbology is two small parallel lines.

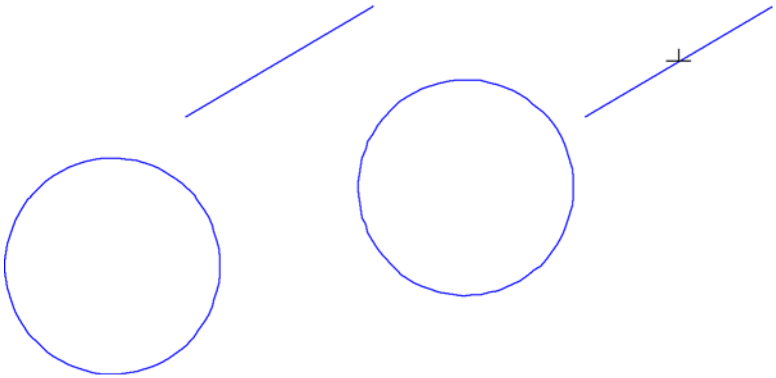


Note: The solver determines which line to move based on a set of rules involving the curve type, other curve relations and minimization of geometry movement. Therefore the order in which you pick the curves is irrelevant as to which curve is actually moved.

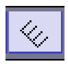
Perpendicular Constraint Tool The perpendicular constraint tool creates a 90° angle between a line and another curve.



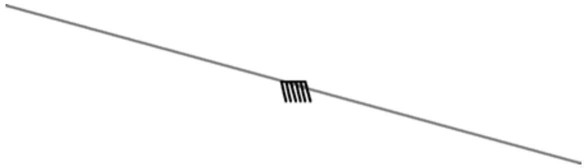
In the example below a coincident constraint is needed to attach the line to the circle. As in other constraints the entity that moves is independent of selection order.



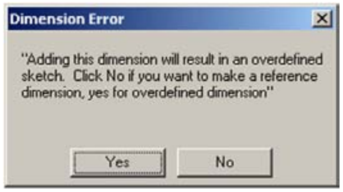
Fixed Constraint Tool The fixed constraint tool locks the entity from being moved by the solver.



The user is allowed to move the entity with the move or translate tools. A fixed object uses the fixed display color which is by default gray and whose symbology consists of a collection of slanted lines.

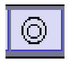


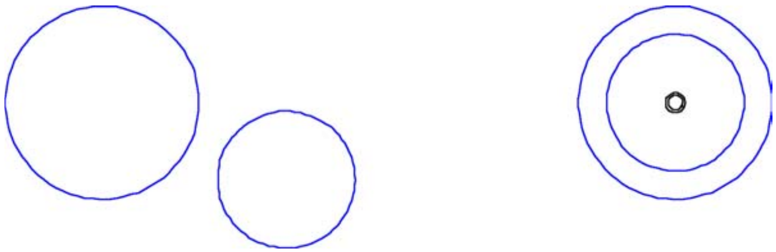
Note: Attempting to dimension a fixed entity will result in the display of the dimension error dialog box.



This error message implies that the position, direction, and length are fixed due to the fix constraint previously applied.

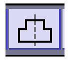
Concentric Constraint Tool The concentric constraint tool creates a constraint that forces circles to share the same center point.

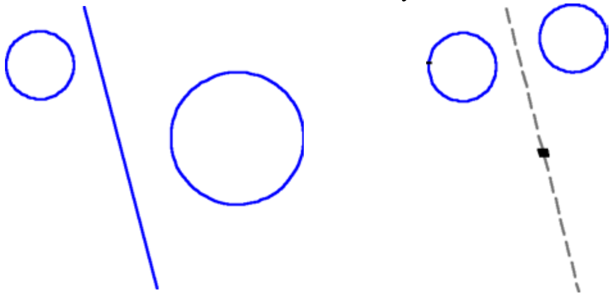




This concentric tool recognizes circles and points as valid selectable entities.

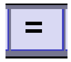
Symmetric Constraint Tool This tool creates a symmetric constraint between entities of asimilar type and symmetry line.

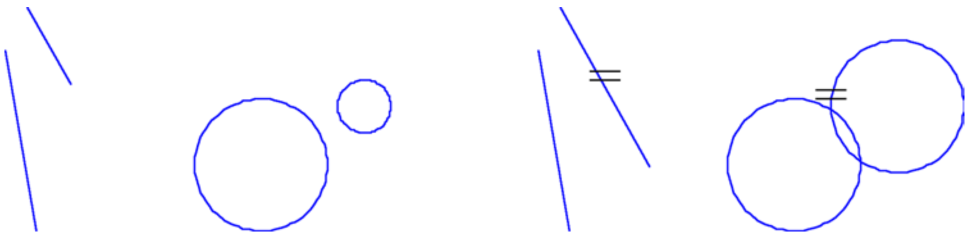




Note: Use the mirror tool in the transformation tool palette while in the sketch mode to automaticaly apply mirror constraints at the time the mirror operation is performed.

Equal Constraint Tool The equal constraint tool applies an equal distant or radius constraint between two entities.

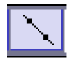


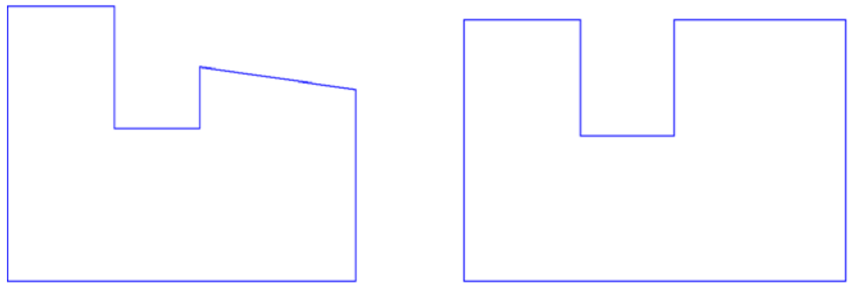


This tool works only with lines or circles. When selecting two lines, the lengths are forced to be the same for the two entities. In the case of circles, the same radius value is applied between the selected circles. As with many constraints, this operation is independent of which curve is selected first due to the method in which the solver finds solutions.

Colinear Constraint Tool

The colinear constraint makes two lines colinear.

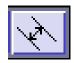


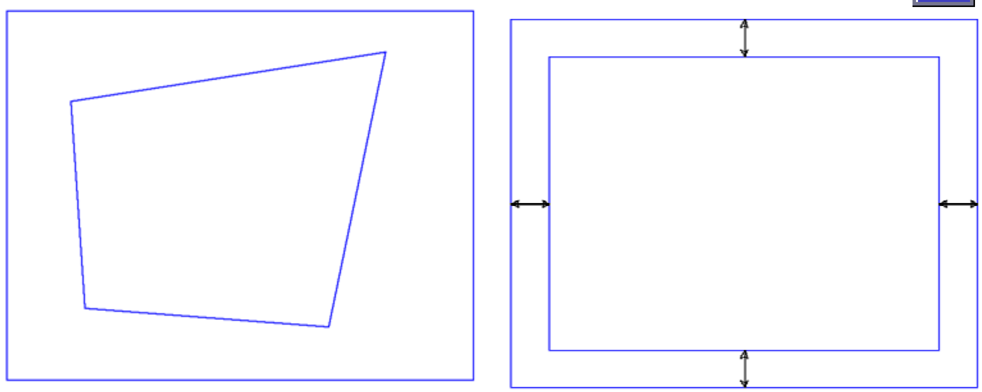


Colinear implies that the two resulting lines have the same direction and line within the same line. It does not imply anything regarding their lengths.

Offset Constraint Tool

This tool adds an offset constraint between two lines or two circles.

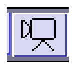




The offset distance is specified through the status line. In addition you can use the edit objects dialog to change an existing offset value. Click on the constraint symbol to display this edit page.

Animating Dimensions Tool

The animation tool will animate a sketch by modifying a dimension value through a range of values. To use the animate tool, select a dimension that was used within a sketch and then choose the Animate Constraint tool.



Note: If you animate a sketch while in sketch mode, only the sketch is updated. If you animate a sketch outside of the sketch mode, dependent surfaces and solids update accordingly.

DCM Animation Settings Dialog

Changing the settings

The DCM Animations Settings dialog box is used to control the behavior of a sketch dimension animation. The available animation settings are Start, End, Steps, Delay, Loop, and Rebound.

The Animation Settings dialog contains the following options:

Start Specifies the starting dimension value.

End Specifies the ending dimension value.

Steps Controls the number of intermediate steps to use when transitioning from the dimension Start value to the End value. For example, if Start is 1.0, End is 4.0, and Steps is 6, the dimension will animate with the values of 1.0, 1.5, 2.0, 2.5, 3.0, 3.5, and 4.0. The delta value is computed by: (End - Start) / Steps.

Delay Controls the number of seconds to wait between sketch dimension animation “frames”.

Loop If checked, the animation will continue to play indefinitely until stopped.

Rebound If checked, the dimension value will transition from Start to End in Step steps, and then continue by “rebounding” from End to Start in Step steps.

Running the Animation

Once the settings are made, select the dimension you want to animate. The animation will start automatically. You may control the execution of the animation using the VCR style buttons in the dialog box.

Deleting Constraints

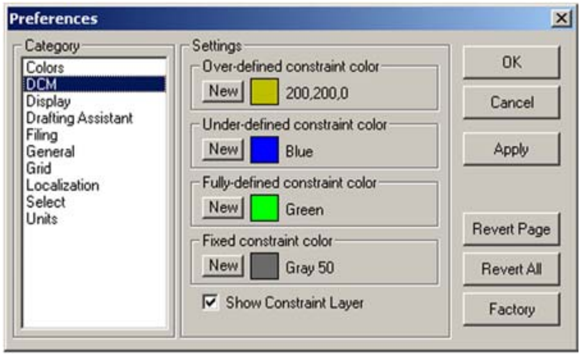
To delete any constraint, simply select the constraint symbol and hit the delete key or delete from the menu.

Verifying Constraint Relations

To confirm the entities used in a constraint relationship, simply move your cursor over the constraint symbol and pause for several seconds. After several seconds the entities involved in the constraint relationship will highlight in red.

Constraint Preferences

The Preferences dialog box under the File menu contains a category for DCM (Dimension and Constraint Management). The DCM Preference Settings includes options for setting:

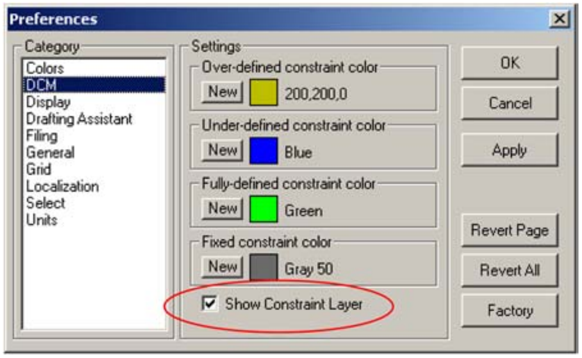


Note: Avoid using the select color as a constraint color to minimize confusion between a constraint color and when an entity is selected.

Over-Defined Constraint Color Any entity in a sketch that has too many constraints or creates an ambiguity for the solver will display in the Over-Defined Constraint color. It is best to resolve any over-defined sketch entities before exiting a sketch.

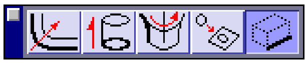
Under-Defined Constraint Color Entities that still have degrees of freedom remaining are displayed in the under-defined constraint color.

Fully Defined Constraint Color	Any entity whose position and size is fully defined is displayed in the fully defined constraint color.
Fixed Constraint Color	Entities that have a fixed constraint associated with their definition are displayed in this color.
Show Constraint Layer	When the show constraint layer check box is enabled, new sketches will automatically turn on the sketch layer. Sometimes complex sketches get visually complicated when all the constraint symbology is displayed.



Reference Edges

When in Sketch Mode, edges of geometry that are not on the sketch plane can be projected onto the plane. In sketch mode, simply select the Explode Edge tool:



Then click on the surface or solid edge you wish to project onto the sketch plane, and the exploded curve will appear on the sketch plane in the Fixed Constraint color. This color means that the object is associatively tied to another piece of geometry. Keep in mind that if you move the original geometry, the exploded curve will move as well.

Projecting a curve into a sketch

This is similar to exploding the edge of a surface or solid and projecting it onto the sketch plane, except that it will work for wireframe geometry. The procedure is the same as for Reference Edges. When in sketch mode, simply select the Explode Edge tool and choose the wireframe geometry you wish to project to the sketch plane. Once again, this geometry is associatively tied to the original curve, and is displayed in the Fixed Constraint color.

Dimension-Driven Geometry

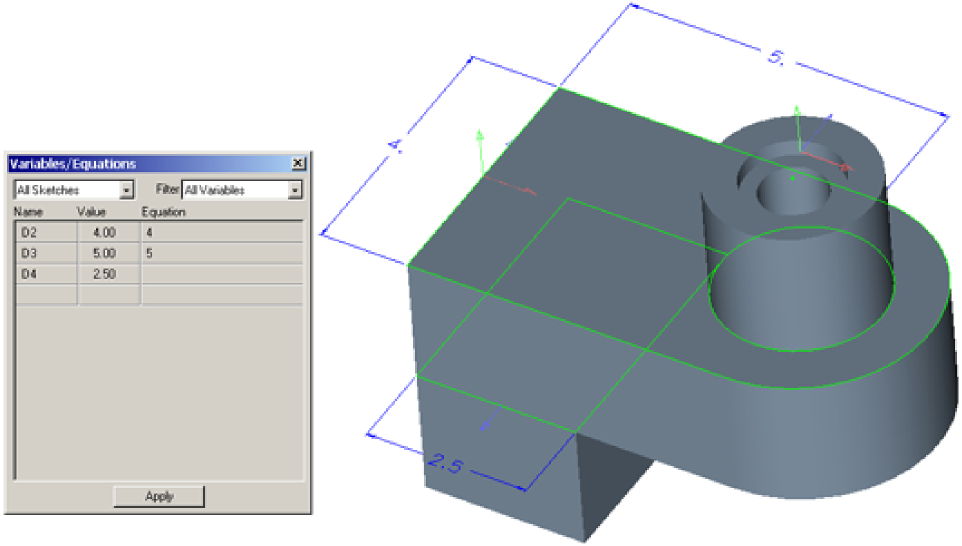
Dimensions created in the sketch mode are by default driving dimensions. This means that changing the dimensional value will force curves associated with the dimension to be updated to the new dimensional value. A dimension that is driven by the curve is called a reference dimension. Outside of the sketch mode, dimensions created are reference dimensions. To change a dimension from dimension-driving to dimension-driven, right click over the dimension.

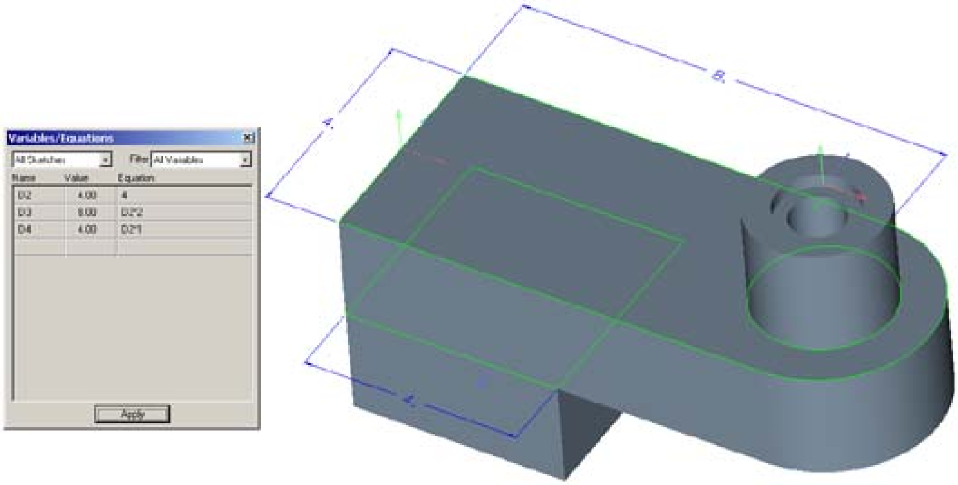
Exiting Sketch Mode

Once a sketch is created and constrained, to exit Sketch Mode simply click the blue Exit icon in the lower right-hand corner of the drawing window:

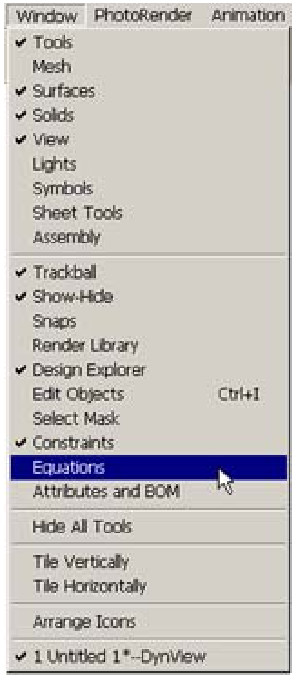
Variables and Equations

This Designer Elements program supports the assigning of variables and equations to dimensions applied while in the Constraints Sketch Mode. When you have geometry that has been constrained using equations is an easy way to manipulate the geometry, see the examples below.





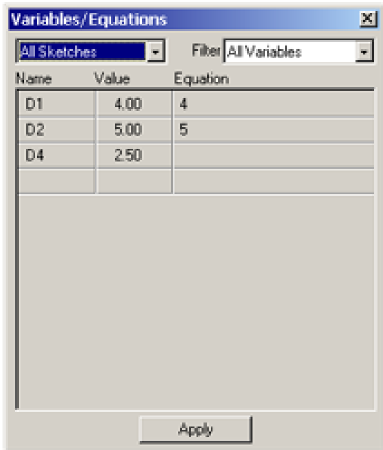
To open the Equations dialog box go to the Window menu and choose Equations (***Window>Equations***).



The equations dialog box supports all manner of mathamatical expressions. If you want to use one dimension as a reference to the other be sure to use its name from the name field as in the previous example. For a list of Mathematical operators that can be used see Appendix A.

Using Equations

Once you have completed your parametrically constrained part you may then edit the dimensions applied to the part using the equations dialog box, seen below.



The Equations dialog contains the following options:

All Sketches This pull down menu contains the list of every sketch within the file.

Filter Allows you to sort what type of dimensions are shown in the equations dialog box.

Name The name of the dimension. You may change the name of the dimension by clicking once in this field and renaming the dimension.

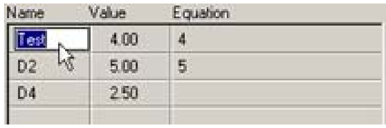
Value Shows the current value for each dimension. By pressing on the mouse while in the value field yo may high-

	light that value on your screen. This is helpful when you have numerous dimensions in your sketches.
Equation	By clicking in this field you may enter the mathematical expression that will define the dimension.

Changing the Dimension Name

Use the following steps to change the name of the Dimensions.

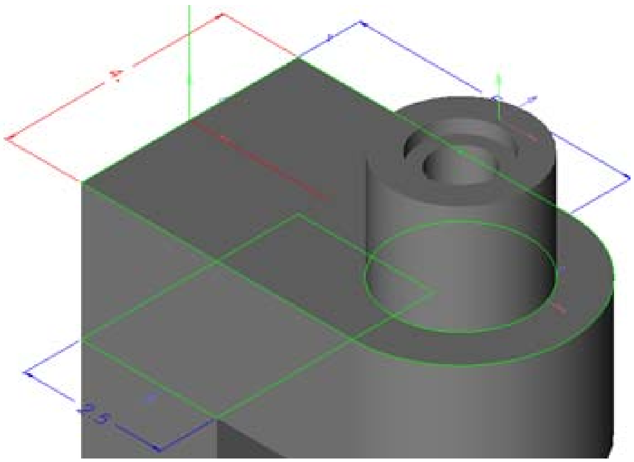
1. Move your cursor to the Dimension Name field you want to change.
2. Click once on the Dimension Name, the dimension highlights.
3. Change the name.



Highlighting Dimensions in the Sketch

Use the following steps to highlight the dimension on the screen.

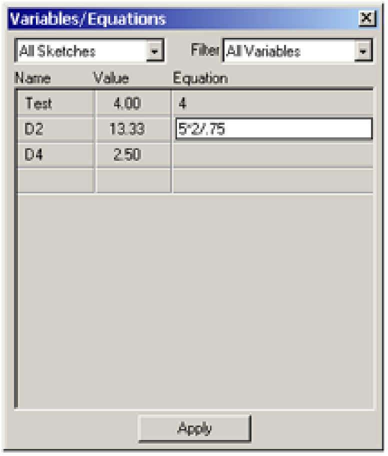
1. Move your cursor to the Dimension Value field you want to highlight
2. Click and hold down the mouse button
3. The dimension highlights.



Adding Equations

Using mathematical expression or operators you can change or resolve the size of your part.

1. Click in the equation field of the dimension you would like to change.
2. Add your mathematical expression, such as D1*2/3.
3. Click Apply for your changes to take place.



Note: You cannot use the undo (Ctrl or Cmd + Z) for expressions applied in the equations dialog box. Repeat the steps above to change the dimension.

Appendix A: Mathematical Operators

This Designer Elements program data fields accept the following mathematical operators.

Addition x+y

Subtraction x-y

Multiplication x*y

Division x/y

x%y - modulo division which returns the remainder of x/y. The resulting value will always be from 0 to y-1.

Angles this Designer Elements program accepts angle entries like the following: 45d30m30s, 45d30"30', 45d30', 30"30', 30m30s, 30"30s and 45°30"30

Absolute Value abs(x) - absolute value of x

Arc tangent atan (degrees)

Arc sine asin(degrees)

Arc cosine acos(degrees)

Cosine cos(degrees)

Sine	sin(degrees)
Tangent	tan(degrees)
Trigonometry	trig functions
Smallest Larger Integer	ceiling(x), or ceil(x) - returns a value representing the smallest integer that is greater than or equal to x.
Largest Smaller Integer	floor(x) - returns a value representing the largest integer that is less than or equal to x.
Degrees to radians	dtor(degrees)
Radians to degrees	rtod(radians)
Factorials	factorial(x), or fact(x) - factorial of x. (e.g. fact(4) = 4*3*2*1)
Logorithms	log(x)
	ln(x) - base 10 log of x, natural ln of x
Exponentials	exp(x) - e to the power of x where ln(exp(x)) = x.
	x^y - exponentiation (x to the power of y)
Negatives	neg(x) - negative of x
π	pi
Round	round(x) - rounds to the nearest whole value. For example, round(1.49) = 1.0, and round(1.51) = 2.0.
Random numbers	rnd - random value between 0.0 and 1.0
Square root	sqrt(x) - square root of x
Remove Fractional Part	truncate(x)
	trunc(x) - truncates to the whole value. For example, trunc(1.01)= 1.0, and trunc(1.99) = 1.0.

Appendix B: Special Characters

You can use special characters and accents which are available with your computer. Usually these characters are described in an appendix of the user manual. Many symbols and characters are from the keyboard.

Windows:

Unlock the keypad with the NUM LOCK key and then hold down the ALT key and enter the numeric code for the character you want.

The character appears in the current font (including the DE Plotter font). The following list includes common symbols which are available in all fonts:

Accent	Key Combination
°	*ALT 0176*
ø	*ALT 0216*
±	*ALT 0177*

Tech Note:

Numeric values must be entered with an enhanced 101 keyboard using the separate number pad.

Macintosh:

To see the characters associated with a particular font, choose *Key Caps* from the Apple menu and select the font from the Font menu. When you press the z (Command), OPTION, SHIFT, or CONTROL key or a combination of these keys, the keyboard graphic changes to show which characters are available.

In addition to those characters you see in *Key Caps*, you can use the OPTION key to generate international accents in the current font, including the DE Plotter font.

Accent	**Key Combination**
é	*OPTION+e* then letter to be accented
è	*OPTION+`* then letter to be accented
ñ	*OPTION+n* then letter to be accented
ü	*OPTION+u* then letter to be accented

The following list includes common symbols which are available in all fonts.

Accent	**Key Combination**
°	*OPTION+ SHIFT 8*
ø	*OPTION+ O* (letter or number)
±	*OPTION+SHIFT =*

Appendix C: DXF/DWG Translator

This Designer Elements program supports the AutoCAD DXF (drawing exchange file) and DWG format. DXF files are standard ASCII text files. DXF files can easily be translated to other CAD systems which likewise support the DXF format. DWG is the native AutoCAD file format. This Designer Elements program supports the following DXF/DWG entities:

Read DXF/DWG	Description
3DFACE	Four points defining the corners of a face.
POLYLINE	A string of vertices defining a line.
LINE	Two points defining a line.
CIRCLE	Center point and circle radius.
ARC	Center, radius, start angle and end angle.
POINT	One point location in space.
TEXT	Collection of characters defining text string.
BLOCK	Symbol.
ACIS DATA	Curves, surfaces, solids generated by ACIS.
ELLIPSE	Valid for R13+.
LIGHTWEIGHT POLYLINE	Valid for R14+.
MULTI-LINE	Valid for R13+.

SOLID	A filled region defined by 4 points.
SPLINE	Smooth spline curve valid for R13+.
LAYERS	Layer names, colors and locking.
LINE TYPES	Line font definition.

AutoCAD is a registered trademark of Autodesk Inc.

Appendix D: IGES Translator

This Designer Elements program supports importing and exporting IGES files. The list of supported entities appears below.

Importing

This list provides the IGES Entity number, Form and IGES Entity name.

IGES	*Form*	*IGES Entity Name*
#100	0	Circular Arc
#102	0	Composite Curve
#104	0	Conic Arc General
#104	1	Conic Arc Ellipse
#104	2	Conic Arc: Hyperbola
#104	3	Conic Arc: Parabola
#106	11	Copious Data 2D Path
#106	12	Copious Data 3D Path
#106	63	Copious Data: Closed 2D Curve
#108	1	Plane Entity Bounded Face
#110	0	Line Straight
#112	0	Parametric Spline Curve

#114	0	Parametric Spline Surface
#116	0	Point
#118	1	Ruled Surface
#120	0	Surface of Revolution
#122	0	Tabulated Cylinder Surface
#123	0	Direction Vector
#124	0	Transformation
#126	0	Rational B-Spline Curve
#128	0	Rational B-Spline Surface

IGES	*Form*	*IGES Entity Name*
#130	0	Offset Curve
#140	0	Offset Surface
#141	0	Boundary Entity
#142	0	Curve on Parametric Surface
#143	0	Bounded Surface
#144	0	Trimmed Surface
#186	0	MSBOBody
#190	0	Plane Surface
#112	0	Rt. Circular Cone
#194	0	Rt. Circular Conical Surf
#196	0	Spherical Surface
#198	0	Toroidal Surface
#502	1	Vertex List
#504	1	Edge List
#508	1	Loop
#510	1	Face
#514	11	Shell

Exporting

The list provides the number, this Designer Elements program/ACIS name, IGES Entity number, Form and IGES Entity name for exporting or writing out an IGES file.

VS/ACIS	*IGES*	*FORM*	*IGES ENTITY NAME*
Straight	#110	0	Line
Ellipse	#100	0	Circular Arc
	#104	1	Conic Arc-Ellipse
	#126	0	Rational B-spline Curve
	#128	0	Rational B-spline Surface
	#120	0	Surface of Revolution
Plane	#190	0	Plane Surface
	#122	0	Tabulated Cylinder
	#109	0	Plane - Unbounded
Cone	#192	0	Cylindrical Surface
	#194	0	Conical Surface
	#120	0	Surface of Revolution
Sphere	#196	0	Spherical Surface

VS/ACIS	*IGES*	*FORM*	*IGES ENTITY NAME*
	#120	0	Surface of Revolution
Straight	#110	0	Line
Torus	#198	0	Toroidal Surface
	#120	0	Surface of Revolution
Point	#116	0	Point
Vector	#123	0	Direction
Transf	#124	0	Transformation
Vertex	#502	1	Vertex
Edge	#504	1	Edge

Loop	#508	1	Loop
	#102	0	Composite Curve
	#106		Copious Data
		11	2D Path
		12	3D Path
		63	Closed Planar Curve
	#142	0	Curve on Parametric Surface
Face	#510	1	Face
	#144	0	Trimmed Surface
Shell	#514	1	Shell
Lump/Body	#186	0	MSBO

Appendix E: Short Cuts

These are the short cut keys currently programmed in this Designer Elements program. There are many short cuts actions in this Designer Elements program which you can program with specific keys. Choose ***File>Short Cut***s to display the Short Cut Manager. See Chapter 6, "Preference Settings," for information on how to program the keys.

Short Cut - Key	Action
Layout:Dec Display...{	Makes the previous layer visible and turns off all other layers except the work layer.
Layout:Inc Display...}	Makes the next layer visible and turns off all other layers except the work layer.
Layout:Isolate Layer...i	Brings up the Isolate Layer dialog box for setting the work layer.
Line:HorzConst...H	Creates a moveable horizontal construction line at your pointer tip. Move the pointer to the desired location and click to place the construction line.
Line:VertConst...V	Creates a moveable vertical construction line at your pointer tip. Move the pointer to the desired location and click to place the construction line.
Snap Alignments...A	Toggles the Drafting Assistant alignment snaps, like align x, y and z, between on and off.

Snap Centers...C	Snaps to the center of an object.
Snap:Edges...E	Snaps to the edges of an object.
Snap:Faces...F	Snaps to the faces of a solid.
Snap:Intersections...I	Snaps to the intersections of objects.
SnapOnOff...Q	Toggles all Drafting Assistant snaps between on and off.
View:Front View...s	Changes the view to Front.
View:ISO View...f	Changes the view to Isometric.
View:Redraw Screen...r	Redraws the screen.
View:Side View...a	Changes the view to Side.
View:Top View...d	Changes the view to Top.
View:TRI View...g	Changes the view to Trimetric.
View:Zoom All...e	Activates the *Zoom All* command.
View:Zoom In...]	Activates the *Zoom In* command.
View:Zoom Out...[	Activates the *Zoom Out* command.
View:Zoom Window...w	Activates the Zoom Window command allowing you to drag a selection fence to specify the zoom area.

Appendix F: Spline Text Files

You can create text files for importing Spline data into this Designer Elements program.

Creating a Text File for Importing a Spline

1. Use a text editor, a word processor or a spreadsheet to create a text file.
2. Input X, Y and Z values for your spline coordinates.

   The text file should be tab or space separated. Each line ends with a return. Line feeds after each return should have no effect.

   The text file should conform to the following columnar format:

1	1	0
2	2	0

   You are allowed to specify decimal coordinates as well:

1.33	1.1	0
2.4	2.5	3.5678

   Be sure that you press ENTER (Windows) or RETURN (Macintosh) after the last coordinate. If not, this Designer Elements program will not import the coordinates specified in the last line.
3. Save the file as Text only and import into this Designer Elements program using the Spline import format to create the spline.

Appendix G: Shader Attribute Definitions

This appendix defines shader attributes for all shader types. Although these terms may be used elsewhere in this Designer Elements program, these definitions only apply to the shader attributes.

Some shader attributes share a common base word like base color and decal color. In these instances, only the base word is listed and defined here. In this example mentioned, color is defined.

Information for these definitions was taken from LightWorks 5.0 Online Reference.

Color Class Attributes

amplitude	Sets the magnitude of an attribute relative to another.
axis	Sets the location of the axis for the attribute.
axis direction	Sets the direction for the axis when applying the simple wood shader.
bands	Sets the total number of bands around the evaluation cylinder when analyzing a surface.
brick height	Sets the brick height.
center	Sets the center of an attribute.
color	Sets the color of the attribute.

color array Sets the colors used for curvature divisions.

coverage Sets the ratio of the area covered by the bands to the area not covered for a surface evaluation.

curvature Sets the degree of curvature to be mapped.

curvature division Sets the number of color divisions used for a curvature evaluation.

curvature type Sets the type used in a geometric curvature evaluation. You have three types: gaussian, mean and absolute.

decal texture space Sets the texture space for the attributes. Checking the Edit button displays a copy of the Render Material Settings dialog box from which you can set the space.

decal transparency Sets the transparency from clear to opaque.

detail Sets the complexity of the texture where a value of 1.0 results in a simple pattern and higher values result in a finer pattern.

draft angle Sets the draft angle required to pull the object out of a mold.

file name Sets the file name containing the image used for the shader.

fuzz Sets the band sharpness.

gnarl Sets the random roughness of the regular rings inside the trunk.

grain Sets the intensity of the random grain effect where 0 (zero) equals no grain.

max cut off Set the maximum curvature value for evaluating an object.

min angle Sets the angle of the normal along the cylinder axis that defines the cylinder length and thus the reflection on the surface you are evaluating. Smaller values create longer cylinders.

min cut off	Sets the minimum curvature value for evaluating an object.
mix	Sets the mix ratio of attributes.
mortar size	Sets the mortar size.
noise	Sets the visibility of the tree rings for the simple wood shader.
offset	Sets relative displacement of odd and even rows of the wood pattern.
plank length	Sets the plank length of the wood shader.
plank variation	Sets the brightness variation between wood planks.
plank width	Sets the plank width for a wood shader.
point on axis	Sets the point on the axis of the tree from which the wood is taken.
pull direction	Sets the direction the object will be pulled from the mold.
radius	Sets the radius of the attribute.
replication type	Sets the pattern of the wrapped image.
ring fuzz grain	Sets the intensity of the high frequency random roughness for the ring edges.
ring fuzz in	Sets the sharpness of the inner ring edges near the trunk center.
ring fuzz out	Sets the sharpness of the outer ring edge.
scale	Sets the scale of the attribute.
separation	Sets the distance between centers of adjacent spheres for the solid polka shader.
size	Sets the attribute size.
softness	Sets the softness of the feature used to define the shading.
strips	Sets the count of the rectangle wood planks.

tolerance angle	Sets the degree tolerance added to the draft angle that allows the object to be pulled from a mold but with difficulty.
trunk direction	Determines the direction of the trunk axis.
trunk center	Specifies the center of the trunk.
vector	Sets the direction of the attribute calculated in determining the appearance of a particular shader.
vein contrast	Sets the color contrast of the marble veins where larger values produce a greater contrast.
width	Sets the attribute width.

Displacement Class Attributes

amplitude	Sets the magnitude of one attribute relative to another.
blend	Sets the size of the blend between the sphere and the surface for the shader.
center depth	Sets the depth of the spheres used for the dimple shader.
dented threshold	Sets the relative contributions made by the displacements and indentations for the casting shader.
detail	Sets the complexity of the texture where a value of 1 results in a simple pattern and higher values result in a finer pattern.
file name	Sets the file name containing the image used for the shader.
frequency	Sets the wavy or curving quality of edges.
irregularity	Sets the pattern shape from a square to an irregular convex shape.
radius	Sets the radius of the attribute.
scale	Sets the scale of the attribute.
separation	Sets the distance between the centers of adjacent spheres for the wrapped dimple shader.

sharpness	Sets the sharpness of surface irregularities.
smooth max	Sets the maximum smoothness of the edges when using the leather shader.
smooth min	Sets the minimum smoothness of the edges when using the leather shader.
softness	Sets the softness of the feature used to define the shading.

Reflectance Class Attributes

absorption	Sets the amount of light absorbed.
ambient factor	Sets the amount of ambient light reflected.
amplitude	Sets the magnitude of one attribute relative to another.
bias	Sets the contribution of the two thread directions for the wrapped woven anisotropic shader. A bias of 0.0 causes all reflectance to be provided by threads along one axis. A bias of 1.0 causes all reflectance to be provided by threads along the other axis.
chrome factor	Sets the amount of chrome light reflected.
color	Sets the color of the attribute.
cylinder distance	Sets the distance between cylinders for the wrapped anisotropic shader. The distance determines the degree of anisotropy of the surface. A distance of 0.0 results in an isotropic (normal) reflection. A distance of 2.0 results in the maximum anisotropy.
decal texture space	Sets the texture space for the attributes. Clicking the Edit button displays a copy of the Render Material Settings dialog box from which you can set the space.
diffuse factor	Sets the amount of diffuse light reflected.

exponent	Sets the sharpness of the specular reflection highlights.
file name	Sets the file name containing the image used for the shader.
floor height	Sets the height of the floor across the cylinders used for the wrapped anisotropic (grooved) shader. A value of 0.0 gives equals no floor. A value of 1.0 creates a flat, isotropic surface.
height	Sets the height of the attribute.
metallic layer factor	Sets the contribution of the metallic layer to the reflectance of the multilayer paint shader.
metallic flakes	Sets the metal for the metal flakes in the paint shader.
mirror factor	Sets the contribution made by light reflected in the mirror direction.
reflectance	Sets the reflectance of the shader.
refraction	Sets the amount of light refracted.
roughness	Sets the sharpness of the reflectance. Smaller values, such as 0.1, produce a sharper reflection.
scale	Sets the scale of the attribute.
selector	Sets the shader used to calculate the decal reflectance with respect to the base object. Clicking the Edit button displays a copy of the Render Material Settings dialog box from which you can set the shader.
shader	Sets the shader.
sharpness	Sets the sharpness of the surface.
softness	Sets the softness of the feature used to define the shading.
specular factor	Sets the amount of specular light reflected.
translucency factor	Sets the degree of translucency.

transmission factor Sets the amount of light that passes though the shader.

transparency Sets the transparency of the shader.

width Sets the width of the attribute.

Transparency Class Attributes

coverage Sets the degree that an attribute covers the object with the associated shader.

color Sets the color of the attribute.

detail Sets the complexity of the attribute.

edge falloff Sets the transparency edge falloff rate.

file name Sets the file name containing the image used for the shader.

fuzz Sets the degree of fuzziness for the edges.

grid size Sets the size of the grid for the wrapped grid shader.

height Sets the height of the attribute.

noise density Sets the density of the roughness or irregularities.

scale Sets the scale of the attribute.

s fuzz Sets the softness of the s edge of the square for the wrapped square shader. The letter "s" is an identifier used to refer to one side of the square.

size Sets the size of the attribute.

s max Sets the maximum s dimension of the square for the wrapped square shader. The letter "s" is an identifier used to refer to one side of the square.

s min Sets the minimum s dimension of the square for the wrapped square shader. The letter "s" is an identifier used to refer to one side of the square.

softness Sets the softness of the feature used to define the shading.

t fuzz Sets the softness of the t edge of the square for the wrapped square shader. The letter “t” is an identifier used to refer to one side of the square.

t max Sets the maximum t dimension of the square for the wrapped square shader. The letter “t” is an identifier used to refer to one side of the square.

t min Sets the minimum t dimension of the square for the wrapped square shader. The letter “t” is an identifier used to refer to one side of the square.

transparency Sets the transparency of the shader.

width Sets the width of the attribute.

zero angle Sets the angle between surface normal and view direction.

Texture Space Class Attributes

aspect ratio Sets the ratio of the texture space which is defined as one unit of its height divided by one unit of its width.

axis direction Sets the direction of the axis.

center point Sets the center point of the cylinder used for mapping a texture space.

origin Sets the origin point of the texture.

scale Sets the scale of the attribute.

scale along axis Sets the factor that an image is scaled along the axis.

scale around axis Sets the factor that an image is scaled around the axis.

vector Sets the direction of the attribute.

Background Class Attributes

angle Sets the angle (radians) over which the environment map is sampled for each background pixel allowing blurring. A 0 (zero) angle (default) means that the pixel's center point determines the color.

color Sets the color of the attribute.

detail Sets the complexity of the attribute.

distance Sets the distance that the infinite planes are in front and back of the eye point. The two background images are placed on these infinite planes.

extrapolation Sets how the background will cover the background area. You have three options: none, smear and tile.

file name Sets the file name containing the image used for the shader.

intensity Sets the brilliance of the reflection to be altered. The color is calculated for each background pixel by multiplying it with the intensity.

keep aspect Sets the use of the pixel aspect ratio for the image. Images are automatically scaled to fit the viewport. Selecting True preserves the pixel aspect ratio.

keep texture Sets the use of the texture for the image. Selecting True results in the image and file name attributes being referenced when no texture has been created yet.

missing ratio Sets the ratio for mixing two shaders.

rotation Sets the angle the image is rotated. The value must be either +/-90°, +/-180°, or +/-270°. Positive angles rotate the image clockwise. Negative angles rotate the image counter-clockwise.

scale Sets the scale of attribute.

shader Sets the shader.

softness Sets the softness used to define the shading.

Foreground Class Attributes

AA level — Sets the depth used for determining the number of samples. A value of zero results in no oversampling. A value of one results in two times the number of samples as the maximum.

AA threshold — Sets the limit value used to determine whether additional sampling calculations are performed to determine the light effect. If the scattered light at two sample points differ more than the threshold value, additional sampling occurs.

amplitude — Sets the magnitude of one attribute relative to another.

bounds — Sets the use of light scattering boundaries for calculating the effect when unable to determine where the light source contribution is negligible. Selecting False results in no bounds.

bounds on — Sets the use of light scattering boundaries for calculating the effect when unable to determine where the light source contribution is negligible. Selecting False results in no bounds.

bounds volume — Sets the volume of the boundary sphere used to calculate the scattering effect when unable to determine where the light source contribution is negligible.

color — Sets the color of the attribute.

density — Sets the density of the attribute.

distance — Sets the distance used in calculating the closeness of the fog to the viewer.

eccentricity — Sets the ellipse eccentricity used when calculating the light scattering effect for the Henyey-Greenstein scattering model. It has no effect on any other light scattering model. A zero eccentricity results in an isotropic scattering. A positive eccentricity results in a forward scattering. A negative eccentricity results in a backward scattering.

error bound Sets the limit for using the max lod attribute when calculating the detail effect using the scattering medium shader. The calculation time can become excessive depending on the detail level. This value determines the trade off point between calculation time and accuracy. The suggested range is between 0.0 and 1.0.

falloff threshold Sets the spherical area of influence of the light sources. Beyond a certain area the light contribution would be negligible, making a large number of samples unnecessary. The default threshold is 0.001. The value's effect depends on the size of the scene and light source intensity. Thresholds that are too high result in spotlight clipping.

far Sets the maximum distance for the foreground shader. Distances greater than the far value display the full background color.

flake size Sets the size of the flake for the snow shader.

fog height Sets the sets fog decrease rate.

ground normal Sets the normal for the ground fog shader.

ground point Sets the ground point for the ground fog shader.

ignore background Sets whether the fog effect is applied to the background. Selecting True results in no background fog effect.

max depth Sets the maximum distance used in calculating the light scattering effect. A smaller depth, near 0.0, results in an image lacking any volumetric effects. The default depth is 1000. The value's effect depends on the size of the scene.

max lod Sets the maximum detail level for calculating scattered light using the scattering medium shader.

medium ambient Sets a uniform light scattering through the medium.

medium attenuation	Sets the attenuation within the medium or how it absorbs light along the way.
medium density	Sets the density of the medium through which scattering occurs.
medium shadows	Sets whether the medium received shadows.
min lod	Sets the minimal detail level for calculating scattered light using the scattering medium shader.
near	Sets the minimum distance that a background color appears. Distances less than the near value will not display the color.
noise gain	Sets the contrast in the noise. High values result in sharp transitions while low values result in smooth transitions.
noise octaves	Sets the number of octaves (frequencies/scales) used to determine the detail of the noise in the light scattering.
samples	Sets the number of samples taken to calculate the atmospheric scattering of light. A higher number results in greater accuracy but requires more calculation time.
scale	Sets the scale of attribute.
scattering model	Sets the model used for scattering the light. You have five options: isotropic, Rayleigh, Mie hazy, Mie murky and Henyey-Greenstein. Choosing Mie murky results in strong anisotropic forward scattering as would appear when looking directly at light sources.
source attenuation	Sets the falloff value for the attribute. Small changes in the attenuation value greatly affect the light scattering effect. Values are typically between 0.1 and 0.5.

Glossary

Accelerators Keyboard Equivalents that invoke commands rather than using the mouse to choose from menus.

Align These commands let you align objects, including: text along the left sides, right sides, tops, bottoms, centers horizontal, centers vertical, to grid and equally spaced vertically.

ACIS This Designer Elements program is based on this kernel, developed by Spatial Technologies.

Ambient Light This light source provides equal illumination on all sides independent of the light source normal.

Annotation Text on drawings, including notes, crosshatching and dimensions.

Alignment Angle The angle of the Drafting Assistant's automatic construction lines. The specification is set in the ***Window>Snaps*** submenu.

Ambiguity Popup This popup menu appears when you attempt to select one object among objects so you can choose the desired object.

Anchor This point defines the direction when placing a distant light source in your drawing.

Anti-Alias Feature Following	This feature performs a ray trace operation to bring out small geometric details and produce smoother images.
Arrange	This command in the Layout menu allows you to change the display of overlapping objects in your drawing.
Arrow Tool	Used for selecting objects to be operated on with subsequent commands. Also used to move selected geometry.
ASCII	An acronym for American Standard Code for Information Interchange.
Aspect Ratio	Refers to a mesh surface and specifies the maximum ratio between triangle edges.
Associativity	A link between an object and its dimensions or parent/child objects. In the case of dimensions, if the object is changed, the dimensions automatically change to match. In the case of parent/child objects, if the parent is changed the child also changes.
Attenuation	The reduction of light intensity with the distance from the source.
Attributes	The data fields associated with a particular object that define that object. This includes such as Layer Name, Pen Style, Color, X-Y-Z Coordinates. This is also a page in the Edit Objects dialog box.
Auto Heal Bodies	This function finds collections of surfaces that define closed volumes and convert them into solids. This occurs when importing Vellum 3D and IGES files into this Designer Elements program.
Auxiliary View	A view created from its parent view at the geometry location that the user specifies.
Axis	Displays the current view orientation of the X, Y and Z axis in the center of your screen.

Bezier Curve	A free form curve. NURB splines are a superset of Bezier curves.
Blend	This is the filleting and rounding of solid edges.
Boolean Tools	These tools allow you to add, subtract and intersect solids.
Border	A frame showing the boundary of a view.
Boss	This is a cylinder extending from a solid and filleted at the intersection of the two.
Boundary	The geometry that defines the limits for operations such as trimming and relimiting.
CAD	An acronym for Computer-Aided Design.
CADD	An acronym for Computer-Aided Design and Drafting.
CADD.LIN	All line patterns are stored in this file in the Environ folder.
CAE	An acronym for Computer-Aided Engineering.
CAM	An acronym for Computer-Aided Manufacturing.
Case	This refers to the text case options in this Designer Elements program and include lower case, UPPER CASE and Title Caps.
Center Mark	A center-line dimension for circles and arcs.
Chamfer	A beveled or sloping edges between two objects.
Characteristics	See Attributes.
Child	An object created from another object or an operation performed on an object.
Circumference	The distance around a circle along its edge: =2 Pi r
Circumscribed	Enclosing a circle. In circumscribed polygons, the *midpoint* of each side of the polygon touches an imaginary circle (i.e. the polygon exactly surrounds the circle).

Clamped Linear
Light intensity that diminishes according to the following formula, *Intensity/(distance+1)*.

Clamped Quadratic
Light intensity that diminishes according to the following formula, *Intensity/(distance^2+1)*.

Click
To press and release the mouse button. When you are told to click an object, move the pointer to the object and press and release the button.

Clipboard
The memory buffer where selections are stored when the *Cut* or *Copy* command is used.

Conic
These objects are used in aerospace design field and create curves defined by start point, end point, shoulder and slope control points.

Construction Lines
Lines, displayed as dotted or gray lines, that you use for exact alignment. The Drafting Assistant creates dynamic, temporary construction lines. You can also create permanent construction lines, which can be used in the geometry or used for alignment and then deleted.

Control Point
The *endpoint* or *midpoint* of an object or "knot" point defining a spline. The Drafting Assistant indicates these positions when the pointer is moved near them.

Coon Patch
A nurb surface with three or four sides.

Coordinates
Positions on axes that specify the point locations. Two-dimensional objects have x,y coordinates; three-dimensional objects have x,y,z coordinates.

Coplanar
This refers to objects that lie in the same two dimensional plane.

Copy
The command that places a duplicate of the selected geometry on the Clipboard. See the descriptions for the Polar Duplicate and Linear Duplicate tools and Transformation tools for additional copying methods.

Cover Surface A surface created from a profile.

Counter Bore A hole created from a hole and a bore where the bore is a straight sided cylinder with a diameter larger than the hole. The hole extends from the end of the bore into the solid to complete the counter bore.

Counter Sink A hole created from a hole and a sink. The sink is an angled hole with a diameter larger than the straight sided hole. The hole extends from the end of the sink into the solid to complete the counter sink.

Crayon Picker This is a Macintosh color display allows you to chose a color from the crayon box.

Custom Colors (Windows only) This button in the color display allows you to define 16 additional colors to show in the partial color display.

Cursor The I-beam position indicator in the text tool and boxes which use text. Elsewhere, the position indicator is called a pointer.

Curvature This command in the Verify menu displays a porcupine plot of selected curves or surfaces representing the direction and order of magnitude of the curvature.

Curvature Plot This analysis displays geometry in a Gaussian Curvature plot which is the product (K_1 - K_2) of the principle curvatures at a point on a surface.

Curve A line, circle, arc, ellipse, or spline.

Cut The command to delete selected entities. The selection is placed on the Clipboard and can be pasted into the same or different documents or into documents created by other applications.

Cutout A profile that has been extruded through a solid and removes all intersecting material.

CYMK Picker	This is a standard color wheel for the Macintosh with the option to specify CYMK values.
Dashed	A line pattern made up of dashes.
Default	Built-in settings that are used by the system if you do not specify your own value or choice.
Defining Points	The X, Y and Z coordinates for the specified points of splines, mesh and slab primitives.
Deformable Faces	These are solid faces whose shapes can be modified by applying a gain pressure.
Degenerative	An object such as a surface or chamfer where the its length at the beginning and/or ending is equal to zero.
Delete	The command to erase selected geometry. The selection is not placed on the Clipboard; however, it can be retrieved within the limits of the *Undo* command.
Delta	A change, usually in position.
Design Explorer	This command displays a dialog box with the history tree of the selected object. It includes its own submenu of editing commands.
Detail View	An enlarged view of a specific area of the geometry displayed in a drawing view.
Dialog Box	A specification box that appears in response to certain commands. A dialog box allows you to provide information that qualifies the execution of those commands.
Diameter	The distance across an arc or circle, passing through the center.
Differential Scaling	Refers to the ability to use different scaling values for along different axes of a curve.
Dimension	A graphic object that displays the distance between two points. A measurement of an object.

Direction This command in the Verify menu displays the direction of the normals of an object.

Distant Light This light source emits parallel rays of light, as if from a very distance source.

Divide To segment a line or curve into equal parts. This is also one of the Trim tools.

Double-click To press and release the mouse button twice, quickly.

Drafting Assistant A unique feature which displays feedback notations and construction lines to aid with snap, alignment and constraint operations. The Drafting Assistant facilitates exact construction without requiring you to be exact. When the pointer is close enough to display feedback, the Drafting Assistant locks onto the exact location for you.

Drag To press and hold the mouse button, move the pointer to a new location, and release the button.

Draft A taper or a specific angle applied to a solid.

Draft Angle The subtraction or addition of material at a specific angle along a solid. Draft angles are used to facilitate the placement or removal of a part from a mold. Positive draft angles add material. Negative draft angles remove material.

DXF An acronym for Data Exchange Format-a format of AutoCAD files.

Drawing A drafted, 2-dimensional drawing of a model. Usually a drawing including an orthogonal view.

Drawing View A view containing 2D geometry created from a 3D model after choosing the *Model to Sheet* command.

Dynamic Render The feature that displays your geometry in the selected render mode as you rotate the geometry.

DWG AutoCAD's native file format.

Endpoint The first and last point of a line or curve. The Drafting Assistant indicates these positions when the pointer is moved near the *endpoint* of an object.

EPS Encapsulated PostScript format for printing to a PostScript printer and for importing into compatible applications.

Estimate Count Refers to a button in the Mesh Parameter dialog box which calculates the approximate number of facets based on specified parameters. The dialog box appears when you using the *Change Object Type* command and convert a surface or solid to mesh.

Export To save a document in a file format that can be used by a different application program.

Extrude Creates a 3D object out of a 2D profile.

Expression Parsing Mathematical, trigonometric and exponential operators that can be used in the Status Line.

Face A surface of a solid.

Facet A way of representing surfaces in DXF and DWG files prior to release 13.

Falloff Angle The angle that controls the sharpness of a spot light's edge.

Falloff Rate This light distribution for a spot light from the center of the spot light cone to the outer edge.

Feature A set of operations that may add material to or subtract material from your solid including blending, chamfering, creating holes, bosses, cutout and protrusions.

Field of View The view angle for a perspective.

File An individual document.

Fill Color The color applied to a selected pattern for a smart polygon.

Fill Pattern The pattern applied to a smart polygon.

Fillet An arc of a specified radius tangent to entities.

First Blind An option for the hole tools which will extend a hole to the first open face.

Flatten View This command in the Drawing View menu flattens 2D geometry within a view onto the Sheet View layer. This operation breaks the 2D geometry's associativity to the 3D model.

Flat Shading Renders your object with a painter's algorithm using constant shading techniques.

Flavor The types of IGES files this Designer Elements program can import and export.

Flip Normal When this option is checked in the Render Options dialog box, the normals of an object are flipped. If light normals are pointed away from the view when rendered, the object will appear dark.

Font The assortment of type used in text.

Gain Pressure applied to a solid face to deform it.

Geometry The objects used to construct parts.

Geometric Characteristics Characteristics that make up the geometry of an object like, length, radius, Defining Points and Rho.

Gouraud Shading Renders your geometry based on calculated light intensities at each vertex. It shades more quickly but with a lower quality than Phong shading. This rendering method uses Open GL (Windows) or QuickDraw 3D (Macintosh).

Gouraud w/Edges Shading Renders your geometry based on calculated light intensities at each vertex and displays the face edge boundaries in a specific color.

Gregory Surface A Nurb surface with more than four sides.

Grid The rectangular array of lines that facilitates measurement and alignment. The grid display can be turned on or off, and the spacing can be specified through the Layout menu.

Group To specify several entities as one unit that will be treated as a single object.

Helix A spiral curve.

Hidden A line pattern used to draw lines that would not be visible in a solid part. Drafters traditionally use hidden lines for geometry that is behind other geometry.

Hidden Shading Renders your geometry such that only visible edges are displayed.

Hidden w/Dimmed Renders your geometry such that the visible edges are displayed and the hidden edges are dimmed.

Hot Spot The point on the wireframe and dimension tool cursors that indicates the next point to click. The hot spot is represented by a dot with crosshairs on the cursor.

Histogram A bar graph representing the frequency of a curvature smoothness (change in a curve over the change in curvature) using the color spectrum. The length of the bar represents the frequency.

Hit Radius The distance, in pixels, detectable by the Drafting Assistant between the object and the pointer.

HSL Picker This is a standard color wheel for the Macintosh with the option to specify Hue, Saturation and Lightness.

HSV Picker This is a standard color display for the Macintosh with the option to specify Hue, Saturation and Value.

IGES An acronym for Initial Graphics Exchange Specification. The U.S. Department of Commerce, National Bureau of Standards issues IGES as the industry standard among CAD/CAM systems for data exchange in a neutral file format.

Import — To load or read in a non-Designer Elements program file.

Infinite Plane — A two dimensional surface with no defining boundaries.

Inflection — A change in slope along a spline.

Influence — The degree of impact a control point has on a spline.

Inscribed — Within a circle. For polygons, all vertices touch the (imaginary) circle.

Instance — Refers to an object that's moved to or placed in a different location after performing an operation on it. If you create a solid, add a blend and move it, the solid is now an instance. The original is still located in the previous position although it is not displayed. If you create a master symbol, an instance occurs when you place the symbol in your drawing. An instance is associative to the original geometry. Any change made to the original is reflected in the instance. If you copy and pasted the object, the associativity is broken.

Intensity — The lighting level for a light source set in the Edit Objects dialog box.

Interference — The shared volume created by two or more intersecting objects.

Intersection — The position where two lines or curves meet. The curves may actually touch or only intersect when they are extended. The Drafting Assistant indicates only actual intersections.

ISO Lines — Control the isopram lines drawn for a surface. These Iso (isopram) lines are constant parameter curves that lie on a surface, typically defined in parameter space. The parameter space coordinate system uses *U* and *V* coordinates. A 0 (zero) in both fields turns off Iso lines. The appropriate *U/V* values may enhance the visual appearance of the surface at the expense of drawing speed. The letters, *U* and *V* are industry standard space coordinate identifiers (U = horizontal, V = vertical).

Isopram The full name for ISO Lines.

Knot Points The points defining a spline, indicated as *vertex* points by the Drafting Assistant.

Lathe Revolve an object a certain angle.

Layer Analogous to transparent media used in conventional manual drafting. Parts can be constructed on several layers which you can make visible or invisible.

Line Pattern Manager Through this dialog box you can modify the scale of all available line patterns.

Linear Duplicate A feature that duplicates an object and places the copies in a line or in an array of multiple lines.

Links This is the associative relationship that exists between parent/child objects in which a modification to the parent also modifies the child.

Local Face Modeling The ability to perform various operations on a specific face of a model including: drafts, move, offset, remove replace and match.

Lock The command to render a selection unchangeable.

Loft A surface that references another surface making it tangent to the referenced surface. Any surface can be used to create the surface.

MxN Curves The number of curves that define a net surface in two perpendicular directions.

Mask To select entities as a group, masking out all others.

Masking — The method used for applying a decal to an object, either stencil or factor.

Mate — This operation matches the planar face of one object to the planar face of another object.

Math Operator — The mathematical, trigonometric, or exponential expression used to indicate values. All value entry boxes in this Designer Elements program accept the math operators listed in Appendix A.

Max Edge — Refers to a conversion of an object type to mesh. This sets the maximum acceptable length of facets.

Message Line — The top line of the drawing area. It names the current tool and provides instructions for using it.

Memory Indicator — The indicator at the bottom left corner of the Designer Elements program window that displays how much RAM is allocated to this Designer Elements program.

Mesh — Planar elements defined by nodes or 3D vertices that can be used to represent surfaces but are not surfaces themselves.

META — The file format used by the Windows Clipboard.

Mnemonics — The key sequence which invokes a command from a menu.

Model — The model is your geometry.

Model Point — The point you can specify in the View Rotation Options dialog box about which to rotate a view.

N# Sections — The number (n) of cross sections used to define a skin surface.

Net Surface — A surface defined by M (number) of rows and N (number) of columns.

Non-planar — Surfaces or points that do not lie in a two dimensional plane.

Normal
A perpendicular to a tangent of a curve, surface or solid face.

Normal Deviation
Refers to a conversion of an object type to mesh. This sets the maximum angular deviation between adjacent facets.

NURB
Non-Uniform Rational B-splines—the type of splines this Designer Elements program creates. NURB splines are a superset of Bezier curves. NURB splines provide designers with two interrelated functions.

First, curvature continuity remains intact even when the curve is changed, so kinks won't develop as the spline is altered. Second, localized control of a complex curve is provided.

Object
An individual piece of geometry, such as a line, arc, circle, surface or solid.

Object Extents
The area defined by an object.

Object Type
This refers to a specific kind of geometry and includes curves, surfaces and solids.

Offset
The distance a curve or surface is placed from the original location.

Origin
The 0,0,0 location on the drawing area. When a new document is opened, 0,0,0 is located in the middle of the screen. The coordinate symbol displays at the origin when the grid is turned on. You can change the origin at any time.

Pan
A horizontal camera movement used when creating movies.

Palette
A group of tools. The general tool palette is always displayed to the left of the drawing area.

Parent
An object from which other objects are created or operations are performed.

Param	The percentage point that a variable blend radius is placed along a solid edge.
Parent/Child	The relationship created between an original object and subsequent objects such that any modifications done on the parent affects the child.
Part	A collection of entities representing an object or structure.
Parting Line	A curve used with the Parting Line tool to divide a solid for applying a draft.
Paste	To place the contents of the Clipboard in the current document.
Perpendicular	At a 90° angle. The Drafting Assistant displays a notation when the current construction is at a 90° angle to an object.
Perspective	Viewing 3D geometry on a two dimensional surface as seen by normal binocular vision.
Phong Shading	Renders your geometry based on calculated light intensities at each pixel location.
Phong w/Edges Shading	Renders your geometry based on calculated light intensities at each pixel location and displays the face edge boundaries in a specific color.
Photorealistic	Images resembling photography in the rendering quality.
Pick Box	The area pixel area defined in Preferences that allows you to select an object.
Pick Objects	This is the dialog box with the Ambiguity Popup menu.
PICT	The Macintosh graphics file format which uses object-oriented bitmaps or resolution-independent graphics.
Pick	To select a location or object by clicking it.

Pipe Solid	A solid created by with an inside and outside diameter using the Pipe Solid tool.
Pitch	The distance between helix peaks.
Planar	Surfaces or points that lie in a two dimensional plane.
Plot	To draw a part on paper using a plotter.
Plotter	A computerized drawing device for hardcopy output.
Point	A location for constructing geometry.
Point Light	This light emits rays in all directions, like a candle or table lamp.
Pointer	The position locator similar to a cursor. When a tool is in effect, the pointer takes on a representative shape while in the drawing area.
Polar Duplicate	A feature that duplicates an object and rotates the copes around a reference point.
Press	To press and hold down the mouse button. This action is most commonly used to view the contents of a menu.
Primitive	A basic solid shape defined by linear or quadratic geometry like: block, cylinder, cone, etc.
Profile	A closed curve or collection of curves connected end to end.
Protruded Feature	Material projections from a solid.
Pull-direction	The direction a part ejects from a mold.
QuickTime Object Movie	A Quick Time movie that keeps the observation point fixed as the eye point is moved at a fixed distance about the observation point.
QuickTime VR Panoramic Movie	A QuickTime movie that keeps the eye point fixed as the observation point is rotated 360°.

Radius Half the distance across an arc, starting from the center.

Rail A curve used to define a sweep direction and distance for a surface.

RAW This translator creates a file containing triangular vertices that define the x, y and z locations of the 3D faces.

Raytrace The function that determines the visibility of objects in a scene by tracing rays from the eye to the objects. The raytrace function calculates the visibility by breaking the scene into smaller pixel areas, producing a photorealistic rendered images.

Redo The command that restores a previous operation. You can redo an infinite number of actions in a specific Designer Elements program session.

Redraw To refresh the screen, recreating all objects.

Reflectivity The ratio of the amount of light falling on a material to the light reflected off it.

Relative Position A location specified as a certain distance from another location. It is often called the delta position.

Relimit Lengthen or shorten lines to the specified object.

Render The command that shades your geometry in a specified mode.

Render Now The render option activated when you choose the *Render* command. You set the mode in the Render Options dialog box.

Render Options The dialog box that allows you to set the shading mode for your geometry.

Resolution The appearance of surfaces and solids when rendered. You have five options: Super Fine,Very Fine, Fine, Medium and Coarse.

Rho	Used in defining a conic object, it is the ratio of the center point - shoulder point distance and the center point - slope control point.
Right-hand rule	A memory aid for the relative directions of the positive axes. With your right palm upturned, the thumb (X) points right, the index finger (Y) points straight ahead and the middle finger (Z) points up. If you move your hand to indicate the X and Y axes, you can easily see the direction of the Z axis.
SAT	A file format for ACIS based programs.
Section View	A view created by making a section cut through your geometry displayed in a drawing view.
Selection Fence	A bounding box that temporarily appears as you drag the Selection tool cursor around an area in your drawing.
Setback	The distance that a blend extends from three or more intersecting edges.
Short Cuts	Key combinations for performing Designer Elements program operations, allowing you to customize your program.
Shelled Solid	A hollowed out solid object.
Shoulder control point	A point used in defining a conic.
Silhouette	A view of the object from the visible edges only. Silhouette edges are view dependent and can cause a significant reduction in drawing speed.
Simplify	This option available in the Stitched Solid tool asks Designer Elements program to determine whether the solid can be defined by analytic surfaces, for which ACIS is optimized, rather than nurb surfaces.
Skin Surface	Nurb patches fitted over a collection of curve cross-sections.
Slab	A solid primitive created from three or more points, a height and draft angle.

Slope	The change of x relative to y between two points on a line. In a spline, slope defines the vector of a line tangent to the spline at a particular knot point.
Slope control point	A point used in defining a conic.
Smart Polygon	A true polygon that is one object with length and width. This can be contrasted with a single line polygon composed of individual lines.
Smart Silhouette	A silhouette that displays only if it does not degrade the performance of this Designer Elements program.
Snap	The command allows you to set the specifications for the Drafting Assistant.
Sphere Trackball	The standard trackball that allows you to rotate the view by dragging the cursor on the sphere.
Spline	A smooth, free-form curve passing through specified points.
Spot Light	This light emits a cone of light from a local source.
Static Render	This render option defines how your geometry will display when the view is stationary.
Status Line	The line at the bottom of the drawing area in which you can enter the specifications for the geometry you are creating.
Stencil	An image that defines the regions through which artwork projects onto an object.
Step Trackball	The trackball that allows you to rotate the view by clicking on a directional arrow.
Stitched Solid	This is a solid created from joining surfaces together.
STL	The file format creates stereolithography files.
Subtract Solid	This solid is subtracted from another to create a final solid.

Sweep	Refers to the extrusion of a profile to a surface or solid.
Surface	Non Uniform Rational B-Splines created for specified boundaries.
Surface Deviation	Referring to the conversion of a surface or solid to a mesh, this sets the maximum acceptable distance between the facet and the surface represented by the mesh.
Symbol	Objects that are created in the Symbol Manager are Master Symbols and can be instanced into the drawing at multiple locations.
Tangent	The point where a line or curve touches a curve without intersecting it. The Drafting Assistant displays the *tangent* notation of a curve when the pointer nears it.
Taper Solid	This is a solid created with draft angles or tapers.
Termination Type	This defines the protrusion type used to create a feature on a solid.
3DMF	The file format of QuickDraw 3D.
Tiling	Breaks up a drawing into a multiple pages to print large drawings.
Tilt	A vertical camera movement used when creating movies.
Torus	A solid primitive generated by the revolution of a conic section, like a circle.
Transparency	The amount that light can pass through a material.
Triad	This illustrates the orientation of the x, y and z axis and the current work plane.
Toggle	To switch between two conditions, for example, *Hide Grid/Show Grid*.
Transformation	The tools to move, rotate, expand, shrink, or mirror an object or group of entities.

Trim	To shorten or remove a portion of a line.
Tube Surface	A surface created in the shape of a tube.
Twist Angle	The angle you want your solid to twist when sweeping a profile to a solid.
Unclamped Linear	Light intensity that diminishes according to the following formula, *Intensity/distance*.
Unclamped Quadratic	Light intensity that diminishes according to the following formula, *Intensity/distance*^{2}.
Undo	The command that reverses the last editing or creation action. You can undo an infinite number of actions in a specific Designer Elements program session.
Uniform Scaling	Scales an object equally in all directions.
Units	Measures used for construction (U.S. or metric).
Vector Splines	The slope, shape, control points and control point influence can be modified for splines of this type.
Verify	A menu listing commands for determining properties of selected objects.
Vertex	The point at which the sides of an angle intersect or a knot point of a spline.
View Rotation Options	This dialog box allows you to specify trackball rotation parameters.
VRML	A file format for exporting virtual reality modeling language files.
Wrap	The mode that artwork projects onto an object, planar, cylindrical or spherical when applying a decal.
Wireframe	A 3-dimensional representation showing boundary lines, edges and intersections, but not surfaces.
World plane	The work plane used at the beginning of the construction of a model. Also known as the world coordinate system.

Work plane The x,y plane used for 2D objects which has an origin of 0,0,0 for all data input. Sometimes referred to as the user or work coordinate system.

Zoom The tool or command that allows you to magnify or reduce an image.

Symbols

Numerics

A

B

C

D

E

F

G

H

I

M

N

O

P

S

T

U