ASHLAR·VELLUM™

Software that works the way you think.

Cobalt™ Xenon™ Argon™

Designer Elemets

3D Modeling v11

User Guide

ASHLAR·VELLUM™

Software that works the way you think.

Cobalt™ Xenon™ Argon™

Designer Elemets

3D Modeling v11

User Guide

Cobalt™, Argon™, Xenon™

March 2019

Vellum Investment Partners, LLC

1-800-877-2745

www.ashlar-vellum.com

CXAGET1903

Table of Contens

Table of Contens i-1
User Guide Documentation 1-1
Menus and Submenus 1-1
Choosing Commands 1-1
Margin Notes 1-1
Style Conventions 1-2
Terminology 1-2
Using the Mouse 1-3
Chapter Breakdown 1-3
Sections 1-4
Appendices 1-4
Other 1-5
Graphics 1-5
First Look 2-1
Parts of the Designer Elements Window 2-1
Tool Palette 2-3
Right Mouse Button 2-5
Selected Drawing View 2-11
Smart Pointer & Wireframe Tools 2-11
Location Indicator 2-12
Message Line 2-12
Status Line 2-12
Coordinate System Axis 2-14
Axis 2-14
Triad 2-14
Drawing Area 2-15
Scroll Bars 2-15
Work Layer Indicator 2-15
Menu Bar 2-15
Update Installer 2-20
Automated Support Ticket Preparation 2-21
The Drafting Assistant 3-1
Snapping onto Geometry 3-1
Customizing the Drafting Assistant 3-4
Drafting Assistant Construction Lines 3-6
Using the Drafting Assistant's Construction Lines 3-7
Permanent Stroke Construction Lines 3-8
The Construction Command 3-9
Creating Construction Geometry 3-10
Removing Construction Lines 3-11
Drafting Assistant and Display 3-11
Selecting Objects 4-1
Objects 4-1
Indicating Selection & Preferences 4-2
Pick Box 4-2
Select Fence 4-2
...... 4-3
Selection Color 4-3
Selection Process 4-4
Selecting Objects with the Selection Tool 4-4
Selecting Points 4-7
Selecting Objects Using Other Tools 4-9
Selecting Objects with Commands 4-9
Deselecting 4-11

Deep Select Tool ... *4-11*
Using the Deep Select Tool to Apply a Color ... 4-13
Eye Dropper Tool ... *4-14*
Using the Eye Dropper Tool ... 4-15
Editing Objects ... 5-1
Basic Editing Commands ... *5-1*
Cut ... 5-1
Copy ... 5-2
Copy Pict/Copy Bitmap ... 5-2
Paste ... 5-2
The Clipboard ... 5-3
Erasing/Deleting Geometry ... 5-3
Undo ... 5-3
Redo ... 5-3
Advanced Editing Commands ... *5-4*
Change Resolution ... 5-4
Change Layer ... 5-6
Grouping Objects ... 5-7
Arrange Command ... 5-8
Align Command ... 5-9
Object Commands ... *5-11*
Change Direction ... 5-11
Change Object Type ... 5-12
Simplify Object ... 5-19
Edit Objects ... 5-19
Links ... 5-25
Design Explorer ... 5-26
Drawing Techniques ... 6-1
Object Creation Methods ... *6-1*
Creating a Line by Clicking ... 6-2
Status Line ... *6-2*
Using the Status Line ... 6-3
Status Line Entries and Tool Operations ... 6-3
Using the Status Line with Drawing Tools ... 6-4
Creating Geometry Offset from a Point ... 6-5
Coordinate System Axis ... *6-5*
Choosing a Coordinate System ... 6-6
Message Line ... *6-6*
Trackball ... *6-6*
Sphere Trackball ... 6-7
Step Trackball ... 6-7
Trackball View Menu ... 6-9
View Rotation Short Cut ... 6-10
Drawing Display Commands ... *6-10*
Show/Hide Triad ... 6-10
Show/Hide Axis ... 6-10
Show/Hide Work Plane ... 6-11
Show/Hide Points ... 6-11
Grid ... 6-11
Drawing at Full Scale ... *6-11*
Default versus Selected Object Settings ... *6-12*
Escape Key ... *6-12*
Introduction to Wireframe Modeling ... 7-1
Wireframe Tools ... *7-2*
Wireframes and the Drafting Assistant ... *7-3*
Wireframes, Selection and Display ... *7-3*

Changing an Object's Resolution 7-3
Objects Characteristics and Edit Objects.................. 7-4
Transforming Geometry 8-1
Transformation Tools.................. 8-1
Geometric Characteristics.................. 8-1
Item vs. Faces.................. 8-1
Move Tool 8-2
Rotate Tools.................. 8-3
Resize/Scale Tool 8-5
Mirror Tool.................. 8-7
Copy along Path Tool.................. 8-9
Linear Duplicate Tool 8-10
Polar Duplicate Tool.................. 8-13
Spherical Duplicate Tool 8-16
Cylindrical Duplicate Tool.................. 8-18
Align Objects Tool 8-21
Transforming Techniques 8-21
Moving Objects with Tools 8-21
Moving Objects to Another Layer 8-23
Copying Objects with Tools.................. 8-23
Architectural Features 9-1
Wall Tool.................. 9-2
Using the Wall Tool 9-2
Door Tool 9-3
Window Tool 9-3
Pen Settings.................. 10-1
Default versus Selected Object Settings.................. 10-1
Changing the Pen Characteristics of an Object 10-1
Changing the Default Pen Characteristics 10-1
Style.................. 10-1
Creating a New Pen Style 10-2
Deleting a Pen Style.................. 10-2
Modifying a Pen Style 10-2
Color 10-2
Displaying the Color Palette.................. 10-3
Specifying the Color of the Current Pen.................. 10-3
Defining Colors.................. 10-3
Weight.................. 10-4
Units 10-4
Pixels.................. 10-4
More 10-5
Pattern 10-5
Line Pattern Manager.................. 10-6
New Patterns.................. 10-6
Construction Line Pattern.................. 10-8
Arrowheads.................. 10-8
Arrowhead Types 10-8
Line Tools 11-1
Single Line Tool 11-1
Using the Single Line Tool 11-2
Drawing a Line Perpendicular to Another Object 11-2
Drawing a Line Tangent or Perpendicular to an Arc, Circle or Ellipse 11-2
Geometric Characteristics.................. 11-2
Mid-point Line Tool 11-2
Using the Mid-point Line Tool 11-3
Geometric Characteristics.................. 11-3

Connected Lines Tool *11-3*
Using the Connected Lines Tool 11-3
Geometric Characteristics 11-4
Parallel Line Tool *11-4*
Using the Parallel Line Tool 11-4
Geometric Characteristics 11-4
Point Tool *11-5*
Using the Point at an XYZ Location Tool 11-5
Using the Points on a Curve Tool 11-5
Using the Points on a Surface Tool 11-6
Geometric Characteristics 11-6
Modifying Lines *11-6*
Arc & Circle Tools 11-1
Arc Tools *11-1*
Center-point Arc Tool 11-1
3-point Arc Tool 11-2
Tangent-point Arc Tool 11-3
Circle Tools *11-3*
Center-point Circle Tool 11-4
Opposite-Point Circle Tool 11-4
3-point Circle Tool 11-5
Ellipse & Conic Tools 12-1
Ellipse Tools *12-1*
2-point Center Ellipse Tool 12-2
Opposite-corner Ellipse Tool 12-3
3-point Center Ellipse Tool 12-3
3-corner Ellipse Tool 12-4
Modifying Ellipses 12-5
Conic Tools *12-5*
2-point Conic Tool 12-5
3-point Conic Tool 12-6
4-point Conic Tool 12-7
Polygon Tools 13-1
Rectangle Tool *13-2*
Center/Corner Rectangle Tool 13-2
Diagonal Rectangle Tool 13-2
Center/Axis Rectangle Tool 13-3
Corner/Axis Rectangle Tool 13-3
Geometric Characteristics 13-4
Inscribed Polygon Tool *13-4*
Using the Inscribed Polygon Tool 13-4
Geometric Characteristics 13-5
Circumscribed Polygon Tool *13-5*
Using the Circumscribed Polygon Tool 13-5
Geometric Characteristics 13-6
Star Polygon Tool *13-6*
Using the Star Polygon Tool 13-6
Geometric Characteristics 13-7
Arbitrary Polygon Tool *13-7*
Geometric Characteristics 13-7
Polygon from Curves Tool *13-8*
Geometric Characteristics 13-8
Spline Tools 14-1
Through-point B-spline Tool *14-2*
Using the Through-Points B-Spline Tool 14-2
Geometric Characteristics 14-3

Vector Spline Tool ... *14-3*
Using the Vector Spline Tool ... 14-4
Geometric Characteristics ... 14-4
Bezier Spline Tool ... *14-4*
Using the Bezier Spline Tool ... 14-5
Geometric Characteristics ... 14-5
Freehand Spline on Surface Tool ... *14-5*
Using the Freehand Spline Tool ... 14-5
Using the Freehand Spline on a Surface Tool ... 14-5
Helix Curve Tool ... *14-6*
Using the Helix Curve Tool ... 14-6
Geometric Characteristics ... 14-7
Add Control Point Tool ... *14-7*
Using the Add Spline Control Point Tool ... 14-7
Remove Control Point Tool ... *14-7*
Using the Remove Control Point Tool ... 14-8
Modify Slope Tool ... *14-8*
Using the Modify Slope Tool - Release Slopes Option ... 14-8
Using the Modify Slope Tool - Reference Option ... 14-9
Using the Modify Slope Tool - Reverse Option ... 14-9
Using the Modify Slope Tool - Explicit Option ... 14-10
Using the Modify Slope Tool - Release Magnitudes Option ... 14-10
Elevate Curve Tool ... *14-10*
Using the Elevate Curve Tool ... 14-11
Fair Curve Tool ... *14-11*
Using the Fair Spline Tool ... 14-11
Wireframe Editing Tools ... 15-1
Fillet and Chamfer Tools ... *15-1*
2-entity Fillet Tool ... 15-2
3-entity Fillet Tool ... 15-2
2-entity Chamfer Tool ... 15-3
Angular Chamfer Tool ... 15-3
Trim Tools ... *15-4*
Simple Trim Tool ... 15-4
Relimit Tool ... 15-5
Segment Tool ... 15-5
Divide Tool ... 15-6
Corner Trim Tool ... 15-7
Divide at Location Tool ... 15-7
Connect Curve Tool ... 15-7
Curve Utility Tools ... *15-9*
Offset Curve Tool ... 15-9
Extrude Curve Tool ... 15-12
Revolve Curve Tool ... 15-12
Project Curve Tool ... 15-13
Explode Edge Tool ... 15-16
Introduction to Surface Modeling ... 16-1
Surface Modeling Tools ... *16-2*
Surface Primitive Tools ... 16-3
Surface From Curves Tools ... 16-3
Feature Surface Tools ... 16-3
Surface Utilites Tools ... 16-3
Local Surface Tools ... 16-3
Surfaces, Selection and Display ... *16-3*
Changing an Object's Resolution ... 16-3
Surface Associativity ... *16-4*
Cutting and Pasting Surfaces ... *16-4*

Exporting Surfaces *16-4*
Objects Types and Edit Objects *16-5*
Surface Control Vertex Modification *16-5*
Performing a Control Vertex Modification 16-5
Surface Primitive Tools **17-1**
Infinite Plane Tool *17-1*
Using the X-, Y- and Z-Station Planes Option 17-2
Using the Normal and Location Plane Option 17-2
Using the Infinite Plane Tool to Create a Three Point Plane 17-2
Geometric Characteristics 17-9
Sphere Surface Tool *17-9*
Block Surface Tool *17-9*
Cylinder Surface Tool *17-9*
Pyramid Surface Tool *17-10*
Surface From Curves Tools **18-1**
Net Surface Tool *18-1*
Using the Net Surface Tool 18-2
Geometric Characteristics 18-2
Ruled Surface Tool *18-3*
Geometric Characteristics 18-3
Skin Surface Tools *18-3*
Standard Skin Surface 18-5
Closed Skin Surface 18-6
Skin Curve Surface Tool 18-6
Skin with Curve Guides Surface Tool 18-7
Geometric Characteristics 18-9
Cover Surface Tool *18-9*
Using the Cover Surface Tool 18-9
Geometric Characteristics 18-10
Revolved Surface Tool *18-10*
Revolved Surface about Two Points Tools 18-11
Revolved Surface about a Curve Tool 18-11
Geometric Characteristics 18-12
Sweep Surface Tool - Along 2 Points *18-12*
Sweep Surface Tool - Along 1 Rail 18-13
Sweep Surface Tool - Along 2 Rails 18-14
Sweep Rules 18-15
Tube Surface Tool *18-15*
Using the Tube Surface Tool 18-15
Geometric Characteristics 18-16
Derived Surface Tools **19-1**
Offset Surface Tool *19-1*
Using the Offset Surface Tool 19-2
Lofted Surface Tool *19-2*
Using the Lofted Surface Tool 19-3
Draft Surface Tool *19-4*
Using the Draft Surface Tool 19-5
Fillet Surface Tool *19-5*
Using the Fillet Surface Tool 19-5
Tangent Cover Tool *19-6*
Surface Utility Tools **20-1**
Surface Editing Tools *20-1*
Plane/Surface Intersection Tool 20-1
Curve/Surface Projection Tool 20-4
Surface/Surface Intersection Tool 20-7
Curve/Surface Intersection Tool 20-7

Boolean Surface Tools 20-8
Connect Surface Tool 20-10
Split Surface Tool 20-10
Silhouette Curves Tool 20-12
Surface Modification Tools 21-1
Match Surface Tool 21-1
Rules and Notations for using the Match Surface Tool and Options 21-1
Match G1 Slope Option 21-2
Match G2 Slope Option 21-3
Rebuild Surface Tool 21-4
Using the Rebuild Surface Tool 21-4
Untrim Surface Tool 21-5
Using the Untrim Surface Option 21-5
Elevate Surface Tool 21-5
Insert Knot Tool 21-6
Using the Insert Knot Tool - At Location Option 21-7
Using the Insert Knot Tool - Across Surface Option 21-7
Introduction to Solid Modeling 22-1
Solid Modeling Tools 22-2
Primitives 22-2
Profiles 22-2
Features 22-3
Solid Utility Tools 22-3
Local Solid Face Tools 22-3
Drafting Assistant and Solids 22-3
Solids, Selection and Display 22-4
Displaying Solid Edges 22-4
Object Resolution 22-4
Solid Associativity 22-5
Solids and Instances 22-5
Instances and Moving Solids 22-5
Instances, Copying Solids and Associativity 22-5
Cutting and Pasting Solids 22-7
Cutting and Pasting on the Macintosh 22-7
Exporting Solids 22-7
Object Types and Edit Objects 22-7
Error Messages 22-7
Solids Primitive Tools 23-1
Sphere Primitive Tool 23-1
Sphere 1 Point Option 23-2
Sphere 2 Point Option 23-2
Sphere by Diagonals Option 23-2
Slab Primitive Tool 23-3
Using the Slab Tool 23-3
Block Primitive Tool 23-4
Block 1 Point Option 23-4
Block 2 Point Option 23-5
Block by Diagonals Option 23-5
Cylinder Primitive Tool 23-6
Cylinder 1 Point Options 23-6
Using the Cylinder 1 Point Option 23-6
Cylinder 2 Point Option 23-7
Cylinder by Diagonals Option 23-7
Cone Primitive Tool 23-8
Cone 1 Point Option 23-8
Cone 2 Point Option 23-9

Cone by Diagonals Option 23-9
Torus Primitive Tool *23-10*
Torus 1 Point Option 23-10
Torus 2 Point Option 23-11
Torus by Diagonals Option 23-11
Prism Primitive Tool *23-12*
Prism 1 Point Option 23-12
Prism 2 Point Option 23-13
Prism by Diagonals Option 23-13
Pyramid Primitive Tool *23-14*
Pyramid 1 Point Option 23-14
Using the 1 Point Pyramid Tool 23-14
Pyramid 2 Point Option 23-15
Using the 2 Point Pyramid option 23-15
Pyramid by Diagonals Option 23-15
Ellipsoid Primitive Tool *23-16*
Ellipsoid 1 Point Options 23-16
Ellipsoid 2 Point Option 23-16
Ellipsoid by Diagonals Option 23-17

Solids from Curves Tools 24-1
Creating Associative Profiles in Cobalt & Xenon *24-1*
Creating Solids from Profiles *24-2*
Lathed Solid Tool *24-2*
Using the Lathed Solid Tool 24-2
Geometric Characteristics 24-4
Extrude Solid Tool *24-4*
Using the Extrude Solid Tool 24-7
Geometric Characteristics 24-8
Swept Solid Tool *24-9*
Using the Swept Solid Tool 24-9
Geometric Characteristics 24-10
Sweep 2 Rail (Paths) Solid *24-11*
Using the Sweep 2 Rail Solid Tool 24-11
Geometric Characteristics 24-11
Cutout Feature Tool *24-11*
Using the Cutout Feature Tool 24-12
Geometric Characteristics 24-12
Protruded Feature Tool *24-12*
Using the Protruded Feature Tool - Vector Option 24-13
Using the Protruded Feature Tool - Face Option 24-13
Geometric Characteristics 24-13
Cutout and Protruded Curve Checks *24-13*
Curve Gap Check 24-13
Planar Curves Check 24-14
Skinned Solid Tool *24-14*
Basic Skinned Solid Option 24-16
Skinned Solid with Guide Curves Option 24-17
Geometric Characteristics 24-17
Solids from Profiles Curve Checks *24-17*
Closed Curves Check 24-17
Pipe Solid Tool *24-18*
Using the Pipe Solid Tool 24-18
Geometric Characteristics 24-19

Solid Feature Tools 25-1
Blend Tools *25-2*
Advanced Blend Capabilities 25-3

Constant Blend Tool 25-5
Variable Blend Tool 25-12
Chamfer Edge Tools 25-16
Advanced Chamfer Capabilities 25-17
Constant Length Chamfer Tool 25-18
Variable Chamfer Edge Tool 25-21
Hole Feature Tools 25-25
Hole Direction 25-27
Simple Hole Tool 25-27
Counter Bore Tool 25-28
Counter Sink Tool 25-28
Boss Feature Tool 25-29
Boss Direction 25-29
Using the Boss Feature Tool 25-30
Geometric Characteristics 25-30
Shell Solid Tool 25-30
Using the Shell Solid Tool 25-31
Geometric Characteristics 25-31
Bend Tool 25-31
Using Bend Center 25-33
Geometric Characteristics 25-34
Twist Solid Tool 25-34
Using the Twist Solid Tool 25-35
Geometric Characteristics 25-35
Solid Utility Tools 26-1
Union Solid Tool 26-2
Using the Union Solid Tool 26-2
Subtract Solid Tool 26-2
Using the Subtract Solid Tool 26-2
Intersect Solid Tool 26-3
Using the Intersect Solid Tool 26-3
Trim Solid Tool 26-3
Split Solid Tool 26-5
Stitched Solid Tool 26-5
Thicken Solid Tool 26-6
Using the Thicken Solid Tool 26-7
Lofted Solid Tool 26-8
Using the Lofted Solid Tool 26-8
Rib Solid Tool 26-9
Using the Rib Solid Tool 26-10
Lip Solid Tool 26-11
Using the Lip Solid Add Feature Tool 26-11
Using the Lip Solid Remove Feature Tool 26-12
Solid Face Modification Tools 27-1
Draft Face Tool 27-1
Rules 27-1
Match Face Tool 27-3
Planar Example 27-3
Elliptical Example 27-4
Using the Match Face Tool 27-4
Geometric Characteristics 27-4
Move Solid Element Tool 27-4
Using the Move Solid Element Tool 27-5
Offset Face Tool 27-6
Using the Offset Face Tool 27-6
Geometric Characteristics 27-7

Remove Face Tool ... *27-7*
Using the Remove Face Tool ... 27-7
Replace Face Tool ... *27-8*
Using the Replace Face Tool ... 27-8
Replace Face Example ... 27-8
Geometric Characteristics ... 27-9
Parting Line Solid Tool ... *27-9*
Rules ... 27-9
Using the Parting Line Solid Tool ... 27-10
Modifying Parting Line Solids ... 27-11
Geometric Characteristics ... 27-11
Deform Face Tool ... *27-11*
Deform Face with Gain Option ... 27-12
Deform Face to Point Option ... 27-13
Deform Face to Curve Option ... 27-14
Stretch Solid Tool ... *27-15*
Using the Stretch Tool ... 27-16
Assembly Modeling Tools ... 28-1
Assembly Connect Tool ... *28-1*
Assembly Mate Tool ... *28-2*
Assembly Align Tool ... *28-3*
Assembly Insert Tool ... *28-4*
Object Properties & Verification ... 29-1
Geometric Properties ... *29-1*
X, Y, Z Command ... 29-1
Angle 3 Points Command ... 29-1
Distance Point-to-Point Command ... 29-2
Minimum Distance Command ... 29-2
Length Command ... 29-2
Area Command ... 29-2
Volume Command ... 29-3
Properties Command ... 29-3
Interference Command ... 29-4
Object Counts Command ... *29-7*
Check Object Command ... *29-7*
Surface Evaluation ... *29-9*
Evaluation Through the Verify Menu and Edit Object Window ... 29-10
Surface Evaluation Through the Advanced Rendering Feature ... 29-22
Draft Evaluation ... *29-23*
Draft Evaluation Through the Verify Menu or Edit Object Window ... 29-24
Draft Evaluation Through Advanced Rendering ... 29-26
Mesh Creation Tools ... 30-1
3-point Mesh Tool ... *30-2*
Using the 3-point Mesh Tool ... 30-2
Geometric Characteristics ... 30-2
4-point Mesh Tool ... *30-2*
Using the 4-point Mesh Tool ... 30-2
Geometric Characteristics ... 30-3
Ruled Mesh Tool ... *30-3*
Using the Mesh Curve Tool ... 30-3
Geometric Characteristics ... 30-4
Extrude Mesh Tool ... *30-4*
Using the Extrude Mesh Tool ... 30-4
Geometric Characteristics ... 30-4
Mesh and Rendering ... *30-5*
Mesh Notations ... *30-5*

Text **31-1**
Text Tools *31-1*
Horizontal Text Tool 31-1
Text Along a Curve Tool 31-2
Text at an Angle Tool 31-2
Geometric Characteristics 31-3
Bounded Box Text Tool 31-3
Text Commands *31-3*
Font 31-3
Size 31-4
Style 31-4
Case 31-5
Text Handling *31-5*
Editing Text 31-5
Moving Text 31-5
Screen Horizontal Text 31-5
Convert Text into Lines, Curves, Poligons. 31-6
Dimensions **32-1**
Dimension Menu *32-1*
Dynamic Dimensions *32-1*
Dimension Tools *32-2*
Dimensions and the Work Plane 32-2
Dimension Smart Pointer 32-3
Dimensioning Objects and Placement 32-3
Moving the Dimension 32-3
Dimension Status Line Fields 32-3
Automatically Placing Dimensions on a Separate Layer 32-4
Dimensions and Edit Objects 32-4
Dimensioning Surfaces and Solids 32-5
Smart Dimension Tool 32-5
Using the Smart Dimension Option 32-5
Horizontal Dimension Tools *32-5*
Horizontal Dimension Option 32-5
Horizontal Base Line Dimension Option 32-6
Horizontal Chain Dimension Option 32-6
Vertical Dimension Tools *32-6*
Vertical Dimension Tools 32-7
Vertical Base Line Dimension Option 32-7
Vertical Chain Dimension Option 32-7
Oblique Dimension Tool *32-8*
Using the Oblique Dimension Tool 32-8
Ordinate Dimension Tools *32-8*
Horizontal Ordinate Dimension Option 32-8
Vertical Ordinate Dimension Option 32-9
Radial Arrow Out Dimension Tool *32-9*
Using the Radial Arrow Out Dimension Tool 32-9
Radial Arrow In Dimension Tool *32-9*
Using the Radial Arrow In Dimension Tool 32-9
Diameter Arrow In Dimension Tool *32-10*
Using the Diameter Arrow In Dimension Tool 32-10
Diameter Arrow Out Dimension Tool *32-10*
Using the Diameter Arrow Out Dimension Tool 32-10
Angular Dimension Tool *32-11*
Using the Angular Dimension Tool 32-11
Center Mark Dimension Tool *32-11*
Using the Center Mark Dimension Tool 32-11

Balloon Dimension Tools *32-12*
Using a Balloon Dimension Tools 32-12
Length Along Curve Dimension Tool *32-12*
Using the Length Along Curve Dimension Tool 32-12
GD&T Feature Control Tool *32-13*
Using the Datum Feature Option 32-13
Using the Datum Target Option 32-13
Using the Feature Control Frame 32-13
Dimension Appearance *32-15*
Linear 32-15
Linear Tolerance 32-16
Angular **32-17**
Text 32-17
Font 32-18
Size 32-18
Style 32-19
Color 32-19
Weight 32-20
Layer 32-20
Arrowheads 32-21
Arrow Size Command 32-21
Witness Lines 32-22
Dimension Standards 32-22
Dimensions and Drawing Views *32-25*
Fill and Crosshatching 25-1
Fill Pattern and Fill Color *25-1*
Fill Pattern 25-2
Fill Color 25-4
Crosshatching *25-4*
Cross Hatch Patterns 25-4
Crosshatch Command 25-6
Hatch Command 25-7
Editing Hatching 25-7
Crosshatching and Section Cuts 25-7
Symbols 34-1
Symbol Palette *34-1*
Symbol Manager *34-2*
Creating Symbols *34-3*
Symbol Tools *34-4*
Symbol 1 Point Tool 34-4
Symbol 2 Points Tool 34-4
Symbol Example *34-5*
Transforming a Symbol *34-6*
Editing a Symbol *34-7*
Symbols and Rendering *34-7*
Symbols and Files *34-7*
Using Symbols in Other Files 34-7
Importing 34-7
Viewing Geometry 35-1
The View Palette *35-1*
Shade Options *35-2*
Shade Now 35-2
Shade Options 35-2
Lights & Facets 35-8
Zooming *35-8*
Zoom Commands 35-8

Zoom, Pan and Rotate Tools 35-9
Using the Dynamic Pan Tool 35-9
Dynamic Zoom Tool 35-9
Stroke Zoom 35-10
Zooming and Views 35-10
Panning a View 35-11
View Displays 35-11
Pre-defined Views 35-11
User-defined Views 35-12
Modifying a View 35-14
Deleting a View 35-15
View Rotation 35-16
Choosing a Predefined View 35-16
Choosing a User-defined View 35-16
Choosing an Undefined View 35-16
Dynamic Rotate Tool 35-16
Using the Shift and Arrow keys 35-17
View Commands 35-17
Redraw Screen 35-17
View the Plane 35-17
Show/Hide 35-17
Tile 35-18
Arrange Icons (Windows only) 35-18
Open File Windows 35-18
Views and Planes 35-19
Views and Zoom Scale 35-19
Layers 36-1
Layer Manager 36-2
Default Layers 36-4
Creating New Layers 36-4
Using the Layer Manager 36-4
Using the Work Layer Indicator 36-5
Using the Isolate Layer Command 36-5
Renaming Layers 36-5
Deleting Layers 36-5
Hiding Layers 36-5
Hiding One Layer 36-5
Hiding all Layers 36-6
Showing Layers 36-6
Showing One Layer 36-6
Showing all Layers 36-6
Locking Layers 36-6
Unlocking Layers 36-6
Making a Layer the Active Work Layer 36-7
Using the Layer Manager 36-7
Using the Work Layer Indicator 36-7
Using the Isolate Layer Command 36-7
Increment Layer 36-7
Decrement Layer 36-7
Layers and Pen Color 36-8
Using the Layer Color Override 36-8
Layer Color Warning 36-8
Layers and Pen Style 36-9
Using the Pen Style Override 36-9
Layers and Copying/Pasting Objects 36-9
Planes 37-1

Work Plane ... *37-2*
Show Work Plane Command ... 37-2
Work Plane ... 37-3
Setting the Work Plane ... 37-5
Work Plane Snapping ... 37-6
Work Plane Dimming ... 37-7
Moving the Origin ... 37-7
Offsetting the Work Plane ... 37-8
Work Plane and Views ... *37-8*
Standard Views ... 37-8
View the Plane ... 37-8
Work Plane and World Coordinates ... *37-8*
Lighting ... 38-1
Directional Lighting ... *38-1*
Light Palette ... 38-1
Ambient Lighting ... *38-14*
Modifying the Lights ... 38-14
Default Lighting ... *38-16*
User-defined Lighting Layouts ... *38-16*
Photo-realistic Sunlight for Stills and Animations ... *38-17*
Lights & Facets ... *38-19*
Display Materials, Decals & Environments ... 39-1
Render Library ... *39-2*
Render Library Items ... 39-2
Using the Materials Library ... 39-3
Using the Decals Library ... 39-7
Using the Backgrounds and Foregrounds Libraries ... 39-12
User-defined Images ... 39-14
Editing the Rendered Scene ... *39-17*
Material Editing ... 39-17
Background and Foreground Editing ... 39-24
Photo-realistic Environment Maps for Stills and Animations ... *39-26*
Using a Photo-realistic Environment Map ... 39-28
Photo-realistic Images ... 40-1
Advanced Rendering Settings ... *40-2*
Custom Render Presets ... 40-4
Interleaved Wireframe Geometry ... *40-5*
Raytrace Rendering ... *40-7*
Photo-realistic Rendering Settings ... 40-7
Photo-realistic Rendering Commands ... 40-9
Render to File ... 40-11
Display Last Image Command ... 40-12
Rendering and Multi-processors ... 40-12
Animation ... 41-1
QuickTime Movies with Camera Movement ... 41-1
QuickTime VR Movies ... 41-7
File Management ... 42-1
Using Documents ... *42-1*
File Use and Operating System Short Cuts ... 42-1
File Menu Commands ... 42-2
Files and Preferences ... *42-5*
Files and Locking ... *42-6*
File Size & Deleting ... *42-6*
Uninstalling This Program on Windows ... *42-6*
Importing & Exporting ... 43-1

Importing *43-1*
Supported Import Formats 43-2
Import Command 43-7
Importing Tips and Notations 43-8
Exporting *43-9*
Supported Export Formats 43-10
Export Command 43-18
Exporting Tips and Notations 43-19
Batch Convert Tool *43-20*
Drawing Composition for Cobalt & Xenon **44-1**
Model to Sheet Command *44-1*
Using the Model to Sheet Command 44-3
Undoing the Model to Sheet Command 44-4
Drawing Views *44-5*
Activating Views 44-6
Drawing View Menu 44-6
Sheet Tools *44-13*
Drawing View Tool 44-13
Auxiliary View Tool 44-13
Section View Tools 44-15
Detail View Tool 44-18
Manual View Regeneration Tool 44-19
Unfolding Views *44-20*
Editing a Drawing View *44-20*
Adding Objects to a View 44-20
Editing Objects in the View 44-21
Editing View Characteristics 44-22
Layout Templates *44-23*
Printing the Layout *44-24*
Attributes and Bill of Materials **45-1**
BOM User Defined Templates *45-3*
Page Setup and Printing **46-1**
Page Layout Command *46-1*
Setup Mode Section 46-2
Drawing Size Section 46-5
Drawing Size and Short Cut Key 46-12
Printing a Drawing *46-12*
Print Setup (Windows); Page Setup (Macintosh) 46-12
Print Command 46-13
Printing/Plotting Region 46-13
Moving the Print Boundaries 46-13
Printing and Rendering 46-14
Print Window *46-14*
Dimensionally Constrained Parametrics **47-1**
Sketch Mode *47-1*
Reference Edges 47-2
Exiting Sketch Mode 47-2
Sketch Mode Tools *47-2*
Sketch Tool 47-2
Auto Constraints Tool 47-2
Horizontal Constraint Tool 47-3
Vertical Constraint Tool 47-3
Coincident Constraint Tool 47-3
Tangent Constraint Tool 47-5
Parallel Constraint Tool 47-5
Perpendicular Constraint Tool 47-5

Fixed Constraint Tool 47-6
Concentric Constraint Tool 47-6
Symmetric Constraint Tool 47-7
Equal Constraint Tool 47-7
Colinear Constraint Tool 47-7
Offset Constraint Tool 47-8
Dimension-driven Geometry 47-8
Constraint Animation Tool 47-8
Dimensional Constraint Animation Controller 47-8
Dimensional Constraint Animation Settings 47-8
Running the Animation 47-9
Deleting Constraints 47-9
Verifying Constraint Relations 47-9
Constraint Preferences 47-9
Variables and Equations 48-1
Using Equations & Variables 48-1
Using the Variables/Equations Dialog Box 48-2
Using the Edit Objects Box for Variables & Equations 48-3
Defining Equations 48-3
Nested Equations 48-3
Conditional Equations 48-4
Preference Settings 49-1
Default versus Selected Object Settings 49-1
Preferences 49-2
Colors 49-3
Dimensional Constraint Manager (DCM) (Cobalt Only) 49-4
Display 49-5
.... 49-6
Drafting Assistant 49-7
File Associations 49-9
Filing 49-9
General 49-11
Grid 49-12
Localization 49-14
Mouse 49-14
Paths 49-14
Select 49-15
Skins 49-17
SpaceMouse 49-17
Units 49-19
Alchemy 49-20
Changing the Preference Settings with the Preferences Command 49-20
Changing the Preference Settings Manually 49-20
Preferences and Object Creation 49-21
Pen and Dimension Preferences 49-21
Shortcuts 49-21
Creating a New Shortcut 49-23
Appendix A: Mathematical Operators A-1
Appendix B: Special Characters B-1
Windows: B-1
Macintosh: B-1
Appendix C: ShortCuts C-1
Short Cut C-1
Key C-1
Action C-1
Appendix D: Spline Text Files D-1

Appendix E: Photo-realism Fundamentals E-1
Creating the Environment *E-2*
Modeling Tips for More Realism E-2
Creating Floor Surfaces and Backdrops E-4
How to View the Scene E-6
Composition: Arranging Objects in the Scene E-7
Making Reflective Objects Look Real E-8
Lighting the Environment *E-9*
The 3-point Light Setup E-9
Specific Lighting Situations E-10
Lighting Color E-11
Shadows E-11
Adding Realistic Materials *E-13*
Basics of Materials E-13
Types of Plastic E-15
How to Make Metal E-16
How to Make Clear Materials E-17
Image-based Textures E-18
Practice Makes Perfect E-19
Appendix F: Shader Attribute Definitions **F-1**
Color Class Attributes F-1
Displacement Class Attributes F-3
Reflectance Class Attributes F-4
Transparency Class Attributes F-5
Texture Space Class Attributes F-6
Background Class Attributes F-6
Foreground Class Attributes F-7
Glossary **G-1**
Index **I-1**

User Guide Documentation

This *User Guide* is written for both WindowsXP/Vista and Power Macintosh and Intel platforms. Before using this *User Guide* section, install any one of the three Designer Elements 3D modeling programs either Cobalt™, Xenon™ or 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-Vellum's Designer Elements 3D modeling programs provide menus and submenus for choosing commands and performing other operations.

Choosing Commands

The manual directs you to choose commands contained in submenus of other menus, like the pull down menu. For example, it might be necessary to select *Lock* in the Group submenu of the Layout menu. That appears in this manual as ***Layout>Group>Lock***.

Margin Notes

This manual includes margin notes that provide information to assist in the using of this program. There are three types of margin notes: Tip, Tech Note and Referral. These notes are given special treatment so that they are instantly recognizable and easy to significance and locate them for future reference.

Tip

A tip provides instructions for getting the most out of the program. Tips may show 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 points 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 an 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 new to surface and solids modeling, there are many terms or phrases that might be unfamiliar 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. While creating a solid, add a blend and move it, the solid is now a translated instance. The original is still located in the previous position although it is not displayed. Creating a master symbol, an instance occurs when placing the symbol in the 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 the object is copied and pasted, the associativity is broken.

Curves Refer to all lines, arcs, ellipses, conics, circles, splines and polygons created using one of the wireframe tools.

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 changing 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, it is possible to go back to the first curve and edit it and the entire object adjusts. Cobalt™ and Xenon™ 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; use it to tell the computer what to do. Use the mouse to indicate locations, choose commands, select tools and construct objects.

This manual uses the following terms for mouse activities:

Chapter Breakdown

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.

Point Move the mouse until the pointer is over the necessary item.

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.

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

Sections

Overview	Chapters 1 through 4 introduce some basic features of the Designer Elements 3D modeling programs, including the program Window, the Drafting Assistant and information on selecting objects.
Setting the Environment	Chapters 5, 6 and 7 explain pen and preference settings and drawing techniques.
Wireframe Modeling	Chapters 8 through 18 cover wireframe modeling including curve tools as lines, arcs, circles, splines, polygons, etc.
Surface Modeling	Chapters 14, 15 and 16 introduce surface modeling and provide information about surface modeling tools such as Infinite Plane, Skin, Net, Tube, Tangent, etc.
Solids Modeling	Chapters 17 through 19 explain solid modeling and contain 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.
Editing	Chapters 20 through 25 explain 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.
Adding Details	Chapters 26 through 29 contain information on text, dimensions, patterns and fills, crosshatching and symbols.
Drawing Display	Chapters 30, 31, 32 and 33 offer information on viewing the geometry, layers, planes and rendering.
Documents	Chapters 34 through 38 contain information on file management, importing, exporting, page setup and producing production drawings.
Advanced Parametrics	Chapters 39 and 40 provide information about using Cobalt's parametric capabilities.

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

Appendices

Operators	Describes all math operators accepted by the data fields.
Special Characters	Lists all special characters not directly available from the keyboard and symbols to use in this Designer Elements program.
Shortcuts	Contains a list of the default Shortcuts
Spline Text Files	Provides instructions for creating spline text files to import.

Operators	Describes all math operators accepted by the data fields.
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	Lists 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.

First Look

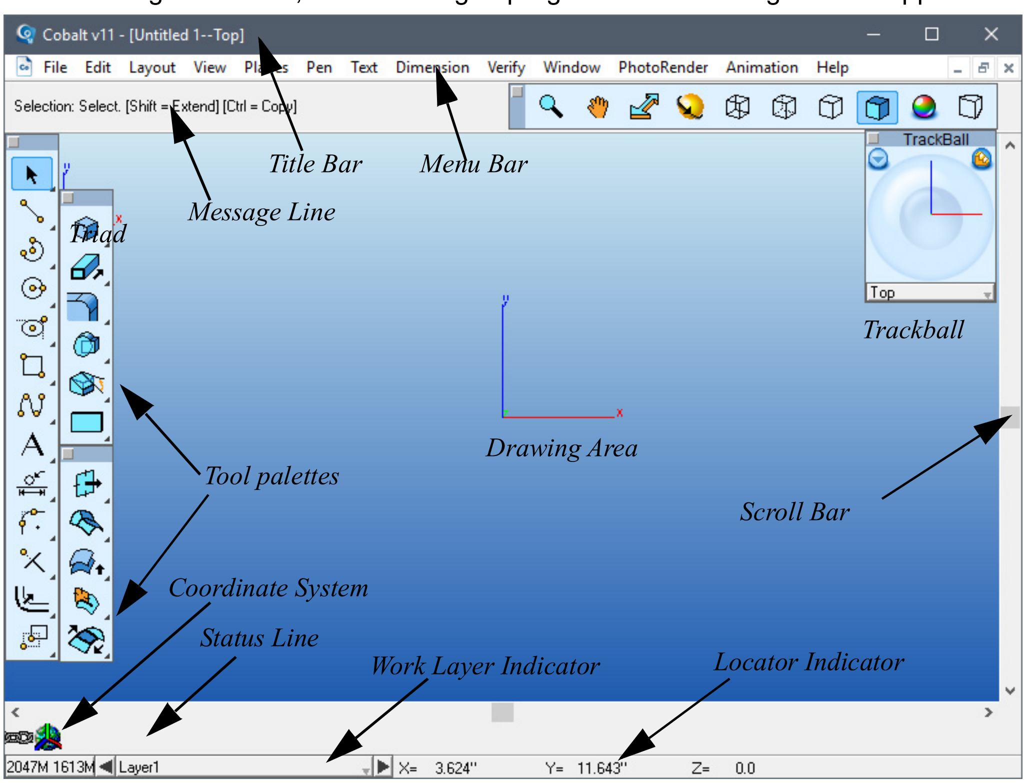
This chapter describes the “first look” components of the Ashlar-Vellum Designer Elements including Cobalt, Xenon and Argon. A brief overview of useful features may be all you need to know if you are familiar with CAD software. The following topics are covered:

- Parts of the Designer Elements Window
- Menu Bar
- Update Installer
- Automated Support Ticket Preparation

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

Parts of the Designer Elements Window

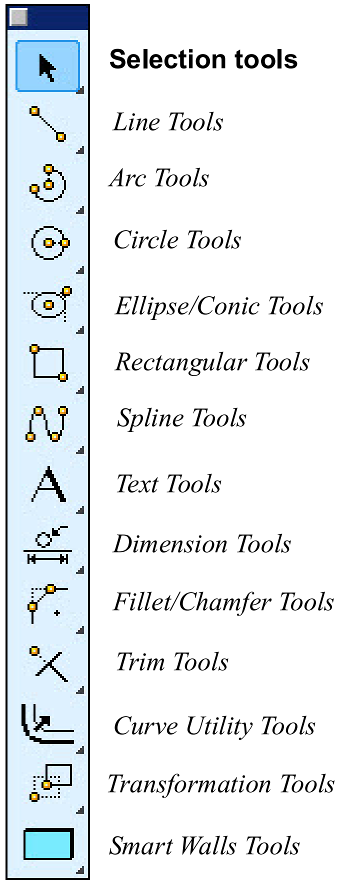
When starting the Cobalt, Xenon or Argon programs the following window appears.



Title Bar	Includes the title of the active document and buttons for controlling the window to minimize, maximize and close it.
Menu Bar	Displays the menus of commands and settings. It is possible to make choices from the menus with the mouse or by using special key combinations.
Tool Palette	Are the drawing and editing tool icons used for constructing, editing and annotating geometry. Only the most basic tool palette is displayed on initial startup. The **Surface** tool palette and **Solids** tool palette are not automatically displayed, unless specified through Preferences. See Preference Settings.
Tool Tip	Displays a brief description of the tool the cursor is held over. Tool Tips are activated or deactivated by choosing ***File>Preferences>General***. If a shortcut is assigned to a tool, the shortcut 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 the 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 the 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 for constructing and annotating geometry.
Status Line	Displays the coordinate location and other geometric characteristics of the current object based on the World coordinate system.
Coordinate System Axis	Shows the current coordinate system. It is possible to choose between the default global or world coordinate system and a user-defined coordinate system.
Scroll Bars	Move around a drawing. The scroll buttons 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.

Tool Palette

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



Selecting a Tool from the Tool Palette

1. Position the arrow pointer on the icon of the tool.
2. Click the mouse button.

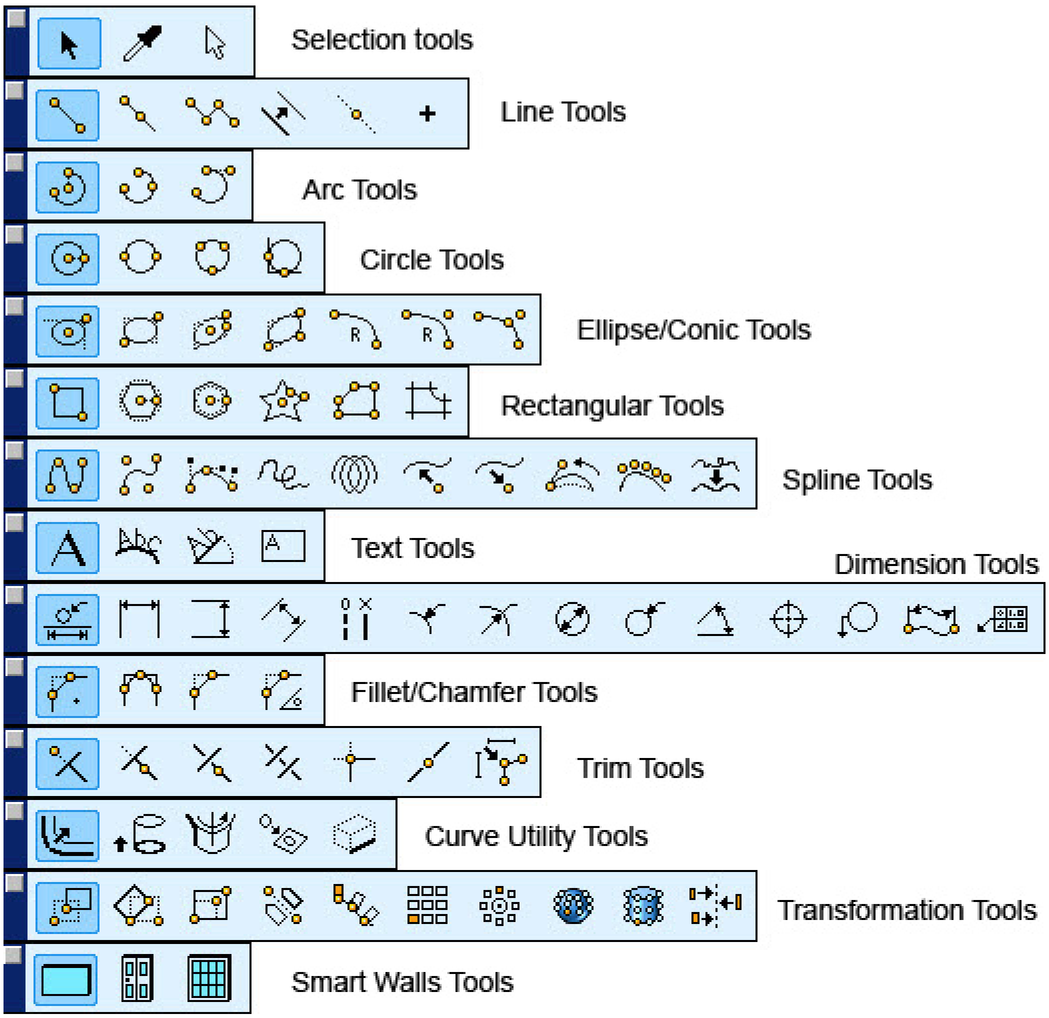
   The icon appears highlighted to indicate its selection.

Floating Tool Palettes

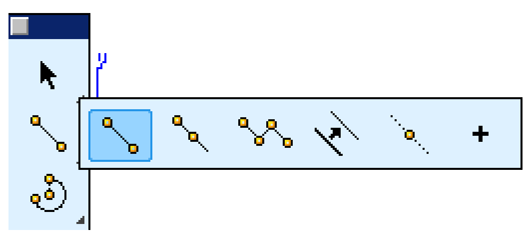
All tool palettes are floating tool palettes. Once a palette is displayed, it is possible to move it to any location in the drawing area simply by dragging the palette by its title bar. Save the palette to that location for future files by choosing ***File>Preferences>General*** and click the Save Now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

Tool Subpalettes

All of the tools in tool palettes contain a subpalette of tools with related functions. The arrow in the lower-right corner of the first icon represents the presence of a subpalette. This graphic shows the **Wireframe** tool palette with all of its submenus.

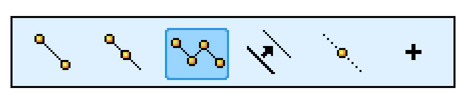


Selecting a Tool from a Subpalette



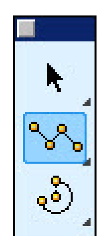
1. Position the arrow pointer on the tool.
2. Press the mouse button.

   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 the selection from the subpalette is the active or current tool.



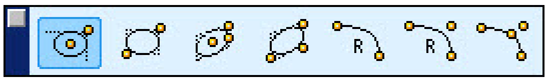
The new tool is visible in the tool palette until another tool is selected from the same subpalette. The tools in the subpalette remain in the same order; only the tool displayed in the tool palette changes.

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

To select a tool already displayed in the tool palette, click it. It is not necessary to select it from the subpalette.

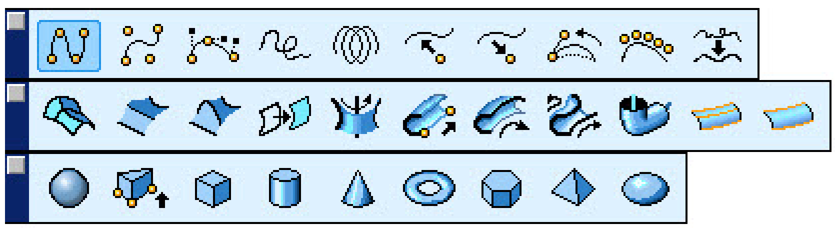
Tear Away Palettes

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



Dragging the mouse to the right across the subpalette beyond the last tool, the subpalette “tears away” from the parent tool palette. 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 tearing away multiple tool palettes and dragging them near each other, they snap to a left alignment.



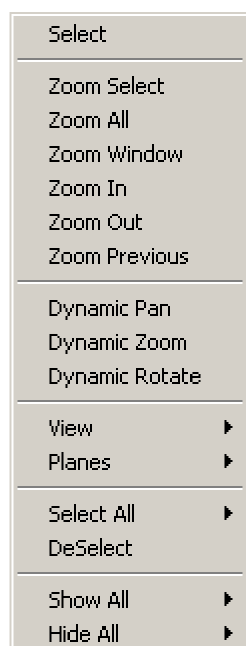
Right Mouse Button

This Designer Elements program accesses commands through the right mouse button. For Macintosh users who do not have a right mouse button, these same commands are available by holding down the CTRL 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

Click the right mouse button (Windows) or CTRL + mouse button (Macintosh) in the white space of the drawing, and the popup menu on the right appears.



The menu includes the following commands:

Zoom Select,
Zoom All,
Zoom Window,
Zoom In
Zoom Out
Zoom Previous

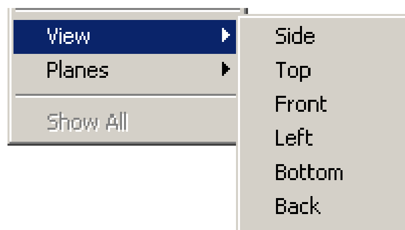
Are the same as those in the View menu.

Dynamic Pan,
Dynamic Zoom,
Dynamic Rotate

Perform the same function as the **Dynamic Pan**, **Dynamic Zoom** and **Dynamic Rotate** tools in the **View** tool palette. When one of these commands is selected, the pointer icon becomes that of the tool. It is possible to perform the operation once and then the icon reverts back to the selected tool. To perform multiple operations, hold down the SHIFT key before selecting the command. When the SHIFT key is released, the icon reverts back to the selected tool.

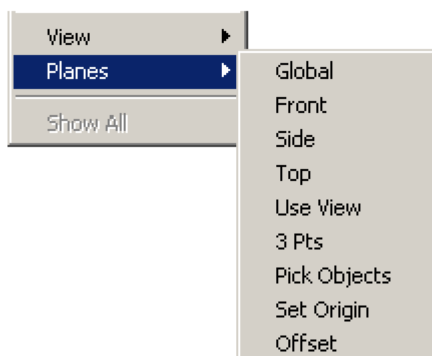
View

Displays all default views and any user-defined views.



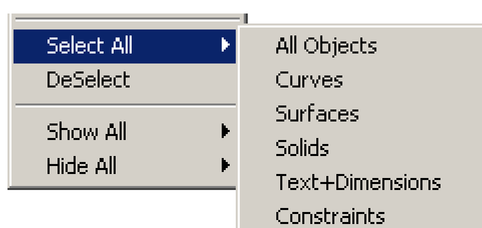
Planes

Displays the same commands as those in the Planes menu, with the exception of the *Show Work Plan* command.



Select All

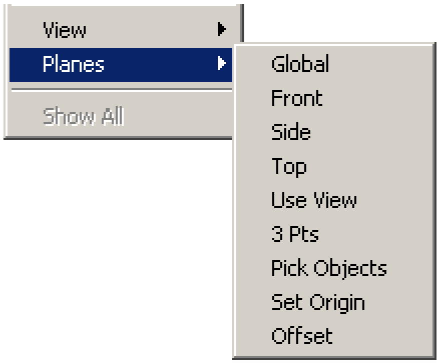
Specifies which object types to select.



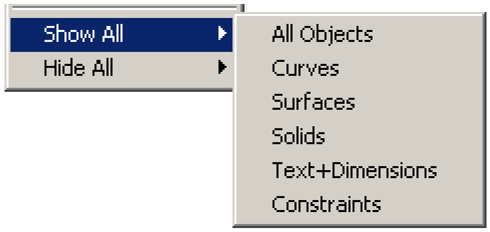
DeSelect

Deselects any selected objects.

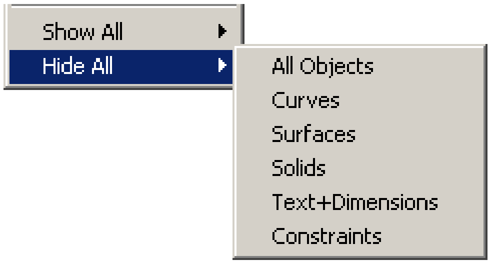
Planes Displays the same commands as those in the Planes menu, with the exception of the *Show Work Plan* command.



Show All Specifies which hidden object types to show.

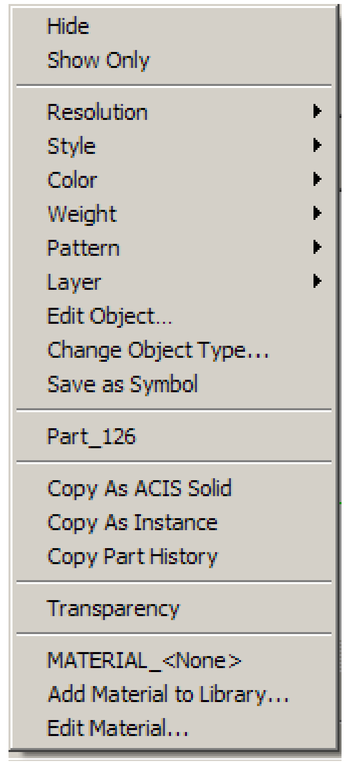


Hide All Specifies which object types to hide.



Over An Object

When an object is selected, place the pointer over the selected object and click the right mouse button (Windows) or CTRL + mouse button (Macintosh), the popup menu on the right appears.



The menu includes the following:

Hide, Show Only — Are the same as those in the Show-Hide dialog box that hide or show the object.

Resolution — Displays a menu to change the object's resolution to Super Fine, Very Fine Medium or Coarse.

Style — Displays a menu of commands for creating and modifying pen styles. A pen style is a collection of pen attributes such as color, weight and pattern.

Color — Displays a menu to change the color or the object. These are the same color options available when choosing ***Pen>Color***. The *More* option displays the color palette.

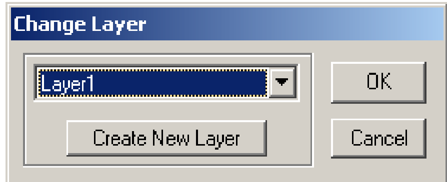
Weight — Sets the pen width of the current pen pattern and the selected lines without changing any other pen characteristics.

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 printing or plotting such lines, they appear with the weights.

It is also possible to set a line weight based on units, pixel thickness and through the More option by the point size or model space.

Layer

Changes 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:



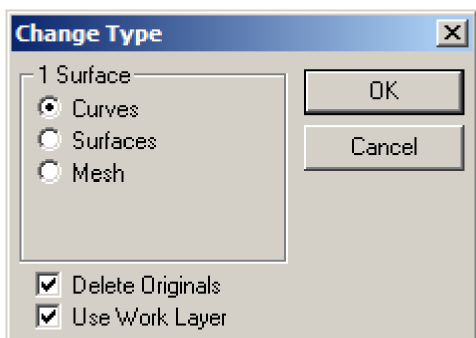
Click on the Create New Layer button and then click OK to move the object to the new layer. The new layer is titled with a default name (for example, Layer3). It is possible to change the name through the Layer Manager (***Layout>Layer Manager***).

Edit Objects

Displays the Edit Objects dialog box.

Change Object Type

Changes 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 program remembers the selected conversion type (Curves, Surface, Mesh or Solids for a Solid object) until the option type is changed. This command converts: Solids to solids, meshes, surfaces or curves. Surfaces to meshes, surfaces or curves. Curves to lines, polylines or splines. When selecting this command a dialog box similar to the one below appears.



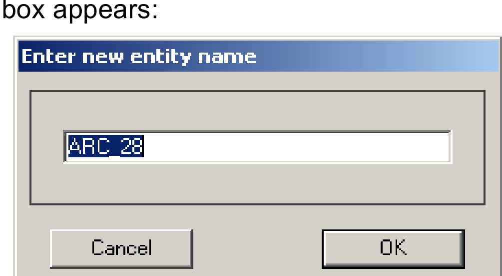
For more information See “Change Object Type” on page 5-15

Save As Symbol

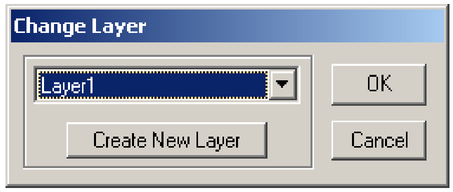
Evokes the Symbol Manager dialogue box to create a new symbol that is used to define 2D or 3D shapes that frequently occur throughout a drawing. Once the symbols are created use the **Symbol** tools to place them into the drawing.

Object Name

Displays the name of the object in the menu. Change the object’s name by clicking on the name. The following dialog box appears:



Layer
Changes 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:



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

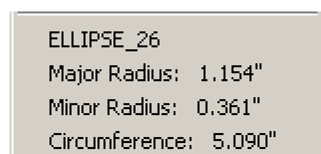
Copy as ACIS Solid
Creates a copy of the part that was right-clicked. The copy is placed directly on top of the original part. This new part contains no history of the original part.

Copy As Instance
Creates a copy of the part that was right-clicked. The copy is placed directly on top of the original part. This new part is an instance of the original, therefore any changes made to the original part will be reflected in the new part.

Copy Part History
Creates a copy of the part that was right-clicked. The copy is placed directly on top of the original part. This new part contains the exact history of the original part but is managed as a completely separate entity.

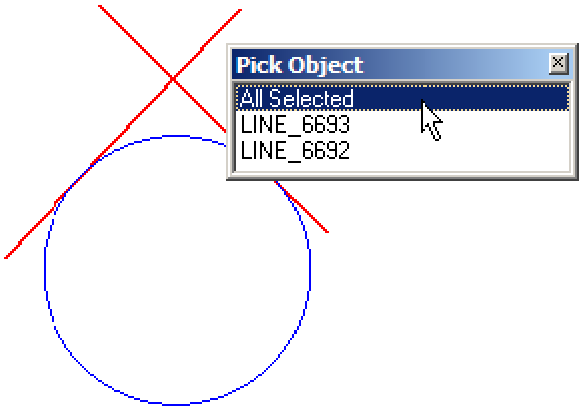
Transparency
Makes it possible to see through the part. To adjust the amount of transparency applied to the part, select ***View>Shade Options...*** and move the transparency slider.

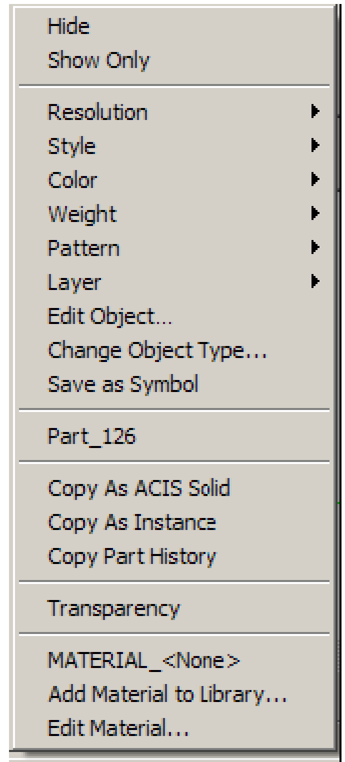
Object Characteristics
Provides geometric characteristics for a wireframe object. For example, an ellipse displays the major radius, the minor radius and circumference.



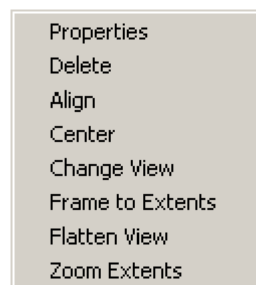
Over Multiple Selected Objects

The common properties of multiple objects can be modified via the right-click menu. The ambiguity dialog box appears if more than one item is clicked. Choose the All Selected option from the Pick Object dialog and the properties modified in the right-click pop-up menu will apply to all the selected objects.





Selected Drawing View



When a drawing view is selected and the right mouse button (Windows) or CTRL + mouse button (Macintosh) clicked, the popup menu appears:

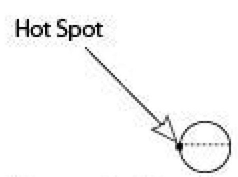
These commands are identical to those in the Drawing View pull-down menu. See “Drawing Composition for Cobalt & Xenon” for information on these commands.

Smart Pointer & Wireframe Tools

Having selected a tool, move the pointer into the drawing area. The pointer shape represents the tool.

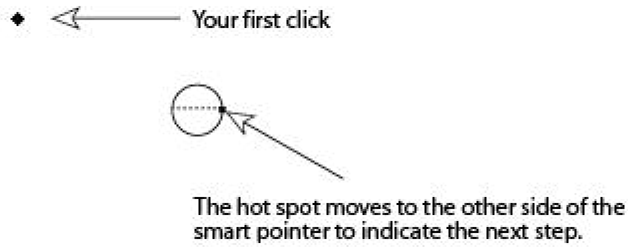
Some of the pointers, like the Single Line pointer, are simple cross-hairs. Others, such as the Opposite-point Circle pointer, resemble the tool.

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 to specify. The dot changes position on the pointer during each step of the construction.

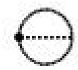


The smart pointer shows where to click next.

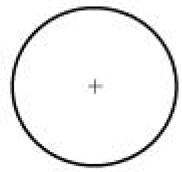
For example, the Opposite-point Circle pointer, illustrated above, shows that the first click of the mouse places a point on one edge of the circle being created. After a location is clicked, the hot spot moves to the other side of the pointer, showing that the next click places a point on the opposite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



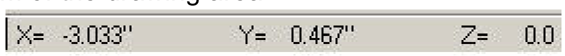
After the second location is clicked, the circle appears. The hot spot moves back to its original position on the pointer, so another circle can be created.



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 a new document is opened. If the Axis are displayed, the origin is at the intersection of the x, y and z axes.

Message Line

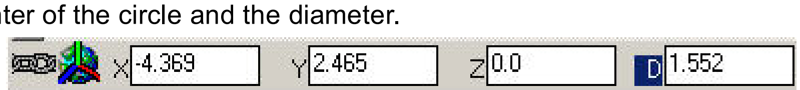
The Message Line across the top of the drawing area provides concise instructions for the use of the current tool.

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, holding 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 clicked.

Status Line

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 the **Center-point Circle** tool is selected, the Status Line shows the X,Y and Z coordinates for the center of the circle and the diameter.



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.*

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

It is possible to change any or all entries in the Status Line. If the entry is changed before the tool operation, the values automatically register when clicking in the drawing area for the particular operation. If a value is entered after the geometry is created and then ENTER (Windows) or RETURN (Macintosh) is pressed, the changes are made to the geometry and no more changes can be made 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. For 1.5 inches, enter 1.5”, 1.5i, 1.5in, 1.5 inch, etc.
- For 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.

Tip: *It is also possible to change objects with the Edit Objects command or by double-clicking on the object to display the Edit Objects dialog box.*

Copying and Pasting Status Line Entries

It is possible to 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. The *Copy* and *Paste* commands in the Edit menu cannot be used.

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

Moving between Status Fields

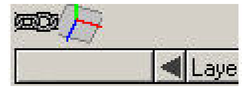
Use the TAB key to move to the right, highlighting the next field. When pressing ENTER (Windows) or RETURN (Macintosh), the entry either changes or the construction redraws according to the new specifications in the Status Line. Use the mouse to activate a Status Line field.

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.

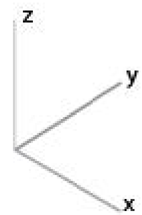
Coordinate System Axis



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

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

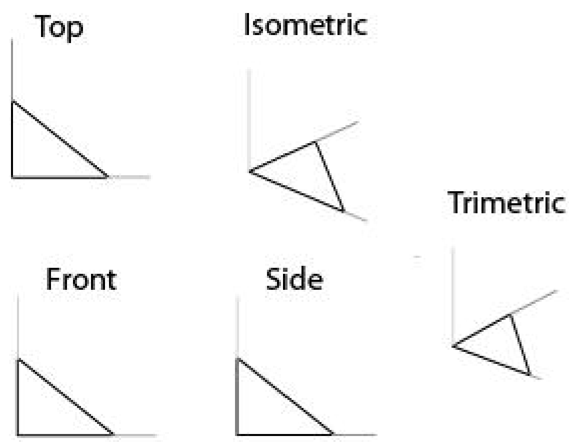
Axis



The Axis displays the current view orientation of the x, y and z axis in the center of the screen. When views are changed, the axis adjust accordingly. Turn off the display by choosing ***View>Show Axis***. If it is not necessary the Axis to display at start up, choose ***File>Preferences>General*** and deselect the Axis option.

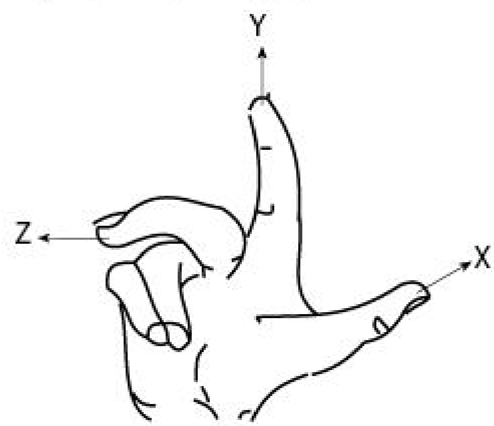
Triad

The Triad displays in the upper-left corner of the 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 the 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 the hand to indicate the x and y-axes, you can easily see the direction of the z-axis.

To turn off the Triad choose ***Planes>Show Triad***. If it is not necessary for the Triad to display at start up, choose ***File>Preferences>General*** and deselect the Triad option.



The Right Hand Rule

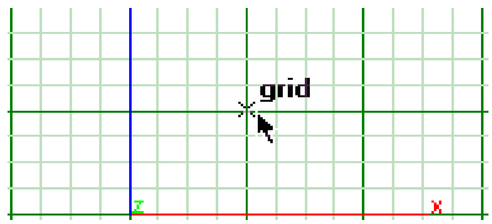
Drawing Area

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 is used to construct full-size unscaled drawings. Use the scroll bars to move the sheet so the portion to work on is visible in the window.

Displaying the Grid

If it is necessary to work with a grid in the drawing area, choose ***Planes>Show Grid***.

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



Scroll Bars

The scroll bars move the sheet up and down or right and left. 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.

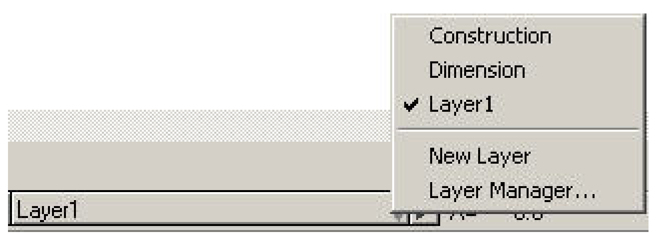
Click the arrows at the end of the scroll bars to move the sheet one line at a time.

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. To make the construction go on a specific layer, first make it the current layer.

Select the work layer in either of 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 a different layer can be selected to be the current work layer. Drag to the new work layer. All new geometry is placed on that layer. It is also possible to create a new layer and display the Layer Manager from the Work Layer Indicator.



Menu Bar

The program menus contain related commands and settings.

File	Contains commands that effect entire documents, including opening and closing files, setting preferences and defining the Print Setup (Windows) or Page Setup (Macintosh).
Edit	Use this menu for commands to select and manipulate objects. These include copying and pasting as well as changing an object's direction, resolution, layer and type.

File	Contains commands that effect entire documents, including opening and closing files, setting preferences and defining the Print Setup (Windows) or Page Setup (Macintosh).
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	Displays the document such as choosing and setting views and zoom options.
Planes	Contains commands for choosing and defining the work plane.
Pen	Specifies pen characteristics (color, weight, and pattern), polygon patterns, polygon fill, crosshatching and arrows.
Text	Sets the font, size, style and case of the text (lower, upper and title capitals).
Dimension	Use this menu to specify dimensions, their format and tolerance.
Verify	Obtains information about the file and specific objects in the file. These include properties of an object, direction, curvature and object counts.
Window	Displays tool palettes, the Design Explorer, Trackball, etc.
PhotoRender	Contains commands for photo-realistically rendering the geometry, setting options for rendering and placing lights.
Animation	Use this menu for generating Quicktime movies.
Help	Offers PDFs of the manuals links to the Ashlar-Vellum Web site for updates and support.

Mouse versus Keyboard

Cobalt, Xenon and Argon menu items can be chosen with the mouse or with a combination of keys on the keyboard. For example, here are 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.

There are also various methods for choosing commands with the keyboard. For example, 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 ⌘ (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.

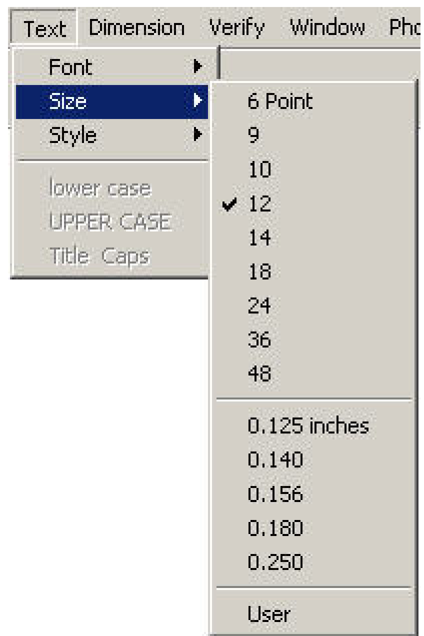
> ***Tip:*** *Windows only: Hold down the ALT key and type a letter to select the first item that begins with that letter, then use the arrow keys to move to the desired selection. Once the choice is highlighted, press the ENTER key.*

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.

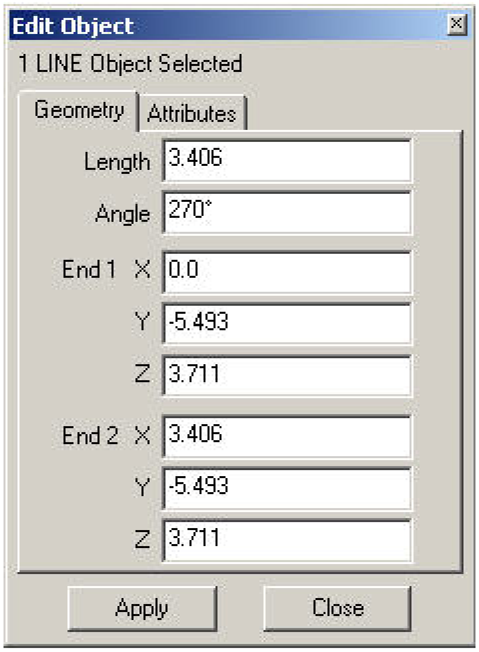


Dialog Boxes

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

Dialog boxes qualify the chosen command by adding information. For example, in the Edit Objects dialog box here, it is possible to change the specifications of the selected object.

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

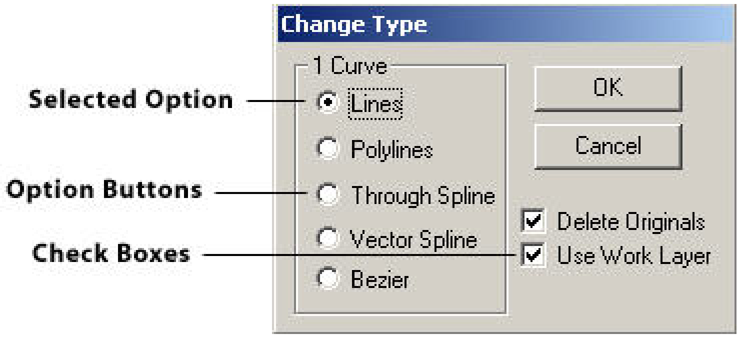


Tech Note: *Windows only: Designer Elements program dialog boxes do not support using the large font setting in the Setting page of the Display Properties dialog box for the Operating System.*

Tech Note: *To copy and paste data between data fields in dialog boxes, the Copy and Paste commands cannot be used. 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. Only one option can be selected at a time. Click the option and the button turns black, as shown by the inches option below.



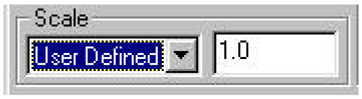
Check Boxes

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

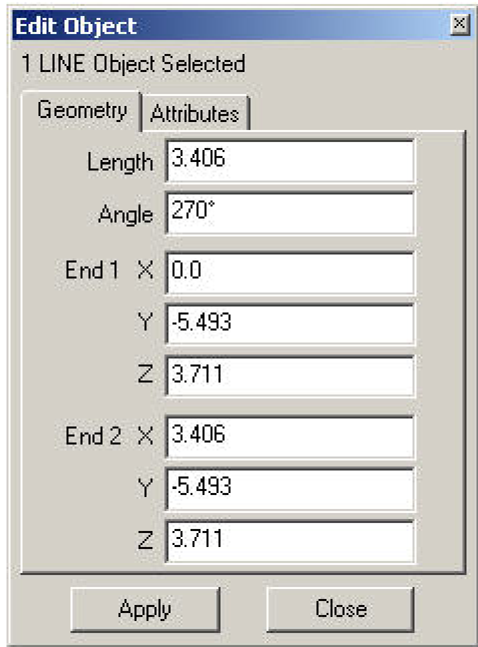
List/Entry Fields

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

Some have fields that allow an entry to be typed. For example, 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.

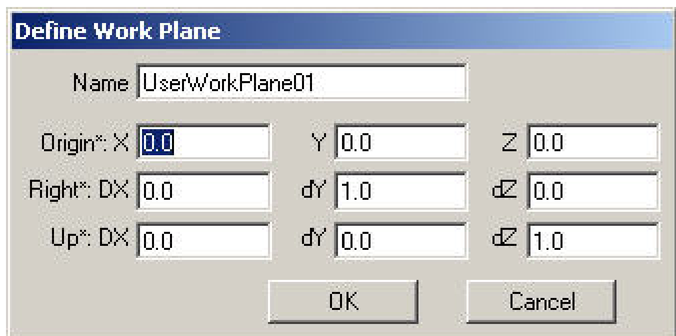


In most cases clicking OK saves the changes.



Asterisks

When an item in the dialog box displays an asterisk (*), it is possible to specify a value by clicking or dragging in the drawing area with the mouse. This feature is particularly useful for specifying location because it is not necessary to know any x, y and z coordinates.



Apply Buttons

Some dialog boxes have an Apply button that applies the specification currently set. The dialog box can be left open to set other specifications.

For example, once an object is changed in the Edit Objects dialog box, 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 the button is clicked. Otherwise, dismiss the dialog box manually by double-clicking the Control Menu (Windows) or clicking the Close Button (Macintosh) in the upper-left corner of the box.

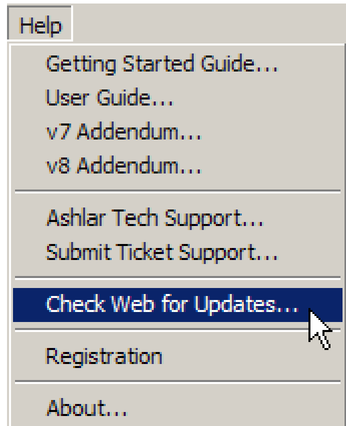
Toggling Commands

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.

Update Installer

Cobalt, Xenon and Argon feature an installation program to quickly check the website for newly available updates. Using this feature under ***Help>Check Web for Updates…*** downloads and installs just the update, not the entire program.



This method keeps in place:

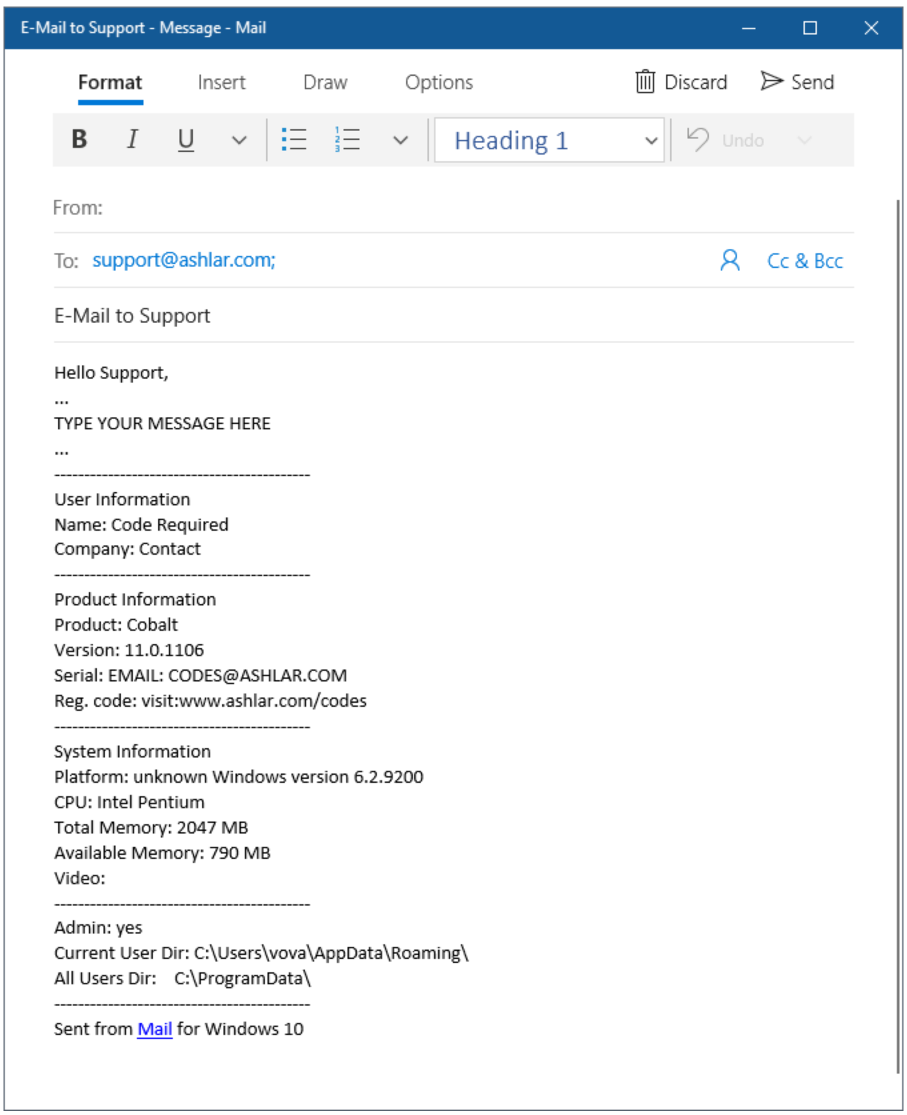
- Designated Preference settings.
- Current registration codes.
- User serial numbers.

It is recommended that this feature be used at least once a month with current software to be sure that the latest build of the major version number is being used.

Automated Support Ticket Preparation

To receive the assistance from our technical support experts, submit a ticket via email directly from your system. The email will automatically capture your particular product and system information, but you MUST include the following within the body of the email:

1. Tell us exactly what the difficulty is.
2. Attach any files or screen captures to help us understand the issue.
3. Include ALL contact information including an alternate email address, your cell phone and desk phone. Our support team does not have regular access to your sales file.
4. Include a subject line descriptive of the problem.
5. Make sure that you use your actual mail client.



The Drafting Assistant

The Drafting Assistant is the feature that makes all Ashlar-Vellum Designer Elements programs unique among design and drafting software products. The Drafting Assistant thinks like a drafter. It automatically knows where construction lines are typically wanted and displays them temporarily when they are necessary.

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 clicked, 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.

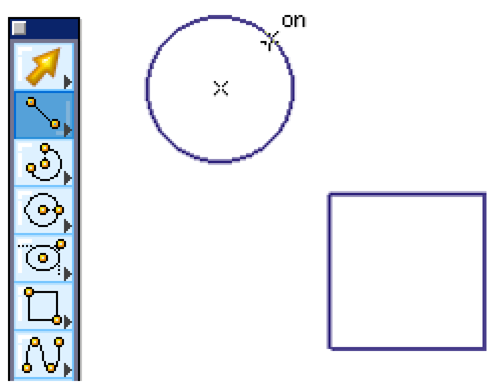
The following topics are covered in this chapter:

- Snapping onto Geometry
- The Drafting Assistant page of Preferences includes the following options:
- Permanent Stroke Construction Lines
- Drafting Assistant and Display

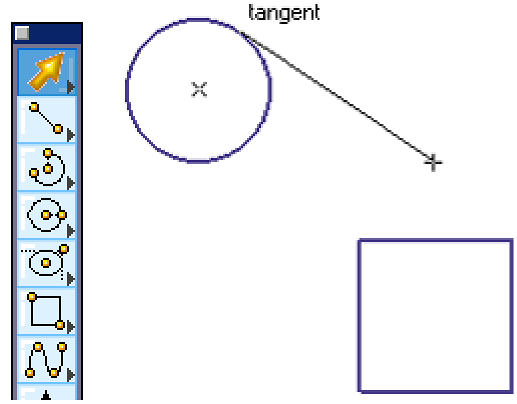
Snapping onto Geometry

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 the pointer is moved near them.

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



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 indicates when a snap point locks onto the following object points.

Center

The center of an arc or circle is indicated here. 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.

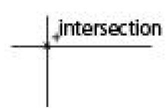
Endpoint

This shows the endpoint of lines, arcs, circles, ellipses and splines.

Midpoint

The midpoint of lines, arcs, circles, ellipses and splines is indicated this way.

Intersection

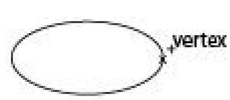


This shows the intersection of two curves including geometry, construction lines and Drafting Assistant alignments.

Quadrant

These designate the quadrant points on an arc or circle displayed at 3 o’clock, 6 o’clock, 9 o’clock and 12 o’clock.

Vertex

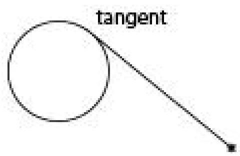


The vertices of an ellipse, spline, or dimension point are shown this way.

Using Tangents and Perpendiculars

Clicking a point *on* an arc or circle and dragging the pointer away at about a 45° angle, the Drafting Assistant locks onto the *tangent*. Dragging away at a 90° angle the Drafting Assistant locks onto a *perpendicular*.

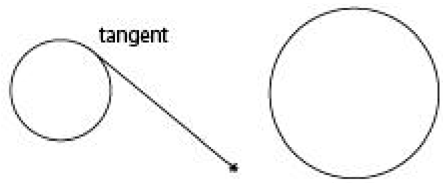
Tangent



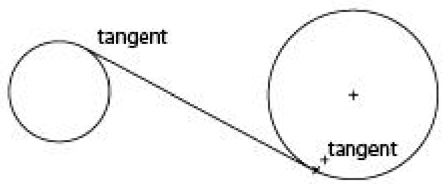
Perpendicular

Holding down the mouse button, the line remains *tangent* or *perpendicular* while the ending point is dragged around the object.

This is a useful feature if, for example, it is necessary 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, it can be dragged 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. It is not possible to begin from a specific point, such as *endpoint*, *quadrant* or *vertex*.

Drafting Assistant Axis Locking

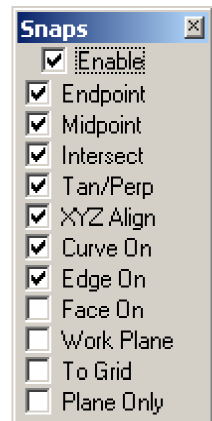
The Drafting Assistant supports locking to the X, Y or Z axis. To lock to one of these axis hold down the corresponding key on the keyboard (X, Y or Z). The Drafting Assistant will only show alignment points in that axis.

Customizing the Drafting Assistant

When first initializing the program, the Drafting Assistant automatically activates. Customize the Drafting Assistant by using the *Snaps* command or changing the preferences in the *Snap* page of the Preferences dialog box.

Snaps

This option box, found in the Window menu, displays the Snaps dialog box and activates and deactivates the different snap modes of the Drafting Assistant.



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

Snaps Enable — This enables the Drafting Assistant. Snaps that are checked display. Deselecting this option turns the Drafting Assistant off.

Endpoints — Displays object endpoints.

Midpoints — Displays object midpoints.

Curve Intersections — Displays intersections between objects display.

Curve On — Displays the point nearest to the pointer with an *on* notation.

XYZ Align — Displays alignments along the x, y and z axes.

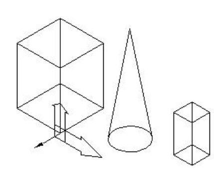
Tangents/ Perpendiculars — Displays tangents and perpendiculars for the geometry.

Face On — Displays a face notation for solid objects.

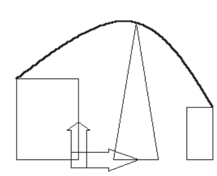
Edge On — Displays all notations normally associated with curves, such as endpoint, midpoint, and vertex for solid objects.

Work Plane — Creates a curve using referencing points on 3D objects but confined to the current 2D work plane. This snap works best when the current work plane is being viewed. See the example here for clarification.

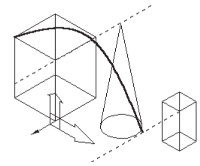
Example: To reference points from three objects, display the Work Plane icon for the Front plane.



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 to see the curve aligned with the current work plane. The dotted lines show that the curve aligns with the object points.



To Grid Snaps to grid alignment points when using the grid.

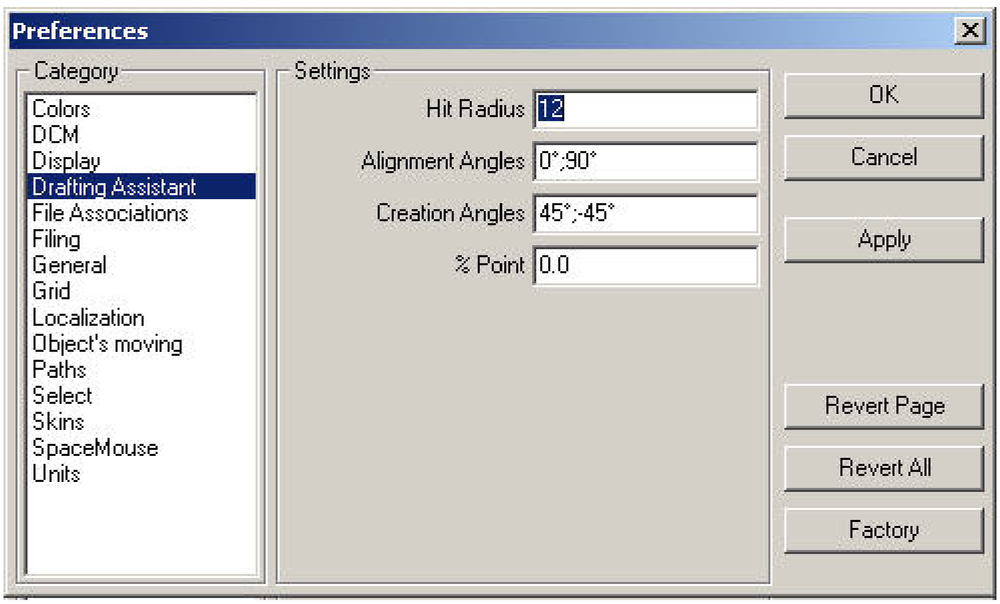
Plane Only Shows snap points for objects **only** on the current work plane.

Each of these snap options have shortcut equivalents. See Shortcuts for more information on using Shortcuts.

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

Choosing ***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 the mouse is clicked. The default Hit Radius is 12 pixels.

If it is complicated to specify locations that are close together because the Drafting Assistant snaps to an existing control point, do any of the following:

•Decrease the Hit Radius.

•Zoom in so more pixels separate the existing point and the point to select.

•Lock on a point by pressing the mouse button and typing the letter that represents the point (*m* for midpoint, for example).

Setting the Hit Radius to zero disables the single click selection of the **Selection** tool. (Dragging a selection fence and double-clicking to select all objects will still work.) Instead, 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. To change the orientation of the drawing, change these specifications. For example, 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.

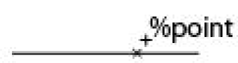
To display a dynamic construction line through a point, move the pointer to the point to activate it (a diamond appears). Construction lines automatically display through the active point. It is possible to have as many as eight active points. Activating the ninth point, the first one in the series deactivates.

Creation Angles

These lines are used by the Drafting Assistant only when 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

This shows the divisions of a line for Drafting Assistant notations. The default is 0.0.



For example, to divide a line into quarters, enter 25. This instructs the Drafting Assistant to show 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 used most frequently—vertical, horizontal and 45° angle lines—display automatically during construction. These construction lines appear temporarily to help align geometry. Once a point is set, the Drafting Assistant construction line disappears so that the drawing is not cluttered with extraneous lines.

Dynamic construction lines extend automatically from the last point created. 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, there is also the option to create *permanent construction lines* and other shapes, as described later in this chapter.

Using the Drafting Assistant's Construction Lines

After indicating the first endpoint of a line, move the pointer horizontally, vertically, or in a 45° direction, and 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

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 while clicking. The dynamic construction line disappears.

> ***Tech Note:*** *It is possible to have as many as eight active points; activating the ninth point in a series deactivates the first point.*

Displaying Dynamic Construction Lines with Existing Geometry

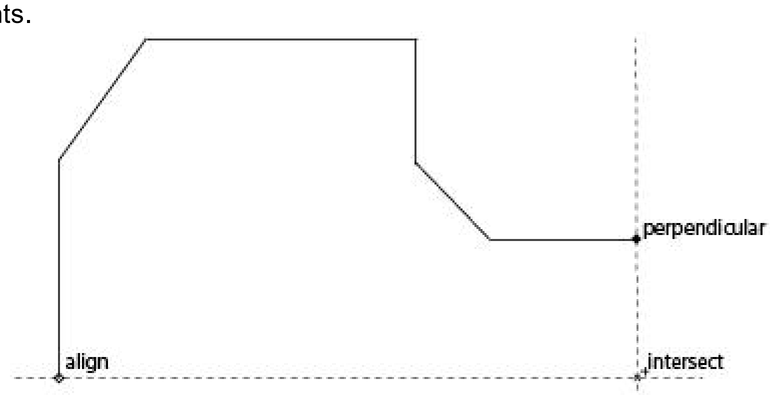
1. On selected geometry, without pressing the mouse button, move the pointer over an existing point.

   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 while clicking. The dynamic construction line disappears.

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



Once the Drafting Assistant become familiar it streamlines design and drafting tasks.

Setting New Drafting Assistant Construction Angles

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 the 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, it is possible to create construction lines that display until they are hidden or removed.

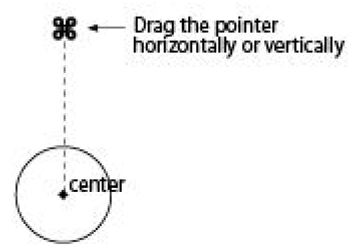
Stroke construction lines are lines created with the mouse. Hold down the SHIFT+CTRL keys (Windows) or the ⌘ key (Macintosh) and drag the mouse horizontally or vertically. Working in the drawing area, the Drafting Assistant helps to place the stroke precisely.

Construction lines automatically appear on the Construction layer, not the work layer of the drawing. Having inadvertently deleted the Construction layer, it recreates the next time a construction line is created.

Hide the construction layer to view or print the drawing without construction lines. Construction lines can be selected 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 Layers).

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 ⌘ key (Macintosh) changes the mouse pointer to the Stroke pointer (⌘).



Tip: *Use strokes to create construction lines 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.*

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, using the **Connected Lines** tool, create a construction line that extends through the center of a circle:

1. Hold down the SHIFT+CTRL keys (Windows) or the ⌘ key (Macintosh). The pointer becomes the stroke pointer (⌘).
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*.

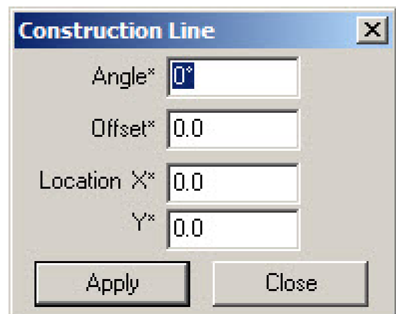
   The construction line appears through the center in the process of creating connected lines after releasing the SHIFT+CTRL keys (Windows) or the ⌘ key (Macintosh).

The Construction Command

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

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

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



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.

Only one set of values can be entered for a new construction line at a time. Multiple angles or offsets cannot be entered to create multiple construction lines.

> ***Tip:*** *Multiple angles or offsets cannot be entered to create multiple construction lines.*

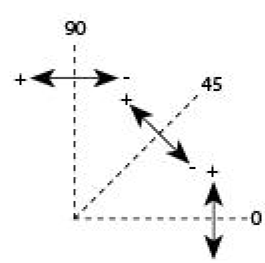
Specifying the Construction Line Angle with the Mouse

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 dragged distance 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 specified point appear in the *X* and *Y* fields, but they can be changed 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

It is possible to create parallel construction lines by creating one construction line using this field, then creating new lines with the **Parallel Line** tool.

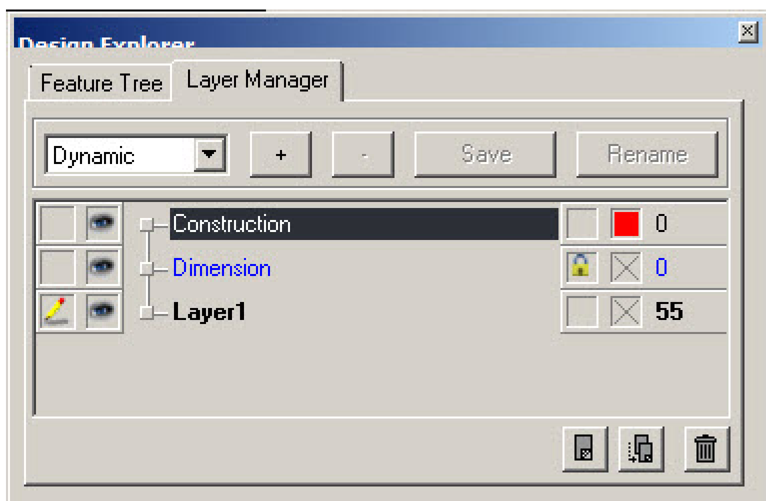
Creating Construction Geometry

Non-construction geometry is placed on the work layer. It is possible to 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 the work.

> ***Tech Note:*** *When creating construction geometry on the construction layer, change layers before creating the actual geometry. Choose **Layout>Delete Constructions** to delete everything on the construction layer and not just construction lines. Choose Undo to restore the geometry and move it to another layer.*

Creating Construction Geometry

1. In the Layout menu, choose *Layer Manager*. (The dialog box will say Design Explorer because 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 the Close button in the upper right corner of the window. The Layer Manager closes and the Construction layer is now the work layer. The *Work Layer Indicator* at the bottom left of the screen shows the current layer.
4. Create the geometry 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 the work.

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

Removing Construction Lines

To delete one or two of the construction lines, select the lines to remove, and then choose the BACKSPACE (Windows) or DELETE (Macintosh) key. All the construction lines, created by choosing ***Layout>Delete Constructions,*** can be removed.

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.

To retrieve deleted construction geometry use the *Undo* command.

Drafting Assistant and Display

The Drafting Assistant accurately creates and places objects. When creating an object that shares a common control point with another object, both control points are in the same X, Y, Z location.

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

Selecting Objects

Once objects are constructed, it is possible to change them. To make changes, select the desired object. In Cobalt, Xenon and Argon, the step at which the object is selected, depends on the operation. 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, 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
- Deep Select Tool
- Eye Dropper Tool

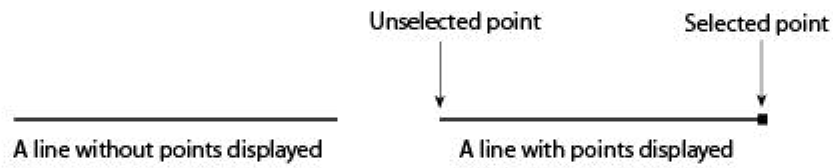
Objects

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 any member in that group is clicked.

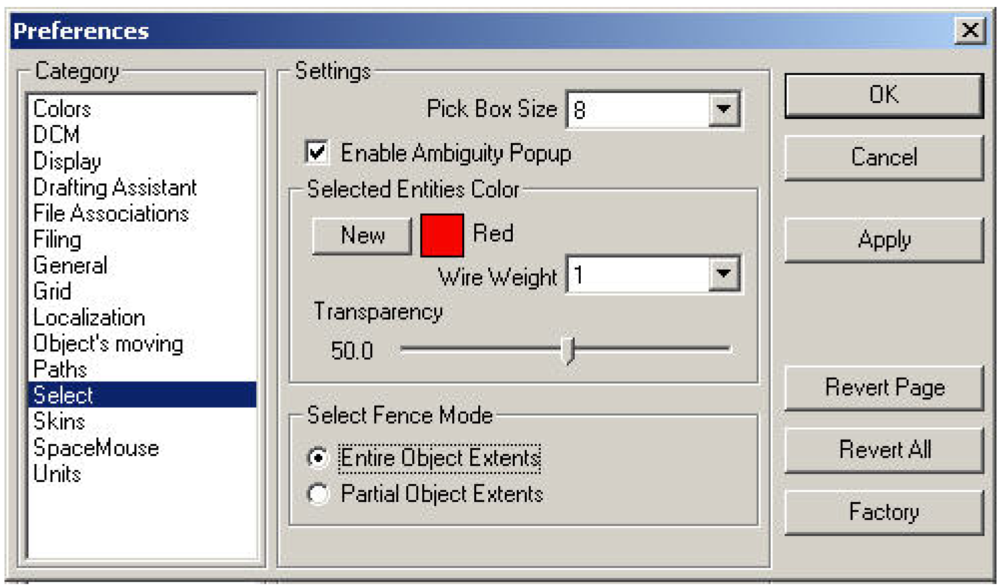
Referral: *Selecting points is described later in this chapter.*

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. By selecting an object and choosing ***Edit>Show Points***, the points are visible on the selected object. When selecting a point without selecting the geometry it defines, Cobalt, Xenon or Argon treats the point as an object.



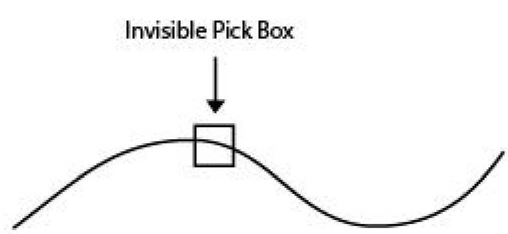
Indicating Selection & Preferences

When selecting an object, its appearance shows that it is selected in a specific color. The default color chosen is red. It is possible to designate another color as well as choose other settings for selecting objects by choosing ***File>Preferences>Select***. The following selection page is displayed.



Pick Box

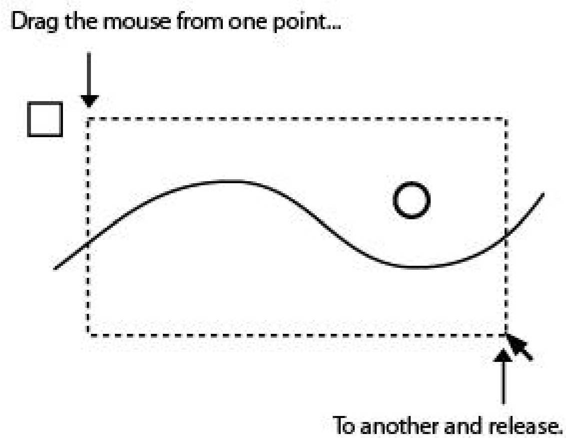
The pick box is an invisible box centered about the pointer tip. The default size of the box is 8 x 8 pixels. The pick box requires the pointer to be moved over the desired object only within the target area. When the mouse button is clicked on objects that are within the bounds of the pick box are selected. See the graphic below.



Select Fence

The selection fence is a rubber banding selection window. 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 even partially within the selection fence window is selected. In the graphic above, both the circle and the spline are selected.

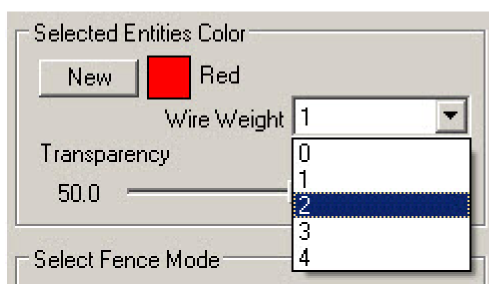
Selection Color

It is possible to choose any color for the selection color. In the *Selected Entities Color* section click New and the standard color display appears. Choose the color, click OK and return 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.

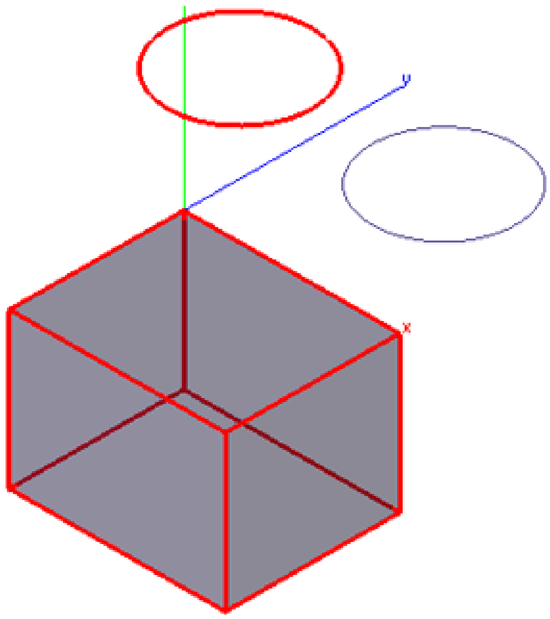
Wire Weight

When selecting a curve the line weight of the curve will increase in accordance with the setting in the drop down box. The line weight is specified in pixels. Choose between 0 (does not increase the weight of the selected curve) and 4 (increases the weight of the selected object to four pixels).



Transparency

When selecting a surface or a solid object, the surface or solid becomes transparent. The Transparency slider controls how transparent the object becomes. A value of 0 means no transparency is added to the selected object. A value of 100 means the object will be fully transparent showing only the edges.



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. 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.

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

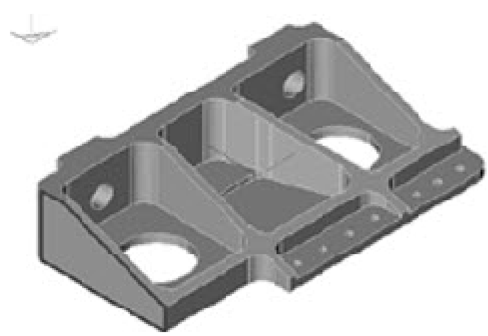
Selecting Objects with the Selection Tool

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

Selection Tool

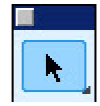
The **Selection** tool in the main tool palette selects one or more objects (curves, solid edges, faces, points, etc.) in the drawing area. The graphic here shows the selection of a solid face.

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.



__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.

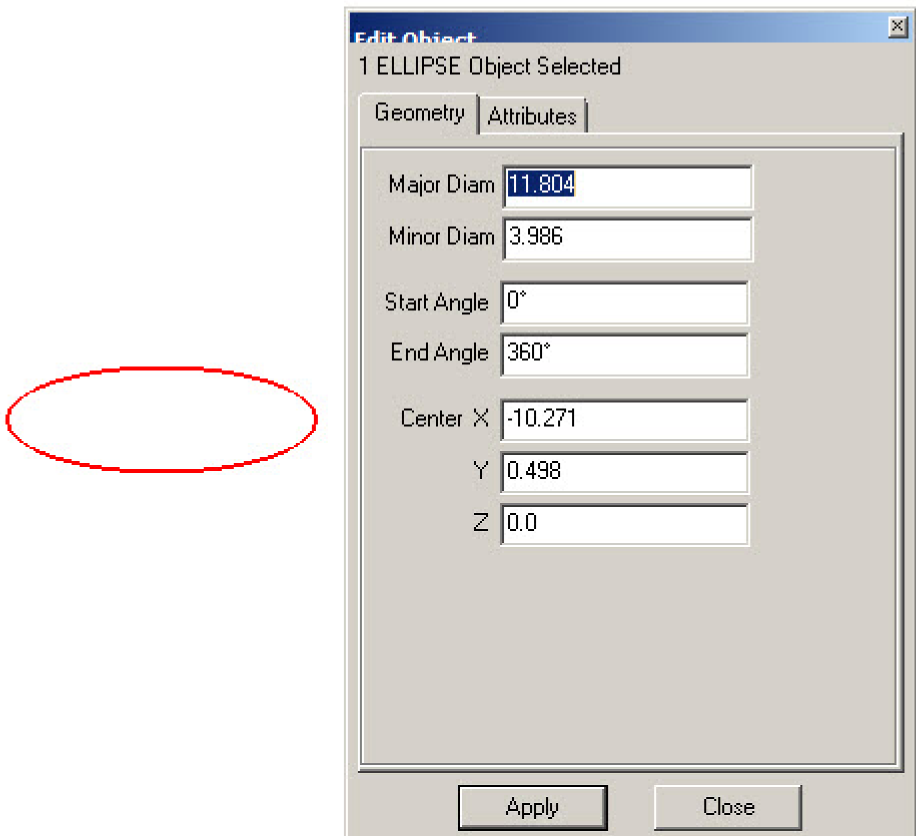
Selecting a Single Object



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 Planes for more information.

Double-selecting a Single Object



Double-click on an object to display that object's characteristics. This brings up the Edit Objects dialog box. Make changes to the object through this box. See "Edit Objects" on page 26. 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 clicked objects are selected. If one of the selected objects is clicked again, it deselects.

5. Release the SHIFT key.

Selecting Multiple Objects by Dragging

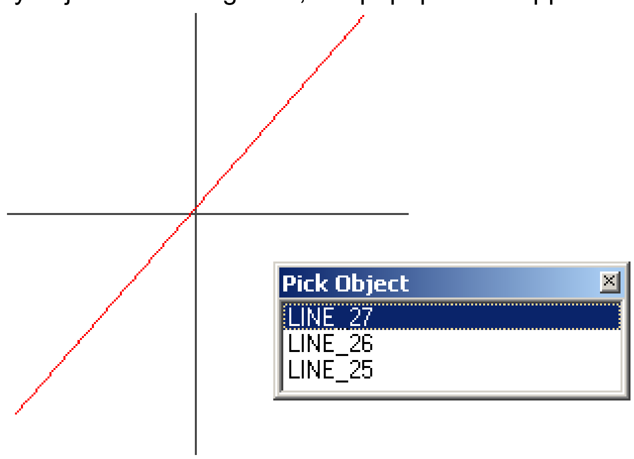
To select more than one object, drag a selection fence around the objects.

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

To select most of the objects within an area, drag a selection fence to select all the objects, and then deselect the objects that do not need to be selected by holding down the SHIFT key and clicking them.

Selecting One or More Objects When They Overlap (Ambiguity Popup Box)

Often 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 attempting to select one object among many objects close together, the popup menu appears.



By moving 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. Move the pointer beyond the extents of the popup window, and the window will automatically move, showing what is underneath the window.

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

There are 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

Tip: *Specify the selection of only certain objects, layers, or colors by setting a selection mask with the Select Mask command.*

Selecting All Objects

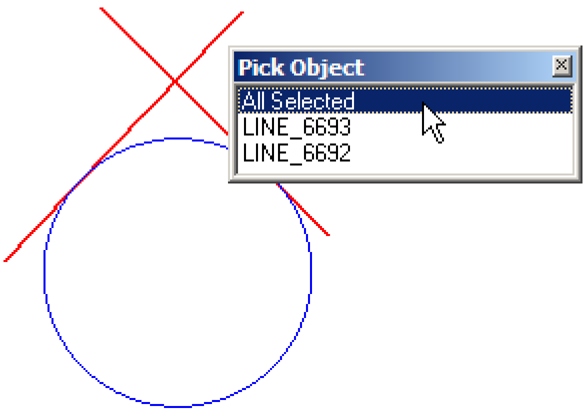
There are two ways to select everything in the document.

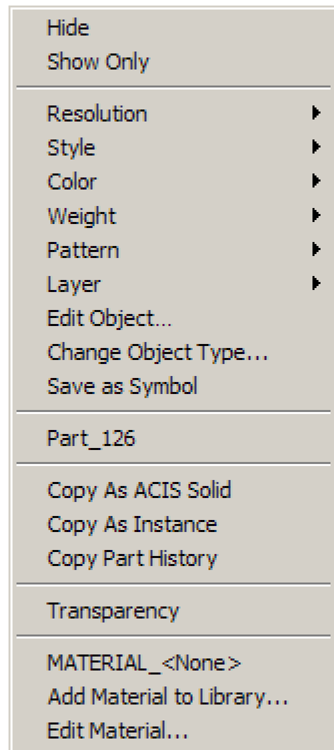
- Select the **Selection** tool and choose ***Edit>Select All*** (CTRL+A (Windows) or ⌘+A (Macintosh).
- Double-click the **Selection** tool.

> ***Tip:*** *Select All is useful to make a global change in a drawing, such as changing the width of all lines.*

Editing Common Properties on Multiple Objects

The common properties of multiple objects can be modified via the right-click menu. The ambiguity dialog box appears if more then one item is clicked. Choose the All Selected option from the Pick Object dialog and the properties modified in the right-click pop-up menu apply to all the selected objects.





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 an instance of the original object is created. All changes made to the original object are automatically reflected in the instance (Cobalt and Xenon only). To not create an instance, select the original object, choose ***Edit>Copy*** and then ***Edit>Paste***.

Selecting Points

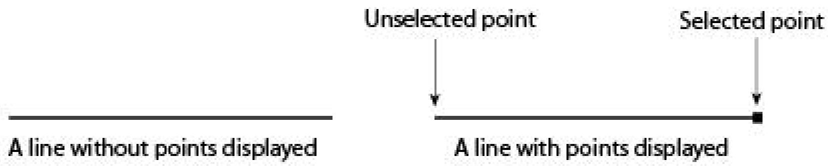
Selecting points differs from selecting objects because points are not always visible.

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, select a point by clicking it. If points are not displayed, select a point by dragging a selection fence around the location of the point.

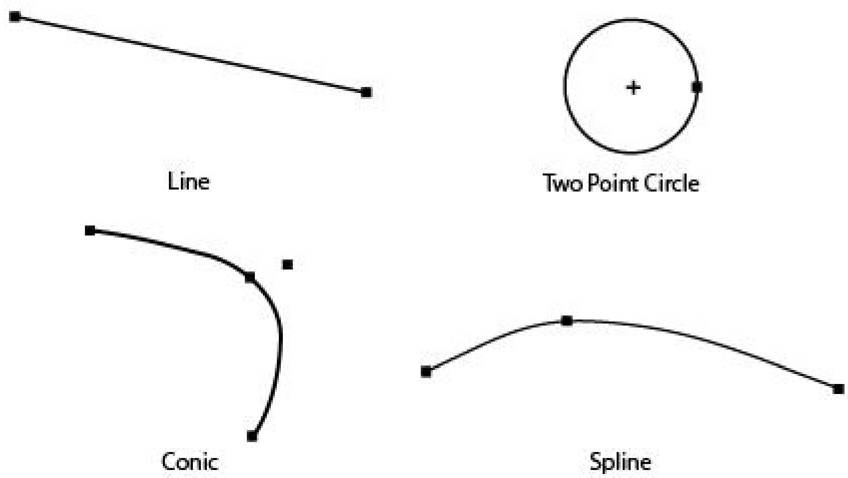
To show points for an individual selection, choose ***Window>Edit Objects*** and change the control points setting from Hidden to Visible.



Selecting and dragging a line, the line and the endpoints move. By selecting and dragging 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 the 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, 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 holding down the SHIFT+CTRL keys (Windows) or the ⌘ key (Macintosh) and clicking an object, the display of the object's points toggles on or off. If the points are hidden when clicking the object, the points display.

Selecting Points

It is possible to 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.

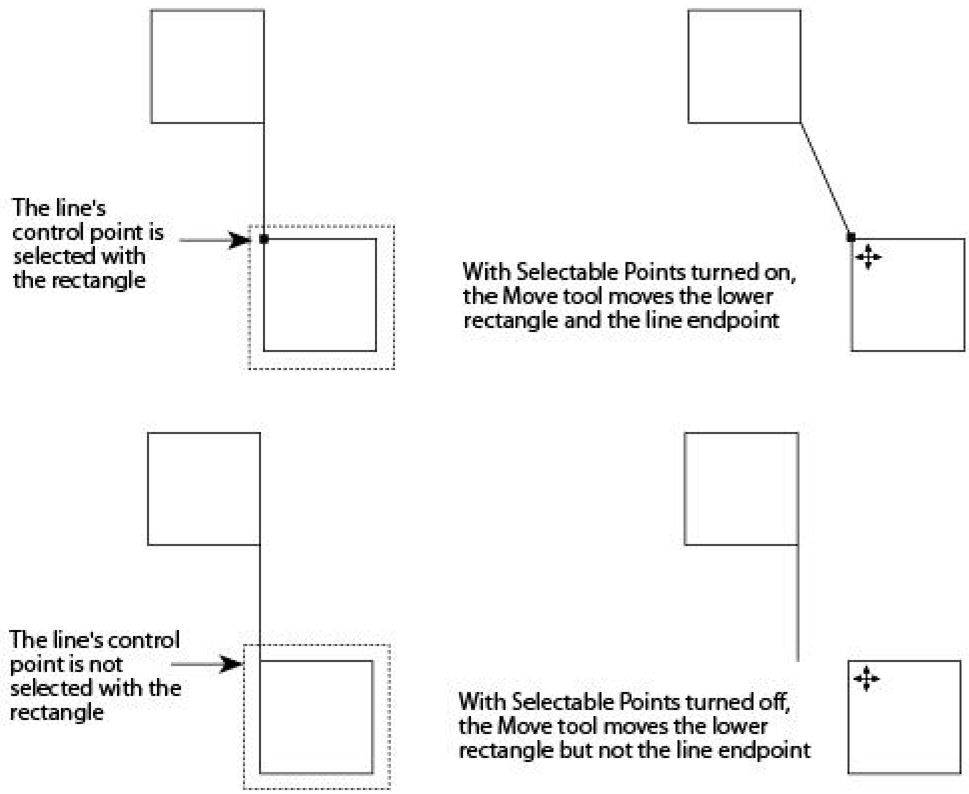
   The selected point displays as a square.

   When a point is selected, the entire curve highlights, however the edits will affect only the selected point.

Selectable Points Command

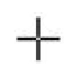
This command in the Edit menu selects points that aren't displayed. When *Selectable Points* is not set, points cannot be selected by dragging a fence.

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



Selecting Objects Using Other Tools

When using editing tools, such as Chamfer or Trim, it is necessary to select the objects on which to perform the operation. As was mentioned earlier in the chapter, select the tool first and then the objects.



Select an editing tool and move the pointer into the drawing area, it becomes a hollow selection arrow.

Having finished selecting, the pointer becomes a general tool cursor.

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

Selecting Objects with Commands

Cobalt, Xenon and Argon provide three selection commands, *Select All*, *Select Mask* and *Select Chain*.

Select All

CTRL+A (Windows); ⌘+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. Double-click the **Selection** tool to select all objects.

Selection Mask Command

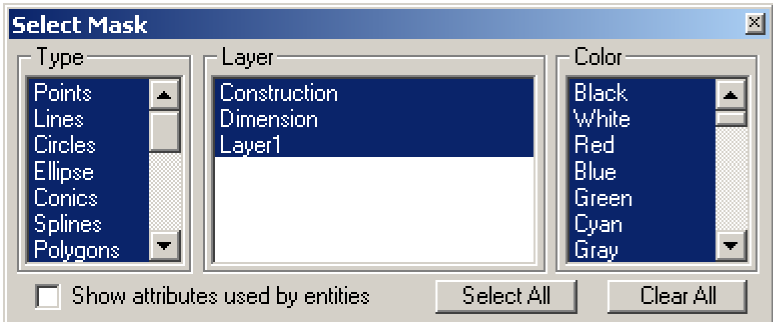
This command in the Window menu limits selection by object type, layer and color. Only objects that are highlighted in the dialog box are selected. The object type list includes such things as points, lines, splines, mesh, surfaces, solids, text, dimensions, images, symbols, groups, lights, decals, draw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

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

Using the Selection Mask

1. Choose ***Window>Selection 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 to select so they are highlighted.

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

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 on the screen.

The Selection Mask is useful in a complex drawing for changing particular groups of objects. 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.

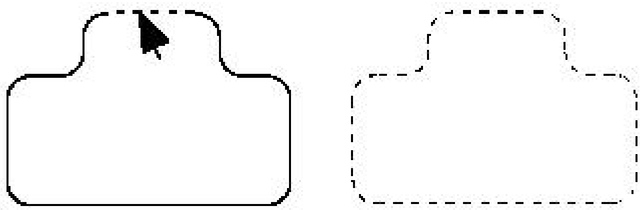
Selecting or Deselecting Listed Items

- 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 ⌘ 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.

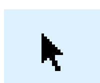


Deselecting

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.

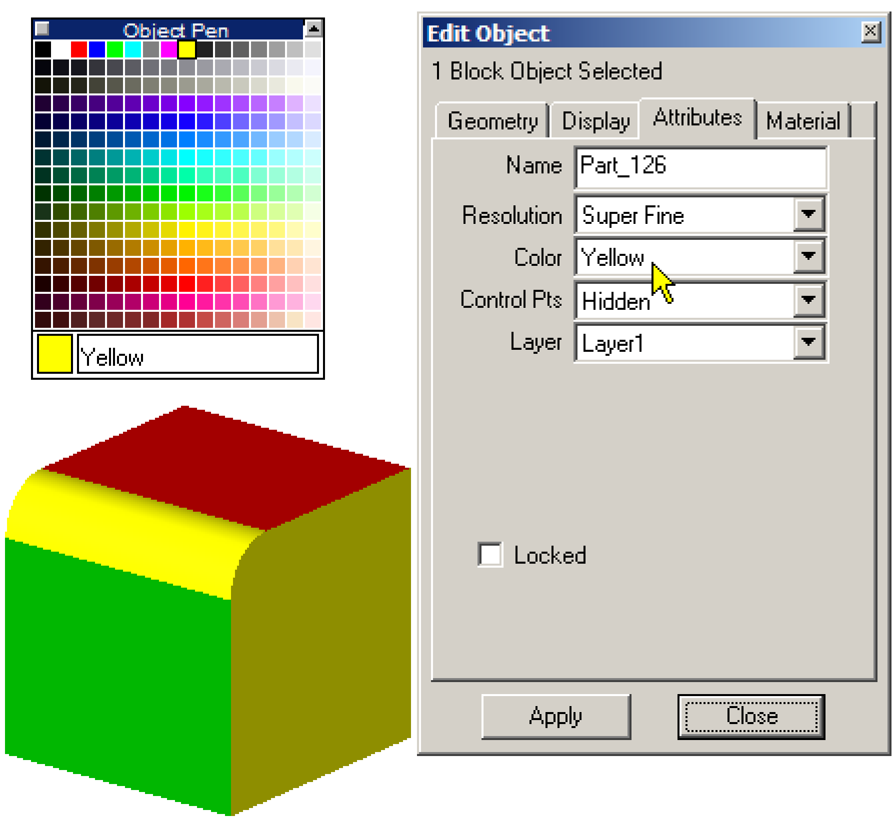
To deselect an object that was selected in a multiple selection operation, while the objects are still selected, hold down the SHIFT key and click the objects to deselect.

Deep Select Tool



This tool applies different colors and materials to each face of an object.

To apply a color to a face, use the **Deep Selec**t tool from the **Selection** tool palette, choose the face and designate the color from the Pen menu.



The Edit Object window shows only the color of the overall object without the changes made by the **Deep Select**.

Some materials adopt the color of the object faces if the Use Object Color check box is checked in the Render Material Settings dialog box.

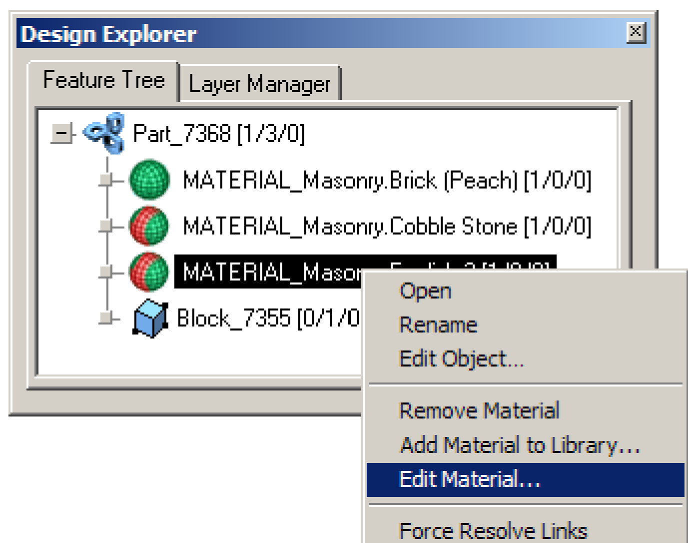
1 2 3

Picture 1: An object with different colors on the faces.

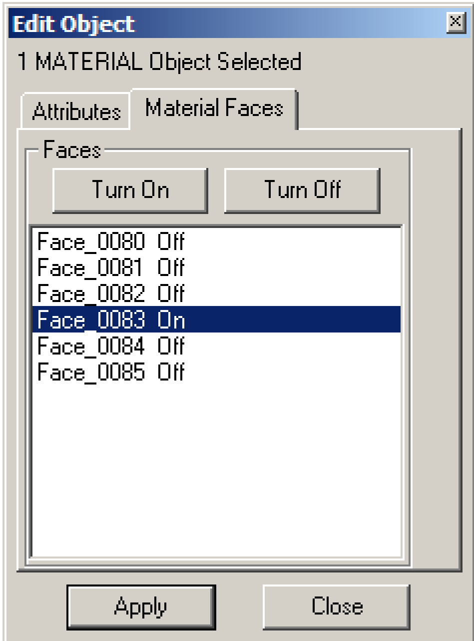
Pictures 2 and **3:** Materials adopt the color of the object faces with the Object Color option checked.

To apply a material to a face, use the **Deep Selec**t tool from the **Selection** tool palette, choose the face and designate the material from the ***Window>Render Library*** menu.

The material for each face appears in the Design Explorer as a ball  and can be edited.

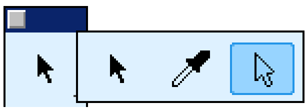


Materials per Face are also editable through the Edit Objects box. Turn on or off the material under the Material Faces tab.



Using the Deep Select Tool to Apply a Color

1. Select the **Deep Select** tool from the **Selection** tools palette. The Message Line reads: *Deep Select: Pick solid faces [Shift = Extend]*.

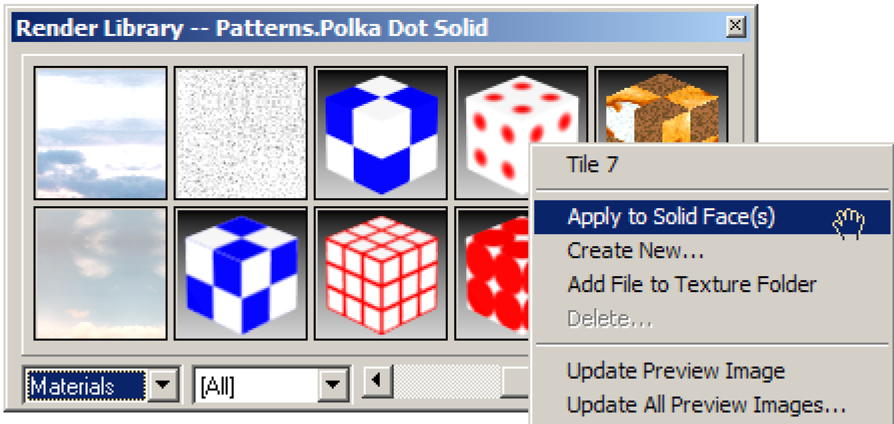


2. Select the face to be changed.
3. Go to ***Pen>Color*** and choose the color desired.

   The selected face changes the color.

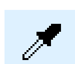
Using the Deep Select Tool to Apply a Material

1. Select **Deep Select** tool, from the **Selection** tools palette. The Message Line reads: *Deep Select: Pick solid faces [Shift = Extend].*
2. Choose the face and designate the material from the ***Window>Render Library*** menu.
3. Right-click on the material and choose Apply to Solid Face(s).



4. Alternatively, choose the material from the Render Library menu, right-click it and select Apply to Solid Face(s) from the context menu. Then choose the face of the solid. The material is applied.
5. Render the object to see the changes.

Eye Dropper Tool

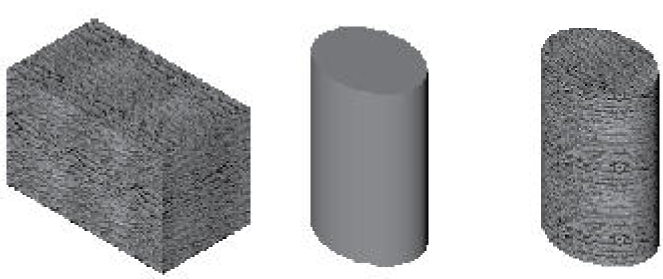


This tool copies object characteristics, such as line font, color, pattern, arrow at start and end, and render materials, from one object to another.

Specify which object characteristic to copy using the Eye Dropper Filter dialog box. To display this box, select the tool then press the CTRL (Windows) or OPTION (Macintosh) key.

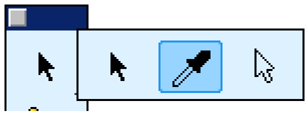
The listed characteristics include: line font, color, layer and render material.

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 to the cylinder without materials (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 copy from. [Ctrl = Filter (Windows) or Option = Filter (Macintosh)]*.



2. Select the object with the characteristics to be copied.

   The Message Line now reads, *Eye Dropper: Select object to modify. [SHIFT = Extend].*
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 step 5.
4. Deselect those items in the dialog box that do not need to be copied.
5. Select the objects to receive the characteristics.

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

It is also possible to select the object before selecting the tool. Doing so, select the reference entity and the object automatically changes. If the change involves applying a material, render the drawing to confirm the change.

There are no Status Line entries.

Editing Objects

This Designer Elements program provides a variety of editing commands to edit geometry quickly and easily. Some commands deal with the placement of the geometry like *Cut* and *Align*. Some commands change the appearance of the object like *Change Resolution*. Still others change the geometry like *Change Object Type* and *Edit Objects*.

The chapter includes these sections:

- Basic Editing Commands
- Advanced Editing Commands
- Object Commands

Basic Editing Commands

These commands provide some standard editing features like: *Cut*, *Paste*, *Undo* and *Redo*.

Cut

CTRL+X (Windows); ⌘+X (Macintosh)

This command in the Edit menu removes the selected object(s) and places them on the Clipboard. Each selection being cut or copied to the Clipboard replaces the previous Clipboard contents.

Cutting Objects

1. Select the object to cut.
2. Choose ***Edit>Cut***.

Once a selection is cut, it can be pasted. Use *Cut* and *Paste* to move geometry or text around the document or from one document to another. The cut selection can even be pasted onto a document in a different application.

Moving Geometry with the Cut Command

1. Select the object(s) to move.
2. Choose ***Edit>Cut***.
3. Display the location where the selection will 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 it can be moved, if necessary.

Copy

CTRL+C (Windows); ⌘+C (Macintosh)

This command in the Edit menu places a copy of the selection onto the Clipboard without deleting the original selection. The copy can be pasted elsewhere in the current document or into a different document. The copied selection can be even pasted 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 the parent/child relationship is broken.* **(Cobalt and Xenon only)** *Use the* **Move** *tool instead. For more information see Move Tool*

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

Use the *Copy Bitmap* command to specify a region to copy as a Raster image. The image can be then pasted back into the same document, to another document or to a different application.

Paste

CTRL+V (Windows); ⌘+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 the *Paste* command is used.

The selection can also be pasted into another document or into a document created with another application.

To create an even distribution of geometry, use one of the Duplicate tools which are located in the Transformation tools palette.

> ***Tech Note:*** *The Copy and Paste commands cannot be used 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

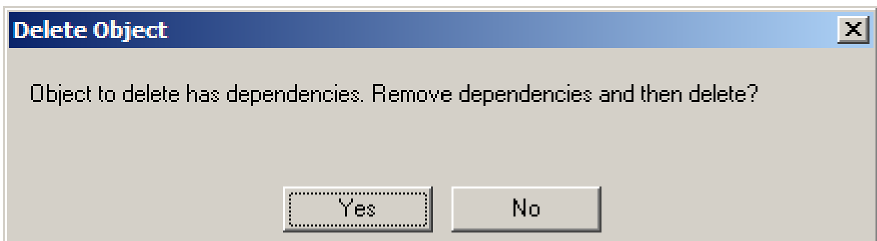
The *Cut* and *Copy* commands place a copy of the selected object on the Clipboard. The Clipboard is a buffer—a temporary storage place that holds the last cut or copied selection. The contents of the Clipboard are objects, not bitmaps (unless *Copy Bitmap* is chosen).

Erasing/Deleting Geometry

This program provides two methods for erasing objects:

- Select the object(s) then press the BACKSPACE (Windows) or DELETE (Macintosh) key. (It is possible to retrieve what is deleted with the *Undo* command.)
- Select the object(s), then choose ***Edit>Cut***. (It is possible to retrieve what is cut by using the *Paste* command, as long as nothing else has been cut or copied.)

Trying to delete an object that is the parent in a parent/child relationship, this warning message will appear (**Cobalt and Xenon only**):



When deleting objects in this program the data is still retained in the file until it is deleted again. If objects are deleted before closing a file, perform the operation again to ensure that the data is removed.

Undo

CTRL+Z (Windows); ⌘+Z (Macintosh)

This command in the Edit menu reverses the last action. An infinite number of times can be undone in this Designer Elements program. When choosing *Undo,* the last action taken is displayed next to the command.

It is possible to *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.

When 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. It is possible to redo an infinite number of times in this Designer Elements program. When choosing *Redo*, the last action taken is displayed next to the command.

It is possible to *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

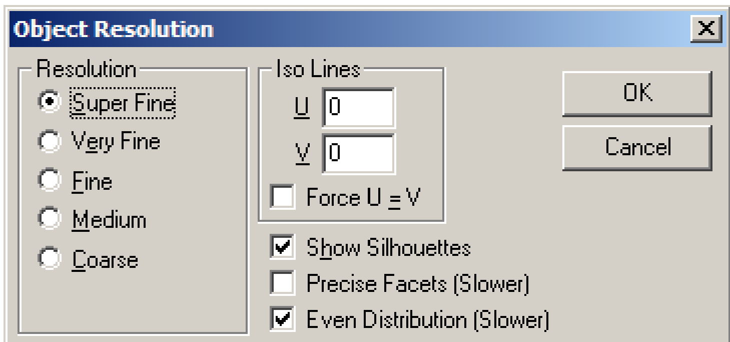
These commands provide more specific editing features/functions involving location, selection and display.

Change Resolution

This command, located in the Edit menu or in the right-click menu, changes the display resolution of curves, surfaces and solids.

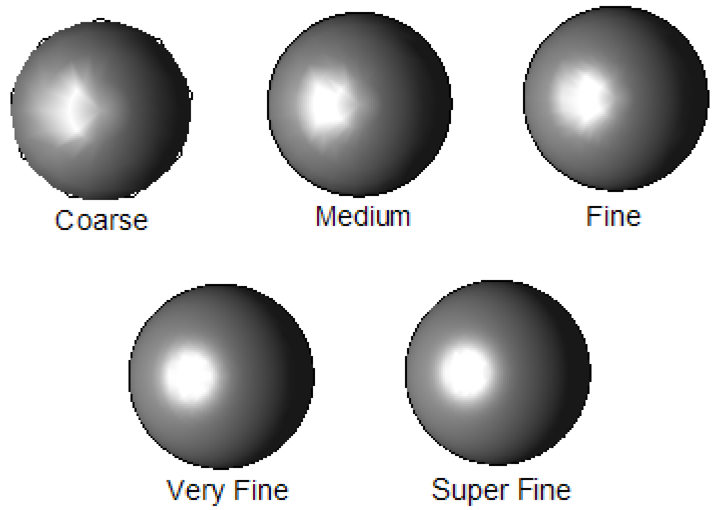
A higher resolution means that this program uses more line segments to display curves on the screen so they look smoother but take longer to draw.

For surfaces and solids a higher resolution also means that the program uses more facets to display the objects. It is possible to control the number of iso lines used to display surfaces with this command.



The dialog box contains the following options:

Resolution Contains radio buttons for five resolution types *Course*, *Medium*, *Fine*, *Very Fine* and *Super Fine.*



Iso Lines Contains the data fields for *U* and *V* isopram (iso) lines and the *Force U=V* check box.

The U and V data fields 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 iso line value.

Show Silhouettes Controls whether the 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.

Precise Facets Enlarges the number of the segments of a solid to make the surface look smoother.

Even Distribution Enlarges the number of the segments of a curve to make the surface look smoother.

OK Closes the dialog box and saves the new settings to make the surface look smoother.

Cancel Closes the dialog box without saving the changes.

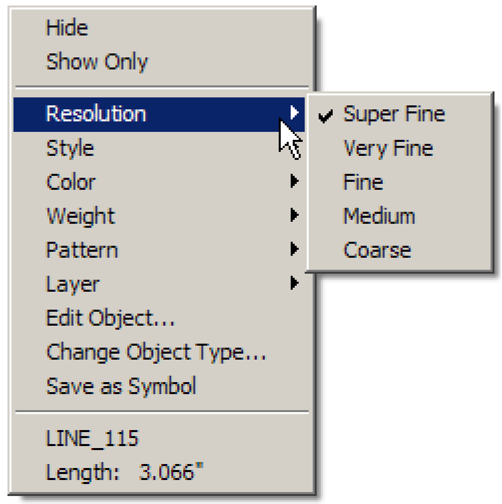
Using the Change Resolution Command

1. Select the object to change resolution.
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.

   or:

1. Select the object to change resolution.
2. Right-click the object. Choose Resolution option from the context menu.
3. Select the desired resolution.

   The object resolution changes.

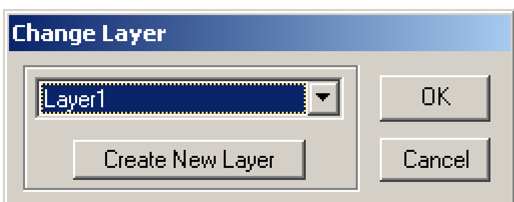


Change Layer

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 to move to a different layer.
2. Choose ***Edit>Change Layer***. The following Change Layer dialog box appears:

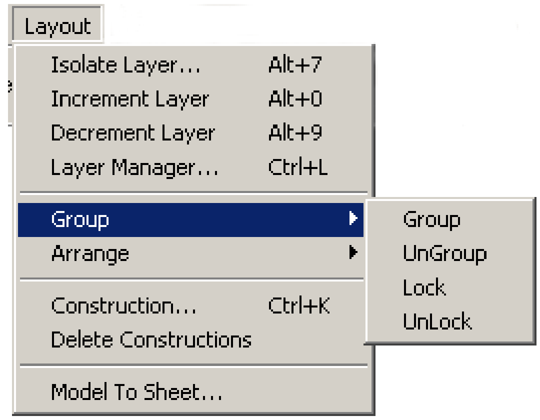


Click on the pull-down menu and select the layer to which to move the selected object. If the layer doesn't exist, click the Create New Layer button. A new layer appears in the layer field. Enter a new name if necessary.

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 shortcuts that this Designer Elements program treats as one unit. When moving a group, all members of the group move as a single unit. Deleting 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. 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. It is possible to create a group and then include it in a second group with other objects or groups.

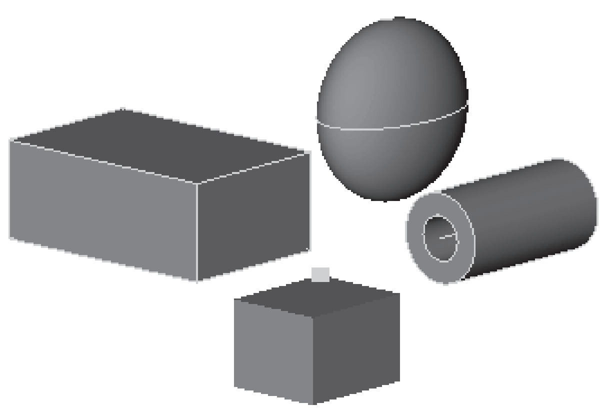
There are two options to add new objects to an existing group. By selecting the group and the new object, then choosing the *Group* command, a group within a group is created. To make all of the objects be in a single group, follow these directions:

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 to add to the group.
4. Choose ***Layout>Group>Group***.

Follow a similar procedure to remove members from a group.

Grouping Rules

- Grouped objects cannot be extruded, revolved or modified.
- If the color of a group is changed and then the objects are ungrouped, 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 using the command the control point is ignored. This is helpful when there are many objects located close to each other making it difficult to select the desired objects without selecting portions of others.



- When grouping objects on different layers, the group layer is the current work layer. If the objects are ungrouped, however, the objects return to their original layers.

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 layers.

Using the Ungroup Command

1. Select the group.
2. Choose ***Layout>Group>Ungroup***.

Lock Command

It is possible to lock any object or shortcut. Once it is locked it cannot be modified, selected, translated, rotated or the position or shape of a locked object cannot be changed in any way.

> ***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 lock.
2. Choose ***Layout>Group>Lock***.

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

Any Designer Elements program object or shortcut that previously has been locked can be unlocked.

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.

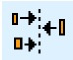
Arrange Command

The *Arrange* command in the Layout menu repositions 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*.

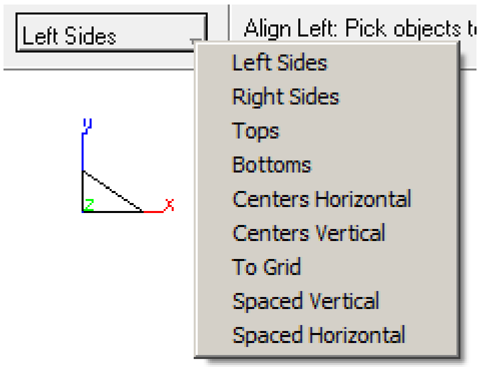
Move Forward	Moves the selected object up one position in the display.
Move to Front	Moves the selected object to the top (or in front) of all other objects in the display.
Move to Back	Moves the selected object to the back of the display.
Move Backward	Moves the selected object down one position in the display.

Align Command

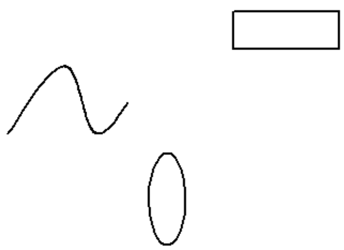


This command in the Transformation tool palette moves selected objects, including text, relative to other objects.

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.

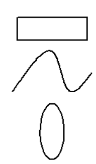
Right Sides Aligns the right sides of selected objects.

Tops Aligns the tops of selected objects.

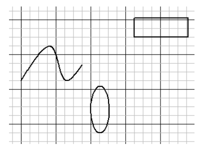
Bottoms Aligns the bottoms of selected objects.

Centers Horizontal Aligns the center points of selected objects to the same x location.

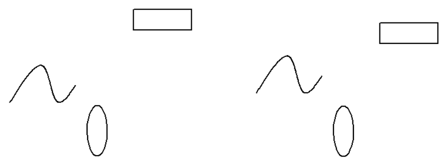
Centers Vertical Aligns the center points of selected objects to the same y location.



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. Choose the alignment tool from the Transformation tool palette. Select the alignment option from the message line. The Message Line reads: *Enter Alignment Point*.
3. Choose the point at which the objects will align.

Object Commands

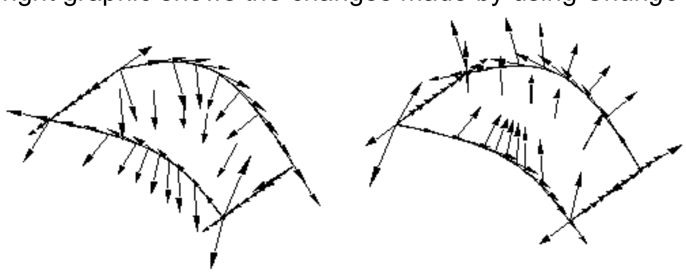
These commands modify various aspects of the geometry including direction, type and size.

Change Direction

This command in the Edit menu reverses the orientation of curves and surfaces.

Sometimes it is 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 to change its direction.
2. Click ***Edit>Change Direction***.

This Designer Elements program displays temporary arrows along the object to show its direction. Pick ***View>Redraw Screen*** to make them disappear.

Referral: *This command can impact the rendering of the object. See Render Option Combinations and View Rotation and Flip Normal Example for more information.*

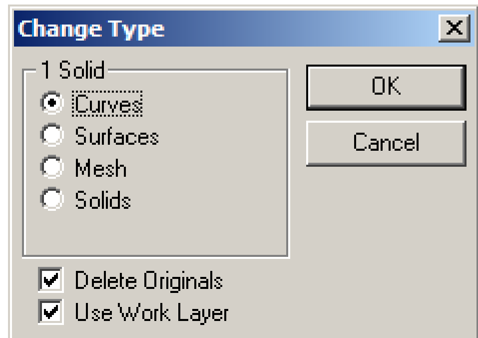
Change Object Type

This command in the Edit menu or in the right-click context menu changes 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 program remembers the selected conversion type (Curves, Surface, Mesh or Solids for a Solid object) until the option type is changed.

This command converts:

- Solids to solids, meshes, surfaces or curves.
- Surfaces to meshes, surfaces or curves.
- Curves to lines, polylines or splines.

When selecting this command a dialog box appears.

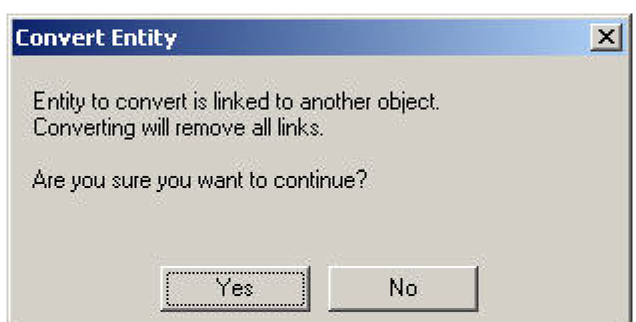


Tech Note: *Only a polysolid, also known as a disjoint solid or multi-lump solid, can be converted to its component, single-lump solids.*
Only a polysurface can be converted to component single surfaces.

The dialog box contains the following standard options:

Selected Object section	This section contains the conversion options. The options differ depending on whether a wireframe, surface or solid object is selected.
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	Convert the object.
Cancel	Cancels the command.

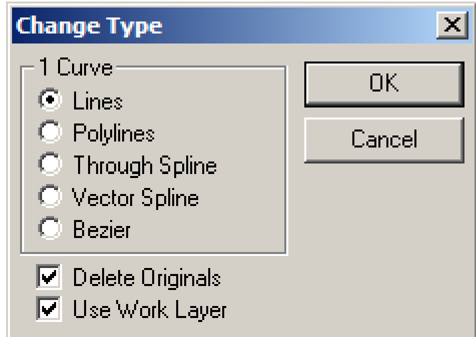
When converting a model containing links, before the conversion proceeds, the dialog box on the right displays for confirmation that the links will be removed by completing the conversion.



Changing Curves

When choosing the *Change Object Type* command after selecting a curve, the dialog box on the right box appears.

It is possible to convert the curve to Line Splines. To delete the original curve click the *Delete Originals* check box.



Curves to Lines Option

When the *Lines* option is selected a curve or set of curves can be converted to lines. 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 the program removes the parent curve after exploding it, leaving only the curves.

Curves to Polylines Option

When the *Polylines* option is selected, curves or a set of curves are converted 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

The *Through Spline* option converts a line, conic, arc, circle, or ellipse into a through spline. A through spline's shape, slope and control points can be edited. The resulting spline is fit to within a 0.001 drawing unit tolerance of the original curve. If the *Delete Originals* option is checked, the parent curve is removed 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

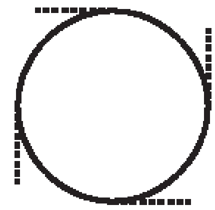
When the *Vector Spline* option is selected, a line, conic, arc, circle, or ellipse is converted into a vector spline. A vector spline's shape, slope and control points can be edited. The resulting spline is fit to within a 0.001 drawing unit tolerance of the original curve. If the *Delete Originals* option is checked, the parent curve is removed 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

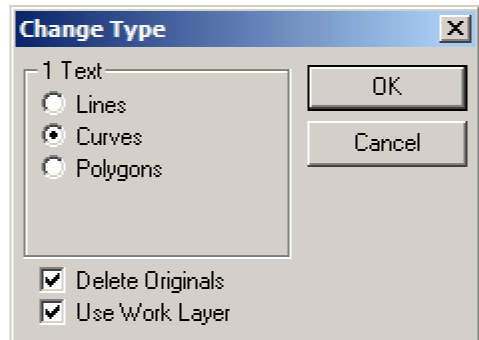
The *Bezier* option converts a line, conic, arc, circle or ellipse into a bezier spline. The bezier spline's shape can be edited by moving the control points and the slope of the spline at those points. If the *Delete Originals* option is checked, the parent curve is removed after exploding it, leaving only the splines.



Converting Text to Curves

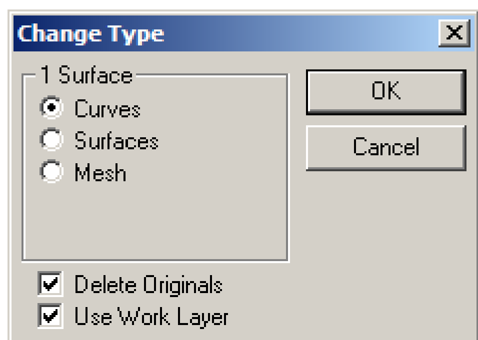
1. Create text using one of the Text tools.
2. Choose ***Edit>Change Object Type***.
3. In the Change Type dialog box choose curves.
4. Click OK.

   Each letter of the text converts into a closed grouped curve. Now any curve operation on the text curve can be performed.



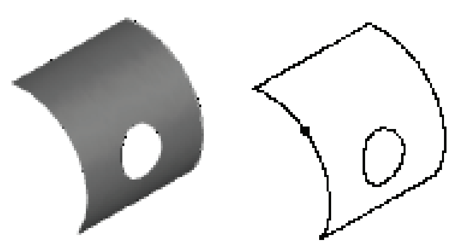
Changing Surfaces

When choosing the *Change Object Type* command after selecting a surface, this dialog box appears:



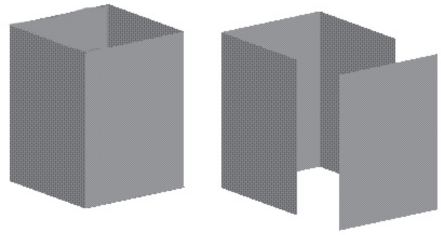
Surface To Curves Option

The *Curves* option converts a surface into a collection of editable vector splines. Modify the shape, slope and location of the spline control points as necessary. If the *Delete Originals* option is checked, the parent surface is deleted after exploding it, leaving only the curves. The resulting curves may consist of lines, arcs, circles, ellipses or splines.



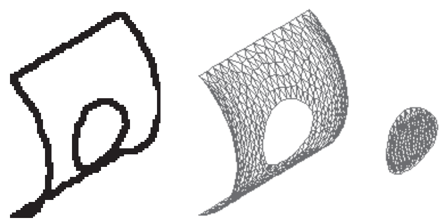
Surface To Surfaces Option

Selecting the *Surfaces* option explodes a surface composed of multiple faces into discrete surfaces. Once the surface is exploded individual surfaces can be edited, moved to a different layer, transform or removed.



Surface To Mesh Option

If the *Mesh* option is checked, the surface is converted to a triangular mesh. Meshes are useful for sharing program data with applications that cannot import precise surfaces. Exploding surfaces to mesh allows editing individual vertices.



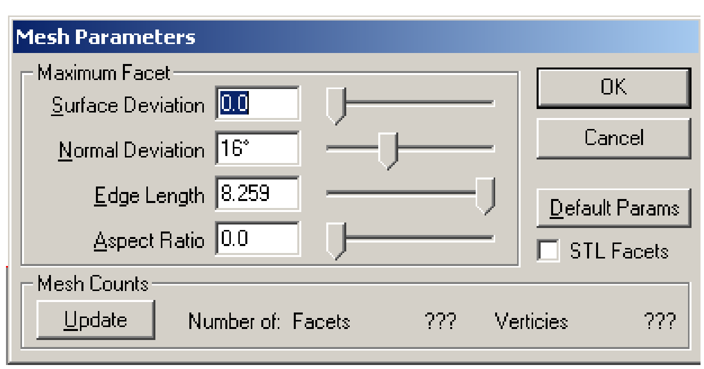
The precise mathematical representation of a solid or surface must often be converted into a collection of imprecise planar facets. These facets, for example, may be used to export a model to the STL format, and when changing a solid or surface (***Edit>Change Object Type***) to a mesh. The amount error 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 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 exploding 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.

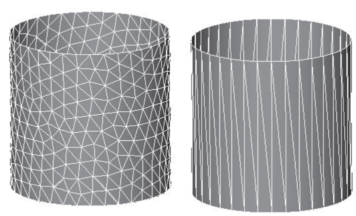
Surface Deviation Sets the maximum acceptable distance between the facet and the surface represented by the mesh. Enter a value in the data field or use the slide to set the value.

Normal Deviation Sets the maximum angular deviation between adjacent facets. The default is 20 degrees. 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 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 question marks until the Update button is clicked. The calculated values replace the question marks.

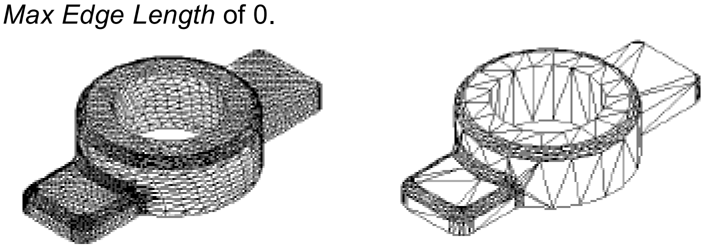
Smoothing Selecting this option creates facets that are regularly spaced. The left graphic here shows an object without smoothing. The right shows the object with mesh smoothing.



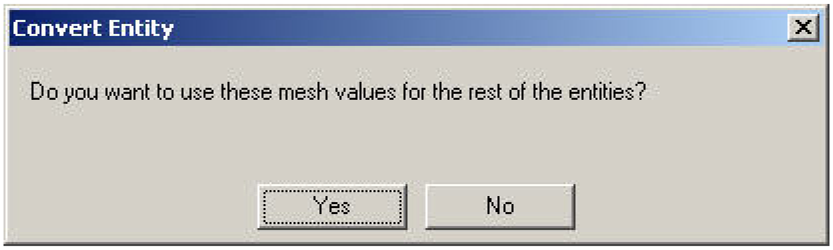
Default Params Reverts the mesh parameters to the program default values.

STL Facets Forces 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.



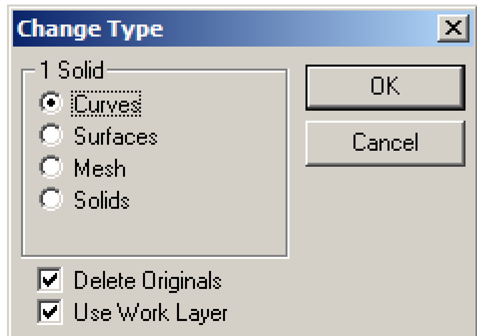
After setting the desired values, click OK and the object will convert. If more than one surface is selected to convert, the dialog box bellow appears:



Click Yes to use the same mesh conversion values for all selected surfaces.

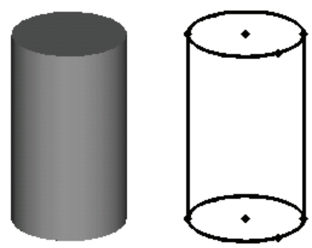
Changing Solids

When choosing the *Change Object Type* command after selecting a solid this dialog box appears:



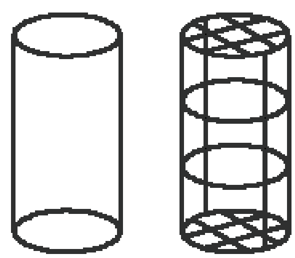
Solid To Curves Option

If the Curves option is selected, the solid is converted into a collection of editable vector splines. The shape, slope and location of the spline control points can be modified. If the *Delete Originals* option is checked, the parent solid is removed after exploding it, leaving only the curves. The resulting curves may be lines, arcs, circles, ellipses or splines.



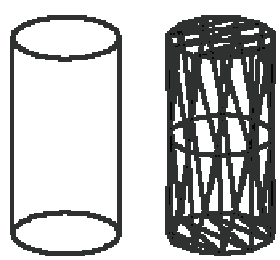
Solid to Surfaces Option

Selecting the Surfaces option explodes a solid into discrete surfaces. Once the solid is exploded, individual surfaces can be modified, moved to a different layer, transformed or removed.

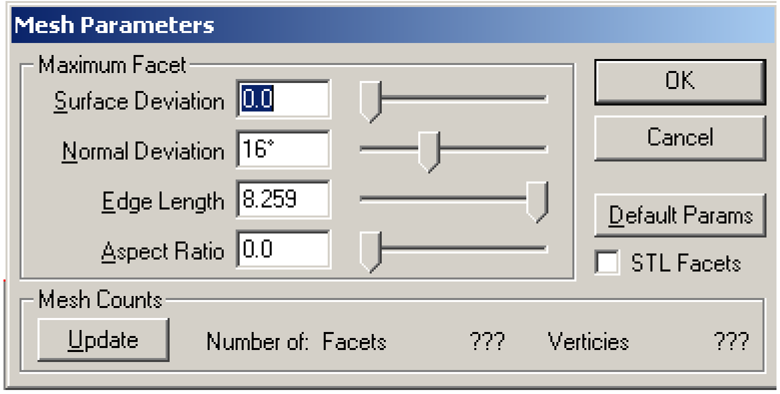


Solid To Mesh Option

If the *Mesh* option is checked, the solid converts to a triangular mesh. Exploding solids to meshes allows editing individual vertices.



When exploding 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. The default is 20 degrees. Use 10 for extremely dense meshes. Enter a value in the data field or use the slide to set the value.

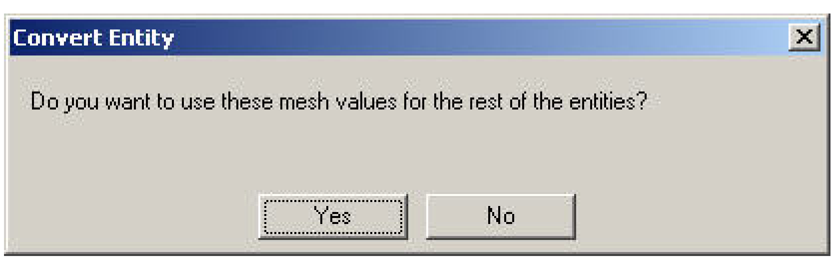
Edge Length Sets the maximum acceptable edge length for facets. Enter a value in the data field or use the slide to set the value.

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 the Update button is clicked. The calculated values replace the question marks.

Default Params Reverts the mesh parameters to the program default values.

STL Facets Forces 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 more than one solid is selected to convert, the following dialog box appears:



Click Yes to use the same mesh conversion values for all selected solids.

Solids To Solids Option

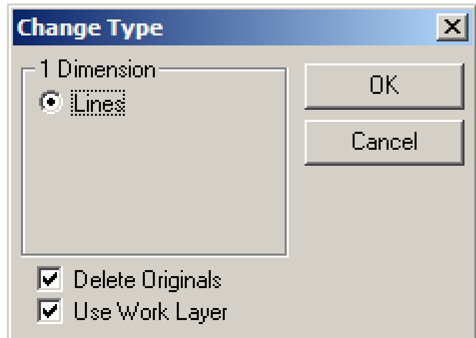
Use the Solids option to break one solid up into several smaller units. This option comes in handy for editing only one part of a larger solid.

To find out the mass properties of several solids as if they are combined into one, first unite them and analyze their properties. After the measurement is completed use the *Change Object Type Solid* option to break the part back into individual pieces.

Change Dimensions

When choosing 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, spheres, tori and planes. Candidate shapes for simplification include data imported via IGES, Step, Rhino and Alias. Also some surfaces created in the Designer Elements programs are initially created as NURBS and may simplify to an analytic shape.

Some advantages of analytic representation of shapes over NURBS that are analytic operations are considered exact whereas NURBS are precise. Analytic operations execute 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.

The Edit Objects dialog box can be displayed in two ways:

- Select the object by choosing ***Window>Edit Objects.***
- Double-click on the object.

Note: It is not possible to 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.)

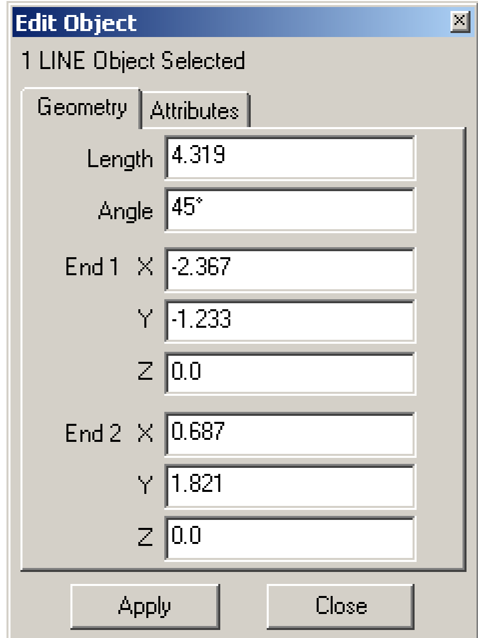
Changes made through this dialog box can be reversed with the *Undo* command.

When choosing 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

Values can be entered in this dialog box in any units desired. These will automatically be converted into the units set in Preferences. See Units.

When units are set to feet and inches (in the Preferences 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 it is necessary 1.5 inches, enter 1.5", 1.5i, 1.5in, 1.5 inch, etc.
- 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.

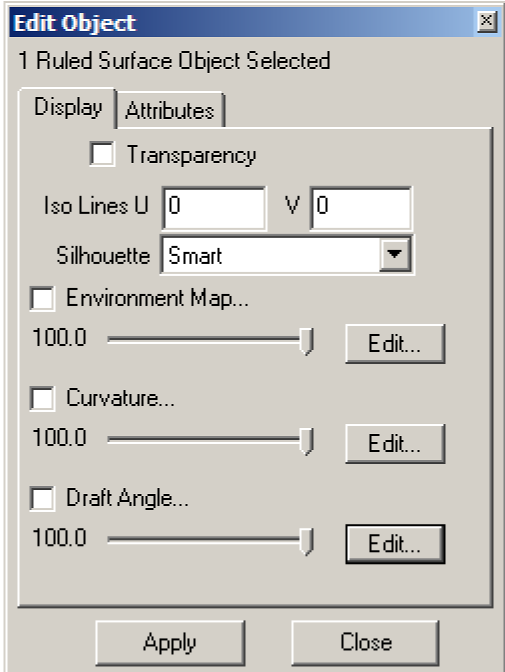
Drawing views created for Layouts have their own Geometry page. Some editing operations like *Split Solid* contain no specific geometry characteristics and therefore do not have a Geometry page.

Every chapter in this manual that deals with 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.



Transparency Makes an object or multiple selected objects to be transparent when it is enabled.

Iso Lines Controls the number of isopram lines drawn for a surface. These 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 coordinate identifiers.

Silhouette Controls whether silhouette edges of objects are displayed. 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.

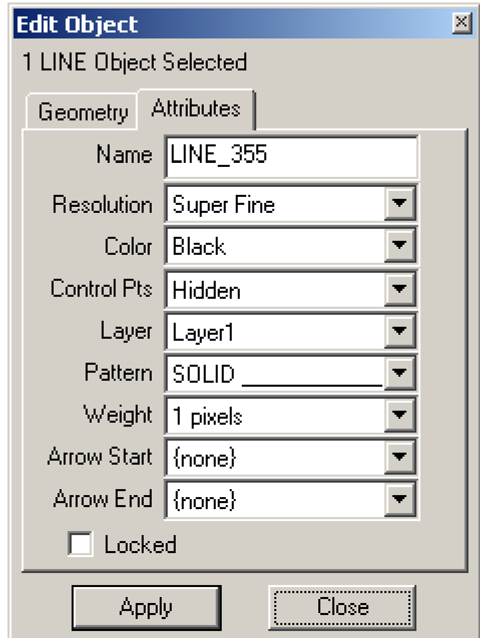
Environment Maps Provide interactive feedback on surface aesthetics. These are used to interactively evaluate curved surfaces in real-time. A real-time environment map is a way to simulate the reflections seen while examining a reflective model. Click the Edit button to use the feature. See Real-Time Environment Maps for the information on how to use the environment maps.

Curvature Creates a curvature plot of the surface. When selecting the command from the ***Verify>Surface Analysis>Curvature Settings*** or on Display tab of the Edit Object window, after pressing the Edit button the Curvature Plot Settings dialog box appears. See Curvature Command for information on using it.

Draft Angle Evaluates the drafts of an object for molding purposes. When pressing Edit button the Draft Angle dialog box appears with the draft angle analysis. See Draft Angle for information on using it.

Attributes Tab

This tab displays a standard list of characteristics for the selected object.



The attributes include:

Name Displays the name of the object. Enter a new name if necessary.

Resolution Displays the resolution setting for the object. 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. Choose any color for the object.

If the 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. Either setting can be chosen.

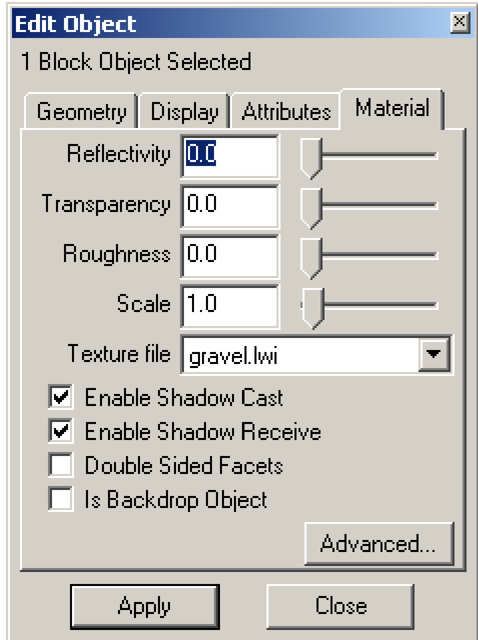
Layer Displays the layer where the object is located. Place the object on any available layer.

Locked Locks the object preventing it from being selected or modified. This is the 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 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 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 Sets the roughness of the material. Values can be entered between zero (0) and one (1).

Scale Aets 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 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 Specifies 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 Specifies 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.

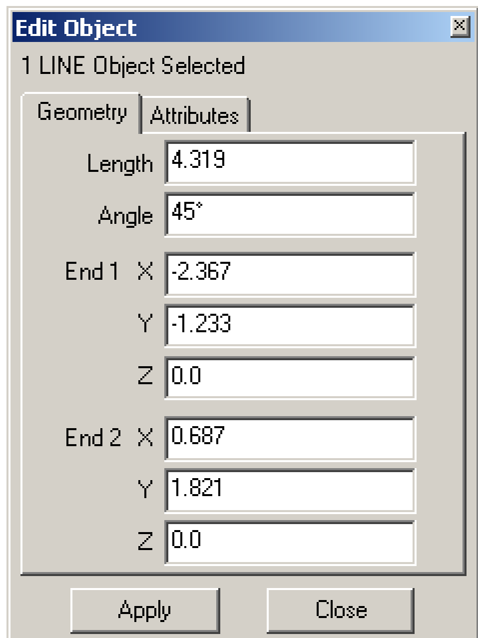
Double Sided Facets Provides additional rendering control for surfaces. Objects with normals facing away from the line of sight are not rendered. When checked all objects are rendered regardless of the normal direction.

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 Optimizes 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, the program does not need to spend much time performing light ray calculations.

Advanced button Displays the Render Material Settings dialog box. Use this dialog to change the advanced rendering settings used on the selected object.

Using the Edit 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 selected object, *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 necessary selection.
5. Make changes in other entry fields as required.
6. Click Apply and the changes automatically occur on the selected geometry.
7. Click Close to close the Edit Objects dialog box.

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.

When doing a lot of editing leave the Edit Objects dialog box displayed. Ths way it is possible to 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 choosing *Edit Objects,* only the common information is displayed. Entries are blank when the information isn't common. For example, if two concentric circles are selected, the center will be displayed in the Edit Objects box but the diameter box will be blank.

When selecting several objects and then choosing *Edit Objects*, all of the objects will reflect the changes made in the dialog box. For example, selecting all dimensions and changing 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

Many of the objects created in Cobalt and Xenon 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. By modifying a parent object (an object that defines another object), and the dependent or child object is automatically regenerated. For example, consider a skinned surface that is defined by two lines. If one of the lines is modified, the skinned surface will automatically regenerate.

Cobalt and Xenon Only

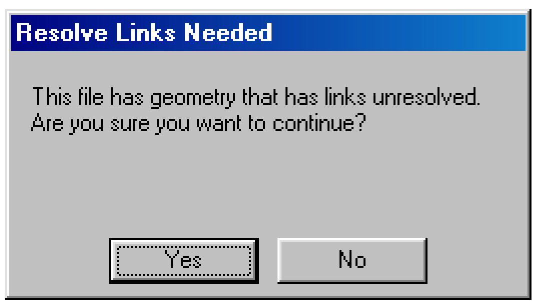
Two commands deal with Links: *Resolve* and *Remove*.

Resolve Links Command

This command is in the Edit menu. When modifying 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, Cobalt or Xenon 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.

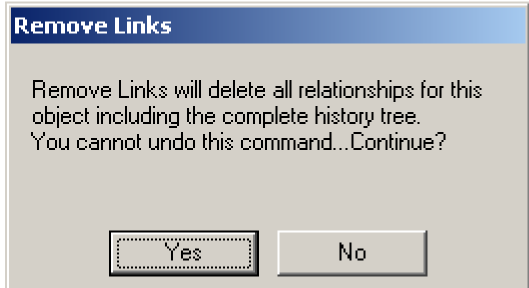
Saving a file containing links that are not resolved, the following warning appears:



Remove Links Command

The *Remove Links* command in the Edit menu deletes all associative links between a parent and child.

Select this command and the following warning appears:



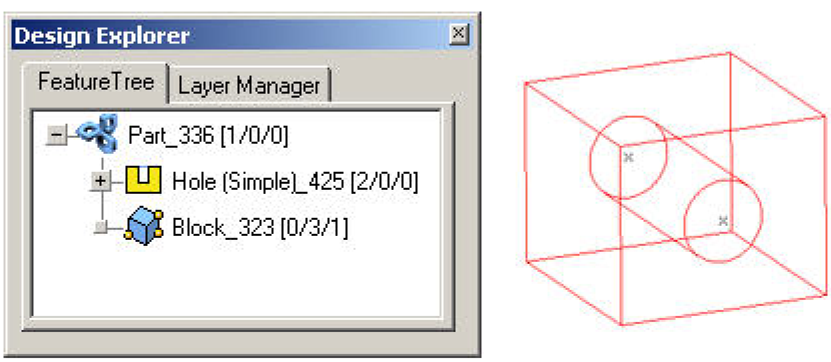
This is a warning that the selected object has a parent/child relationship to another object and that deleting this object not only deletes the history tree but cannot be undone.

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 the geometry. Use the Design Explorer instead. See a later section for information on the Design Explorer.*

Design Explorer

The Design Explorer, found in the Windows menu, is a valuable feature for traversing the drawing's design history. It offers a Windows-style tree structure with expandable branches that display the associative geometry used in the creation of all of the entities in the drawing. The Design Explorer Feature Tree displays the most recent items at the top of the list. Think of stacking new items or features on top of old ones.

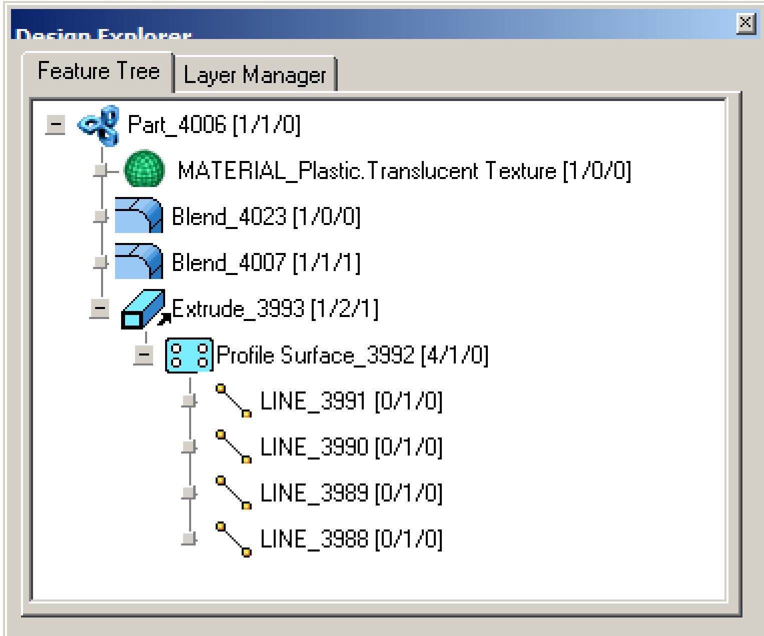


The figure shows a solid cube with a hole in one face. The Design Explorer window identifies the entity as a part and shows that it consists of a block and a hole feature. Click on items in the tree and the corresponding part in the drawing will highlight.

If an item is selected in the Design Explorer when the Edit Objects dialog box is displayed, the dialog box automatically updates to reflect the selected item.

Cobalt and Xenon Only

In/Out/History Links



Notice the numbers in brackets [ ] after each feature in the tree such as:

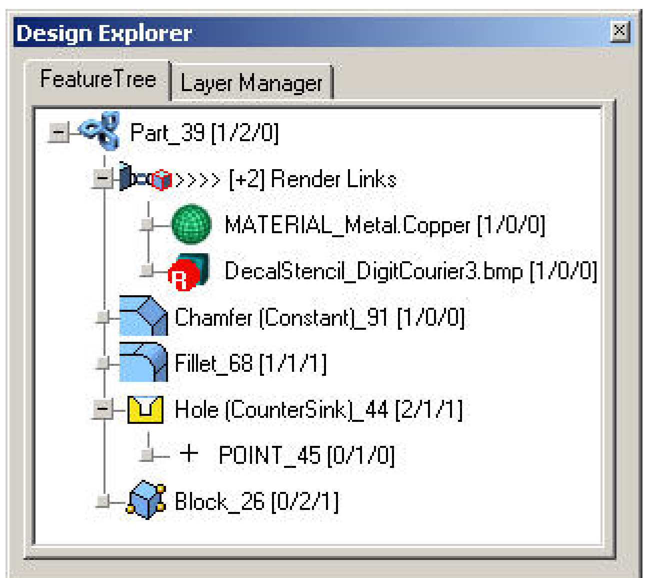
LINE_3991 [0/1/0]

These numbers represent links that go into and out of the feature, or that form a separate history. Out Links are always above In Links.

Render Links

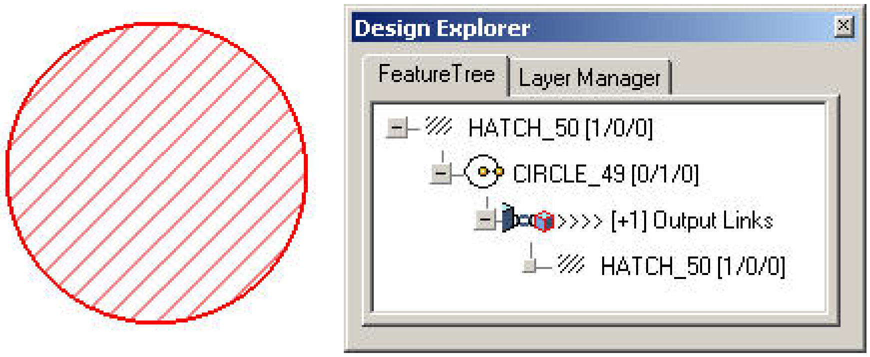
The Render Links feature organizes all persistent parts of the model in the Design Explorer. If material or decals are placed on the model, or if drawing views are created, they group separately from any geometry operations under the Render Links category.

This makes it possible to modify the part without having to replace material or recreate views.



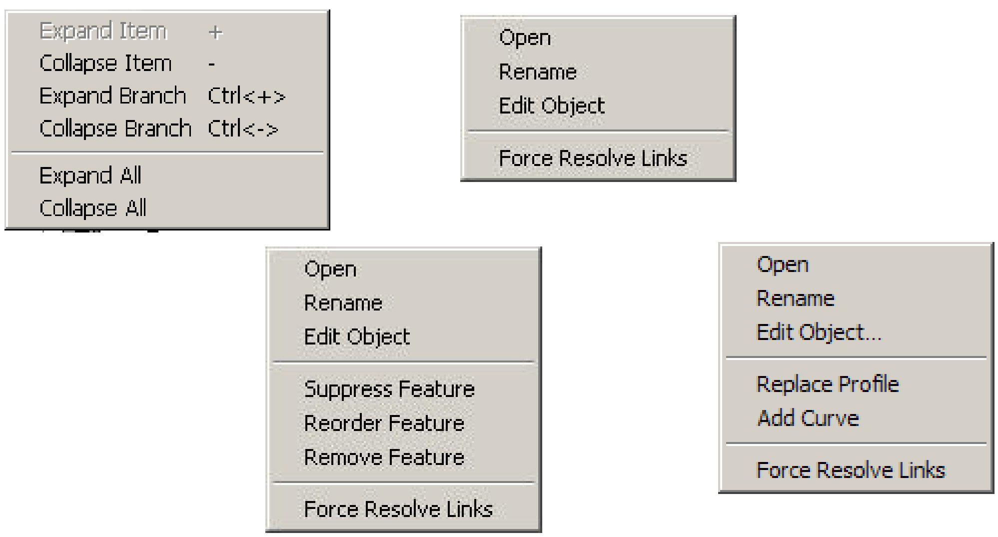
Output Link

In the picture bellow the circle has an output link to hatch.



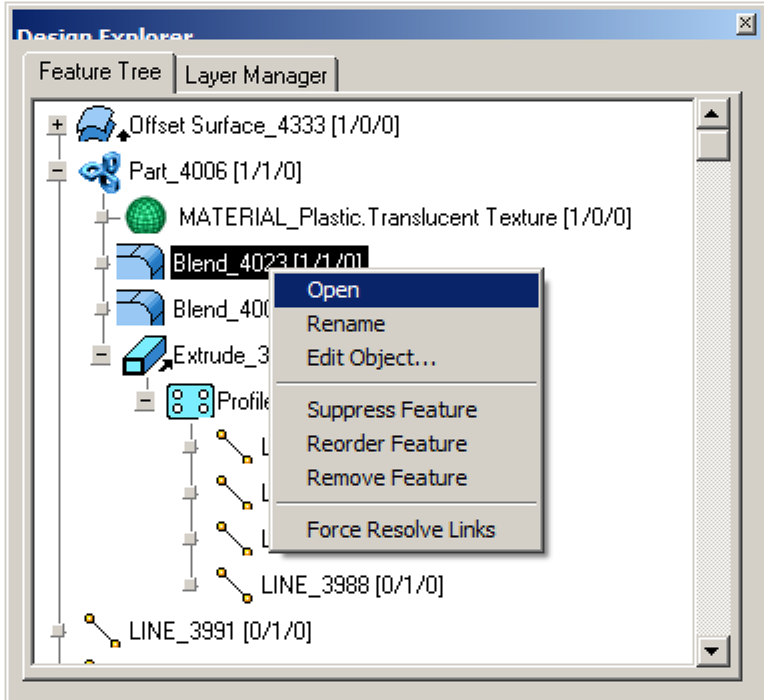
Design Explorer Commands

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 a one-button mouse, hold down the CTRL key and click on the feature. Depending on where the right click is and when, one of the following menus appear:

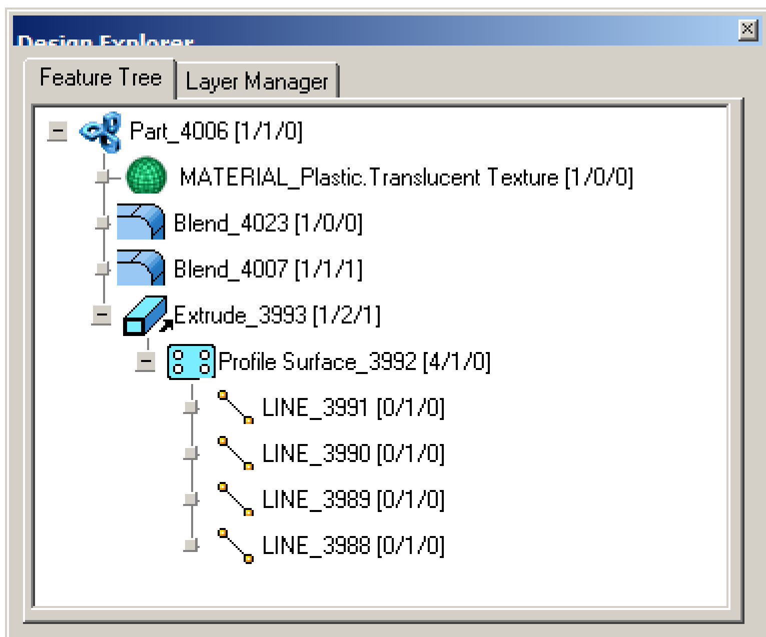


These commands are described on the following pages.

Open



The *Open* command accesses the history link as a top-level feature, as shown in the picture below.

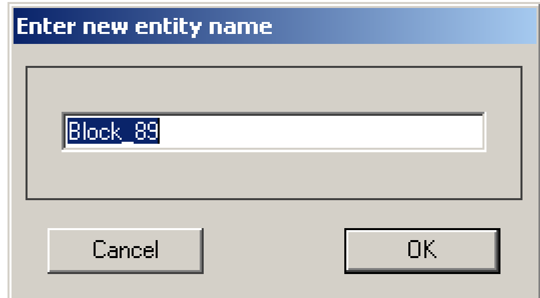


Cobalt and Xenon Only

Rename

The *Rename* command renames any of the entities. It displays this dialog box.

Enter the new name and click OK.

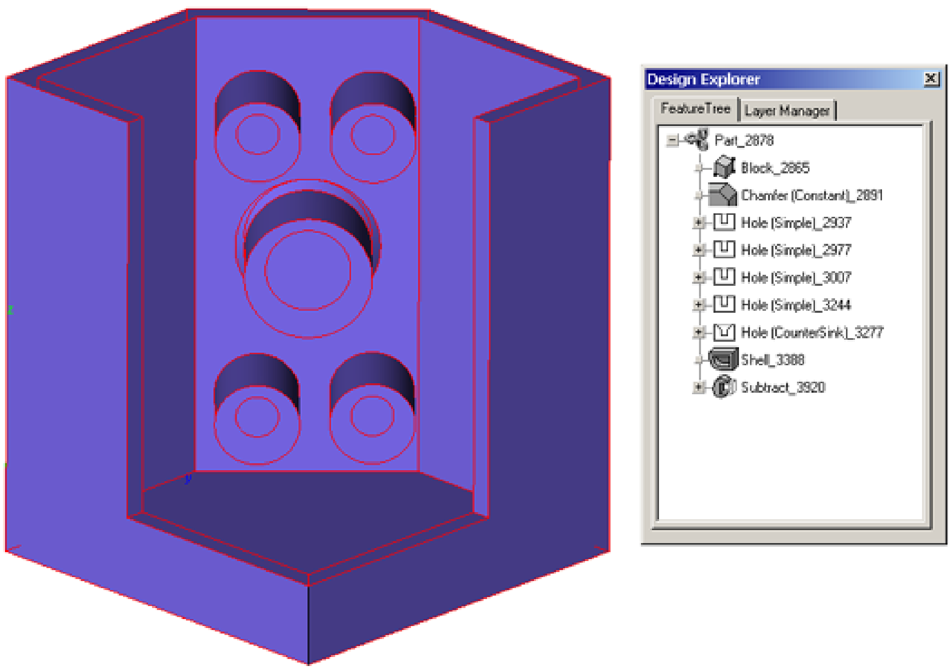


Cobalt and Xenon Only

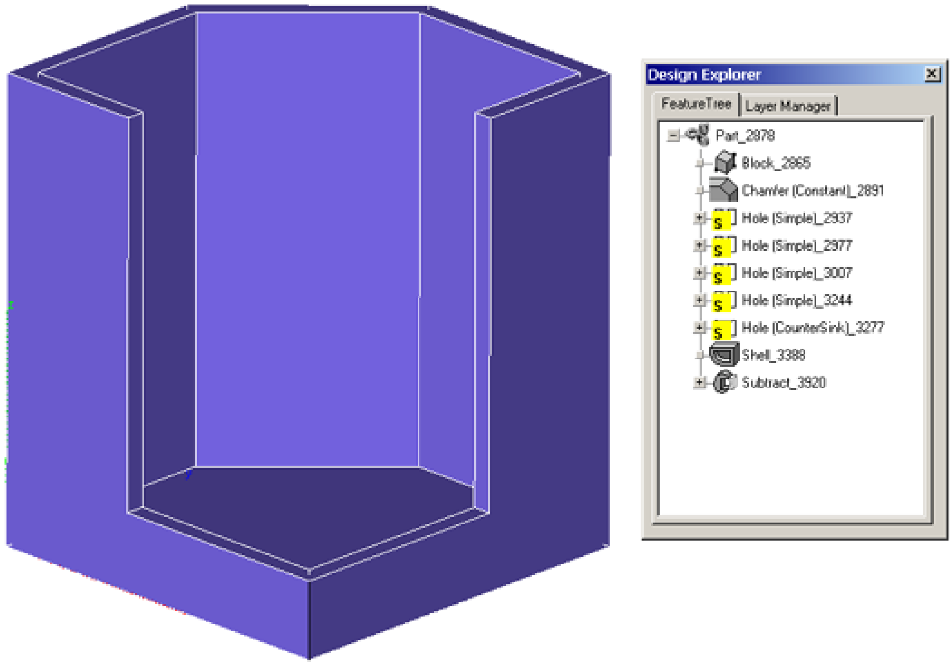
Suppress Feature/Unsuppress Feature

These commands turn off or on a particular feature in the solid history tree. They toggle between each other depending on whether an operation is suppressed.

When a feature is suppressed, the solids rebuilt 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.



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, certain features may not need to be 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, turn off blends and other features that may prevent a successful shelling operation. Then, using the *Reorder Feature* command, the shelling operation can be moved before the suppressed feature in the history tree so that shelling succeeds. Finally, unsuppress the selected feature using the *Unsuppress Feature* command.

Using the Suppress Feature Command

1. Select the model which contains the feature 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.

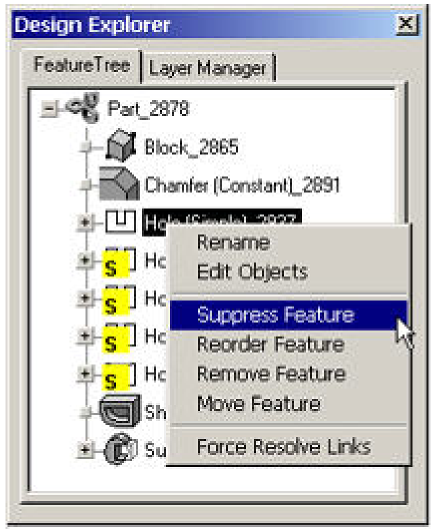
Cobalt and Xenon Only

4. Select the desired feature.

   The feature highlights.

5. Click the right mouse button on a two-button mouse or press the CTRL key with a one-button mouse to display the Design Explorer menu.
6. Choose the *Suppress Feature* command.

   The feature suppresses and the model regenerates without the feature. The Design Explorer now displays an “S” over the operation icon.

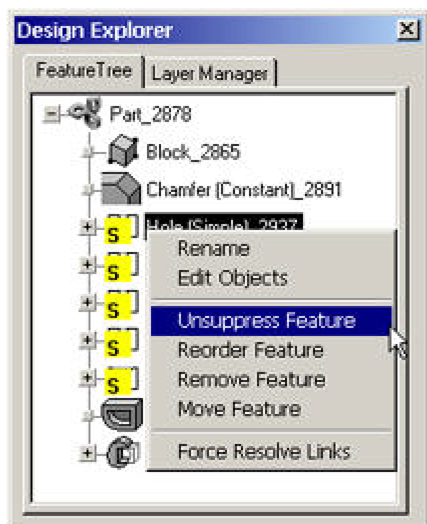


Using the Unsuppress Feature Command

The *Unsuppress Feature* command replaces the *Suppress Feature* command in the Design Explorer menu for suppressed features only.

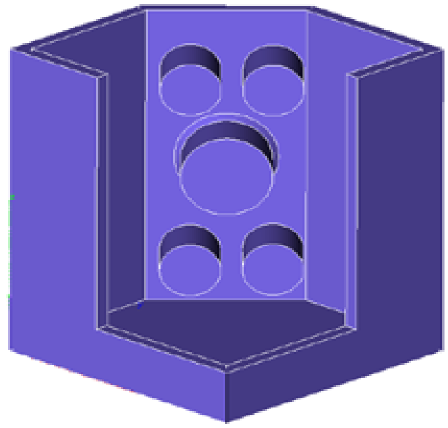
1. In the Design Explorer select the suppressed feature.
2. Click the right mouse button (Windows) or press the CTRL key with a one-button mouse to display the Design Explorer menu.
3. Choose the *Unsuppress Feature* command.

   The feature reappears and the model regenerates. The feature icon no longer displays an “S.”



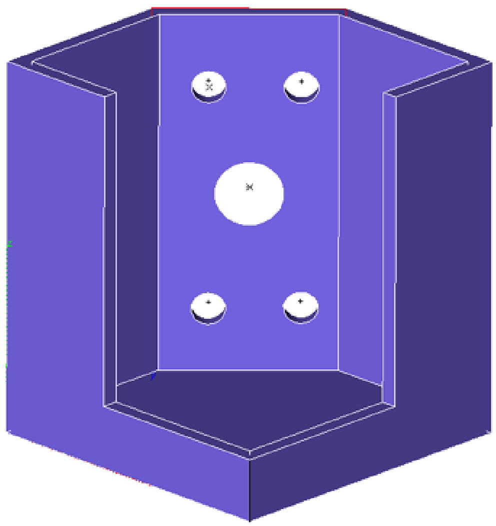
Reorder Feature

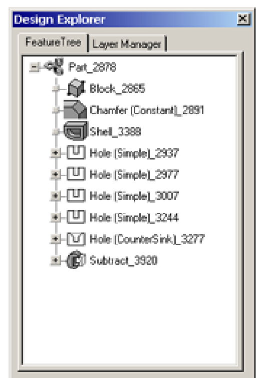
A part may be modified by rearranging its features. The Reorder Feature command moves features up or down in the history tree. In the graphic on the right, the shell operation occurred after the holes were placed.



Cobalt and Xenon Only

Using the *Reorder* command, the shelling operation was moved before the hole operation, resulting in the model shown here.





Using the Reorder Feature Command

1. Select the part.
2. Open the Design Explorer and display the tree to expose the feature 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 to move the selected operation.

   The operation moves and the part regenerates.

Cobalt and Xenon Only

> ***Tech Note/Tip:*** *It is not possible to move or reorient a feature to any side of a solid that is not visible. The operation automatically believes that one of the visible faces is chosen. Rotate the solid so that the face is visible before choosing the Move Feature command.*

Move Feature

The *Move Feature* command repositions holes and bosses within the assembly.

Using Move Feature Command

1. Select the feature in the Design Explorer to move. The feature is selected in the drawing area.
2. Display the Design Explorer menu by clicking the right mouse button on a two-button mouse or pressing the CTRL key with a one-button mouse.

3. Select the *Move* command. The Message Line appears guiding the appropriate steps.
4. Click a new center on the solid. The feature is repositioned.

Add Curve

This command adds a curve to a profile or the elements of a profile in the **Extrude, Skin, Lathe, Cutout, Protrude, Boss** and **Hole** tools.

Replace Profile

This command replaces a profile or the elements of a profile in the **Extrude, Skin, Lathe, Cutout, Protrude, Boss** and **Hole** tools.

Remove Feature

This command deletes a feature from the assembly.

Edit Parameters

Cobalt and Xenon Only

This command displays the Edit Objects dialog box. 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.

Expand Branch

This command expands the entire branch that includes the selected item.

Collapse Branch

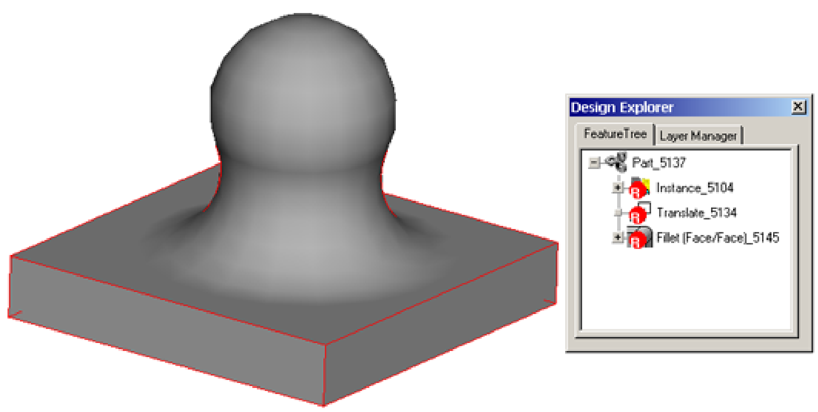
This command collapses the entire branch that includes the selected item.

Close Window

This command closes the Design Explorer.

Force Resolve Links

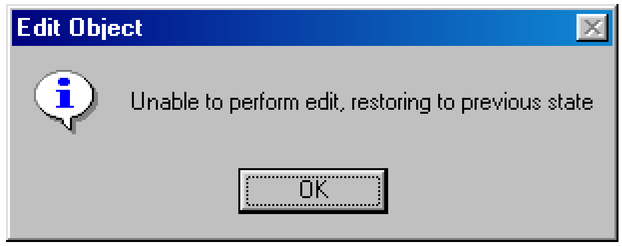
This command resolves all unresolved child geometry. Selecting the part and displaying the Design Explorer, a red “R” appears over any child operation icons that is not resolved.



This will occur if the *Enable Auto Resolve* option in the General page of Preferences is turned off. When activated, all child geometry is automatically resolved when a parent is modified. When deactivated use the *Force Resolve Links* command to resolve child geometry.

Design Explorer Rules

- Within the Design Explorer, a curve within a profile cannot be selected and its length cannot be edited through the Edit Objects dialog box. Attempting this will make a later operation invalid. The message here appears when Apply in the Edit Objects dialog box is chosen.



  Click OK to restore the original length.
  For example, 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.

Drawing Techniques

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

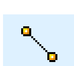
- Object Creation Methods
- Status Line
- Coordinate System Axis
- Message Line
- Trackball
- Drawing Display Commands
- Drawing at Full Scale
- Default versus Selected Object Settings
- Escape Key

Object Creation Methods

In Vellum 3D and Vellum Draft, there were two methods for creating objects, clicking and dragging. In Cobalt, Xenon and Argon surface and solid objects are created by clicking only. Wire frame geometry can be created with either method.

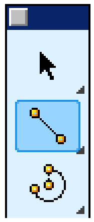
As geometry is created, a rubberband image of it appears showing how the object will look when the last point is clicked.

To make object creation easier, the wireframe tools contain a feature called Smart Pointers. When working with a tool, the cursor icon shows the points that must be indicated to create an object with that tool.



Each dot on the icon represents a point to be placed by clicking. The smart pointer indicates the order for designating points.

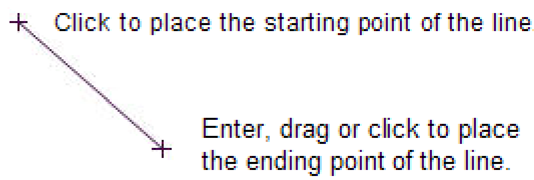
Creating a Line by Clicking



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 or drag the mouse to the desired location for the endpoint of the line. When doing so, a rubberband image of the line appears.



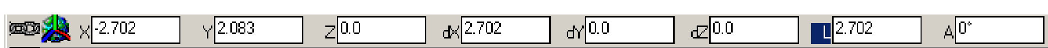
4. Release the mouse button or click to set the endpoint.

Immediately after construction, it is possible to 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.

> ***Note:*** *With the wire frame tools, it is possible to use either the click and drag method of creating objects or the click, click method as described here.*

Status Line

When a tool from a tool palette is selected, Status Line entry boxes appear along the bottom of the drawing area. The Status Line 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 delta X, Y and Z values. It also contains the value of the length and angle of the line.



The Status Line can be used in three ways:

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

When constructing an object, the status data field containing the specification that is most likely to be changed is active (the data field is highlighted). For example, when a line is drawn with the **Single Line** tool, the Length data field is highlighted so that a new value for the length can be entered. Just type a value. Press ENTER (Windows) or RETURN (Macintosh) and the line is redrawn at the new length.

The number of decimal places displayed in the data field can be set by going to ***File>Preferences>Units***.

Using the Status Line

To make an entry in a different status data field, 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 (e.g. 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, such as “ for inches. Entering a 1.5 in the field is read as 1.5 feet. To set 1.5 inches, enter 1.5”, 1.5i, 1.5in, 1.5 inch, etc.
- 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

Status Line text can be copied and pasted or used 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. Do not use the *Copy* and *Paste* commands in the Edit menu.

For Macintosh, use ⌘+C to copy and ⌘+V to paste text. The *Copy* and *Paste* commands in the Edit menu are unavailable. These function only for the Designer Elements object data (i.e. line, circle, etc.)

Status Line Entries and Tool Operations

Information entered in the Status Line is registered in two different ways depending on when in the geometry creation process it occurs.

- When a tool is selected, it is possible to immediately enter values in the Status Line data fields. When the tool operation is performed, those values will be used to create the object.
- When the object is created with a tool and is still selected, enter new values in the Status Line data fields. To move from field to field use the TAB key. When ENTER (Windows) or RETURN (Macintosh) is pressed, the object changes to reflect those values. Once ENTER (Windows) or RETURN (Macintosh) is pressed, 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 the geometry depends on the number of decimal places entered by the user.
- 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.

***Tech Note:** There are two methods for making Status Line entries in Cobalt, Xenon and Argon. The first is to 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, select the tool, type the desired values in the Status Line and press ENTER (Windows) or RETURN (Macintosh). The values are now registered. Then create the geometry with the tool.*

Using the Status Line with Drawing Tools

Once an object is drawn, the specifications can be adjusted in the Status Line including the length, angle and location. It is possible to make these changes immediately before constructing another object, selecting a different tool or choosing a command, however, it is not necessary.

It is possible to make only one series of changes in the Status Line (in as many fields as necessary); after ENTER (Windows) or RETURN (Macintosh) is pressed the program redraws the object to the specifications. After this, subsequent changes must be made 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.
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 ENTER (Windows) or RETURN (Macintosh) is pressed, the program constructs the object based on the specifications in the Status Line.

Creating Additional Geometry with the Status Line

1. With the **Single Line** tool still selected from the previous example, click two 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 a value for the X coordinate of the line's beginning point may be entered.

   Do not press ENTER (Windows) or RETURN (Macintosh) until 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.

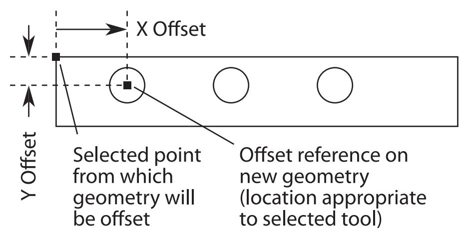
   The dX data field highlights. The dX value is the delta X, the numeric difference between the beginning and ending X coordinate.
9. Type **2**.
10. Press the TAB key. The dY data field highlights.
11. Type **2**.
12. Press the TAB key. The dZ data field highlights.
13. Type **0**.
14. Press ENTER (Windows)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

Creating Geometry Offset from a Point

To create geometry offset from an existing point, use the Status Line to specify the offset.

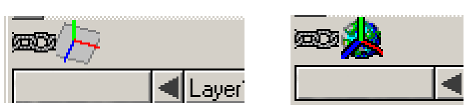
Creating Geometry Offset from a Point

1. Select the tool to be used.
2. Move the pointer over the control point from which the offset is desired.
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 the construction.



Coordinate System Axis

The coordinate system axis, at the left of the Status Line, displays the coordinate system currently set for the file. 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.

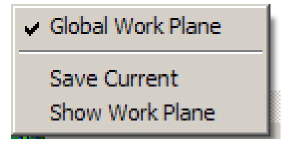


The global system aligns with the X, Y and Z axes (X = 1, 0, 0; Y = 0, 1, 0; Z = 0, 0, 1). The user-defined coordinate system is set by the user. Planes labeled as *DynWorkPlane* or *UserWorkPlane#* are part of the user-defined coordinate system. See "Work Plane" for information on the Work Plane Manager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

   A menu displays.



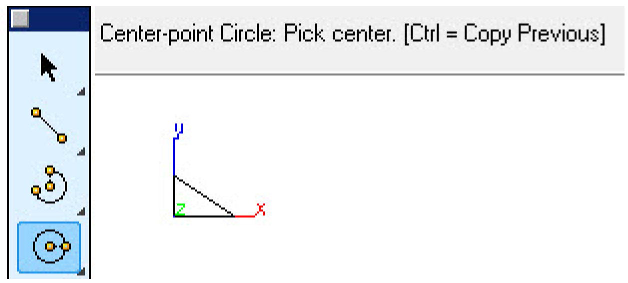
2. Choose the Global Work Plane or any user-defined work plane.

   The work plane and coordinate system are now set.

3. Continue designing.

> ***Referral:*** *See Work Plane for information on defining a work plane.*

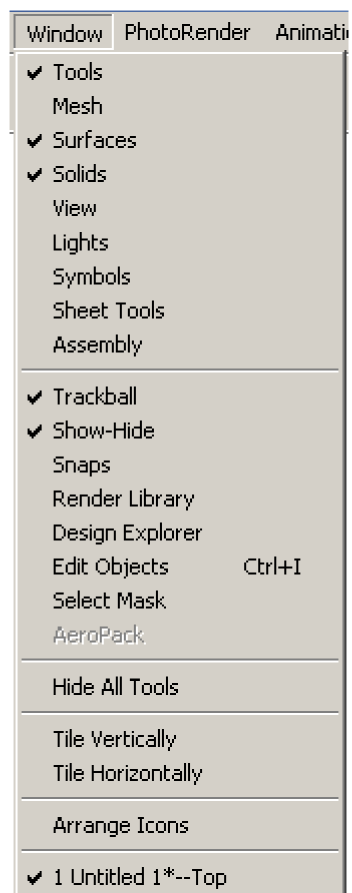
Message Line



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 each step is finished, 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* rotates the view orientation of the geometry in the drawing area. It is also possible to rotate the view around an object by selecting it before using the Trackball. If the object includes features such as holes, 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. The Trackball can be dragged to any location in the drawing area.

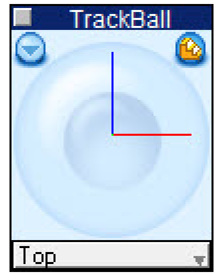
Toggle the Trackball between the sphere display and the step display. Both displays include a Views pull-down menu.

Axis Locking

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

> ***Tip:*** *Axis locking applies to geometry also. Hold down the X,Y or Z key while creating geometry and it will be locked to that axis.*

Sphere Trackball



The *Sphere Trackball* drags your view to the desired rotation. Drag beyond the boundary of the trackball and continue the rotation. The sphere trackball is the default display.

Using the Sphere Trackball

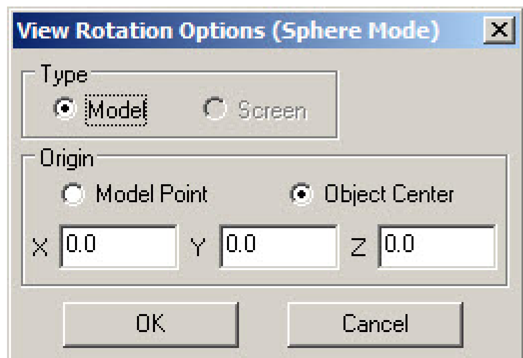
1. Choose ***Window>Trackball***.
2. Drag the pointer on the Trackball to rotate the view.

   The model rotates while dragging. See the section below for “Setting View Rotation Options”. The model continues rotating if the pointer is dragged past the edge of the trackball display and until the mouse button is released.

Setting View Rotation Options

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 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 *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 Sets the origin at either the *Model Point* or the *Object Center*.

The *Model Point* option determines the rotational point. Enter either the values in the x, y and z data fields or click on the geometry to set the location (values are entered automatically).

The *Object Center* option rotates the geometry around the center of the objects in the drawing. This center is calculated by the program. When this option is selected, the x, y, and z data fields are unavailable.

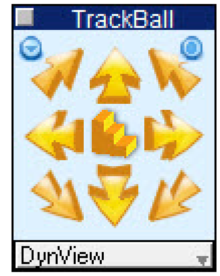
Choose only one of the four rotational options at a time, *Model Type*, *Screen Type*, *Model Point Origin* or *Object Center Origin*.

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* rotates the view in regular angle increments or in a continuous movement and provides 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.



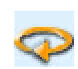
The Step Trackball includes the following icons:

Directional Arrows Rotate the view in a specific direction. Clicking the vertical arrows rotates the geometry about the x axis. Clicking the horizontal arrows rotates the geometry about the y axis. Clicking the angled arrows rotates the geometry about the z axis. (In the View Rotation Options dialog box, choose either the model or screen axis to reference the rotation when using these arrows. See the section bellow, “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 selected direction.

Set the degree increment for the steps in the View Rotation Options dialog box. See the section bellow, “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 selected direction.



Temporarily halt the rotation by placing the cursor over the arrow and pressing. When the mouse is released, the rotation continues. To stop the continuous rotation, click the circular arrow icon.

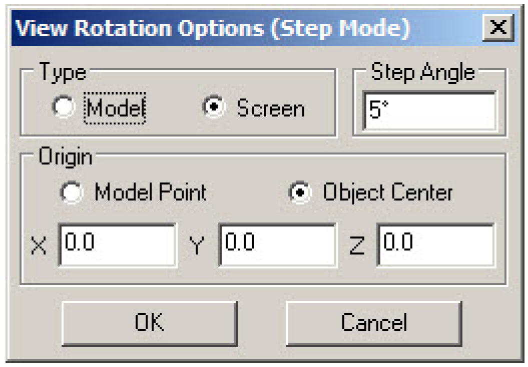
Using the Step Trackball

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

Set view rotation options for the Step Trackball in the View Rotation Options dialog box.

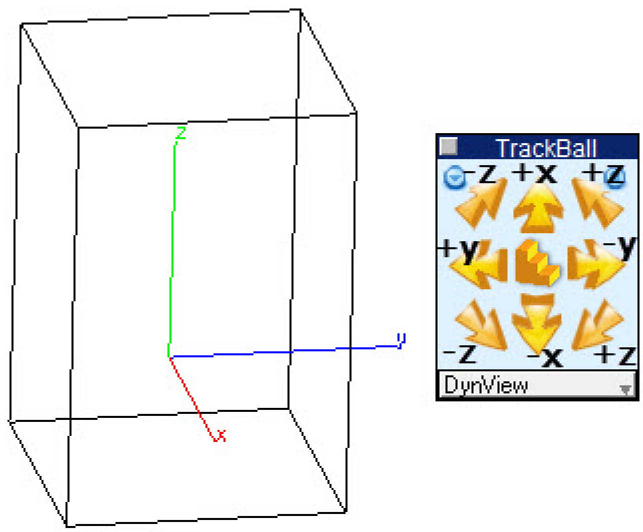


To display the dialog box, double-click on the black area of the Step Trackball. View Rotation Options containing these options:

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 rotates 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

This sets the rotation angle for the Step Trackball.

Origin

This sets the origin at either the *Model Point* or the *Object Center*.

The *Model Point* option determines the rotational point. Enter either the values in the x, y and z data fields or click on the geometry to set the location (values are entered automatically.

The *Object Center* option rotates the geometry around the center of the objects in the drawing. This center is calculated by the program. When this option is selected, the x, y, and z data fields are grayed out.

Choose only one of the four rotational options at a time, *Model Type*, *Screen Type*, *Model Point Origin* or *Object Center Origin*.

Click OK to accept the 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 to be displayed in the current window.

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

View Rotation Short Cut

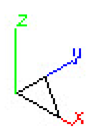
Rotate the view by holding down the SHIFT key and pressing one of the keyboard arrows. This rotates the view a specified number of degrees.

It is also possible to 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

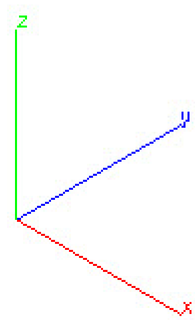
There are a number of commands that display features that may assist in creating geometry. These include: *Show/Hide Triad*, *Show/Hide Axis*, *Show/Hide Work Plane, Show/Hide Points* and the Grid commands.

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 Triad for more information.

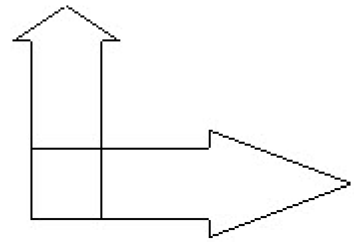
Show/Hide Axis



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 the 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 Selecting Objects for more information.

Grid

The grid overlays the drawing and can help to 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 the 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. It is possible to 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, it is impossible to construct an object closer than .25 inch to another object.

> ***Tip:*** *If a gray rectangle appears in the drawing area when the program first starts, 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 designing or drafting a highly detailed blueprint, the geometry should be created at its actual size. This Designer Elements program constructs the part using full-scale specifications and then sets 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. (It is necessary to dimension manually if the drawing is not at full scale.)
- Associative dimensions in Cobalt and Xenon update when the object is edited (manual dimensions do not).
- The size relationship of imported parts is compatible.

Once the project is drawn, it can be dimensioned and scaled visually to fit into a standard drawing size, if needed, with the *Print Setup* command (Windows) or *Page Setup* command (Macintosh) in the File menu. Regardless of how the drawing is set up, the actual size of the geometry remains constant unless it is edited.

To ensure that the geometry is contained within a specified page bounds at a 1:1 scale, the page bounds can be displayed. See Page Setup and Printing for more information.

When a new Designer Elements program document is opened, the drawing area is an infinitely large sheet so that anything can be designed at full size. As a simple 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, it is easy to create accurate full-scale drawings which are displayed at any magnification. The actual size of an object is not affected by zoom magnification or reduction.

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

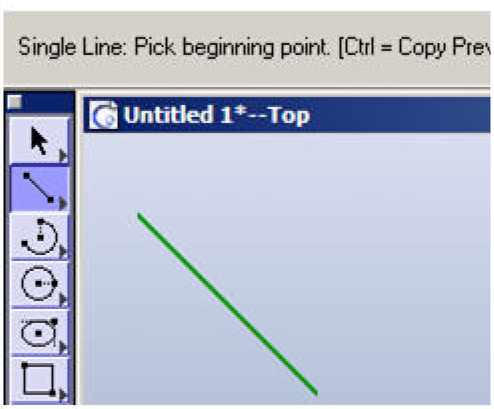
Default versus Selected Object Settings

When no object is selected, any setting changes made 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 session. When an object is selected, any change made will only affect the object.

Escape Key

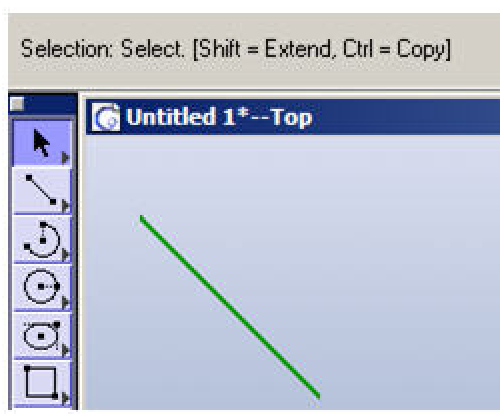
When geometry becomes quite complex, the time required for operations to complete will inevitably lengthen. It is possible to interrupt the command by pressing the ESC key.

Use ESC to deselect both an object and the tool as outlined here.

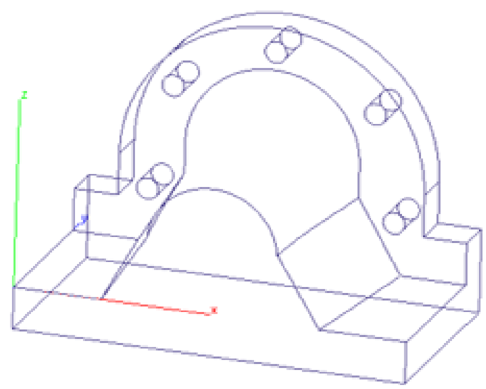


Tap ESC the first time to deselect an object, such as this line.

Tap it the second time to deselect the current tool, returning to the Select tool.



Introduction to Wireframe Modeling



Cobalt, Xenon and Argon all 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 the Desiger Elements 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:

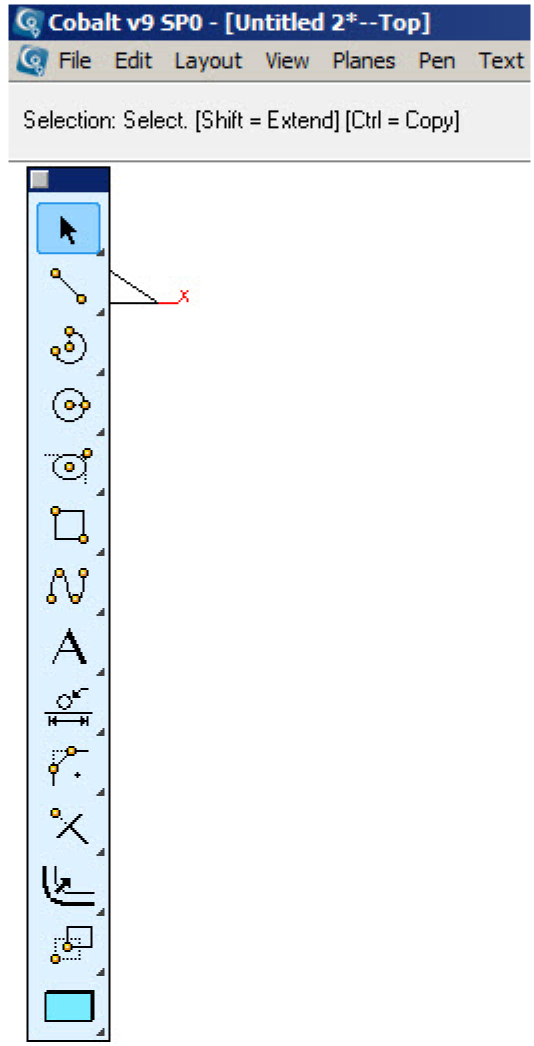
- Wireframe Tools
- Wireframes and the Drafting Assistant
- Wireframes, Selection and Display
- Object Types and Edit Objects

Wireframe Tools

The next few chapters in this section describe how to create wireframe geometry with the Desiger Elements program tools. They also provide information on how the Drafting Assistant makes designing faster and easier.

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

It is possible to change the default status (open or closed) and location of the main tool palette by choosing the status and location and quitting the Designer Elements program. The next time the program is launched, the palette will retain the status and location chosen.



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

Line tools

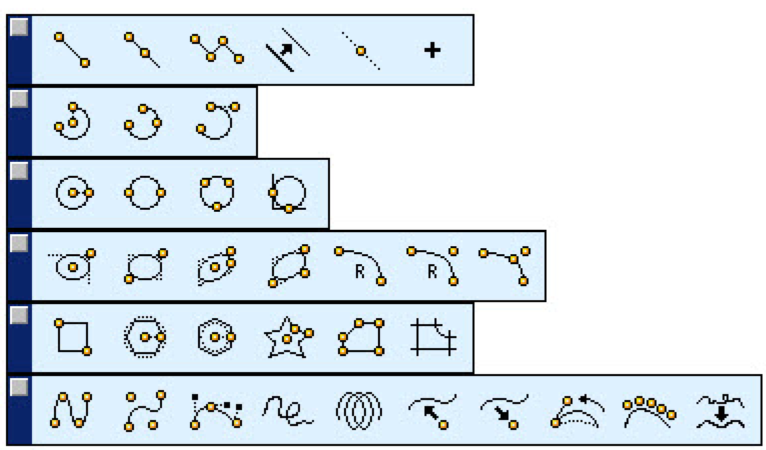
Arc tools

Circle tools

Ellipse/Conic tools

Polygon tools

Spline/Helix tools



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.

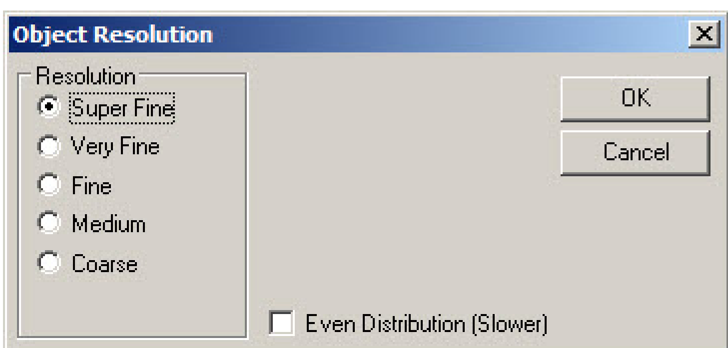
Wireframes, Selection and Display

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

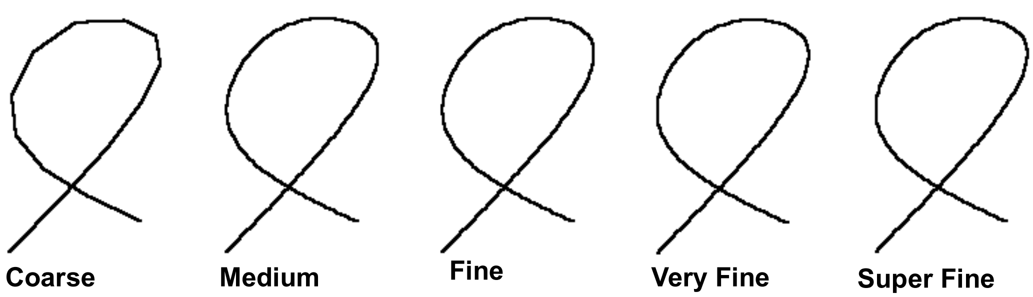
Changing an Object's Resolution

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

   The Object Resolution dialog box appears.



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



Objects Characteristics and Edit Objects

All wireframe objects created 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 10 through 14 introduce 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 Editing Objects.

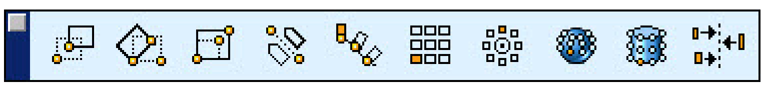
Transforming Geometry

This Designer Elements program provides tools and commands to transform the geometry with respect to scale, location and orientation.

This chapter covers the following topics:

- Transformation Tools
- Transforming Techniques

Transformation Tools



The tools on the **Transformation** tool palette include: move, rotate, expand or shrink and mirror objects; polar, linear, spherical and cylindrical duplicate; and align. Select the object to transform before or after selecting a **Transformation** tool.

Objects can be copied and transformed at the same time by holding down the CTRL (Windows) or the OPTION (Macintosh) key while specifying the transformation.

Press the SHIFT key to select additional objects after selecting a transformation tool (see the Message Line). As soon as the SHIFT key is released the **Transformation** tool is active again.

Enter values for each tool in the Status Line to perform the operation either before or after the operation is complete. If the values are entered after selecting the tool but before performing the operation, the first click in the drawing area automatically registers all Status Line values. If values are entered 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.

Item vs. Faces

All of the **Transformation** tools, with the exception of the **Alignment** tool, support the movement of individual faces in addition to entire items.

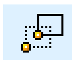
When one of these tools is selected, the Item/Faces buttons appear in the Message Line and the Message Line changes accordingly.

To transform or copy only certain faces of an object:

1. Select the appropriate transformation tool.
2. Click the **Faces** button in the Message Line.
3. Select the face or faces to transform. To copy press CTRL.
4. Select the beginning and ending reference points as directed by the Message Line.

After completing the action, while the object is still selected, enter new values in the Status Line fields to modify the face transformation. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

Move Tool



The **Move** tool moves selected objects, including control points to a new location. Copy the selection by holding down the CTRL (Windows) or OPTION (Macintosh) key. Selecting more than one object remain them in the same position relative to each other.

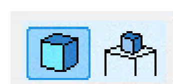
Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when moving objects with this tool.

> ***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 performing an operation on a solid or surface and then moving it to another location. This instance object displays its own unique Geometry page in Edit Objects.*

Using the Move Tool

The object may be selected before or after selecting this tool. The Message Line adjusts to accommodate this ability.

1. Select the **Move** tool.
2. Click the Item or Face icon in the Message Line. The Message Line reads: *Move: Select objects to move. [Shift = Extend]*

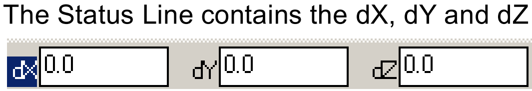
   

   Click the Item or Face icon in the Message Line. If the objects are already selected skip to step 4. The Message Line reflects the previous selection.
3. Select the object(s) to be moved.

   The Message Line reads: *Move: Pick beginning reference point. [Shift=Select, Ctrl=Copy* (Windows) or *Option=Copy* (Macintosh)*]*
4. If necessary, use SHIFT-click to select more objects, and then pick a beginning point. The Message Line now reads, *Move: Pick 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) being moved. The move is performed relative to the specified points.

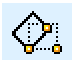
The Status Line contains the dX, dY and dZ values of the move distance and direction.



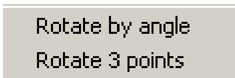
After the move and while the object is still selected, change a value in a data field and press ENTER (Windows) or RETURN (Macintosh) to accept the value.

__Tech Note:__ Be aware that when using this tool on an object that has had a subtraction boolean operation, it is necessary also to move the subtracted object. Otherwise, the subtraction operation may be altered or lost.

Rotate Tools



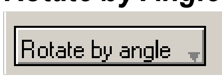
When selecting the **Rotate** tools icon a drop down menu appears in the Message Line containing two tools, **Rotate by Angle** (or by one point) and **Rotate by 3 Points**.



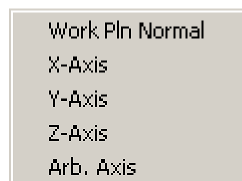
Use them to rotate a selected set of objects around a point in any axis, or to create a circular array.

When rotating by one point or by three points, the selection can be copied by holding down the CTRL (Windows) or OPTION (Macintosh) key while selecting the objects. When selecting more than one object, they remain in the same position relative to each other.

Rotate by Angle Tool

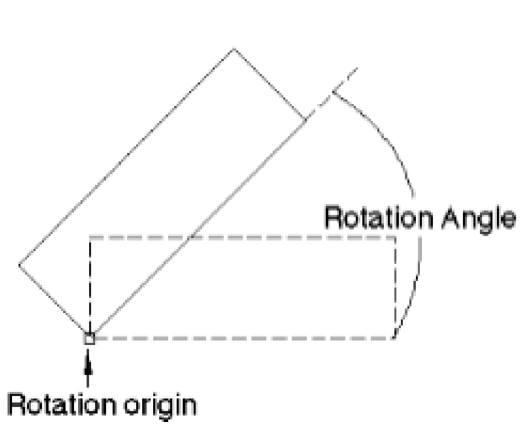


This tool rotates one or more objects around a specified point. The Message Line contains a pull-down menu for specifying the rotation axis.



The options include:

Work Plane 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.
Arbitrary	Rotates the objects about an axis specified by picking two points on screen.

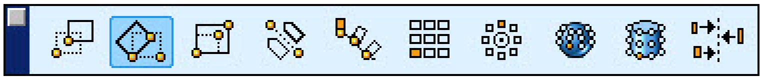


Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when rotating objects with this tool.

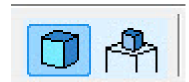
Using the Rotate by Angle Tool

Select the object before or after selecting this tool. The Message Line adjusts accordingly.

1. Select the **Rotate** tool.



2. Click the Item or Face icon in the Message Line.



3. Select the **Rotate by Angle** tool in the Message Line. The Message Line reads: *Rotate: Select items [or feature faces] to rotate. [Shift = Extend]*.

   If the objects are already selected, skip to step 4. The Message Line reflects the previous selection.

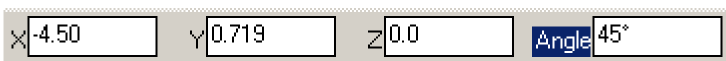
4. Select the objects to rotate.

   The Message Line reads: *Rotate by Angle: Enter location to rotate about [Shift=Select, Ctrl=Copy* (Windows) or *Option=Copy* (Macintosh)*]*.

5. Use SHIFT-click to select more objects.
6. Select the rotation axis from the pull-down menu.
7. Specify the center of rotation.

   The object is rotated.

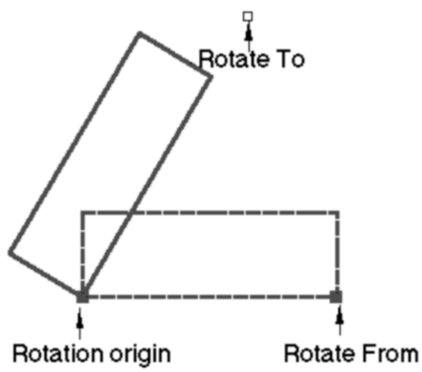
The Status Line contains the X, Y and Z values of the rotation point and the rotation angle. After completing the rotation click in any data field and enter a new value.



Rotate 3 Points Tool

The **Rotate 3 Points** tool rotates an object based on three defining points, the center of rotation, a beginning reference point, and an ending reference point. The program determines the rotation angle from the two reference points. The two reference points need not be on the object.

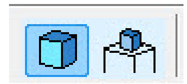
Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when rotating objects with this tool.



Using the Rotate 3 Points Tool

Select the object before or after selecting this tool. The Message Line adjusts to accommodate this ability.

1. Select the **Rotate** tool.
2. Click the Item or Face icon in the Message Line.



3. Select the **Rotate 3 Points** tool in the Message Line. The Message Line reads: *Rotate: Select items [or feature faces] to rotate. [Shift = Extend]*

   If the objects are already selected, skip to step 4. The Message Line reflects the previous selection.
4. Select the objects to rotate.

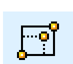
   The Message Line reads: *Rotate: Pick center of rotation [Shift=Select, Ctrl=Copy* (Windows) or *Option=Copy* (Macintosh)*]*. If necessary, use SHIFT-click to select more objects.
5. Click the rotation point.
6. Click the first reference point.
7. Click the second reference point.

   The object is rotated.

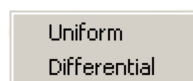
The Status Line contains the rotation angle.

After the rotation is completed and while the object is still selected, enter a new value if necessary. Press ENTER (Windows) or RETURN (Macintosh) to accept the value.

Resize/Scale Tool



The **Resize/Scale** tool enlarges or reduces objects. The Message Line contains a pull-down menu with two options:



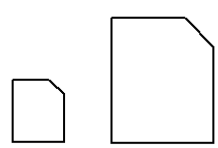
Uniform	Maintains the proportions of the stretched objects. It can be used for curves, surfaces and solids.
Differential	Specifies separate scales in the X, Y and Z axes. It can be used for curves, surfaces and solids.

It is possible to copy the selection as it is scaled by holding down the CTRL (Windows) or OPTION (Macintosh) key. If more than one object is selected, they remain in the same position relative to each other.

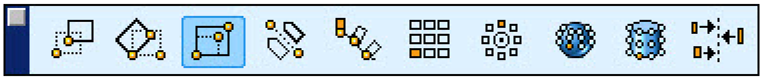
Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when stretching objects.

Using the Resize/Scale Tool - Uniform Option.

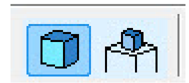
This option/tool expands or shrinks the curves, surfaces and solids uniformly. Select the object before or after selecting this option. The Message Line adjusts accordingly. The graphic here shows uniformly scaled curves.



1. Select the **Resize and Scale** tool. In The Message Line reads: *Resize and Scale: Select items [or feature faces] to scale [Shift = Extend].*



2. Click the Item or Face icon in the Message Line.

   If the objects are already selected skip to step 4. The Message Line reflects the previous selection.
3. Select the object(s) to scale. The option pull-down menu appears.
4. Select the *Uniform* option from the menu. When moving the pointer into the drawing area the pointer becomes the expand/shrink icon, shown here. 

   The Message line now reads, *Resize/Scale: Pick anchor point [Shift = Select, Ctrl = Copy* (Windows) or *Option = Copy* (Macintosh)*].* It is possible to select more objects.
5. Click the anchor point on the 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, *Resize/Scale: Pick beginning reference point [Ctrl = Copy* (Windows) or *Option = Copy* (Macintosh)*].*

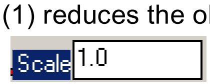
   The beginning reference point is a point on the geometry used in conjunction with the ending reference point to set the scale of the operation. Rather than specifying reference points just enter a value in the Scale data field and press ENTER (Windows) or RETURN (Macintosh) and the object will scale.
6. Click the beginning reference point.

   The Message Line reads: *Resize/Scale: Pick ending reference point [Shift=Select, Ctrl = Copy* (Windows) or *Option = Copy* (Macintosh)*].*
7. 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.

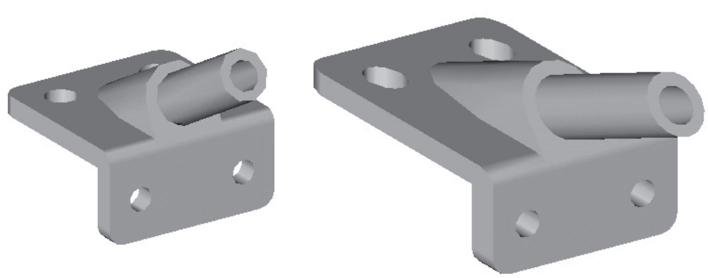
After completing the scale and while the object is still selected, enter a new value scale. Press ENTER (Windows) or RETURN (Macintosh) to accept the new value.

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.

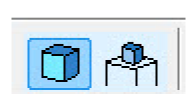


Using the Resize/Scale Tool - Differential Option

This option/tool scales the curves, surfaces and solids differentially. 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.



1. Select the **Resize/Scale** tool. The Message Line reads: *Resize/Scale: Pick items [or feature faces] to scale. [Shift = Extend]*
2. Click the Item or Face icon in the Message Line.

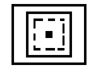
   

   If the objects are already selected skip to step 3. The Message Line reflects the previous selection.

3. Select the object(s) to scale.

   The option pull-down menu appears.

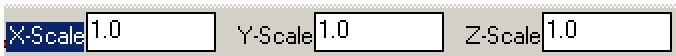
4. Select the *Differential* option from the menu. When moving the pointer into the drawing area the pointer becomes the scale icon, shown here:



   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.

5. Enter the X, Y and Z Scale values in the Status Line.
6. Click a point about which the object will be scaled.

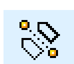
The Status Line contains the X-Scale, Y-Scale and Z-Scale data fields.



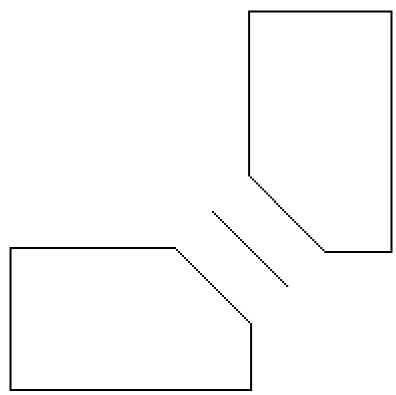
After completing the scale, while the object is still selected, new values can be entered. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

> ***Tech Note:*** *When using the Differential option of the Resize/Scale tool for surfaces and solids, the stretched object displays its own* Geometry *page in Edit Objects.*

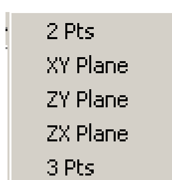
Mirror Tool



The **Mirror** tool creates a mirror image of an object or objects on the opposite side of a reference line. Copy the selection by holding down the CTRL (Windows) or OPTION (Macintosh) key. Selecting 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 Points	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 Points	Mirrors the object around the mirror plane specified by three points.

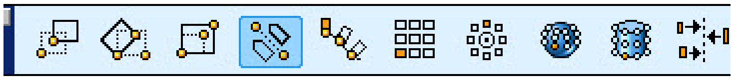
> ***Tip:*** *When using the plane options, it may be helpful to display the Axis for reference. Choose* ***Layout>Show Axis****.*

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.

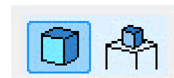
Using the Mirror Tool - 2 Points Option

The object can be selected before or after selecting this option. The Message Line adjusts to accommodate this ability.

1. Select the **Mirror** tool. The Message Line reads: *Mirror: Pick items [or feature faces] to mirror [Shift = Extend]*.



2. Click the Face or the Item icon in the Message Line.



   If the objects are already selected, skip to step 4. The Message Line you see reflects the previous selection.
3. Select the object(s) to be mirrored.

   An option pull-down menu appears.
4. Select the *2 Points* option in the pull-down menu. The Message Line reads: *Mirror: Pick beginning of reference line. [Shift=Select, Ctrl=Copy* (Windows) or *Option=Copy* (Macintosh)*]*

   If necessary, use SHIFT-click to select more objects.
5. Click a location to set the beginning of the reference line around which the object mirrors. The Message Line now reads, *Mirror: Pick end of reference line. [Ctrl = Copy* (Windows) or *Option = Copy* (Macintosh)*]*

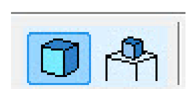
   Drag to specify the beginning and endpoints and the object mirrors.
6. Click to locate the end of the reference line.

   The reference line need not be parallel to the object.

Using the Mirror Tool - XY Plane Option

The object can be selected before or after selecting this option. The Message Line adjusts to accommodate this ability.

1. Select the **Mirror** tool.
2. Click the Item or Face icon. The Message Line reads: *Mirror: Pick items [or feature faces] to mirror. [Shift = Extend]*



   If the objects are already selected skip to step 4. The Message Line reflects the previous selection.
3. Select the object(s) to be mirrored.

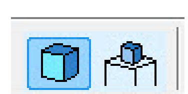
   An option pull-down menu appears.
4. Select the *XY Plane* option in the pull-down menu. The Message Line reads: *Mirror 1 Point: Enter 1 point for mirror origin. [Shift=Select, Ctrl=Copy* (Windows) or *Option=Copy* (Macintosh)*]*

   Use SHIFT-click to select more objects.
5. Click a point in the XY plane to specify a reference point for the mirror.

Using the Mirror Tool - ZY Plane Option

The object can be selected before or after selecting this option. The Message Line adjusts to accommodate this ability.

1. Select the **Mirror** tool.
2. Click the Item or Face icon. The Message Line reads: *Mirror: Select items to mirror [Shift = Extend]*.



   If the objects are already selected skip to step 3. The Message Line reflects the previous selection.

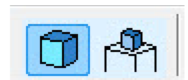
3. Select the object(s) to be mirrored. An option pull-down menu appears.
4. Select the *ZY Plane* option in the pull-down menu. The Message Line reads: *Mirror 1 Pt: Enter 1 point for mirror origin [Shift= Select, Ctrl=Copy* (Windows) or *Option=Copy* (Macintosh)*]*.

   Use SHIFT-click to select more objects.
5. Click a point in the ZY plane to specify a reference point for the mirror.

Using the Mirror Tool - ZX Plane Option

The object can be selected before or after selecting this option. The Message Line adjusts to accommodate this ability.

1. Select the **Mirror** tool.
2. Click the Item or Face icon. The Message Line reads: *Mirror: Pick items [or feature faces] to mirror. [Shift = Extend]*



   If the objects are already selected skip to step 4. The Message Line reflects the previous selection.
3. Select the object(s) to be mirrored. An option pull-down menu appears.
4. Select the *ZX Plane* option in the pull-down menu. The Message Line reads: *Mirror 1 Point: Enter 1 point for mirror origin. [Shift=Select, Ctrl=Copy* (Windows) or *Option=Copy* (Macintosh)*]*

   Use SHIFT-click to select more objects.
5. Click a point in the ZX plane to specify a reference point for the mirror.

Using the Mirror Tool - 3 Pts Option

The object can be selected before or after selecting this option. The Message Line adjusts to accommodate this ability.

1. Select the **Mirror** tool.
2. Click the Item or Face icon. The Message Line reads: *Mirror: Pick items [or feature faces] to mirror. [Shift = Extend]*

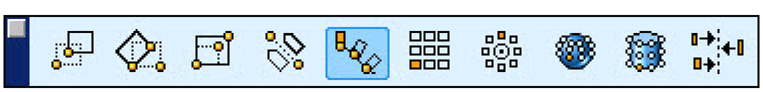
   If the objects are already selected skip to step 4. The Message Line reflects the previous selection.
3. Select the object(s) to be mirrored.

   An option pull-down menu appears.
4. Select the *3 Points* option in the pull-down menu. The Message Line reads: *Mirror 3 Points: Enter 3 points for mirror plane. [Shift=Select, Ctrl=Copy* (Windows) or *Option=Copy* (Macintosh)*]*

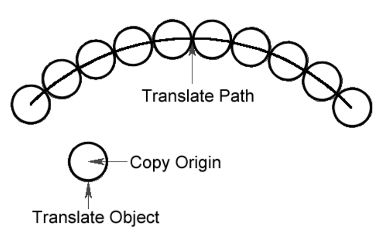
   Use SHIFT-click to select more objects.
5. Click three points to specify a reference plane for the mirror.

There are no Status Line entries for the **Mirror** tools.

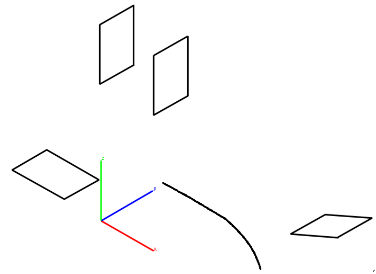
Copy along Path Tool



The **Copy along Path** tool copies an object along a path by the number of specified times. The objects will be equally spaced along the curve.



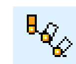
Align the copies along the X, Y and Z axis (the alignment axis). The graphic here shows a polygon created in the Top plane with a Z alignment axis.



Using the Copy Along Path Tool

Select the object before or after selecting this tool. The Message Line adjusts to accommodate this ability.

1. Select the **Copy along Path** tool.



2. Select the Item or Face icon in the Message Line. The Message Line reads: *Copy Along Path: Select items [or feature faces] to duplicate. [Shift = Extend]*

   If the objects are already selected skip to step 3. The Message Line reflects the previous selection.

3. Select the objects to copy.

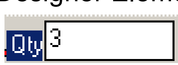
   The Message Line now reads: *Copy along Path: Select the translation path for objects*.

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*.

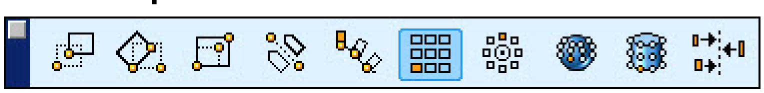
   The copies can be aligned along the X, Y or Z axis. Use the Drafting Assistant to choose the desired axis.

7. Pick two points on the screen to show where to align the object(s).

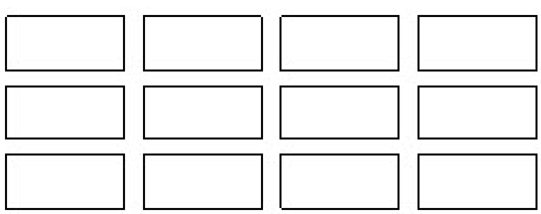
The Status Line contains the Qty (Quantity) field displaying the number of objects this Designer Elements program will create. Set this value before copying the objects.



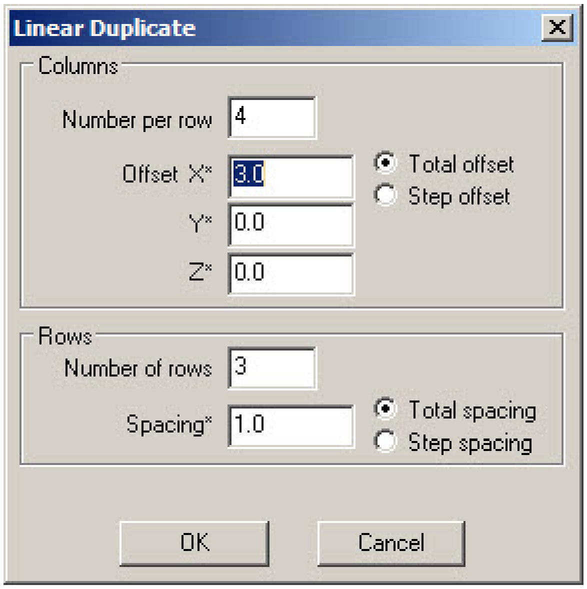
Linear Duplicate Tool



The **Linear Duplicate** tool creates copies of an object in a rectangular array.



To complete the array, 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.



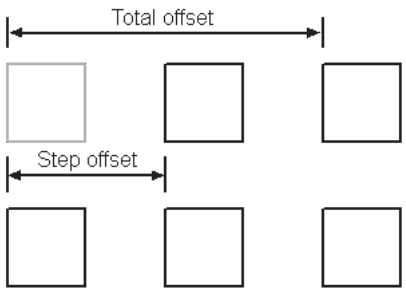
The dialog box contains the following options:

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 the number is unknown, use math operators to determine the number of copies. For example, to place studs 16 inches apart on a 17 foot wall, enter (17*12)/16. The program will truncate the result to 12 copies.

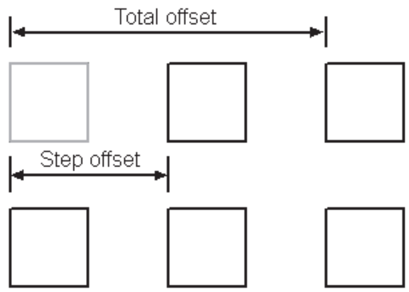
Offset X*, Y*, Z* (Columns section) These fields set the offset length in the X, Y or Z direction for the duplicated objects. These values work with the Total offset and Step offset options. Enter these values or drag in the drawing area to enter the values.

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.



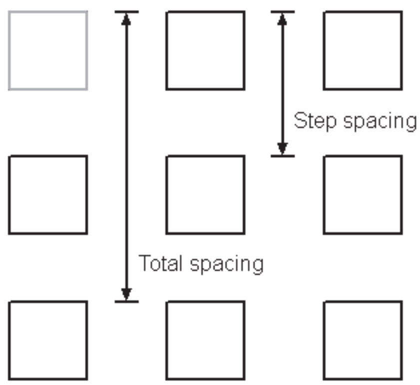
Step offset (Columns section) When selected, this specifies that the values in the X, Y and Z fields are step offset values from the selected object to the first copy.

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.



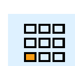
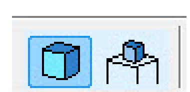
Step spacing (Rows section) When selected, this option specifies that the Spacing value is the distance from the first selected object to the first copy. This option works with the Spacing field.

OK/Cancel Click OK to complete the operation. Click Cancel to discontinue the operation.

The asterisk (*) indicates that the value automatically fills in when dragging the pointer in the drawing area to indicate the offset or spacing.

Using the Linear Duplicate (Rectangular Arrays) Tool

Select the object before or after selecting this tool. The Message Line adjusts to accommodate this ability.

1. Select the **Linear Duplicate** tool. 
2. Select the Items or Faces icon in the Message Line. 
3. The Message Line reads: *Linear Duplicate: Select objects for linear duplicate. [Shift = Extend]*

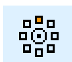
   If the objects are already selected the Linear Duplicate dialog box appears. Skip to step 4.
4. Select the object(s) to be duplicated. The dialog box appears.

   The Message Line now reads: *Input Set* (followed by hints for the highlighted field in the dialog box)

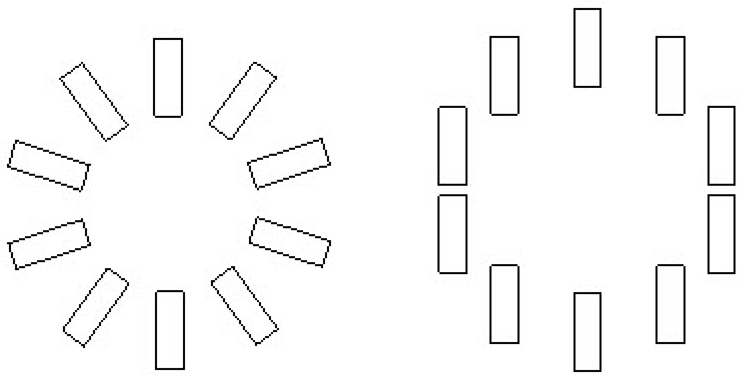
5. Enter the number of columns in the *Number per row* data field.
6. Select offset option, either Total or Step.
7. If the *Offset X** field is not selected, click in the field.
8. Move the pointer to the drawing area and drag to indicate the offset for the row of copies. Values automatically appear in the X*, Y* and Z* data fields.
9. Select the spacing option, either Total or Step.
10. If the *Spacing** field is not selected, click in the field.
11. Move the pointer to the drawing area and drag to indicate the spacing for the spacing for the rows.
12. Click OK to close the dialog box and create the duplicates.

There are no Status Line entries.

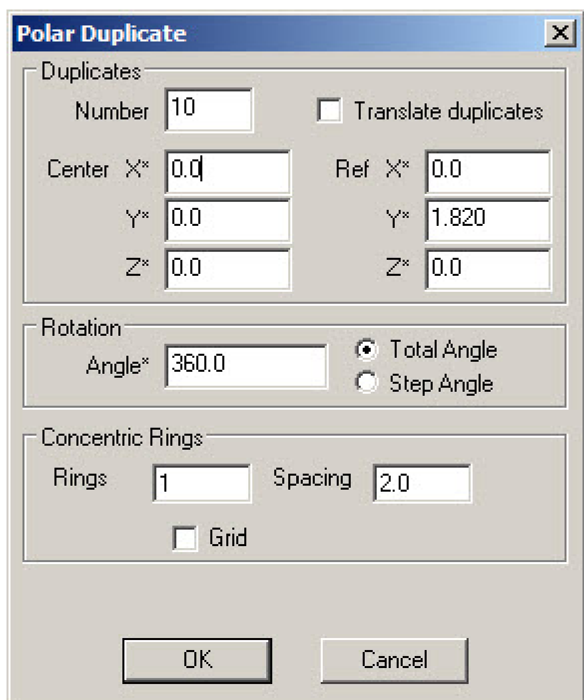
Polar Duplicate Tool



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.

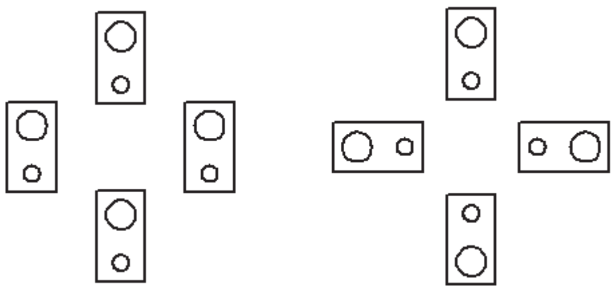


The dialog box contains the following options:

Number Sets the total number of objects. Be sure to include the selected object in this total count.

Center X*, Y*, Z* Set the X, Y and Z location for the center of the circular array. It is possible to enter these values or click the place in the drawing area for the center.

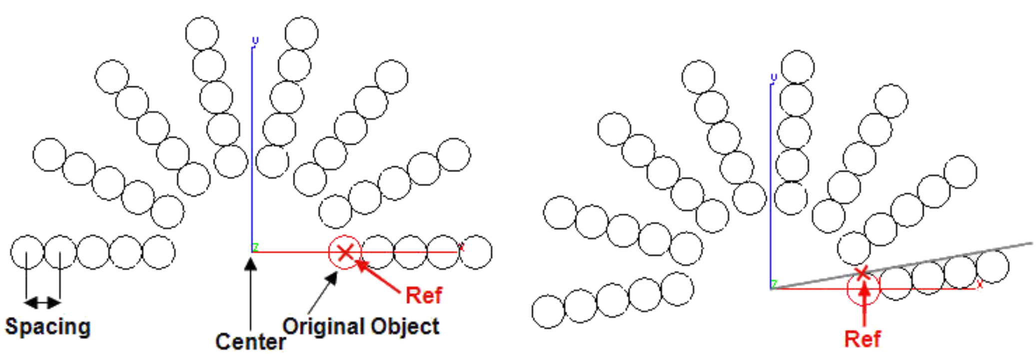
Translate duplicates When selected, creates copies in the same orientation as the original object. When this option is not selected, copies are rotated around the center specified.



Translate Duplicates Option On **Translate Duplicates Option Off**

Ref X*, Y*, Z* Represent the reference point for the Translate duplicates option.

Ref X, Y and Z determine the imaginary point duplicated around the center 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.

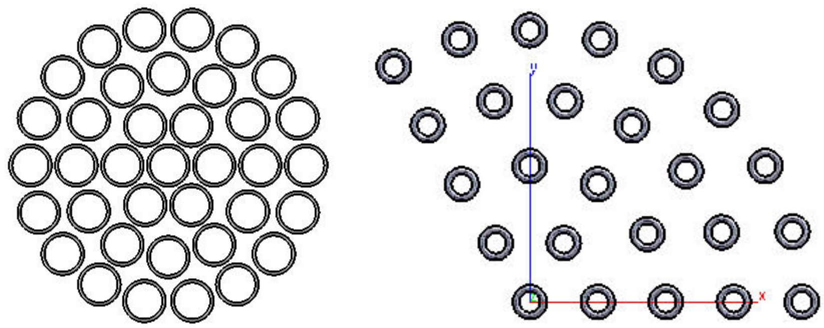


Specify the Ref point by entering values or clicking at the desired location in the drawing area.

Angle Specifies the angle of rotation. This field operates with the Total angle and Step angle options.

Ref X*, Y*, Z* Represent the reference point for the Translate duplicates option.

Total angle 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.) The result on the left has a rotation angle of 360^0, where the result on the right changes the rotation angle to 120^0



360° Rotation Angle **120° Rotation Angle**

Step angle (Rotation section) When selected this option specifies that the angle value is the angle between the center of two adjacent objects.

Grid Fills the area of the pattern with the maximum amount of the features.

Rings Designates the number of concentric rings in a pattern.

Spacing Designates the spacing between the rings.

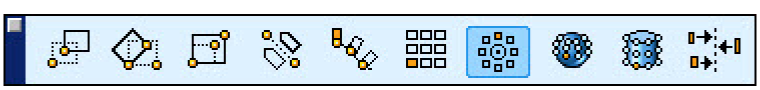
OK/Cancel Click OK to complete the operation. Click Cancel to discontinue the operation.

The asterisk (*) indicates that the value automatically fills in when dragging the pointer in the drawing area to indicate the offset or spacing.

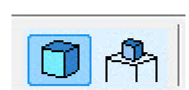
Using the Polar Duplicate Tool

Select the object before or after selecting this tool. The Message Line adjusts accordingly.

1. Select the **Polar Duplicate** tool



2. Select the Items or Faces icon in the Message Line. The Message Line reads: *Polar Duplicate: Select items [or feature faces] to polar duplicate. [Shift = Extend]*



   If the objects are already selected the Polar Duplicate dialog box appears. Skip to step 4.

3. Select the object to be duplicated. The dialog box appears.

   The Message Lines reads, *Input Set* (followed by hints for the highlighted field in the dialog box)

4. Enter the number of objects in the circular array.
5. If the *Center X** data field is not selected, click in the field.

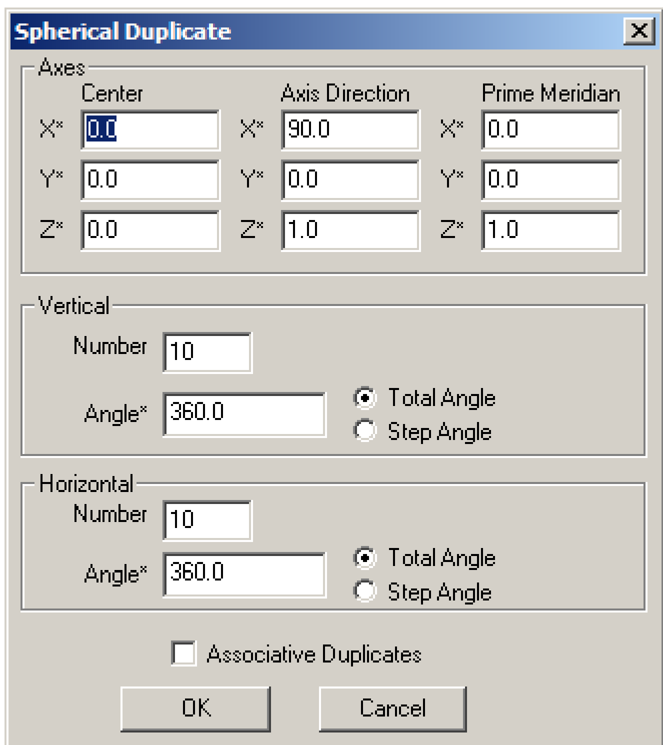
6. Move the pointer to the drawing area and click the center point of the array in the drawing area. Values automatically appear in the X*, Y* and Z* data fields.
7. To make objects to be upright in the same orientation as to the selected objects, select the *Translate Duplicates* option.
8. If Translate Duplicates is specified, 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 the drawing area along the desired angle using the Draft Assistant.
11. Click OK.

There are no Status Line entries.

Spherical Duplicate Tool

The **Spherical Duplicate** tool copies a selection of either items or faces in a sphere around a central point.

The number and location of duplicates are controlled through the Spherical Duplicate dialog box.



The dialog box contains the following options:

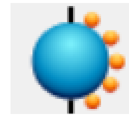
Center X*, Y*, Z* Sets the X, Y and Z location for the center of the spherical array. Enter these values or click the location in the drawing area for the center.

Axis Direction X*, Y*, Z* Sets the direction of the axis from which all the parallels will be calculated.

Prim Meridian X*,Y*, Z* Sets the principal meridian which is the beginning point of calculating of location of all the other meridians.

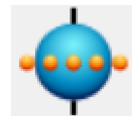
Center X*, Y*, Z*
Sets the X, Y and Z location for the center of the spherical array. Enter these values or click the location in the drawing area for the center.

Vertical Number
Sets the total number of faces or items in the plane, going through the axis of the sphere. Be sure to include the selected one in this total count.



Vertical Angle
Sets the angle between objects in the plane that goes through the axis of the sphere.

Horizontal Number
Sets the number of repetitions of the item or face in the horizontal plane along the specified axis of the sphere.



Horizontal Angle
Designates the angle by which the objects, specified in the Vertical Number and Vertical Angle, are twisted.

Total Angle
Defines the angle to which the duplication will be spread. A 360° angle spreads the features or items all along the sphere.

Step Angle
Sets the angle of the single step for which the feature/item will be duplicated.

Associative Duplicates
Creates each duplicate as an instance of the original.

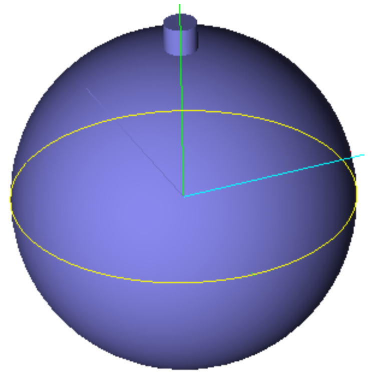
The asterisk (*) indicates that the value is automatically filled in when dragging the pointer in the drawing area to indicate the offset or spacing.

Using the Spherical Duplicate Tool

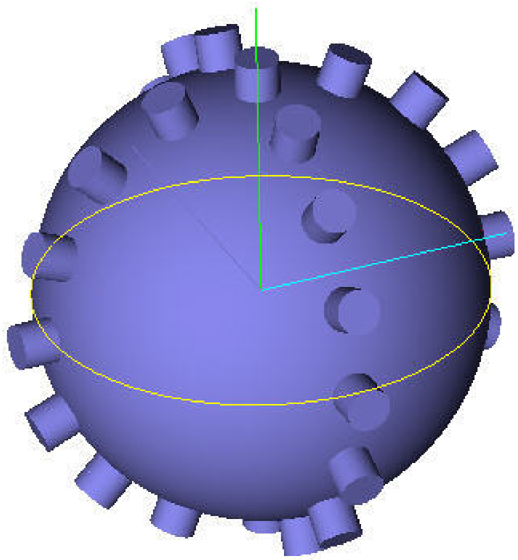
1. Select the **Spherical Duplicate** tool. 
2. Select Item or Faces from the Message Line. 
3. The Message Line reads: *Spherical Duplicate: Select Feature Faces [or Items] to spherically duplicate.*
4. Select the item or faces to duplicate. The dialog box appears.
5. Enter the values for axes location or click them with the pointer in the drawing area and the values will appear in the appropriate field.
6. Specify the number of features/items to be duplicated in vertical and horizontal directions and the angle for which the feature/item to be duplicated on the sphere's surface.
7. Click OK.

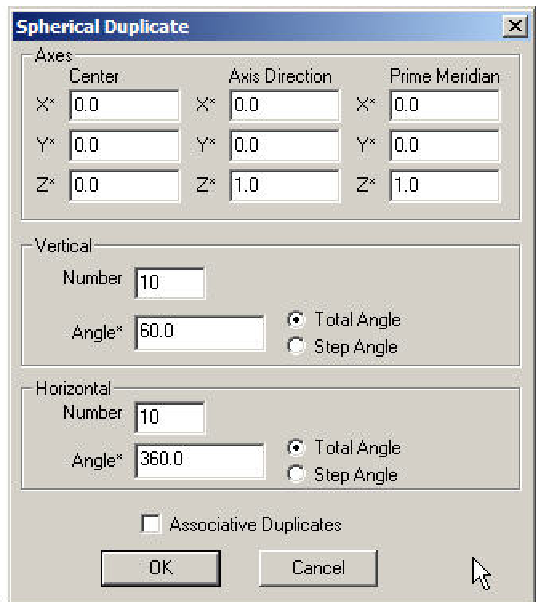
The face/item duplicates according to the specified values.

To modify the duplication, use Status Line entry fields, the Feature Tree, or double click the object to open Edit Object dialog box.

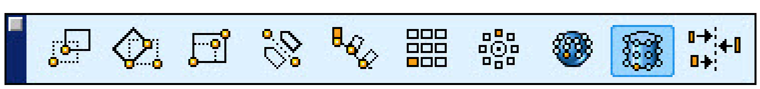


The **Spherical Duplicate** tool duplicates a face in accordance with the data set in the Spherical Duplicate dialogue box.





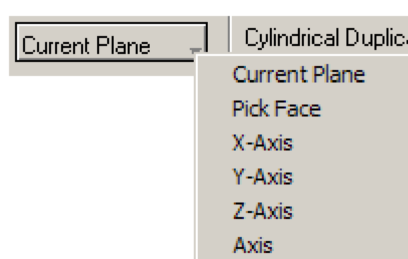
Cylindrical Duplicate Tool



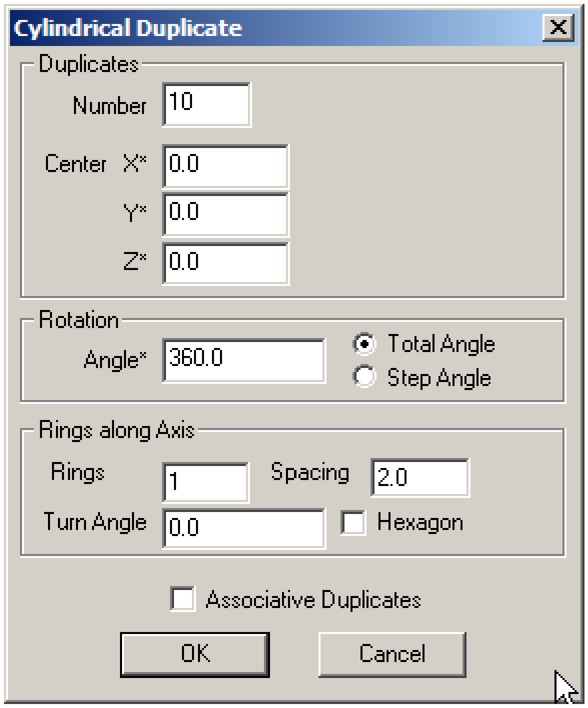
The **Transformation** tools palette now includes a **Cylindrical Duplicate** tool for items and faces. The **Cylindrical Duplicate** tool copies a selection of either items or faces in a cylinder around a central axis.

The **Cylindrical Duplicate** tool has options found in the Message Line when the tool is selected:

These options set the axis of the cylinder.



The number and location of duplicates are controlled through the Cylindrical Duplicate dialog box.



The dialog box contains the following options:

Number Sets the number of items/faces in the ring.

Center X*, Y*, Z* Sets the X, Y and Z location for the center of the cylindrical array. Enter these values or click the location in the drawing area for the center.

Rotation Angle* Designates the area used for the duplication, measured in degrees.

Rings Sets the number of circles of items/faces in the cylinder array.

Spacing Specifies the distance between the rings.

Turn Angle Designates the shift of the following ring relative to the previous around the axis.

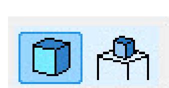
Hexagon Designates that each ring of the items/faces is located one-half twist relative to the previous one.

Associative Duplicates Creates each duplicate as an instance of the original.

The asterisk (*) indicates that the value is automatically filled in when dragging the pointer in the drawing area to indicate the offset or spacing.

Using the Cylindrical Duplicate Tool

1. Select the **Cylindrical Duplicate** tool. 

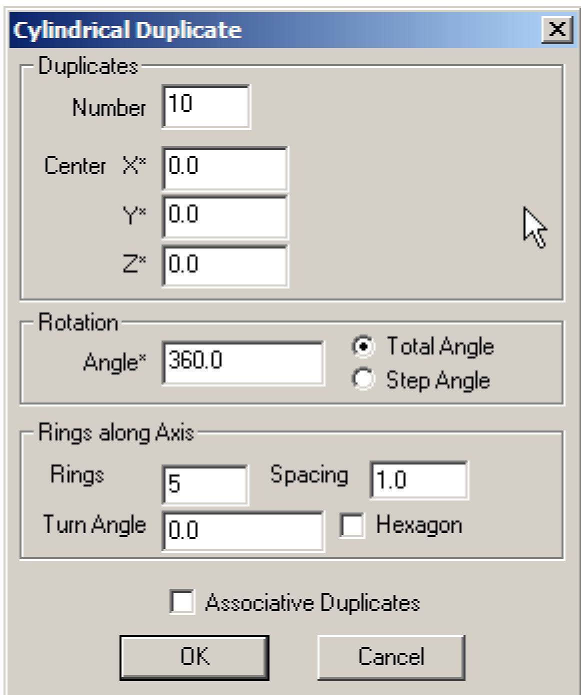
2. Select Item or Faces from the Message Line. 

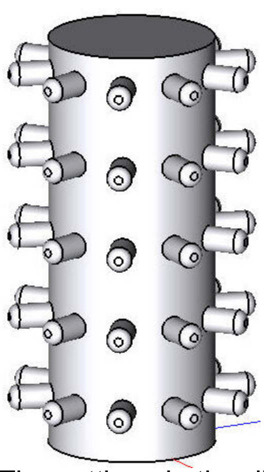
3. The Message Line reads: *Cylindrical Duplicate: Select items/[or feature faces] to cylindrically duplicate.*
4. Select the item or faces to duplicate. The dialog box appears.
5. Enter the values for the center of the cylinder duplicate and number of the items/ faces in a ring of the cylinder duplicate. It is possible to click in the drawing area to specify the center.
6. Fill in the Rotation and Rings along Axis fields as desired.
7. Click OK.

The duplication is performed reflecting the values specified in the Cylindrical Duplicate dialog box.

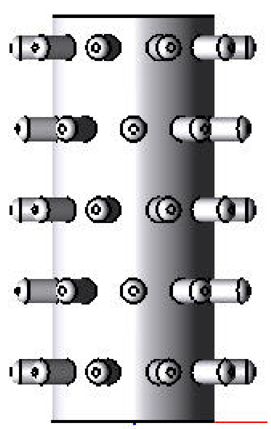
To modify the feature duplication, use the Status Line entry fields, the Feature Tree, or double click the feature to open Edit Object dialog box.

This graphic shows an object with a face to be duplicated with the **Cylinder Duplicate** tool.



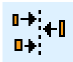


The settings in the dialog box duplicate the features as in this picture.

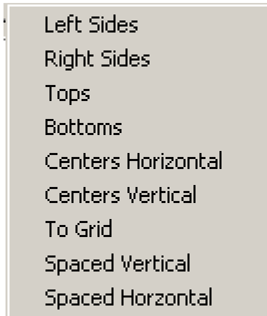


This graphic shows the same features duplicated with the Hexagon option checked.

Align Objects Tool



Cobalt, Xenon and Argon can align any object created using the **Align** tool. This includes aligning text objects with other text objects and text objects with geometry.



Using the Align Command

1. Select the **Align** tool.
2. Select the alignment type from the pull down menu in the Message Line.
3. Hold down the SHIFT key and select the text and geometry to align.
4. Click in the drawing area to pick an alignment point.

The objects align.

For more detailed information regarding the **Align** Tool see Editing Objects

Transforming Techniques

There are many ways to accomplish the same transformation task in this Designer Elements program. This section includes some of those ways.

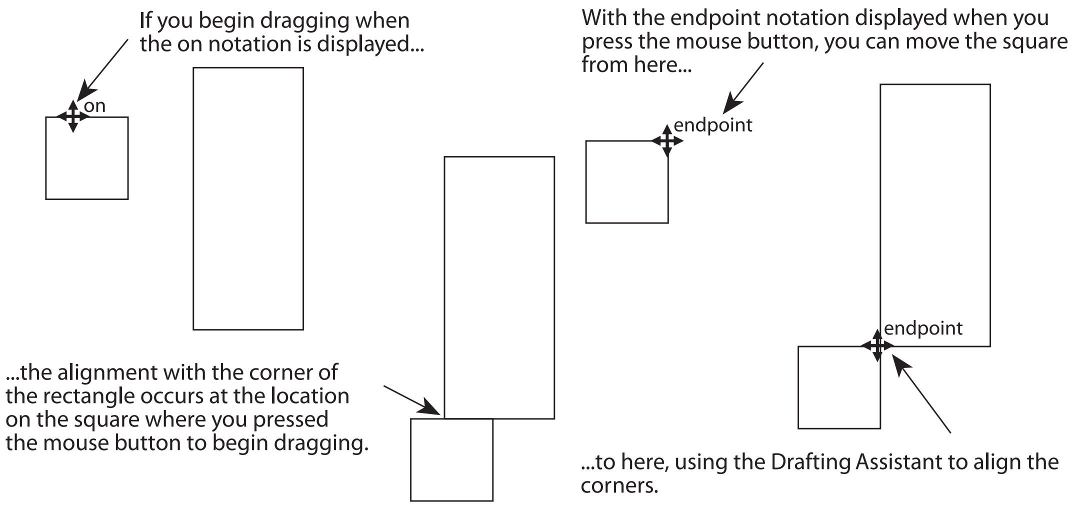
Moving Objects with Tools

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 the **Selection** tool is active.

Using the Drafting Assistant for Moving

When the pointer becomes the 4-way Move symbol, drag the object around and see the Drafting Assistant's notations relative to the object's location when pressing the mouse button. This way, if the pointer is moved over a control point so the Drafting Assistant activates it, it is possible to align the object being moved with that point.

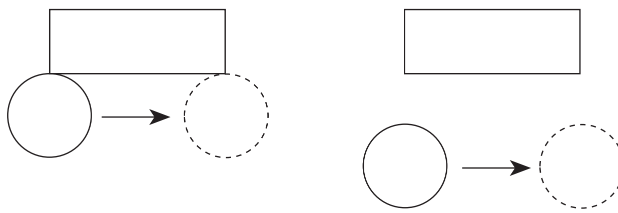


Selected Move versus Move Tool

Objects can be moved either with the **Selection** tool or with the **Move** tool.

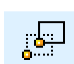
Moving with the Selection Tool

The move function of the **Selection** tool allows free movement of the selected object.



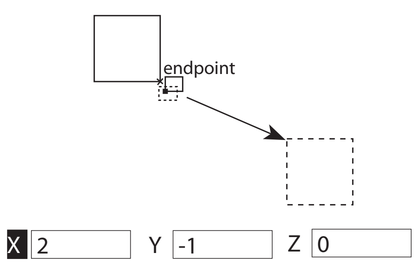
To move a circle from one location to another as shown above, use the **Selection** tool and the Drafting Assistant to align the 90° *quadrant* with the corner of the rectangle. The circles do not need to touch the rectangle to be aligned with it.

Moving with the Move Tool



The **Move** tool in the Transformation subpalette moves 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 the object will be relocated.
3. Click OK. The object is now located on the new layer.

Copying Objects with Tools

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

Hold down the CTRL (Windows) or OPTION (Macintosh) key and drag a copy of the selection to a new location.

When using the **Selection** tool to copy a surface or solid using this technique, 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***.

1. Choose the **Selection** tool.
2. Select the object(s) to copy.
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

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 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

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 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.

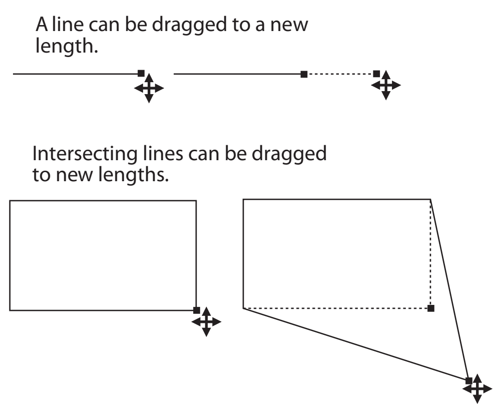
Sizing Objects with Tools

Normally an object is sized 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

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 to stretch.
4. Drag the points to a new location.

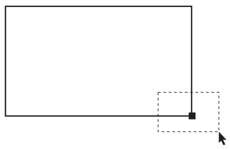


Sizing an Object with the Move Tool

The **Move** tool specifies 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:

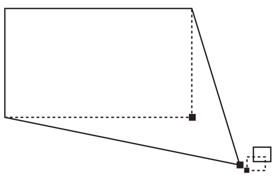
1. In the Edit menu, be sure that *Selectable Points* is set.
2. Click the **Selection** tool.
3. Drag a selection fence around the lower right corner of the rectangle.



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.

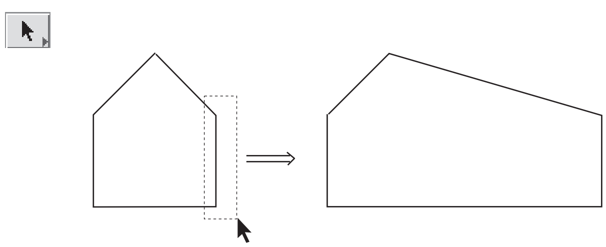
   Press the ENTER (Windows) key or the RETURN (Macintosh) key.

   The corner of the rectangle is moved the distance specified in the Status Line.

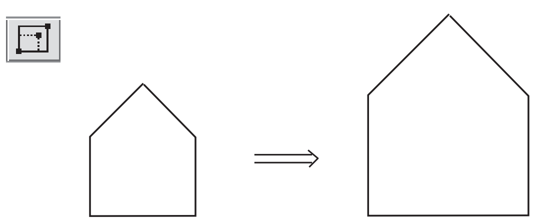


Selected Sizing versus Expand/Shrink Tool

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.



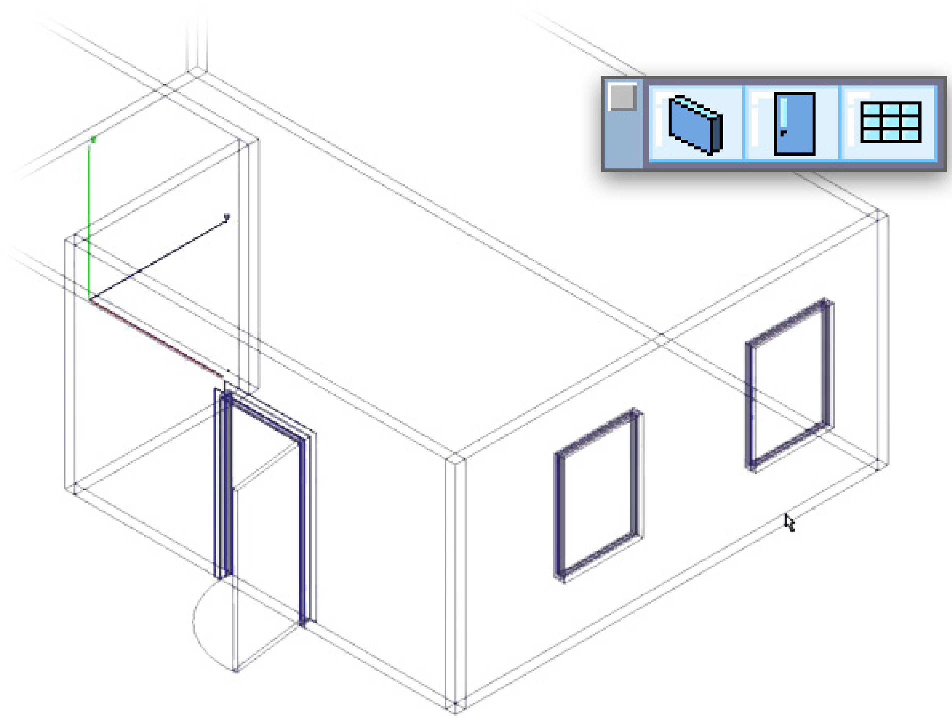
The **Resize/Scale** tool on the Transformation subpalette enables resizing geometry while maintaining its proportions.



Use the Resize/Scale tool to enlarge or shrink a shape proportionally

In addition, proportions can be specified by clicking points on other geometry. For example, resize an object to fit within another object by clicking the boundary into which the resized object must fit.

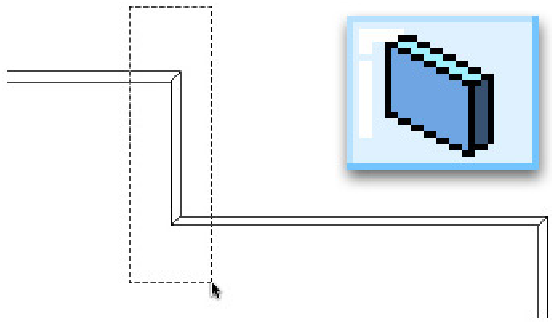
Architectural Features

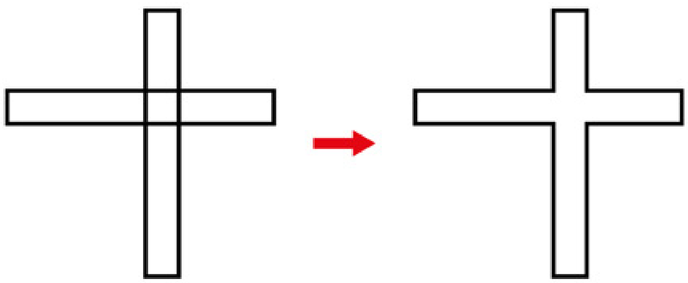


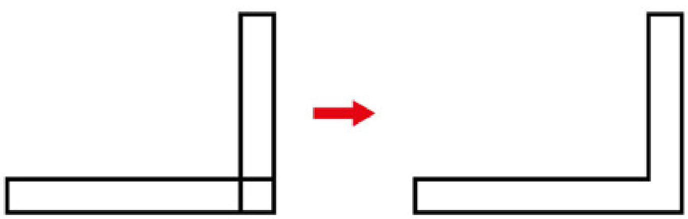
Architectural features such as walls, doors and windows are introduced further in this chapter.

Wall Tool

Similar to those found in Graphite, Smart Walls are double lines that are automatically trimmed at the intersections when the walls are on the same layer. Thickness, height, angle and length are specified in the Status Line. Optional properties include hatch pattern, hatch color, trimming, single or double lines and wall orientation such as center line, inside or outside.





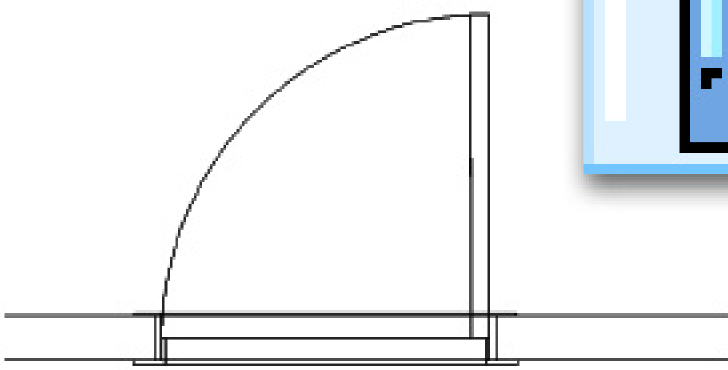


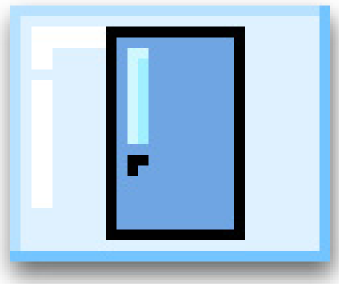
Using the Wall Tool

1. Select the Single Wall tool.
2. Choose one of the
3. The Message Line reads: *Single Wall: Pick the first wall point.*
4.

Door Tool

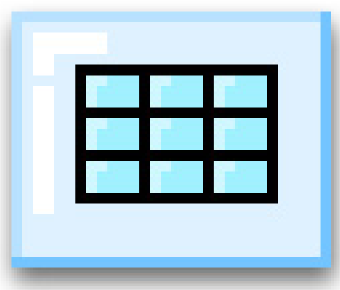
Doors are inserted in wall segments. When viewed from the top, the arc and width of the door appear. From all other views, the door appears as a 3D object. Options include single and double doors.

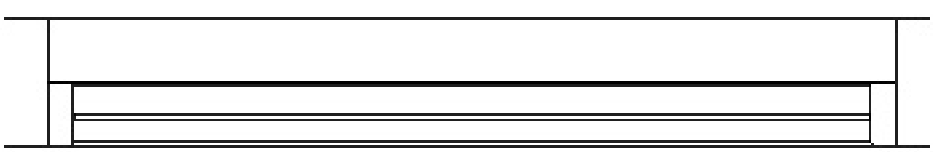




Window Tool

The Windows tools if for adding windows to a wall segment. Like the doors, the windows appear appropriately in top vs. other three-dimensional views. Windows can have overlapping, tight or standard sills.





Pen Settings

This chapter describes various options for adapting Cobalt, Xenon and Argon 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

There are three options for setting pen characteristics, pen color, weight and pattern.

It is possible to change the pen characteristics for selected objects or choose one or more of these characteristics as the default setting for a file.

Changing the Pen Characteristics of an Object

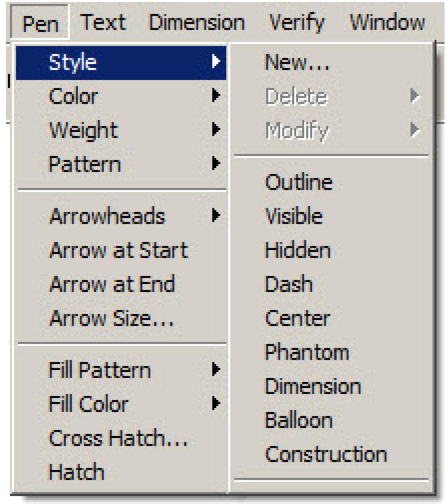
1. Select the object.
2. Choose ***Pen>Color***, ***Weight*** or ***Pattern*** and select any one for the selected object.

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

Changing the Default Pen Characteristics

To change a pen characteristic so that all future objects 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.

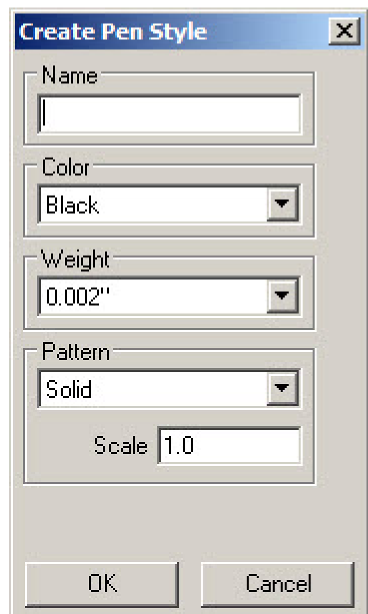
User Pen Styles are available globally and not just per drawing. New pen styles are automatically saved in a *PenStyles.ini* file so they are available for all drawings once a new style is created.

Pen Styles are dynamic. This means that after pen styles are created and assigned to a drawing, it is possible to go back and modify the pen style and have all objects using that pen style update.

The first three menu options provide tools to easily create, delete and modify user defined pen styles. The next nine pen styles are factory-set and can only be changed by hand editing the *PenStyles.ini* file.

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 to create the new style. All pen styles created are listed below the factory-set 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 user-created styles. The factory-set pen styles cannot be deleted.

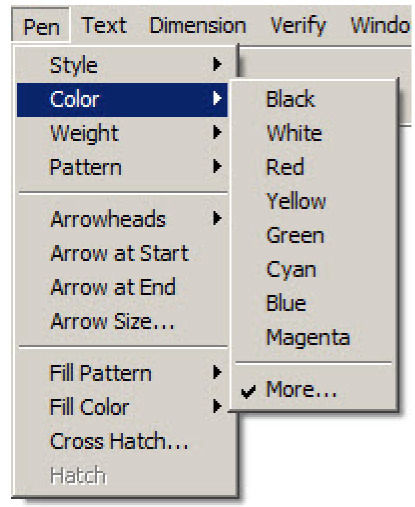
Modifying a Pen Style

To modify a pen style choose ***Pen>Style>Modify*** from the drop down menu. Change the style in the dialog box and press Apply for the 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 *More* option, which displays a color palette with more colors.



Tip: *The fill color for polygons is set in the Pen Color. See Fill and Crosshatching.*

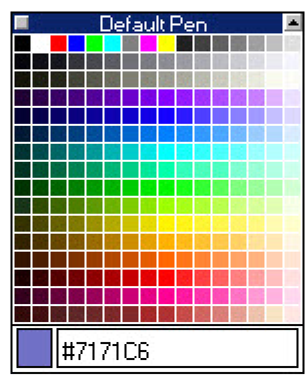
Displaying the Color Palette

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

   The Color palette appears.

2. 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.

   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 desired.

   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

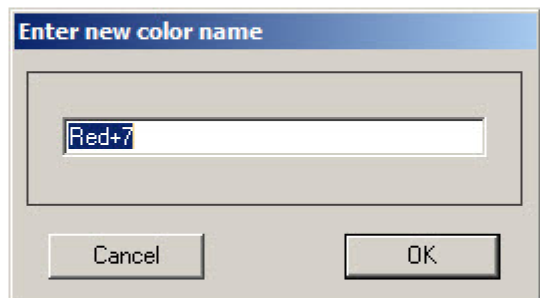
It is possible to define as many as 239 colors. When a color is defined it is defined for both the file and for the program.

Because the color displays are different for Windows and Macintosh machines, the process for defining a color varies slightly. See your Windows or Mac documentation for information on how to define custom colors on your platform.

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 color display dialog box for your platform appears. Follow the directions in your Windows or Mac operating system documentation on how to create a custom color.
3. Once the color is selected, click OK, the color display closes and returns to color palette.
4. Double-click on the color name field, where the old name currently displays. The following dialog box appears.



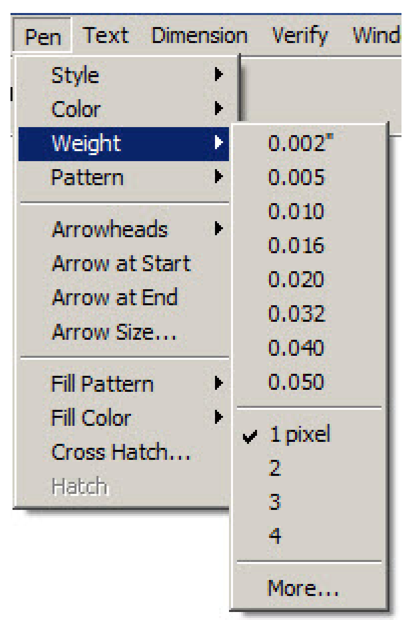
5. Enter the new name for the color and click OK.

   The color palette returnes once again with the new color name shown.
6. Repeat this process to define up to 239 colors.

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

Weight

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



It is also possible to set 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 metric units are selected, the line weight shows in millimeters instead of inches.

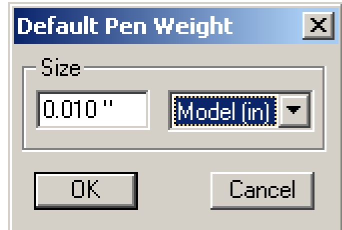
Pixels

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

More

Choose *More* and the Pen Weight dialog box appears. Enter the size and select the units of measure from the drop down menu.

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 printing or ploting such lines, they appear with the different weights.*

Tip: *Even if the preference settings are set to one type of unit, such as inches, in this dialog box the pen units can be set to a different unit of measure, such as millimeters, if desired.*

Specifying a New Weight for the Current Pen

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

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

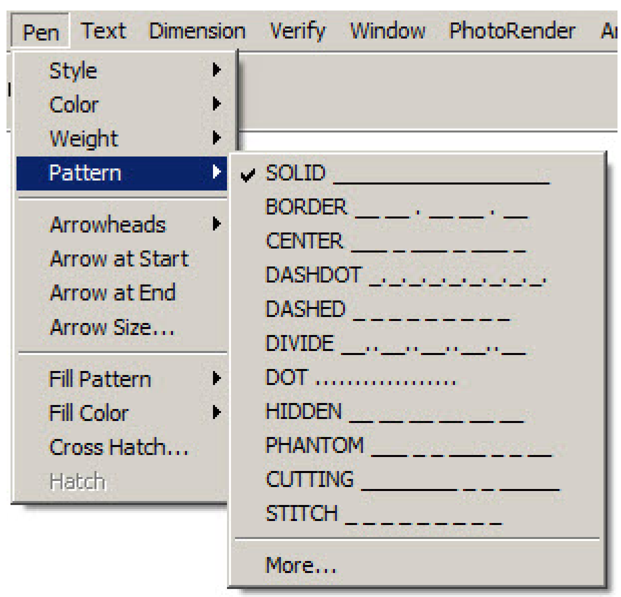
Pattern

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 printing or ploting such lines, they appear with the weights.

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

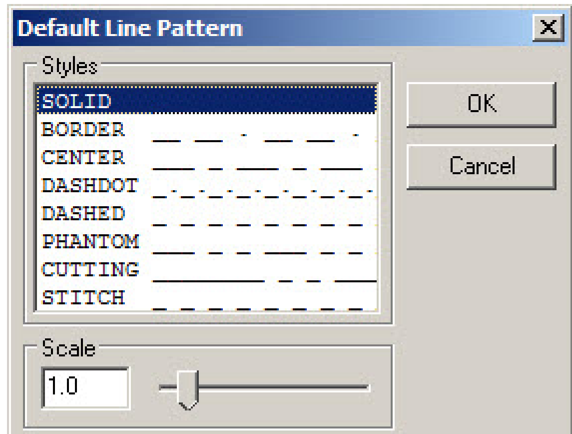
There are nine default pen patterns in the Pattern submenu of the Pen menu.

Create custom 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 ***Pen>Pattern>More*** is chosen. 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 for creating and editing geometry. It is possible to edit 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. 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
```

Add new patterns 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 in an ASCII text editor.
3. After the last line pattern, enter the name of the 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.

Tech Note: *The location of the* Cadd.lin *file for the line patterns varies with the program, the version, the platform and the version of the operating system. It can be found under User, Workgroup, System (Mac only) and/or Program. For specific information on your version try the knowledgebase in the Support Center on the website at www.ashlar.com.*

Editing Patterns

Any pattern can be edited in two ways:

- Make changes to the *Cadd.lin* file in an ASCII editor.
- Choose settings in the Line Pattern Manager.

Making Changes in the *Cadd.lin* file

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

Tip*: 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 it is necessary to return to the original factory settings, simply replace the* Cadd.lin *file with this original file.*

Tech Note: *The location of the Cadd.lin file for the line patterns varies with the program, the version, the platform and the version of the operating system. It can be found under User, Workgroup, System (Mac only) and/or Program. For specific information on your version try the knowledgebase in the Support Center on the website at www.ashlar.com*.

Choosing Settings in the Line Pattern Manager

1. Choose ***Pen>Pattern>More***. The Line Pattern Manager displays.
2. Select the pattern 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.

   If any objects are selected, they will reflect the changes. The current pen pattern will not be affected unless it was the pattern that was edited.

   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 desired pattern. The pen takes on the new pattern, selected in the submenu.

Specifying a New Pattern for a Selected Object

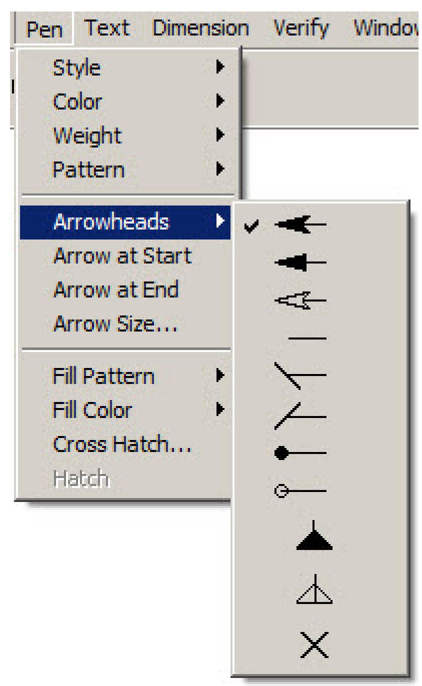
1. Select the object.
2. Choose ***Pen>Pattern***.
3. Drag to the desired pattern. The pen takes on the new pattern, selected in the submenu.

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. To create construction geometry on the construction layer, use the dotted line pattern.

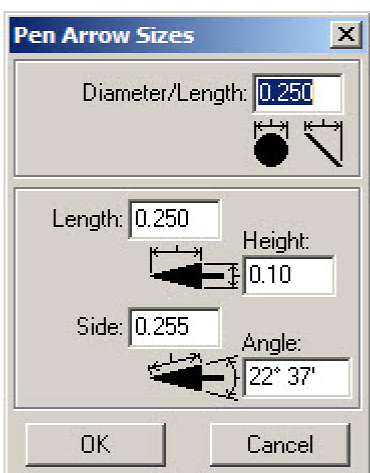
Arrowheads

To use arrowheads on lines or circular arcs that are not a part of dimensions, specify the placement of arrowheads in the Pen menu. Have an arrowhead at the beginning or end of a line or circular arc, or at both the beginning and end.



Arrowhead Types

The Arrowheads submenu offers eight arrowheads for arrow lines. Select the preferred style.



Arrow Size Command

This command in the Pen menu specifies the size of the selected arrowhead.

Diameter/Length Specifies the diameter or length of circular arrow styles and the 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 Specifies the length of the arrowhead as the horizontal distance from its tip to the furthest extension of its base.

Height Specifies the height of the arrowhead as the vertical distance of its base.

Side Specifies the length of the edge of the arrowhead.

Angle Specifies the angle of the tip of the arrowhead.

By changing any value in the *Length*, *Height*, *Side*, or *Angle* entry fields, the values are changed in the other entry fields accordingly.

Arrow At Start

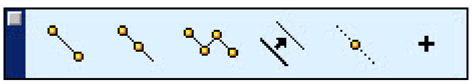
This command in the Pen menu places an arrowhead at the beginning of selected and subsequent lines and circular arcs. Choose the type of arrowhead from the Pen menu. A check indicates the current arrowhead setting. Change the default setting by saving changes in the preferences file.

Arrow At End

This command in the Pen menu places an arrowhead at the end of selected and subsequent lines and circular arcs. Choose the type of arrowhead from the Pen menu. A check indicates the current arrowhead setting. 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.

Line Tools



The Line tools in the **Line** tool palette create line segments, connected lines, lines parallel to existing lines and points. Creating 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.

Values for each tool can be entered in the Status Line to define a line after it is created. If values are entered 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.

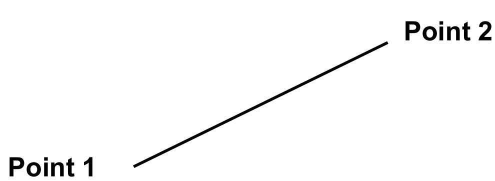
The topics explained here include

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

Cobalt, Xenon and Argon do not support Smart Wall entities found in Graphite. All imported Smart Walls are converted into individual curves.

Single Line Tool

This tool draws a line between two points. Click points to draw a line.

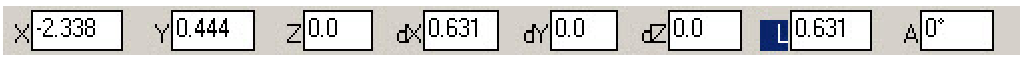


Using the Single Line Tool

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. A rubberband image appears that previews the 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 it is 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 (*end-point*, *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.

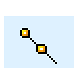
Drawing a Line Tangent or Perpendicular to an Arc, Circle or Ellipse

1. Construct an arc, circle, spline or ellipse.
2. Choose the **Single Line** tool.
3. Move the pointer to the arc until a Drafting Assistant *on* notation appears (*endpoint*, *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.

Mid-point Line Tool



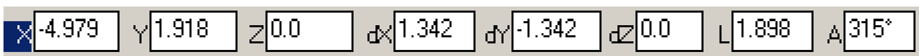
This tool creates a line object by specifying the mid point and end point.

Using the Mid-point Line Tool

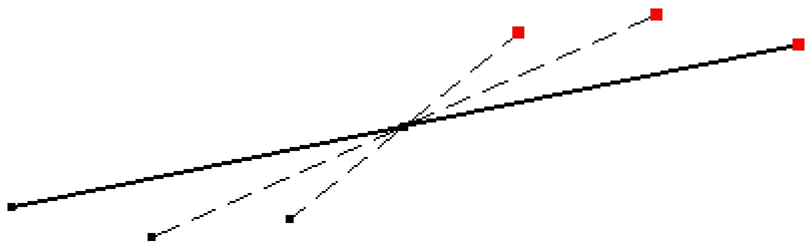
1. Select the **Mid-point Line** tool from **Line** tool palette. The Message Line reads: *Mid-point Line: Pick center point [Ctrl = Copy previous]* (Windows) *or [Option = Copy Previous]* (Macintosh).
2. Pick the center point. The Message Line reads: *Mid-point Line: Pick end point*.
3. Pick the end point of the line.

   To copy this just-created line, hold down the CTRL (Windows) or the OPTION (Macintosh) key and click once in the drawing area to set the starting point. An identical line appears beginning at the point clicked 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. Once a line is drawn, the Length field is active in the Status Line. Specify the end point for the region to stretch.



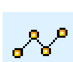
Mid-point Line object has two end control points and a mid control point. Move the end control point to change line length or/and angle from the horizontal.

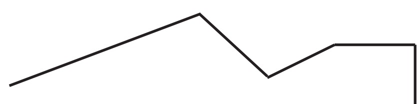


Geometric Characteristics

A mid point line is created by clicking the center point and end point 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

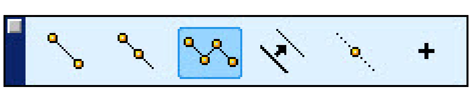




This tool draws lines in which the endpoint of one line segment is the beginning point of the next.

Using the Connected Lines Tool

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

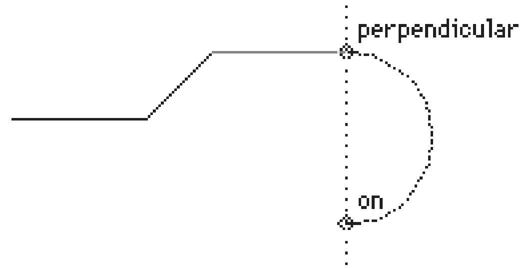


2. Click to indicate the endpoints of the line segments.

   Press the ESC key, or choose *Undo* to remove the last segment if a point within the process is clicked by mistake.

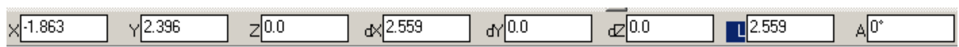
   After the beginning point is set, the Message Line changes to reflect the next step.
3. Indicate the last point by double-clicking, hitting the ESC key or by choosing another tool.

After completing at least one segment with the **Connected Lines** tool, 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.

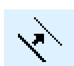
The Status Line contains the X, Y and Z coordinates of the beginning, the relative location of the end (delta X, delta Y and delta Z), the line length and the angle from horizontal. Once a line segment is drawn, the Length field is active in the Status Line.

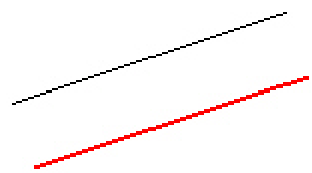


Geometric Characteristics

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

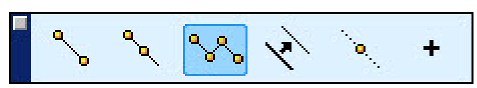




This tool constructs a line parallel to an existing line in the X or Y axis of the current work plane.

Using the Parallel Line Tool

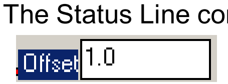
1. Select the tool. The Message Line reads: *Parallel Line: Drag new line off existing line*.



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

   If the value is typed in and ENTER (Windows) or RETURN (Macintosh) is pressed, the parallel line moves the offset distance specified.

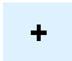
   The Status Line contains the Offset data field.



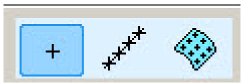
Geometric Characteristics

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.

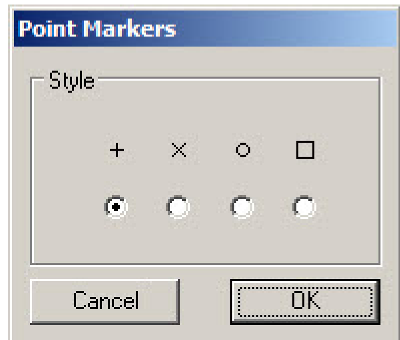
Point Tool



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

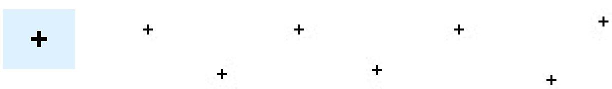


Choose the appearance of the points using any of these tools. Press the CTRL (Windows) or the OPTION (Macintosh) key to display the Point Markers dialog box.



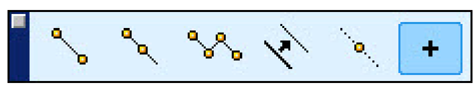
Click the radio button for the desired marker (crosshairs plus, crosshairs x, empty diamond or empty square) and click OK to close the dialog box. When clicking to place the point in the drawing area, the point is represented by the new marker. The Point Marker style cannot be saved as a preference.

Using the Point at an XYZ Location Tool



Create individual points in the drawing area with this tool.

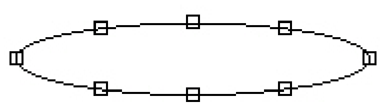
1. Select the **Point** tool from the palette.



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

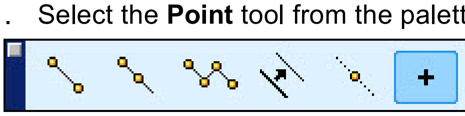
The Status Line contains the X, Y and Z location of the point.

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). This tool cannot be used on polygons created using one of the **Polygon** tools.

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



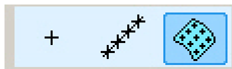
2. Select the **Point on a Curve** tool in the Message Line. The Message Line reads: *Point on Curve: Select curves to distribute points on. [Ctrl = Markers* (Windows) or *Option = Markers* (Macintosh), *Shift = Extend]*.
3. Enter the number of points that need to be distributed along the curve in the #Pts data field of the Status Line.

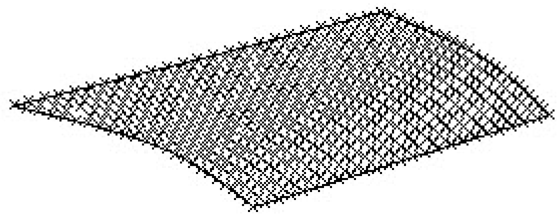
4. Select the curve. To select more than one curve, hold down the SHIFT key before selecting the first curve and while selecting succeeding curves.

   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.

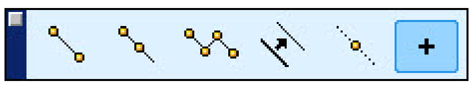
Using the Points on a Surface Tool





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

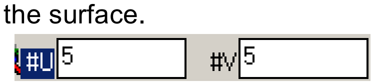
1. Select the **Point** tool from the palette.



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 = Markers* (Windows) or *Option = Markers* (Macintosh), *Shift = Extend].*
3. Enter the number of points to distribute 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.   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

A line can be modified in a number of ways. See Editing Objects for information on the Editing tools. After creating a line, change any value including length if the line is still selected.

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

Arc & Circle Tools

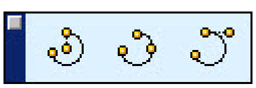
This chapter contains information on the Arc and Circle tools available in the Designer Elements program. Creating an arc or circle, the coordinate locations and radius/ diameter appear in the Status Line. The arc or circle is also drawn with the current pen specifications for color, weight and pattern.

To define an object, after it is created, enter values in the Status Line for each tool. If values are entered 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:

- **Arc Tools**
- **Circle Tools**

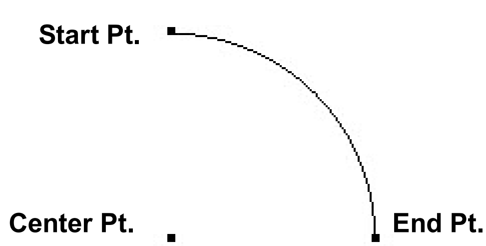
Arc Tools



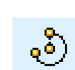
The Designer Elements program features three arc tools: **Center-point Arc**, **3-point Arc** and **Tangent-point Arc**.

Center-point Arc Tool

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



Using the Center-point Arc Tool

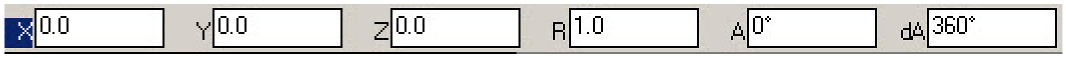


1. Select the tool. The Message Line reads: *Center-point Arc: Pick Center*.
2. Click the center point of the arc. The Message Line now indicates the next step in using the tool.
3. Click the beginning point of the arc which will also define the radius.

4. Moving the cursor to place the final point, a rubberband arc appears. Click the final point.

To construct an arc greater than 180°, 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.



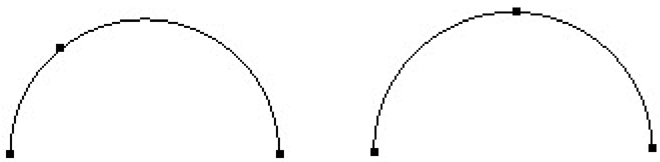
Geometric Characteristics

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.

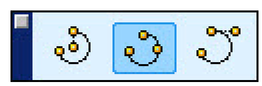
The Start and End Angles refer to the angles tangent to the arc point relative to the work plane.

3-point Arc Tool

This tool draws an arc through three specified points.



Using the 3-point Arc Tool

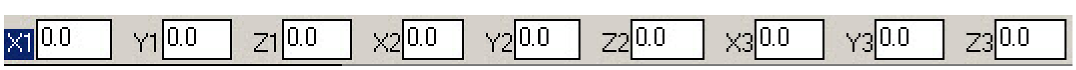


1. Select the tool. The Message Line reads: *3-point Arc: Pick first point. [Ctrl = Tangent to object* (Windows) or *Option = Tangent to object* (Macintosh)*]*.
2. Click the first endpoint of the arc.

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

The arc is drawn from the first indicated position, 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 some other object is clicked, the program defines 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.

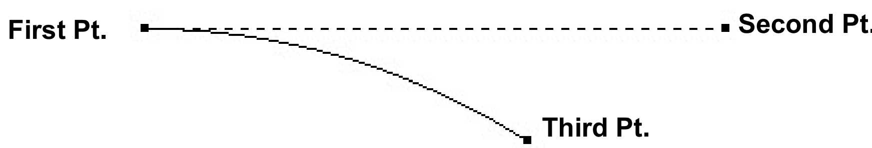


Geometric Characteristics

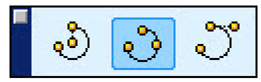
A 3-point arc is created by placing three points and are made up of the following characteristics: Diameter, Start Angle, End Angle, Start (X, Y and Z values) and End (X, Y and Z values). This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select 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 specified point. The second specified point is the direction vector and the third pecified point 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.



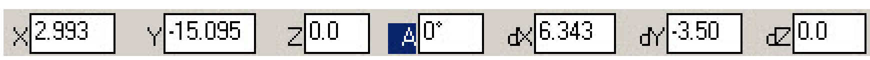
Using the Tangent-Point Arc Tool



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 clicked points, tangent to the line between the first and second points.

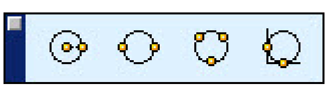
The Status Line contains the X, Y and Z coordinates of the endpoints of the arc and the angle of the tangent line.



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



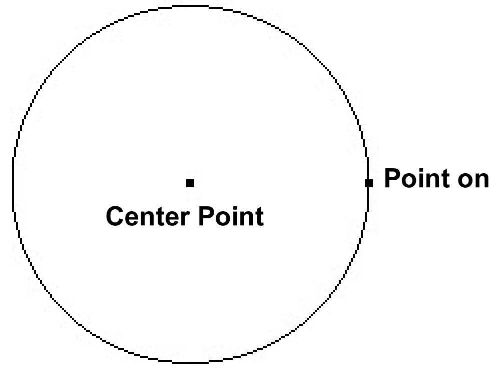
The Circle tools include the **Center-point Circle**, **Opposite-Point Circle**, **3-point Circle** and **Tangent Circle**.

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.

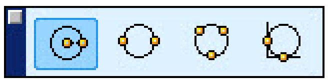
Center-point Circle Tool

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



Using the Center-Point Circle Tool

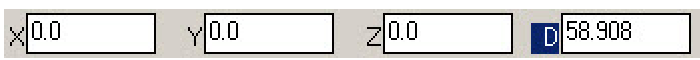
1. Select the tool. The Message Line reads: *Center-point Circle: Pick center. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*]*.



2. Click the center of the circle. A rubberband circle appears guiding the construction as the cursor is moved.
3. Click the second point which determines the circle's radius.

To create a copy of the last circle hold down the CTRL (Windows) or the OPTION (Macintosh) key and click once in the drawing area to set the center point. An identical circle appears centered at the point clicked on the current work layer.

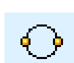
The Status Line contains the X, Y and Z coordinates of the center and the diameter of the circle. Diameter is the active status field.



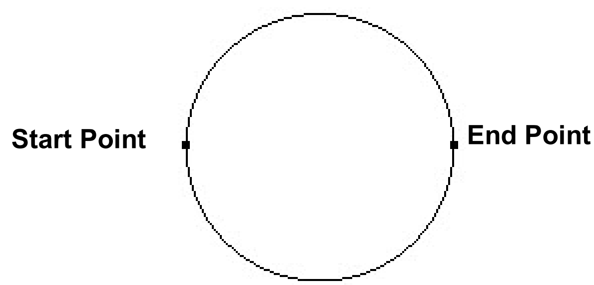
Geometric Characteristics

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.

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 the first point is clicked, a rubber-band circle appears to guide in the construction.

To create a copy of the last circle hold down the CTRL (Windows) or the OPTION (Macintosh) key and click once to place the center point of the new circle. An identical circle appears centered at that point on the current work layer.

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

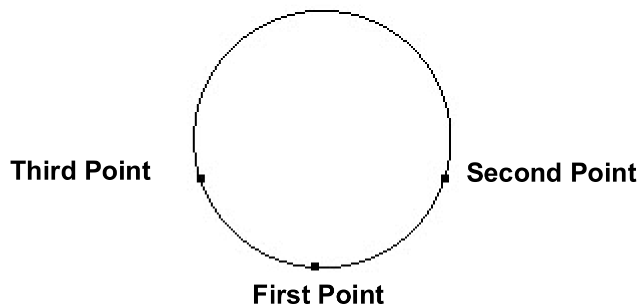
> ***Tip:*** *It is possible to select the center point of an opposite point circle if its control points are displayed using the* Show Points *command in the Layout menu.*

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 selected points.



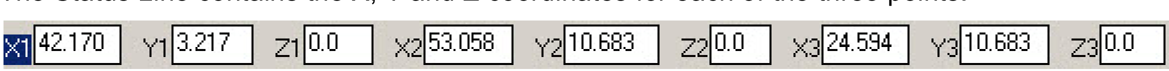
Using the 3-point Circle Tool

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 the construction.

Placing any of the three points on an existing object, the circle is drawn through that point. If an object is clicked while holding down the CTRL (Windows) or OPTION (Macintosh) key, the circle is drawn tangent to the object rather than through the indicated point. 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.

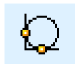


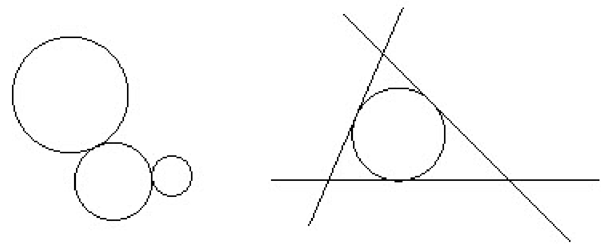
Geometric Characteristics

A three-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 Circle Tool





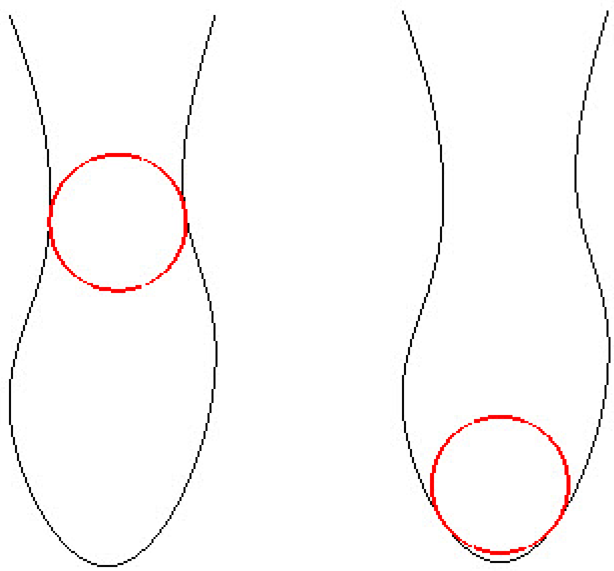
This tool draws a circle tangent to two objects.

Using the Tangent Circle Tool

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

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

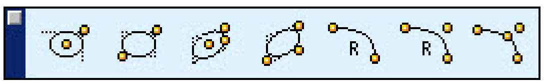
Tap the CTRL (Windows) or OPTION (Macintosh) key to flip the circle to any alternative positions possible for placement tangent to the two objects.



Geometric Characteristics

A tangent 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. While creating an ellipse, the coordinate locations, lengths of the control rectangle and angles appear in the Status Line. While creating 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.

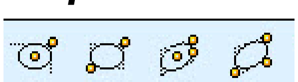
The values for each tool can be entered in the Status Line to define an object, after the object is created. If values are entered in the selected Status Line data field after creating the object and while the object is still selected, pressing ENTER (Windows) or RETURN (Macintosh) updates the object to reflect the new values.

The tools explained here include:

- **2-point Center Ellipse Tool**
- **Opposite-corner Ellipse Tool**
- **3-point Center Ellipse Tool**
- **3-corner Ellipse Tool**
- Modifying Ellipses
- **2-point Conic Tool**
- **3-point Conic Tool**
- **4-point Conic Tool**

> ***Refferal:*** *The rho value definition can be found in Conic Tools and 2-point Conic Tool.*

Ellipse Tools



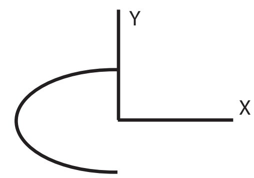
Create ellipses with four tools: **2-point Center**, **Opposite-corner**, **3-point Center** and **3-corner**. These tools in the main tool palette construct ellipses inscribed within an invisible rectangle or parallelogram. The **2-point Center Ellipse** tool uses the center point and one corner location 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.

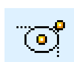
> ***Tip**: Graphite users: Ellipses are enhanced in this program and are defined by centers.*

The graphic here shows an ellipse with a start and end angle of 90° and 180°.

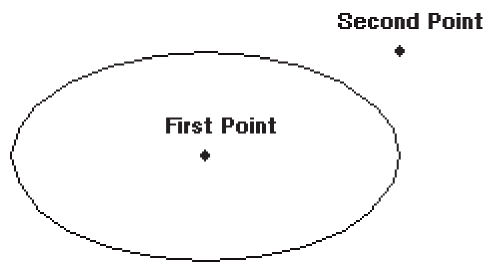
The ellipse is drawn with the current pen specifications for color, weight and pattern.



2-point Center Ellipse Tool

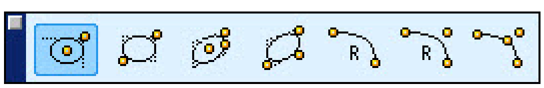


This tool constructs an ellipse inscribed within a rectangle defined by two specified points: the center point and one corner of the rectangle.



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. Click to establish the center of the ellipse.

   Notice that the Message Line displays the next step for using the tool.
3. Click to establish the corner of the rectangle defining the ellipse.

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

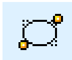
Copy the last ellipse by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking where the center will be. The copy is placed on the current work layer.

In the Status Line, 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.

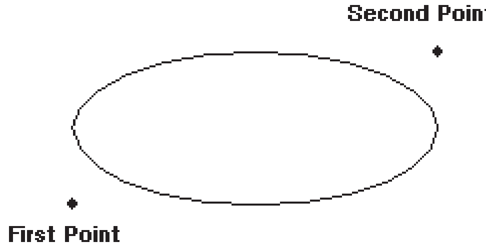
Geometric Characteristics

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

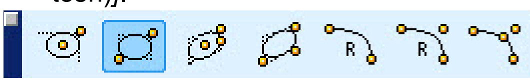


This tool draws an ellipse inscribed in a rectangle specified by opposite corners.



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. Click to establish one corner of the rectangle defining the ellipse.

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

3. Click to establish the opposite corner of the defining rectangle.

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

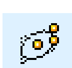
Copy the last ellipse by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking to place the lower-left point. The copy is placed on the current work layer.

In the Status Line, 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.

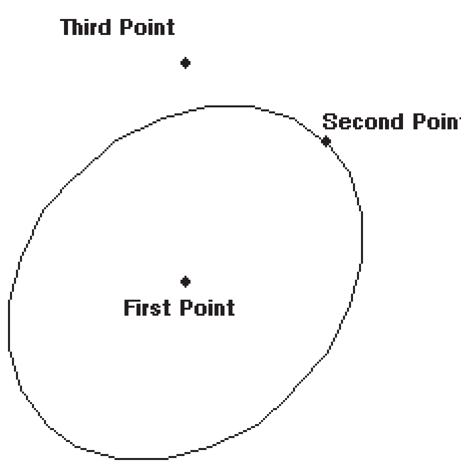
Geometric Characteristics

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.

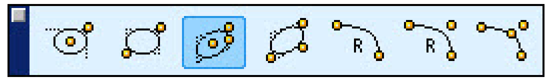
3-point Center Ellipse Tool



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 guides each successive step.
2. Click to establish the center of the ellipse.
3. Click to establish the midpoint of the side of the control parallelogram defining the ellipse.
4. Click to establish the corner of the control parallelogram defining the ellipse.

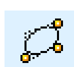
If the three points are on the vertical or horizontal axis, a straight line is drawn. Copy the last ellipse by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking where the center will be. The copy is placed on the current work layer.

In the Status Line, specify the X, Y and Z coordinates of the center point and the length and angle of the sides of the parallelogram.

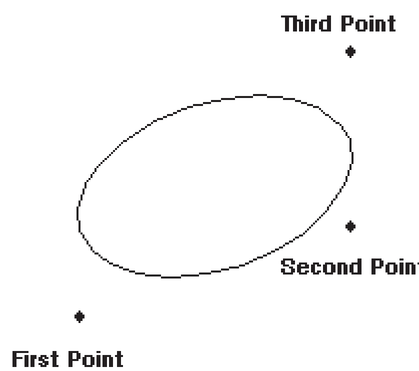
Geometric Characteristics

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.

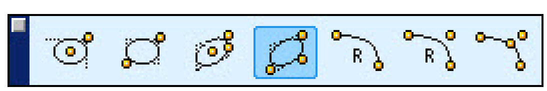
3-corner Ellipse Tool



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 guides each successive step.
2. Click to establish one corner of the parallelogram defining the ellipse.
3. Click to establish another corner of the defining parallelogram.
4. Click to establish the final corner of the defining parallelogram.

If the three points are on the vertical or horizontal axis, a straight line is drawn. Copy 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 specifies the X, Y and Z coordinates of a corner and the length and angle of the sides of the parallelogram.

Geometric Characteristics

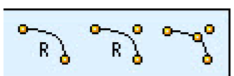
A 3-corner ellipse is created by placing the three points that represent the corners of the control rectangle and is made up of the following characteristics: Major Diameter, Minor Diameter, Start Angle, End Angle and Center (X, Y and Z values). This information is listed in the Edit Objects dialog box 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

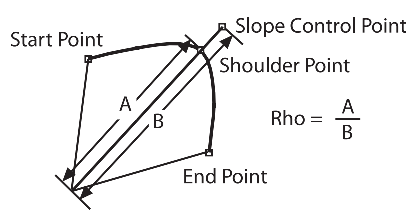
Modify any ellipse in a number of ways:

- After it's created and while it's still selected, by entering new values in the Status Line.
- By using the **Transformation** tools (see Transforming Geometry).
- By using Edit Objects.
- By selecting 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

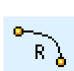


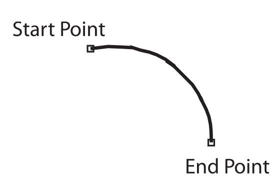
Create conics with three tools: **2-point Conic, 3-point Conic** and **4-point Conic**. The **2-point Conic** tool uses the start point and end point to define the conic. The **3-point Conic** tool uses the start point, end point and slope control point to define the conic. The **4-point Conic** tool 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 placing all but the last point. This shows the ellipse before it is drawn. The conic is drawn with the current pen specifications for color, weight and pattern.

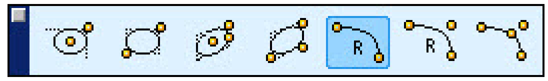
2-point Conic Tool





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 guides each successive step.
2. Pick the start point of the conic. As the cursor is moved to place the second point, a rubberband image appears.
3. Pick the end point to complete the conic.

   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.

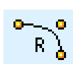
Create a copy of the last conic by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking to indicate the start point. The copy is placed on the current work layer.

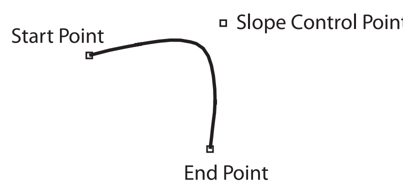
The Status Line specifies the rho value of the conic.

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

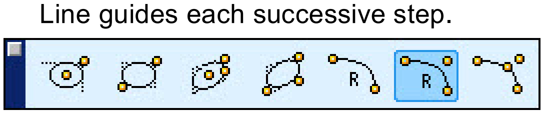




This tool constructs a conic calculated from three specified points: the start point, end point and slope control point.

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 guides each successive step.



2. Pick the start point of the conic.
3. Pick the end point of the conic. Once this third point is placed, a rubberband image appears.
4. Pick the shoulder point to complete the conic.

   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.

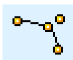
Create a copy of the last conic by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking to indicate the start point. The copy is placed on the current work layer.

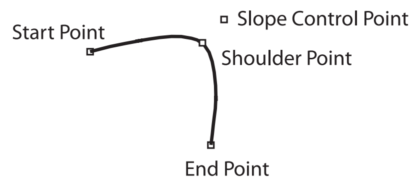
The Status Line specifies the rho value of the conic.

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.

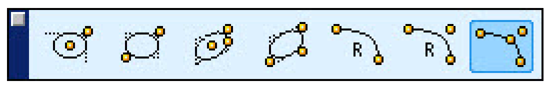
4-point Conic Tool





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

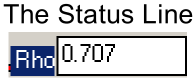


1. Select the tool. The Message Line reads: *4-point Conic: Pick start point. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*]*. The Message Line guides each successive step.
2. Pick the start point of the conic.
3. Pick the end point of the conic.
4. Pick the slope control point of the conic. Once this third point is placed, a rubber-band image appears.
5. Pick the shoulder point to complete the conic.

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 creates a straight line. A rho value of .999 creates a conic with a 90° angle.

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.

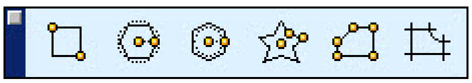
The Status Line specifies the rho value of the conic.



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.

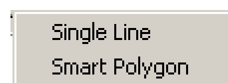
Polygon Tools



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.

After the polygon is created, and while it is still selected, it is possible to refine its size and shape by entering values in the Status Line. Press ENTER (Windows) or RETURN (Macintosh) the polygon updates 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.



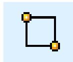
Polygons created with the *Single Line* option, are composed of individual lines that can be modified independently. These polygons can be filleted/chamfered, extruded, revolved, offset or relimited.

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, it is possible to 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 “Change Object Type” on page 15. for more information.

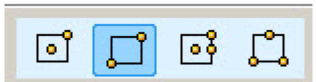
The tools explained here include:

- **Rectangle Tool**
- **Inscribed Polygon Tool**
- **Circumscribed Polygon Tool**
- **Star Polygon Tool**
- **Arbitrary Polygon Tool**
- **Polygon from Curves Tool**

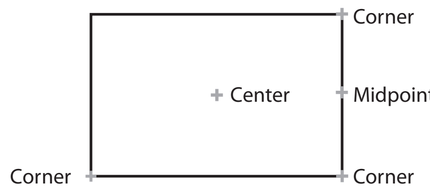
Rectangle Tool



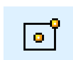
When the **Rectangle** tool is selected, a subpalette appears in the Message Line containing four tools for creating rectangles: by center and corner point, by diagonals, by center and axis, and by corner and axis.



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 specified.

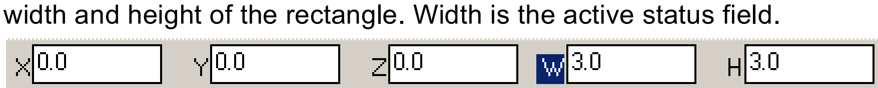
Using the Center/Corner Rectangle Tool

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 the cursor is moved, a rubberband image appears.
4. Pick rectangle corner.

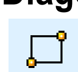
With the rectangle still selected, the width and height of the rectangle can be changed by entering the values in the Status Line fields and pressing ENTER (Windows) or RETURN (Macintosh). The width and the height of a rectangle can be also changed in the Edit Objects dialog box.

Create a copy of the last rectangle by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking where the upper-left corner will be.

The Status Line specifies 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 the cursor is moved a rubberband image appears.
4. Click the opposite corner of the rectangle.

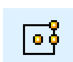
With the rectangle still selected, the width and height of the rectangle can be changed by entering the values in the Status Line fields and pressing ENTER (Windows) or RETURN (Macintosh). The width and the height of a rectangle can be also changed in the Edit Objects dialog box.

A copy of the last rectangle can be created by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking where the upper-left corner will be.

The Status Line specifies the X, Y and Z coordinates of the first point, and the rectangle's width and height. Width is the active field.

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. To draw a square from the center rather than opposite corners, use one of the other polygon tools, specifying four sides.

Center/Axis Rectangle Tool



This tool draws a rectangle using the center, midpoint and corner.

Using the Center/Axis Rectangle Tool

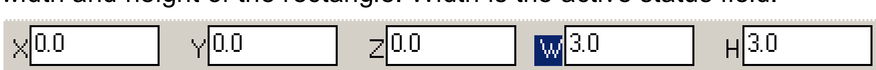
1. Select the **Rectangle** tool from the palette.
2. Select the **Center/Axis** tool in the Message Line. The Message Line reads: *Rectangle: Pick center point of rectangle. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*]*.
3. Click the center point of the rectangle.
4. Click the midpoint of the rectangle side. As the cursor is moved a rubberband image appears.
5. Click the opposite side of the rectangle.

With the rectangle still selected, the width and height of the rectangle can be changed by entering the values in the Status Line fields and pressing ENTER (Windows) or RETURN (Macintosh).

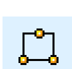
The width and the height of a rectangle can be also changed in the Edit Objects dialog box.

To create a copy of the last rectangle, hold down the CTRL (Windows) or the OPTION (Macintosh) key and click where the upper-left corner will be.

The Status Line specifies 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.



Corner/Axis Rectangle Tool



This tool draws a rectangle using the specified corners.

Using the Corner/Axis Rectangle Tool

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

2. Select the **Corner/Axis 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 the first corner of the rectangle.
4. Click the second corner of the rectangle. As the cursor is moved, a rubberband image appears.
5. Click the opposite point.

With the rectangle still selected, the width and height of the rectangle can be changed by entering the values in the Status Line fields and pressing ENTER (Windows) or RETURN (Macintosh).

The width and the height of a rectangle can also be changed in the in Edit Objects dialog box.

To create a copy of the last rectangle hold down the CTRL (Windows) or the OPTION (Macintosh) key and click where the upper-left corner will be.

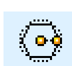
The Status Line specifies the X, Y and Z coordinates of the first point, as well as the width and height of the rectangle. Width is the active status field.

Geometric Characteristics

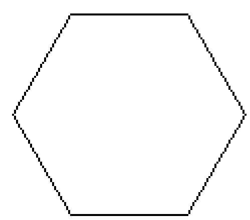
A rectangle polygon is created by picking the appropriate points as directed by the selected tool in the rectangle subpalette. A rectangle is made up of the following characteristics according to the Edit Objects dialog box: Length and Width and includes the option to Show Frame and Show Fill.

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, the geometry will become invisible when Apply is clicked.) 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

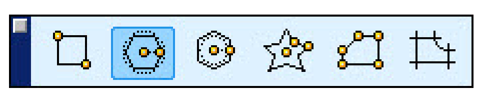


This tool creates a polygon where the radius of the circumscribing circle determines the location of the polygon's vertices.



The default polygon is a hexagon, but the number of sides can be specified 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 the cursor is moved, a rubberband polygon appears guiding the construction.

3. Pick a point on the circumference of the circumscribing circle to complete the polygon.

To create a copy of the last inscribed polygon hold down the CTRL (Windows) or OPTION (Macintosh) key and click where the center will be.

The Status Line specifies 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.

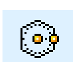
Geometric Characteristics

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.

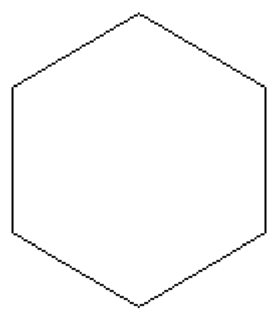
Scribe Type refers to circle type used to define the polygon. Choosing Circle Outside places the polygon within the circle, defining an inscribed polygon. Choosing Circle Inside places the polygon on the outside of the circle, defining a circumscribed polygon.

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, the geometry will become invisible when Apply is clicked.) 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 the number of sides can be specified in the Status Line. Notice that the Status Line shows the diameter for a circle, the standard way of describing a polygon circumscribed around a circle.



Using the Circumscribed Polygon Tool

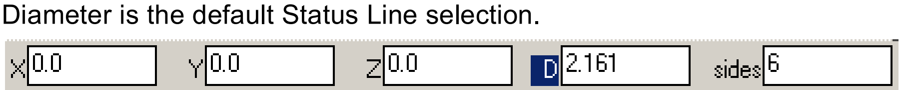
1. Select the tool. The Message Line reads: *Circumscribed Polygon: Pick center of polygon. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*]*.

2. Click the center of the polygon and the midpoint of one of the sides, as directed by the Message Line.

   After the first point is placed, a rubberband polygon appears guiding construction.

To create a copy of the last circumscribed polygon hold down the CTRL (Windows) or OPTION (Macintosh) key and click where the center will be.

The Status Line specifies 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

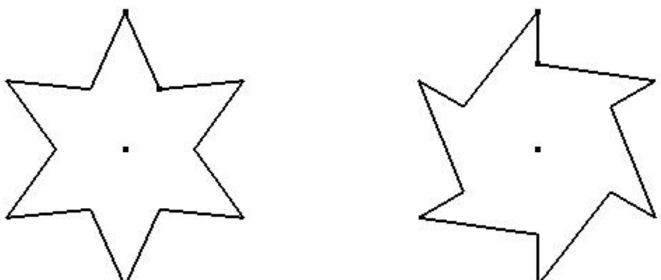
A circumscribed polygon is created by picking the center and the midpoint of the polygon side. It is made up of the following characteristics according to the Edit Objects dialog box: Diameter, Sides, Scribe Type and includes the option to Show Fill and Show Frame.

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, the geometry will become invisible when Apply is clicked.) 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.

Star Polygon Tool

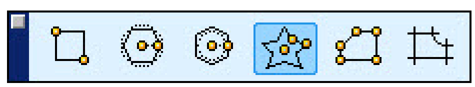
The **Star Polygon** tool creates and object with three control points including center, inside and outside vertexes.



Symmetric Star **Non-symmetric Star**

Using the Star Polygon Tool

1. Select the **Star Polygon** tool from **Polygon Tools** palette. The Message Line reads: *Star Polygon: Pick center of star polygon. [Ctrl = Copy previous]* (Windows) or *[Option = Copy Previous]* (Macintosh).



2. Pick the center point. The Message Line reads: *Star Polygon: Pick outside vertex of star polygon.*
3. Pick the outside vertex. The Message Line reads: *Star Polygon: Pick inside vertex of star polygon. [Shift = Deform star].*
4. Pick the inside vertex. Press the SHIFT key to create a non-symmetric star polygon.
5. The Status Line contains the Vertices field. Enter the desired number for the star polygon.

The Status Line contains the X, Y, and Z coordinates of the star center, inside and outside diameters of the star (D1 and D2), star vertices number.

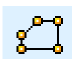
Geometric Characteristics

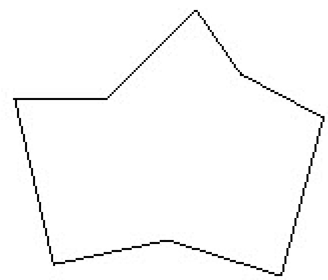
A star polygon is created by placing the center, picking the outside and inside vertexes. It is made up of the following characteristics according to the Edit Objects dialog box: Show Frame, Show Fill, Diameter 1, Dameter 2, Vertices.

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, the geometry will become invisible when Apply is clicked.) Setting Show Fill to Yes, places a fill within the polygon, based on the default pattern and fill currently selected in the Pen menu.

Diameter 1 represents the value of the diameter of an imaginary circle inscribed within the inside vertexes. Diameter 2 represents the value of the diameter of an imaginary circle circumscribed around the outside vertixes of the star poligon. The vertexes option sets the number of points out of the star poligon. 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.

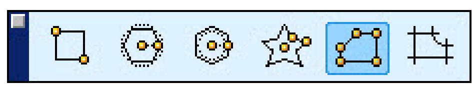
Arbitrary Polygon Tool





This tool draws a polygon with the number of sides determined by the chosen points.

Using the Arbitrary Polygon Tool

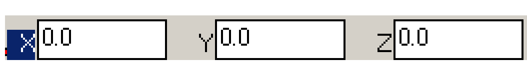


1. Select the tool. The Message Line reads: *Arbitrary Polygon: Pick first point. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*]*.
2. Click the next point as directed by the Message Line.
3. Continue clicking points to complete the desired polygon. Single click the last point to close the polygon at its starting point.

   Double-click to place the last point that is not the starting point of the polygon, and the program closes the polygon automatically.

To create a copy of the polygon hold down the CTRL (Windows) or OPTION (Macintosh) key and click where the center will be.

The Status Line specifies the X, Y and Z coordinates of the first point. X is the default Status Line selection.



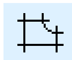
Geometric Characteristics

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, the geometry will become invisible when Apply is clicked.) 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.

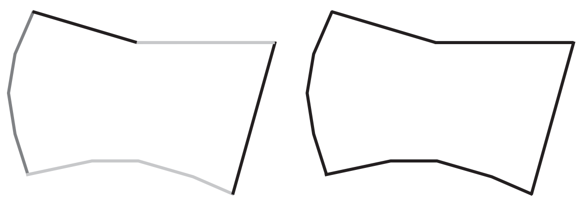
Polygon from Curves Tool



This tool creates a polygon from the chosen curves. 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.

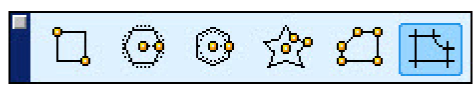
Once the polygon is created a parent/child relationship has been established between the original curves and the polygon. The original curves selected are still available for editing and any change made will affect the polygon. Due to this same parent/child relationship, in order to move the polygon and retain the relationship, the curves and the polygon must be selected. Break the relationship by selecting them and choosing ***Edit>Remove Links***. See Remove Links Command 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 the curves are selected individually, hold down the SHIFT key before beginning the selection.

   A polygon is created from those curves.

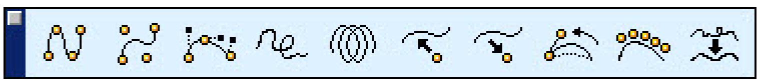
There are no entries in the Status Line.

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, the geometry will become invisible when Apply is clicked.) 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:*** *Cobalt, Xenon and Argon create 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 making minor modifications without adversely affecting the shape. For example, if a new, bigger engine wouldn't fit under a perfectly-designed hood, 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.

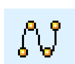
The spline creation tools include **Through-points B-spline**, **Vector Spline**, **Bezier Spline** and **Freehand Spline**. At the most basic level these splines are the same. For the Through-points B-spline and the Bezier Spline, there is additional step of interpolating the points to calculate a control polygon which is hidden from view. For the Vector Spline the interpolation step is eliminated because the accessible control polygon is defined.

Spline control points automatically lock after placement. One point can be selected and moved without affecting the other control points.

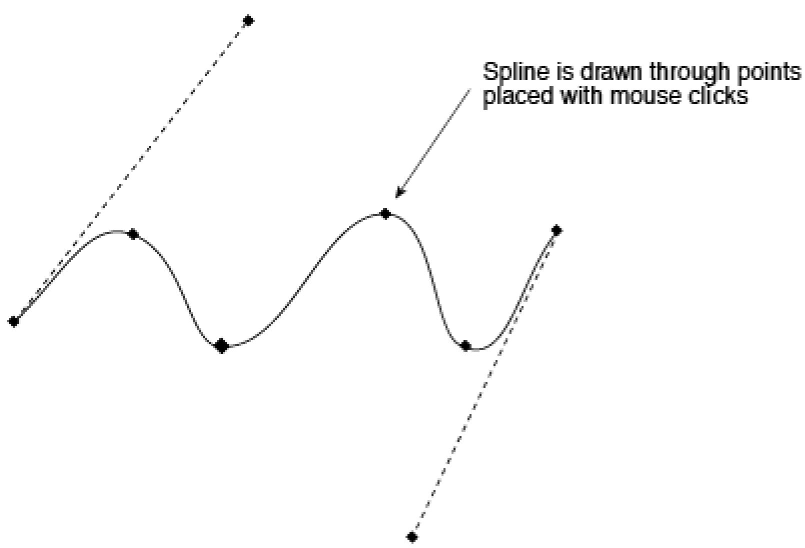
Enter values in the Status Line to define a spline after the spline is created. If values are entered in the selected Status Line data field after creating the spline and while the spline is still selected, pressing ENTER (Windows) or RETURN (Macintosh) updates the spline to reflect the new values. The tools explained here include:

- **Through-point B-spline Tool**
- **Vector Spline Tool**
- **Bezier Spline Tool**
- **Freehand Spline on Surface Tool**
- **Helix Curve Tool**
- **Add Control Point Tool**
- **Remove Control Point Tool**
- **Modify Slope Tool**
- **Elevate Curve Tool**
- **Fair Curve Tool**

Through-point B-spline Tool

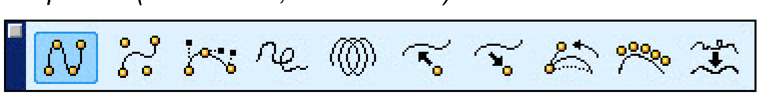


The **Through-points B-spline** tool draws a spline through the specified points. Use this tool to make the B-spline to interpolate a collection of points. The interpolation algorithm is based on predefined B-spline blending functions, control point locations and imposing curvature continuity across the curve length. This results in a smooth spline created through the points.



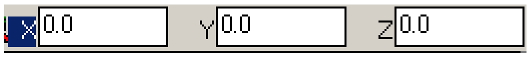
Using the Through-Points B-Spline Tool

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.



When creating a spline that crosses over itself, the Drafting Assistant is not be able to find that intersection. This is by design.

Geometric Characteristics

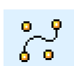
A Through-points B-spline is created by picking the desired points. A spline is made up of the following characteristics according to the Edit Objects dialog box: Start and End Angles (relative to work plane) and Defining Point.

Angles refer to the degrees of the tangent relative to the work plane for the spline. 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.

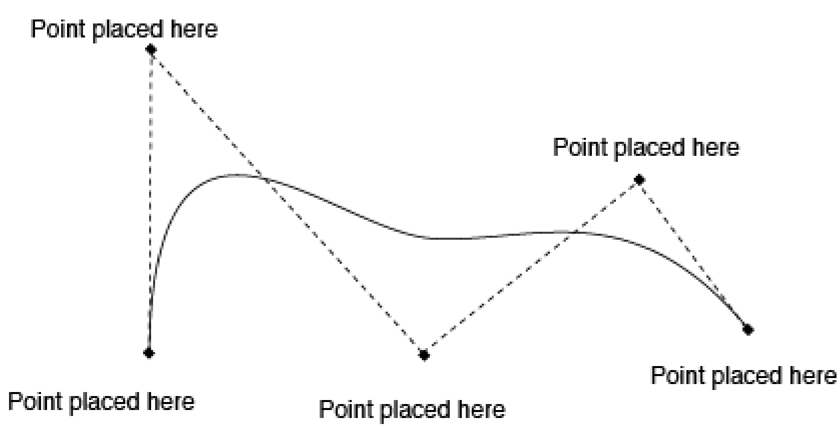
This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the spline and choose ***Window>Edit Objects*** or double-click on the spline.

Vector Spline Tool



This tool draws a spline using vectors determined by the specified points. The **Vector Spline** tool uses each placed point that as the vertex of a vector for the spline it creates. Vector splines are smooth curvature, continuous B-splines created from the defined control polygon. 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.

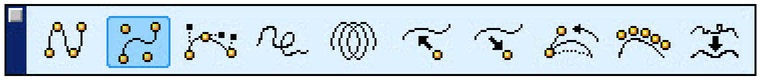
Choosing ***Edit>Show Points*** displays all the control points used to define the spline and can be selected to edit the spline. This provides the additional benefit of being able to smooth the spline to eliminate inflections. To determine whether or not the spline has inflections, choose ***Verify>Curvature***. An inflection is located where the curvature changes from one side of the spline to the other.



Tip: *With the* **Vector Spline** *tool all original points used to create the spline will display when the spline is selected.*

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.

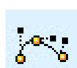
The program uses these vectors as the control points. The spline is tangent to the first and last vectors and passes between the remaining control points.

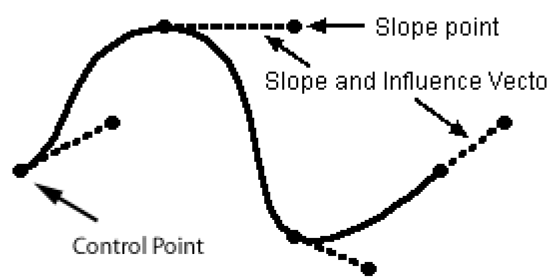
The Status Line shows the X, Y and Z coordinates of each point as they are placed.

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.

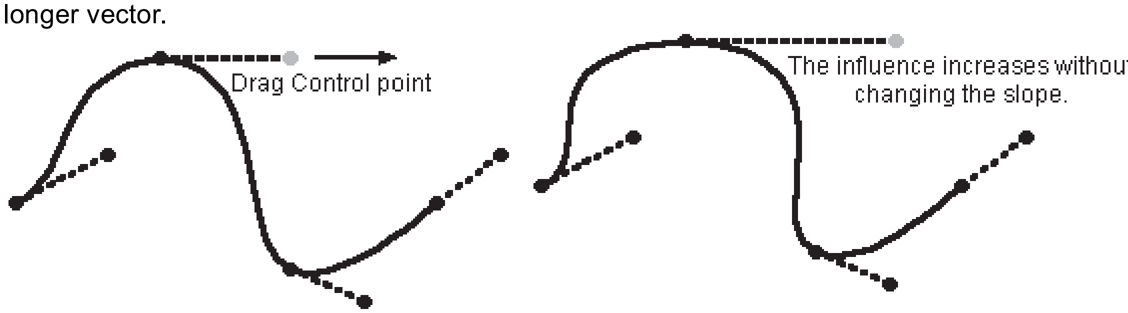
Bezier Spline Tool





This tool creates curvature continuous (C2) splines through a collection of user defined points. At each control point, it is possible to control the direction and the magnitude of the tangent. After creating a spline with this tool, it is possible to move the control points, change the slope (direction) and the influence of a control point using the spline vector.

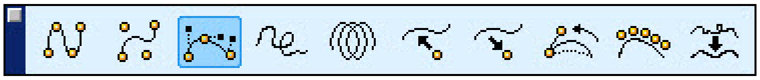
The influence of a control point is determined by the length of the control point's vector. The longer the vector, the greater the influence a control point has on the spline. It is possible to adjust the length of control point vector but maintain the slope. The left graphic shows a Bezier spline, a 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.



To change the spline use **Add Control Point**, **Remove Control Point** and **Modify Slope** tools.

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.

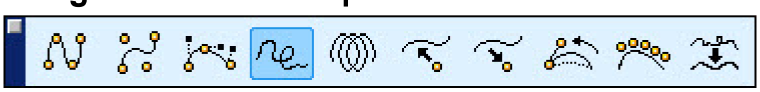
Geometric Characteristics

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.

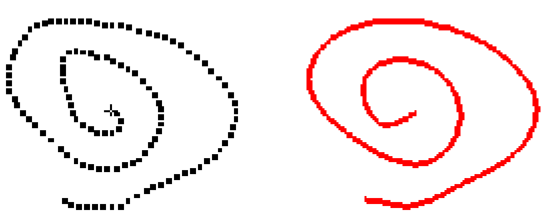
Freehand Spline on Surface Tool

The **Freehand Spline** tool samples points as the cursor moves and creates a smooth spline through the sampled data. Once selected this tool has two options located in the Message Line: the **Freehand Spline** tool and the **Freehand Spline on a Surface** tool. While the Freehand Spline tool creates a spline on the current workplane, the Freehand Spline on a Surface tool creates the spline on a designated surface or on one face of a solid or polysurface.

Using the Freehand Spline Tool

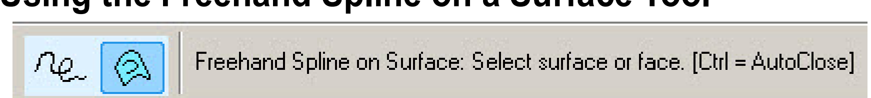


1. Select the tool. The Message Line reads: *Freehand Spline: Press and drag mouse. [Ctrl = AutoClose* (Windows) *Option = AutoClose* (Macintosh)*]*
2. Click and drag the mouse on the screen.
3. Release the mouse button to create the freehand spline.



Freehand tool - click and drag   Freehand tool - result

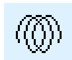
Using the Freehand Spline on a Surface Tool



1. Select the tool. The Message Line reads: *Freehand Spline on Surface: Select surface or face. [Ctrl = AutoClose* (Windows) *Option = AutoClose* (Macintosh)*]*

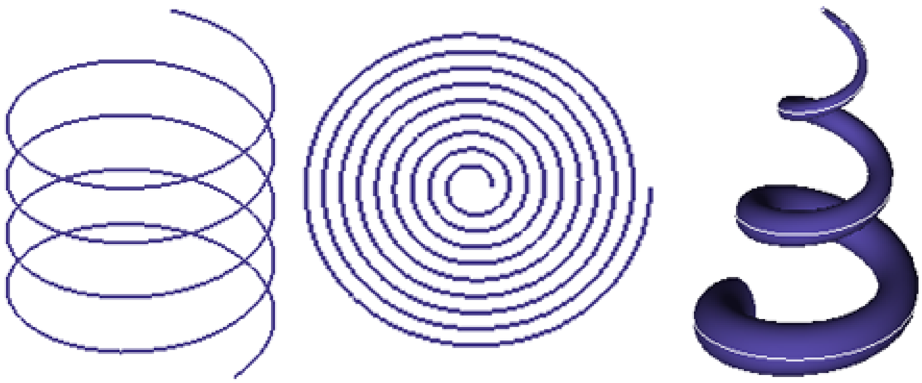
2. Select the surface or solid on which to draw.
3. Click and drag the mouse over the selected surface, keeping the points on the surface.
4. Release the mouse button to complete the spline.

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°.

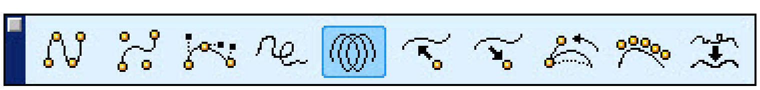
Create a standard helix (left graphic, below), a spiral (middle graphic, below) by entering zero for the helix length, or a swept solid using a helix with a draft as the sweep path (right graphic, below).



> ***Tech Note:** To sweep a surface on a helix, the helix must be created using this tool. See Sweep Surface Tool - Along 2 Points for information on using the **Sweep Surface** tool.*

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, the Status Line values can be changed. Type the new values and press ENTER (Windows) or RETURN (Macintosh) and the helix updates.

Geometric Characteristics

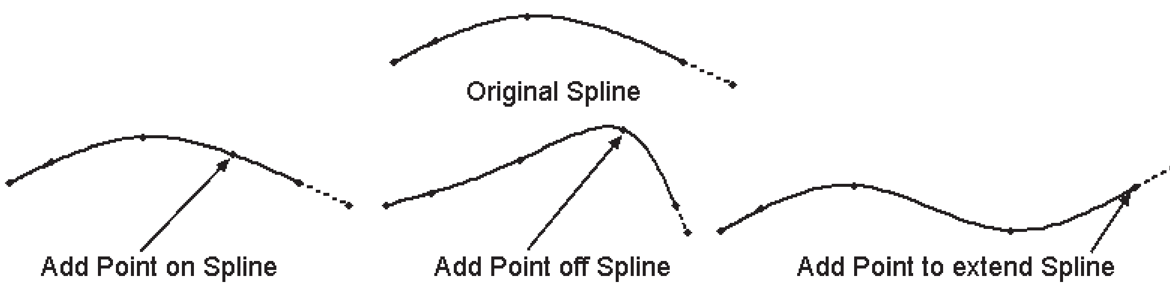
A helix is created by picking the start and end point and specifying the pitch and diameter. A helix is made up of the following characteristics according to the Edit Objects dialog box: the Diameter, Pitch, Length and Draft Angle.

There is also a check box to change the helix direction. By default, using this tool creates a right-handed helix. The helix can be changed to a left-handed helix by deselecting the *Right-handed Helix* option and clicking Apply or Close. If a left-handed helix is already created, 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.

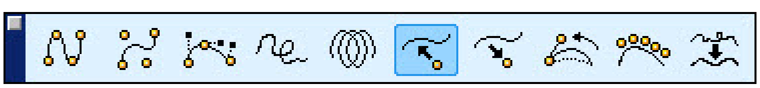
Add Control Point Tool

This tool adds 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)*.



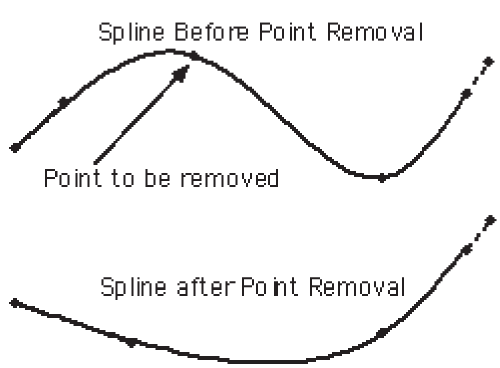
2. Click on the spline to add a point.
3. Pick the desired location(s) for the new control point(s).

Note: To see the new control point(s), as well as the existing control points, 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.

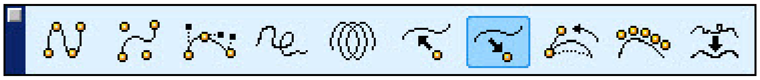
Remove Control Point Tool

This tool removes 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 to be removed.

There are no Status Line entries for this tool.

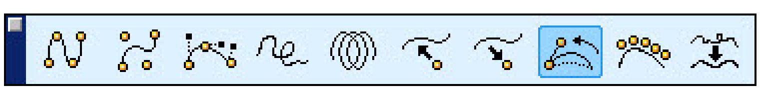
Modify Slope Tool

This tool changes the endpoint slope of an existing spline created using the **Through-points B-spline** tool. There are five options for changing the slope: *Release Slopes*, *Reference*, *Reverse, Explicit* and *Release Magnitudes*.

Release Slopes	Releases the spline slope returning it to its originally created slope control point. This is helpful if you've adjusted the slope of a spline has been adjusted in numerous locations along and needs to be returned to the shape 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	Sets the exact slope of the curve relative to the work plane (between 0° and 360°). A slope of 0° makes the curve tangent to the plane. Choose ***Planes>Show Work Plane*** when using this option.
Release Magnitudes	Releases the spline slope magnitude returning it to its originally created magnitude. This is helpful if the magnitude is adjusted in numerous locations along the spline and it is necessary to return to the spline that was originally created. (This only works with Bezier splines.)

Using the Modify Slope Tool - Release Slopes Option

1. Select the tool.



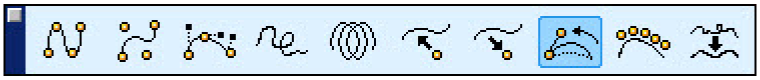
2. Choose the Release Slope option in the pull-down menu of the Message Line. The Message Line reads: *Modify Slope: Select spline slope(s) to release.*
3. If the control points of the spline are not displayed, select the spline and choose ***Edit>Show Points***.
4. Pick the spline slope control endpoint to be freed.

   The slope control endpoint returns to its original position.

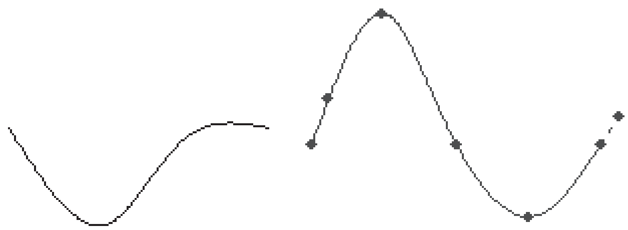
There are no Status Line entries for this tool.

Using the Modify Slope Tool - Reference Option

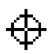
1. Select the tool.

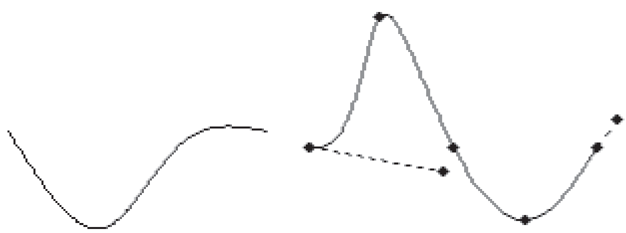


2. Choose the *Reference* option in the pull-down menu of the Message Line. The Message Line reads: *Modify Slope: Select the spline end to set slope*.
3. If the control points of the spline that need to be modified are not displayed, select the spline and choose ***Edit>Show Points***.
4. Pick the spline to modify. In the graphic here, it's the spline displaying points.



5. Pick the curve, surface or solids whose slope is to be referenced.

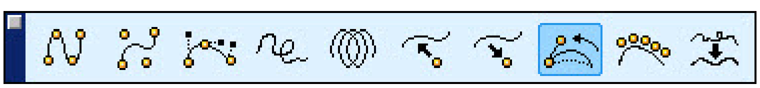
   The icon now becomes a target icon. 
6. Click on the endpoint of the curve to be modified, (The curve and the reference object do not have to connect.) The curve slope adjusts.



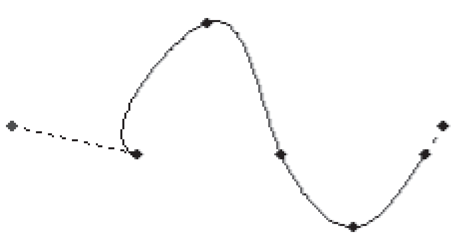
There are no Status Line entries for this tool.

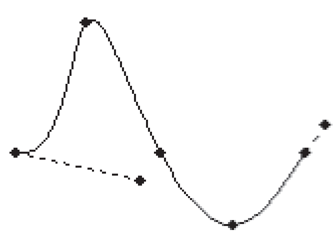
Using the Modify Slope Tool - Reverse Option

1. Select the tool.



2. Choose the *Reverse* option in the pull-down menu of the Message Line. The Message Line reads: *Modify Slope: Select spline slope point to reverse.*
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 to be reversed, as in the left graphic below. The slope control endpoint reverses 180° (right graphic).

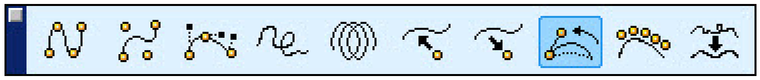




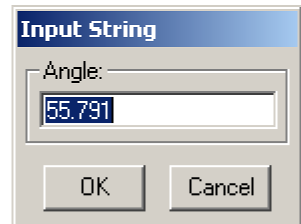
There are no Status Line entries for this tool.

Using the Modify Slope Tool - Explicit Option

1. Select the tool.



2. Choose the *Explicit* option in the pull-down menu of the Message Line. The Message Line reads: *Modify Slope: Select spline slope point for explicit set.*
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 to be set. An Input String dialog box appears.

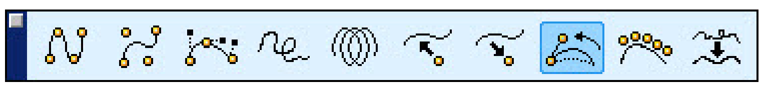


5. Enter the Angle for the slope in the data field and click OK. The dialog box closes and the slope of the spline is adjusted. (Click Cancel to close the dialog box without changing the angle value.)

There are no Status Line entries for this tool.

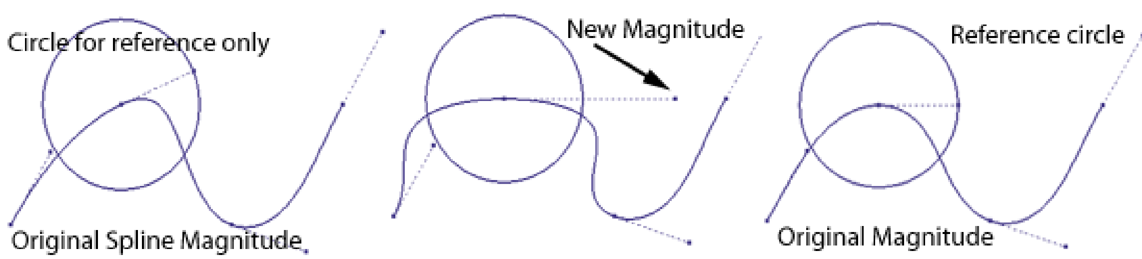
Using the Modify Slope Tool - Release Magnitudes Option

1. Select the tool.
2. Choose the Free Magnitudes option in the pull down menu. The Message Line reads: *Modify Slope: Select spline magnitude to release*.



3. If the control points of the spline are not displayed, select the spline and choose **Edit>Show Points**.
4. Select the bezier curve slope or control point to be released.

The Bezier slope resets to a natural magnitude.

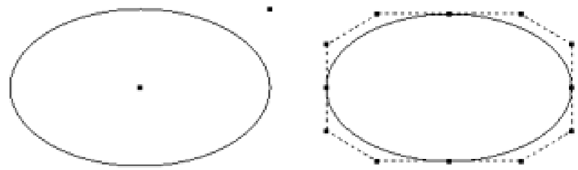


Elevate Curve Tool

This tool elevates the degree of a curve, introducing more control points. These control points can be used to modify the curve.

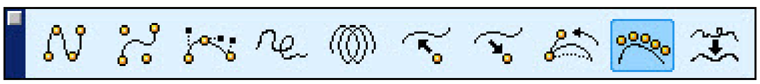
> ***Note:*** *The* **Elevate Curve Tool** *changes the object type of the curve being elevated to a vector spline.*

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 tool raises 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 to never elevate a curve higher 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.*



2. Select the curve to elevate.

The curve elevates one degree.

There are no entries in the Status Line.

Fair Curve 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 the spline is fired with the **Edit>Show Points** command. Use the **Verify>Curvature** tool to visually inspect the curvature graph of a curve.

Using the Fair Spline Tool

1. Select the tool.
2. The Message Line reads: *Fair Spline: Select curve to fair*.
3. Click on the spline 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.

Wireframe Editing Tools

Once an object is created it can be selected and altered 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. 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 enter values in the Status Line to perform the editing operation before or after the operation is complete. If the values are entered after selecting the tool but before performing the operation, the first click in the drawing area automatically registers all Status Line values. If values are entered 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.

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 the first step is completed any other objects cannot be selected to include in the same operation. To select additional objects reset the tool and start again.

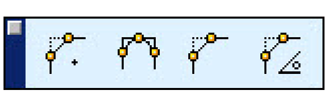
> ***Tech Note:*** *These editing tools cannot be performed on grouped objects, only curves.*

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

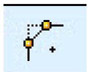
Fillet and Chamfer Tools



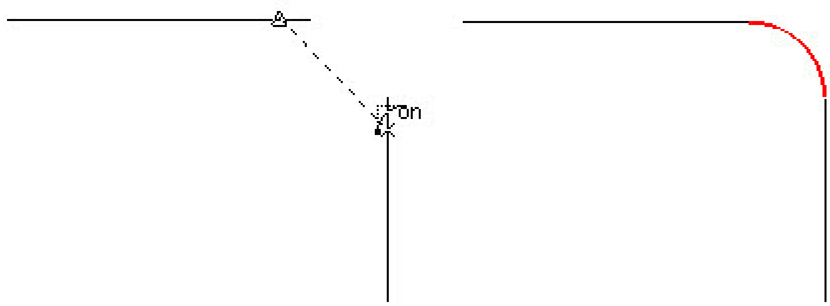
These tools on the main tool palette construct fillets and chamfers from corners formed by nonparallel lines or curves. The program automatically trims fillets and chamfers by default. If they do not need trimming, hold down the CTRL (Windows) or the OPTION (Macintosh) key while selecting the objects to fillet or chamfer.

All filleted curves being trimmed must be in the same plane.

2-entity Fillet Tool

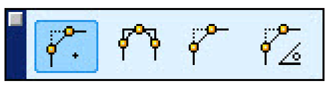


This tool constructs an arc tangent to the two specified objects.



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)]*.

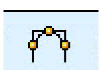


2. Enter the arc radius in the Status Line. The default radius is .50 inch.
3. Click the first curve. The Message Line reads: *2-entity Fillet: Pick second entity. [Ctrl=No trim]* (Windows) or *[Option=No Trim* (Macintosh)*]*.

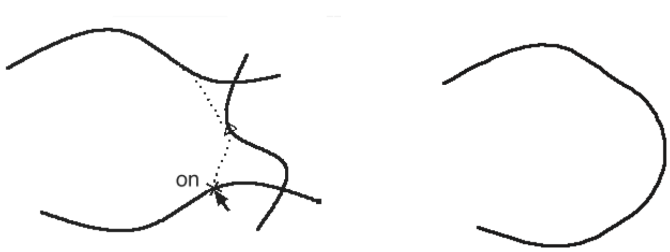
It is also possible to hold down the SHIFT key and click once inside the corner to fillet. (See the Message Line at the top of the drawing area for a reference to this feature.) Holding down the CTRL (Windows) or OPTION (Macintosh) key while selecting the objects to fillet, the objects are not trimmed. Fillets create the smallest possible arc between the selected geometry.

The Status Line contains the Radius of the fillet.

3-entity Fillet Tool

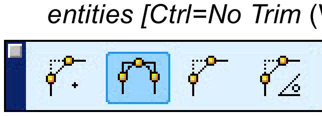


This tool constructs a fillet tangent to the three chosen objects.



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)*]*.

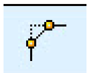


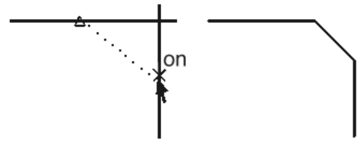
2. Click the three objects to fillet.

Holding down the CTRL (Windows) or OPTION (Macintosh) key while selecting the objects to fillet, the objects are not trimmed.

There are no Status Line entries.

2-entity Chamfer Tool



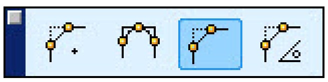


This tool creates a chamfer across a corner at the specified distance from the intersection of two lines.

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. [Shift = Corner, Ctrl = No trim* (Windows) or *Shift = Corner, Option = No trim* (Macintosh)*]*.

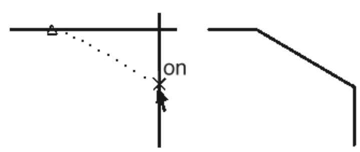
It is also possible to hold down the SHIFT key and click once inside the corner to chamfer.

The face of a part, rather than just the edges, may be selected for a chamfer. Use the Ambiguity box to specify the edge or face when performing the operation.

The lines are automatically trimmed or extended. Holding down the CTRL (Windows) or the OPTION (Macintosh) key while selecting the objects to chamfer, the objects are not trimmed.

The Status Line specifies the distance (Length) from the chamfer to the intersection of the corner lines.

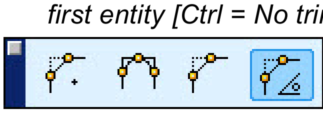
Angular Chamfer Tool



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. The default length is .50 inch.
3. In the Status Line also enter the angle 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)*]*.

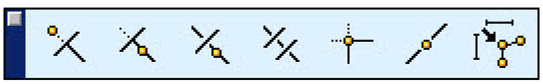
It is also possible to hold down the SHIFT key and click once inside the corner to chamfer.

The face of a part, rather than just the edges, may be selected for a chamfer. Use the Ambiguity box to specify the edge or face when performing the operation.

The lines are automatically trimmed or extended by default. Holding down the CTRL (Windows) or the OPTION (Macintosh) key while selecting the objects to chamfer, the objects are not trimmed.

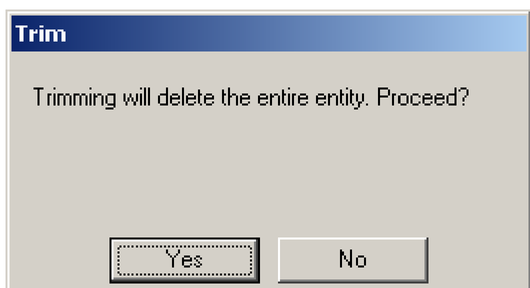
The Status Line sets the distance (Length) from the intersection as well as the Angle.

Trim Tools



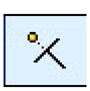
The **Trim** tools on the tool palette lengthen or shorten lines, curves and polylines. Select the limiting object(s) before or after selecting the tool. If every object that needs selecting is not selected, hold down the SHIFT key for the tool becomes a temporary **Selection** tool and click additional objects after the **Trim** tool is selected.

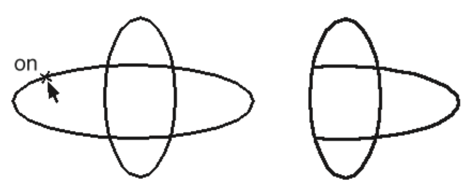
When trimming curves using one of these tools the following warning will appear if the operation will delete the entire curve.



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. Holding down the CTRL (Windows) or the OPTION (Macintosh) key while using this tool, it becomes the **Relimit** tool.

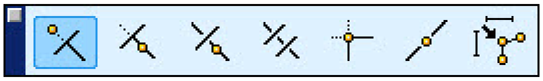
When trimming a Through Point B-spline or Bezier spline an editable vector spline results. It is possible to move its control points and change its shape and slope. In some situations trimming a curve results in a spline. This spline is also an editable vector spline.

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 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 the boundaries are already selected, proceed to step 3.

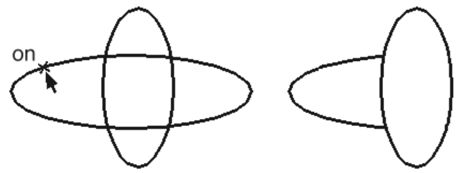
2. Select the object that limits the trim.

   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



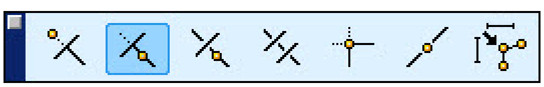
This tool lengthens or shortens a curve to the specified boundary. Hold down the CTRL (Windows) or the OPTION (Macintosh) key while using this tool to make it the **Trim** tool. The **Relimit** tool will not relimit arcs.

When relimiting a Through Point B-spline or Bezier spline an editable vector spline results. 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 to keep.

Using the Relimit Tool



1. Select the **Relimit** tool. The Message Line reads: *Relimit: Select boundaries for relimit [Shift = Extend].*

   If the boundaries are already selected, proceed to step 3.

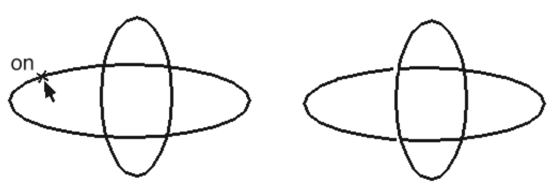
2. Select the object that limits the change.

   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 to retain.

There are no Status Line entries.

Segment Tool

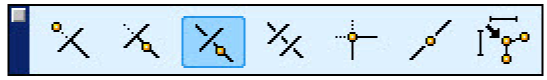


This tool divides a curve at intersections with other lines or curves.

When segmenting a Through Point B-spline or Bezier spline an editable vector spline results. It is possible to 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 the boundaries are already selected, proceed to step 3.
2. Select the objects that limit 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 the segmentation cannot be seen on the screen, parts of the segmented line can be selected by choosing it with the **Selection** tool. In order to see the segmentation on the screen, first select the boundaries and the object to segment, then choose ***Edit>Show Points***. The endpoints of the segmented parts are displayed.

Holding 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.

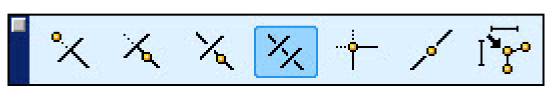
There are no Status Line entries.

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 a Through Point B-spline or Bezier spline is divided, an editable vector spline results. Its control points can be moved to 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

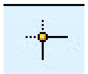


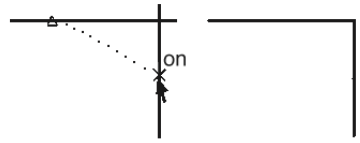
1. Select the **Divide** tool. The Message Line reads: *Divide: Pick curve to divide. [Shift=Extend]*
2. Set the No. of Segments value in the Status Line.
3. Select the object to divide.

   The object divides. The individual segments are not indicated unless their control points are turned on.

The Status Line sets the number of segments in the curve.

Corner Trim Tool



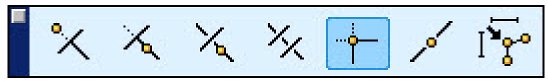


This tool creates a corner from two specified objects. Lines are extended or shortened to create the corner.

Conduct a corner trim on a Through Point B-spline or Bezier spline to result in an editable vector spline. Move its control points to 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].*

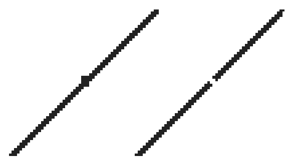


2. Select the first curve.
3. Select the second object.

As an alternative to picking two entities, 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 chosen point.

Dividing a Through Point B-spline or Bezier spline using this tool an editable vector spline results. Move its control points to change its shape and slope. In some situations dividing a curve results in a spline. This spline is also an editable vector spline. It is also possible to divide 360° circles and ellipses.

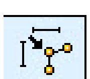
Using the Divide at Location Tool

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 the division will be made.

There are no Status Line entries.

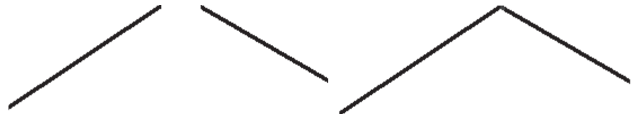
Connect Curve Tool



This tool connects curves. The program provides 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.)

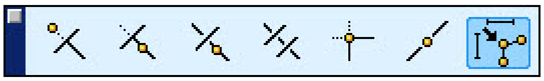
Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when connecting curves with this tool.

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 to 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)*]*.



2. Select the first curve near the endpoint where it will connect the curve.
3. Select the second curve near the endpoint that needs moving.

   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.

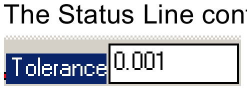
Using the Connect Curve Tool - Join Option

The *Join Curve* option replaces the two selected curves 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 curves are converted into polylines using the *Change Object* command and later it is necessary to join them. Select the *Join Curve* option as picking the two curves.

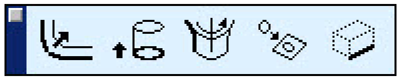
1. Select the **Connect Curve** tool. The Message Line reads: *Connect Curve: Select two curves to connect end points [Ctrl=Join* (Windows) or *Option=Join* (Macintosh)*]*.
2. If desired, specify a tolerance distance in the Status Line to which the resulting curves will fit.
3. Hold down the CTRL key (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.



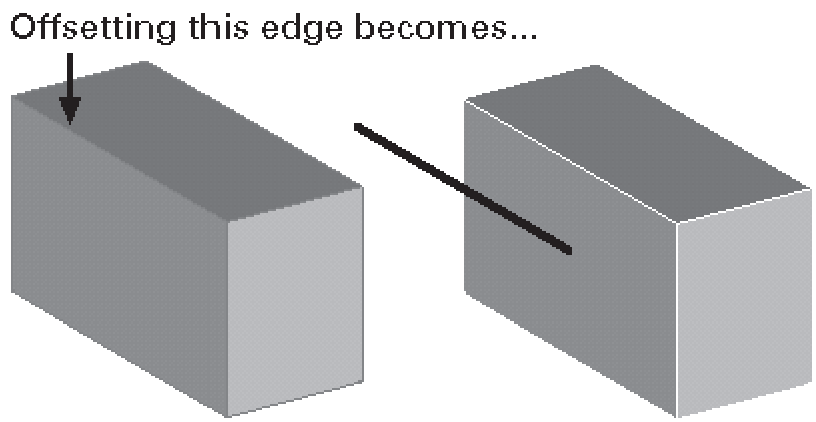
Curve Utility Tools



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 specified distance. 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.

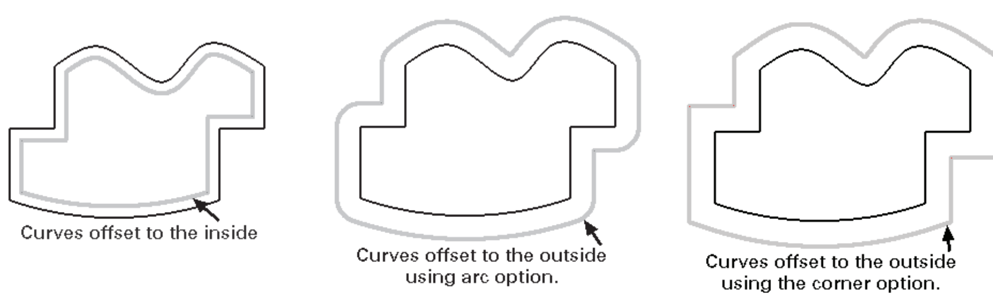


One or more curves can be offset using the following options.

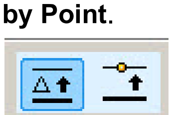
Automatic trimming for curves offset inside curves.

Curves offset outside of curves with the arc corners for the offset.

Curves offset outside of curves with extended corners for the offset.



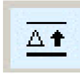
The **Offset Curve** tool includes a subpalette with two tools, **Offset to Value** and **Offset by Point**.



When offsetting a Through Point B-spline or Bezier spline an editable vector spline results. Move its control points to 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



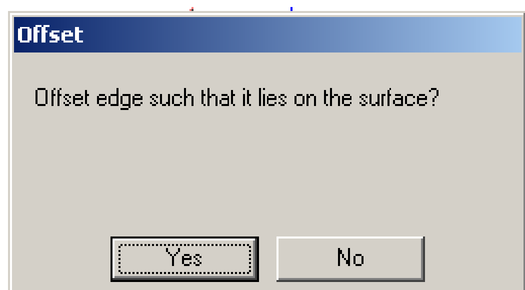
This tool creates an offset curve at a specified distance from the original curve, surface edge or solid edge. When offsetting surface edges there is the option of placing the offset on the surface. It is possible to offset one curve or edge or connected curves or edges.

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*.

   Selecting a surface edge, the Offset dialog box appears asking if the offset lie on the surface.

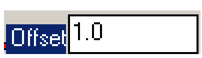
   Click Yes to place the offset on surface. Click No and proceed with the next step.



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. The program offsets the curve by the distance set in the data field.

While the offset curve is selected a new offset distance may be specified 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 there are 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.

   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.

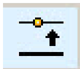
   Interior Offset: Pick a point on the inside of the curves or edges.

   The program offsets the curves or edges by the distance set in the Status Line data field.

While the offset curves are selected a new offset distance may be entered in the Status Line. Press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the Offset distance.

Offset by Point Tool



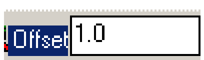
This tool creates an offset at the point specified. It is possible to offset one curve or solid edge or connected curves or solid edges.

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 Corner* (Windows) or *Option=Extend Corner* (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. The 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 it is possible to 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 Corner* (Windows) or *Option=Extend Corner* (Macintosh)*] [Shift=Extend]*
3. Select the curves to offset. The Message Line reads: *Offset Curve: Enter offset point*.
4. Select the type of offset.

   Exterior Offset - Arc Intersection: Pick a point on the outside of the curves.

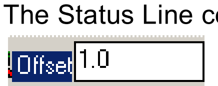
   Exterior Offset - Corner Intersection: Hold down the CTRL key (Windows) or the OPTION key (Macintosh) and pick a point on the outside of the curves.

   Interior Offset: Pick a point on the inside of the curves.

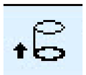
   The program offsets the curves at that point. The offset value is displayed in the Offset data field in the Status Line.
5. Pick the desired point for the offset. The Designer Elements program offsets the curve to that point. The offset value is displayed in the Offset data field in the Status Line.

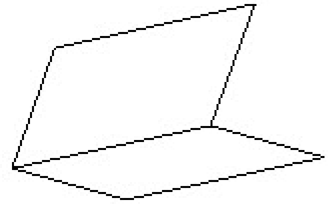
While the offset curves are selected it is possible to enter a new offset distance in the Status Line. Press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the offset distance.



Extrude Curve Tool

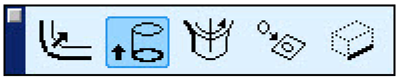




This tool copies a curve at a distance and in the specified direction. It connects its endpoints by two lines back to the original curve. In the graphic to the right the lower curve is extruded to form the polygon.

Using the Extrude Curve Tool

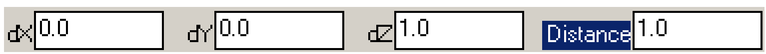
1. Click the **Extrude Curve** tool. The Message Line reads: *Extrude Curve: Pick curves to extrude [Shift=Extend]*.



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 the points are picked. This Designer Elements program applies the specified distance and direction.

While the object is selected it is possible to change the extrusion distance in the Status Line. Press ENTER (Windows) or RETURN (Macintosh).

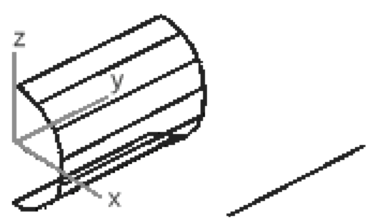
The Status Line contains the extrusion's dX, dY and dZ values and the Distance.



***Tip:** To extrude an object 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.

***Tech Note:** A Smart Polygon cannot be revolved. It must first be converted into single lines using the Change Object Type command. See Polygon Tools for more information on polygon types.*
To define the axis of revolution click the points rather than drag.

Revolve Curve Tool



This tool revolves a curve around an axis.

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 two points for the revolution axis.*
3. Click two points to indicate the axis of revolution. The direction of the revolution is specified by the order the points are picked. The 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 it is possible to change the number of steps and the angle of revolution in the Status Line. Press ENTER (Windows) or RETURN (Macintosh).

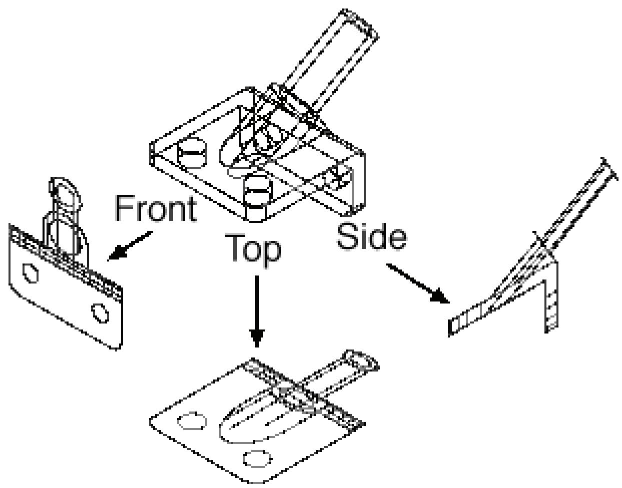
The Status Line contains the number of steps and the angle of revolution.

Project Curve Tool

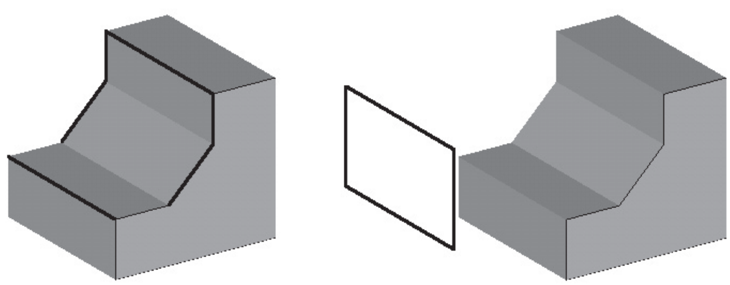
The **Project Curve** tool projects 3D curves, including solid edges onto a 2D plane. Arcs, circles and ellipses maintain their precise shape when being normally projected to their definition. Being projected otherwise, a spline is used with a default curve tolerance of 0.001 inches. This spline is an editable vector spline.

When projecting a Through Point B-spline or Bezier spline an editable vector spline results. Move its control points to 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 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.

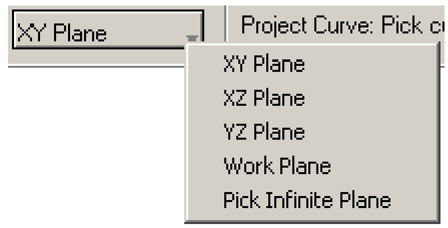


The left graphic below shows a model with four solid edges selected (displayed in black). The right graphic shows the edges projected to the XZ Plane.

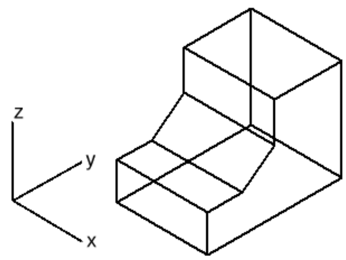


Plane Options

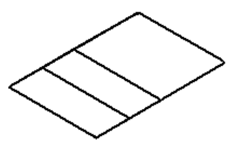
The pull-down menu in the Message Line provides five options for choosing the plane: *XY Plane*, *XZ Plane*, *YZ Plane*, *Work Plane* and *Pick Infinite Plane*.



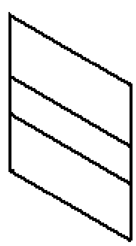
The following figures, displayed in the Isometric view, explain the projection options. All graphics used for the individual planes are also displayed in the Isometric view.



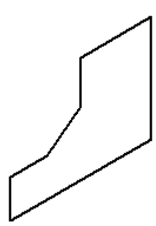
XY Plane Choosing this option projects the selected curves into the XY plane (Top) where X=0 and Y=0.



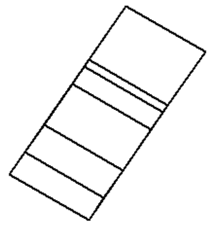
XZ Plane Choosing this option projects the selected curves into the XZ plane (Front) where X=0 and Z=0.



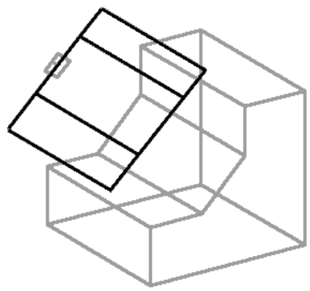
YZ Plane Choosing this option projects the selected curves into the YZ plane (Side) where Y=0 and Z=0.



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 projects the selected curves to the specified infinite plane.



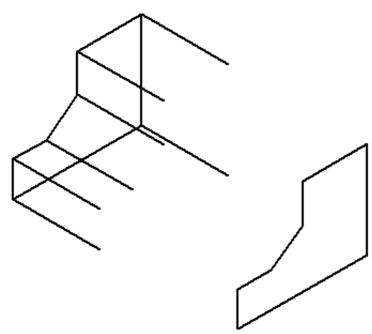
Use the Plane commands in the Plane menu to define the plane. Set the work plane to the desired plane before using this option.

Projection Options

When projecting curves this Designer Elements program takes those curves and creates new geometry on the specified plane. There are two options, Default and Copy.

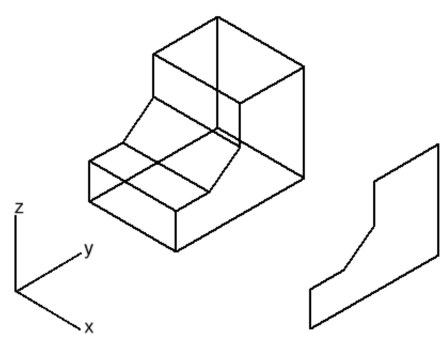
Project Curves - Default Option

This option takes the selected geometry. See the graphic here. Notice that the selected geometry no longer appears on the original part.



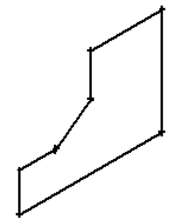
Project Curves - Copy Option

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 selecting the tool, it is possible to select multiple lines for projection by holding down the SHIFT key and selecting the desired lines, as noted in the Message Line.

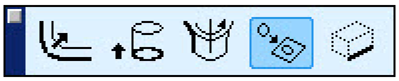


Select multiple lines or an entire object by dragging a selection fence around the entire object. When projecting the object, all curves that are normal to the projection plane are recreated as points in the new curve. The graphic here shows a YZ projection. The cross hairs represent the points.

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]*



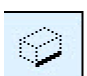
2. From the pull-down menu in the Status Line choose the projection plane.
3. Select the curves to project. Hold down the SHIFT key or drag a selection fence to select multiple lines. The Message Line now reads, *Project Curve: Specify point in projection plane. [Ctrl=Copy* (Windows) or *Option=Copy* (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.

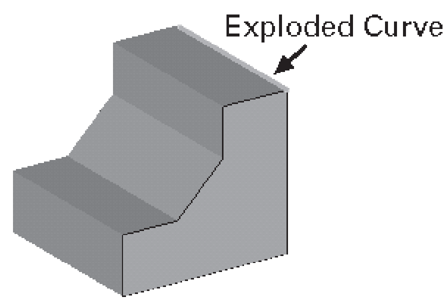
   To project a copy of the lines hold down the CTRL key (Windows) or OPTION key (Macintosh) before selecting the point.

There are no Status Line entries.

> ***Tech Note:** If ENTER (Windows) or RETURN (Macintosh) is pressed instead of specifying a point, the projection is placed at the origin.*

Explode Edge Tool



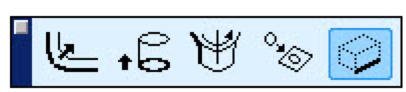


This tool creates a curve from a surface or solid edge. For surfaces and solids, Iso lines can be also exploded when displayed on the geometry. This ability is useful when constructing geometry off an object.

When exploding 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. Move its control points to 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 to create a curve.

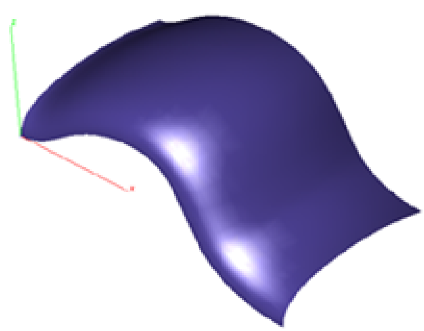
   The curve is created at the same location as the selected edge.

There are no entries in the Status Line.

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 way to create complex shapes such as cars, ships and aircraft.

In Cobalt, Xenon and Argon, 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 the Designer Elements programs are suitable for precise geometric analysis and can be passed to computer-aided manufacturing applications that support SAT or IGES file formats.



The Designer Elements programs have 5 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.

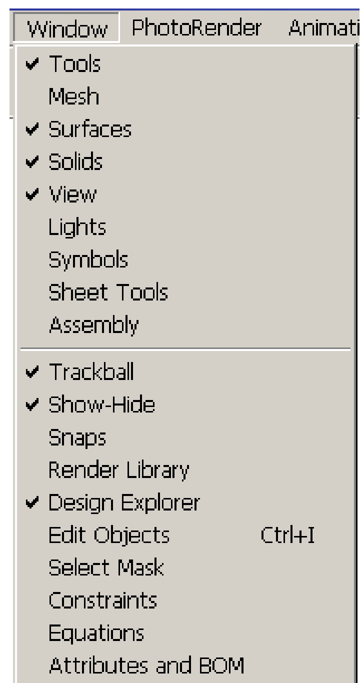
Values for each tool can be entered in the Status Line to define a surface, either before or after creating the surface. If the values are entered after selecting the tool but before creating the surface, the first click in the drawing area automatically registers all Status Line values. If values are entered 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 the first step is completed (which may involve holding down the SHIFT key to select multiple objects), it is not possible to select more objects to include in the same operation. To select additional objects, reset the tool and start again.

The topics discussed in this chapter include:

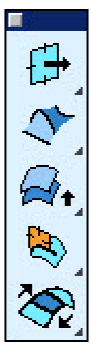
- Surface Modeling Tools
- Surfaces, Selection and Display
- Surface Associativity
- Cutting and Pasting Surfaces
- Exporting Surfaces
- Objects Types and Edit Objects
- Surface Control Vertex Modification

Surface Modeling Tools



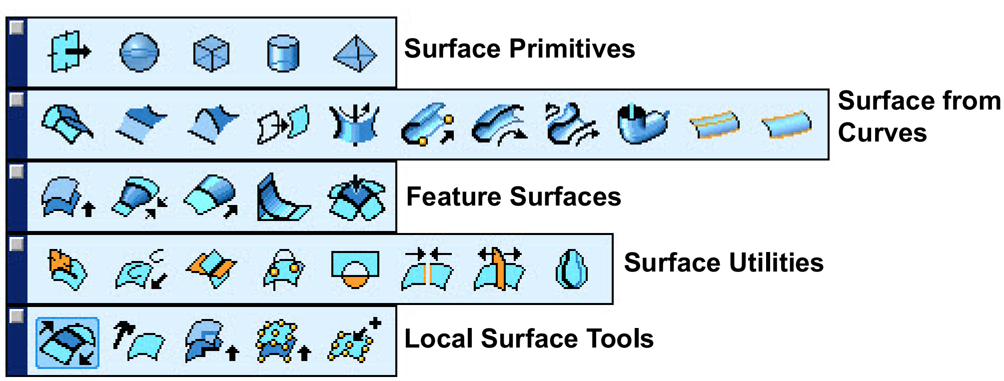
The surface modeling tools are accessible from the **Surfaces** tool palette. The **Surfaces** tool palette automatically displays when Cobalt, Xenon or Argon is first launched. If the palette is closed and it is necessary to display the palette again, choose ***Window>Surfaces***.

The palette will appear under the main tool palette on the left side of the screen.

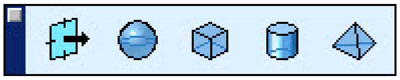


Save the open status of the palette and its location, simply by quitting the program. The next time the program is launched, the palette will be open and placed in the last location.

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

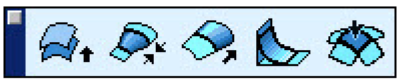


The **Surface Primitives** tool palette has tools for quickly creating surfaces from simple shapes and parameters. This tool palette offers **Infinite Planes**, **Speheres**, **Blocks**, **Cylinders** and **Pyramids**. (For more information on these tools see Surface Primitive Tools.)

Surface From Curves Tools

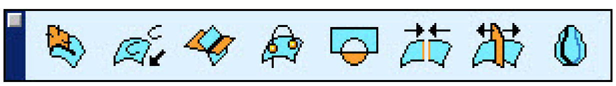
Use the **Surfaces From Curves** tools to construct surfaces built upon curves. This tool palette has the **Net**, **Ruled**, **Skin**, **Cover**, **Revolution** (lathe), **Extruded**, **Swept**, **Swept 2 rail** and **Pipe** surface tools. (For more information see Surface From Curves Tools..)

Feature Surface Tools



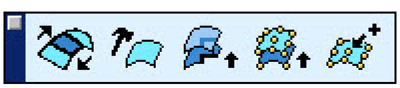
The **Feature Surface** tool palette constructs surfaces built upon other surfaces. The tools in this palette include **Offset**, **Loft**, **Draft**, **Fillet** and **Tangent Cover Surface**. (For more information on these tools see Derived Surface Tools.)

Surface Utilites Tools



After surfaces are constructed or imported into the model, they can be refined or combined with the surfaces with other entities. The program provides several tools which make these tasks easy to complete. The **Surface Utilities** palette has tools such as **Entity Intersections**, **Projections**, **Joining**, **Trimming** and **Silhouette Curves** commands. (For more information see Surface Utility Tools.)

Local Surface Tools



Tools in the **Local Surface** palette 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 Surface Modification Tools.)

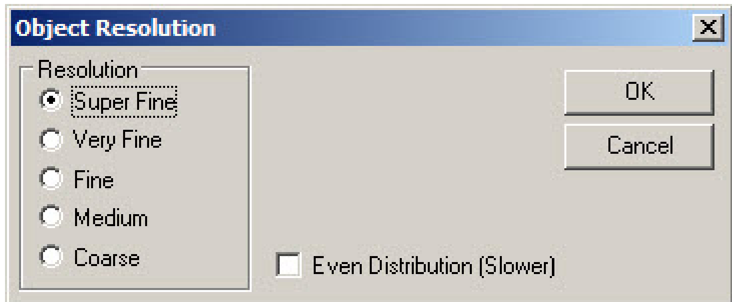
Surfaces, Selection and Display

To select a surface click anywhere within its boundary, however, the fastest way is if the surface's edge is selected. A surface is displayed by its boundary edges and any existing interior holes. A surface's edge resolution can be increased 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.



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.

3. Enter the desired number of Iso lines. Check the *Show Silhouettes* box.

   The number of *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 on the silhouette.

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, creating a skin surface from three arcs and changing 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 curve changes should not 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 a surface is cut 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. This data can be pasted into other Windows or Macintosh programs that support the clipboard.

Exporting Surfaces

The Designer Elements programs provides several methods for exporting surfaces into other applications. Our primary goal is 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 created with a Designer Elements program tool is defined by its own characteristics, and includes 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.

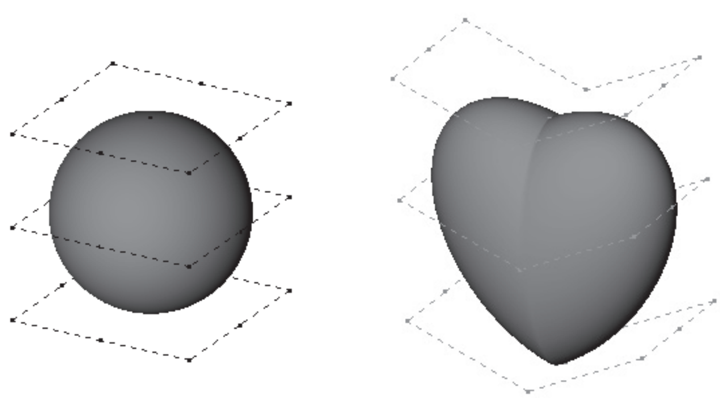
Chapters 16, 17, 18, 19, 20, and 21 describe the surface tools. Included with every tool description is a list of the characteristics displayed for the object when Edit Objects is chosen. For information on using the *Edit Objects* command and the dialog box, see Editing Objects.

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 Surface Evaluation.

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.

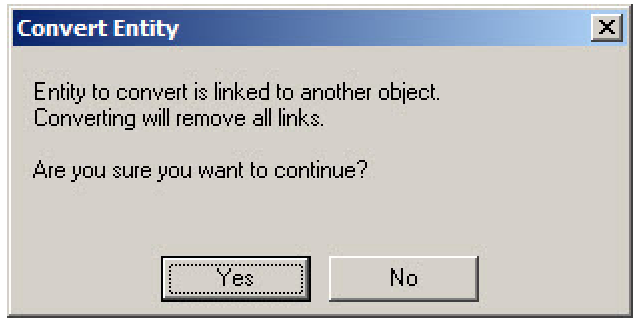


Performing a Control Vertex Modification

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.

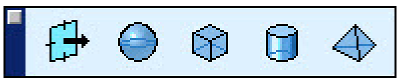
If the surface you selected is linked to another object the following warning box appears:



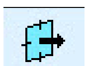
Click Yes to continue conversion.

5. Choose ***Edit>Show Points*** to display the surface control points.
6. Using the **Selection** tool, drag the control points as desired.

Surface Primitive Tools



Infinite Plane Tool

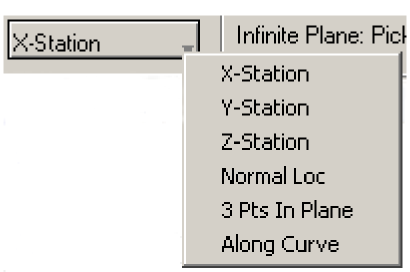


The *infinite plane* is the simplest surface supported by the Designer Elements programs. An infinite plane is defined by a location and a normal. The plane surface is useful for generating cross-sections through meshes, surfaces or solids.

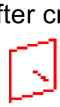
The **Infinite Plane** tool has six creation methods listed in the pull-down menu of the Message Line.

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 X-, Y- and Z-Station Planes Option

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*.

   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 on the screen where the plane will be. An icon for the plane appears at that location.

The Status Line displays the X, Y and Z location for the plane.

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.

> ***Tip:*** *Determine the distance between two infinite planes by selecting the planes and choosing* ***Verify>Minimum Distance.***

Using the Normal and Location Plane Option

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).*

   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 the plane's normal will end, the point where the plane's normal will start, and the point where the plane will be placed.

There are no entries in the Status Line.

Using the Infinite Plane Tool to Create a Three Point Plane

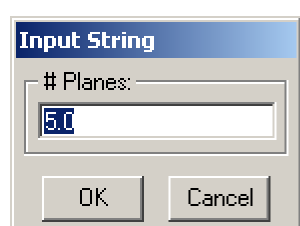
1. Select the **Infinite Plane** tool.
2. Pick *3 Pts In Plane* option from the drop down list on the Message Line. The Message Line reads: *Infinite Plane: Enter 3 points in the plane*.

   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 appears at the specified location.

There are no entries in the Status Line.

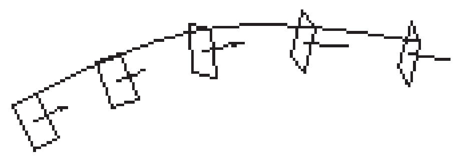
Using the Infinite Plane Tool to Create a Plane Along a Curve

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.



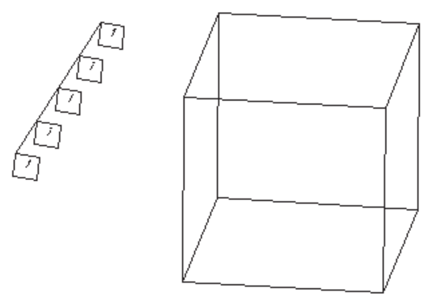
If the curve is selected, the Input String dialog box reappears to change the number of planes placed along the curve.

There are no entries in the Status Line.

Infinite Plane Tool - Along the Curve Option and the Plane/Surface Intersection Tool

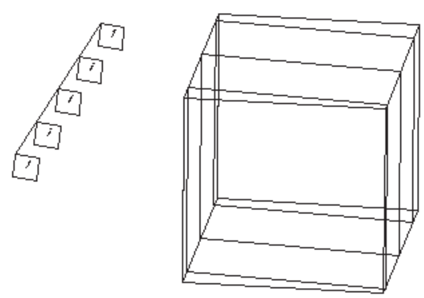
Use the *Along Curve* option of the **Infinite Plane** tool with the **Plane/Surface Intersection** tool to quickly create cross sections through solid models.

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 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. The Designer Elements program puts each projected plane on a separate layer after clicking the *AutoLayers* box on the Message Line before creating them.
5. The intersections display.

There are no entries in the Status Line.

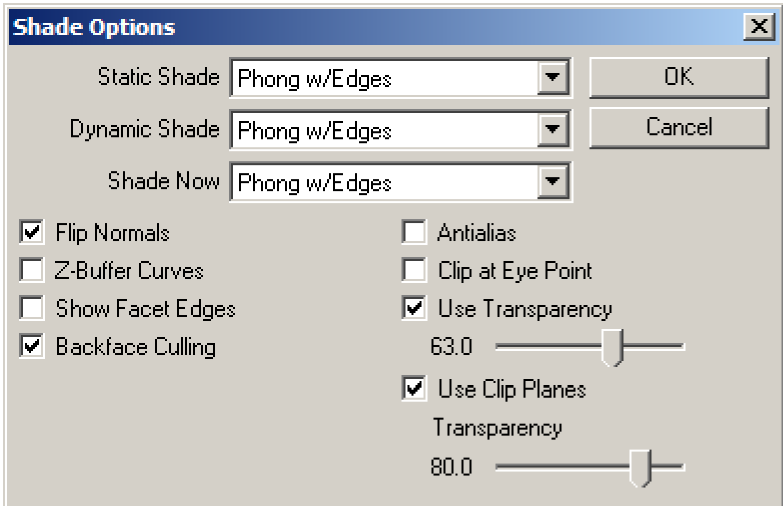


Referral: *See Plane/Surface Intersection Tool for information on using the* **Plane/Surface Intersection Tool** *.*

Clipping Planes

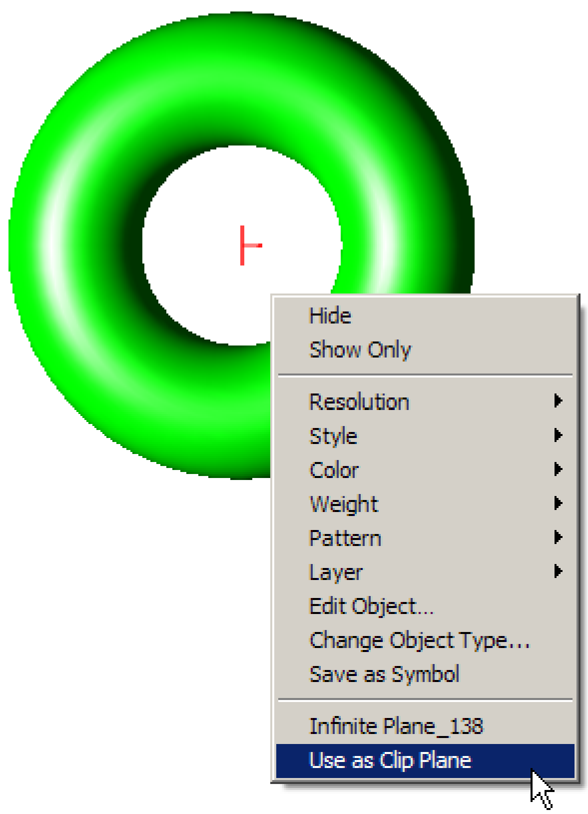
Dynamic display clipping planes provide a means to slice through the model and hide the display to either side of the plane.

In order to use an **Infinite Plane** as a **Clipping Plane** bring up the **Shade Options** dialog box located in the **View Menu** and select the *Use Clip Planes* option.

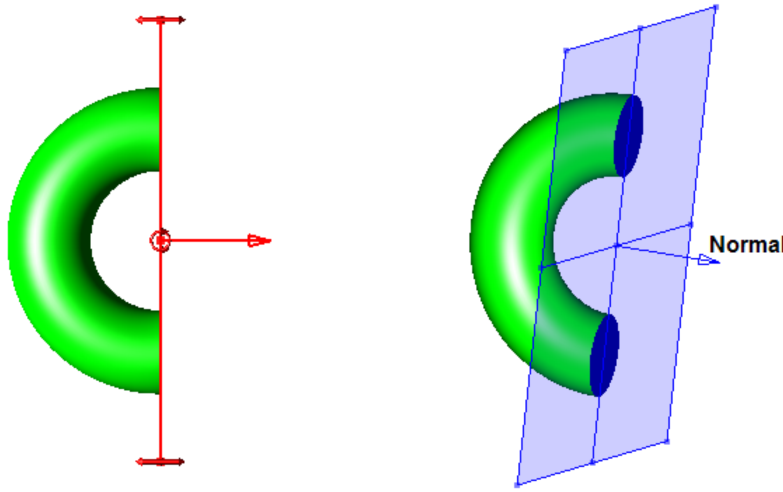


Using an Infinite Plane as a Clipping Plane

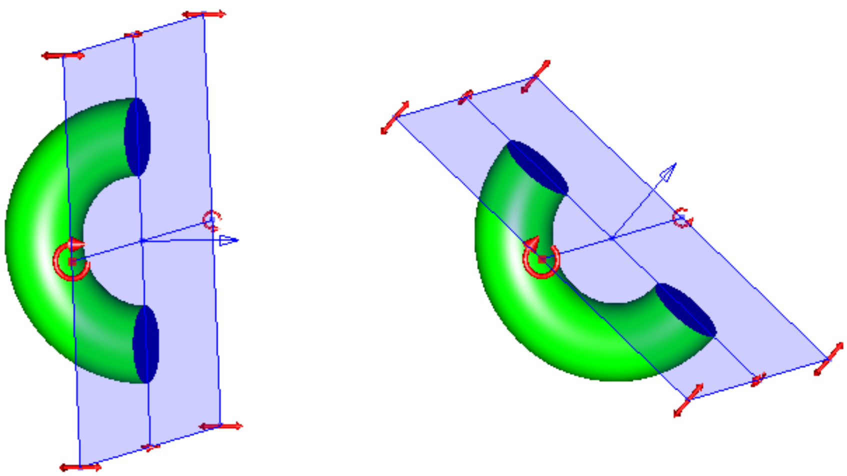
1. Select the **Infinite Plane** tool. The Message Line reads: *Infinite Plane: Pick location for plane*.
2. Place the infinite plane in the drawing.
3. Right-Click on the infinite plane and select the *Use a Clip Plane* menu option.



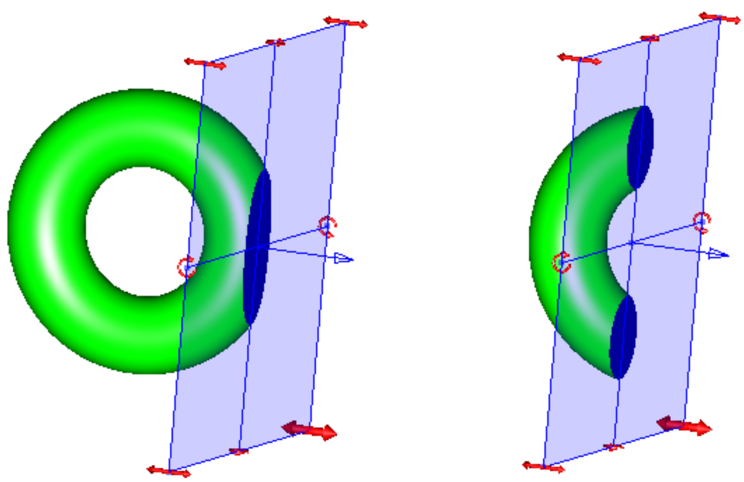
4. All objects are visually clipped and sectioned. The perpendicular arrow represents the normal, which indicates the direction of the clipped part of the unit.



5. Adjust the plane visually using the handles on the plane indicator. There are 3D handles on the corners, midpoints and center of the clip plane. Midpoint markers rotate the clip plane. Hold the SHIFT key to rotate in 45-degree steps.



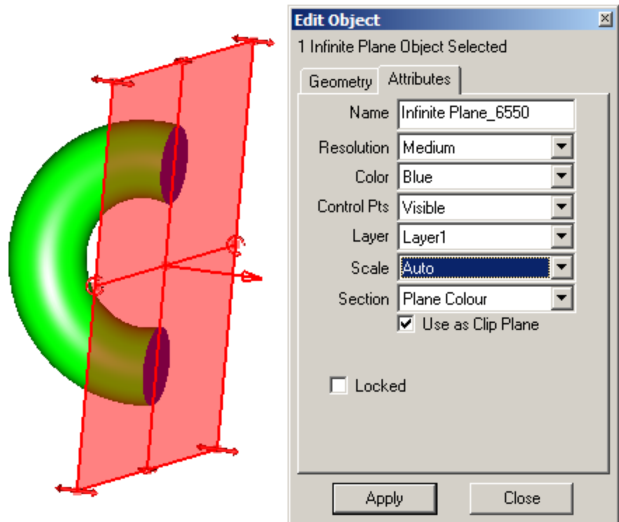
Corner markers pan the plane along the normal. The central marker moves the clip plane to any location.

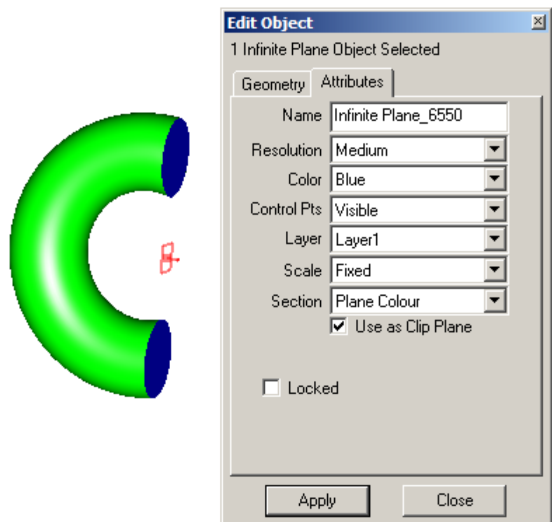


6. Edit the plane properties with the Edit Object dialog box. Double click the plane to open the Edit Objects. Use the *Scale* option to size the plane.

Auto The diagonal of the clip plane is equal to the length of the diagonal of an imaginary bounding box of all the objects to be clipped.

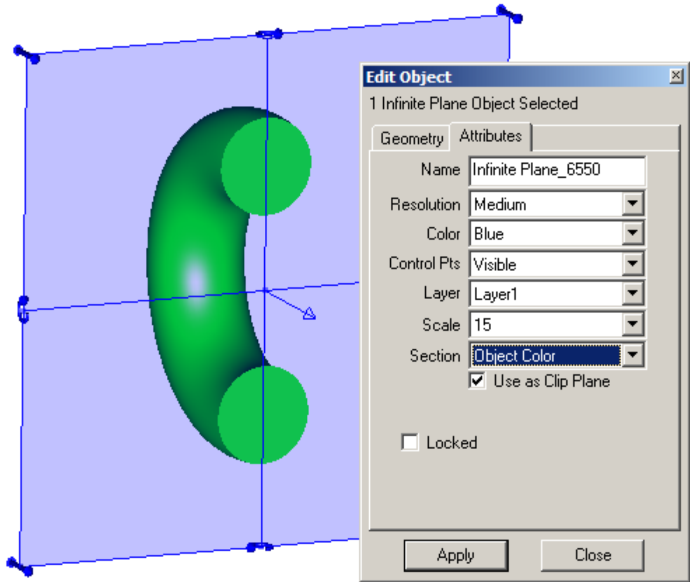
Fixed Shows just the plane object in the plane origin.





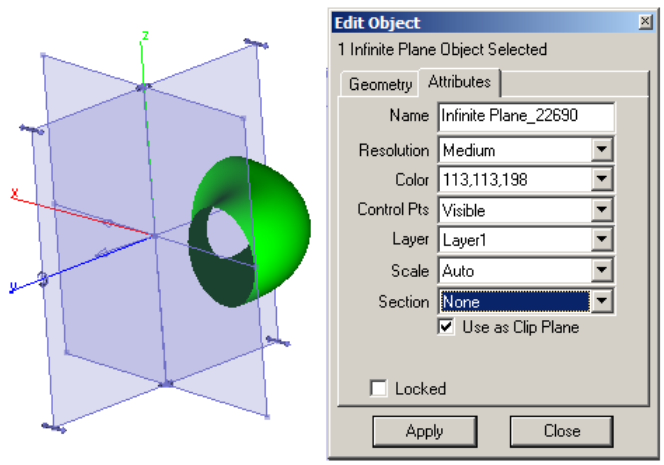
7. Modify the color of the clip plane cut with the Section drop down menu.

Object Color Makes the section the same color as the object, thus the clipped part looks like the solid object as in the picture below.

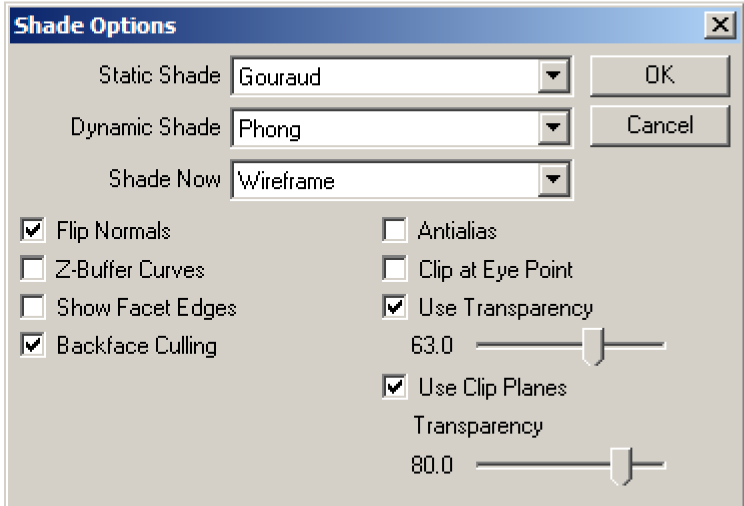


Plane Color Makes the section the same color as the plane.

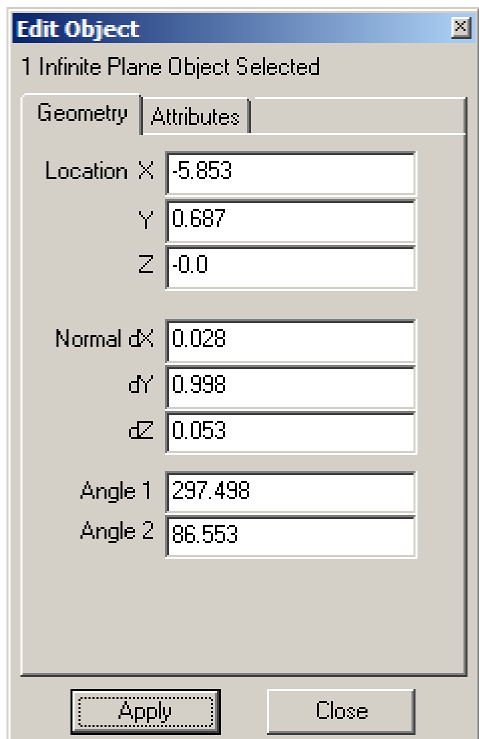
None The section area appears transparent.



8. Set the transparency of the clip plane in the ***View>Shade Options*** dialog.



9. The geometry tab of the Edit Object dialog box provides the following options:



Location Sets the location of the plane origin.

Normal Sets the direction of the normal.

Angle 1 Sets the turn angle along the X axis.

Angle 2 Sets the turn angle along the Y axis.

Apply Activates edited settings.

Set the options and click the Apply button to activate the settings.

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 on.

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 Surface Tool

This tool creates a sphere surface using a center and radius point.

Using the Sphere Surface Primitive Tool

1. Select the **Sphere Surface 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 Surface Tool

The Block Surface Primitive creates a cubic or rectangular shaped object with six surfaces in a group.

Using the Block Surface Primitive Tool

1. Select the Block Surface 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 clicked is the center point. This is the bottom center of the resulting block. The second point clicked controls not only the length, but the 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, it is possible to specify L and W for the length and width of the block.

Cylinder Surface Tool

This tool creates a cylindrical surface primitive.

Using the Cylinder Surface Primitive

1. Select the **Cylinder Surface 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 clicked is the cylinder base center point. The second point clicked controls not only the length, but the 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, specify D for the diameter of the cylinder.

Pyramid Surface Tool

This tool creates a pyramid surface.

Using the Pyramid Surface Primitive

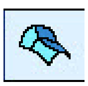
1. Select the Pyramid Surface Primitive tool. The Message Line reads: *Pyramid Primitive: Enter start and end points for pyramid*.
2. In the drawing area click the start and end points for the pyramid.

The first point clicked is the pyramid base center point. The second point clicked 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, specify D for the pyramid base diameter and the #Sides for the number of sides.

Surface From Curves Tools.

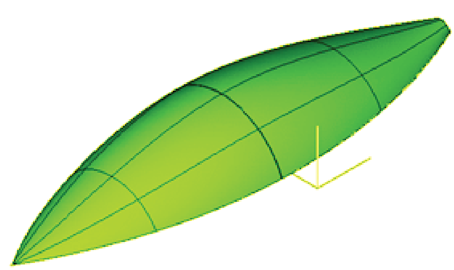
Net Surface Tool



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.

The curves may be in different planes. If the curves lie in the same plane, the Designer Elements program creates a simple planar surface within the curve boundary. If the curves lie out of plane, the 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.

It is also possible to specify a point entity for the start or end of the net surface, creating a degenerative surface.

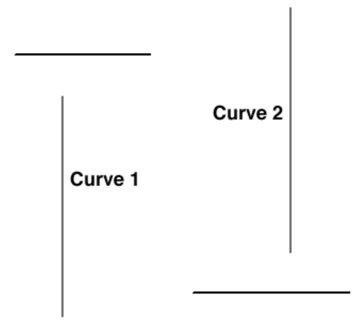


Tech Note: *A net surface is not the same as a mesh. Net surfaces are nurb surfaces. A mesh is defined by nodes or 3D vertices.*

Using the Net Surface Tool

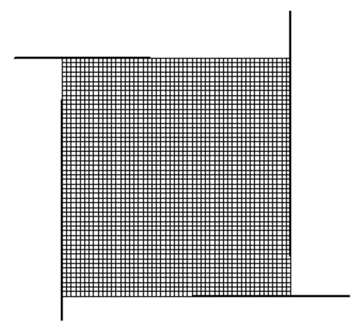
1. Select the **Net Surface** tool. The Message Line reads: *Net Surface: Pick (M) curves defining surface [Shift=Extend]*.
2. Hold down the SHIFT key and pick two or more curves along one direction. The objects appear selected as in the graphic here (Curves 1 and 2).

   When the SHIFT key is released, 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 the mouse button is released 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. The 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.

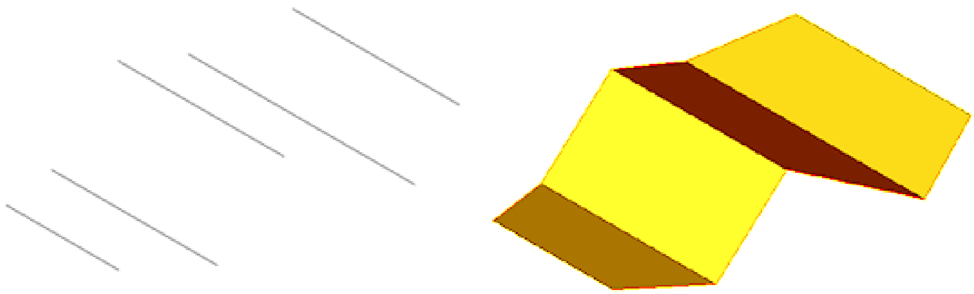
Display contains settings for Iso Lines and Silhouette. It is possible to display Iso Lines on the surface by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on the surface. There are options for the Silhouette setting, Smart, Off or On.

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 the curves. When the SHIFT key is released the surface is created



As the example shows, the curves defining the surface do not have to be connected. The Designer Elements program calculates the intersecting area of the selected curves and creates the ruled surface

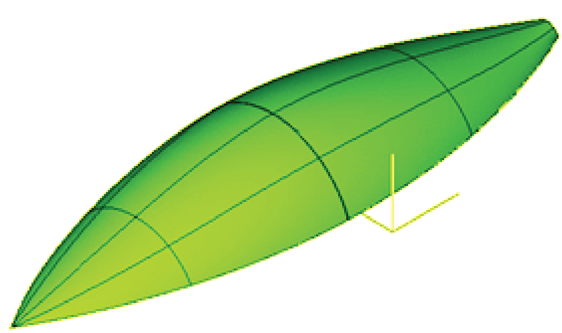
Geometric Characteristics

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. Display Iso Lines on the 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 the surface. There are options for the Silhouette setting, Smart, Off or On.

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. Create a standard skin surface or closed (tangent) skin surface. It is also possible to specify a point entity for the start or end of the surface, creating a degenerative surface.

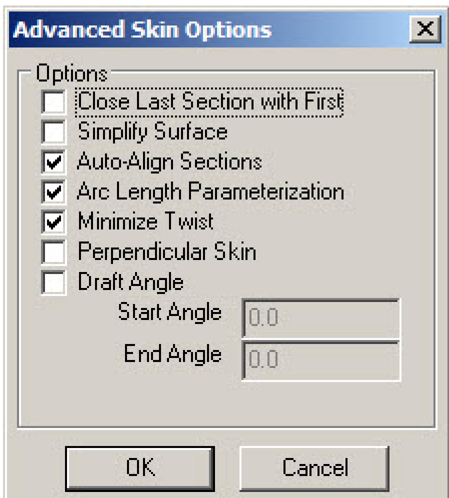


Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when creating a skin surface.

> ***Tech Note:*** *Do not confuse the* **Skin Surface** *tool with the* **Cover Surface** *tool. The curves selected for this tool are used as cross-sections. They cannot be connected or intersecting. A warning message will appear if invalid curves for the operation are chosen. To surface curves that are connected or closed use the* **Cover Surface** *tool.*

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 more control of how the 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 a set of closed profiles is provided, the face normals of the skin or loft body point outside, away from the body material. When a set of open profiles is provided, 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 profiles are not to be provided 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 may not be 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.

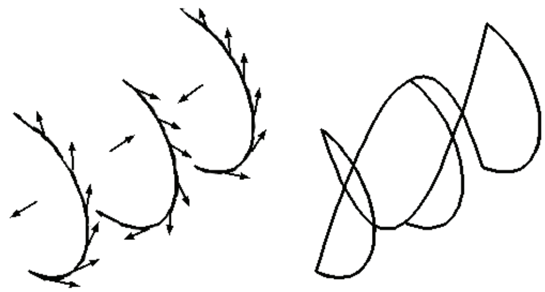
Draft Angle Skinning with draft angles provides the ability to control the take–off vectors of the two outer skinning profiles. The "draft" angle is defined as an angle off the plane of the wire at every point along the skinning profile. In addition to the user supplying the angle itself, one may also supply a magnitude for the take–off vector. The draft angle and magnitude is constant for the entire profile. However, different draft angles may be applied 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

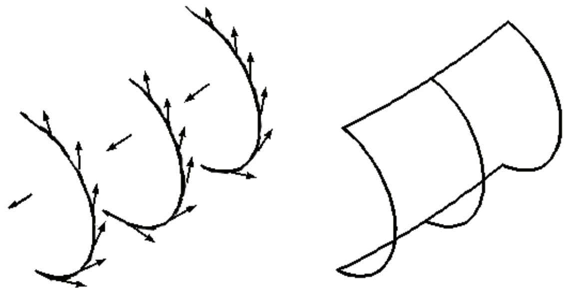
An open skin surface is a surface that has open ended in all directions. Select the curves in the order that they need to be skinned. When selecting the successive curves (or *n#* sections), create a surface between the curves. The individual curves are not surfaced. To surface curves connected end to end use the **Cover Surface** tool. See a later section in this chapter.

Tech Note: *The term, N# sections, refers to the curve sections selected to create a surface. The letter N is the generic identifier.*

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.



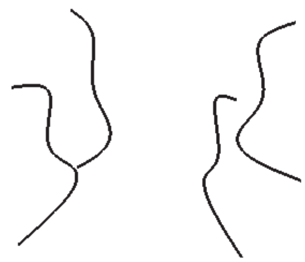
To avoid creating twisted surfaces, check the curves' directions by choosing ***Verify>Direction***. If the curves do run in different directions, adjust or redraw them before creating the surface. The graphic here shows curves with the same direction and the resulting surface.



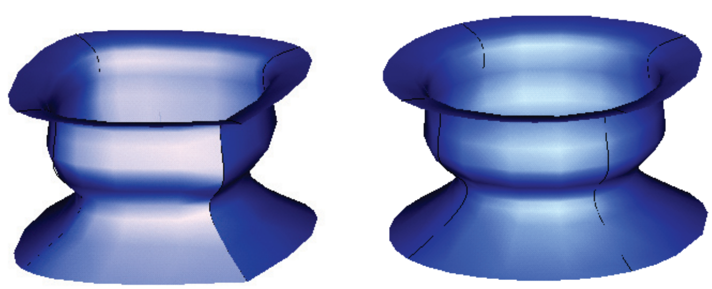
This tool also provides an option for creating closed tangents.

Closed Skin Surface

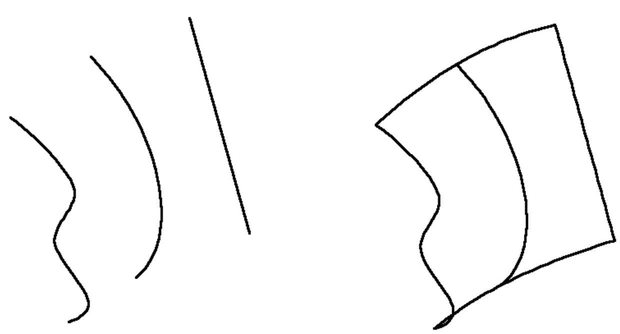
A closed or tangent skin surface is continuous end to end. When using this option, the Designer Elements program automatically closes the skin such that the first section equals the last section making the slopes along the edge tangent continuous. The graphic here shows spline curves used to create a 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. A standard skin surface or a closed skin surface can be created. 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. A selection fence can not be used 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. (The selection fence can not be used to select the curves.) After selecting 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. The selection fence can not be used to select the curves.

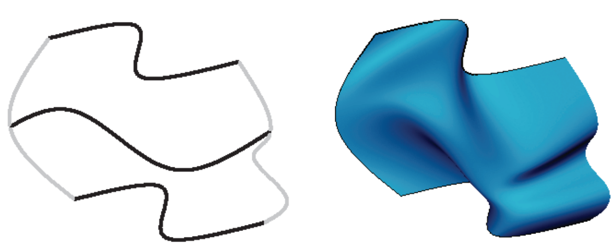
   The tangent surface is created.

   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 gives the control over the skin surface. 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 is to be used 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.

It is possible to 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. A selection fence can not be used 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. (The selection fence can not be used to select the curves.) After the last curve is selected 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. The selection fence can not be used 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

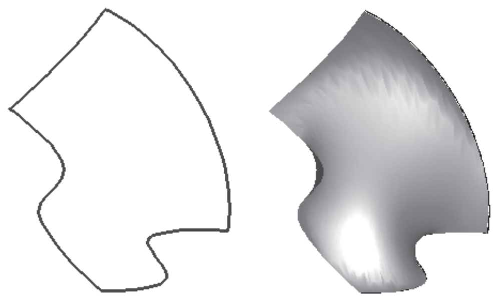
A skin surface is created by choosing curves as directed by the Message Line. A skin surface has no specific geometry characteristics according to the Edit Objects dialog box. There are tabs for both Display and Attributes.

Display contains settings for Iso Lines and Silhouette. Display Iso Lines on the 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 the surface. There are 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 this Designer Elements program creates a simple planar surface within the curve boundary. If the curves lie out of plane, this 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

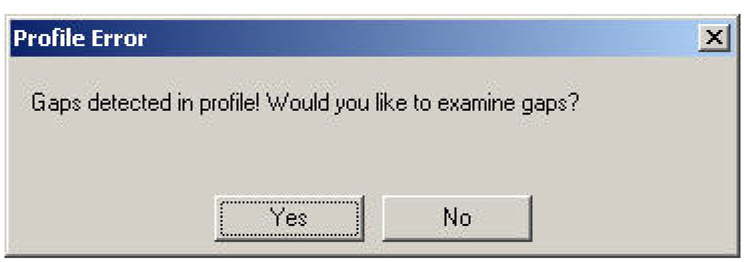
1. Select the **Cover Surface** tool. The Message Line reads: *Cover Surface: Pick curves for cover surface [Shift=Extend]*.
2. To select more than one curve hold down the SHIFT key before selecting the first curve.
3. Select the curves. Do not worry about the order in which they are selected.

   The cover surface is created.

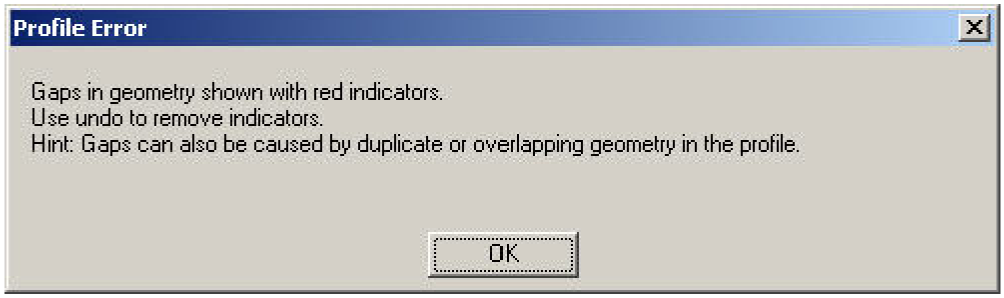
If not all of the curves selected are connected end to end, as shown in the figure below, the program warns with a Profile Error dialog box and marks the gaps in the selected curves.

If gaps are detected, two warning messages appear. The first one says, "Gaps detected in profile. Would you like to examine gaps?"

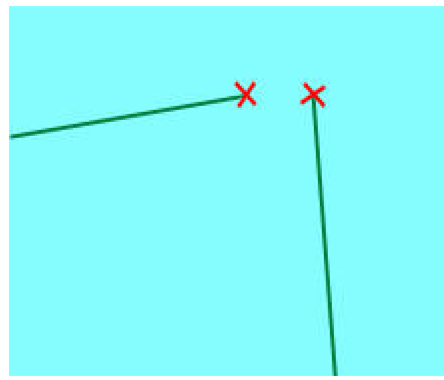
Click No to ignore the gaps and continue.



Click Yes, and another message appears: "Gaps in profile are shown with red indicators. Use Undo to remove indicators. Hint: gaps can also be caused by duplicate or overlapping geometry in the profile."



Click OK to remove the message and view the profile with red Xs where gaps are detected. After noting the gaps, use ***Edit>Undo*** to clear the indicators or drag a selection box around them and press Delete. The indicators are created as geometric points, adding geometry to the drawing.



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. Display Iso Lines on the 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 the surface. Options for the Silhouette setting include Smart, Off or On

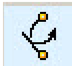
Revolved Surface Tool

The **Revolve Surface** tool creates a surface by revolving a set of curves (called the profile) around a central axis.

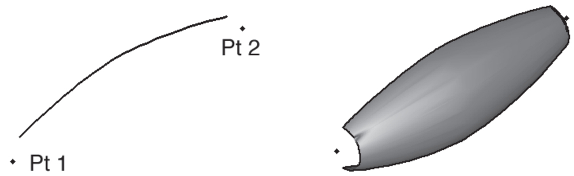
In the Message Line, there is a subpalette displaying two tools for revolving profiles: **Revolved Surface about Two Points** and **Revolved Surface about a Curve**.

Revolving curves connected at their endpoints the resulting revolution creates one surface.

Revolved Surface about Two Points Tools



This tool revolves a curve about two chosen points. The direction of the Angle of Revolution is determined by the order in which the points that define the axis are picked. Use the Right Hand Rule to determine the direction that the surface will revolve.



See Triad 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 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:

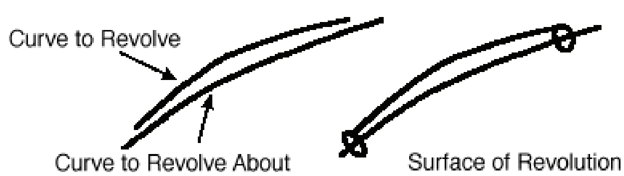
   Pick a new non-intersecting axis to complete the surface of revolution.

   The surface is created.

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 revolves a curve about an existing curve that is chosen. 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.

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

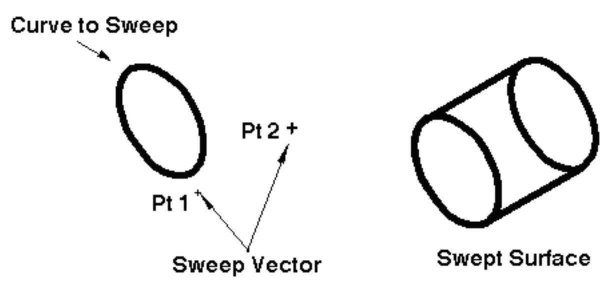
A revolved surface is created by choosing a curve to revolve and the axis of revolution. A revolved surface has the following geometry characteristics according to the Edit Objects dialog box: Rot. Angle, End 1 (X, Y and Z values) and End 2 (X, Y and Z values).

Display contains settings for Iso Lines and Silhouette. Display Iso Lines on the 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 the 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 the 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 chosen points. The resulting surface is associative to the original sweep curve. Change the curve and the surface automatically updates.



To check information on the sweep use commands in the Verify menu. Grouped curves can be sweeped with this tool.

> ***Tip:*** *To sweep a curve at an angle not perpendicular to the curve, change the view to specify the sweep points. Example: drawing a rectangle in the top plane, change the view to front and specify the points at the desired angle.*

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 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 the points are clicked specifies the direction of the sweep.

   The vector cannot run in the same direction as the curve. If a vector is defined in that direction this message appears:

   Select the curve again and click two other points to define the vector.

   The sweep surface is created.

The values in these data fields can be changed 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 made 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

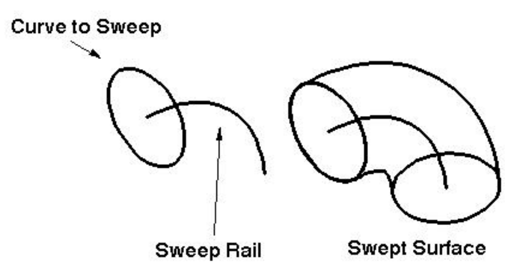
A swept surface is created by choosing a curve to sweep and the points for the two points defining the vector. A swept surface along two points has the following geometry characteristics according to the Edit Objects dialog box: Distance and Vector (dX, dY and dZ values).

Display contains settings for Iso Lines and Silhouette. Display Iso Lines on the 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 the surface. There are options for the Silhouette setting, Smart, Off or On.

Sweep Surface Tool - Along 1 Rail

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. Only one curve can be sweeped 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.

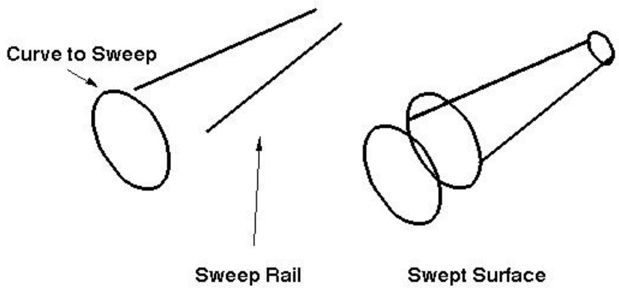
Geometric Characteristics

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.

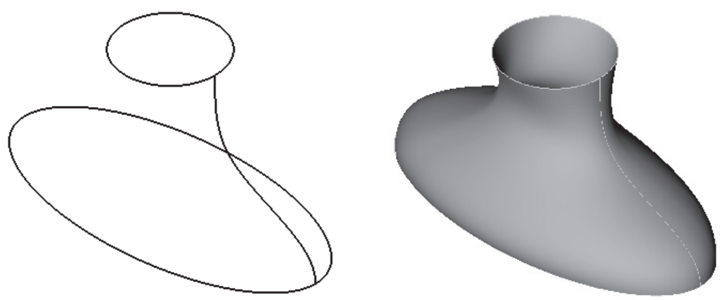
Display contains settings for Iso lines and Silhouette. Iso Lines can be displayed on the surface by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on the surface. There are options for the Silhouette setting, Smart, Off or On.

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, it is also possible to 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 to sweep and then choose the two rails for the sweep path.

   A swept surface is created.

   Hold down the Ctrl or Option key to maintaine the profile height.

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.

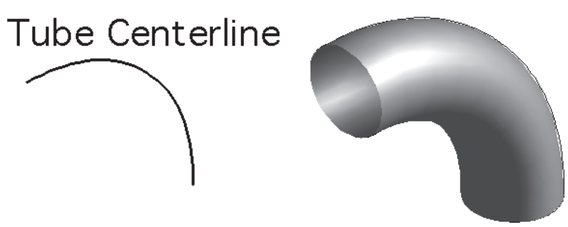
Display contains settings for Iso Lines and Silhouette. Iso Lines can be displayed on the surface by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on the surface. There are options for the Silhouette setting, Smart, Off or On.

Sweep Rules

- Double-click the surface to display the Edit Objects dialog box. Picking 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.
- 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 sweped curves are connected at their endpoints, the resulting sweep creates one surface.

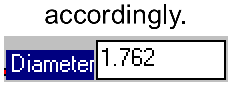
Tube Surface Tool

Tube Surface tool quickly and efficiently creates a tube or pipe along a reference curve. This tool support grouped curves. A radius must be specified 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.



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.

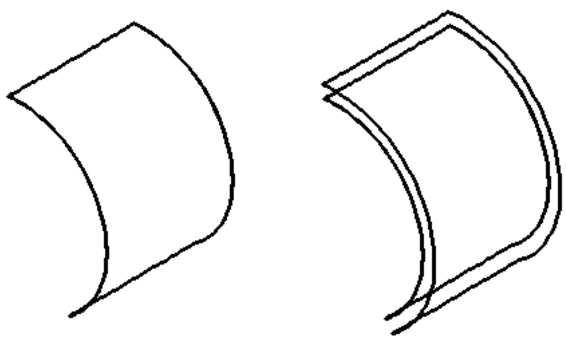
Display contains settings for Iso Lines and Silhouette. Iso Lines can be displayed on the surface by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on the surface. There are options for the Silhouette setting, Smart, Off or On.

Derived Surface Tools

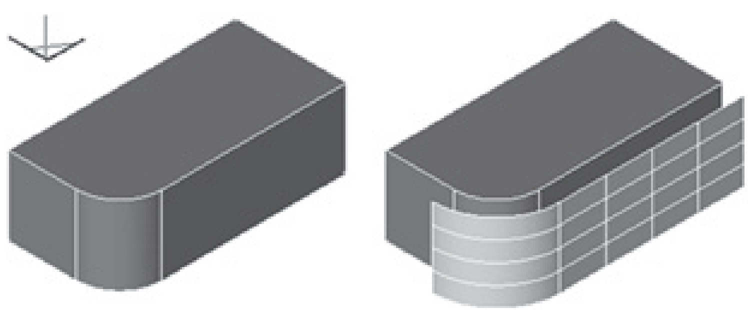
Offset Surface Tool

Offset Surface tool makes a new surface based on an existing or parent surface, offset by a chosen amount. An infinite plane can be also offset.

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. It is possible to specify negative offsets. Do not offset a surface a distance greater than the smallest radius of curvature or a degenerate surface can be created.



If a zero offset is specified, the Designer Elements program creates an associative copy of the selected surface at the same location.



it is also possible to offset solids faces. 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.

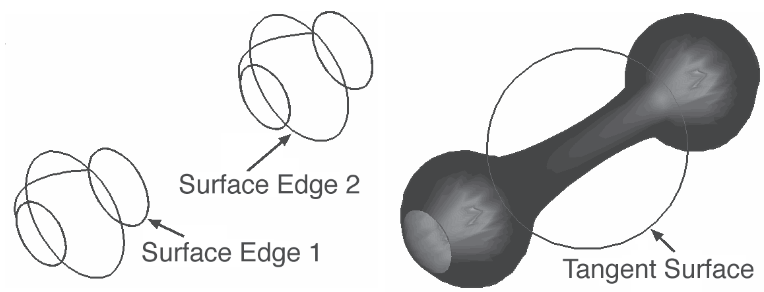
Geometric Characteristics

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. Display Iso Lines on the 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 the surface. There are options for the Silhouette setting, Smart, Off or On.

Lofted Surface Tool

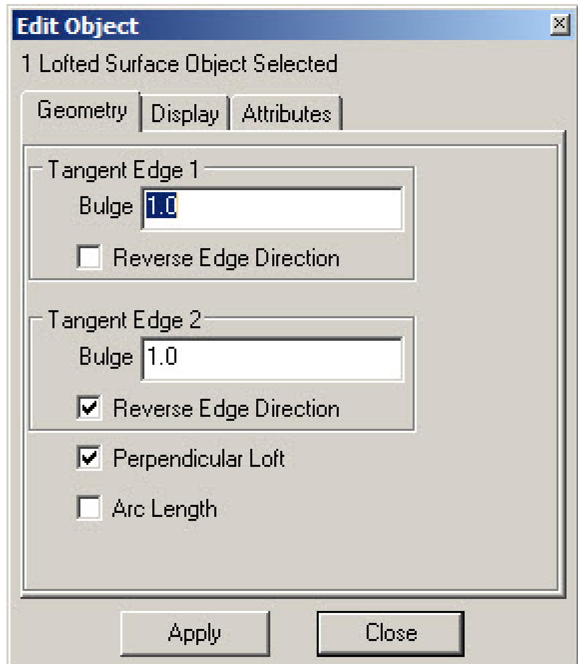
The **Lofted Surface** tool creates a smooth, transitional surface between surfaces edges and any number of interior non-surface edges. To create a tangent surface select the edge of the start surface and the edge of the end surface.



Using the Lofted Surface Tool

1. Select the **Lofted Surface** tool. The Message Line reads: *Lofted Surface: Pick edges for lofted surface.[Shift=Extend]*.

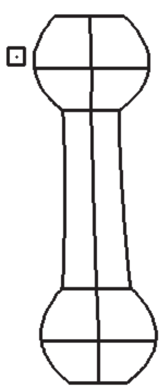
2. Enter a new bulge factor in the Bulge data field in the Status Line, if desired. The bulge factor controls the magnitude of the slope of the surface edges.



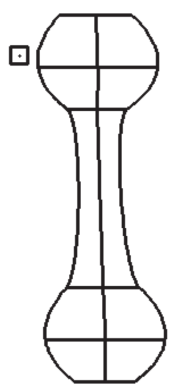
3. Click an edge on each surface to be joined by the tangent surface. When a tangent surface is created, the screen shows only one connecting line. Choose ***View>Shade Now*** to see the tangent surface. If *Shade Now* is set to anything other than wireframe in the Shade Options dialog box, the lofted 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.

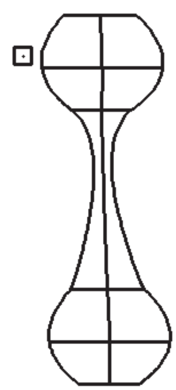
The figure shows two bulb surfaces joined with a tangent surface using different bulge factors. The left figure uses a 0.25 bulge factor; the center figure uses a 10.0 bulge factor; the right figure uses a 3.0 start bulge factor and a 0.5 end bulge factor.



A bulge factor of .25 is used at both ends.

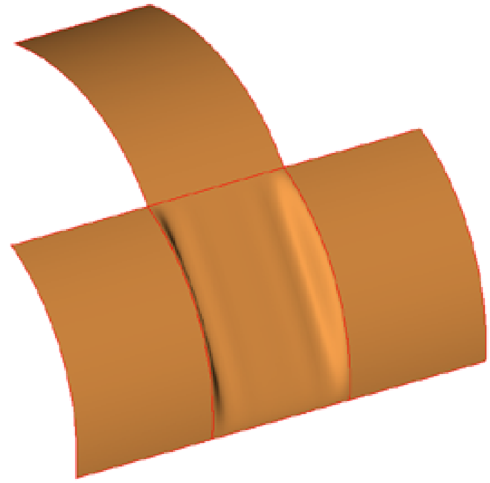


A bulge factor of 10 is used at both ends.



A bulge factor of 3 is used at the top end and 0.50 factor at the bottom end.

Geometric Characteristics



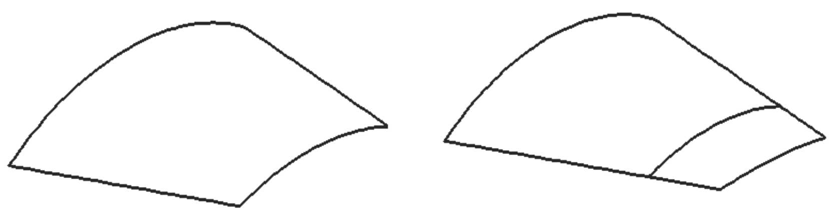
A lofted surface is created by choosing the two connecting surfaces. The bulge factor for the lofted surface edges can be edited in the Edit Objects dialog box. Tangent Edge 1 controls the first edge chosen and Tangent Edge 2 controls the last edge chosen.

Display contains settings for Iso Lines and Silhouette. Display Iso Lines on the 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 the surface. There are options for the Silhouette setting, Smart, Off or On.

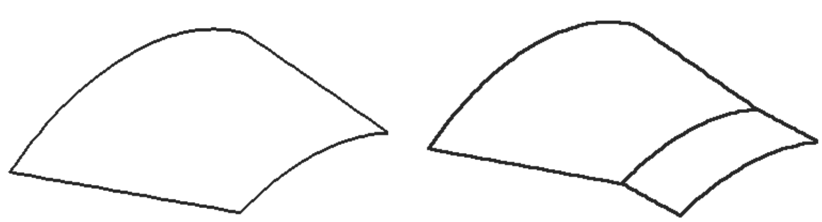
Draft Surface Tool

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 the 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.

There are 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), choose the *Align Curve* option listed in the Message Line. When using 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 necessary.

3. Click a curve (edge) on the surface to draft or extend the surface. When the edge is seleced, the whole surface is highlighted until this Designer Elements program creates the draft.

Both the angle and the length of the Draft can be changed 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

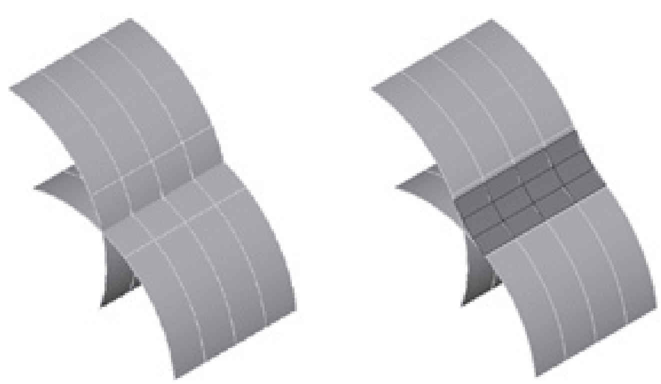
A draft surface is created by choosing a curve to draft and entering the desired angle and length of the draft. A draft surface has the following geometry characteristics according to the Edit Objects dialog box: Angle and Length.

Display contains settings for Iso Lines and Silhouette. Display Iso Lines on the 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 the surface. There are 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***. 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.

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.

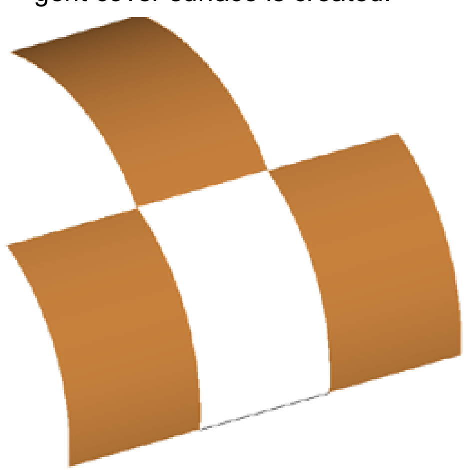
The Display page of Edit Objects contains settings for Iso Lines and Silhouette. Display Iso Lines on the surface by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on the surface. There are options for the Silhouette setting, Smart, Off or On.

Tangent Cover Tool

The **Tangent Cover** surfaced creates a surface that is tangent to four neighboring surfaces. The surfaces must form a closure. Curves can be selected for some of the boundaries however if there is a choice, use surface edges for the boundary of the tangent cover surface.

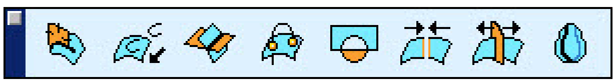
Using the Tangent Cover Surface Tool

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.

Surface Utility Tools



Cobalt, Xenon and Argon all provide several editing tools in the **Surface Editing** tool palettes. The tools 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.

Surface Editing Tools

Many of the editing tools require a two step process for their use. The first step after choosing the tool may be to select one or more objects. The second step performs the editing operation. Once the first step is completed other objects cannot be selected 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 select the objects for performing an operation. When selecting the tool and moving the cursor into the drawing area it becomes a hollow selection arrow (shown here).

After finishing selection the pointer becomes a general tool pointer (shown here.)

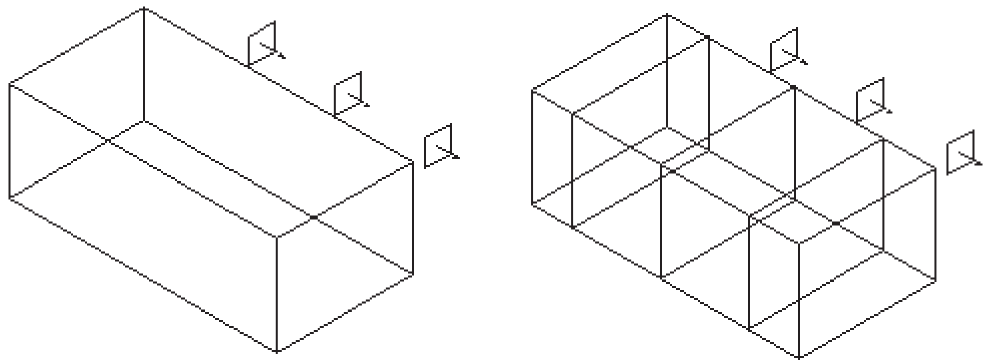
This hollow selection arrow must not be confused with the **Selection** tool.

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 either entity is modified, the cut will update (**Cobalt and Xenon only**).

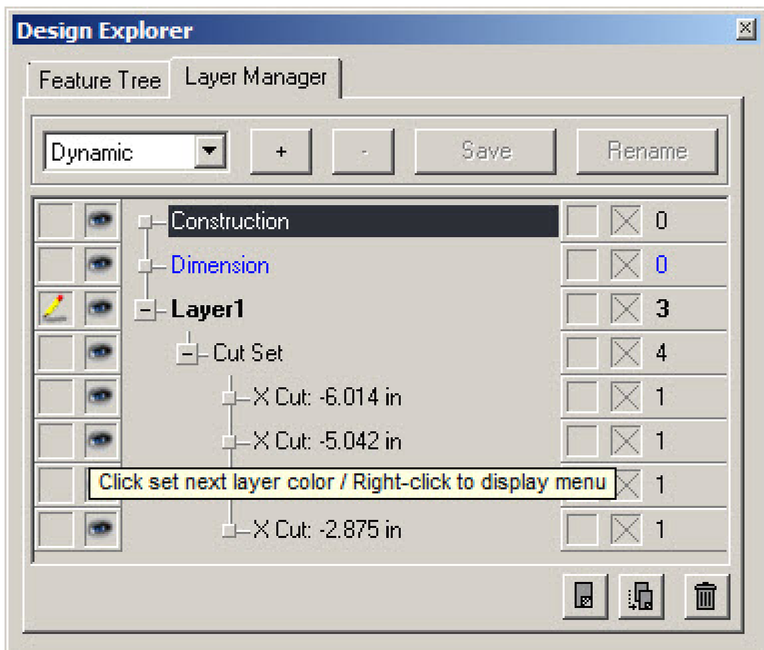
Auto Layers Option

This tool provides an *Auto Layer* option in the Message Line. When this option is enabled, the 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.

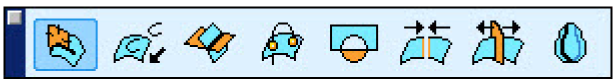


Tip: When the Auto Layers option is used with the Increment Layer command (ALT+0 - Windows, z+0 - Macintosh) and Decrement Layer (ALT+9-Windows, z+9-Macintosh), to change layers, it provides a convenient way of examining every 2D cut of a 3D model.

Intersecting a Plane with a Curve

When a plane intersects with a curve, a point intersection results. The left graphic below 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. [Ctrl = Auto Layers] [Shift = Extend]*



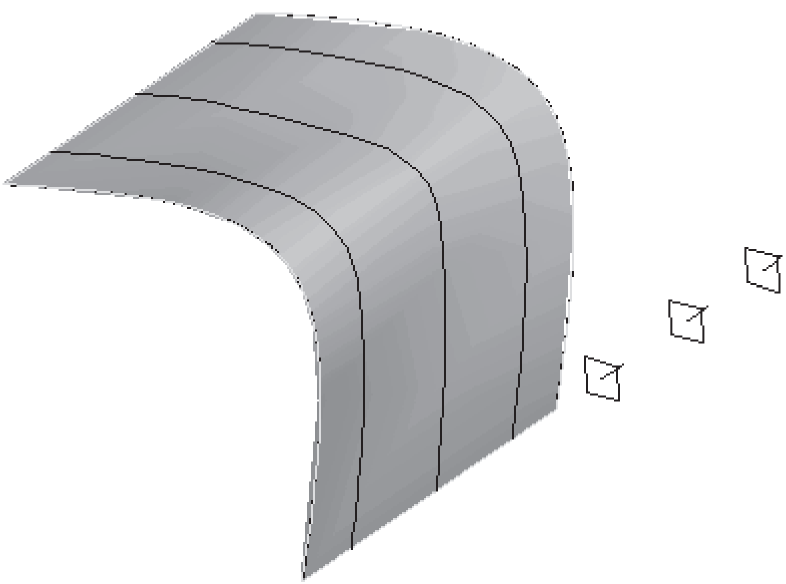
2. Click the plane to use for intersecting the curve. Hold down the SHIFT key to pick more than one plane. The Message Line reads: *Plane/Surface Intersection: Pick objects to intersect with plane. [Shift=Extend]*
3. Click the curves to intersect with the plane.

   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. **(Cobalt and Xenon only)**

There are no entries in the Status Line.

Intersecting a Plane with a Surface

When a plane intersects 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. [Ctrl = Auto Layers] [Shift=Extend]*

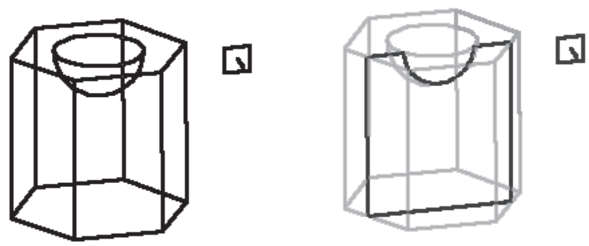
2. Click the plane to use to intersect the surface. Hold down the SHIFT key to pick more than one plane. The Message Line reads: *Plane/Surface Intersection: Pick objects to intersect with plane. [Shift=Extend]*
3. Click the surface to intersect with the plane.

   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 a plane is intersected 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. [Ctrl = auto Layers] [Shift=Extend]*.
2. Click the plane 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 to intersect with the plane.

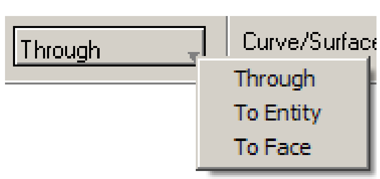
   The intersection curve displays. The curve is associative to the plane and solid. Modifying either the plane or the solid automatically updates the location of the curve (**Cobalt and Xenon only**).

There are no entries in the Status Line.

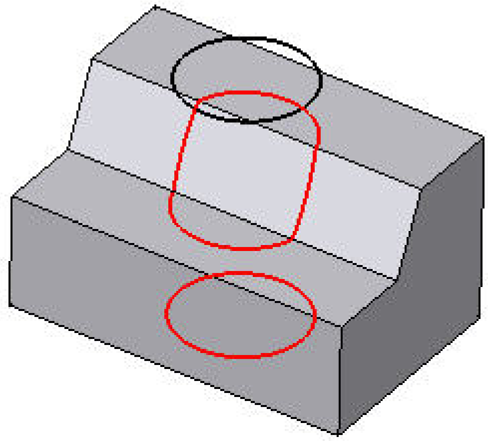
Curve/Surface Projection Tool

The **Curve/Surface Projection** tool supports projecting curves onto surfaces and solids along a direction vector. Specify the direction vector by clicking two points. The new curve is associative to the object. When modifying the object the projected curve will automatically update (**Cobalt and Xenon only**).

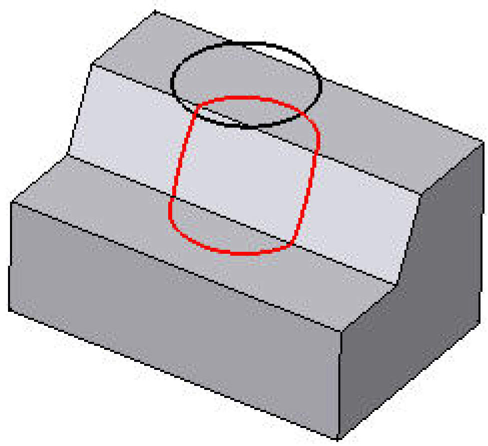
Select the **Curve/Surface Projection** tool from the **Surface Utilities** tool palette and these options appear in the Message Line:



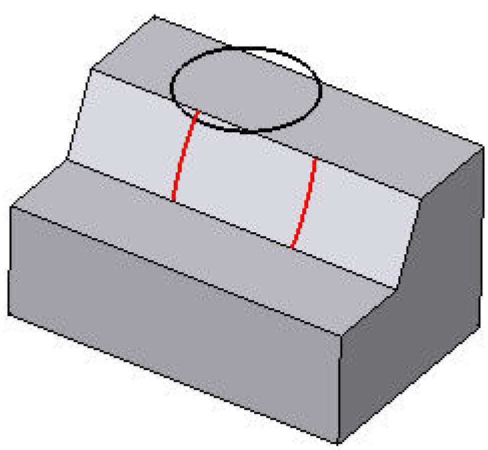
Through Projects a curve on a surface that intersects the entire object.



To Entity Projects a curve where it first touches the object.

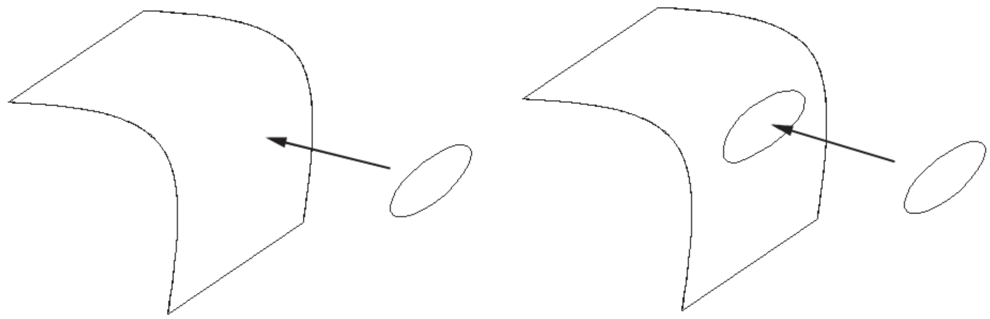


To Face Projects a curve only on the specified face.



Projecting a Curve to a Surface

With this tool a curve projects onto a surface. If the curve is a spline, it is a vector spline capable of being edited.



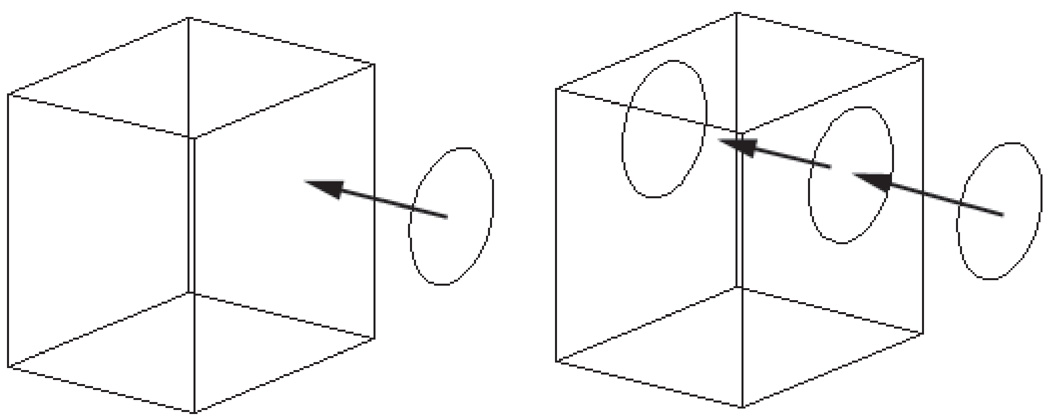
1. Select the tool in the tool palette. Choose the necessary option in the Message Line. The Message Line reads: *Curve/Surface Projection: Pick curves to project onto surface. [Ctrl = Imprint] [Shift=Extend]*
2. Click the curve to project onto the surface (it does not highlight). Hold down the SHIFT 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 to project the curve.
4. Enter two points for the projection vector.

   The curve projects onto the surface.

There are no entries in the Status Line.

Projecting a Curve to a Solid

This tool projects 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. Select the necessary option in the Message Line. The Message Line reads: *Curve/Surface Projection: Pick curves to project onto surface. [Ctrl = Imprint] [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 to project the curve.
4. Enter two points for the projection vector.

   The curve projects onto the solid.

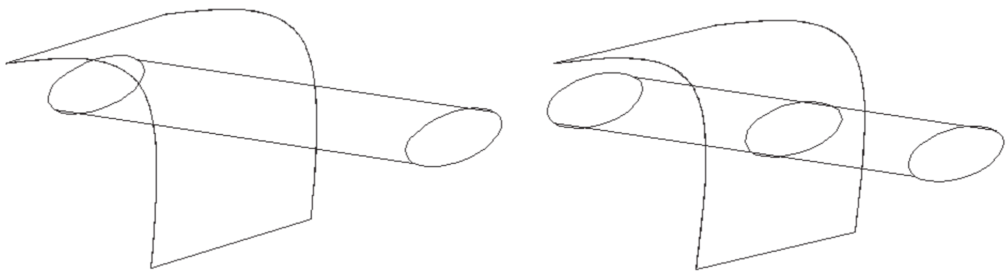
There are no entries in the Status Line.

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 changing 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.

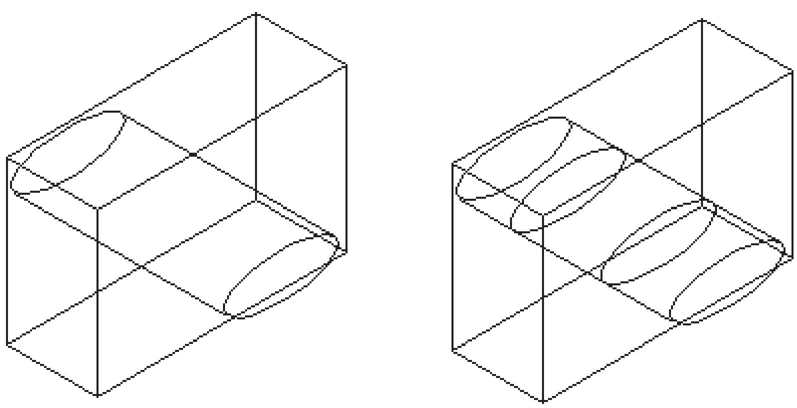


1. Select the tool in the tool palette. The Message Line reads: *Surface/Surface Intersection: Pick two surfaces or solids for intersection*.
2. Click on the two surfaces. An intersection curve displays.

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*.
2. Click the two surfaces. An intersection curve displays.

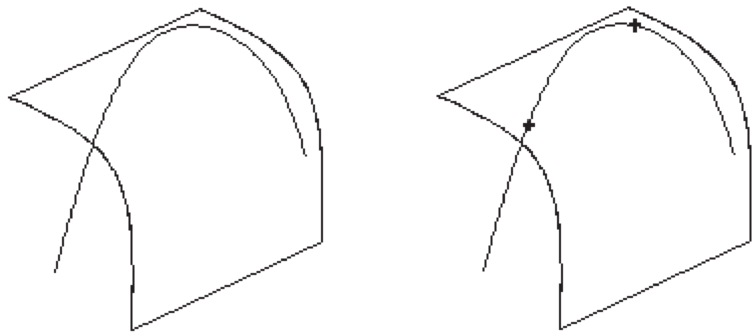
There are no entries in the Status Line.

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

This tool intersects a curve with a surface.

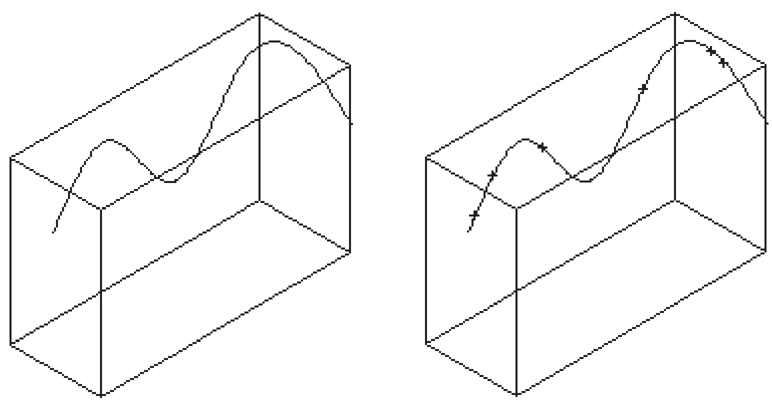


1. Select the tool in the tool palette. The Message Line reads: *Curve/Surface Intersection: Select curves to intersect surface/solids. [Shift=Extend]*
2. Click the curve to intersect the surface. Hold down the SHIFT key to pick more than one curve. The Message Line reads: *Curve/Surface Intersection: 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

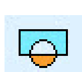
Intersect a curve with a solid with this tool.



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.

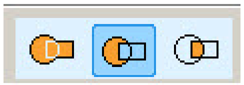
There are no entries in the Status Line.

Boolean Surface Tools



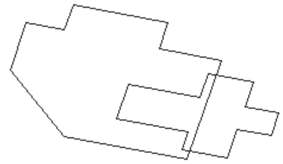
Cobalt, Xenon and Argon all support 2D boolean operations on planar surfaces created with the cover surface command. The **Boolean Surface** tool adds surfaces together, subtracts surfaces from one another, and finds the intersection of two or more surfaces.

Use the new curves created from boolean operations for 2D property analysis or as profiles for solid extrusions, sweeps and lathe operations.

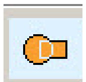


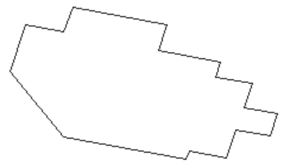
There are three tools for performing boolean operations: **Add Boolean Surface**, **Subtract Boolean Surface** and **Intersect Boolean Surface**.

For the purposes of explaining these tools the objects to the right are used.



Add Boolean Surface Tool





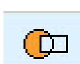
This tool adds two objects to create a third object.

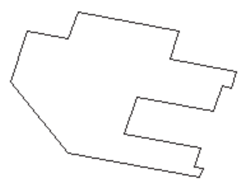
1. Select the **Boolean Surface** tool.
2. Select the **Add Boolean Surface** tool in the Message Line. The Message Line reads: *Boolean Surface: Pick planar surface to add to. [Shift=Extend]*
3. Select the surface to which another will be added. The Message Line reads: *Boolean Surface: Pick planar surface to add. [Shift=Extend]*
4. Select the surface to be added.

There are no entries in the Status Line.

> ***Tip:*** *To get volume, density and other properties of geometry in the drawing, combine them with this tool whether or not they are connected.*

Subtract Boolean Surface Tool



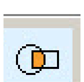


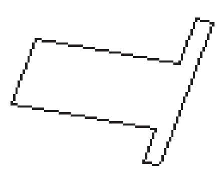
This tool subtracts one surface from another.

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





This tool retains the intersecting portion of two surfaces.

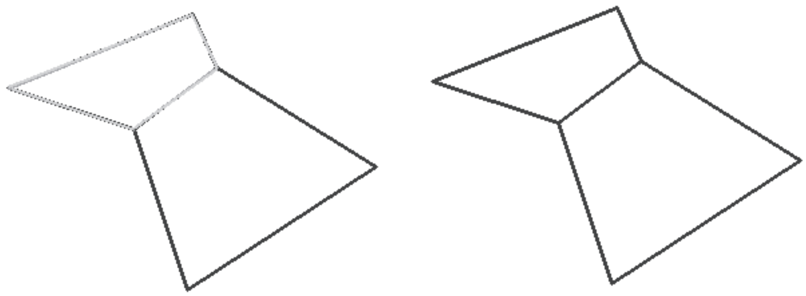
1. Select the **Boolean Surface** tool.
2. Select the **Intersect Boolean Surface** tool in the Message Line. The Message Line reads: *Boolean Surface: Pick planar surface to intersect with. [Shift=Extend]*

3. Select the surface with to 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

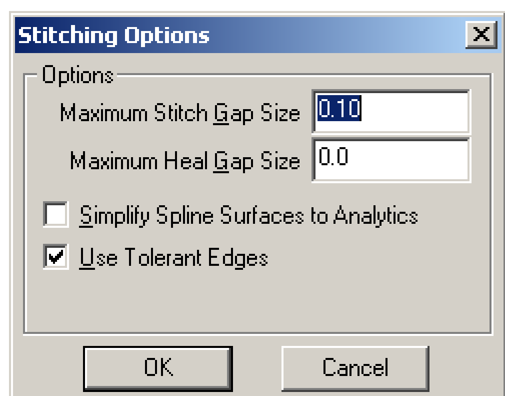
The **Connect Surface** tool merges two surfaces into one. The resulting surface retains the original surfaces mathematical definitions, but a selection operation treats both surfaces as one. Unlike the **Add Boolean Surface** tool the **Connect Surface** tool can be used with non-planar surfaces. The 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. [CTRL = Advanced (Windows) or OPTION = Advanced (Macintosh)] [Shift=Extend]*.
2. For advanced settings tap the CTRL key (Windows) or the OPTION key (Macintosh) to display the Stitching Options dialog box to enter the maximum stitch and heal gap sizes, simplify spline surfaces or use tolerant edges.



3. Click the two surfaces to connect. The surfaces become one surface but retain their individual mathematical definitions.

There are no entries in the Status Line.

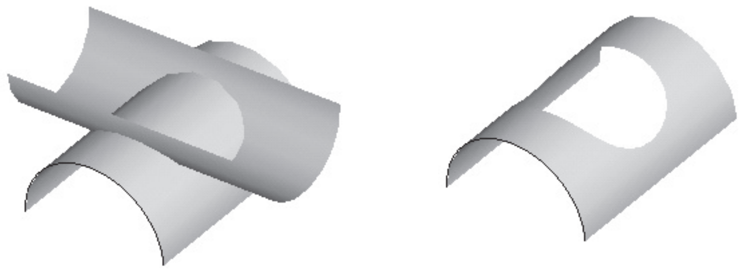
Split Surface Tool

Use the **Split Surface** tool to cut away portions of a surface. It is possible to 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.

Tech Note: *The* **Split Surface** *tool will not work with grouped surfaces.*

Using the Tool to Split a Surface with a Surface

Use this tool to trim a surface to a surface.

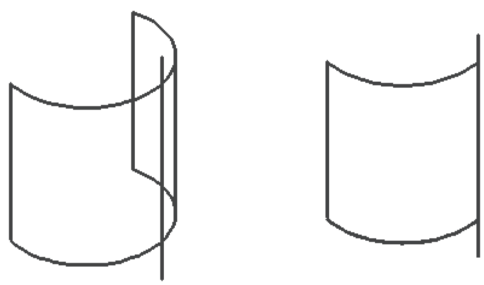


1. Select the **Split Surface** tool in the tool palette. The Message Line reads: *Split Surface: Pick surface to split*.
2. Select the surface to split. The Message Line reads: *Split Surface: Pick curves, surface, or solid to split with*.
3. Select the entity that defines the trim area. The program splits the surface at the intersection of the two entities. The objects won't appear differently on the screen. Delete any unnecessary portions.

There are no entries in the Status Line.

Using the Tool to Split the Surface to a Curve

Use this tool to trim a surface to a curve.

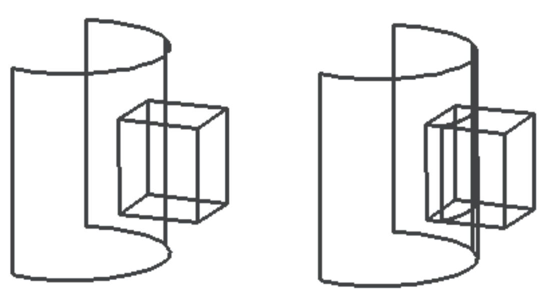


1. Select the **Split Surface** tool in the tool palette. The Message Line reads: *Split Surface: Pick surface to split*.
2. Select the surface to trim. The Message Line reads: *Split Surface: Pick curves, surface, or solid to split with. [Shift=Extend]*
3. Select the entity that defines the trim area. The program splits the surface at the intersection of the surface and curve. The objects won't appear differently on the screen. Delete any unnecessary portions.

There are no entries in the Status Line.

Using the Tool to Split a Surface with a Solid

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*.
2. Select the surface to split. The Message Line reads: *Split Surface: Pick curves, surface, or solid to split with*.

3. Select the entity that defines the split area. The program splits the surface at the intersection of the surface and the solid. The objects won't appear differently on the screen. Delete any unnecessary portions.

There are no entries in the Status Line.

Silhouette Curves Tool

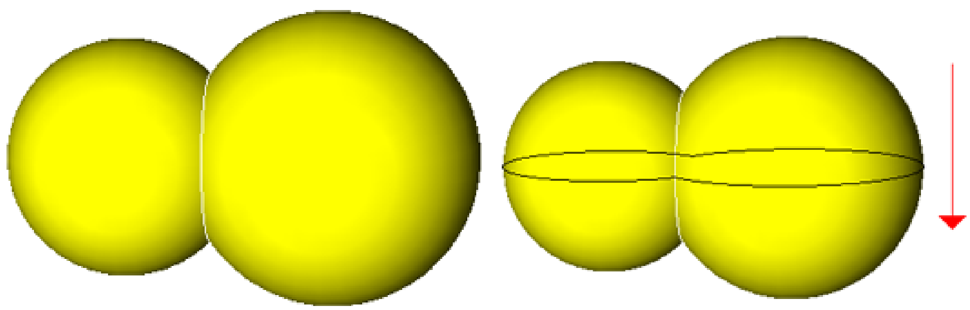
The **Silhouette Curves** tool creates curves from the bounding edge of an object relative to a user-defined projection direction. The curves produced by this tool are, in most cases, non-planar. This tool is useful for producing injection mold drawings.

Using the Silhouette Curves Tool

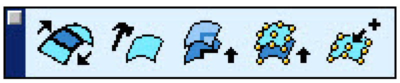
1. Select the Silhouette Curves tool, The Message Line reads: *Silhouette Curves: Pick surface/solids for silhouettes. [Shift = Extend]*

2. Select the objects on which silhouette curves are necessary. 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.



Surface Modification Tools



Match Surface Tool

Cobalt, Xenon and Argon provide tools for modifying and evaluation the local faces of surfaces.

This tool imposes a smooth transition from one surface to another neighboring surface. It also performs greater surface manipulation by inserting a knot and elevating the degree of the surface.

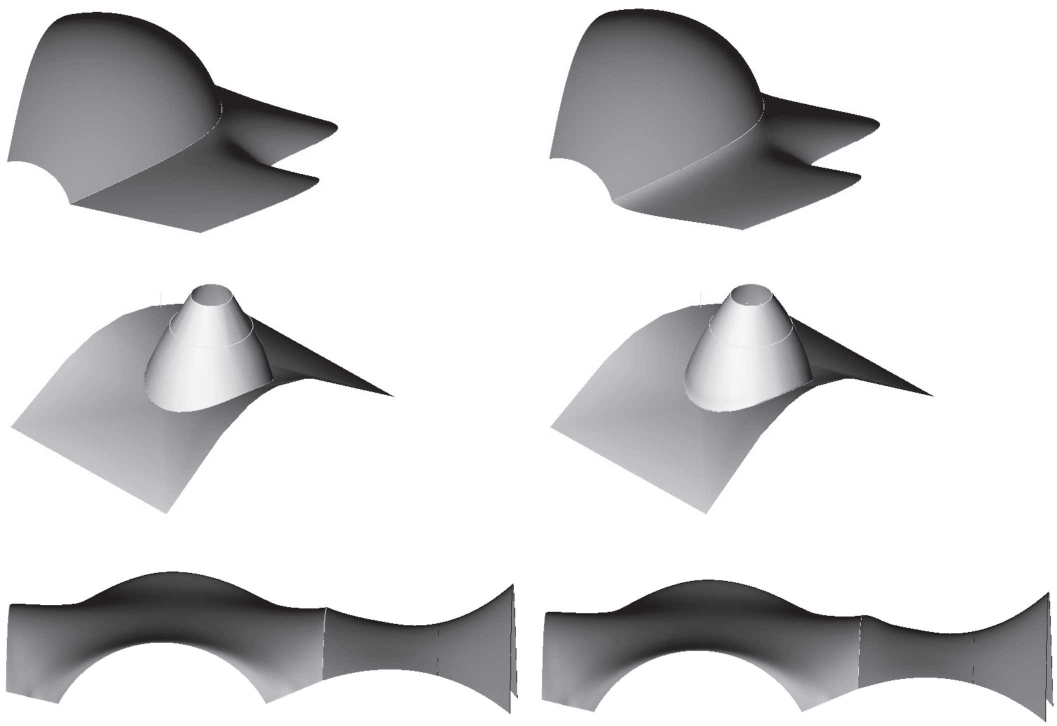
There are two options available for this tool: *Match G1 Slope and Match G2 Slope*, *Insert Knot*. These are accessed through the pull-down menu in the Message Line.

Rules and Notations for using the Match Surface Tool and Options

- Surfaces cannot be trimmed. (Any surface that has had a relimit or another editing operation conducted on it is a trimmed surface.) To match untrimmed surfaces, first untrim them using the *Untrim Surface* tool before matching them. See a later section about this.
- 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 Surface* tool before matching the surfaces. See a later section about this tool.

Match G1 Slope Option

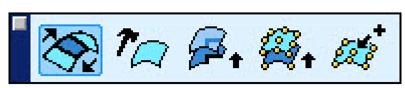
Use this option to match two untrimmed surfaces with G1 continuity. G1 matched surfaces are tangent continuous at their shared edge. The graphics on the left below are the two original surfaces. The graphics on the right below are the same surfaces using the *G1 Slope* option



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 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 the first surface is being matched.

   The slope of the first surface edge now matches the second surface edge with a G1 continuity. Verify this continuity by using the surface evaluation feature. In a surface evaluation plot, stripes represent the slope of the surface.

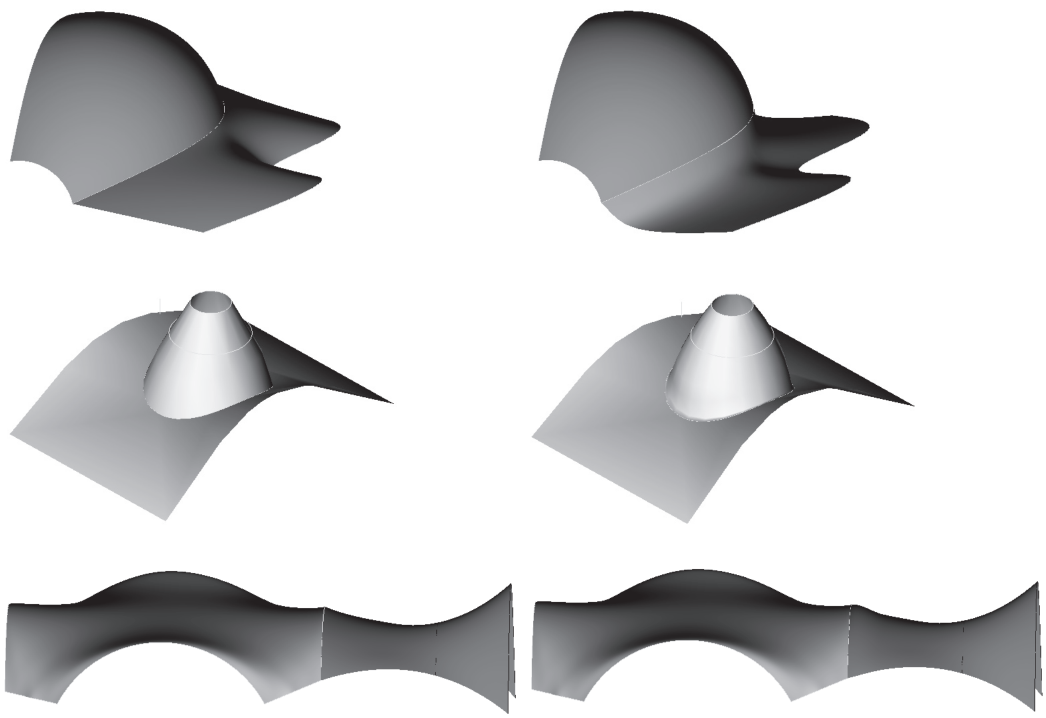
   These stripes line up from the first surface to the second surface for G1 continuous surfaces.

   See the Surface Evaluation section at the end of this chapter for information.

   There are no entries in the Status Line.

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.



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.*

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 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 the first surface is being matched.

   The slope of the first surface edge now matches the second surface edge with a G2 continuity. 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.

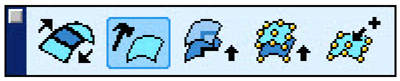
   See the Surface Evaluation section at the end of this chapter for information.

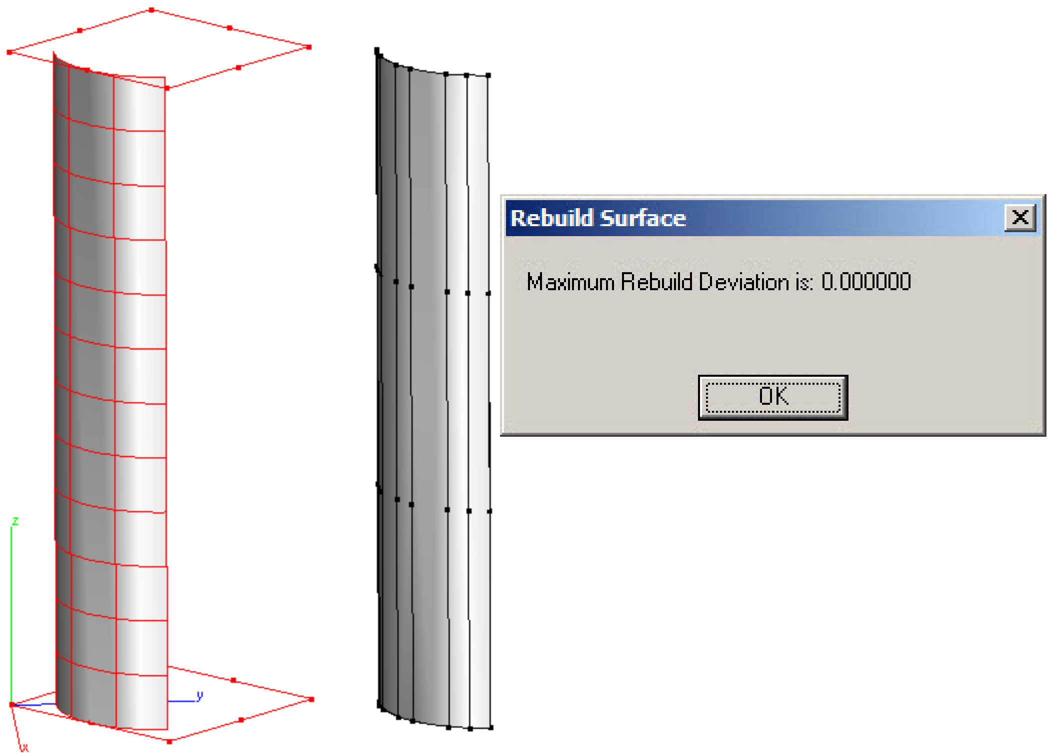
There are no entries in the Status Line.

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.





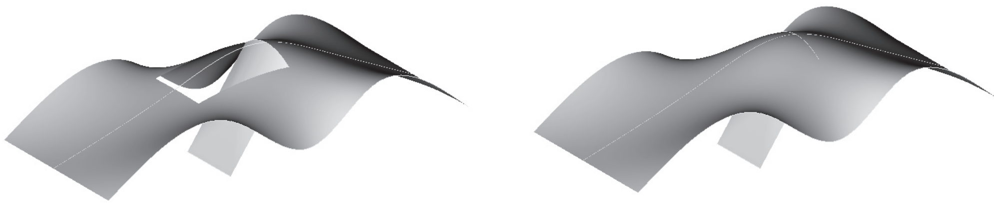
Using the Rebuild Surface Tool

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. 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.*

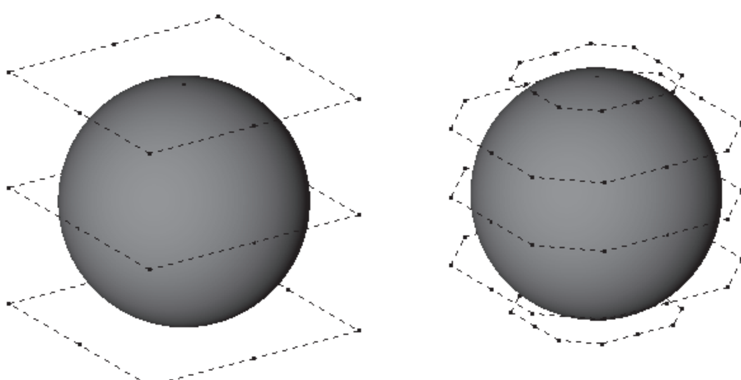
   The surface untrims.

There are no entries in the Status Line.

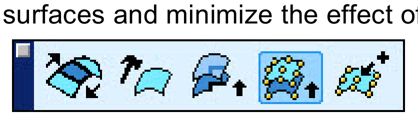
Elevate Surface Tool

This tool 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 not to 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.

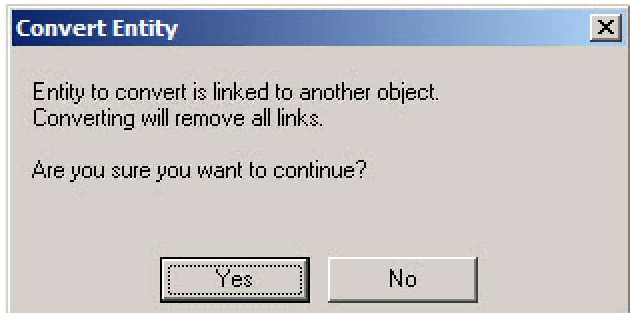


Using the Elevate Surface Tool

1. Select the surface for which to add control points.
2. Choose ***Window>Edit Objects*** to verify that the surface is a surface face.

   If the 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 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.

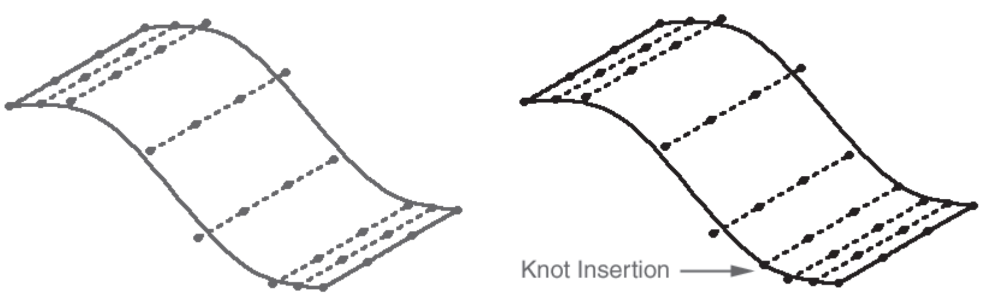
   The surface elevates to the next degree.

There are no entries in the Status Line.

Insert Knot Tool

This tool adds 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. Verify the surface type by double-clicking on the surface to display the Edit Objects dialog box.

Knot insertion is valuable 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, other control points may have shifted. This is automatically done to preserve the surface shape.

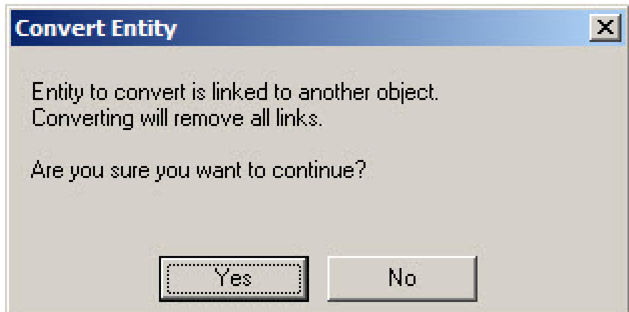
There are two options in the Insert Knot tool: At Location and Across Surface. At Location inserts one row or column of knots near designated spot, depending on the edge selected. Across Surface inserts multiple rows and columns of knots equally spaced across a surface.

Using the Insert Knot Tool - At Location Option

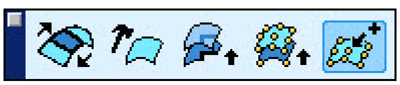
1. Select the surface to which control points are being added.
2. Choose ***Window>Edit Objects*** to verify that the surface is a surface face.

   If it is continue on with step 5.
3. Choose ***Edit>Change Object Type***.
4. Select the Surface option and click OK.

   If the selected surface 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 **At Location**. 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. To add another row or column of knots select the surface again.

There are no entries in the Status Line.

Using the Insert Knot Tool - Across Surface Option

1. Select the surface to which control points are being added.
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 selected surface 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 **Across Surface**.
8. In the Status Line enter the number of rows and columns of knots to be inserted across the surface in both directions.
9. The Message Line reads: Insert Knot: Pick surface to insert a new knot. [SHIFT = Extend]

   Select the surface.

There Status Line contains the Number of Knots data field specifying the number of rows or columns of knots in both directions.

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 the 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.

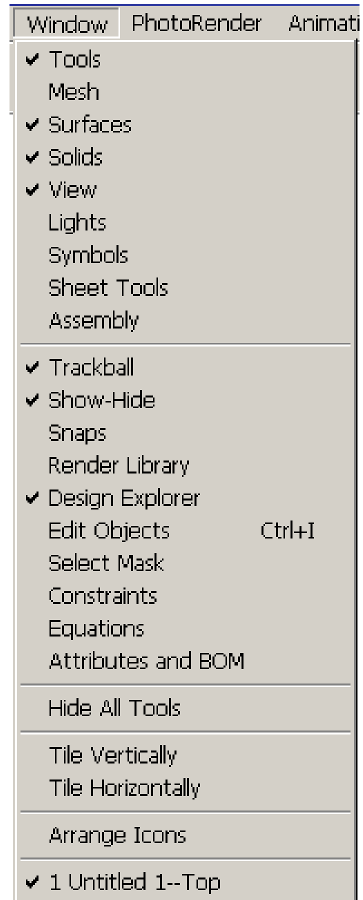
Ashlar-Vellum's Cobalt, Xenon and Argon are all based on Spatial Technologies' ACIS Geometry Engine. ACIS provides a precise boundary representation of solids suitable for design projects sensitive to accuracy. In ACIS, linear and quadric geometry is exactly represented analytically while free-form geometry is represented as Non Uniform Rational B-Splines (NURBs). Unlike faceted solid modelers, ACIS maintains a high level of accuracy as boolean operations are performed or features are added. Because the ACIS engine is so accurate, parts modeled with it are suitable for applications that demand precision, like numerical control. All of the Designer Elements programs provide several sets of tools with which to create or modify solids: Primitives, Profiles, Features and Editing.

The topics explained here include:

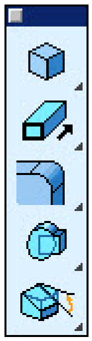
- Solid Modeling Tools
- Drafting Assistant and Solids
- Solids, Selection and Display
- Solid Associativity
- Solids and Instances
- Cutting and Pasting Solids
- Exporting Solids
- Object Types and Edit Objects
- Error Messages

Solid Modeling Tools

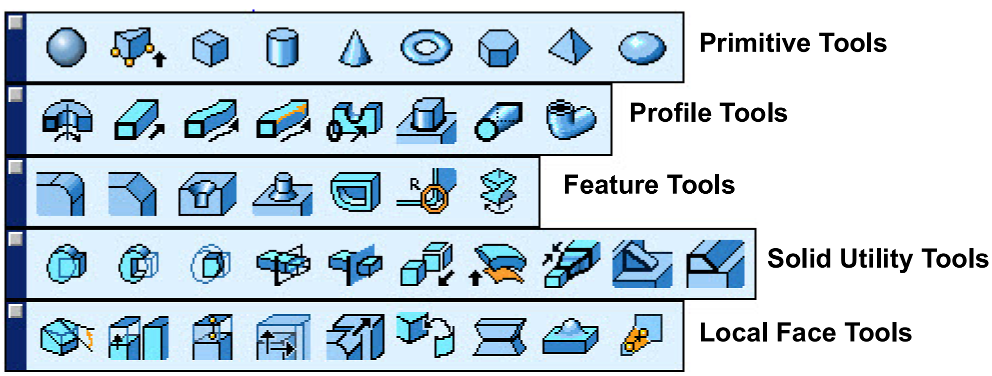
The solid modeling tools are accessible from the Solids tool palette. The Solids tool palette displays automatically when the program is first launched. To toggle the palette display on and off, use ***Window>Solids***.



The palette appears under the main tool palette on the left side of the 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. Sphere Surface Tool explains these tools.

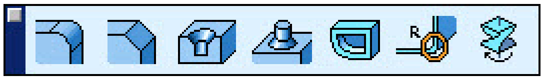
Profiles

A profile is a curve or collection of curves that enclose an area. 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, Sweeps (path, 2-rail), Cutout, Protruded Feature, Skinned Solid and Pipe Solid. Solids from Curves Tools explains these tools.

Features

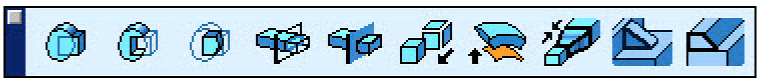
A feature simplifies building or modifying a solid by constructing common design elements with a single command.



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. Solid Feature Tools explains these tools.

Solid Utility Tools

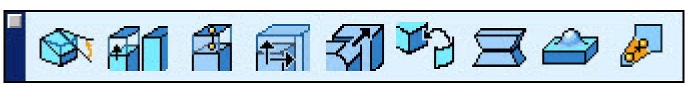
When creating the geometry it is possible to perform various operations such as splitting a solid or placing a draft. The Solid Utility tools provide various ways of editing solid objects.



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). Solid Utility Tools explains these tools.

Local Solid Face Tools

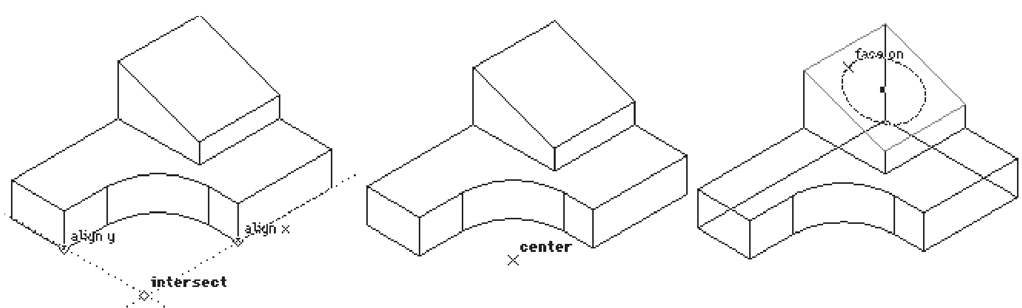
When creating or importing geometry various operations can be performed. The Local Solid Face Tools provide a means to modify individual faces of the solid geometry.



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 distance, to a point or to a curve). Solid Face Modification Tools 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

A solid can be selected by clicking on an edge or within a face. Selecting the solid on its edge is considerably faster than selecting within a face (the 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.

It is possible to set one of these as the default by choosing ***File>Preferences>Display*** and selecting the option.

Object Resolution

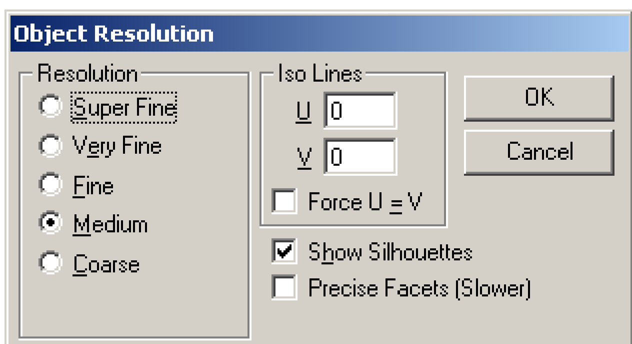
A solid's edge resolution can be increased with the Change Resolution command in the Edit menu.

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.

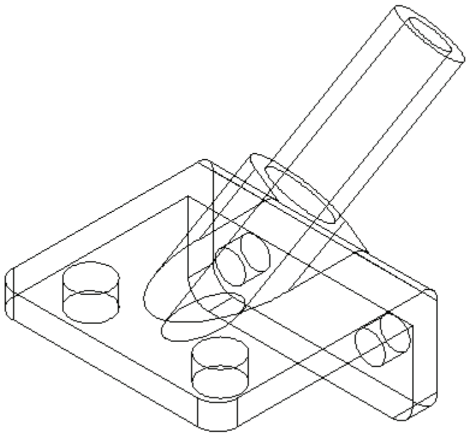


3. Enter the desired number of Iso lines. Check the Show Silhouettes box if desired.

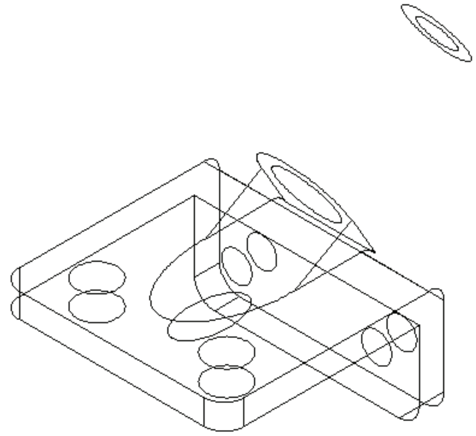
   The number of Iso Lines control the isopram lines drawn for a solid face. These Iso (isopram) lines are constant parameter curves that lie on a surface, typically defined in parameter space. The parameter space coordinate system uses U and V coordinates. A 0 (zero) in both fields turns off Iso lines. 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 Change Resolution for more information.

Solid Associativity

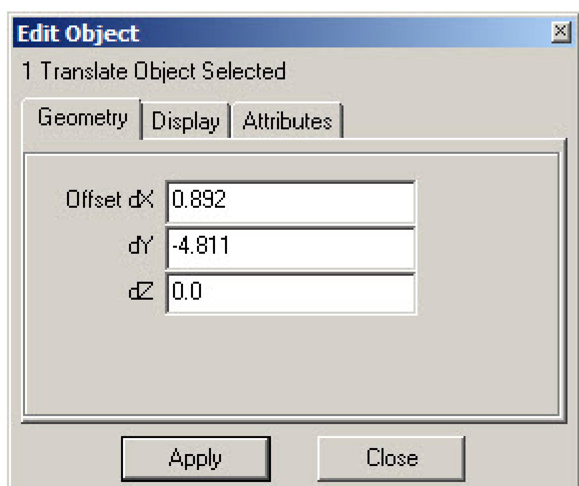
Cobalt and Xenon Only

Solid objects are associative to the original curves used to create them. Changing the curve automatically changes the dependent solid. For example, creating a lathed solid from four curves and changing 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 curve changes must not 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 performing an operation on a solid and moving it to another location, or when copying a solid, an instance of the original is created. Displaying the Edit Objects dialog box, the object is referred to as an instance.

Instances and Moving Solids

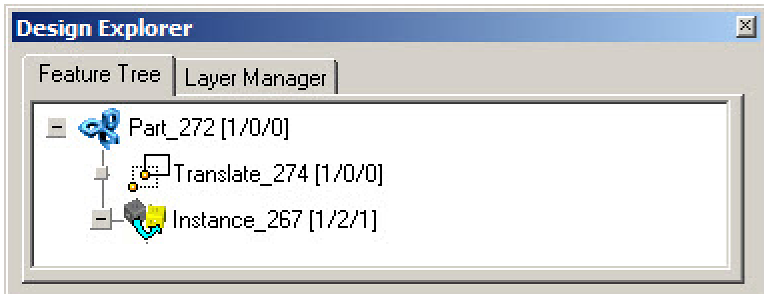


When creating an instance by moving the modified solid to another location, the information appears as an instance operation in the Design Explorer. The original object is still located in previous position, although it is not displayed.

When selecting the Instance Translate item in the Design Explorer and then open the Edit Objects dialog box, the only geometry characteristics provided are the offset distances 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. There are two instance operations in the history tree in the Design Explorer. 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.

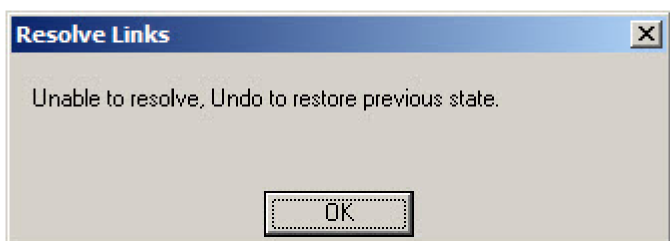


Cobalt and Xenon Only

Displaying the Edit Objects dialog box for the Instance item shows there are no unique geometric characteristics. Opening the history tree further the original solid is exposed and its characteristics can be viewed through the Edit Objects dialog box.

By copying a solid and thereby, 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 a hole is added to the original after creating the instance, the hole is also added to all instances. Choose ***Edit>Resolve Links*** for this to occur.

If an operation is performed on the original after an instance is created that conflicts with an operation already performed on that instance, two successive errors are received, a Reorder Error and a Resolve Links error (shown below). Choose ***Edit>Undo*** to return the part to the original state.



When performing operations on an instance, they are listed in the Design Explorer and do not affect the original. If the link between the original and the instance is broken, changes made to the original will not affect the instance.

Be aware when creating multiple instances, whether instancing the original part or another instance. This is important if it is necessary to break the link between the original and an instance. If there are any instances created from that instance, their associativity to the original is broken as well. So, to break the link between one instance and the original, all other instances should be created from the original solid.

If it is not necessary to create an instance, select the original solid, choose ***Edit>Copy*** and then ***Edit>Paste***. It is also possible to select the instance and choose ***Edit>Remove Links***, however, the object geometry cannot be edited using the technique.

Cutting and Pasting Solids

By cutting and pasting a solid all associativity relationships are lost. After cutting and pasting a part, it is not possible 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

By cutting and pasting a solid object into is Designer Elements program on a Macintosh computer, the object is represented in both PICT2 and VS Object form. When both forms are present, cut and paste a representation of the solid into other applications and cut and paste a true solid back into the Designer Elements program drawing.

Exporting Solids

To export a solid model created in this program to another application, convert the model to a standard format. Three formats are supported:

1. When exporting to an application that supports ACIS, 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 Cobalt, Xenon or Argon.
2. 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.
3. Export solids through the IGES translator as solid entities or as a collection of trimmed surfaces.

Object Types and Edit Objects

Every solid object created with Cobalt, Xenon or Argon 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.

Solids Primitive Tools and Solids from Curves Tools introduce the solids tools 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 Design Explorer Commands.

Error Messages

Error Messages provide more feedback when trying to perform operations such as shelling, local face modeling and so on. The Designer Elements program determines why the operation failed and provides a hint to the feature that may have caused the problem.

Solids Primitive Tools

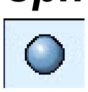
Primitives are simple solid shapes. Primitives can be created by picking one point, two points or diagonals. Each Designer Elements program primitive is defined by a unique set of characteristics which can be edited.

Values for each tool can be entered in the Status Line to define a solid either before or after creating the solid. If the values are entered after selecting the tool but before creating the solid, the first click in the drawing area automatically registers all Status Line values. Entering values in the selected Status Line data field after creating the solid and while the solid is still selected, then 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 Primitive Tool**
- **Slab Primitive Tool**
- **Block Primitive Tool**
- **Cylinder Primitive Tool**
- **Cone Primitive Tool**
- **Torus Primitive Tool**
- **Prism Primitive Tool**
- **Pyramid Primitive Tool**
- **Ellipsoid Primitive Tool**

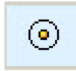
Sphere Primitive Tool



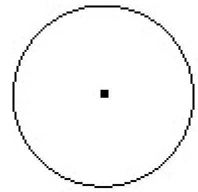
Cobalt, Xenon and Argon create a sphere from a center point and radius or from two diagonal points on a bounding box. When selecting the **Sphere Primitive** tool, a subpalette appears in the Message Line containing three options for creating spheres: **Sphere 1 Point**, **Sphere 2 Point** and **Sphere by Diagonals**.

The Drafting Assistant recognizes the points picked to define the sphere.

Sphere 1 Point Option

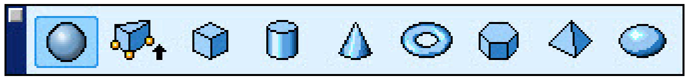


This tool draws a sphere using the specified center point and the radius entered in the Status Line.



Using the Sphere 1 Point Option

1. Select the **Sphere Primitive** tool.

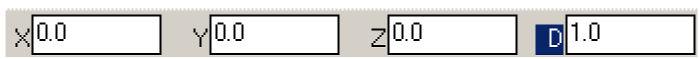


2. Select the **Sphere 1 Point** option in the Message Line. The Message Line reads: *Sphere Primitive: Enter center point for sphere*.
3. Click a point in the drawing area.

   The sphere is created in the specified location.

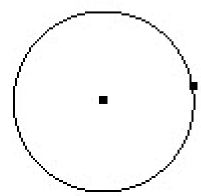
While the sphere is still selected, change the X, Y, Z location of the center point and the radius, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

The Status Line contains X, Y and Z data fields for the sphere center point and the Diameter data field.



The program creates one point spheres independently of the construction plane's orientation.

Sphere 2 Point Option



This draws a sphere from the center point and specified radius.

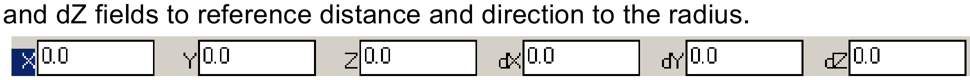
Using the Sphere 2 Point Option

1. Select the **Sphere Primitive** tool.
2. Select the **Sphere 2 Point** option 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.

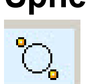
   The sphere is created.

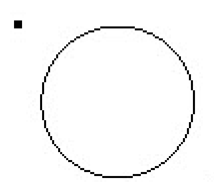
While the sphere is still selected, change the X, Y, Z location of the center point and dX, dY and dZ values for the radius, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

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 Option





This draws a sphere based on the two corners of the sphere's specified bounding box.

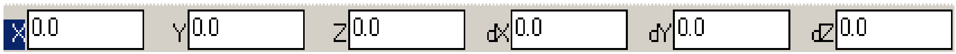
Using the Sphere By Diagonals Option

1. Select the **Sphere Primitive** tool.
2. Select the **Sphere by Diagonals** option 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.

   The 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, 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.



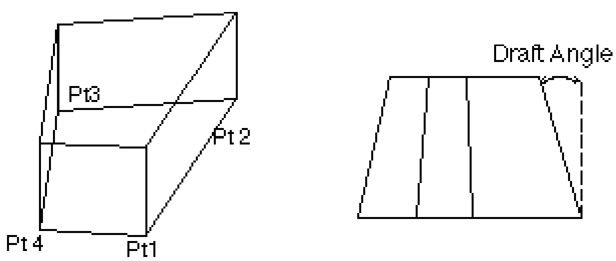
Geometric Characteristics

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

Create a solid slab primitive by extruding a series of closed planar points. If the points do not close, the program automatically closes them. It also checks for invalid or overlapping slab points. If one or more are detected, no slab is created.

Cobalt, Xenon or Argon determines the extrusion direction from the order in which the points are entered. It constructs the slab in the direction of the right hand rule (see Triad 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. Specify only 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, the height and draft angle can be changed in the Status Line, if desired. Press ENTER (Windows) or RETURN (Macintosh) to adjust the slab to the new values.

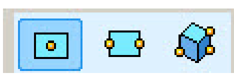
The Status Line contains the Height and Draft Angle data fields.

Geometric Characteristics

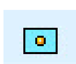
A solid slab primitive is created by picking three or more points for the slab’s corners and is made up of the following characteristics: the Height and Draft Angle, and the Defining Points (X, Y, and Z values) of the slab’s corner points. This information is listed in the Edit Objects dialog box under 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

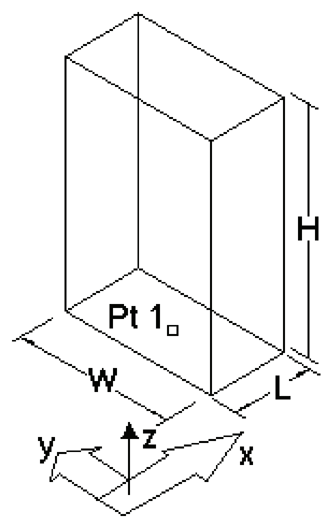
The block primitive is a cubic or rectangular solid. When selecting the **Block Primitive** tool a subpalette appears in the Message Line containing three block options: **Block 1 Point**, **Block 2 Point** and **Block by Diagonals**.



Block 1 Point Option



This draws a block using the specified center point and the Height entered in the Status Line.



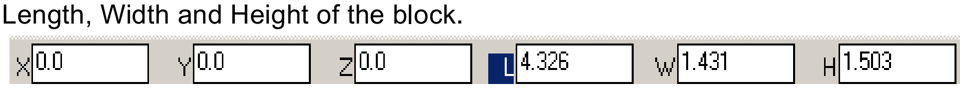
Using the 1 Point Block Option

1. Select the **Block Primitive** tool.
2. Select the **Block 1 Point** option in the Message Line. The Message Line reads: *Block Primitive: Enter center point for block base*.
3. Click a point in the drawing area.

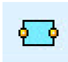
   The block is created with its base centered at the chosen point. The block’s orientation is determined by its length, width, and height along the x, y, and z axes of the work plane.

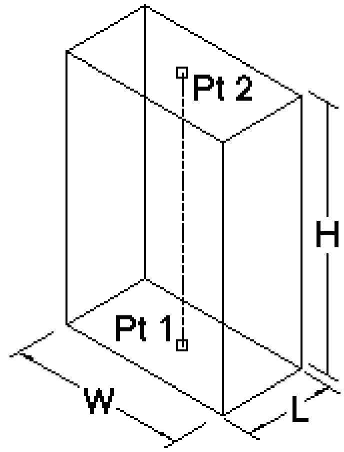
While the block is still selected, it is possible to change the X, Y, Z location of the base center point and length, width and height of the block, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

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 Option





This option draws a block between the center points of each end.

Using the 2 Point Block Option

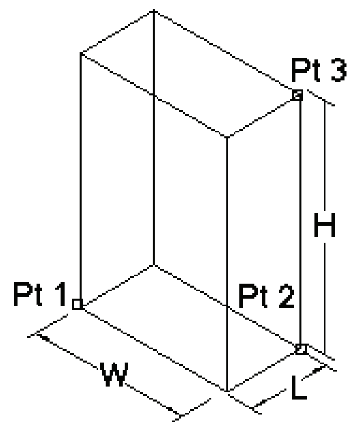
1. Select the **Block Primitive** tool.
2. Select the **Block 2 Point** option in the Message Line. The Message Line reads: *Block Primitive: Enter centerline points for block*.
3. Click two points in the drawing area.

   The block is created centered on the point chosen with a height equal to the distance between the two chosen points.

While the block is still selected, it is possible to 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 Option



This draws a block based on the two base corners and the specified height.

Using the Block by Diagonals Option

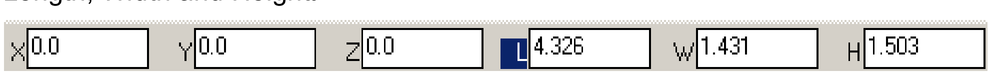
1. Select the **Block Primitive** tool.
2. Select the **Block by Diagonals** option in the Message Line. The Message Line reads: *Block Primitive: Enter start diagonal point. [Shift=Square]*
3. Click two points in the drawing area to specify two diagonal corners of the block's base.

   Hold down the SHIFT key to draw a square.
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. Working in the top, front or side view, pick a point, and the program will make the block's height equal to its width. Working in 3D view, select the point to set the desired height.

While the block is still selected, it is possible to 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.

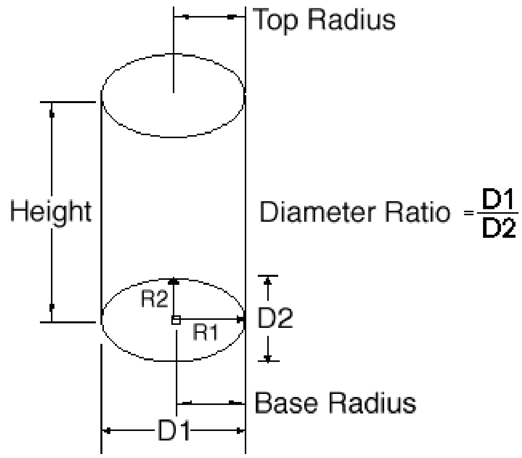
The Status Line contains the X, Y and Z values for the first chosen corner and the solid's Length, Width and Height.



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.

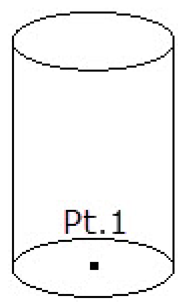
Cylinder Primitive Tool



Cobalt, Xenon and Argon create 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 a visual representation of these definitions.

When the **Cylinder Primitive** tool is selected, a subpalette appears in the Message Line containing three cylinder options: **Cylinder 1 Point**, **Cylinder 2 Point** and **Cylinder by Diagonals**.

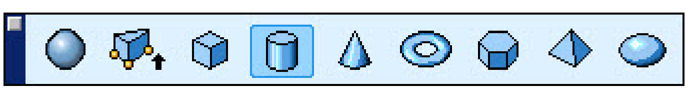
Cylinder 1 Point Options



This option draws a cylinder using the specified center point and the radius and height entered in the Status Line.

Using the Cylinder 1 Point Option

1. Select the **Cylinder Primitive** tool.



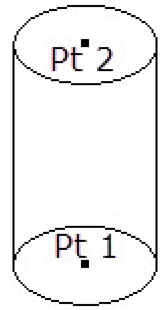
2. Select the **Cylinder 1 Point** option in the Message Line. The Message Line reads: *Cylinder Primitive: Enter center point for cylinder*.
3. Click a point in the drawing area.

   The cylinder is created with its base centered at the chosen point. The cylinder extends along the z axis of the current construction plane.

While the cylinder is still selected, it is possible to 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 Diameter and Height.

Cylinder 2 Point Option



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 Option

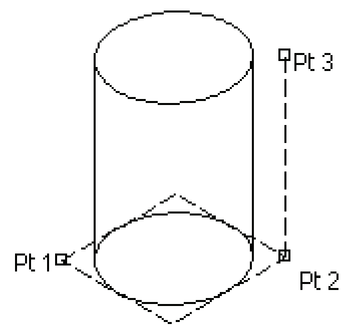
1. Select the **Cylinder Primitive** tool.
2. Select the **Cylinder Two Point** option in the Message Line. The Message Line reads: *Cylinder Primitive: Enter start and end points for cylinder*.
3. Click two points in the drawing area.

   The cylinder is created with its height is equal to the distance between the two chosen points and with its top and bottom centered on those points.

While the cylinder is still selected, it is possible to 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 Diameter.

Cylinder by Diagonals Option



This option draws a cylinder based on the two base corners and the height.

Using the Cylinder by Diagonals Option

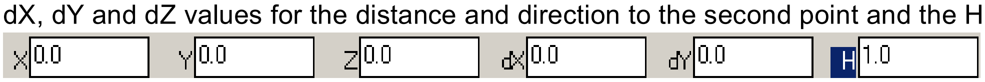
1. Select the **Cylinder Primitive** tool.
2. Select the **Cylinder by Diagonals** option in the Message Line. The Message Line reads: *Cylinder Primitive: Enter start diagonal point. [Shift=Square]*
3. Click two points in the drawing area to specify two diagonal corners of the cylinder base's bounding box.

   Hold down the SHIFT key to draw a circular base.
4. The Message Line reads: *Cylinder Primitive: Enter height*.
5. Drag to set the height of the cylinder.

   The cylinder is created. Working in the top, front or side view, pick a point and the program makes the cylinder's height equal to that of the last drawn cylinder (or equal to one inch if this is the first drawn cylinder). Working in a 3D view it is possible to select the point to set the desired height.

While the cylinder is still selected, it is possible to 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.



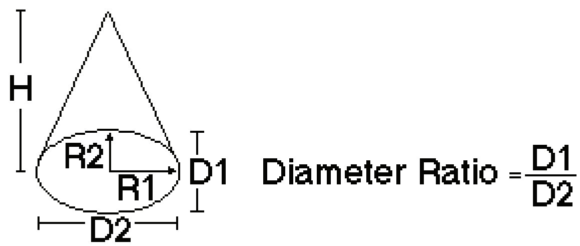
Geometric Characteristics

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.

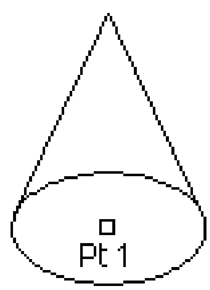
The ratio defines the relationship between the major radius (R1) and minor radius (R2). A circular cone's base ratio equals one. Any other value yields an elliptical cone.



When selecting the **Cone Primitive** tool a subpalette appears in the Message Line containing three cone options: **Cone 1 Point**, **Cone 2 Point** and **Cone by Diagonals**.

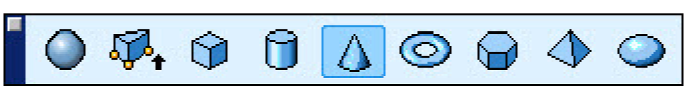
Cone 1 Point Option

This option draws a cone using the specified center point and the radius and height entered in the Status Line.



Using the Cone 1 Point Option

1. Select the **Cone Primitive** tool.



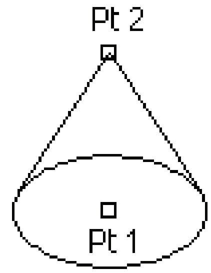
2. Select the **Cone 1 Point** option in the Message Line. The Message Line reads: *Cone Primitive: Enter center point for cone*.
3. Click a point in the drawing area.

   The cone is created with its base centered at the chosen point. The cone extends along the z axis of the current work plane.

While the cone is still selected, 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 Diameter and Height.

Cone 2 Point Option



This tool draws a cone from the center point of the base at the height and specified radius.

Using the Cone 2 Point Option

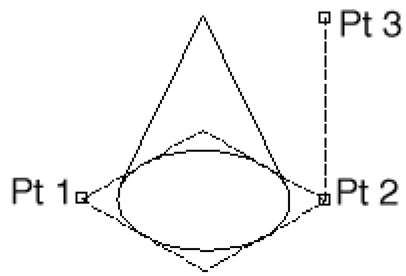
1. Select the **Cone Primitive** tool.
2. Select the **Cone 2 Point** option in the Message Line. The Message Line reads: *Cone Primitive: Enter start and end points for cone*.
3. Click in the drawing area to set the start (center) point and the end (tip) point of the cone.

   The cone is created with its height equal to the distance between the two chosen points and with its top and bottom centered on those points.

While the cone is still selected, 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 Diameter.

Cone by Diagonals Option



This tool draws a cone based on the two base corners and the specified height.

Using the Cone by Diagonals Option

1. Select the **Cone Primitive** tool.
2. Select the **Cone by Diagonals** option 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.

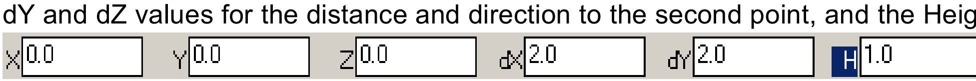
   Hold down the SHIFT key to draw a circular base.

   After doing this step, the Message Line reads: *Cone Primitive: Enter height*.
4. Drag to set the height of the cone.

   The cone is created. Working in the top, front or side view, pick a point, and the program makes the cone's height equal to that of the last drawn cone (or equal to one, if this is the first drawn cone). Working in a 3D view, select the point to set the desired height.

While the cone is still selected, 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.

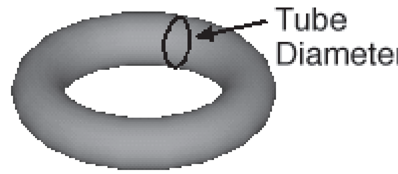
The Status Line contains the X, Y and Z values for the cone's base center point, the dX, dY and dZ values for the distance and direction to the second point, and the Height.



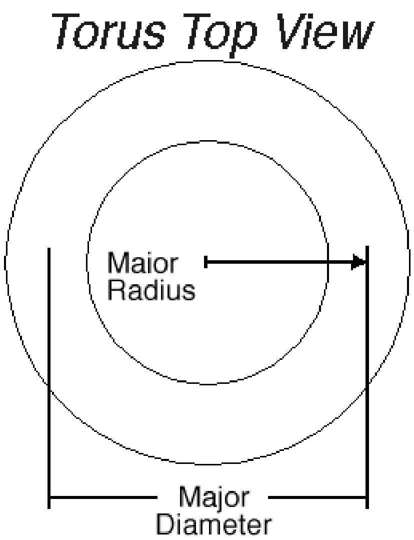
Geometric Characteristics

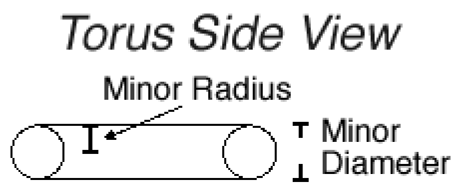
A cone primitive is created by picking one point to specify the center point of the cone's base, two points for the center point and height, or three points for the corners of the bounding box and the cone's height. It is made up of the following characteristics: Height, Top Diameter, Diameter Ratio (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.

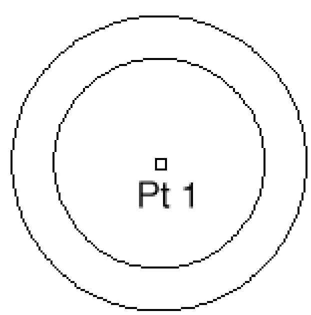




When selecting the **Torus Primitive** tool, a subpalette appears in the Message Line with three torus options: **Torus 1 Point**, **Torus 2 Point** and **Torus by Diagonals**.

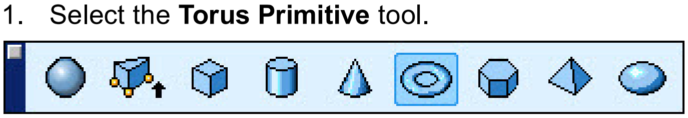
Torus 1 Point Option

This draws a torus using the specified center point and the Major Diameter (D1) and the Minor Diameter (d2) of the Status Line.



Using the Torus 1 Point Option

1. Select the **Torus Primitive** tool.



2. Select the **Torus 1 Point** option 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.

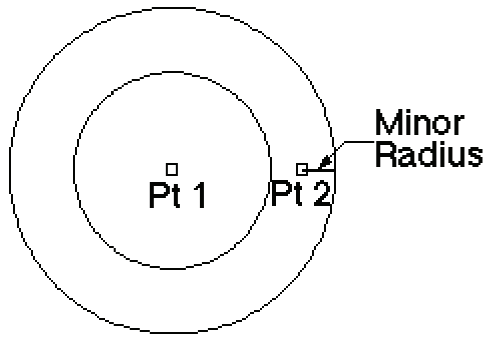
   A torus is created with its center at the chosen point. The torus lies perpendicular to the z axis of the current work plane.

While the torus is selected, 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 Option

This draws a torus from the center point and radius.



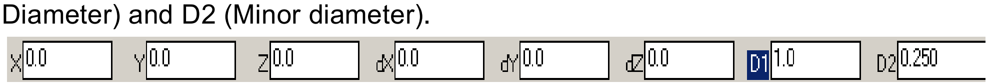
Using the 2 Point Torus Option

1. Select the **Torus Primitive** tool.
2. Select the **Torus 2 Point** option 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.

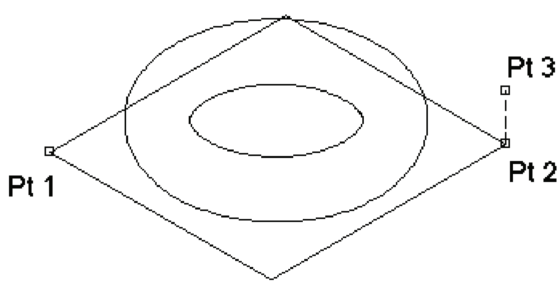
While the torus is still selected, change the X, Y, Z location of the base center point, the dX, dY and dZ to the second point for defining the distance and direction, the Major Diameter (D1) and Minor Diameter (d2), if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the values.

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 Option

This option draws a torus from the center point, major radius and minor radius.



Using the Torus by Diagonals Option

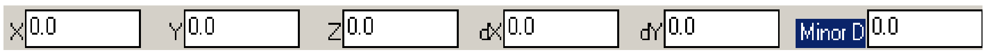
1. Select the **Torus Primitive** tool.
2. Select the **Torus by Diagonals** option in the Message Line. The Message Line reads: *Torus Primitive: Enter start diagonal point. [Shift=Square]*
3. Click twice to specify two diagonal corners of the torus' bounding box.

   Hold down the SHIFT key to draw a square bounding box.
4. The Message Line reads: *Torus Primitive: Enter height*.
5. Drag to set the height of the torus.

Working in the top, front or side view, pick a point. The Designer Elements program calculates the torus' Minor Diameter based on the location of that point. Working in a 3D view, it is possible to select the point to set the Minor Diameter.

While the torus is still selected, 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.

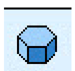
The Status Line contains the X, Y and Z values for the torus center point, the dX, dY and dZ values for the distance and direction to the second point D1 (Major Diameter) and the D2 (Minor Diameter).

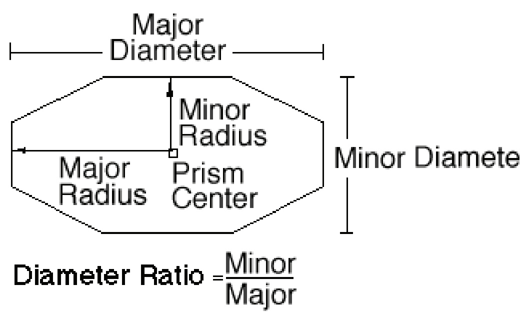


Geometric Characteristics

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

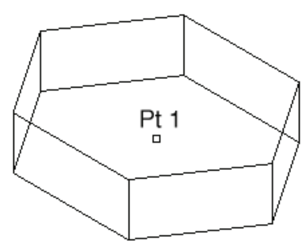




The prism primitive is composed of three or more sides, all perpendicular to the solid's base.

Selecting the **Prism Primitive** tool, a subpalette appears in the Message Line containing three prism options: **Prism 1 Point**, **Prism 2 Point** and **Prism by Diagonals**.

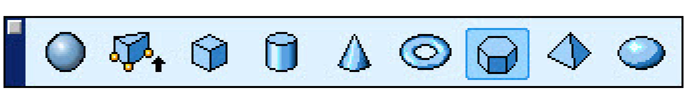
Prism 1 Point Option



This option draws a prism using the specified center point and the radius, height and number of sides listed of the Status Line.

Using the Prism 1 Point Option

1. Select the **Prism Primitive** tool.

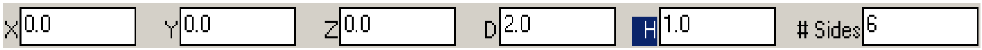


2. Select the **Prism 1 Point** option in the Message Line. The Message Line reads: *Prism Primitive: Enter center point for prism.*
3. Click in the drawing area to set the base's center point.

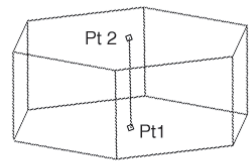
   A prism is created with its base centered at the chosen point. The prism extends along the z axis of the current work plane.

While the prism is still selected, 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 Diameter, Height, and Number of Sides.



Prism 2 Point Option



This draws a prism from the center point and at the specified height.

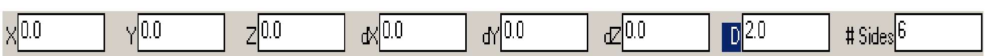
Using the 2 Point Prism Option

1. Select the **Prism Primitive** tool.
2. Select the **Prism 2 Point** option 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).

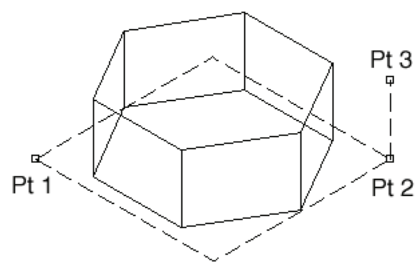
   A prism is created with its height equal to the distance between two points of the choice and its top and bottom centered on those points.

While the prism is still selected, 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 Diameter and Number of Sides.



Prism by Diagonals Option



This tool draws a prism from two corner points, representing the prism's bounding box and a third point for the height.

Using the Prism by Diagonals Option

1. Select the **Prism Primitive** tool.
2. Select the **Prism by Diagonals** option 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.
4. The Message Line reads: *Prism Primitive: Enter height*.
5. Drag to set the height of the prism.

   Working in the top, front or side view, pick a point, and the program makes the prism's height equal to the last drawn prism (or equal to one inch if this is the first drawn prism). Working in 3D view it is possible to select the point to set the desired height.

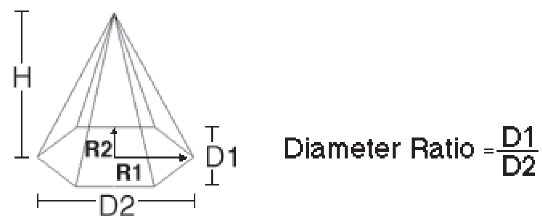
While the prism is still selected, change the X, Y, Z location of the prism's base center point, the dX, dY, and dZ values for the distance and direction to the second point and the prism's height, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

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 and Number of Sides.

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.

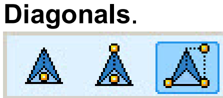
Pyramid Primitive Tool



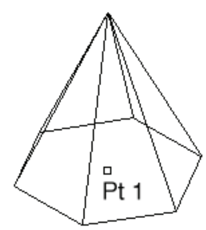
A pyramid contains three or more flat faces which converge to a common vertex at the tip.

A pyramid is created by defining the diameter of the base, the height and the number of sides.

Selecting the **Pyramid Primitive** tool a subpalette appears in the Message Line containing three pyramid options: **Pyramid 1 Point**, **Pyramid 2 Point** and **Pyramid by Diagonals**.



Pyramid 1 Point Option



This draws a pyramid using the specified center point and the radius, height and number of sides listed of the Status Line.

Using the 1 Point Pyramid Tool

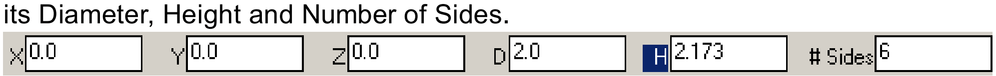
1. Select the **Pyramid Primitive** tool.

2. Select the **Pyramid 1 Point** option 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.

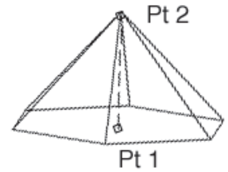
   A pyramid is created with its base centered at the chosen point. The pyramid extends along the z axis of the current work plane.

While the pyramid is still selected, 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 Diameter, Height and Number of Sides.



Pyramid 2 Point Option



This option draws a pyramid from the base center point and the height.

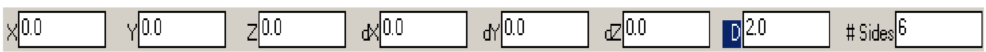
Using the 2 Point Pyramid option

1. Select the **Pyramid Primitive** tool.
2. Select the **Pyramid Two Point** option 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.

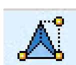
   A pyramid is created whose height is equal to the distance between the two chosen points and whose top and bottom are centered on those points.

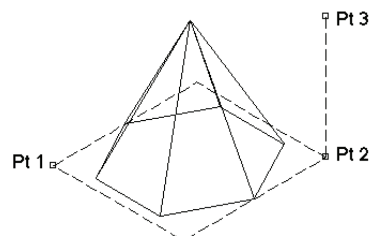
While the pyramid is still selected change the X, Y, Z location of the base center point, the radius, height and number of sides, if desired. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

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 Diameter and Number of Sides.



Pyramid by Diagonals Option





This option draws a pyramid from the corner of the pyramid's bounding box and the specified height.

Using the Pyramid By Diagonals Option

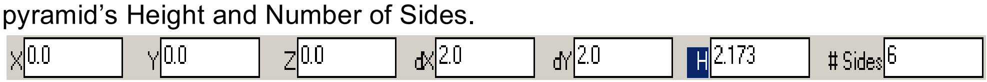
1. Select the **Pyramid Primitive** tool.
2. Select the **Pyramid by Diagonals** option in the Message Line. The Message Line reads: *Pyramid Primitive: Enter start diagonal point. [Shift=Square]*
3. Click two points in the drawing area to specify two diagonal corners of the pyramid base's bounding box.

   Hold down the SHIFT key to draw a square bounding box.
4. The Message Line reads: *Pyramid Primitive: Enter height*.
5. Drag to set the height of the pyramid.

   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 drawn pyramid (or equal to one inch if this is the first drawn pyramid). Working in a 3D view it is possible to select the point to set the desired height.

While the pyramid is still selected 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 and Number of Sides.



Geometric Characteristics

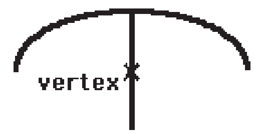
A pyramid primitive is created by picking one point to specify the center point of the pyramid's base, two points for the center point and height, or three points for the corners of the bounding box and the pyramid's height. It is made up of the following characteristics: Height, Top Diameter, Diameter 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.

Selecting the **Ellipsoid Primitive** tool a subpalette appears in the Message Line containing three options for creating ellipsoids: **Ellipsoid 1 Point**, **Ellipsoid 2 Point** and **Ellipsoid by Diagonals**.

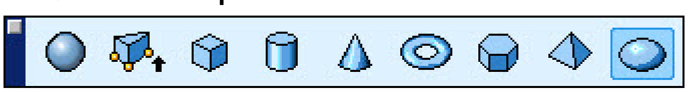
Ellipsoid 1 Point Options



This option creates an ellipsoid using the specified center point 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 Option

1. Select the **Ellipsoid Primitive** tool.



2. Select the **Ellipsoid 1 Point** option 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, it is possible to 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 Option

This creates an ellipsoid using the center point and the specified radius. The Drafting Assistant displays *vertex* notations at the center point and radius.

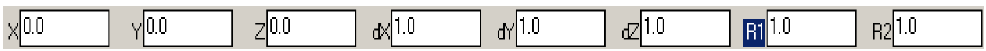
Using the Ellipsoid 2 Point Option

1. Select the **Ellipsoid Primitive** tool.
2. Select the **Ellipsoid 2 Point** option 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.

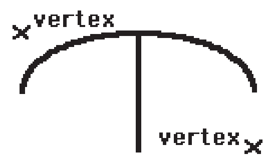
   The ellipsoid appears.

While the ellipsoid is still selected, change the X, Y and Z location of the center point, the dX, dY and dZ values for the first radius and the two radiuses not specified in the operation (R1 and R2). Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

TheStatus Line contains the X, Y and Z values for the center point, the dX, dY, dZ, R1 and R2 values.



Ellipsoid by Diagonals Option



This creates an ellipsoid based on two corners of the ellipsoid’s bounding box and the specified height. The Drafting Assistant displays *vertex* notations at the bounding box corners.

Using the Ellipsoid by Diagonals Option

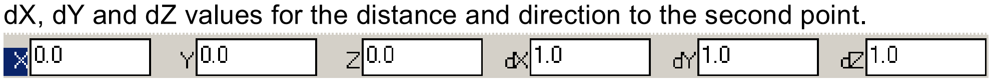
1. Select the **Ellipsoid Primitive** tool.
2. Select the **Ellipsoid by Diagonals** option in the Message Line. The Message Line reads: *Ellipsoid Primitive: Enter start diagonal point. [Shift = Square]*
3. Click two points in the drawing area to specify two diagonal corners of the ellipsoid’s bounding box.

   Hold down the SHIFT key to draw a rectangular base.
4. The Message Line reads: *Ellipsoid Primitive: Enter height*.
5. Drag to set the height of the ellipsoid.

   The ellipsoid appears. Working in the top, front or side view, pick a point, and the program makes the ellipsoid’s height equal to that of the last drawn ellipsoid (or equal to one inch if this is the first drawn ellipsoid). Working in a 3D view it is possible to select the point to set the desired height.

While the ellipsoid is still selected, it is possible to change the X, Y, Z location of the ellipsoid’s base 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.

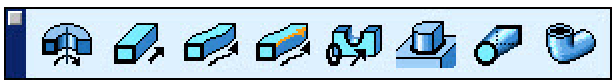


Geometric Characteristics

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.

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 from Curves Tools

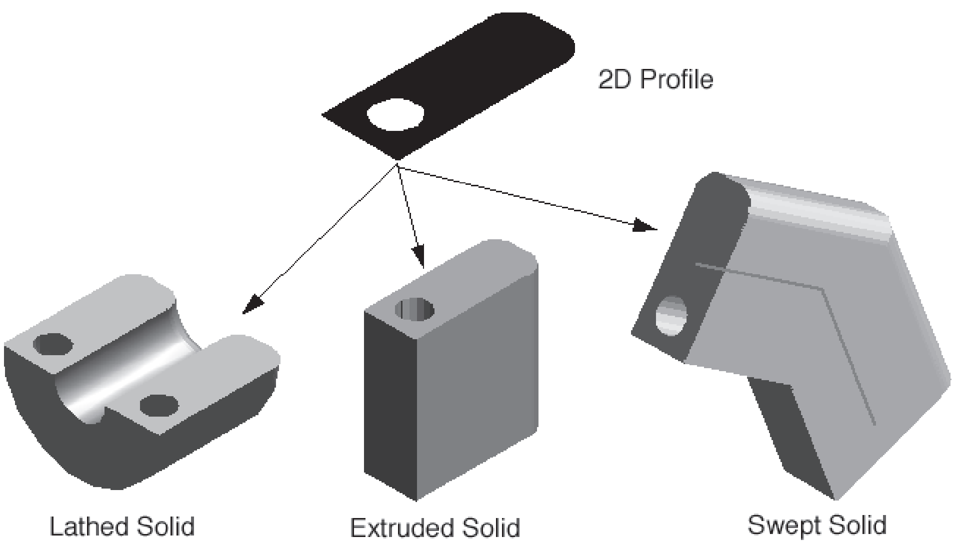


A profile is a closed section created from curves, polygons or surfaces (planar and non-planar). Lathe, extrude, or sweep a profile to create a solid.

Creating Associative Profiles in Cobalt & Xenon

Cobalt and Xenon Only

When making a solid from a profile, Cobalt and Xenon create a parent/child relationship between the profile (the parent) and the solid (the child). By changing the parent, Cobalt or Xenon automatically regenerates the child. For example, by lathing a rounded rectangle polygon, then changing the polygon's radius, the solid will update to the new size. The graphic here shows how to use the same 2D profile to create three different objects.



Tech Note: *The ASIC kernel supports grouped profiles as well as grouped curves for profiles.*

Creating Solids from Profiles

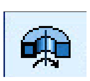
Cobalt, Xenon and Argon all support five tools for creating solids from profiles. For each tool enter values in the Status Line to define a solid, either before or after creating the solid. By entering the values after selecting the tool but before creating the solid, the first click in the drawing area automatically registers all Status Line values. When entering 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.

> ***Tech Note:*** *These tools do not support using the Select Chain command to select multiple grouped profiles, however, multiple grouped profiles can be extruded at one time.*

The topics explained in the chapter include the following:

- **Lathed Solid Tool**
- **Extrude Solid Tool**
- **Swept Solid Tool**
- **Sweep 2 Rail (Paths) Solid**
- **Cutout Feature Tool**
- **Protruded Feature Tool**
- **Cutout and Protruded Curve Checks**
- **Skinned Solid Tool**
- **Solids from Profiles Curve Checks**
- **Pipe Solid Tool**

Lathed Solid Tool



The **Lathed Profile** tool creates a solid by revolving a profile around an axis line. Lathe profiles are composed of individual, grouped or nested group curves. A profile that intersects itself cannot be lathed, as can happen with splines and polygons.

Using the Lathed Solid Tool

Before using this tool 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 the lathed solid. Tab between data fields.

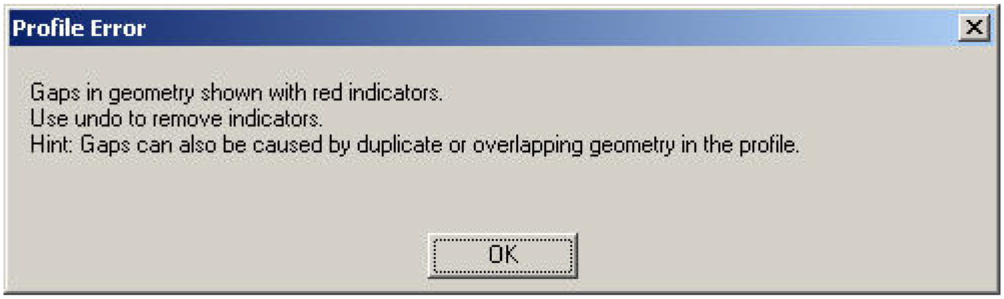
   

   When the draft angle is zero, the sides of the solid lie parallel to the profile normal. If the draft angle is positive, the sides will be forced outward from the normal. Negative draft angles force the sides inward toward the normal.
3. Pick a curve or curves to lathe. If selecting more than one curve, hold down the SHIFT key before selecting the first curve.

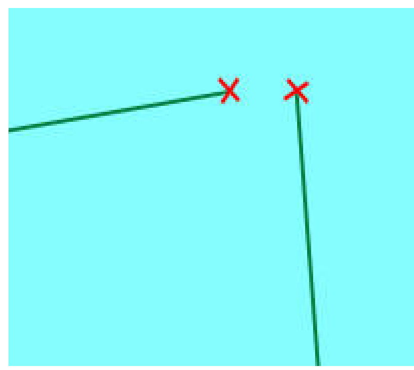
   The curves making up the profile must be closed. If they are not, two warning messages appear. The first one says, "Gaps detected in profile. Would you like to examine gaps?"

Click No to ignore the gaps and continue.

Click Yes, and another message appears: "Gaps in profile are shown with red indicators. Use Undo to remove indicators. Hint: gaps can also be caused by duplicate or overlapping geometry in the profile."



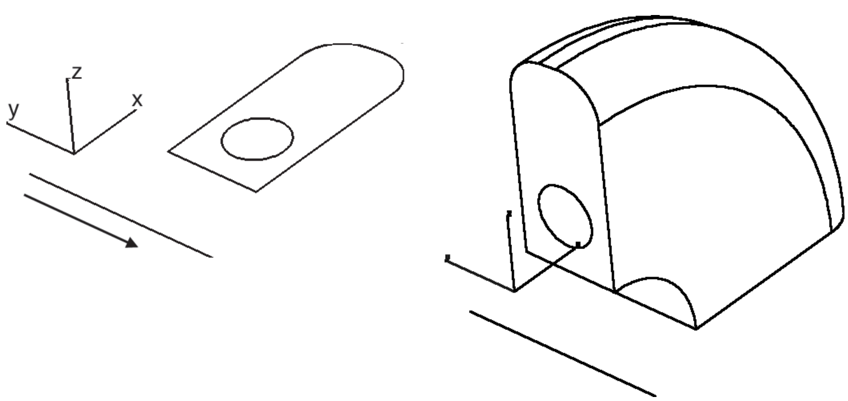
Click OK to remove the message and view the profile with red Xs where gaps are detected. After noting the gaps, use ***Edit>Undo*** to clear the indicators or drag a selection box around them and press Delete. The indicators are created as geometric points, adding geometry to the drawing.



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.

   Cobalt, Xenon or Argon 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 clicking the line, the object is lathed.

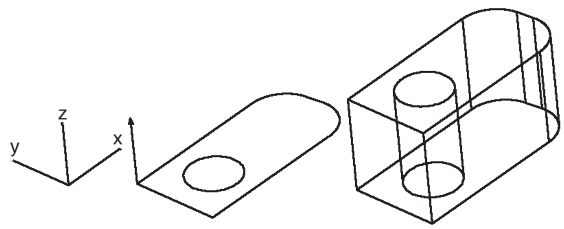
While the object is still selected it is possible to edit the number of degrees in the revolution angle and the draft angle for the entire solid. Enter the new values and press ENTER (Windows) or RETURN (Macintosh) and the solid updates.

Geometric Characteristics

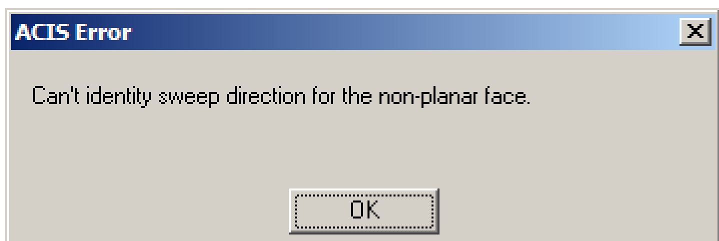
A lathed profile solid is created by picking one or more curves from which to construct the solid and a line about which to rotate the curves. The lathed solids is made up of the following characteristics: Lathe Angle and Draft Angle. This information is listed in the Edit Objects dialog box under the 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

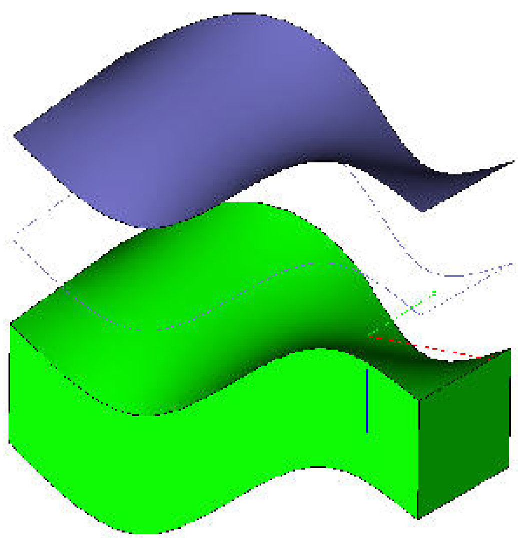
The **Extrude Solid** tool extends the selected profile along a vector. Extruded profiles are composed of lines, circles, ellipses, closed splines, polygons, grouped curves (including nested groups) and surfaces.



Be careful when selecting circles for extrusion. If one of them is actually a sphere, the following error message appears:

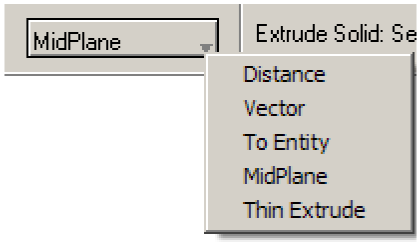


This tool supports extruding a non-planar surface.



The **Extrude Solid** 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 a specified distance normal to the plane of the profile. The distance can be positive or negative.

Vector Extrudes a profile along a vector whose distance is defined by the user supplied points.

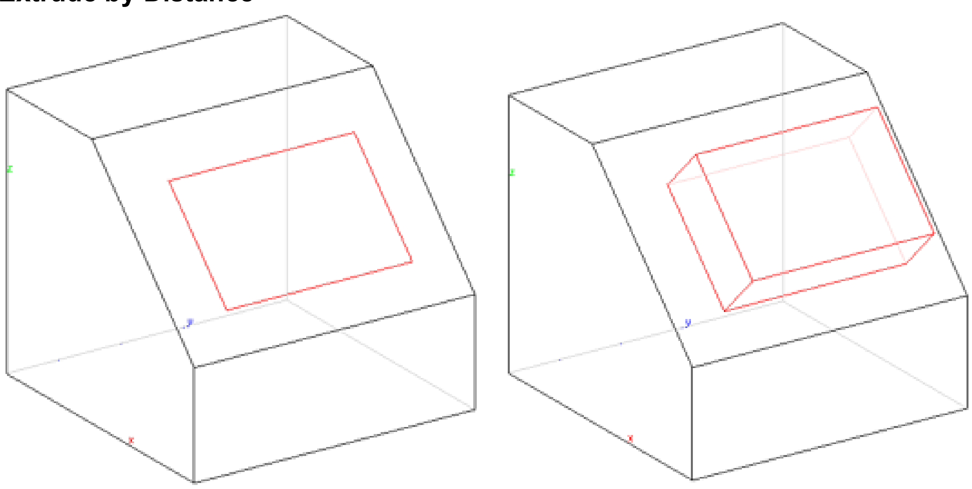
Mid Plane Extrudes a profile in both directions using the plane of the profile.

To Entity Extrudes along a vector terminating at a surface or solid. The terminating body is not unioned with the extruded body.

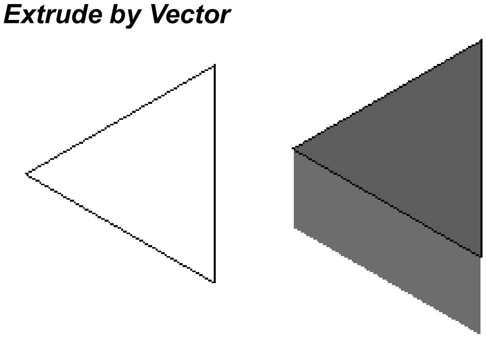
Thin Extrudes an open or closed profile by a vector and thickness.

The following are examples of each type of extrusion:

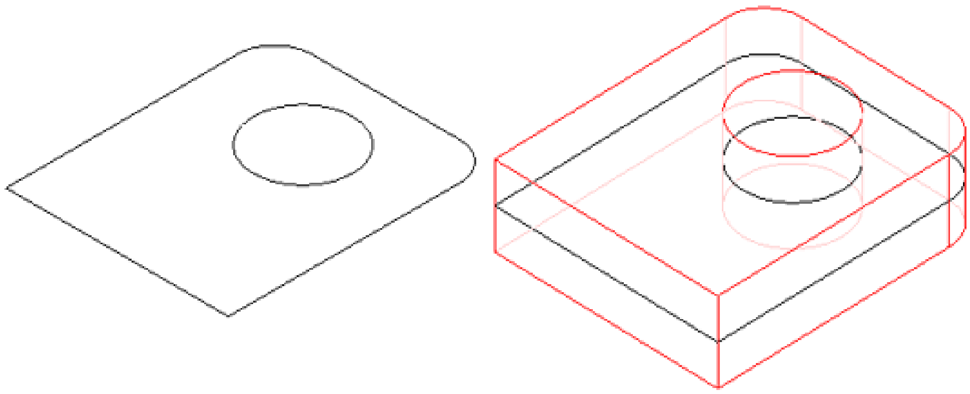
Extrude by Distance



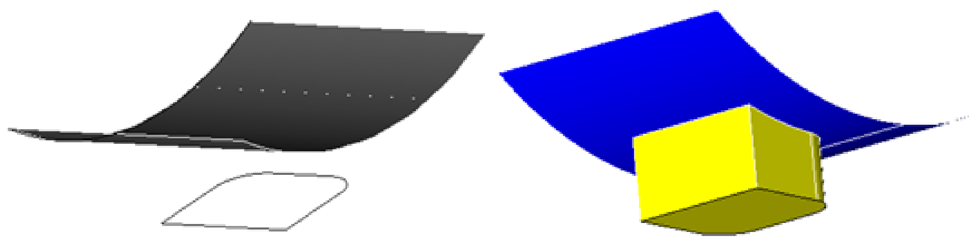
Extrude by Vector



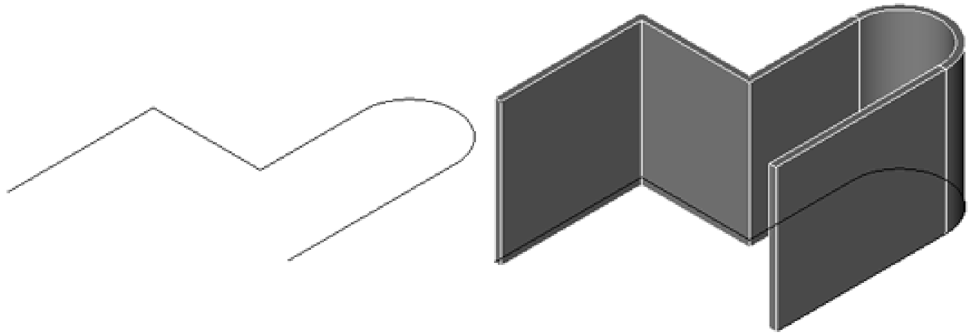
Extrude by Mid Plane



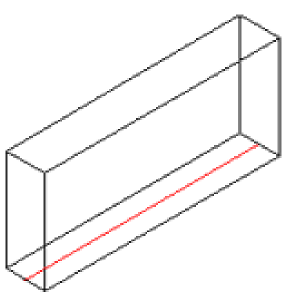
Extrude to Entity



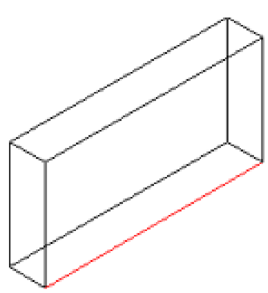
Thin Extrude



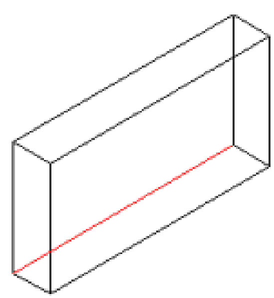
Note: The Thin Extrude tool has a toggle key that changes the location of the extrude. Use the CRTL key (Windows) or the OPTION key (Macintosh) to toggle the extrusion between the right side, left side or offset (mid plane) directions. Examples of the toggle directions are below.



Offset/Mid Plane

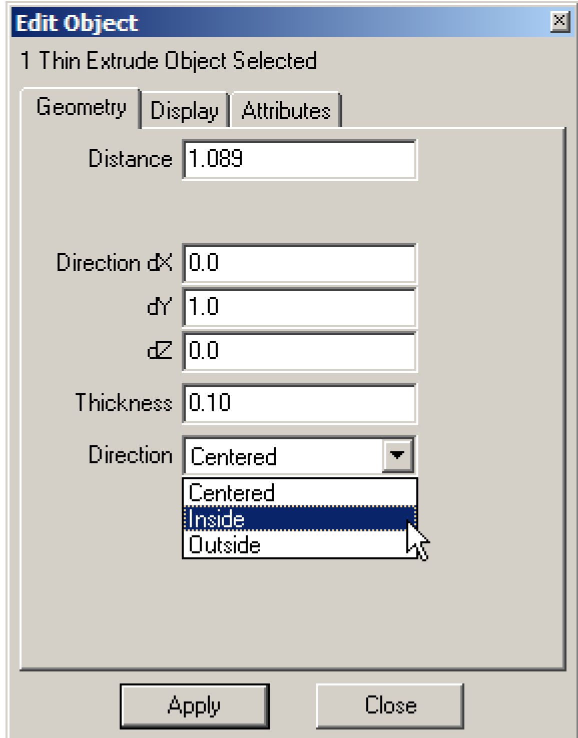


Right Side Extrude



Left Side Extrude

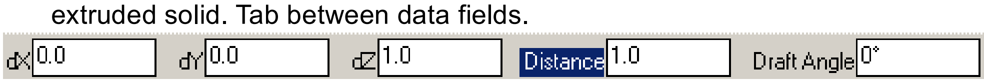
Access these toggle directions in the Edit Objects dialog box. It is necessary, however, to access the Edit Objects box through the Design Explorer in order to change these directions.



Using the Extrude Solid Tool

Before using this tool 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 the extruded solid. Tab between data fields.

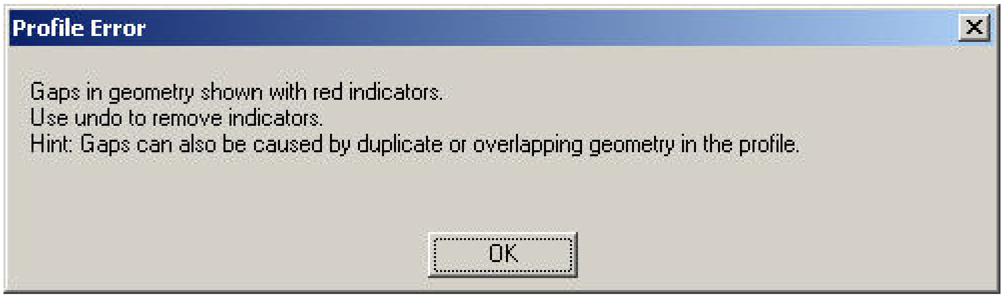


   When the draft angle is zero, the sides of the solid lie parallel to the profile normal. If the draft angle is positive, the sides will be forced outward from the normal. Negative draft angles force the sides inward towards the normal.

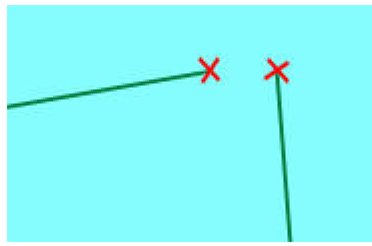
3. Pick one or more curves to extrude. As the Message Line indicates, they must be closed. If they are not, two messages appear. The first one says, "Gaps detected in profile. Would you like to examine gaps?" Click No to ignore the gaps and continue.

> ***Tech Note:*** *Some draft angles and extrusion heights will cause the solid to self-intersect. If this happens, reduce the draft angle or increase the extrusion height.*
> *When selecting multiple profiles if one of them is not planar, a warning will appear. Correct this or an ACIS error will occur.*

Click Yes, and another message appears: "Gaps in profile are shown with red indicators. Use Undo to remove indicators. Hint: gaps can also be caused by duplicate or overlapping geometry in the profile."



Click OK to remove the message and view the profile with red Xs where gaps are detected. After noting the gaps, use ***Edit>Undo*** to clear the indicators or drag a selection box around them and press Delete. The indicators are created as geometric points, adding geometry to the drawing.



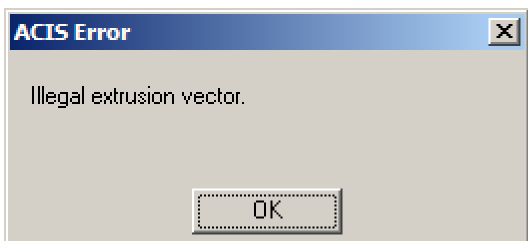
Cobalt, Xenon and Argon place markers at the gap locations so it is easy to find and close them.

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 its 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 choosing points that are tangential to the profile:



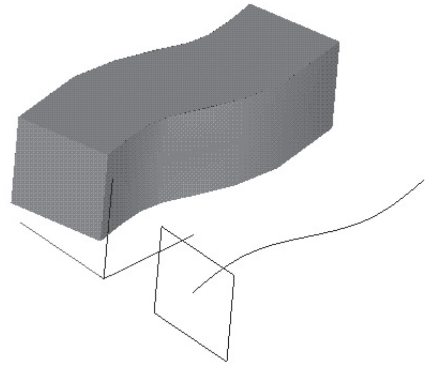
While the object is still selected edit the dX, dY, and dZ values for the extrusion vector's distance and the draft angle for the entire solid. Enter the new values and press ENTER (Windows) or RETURN (Macintosh) and the solid updates.

> ***Tech Note:*** *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.

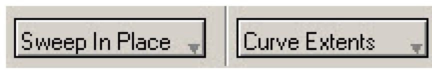
Swept Solid Tool



The **Swept Solid** tool creates a solid by sweeping the profile along a curve. Extruded profiles are composed of lines, circles, ellipses, closed splines, polygons, grouped curves, curves and surfaces (including nested groups).

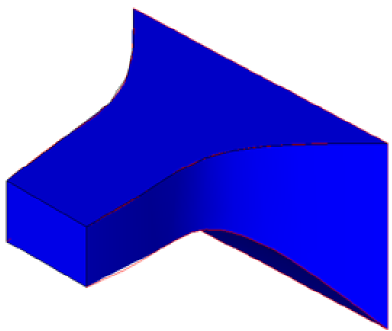
Important: Be careful that there are no gaps or overlapping curves in the profile.

The **Swept Solid** tool has two pull down options in the Message Line. The first pull down controls 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 Perpendicular 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 behavior in some previous versions of Cobalt, Xenon, Argon and Vellum Solids.

Using the Swept Solid Tool

Before using this tool, create a profile and a curve for the sweep.

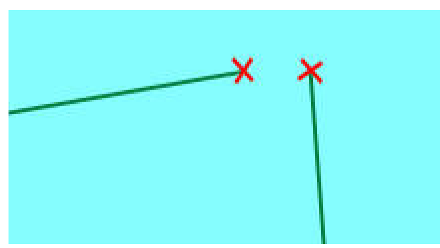
1. Click the **Swept Solid** tool. The Message Line reads: *Swept Solid: Select closed profile to sweep. [Shift=Extend]*
2. The Status Line contains the Twist Angle and Draft Angle data fields. Enter the twist angle through which the solid will twist from beginning to end, and a draft angle, if desired.

3. Pick one or more curves for the profile. As the Message Line indicates, they must be closed. If they are not, two messages appear. The first one says, "Gaps detected in profile. Would you like to examine gaps?" Click No to ignore the gaps and continue.

   Click Yes, and another message appears: "Gaps in profile are shown with red indicators. Use Undo to remove indicators. Hint: gaps can also be caused by duplicate or overlapping geometry in the profile."

   Click OK to remove the message and view the profile with red Xs where gaps are detected. After noting the gaps, use ***Edit>Undo*** to clear the indicators or drag a selection box around them and press Delete. The indicators are created as geometric points, adding geometry to the drawing.

   Cobalt, Xenon and Argon place markers at the gap locations so it is easy to 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 the sweep will 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 edit the twist and draft angles. Enter the new values and press ENTER (Windows) or RETURN (Macintosh) and the solid is adjusted.

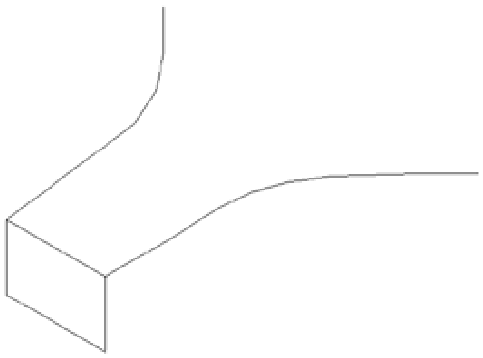
Geometric Characteristics

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 curves. Extruded profiles are 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 using this tool, 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 to sweep.
3. Choose the first rail that the profile will be swept along.
4. Choose the second rail that the profile will be swept along.

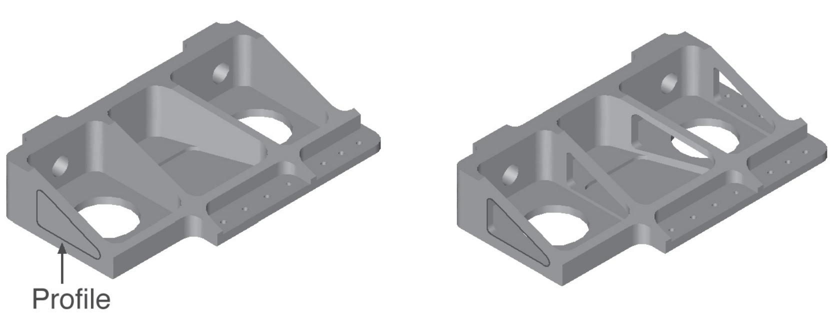
There are not status line entries for this tool.

Geometric Characteristics

A swept two rail profile solid is created by picking a closed profile then picking the two rails (curve or group of curves) along which the profile will be swept. There are only two tabs in the Edit Objects dialog box for this tool: Display and Attributes.

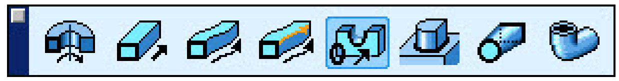
Cutout Feature Tool

This tool subtracts material from a solid. The cutout is formed from a 2D profile. This profile is located on or in the solid and composed of grouped curves. 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 to indicate the direction and length of the cutout.

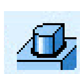
While the solid is still selected, change the distance, draft angle and direction of the cutout in the Status Line. Type the desired values in the data fields and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the extrusion dX, dY and dZ values, the extrusion Distance and the Draft Angle.

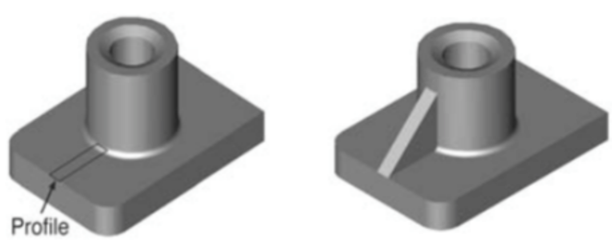
Geometric Characteristics

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.

Protruded Feature Tool

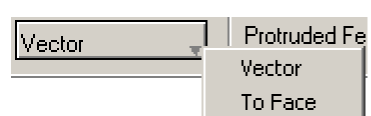


This tool adds a piece to an existing solid. The protrusion is formed from a 2D profile. The profile does not need to be attached to the original solid and can be grouped curves.



Termination Types

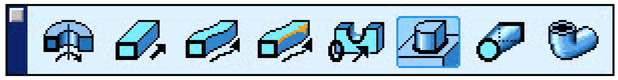
The Message Line contains a pull-down menu to choose where the protrusion will end. There are two options: *Vector* and *To Face*.



Vector	Defined by two chosen points, the vector determines the length and direction of the protrusion.
To Face	This option 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, 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 the protrusion will end.
7. Specify two points for the protrusion direction. By specifying the direction, not the length, it is not necessary to reference the curve or the solid.

While the solid is still selected, 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 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.

Cutout and Protruded Curve Checks

To assist in performing cutout and protrude operations, Cobalt, Xenon and Argon provide two curve checks: **gaps** and **planar**.

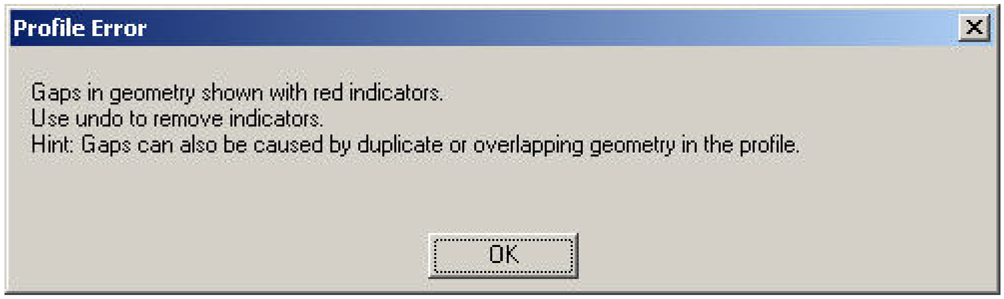
Curve Gap Check

The software provides functionality to assist in finding gaps in extruded profiles and fixing them. If gaps are detected, two warning messages appear.

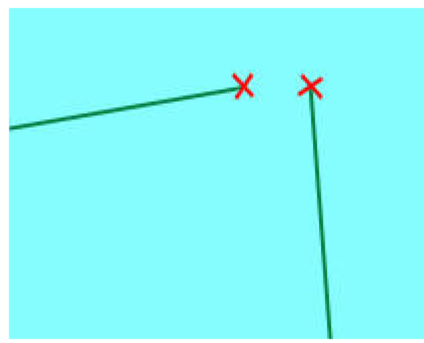
The first one says, "Gaps detected in profile. Would you like to examine gaps?"

Click No to ignore the gaps and continue.

Click Yes, and another message appears: "Gaps in profile are shown with red indicators. Use Undo to remove indicators. Hint: gaps can also be caused by duplicate or overlapping geometry in the profile."

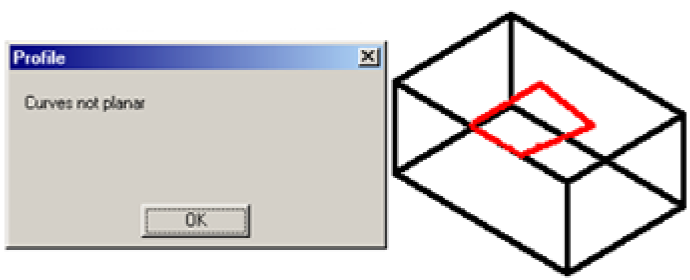


Click OK to remove the message and view the profile with red Xs where gaps are detected. After noting the gaps, use ***Edit>Undo*** to clear the indicators or drag a selection box around them and press Delete. The indicators are created as geometric points, adding geometry to the drawing.



Planar Curves Check

The planar check warns if the selected curves for the cutout or protrusion are not planar.

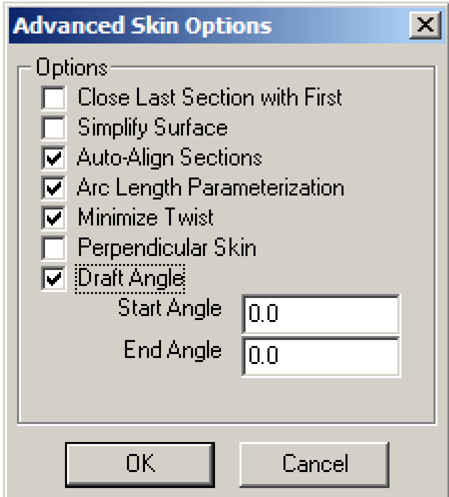


Skinned Solid Tool

The **Skinned Solid** tool creates a solid from a collection of closed profiles. 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 options in the Message Line: **Basic Skinned Solid** and **Skinned Solid with Guide Curves**.

The Advanced Skin Options dialog box provides additional control of the appearance of skinned objects. To activate the box tap the CTRL key (Windows) or Option key (Macintosh). .



It contains the following options:

Close Last Section with First Constructs a body closed in one direction (v direction). The default is an open (not closed) body. The surface is continuous at each profile. If a set of closed profiles is provided, the face normals of the skin or loft body point outside, away from the body material. When a set of open profiles is provided, the surface normals of the skin surface are oriented along the surface normals and no attempt is made to change the surface normal orientation. When using this option profiles are not to be provided in the same position.

Simplify Surface 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 Allows 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 Specifies the 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.

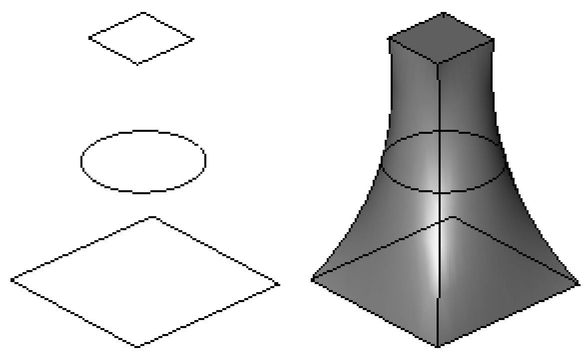
Minimize Twist Minimizes the twist of the surface produced. Twist minimization aligns closed curves such that the start of the second curve is aligned to the start of the first curve. Even if a body's shape is unaffected by twisting, a surface with a twist could produce unexpected results when faceting and rendering. By default, twist minimization is on. Twist minimization is also an involved calculation that may not be carried out.

Perpendicular Skin Specifies the direction of the take–off vector, perpendicular to the edge. The take–off vector is a tangent vector going out of the starting edge or surface and into the skinned surface. The default is in the loft direction because a perpendicular take–off vector can cause self–intersections to the surface.

Draft Angle 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 specifying the angle itself, it is also possible to supply a magnitude for the take–off vector. The draft angle and magnitude is constant for the entire profile. However, different draft angles may be applied 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.

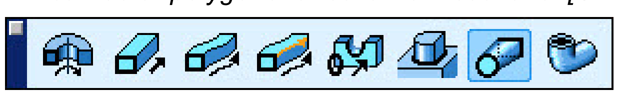
Basic Skinned Solid Option

This option creates a skinned solid between two or more sections. The solid edges are defined by the limits of the selected curves within each section. The figure here shows a skinned solid created from three sections.



Using the Basic Skinned Solid Option

1. Select the **Skinned Solid** tool. The Message Line reads: *Skinned Solid: Pick closed curves or polygons for solid from sections. [Shift = Extend]*



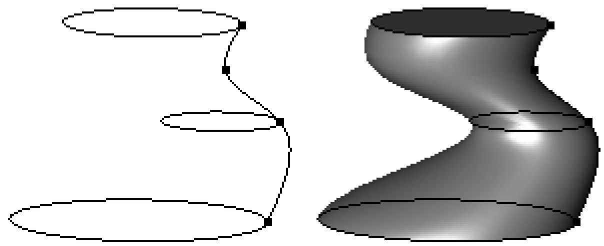
2. Select the **Basic Skinned Solid** option in the Message Line.
3. Hold down the SHIFT key and select the closed curves.

   The solid is created.

There are no entries in the Status Line.

Skinned Solid with Guide Curves Option

This option creates a skinned solid between two or more sections using additional curves as guides to define the skin. Guide curves provide control over the skinned solid. Use one or more sets of 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 but the skin behaves better if the directions are consistent.
- Curves cannot loop and must be “well-behaved.”
- The curves must connect with each profile used for the solid creation.

Using the Skinned Solid with Guide Curves Option

1. Create the section curves and guide curve.
2. Select the **Skinned Solid** tool. The Message Line reads: *Skinned Solid: Pick closed curves or polygons for solid from sections. [Shift = Extend]*
3. Select the **Skinned Solid with Guide Curves** options in the Message Line.
4. Hold down the SHIFT key and select the closed curves for all of the sections.

   Release the SHIFT key and 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 sections and is made up of the same characteristics as any solid object.

Solids from Profiles Curve Checks

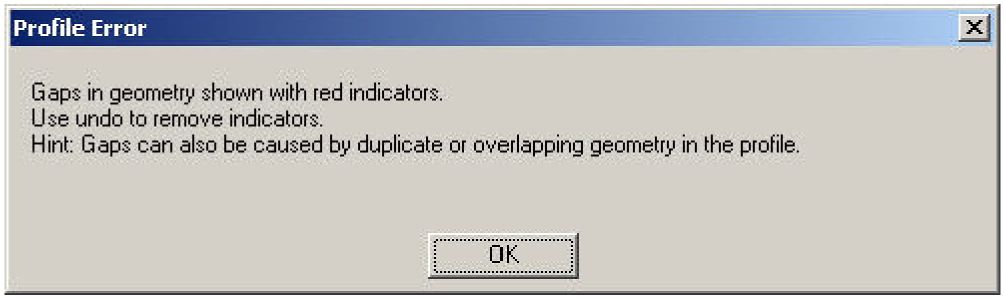
In order to create objects using the **Lathed Solid**, **Extruded Solid** and **Swept Solid** tools, the profiles must be closed and planar.

Closed Curves Check

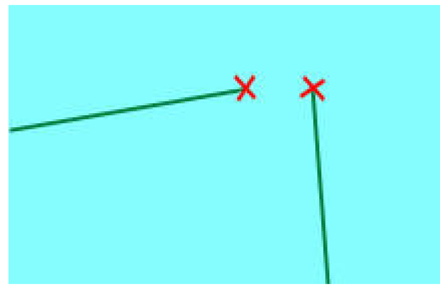
The Closed Curves Check warns if the profile contains gaps. If gaps are detected, two warning messages appear. The first one says, "Gaps detected in profile. Would you like to examine gaps?"

Click No to ignore the gaps and continue

Click Yes, and another message appears: "Gaps in profile are shown with red indicators. Use Undo to remove indicators. Hint: gaps can also be caused by duplicate or overlapping geometry in the profile."



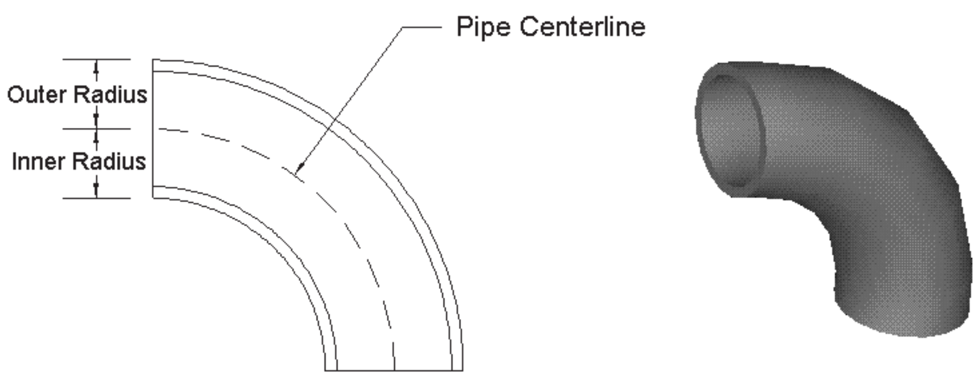
Click OK to remove the message and view the profile with red Xs where gaps are detected. After noting the gaps, use ***Edit>Undo*** to clear the indicators or drag a selection box around them and press Delete. The indicators are created as geometric points, adding geometry to the drawing.



Pipe Solid Tool

The **Pipe Solid** tool provides a quick and easy way to create pipes and tubes. Select a curve for the pipe's center line and specify the pipe's inner and outer diameter values. Create a pipe by selecting curves connected end to end. Create one pipe at a time with this tool.

When making a pipe solid, the program creates an intelligent link between the solid and its parent center line. When moving or modifying the center line, the pipe updates automatically. A pipe with an Inner Diameter of zero is a solid cylinder.



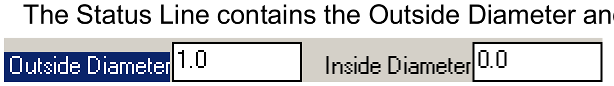
This tool does not support grouped curves for creating a pipe solid.

Using the Pipe Solid Tool

Before using this tool 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 solid cylinder. Tab between the fields.

3. Select the curve. To select more than one curve, hold down the SHIFT key before selecting the first curve.

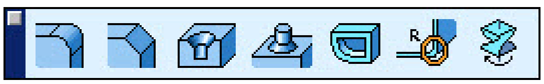
   A pipe is drawn.

While the object is still selected 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.

Solid Feature Tools



After creating a solid model from a primitive or a profile, use any of the feature tools to modify it. These tools cannot be used on grouped objects.

Values for each tool can be entered in the Status Line either before or after creating the feature. When entering the values after selecting the tool but before creating the feature, the first click in the drawing area automatically registers all Status Line values. If values are entered 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 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 the first step is completed (which may involve holding down the SHIFT key to select multiple objects), other objects cannot be selected to include in the same operation. To select additional objects reset the tool and start again.

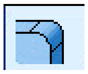
After adding features to the solid, they can be modified by selecting them in the Design Explorer and choosing ***Window>Edit Objects*** or double-clicking on their names in the Design Explorer **(Cobalt and Xenon only)**.

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:

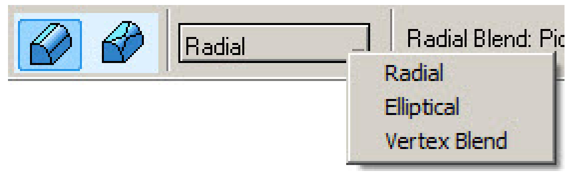
- **Blend Tools**
- **Chamfer Edge Tools**
- **Hole Feature Tools**
- **Boss Feature Tool**
- **Shell Solid Tool**
- **Bend Tool**
- **Twist Solid Tool**

Blend Tools



The **Blend** tools create fillets or rounds along the edge of a solid. When selecting the **Blend** tool, a subpalette appears in the Message Line containing two subtools for creating blends: **Constant Blend** and **Variable Blend**.

Each of these subtools includes a pull-down menu containing options for creating that type of blend.



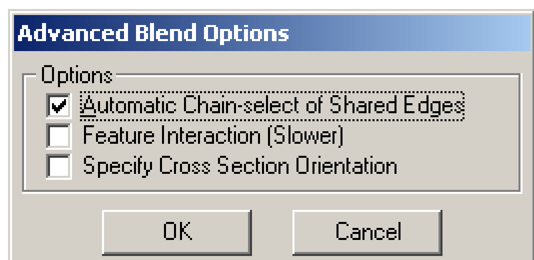
These options include for the **Constant Blend** tool:

- **Radial Blend**
- **Elliptical Blend**
- **Vertex Blend**

For the **Variable Blend** tool they are:

- **Linear Blend**
- **By Position Blend**
- **Fixed Width Blend**
- **Hold Line Blend**

Many of these options also have advanced capabilities that are accessed by tapping the CTRL (Windows) or OPTION (Mac) key. These advanced settings are outlined next.



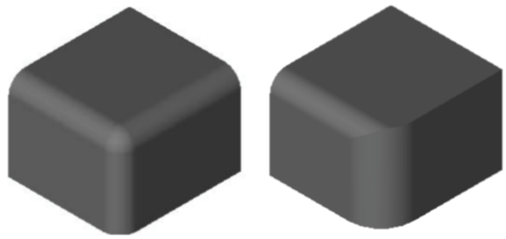
Tech Note: *The* **Blend** *tools perform a similar operation on solids to what the fillet tool performs on a wireframe.*
In the solids world, blend refers to both filleting and rounding. A filleting operation adds material to a concave intersection. A rounding operation removes material from a convex intersection.

Advanced Blend Capabilities

The Advanced Blend Options dialog box contains the following:

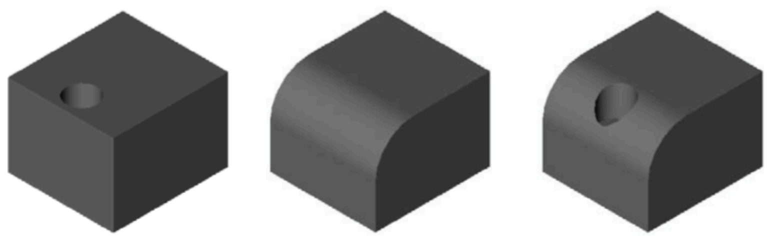
Automatic Chain-select of Shared Edges

Checking this box blends 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

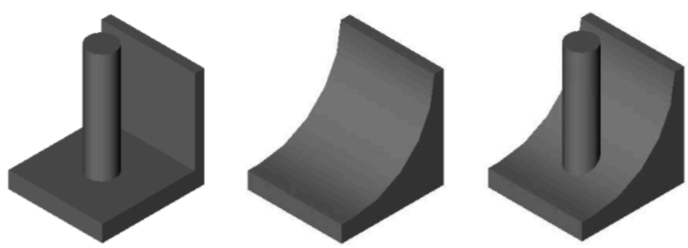


Feature Interaction (Slower)

Checking this box applies 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 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.

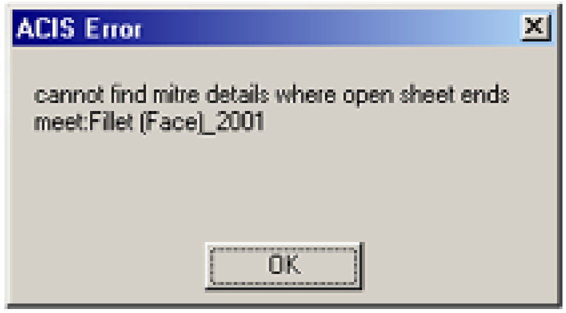


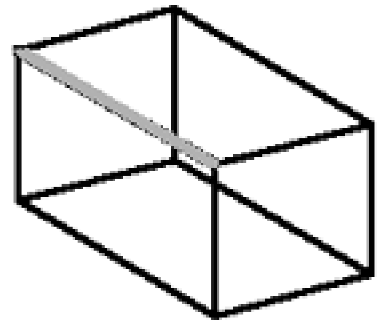
Using this option results in a longer blending time.

Specify Cross Section Orientation

Checking this box specifies the orientation of the cross-section for the blend. This is especially valuable for blending an edge between faces containing a draft and to maintain the arc. See the next section for instructions on using this option.

If the model contains a problem that prevents a successful blending operation, an ACIS Error dialog box appears highlighting the location of the problem.

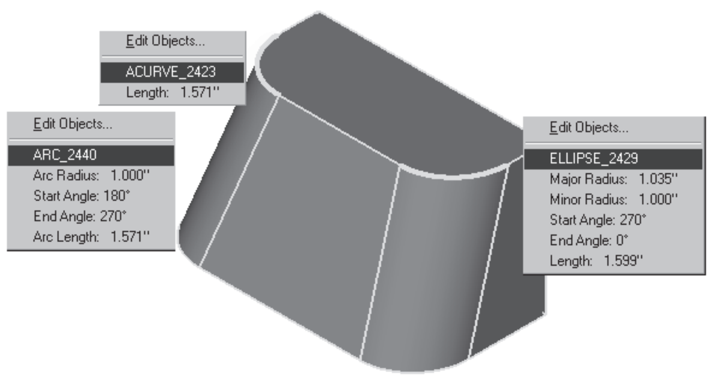




Specifying a Cross Section Orientation for a Blend

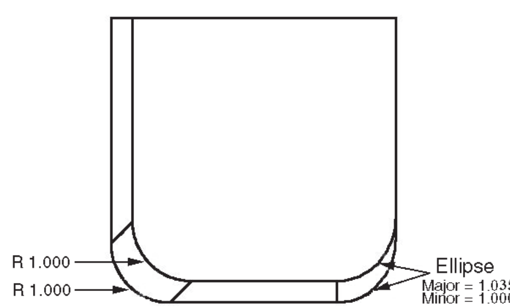
By default, this Designer Elements program uses the selected edge to define the orientation of the cross-section resulting in an arc shape for the blend. However, when applying a blend to an object with a draft or taper, the cross-section of the resulting blend is elliptical because the blended edge is not perpendicular to the intersecting faces. 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. By holding down the right mouse button (Windows) or CONTROL and the mouse button (Macintosh), the object shape is verified. (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.

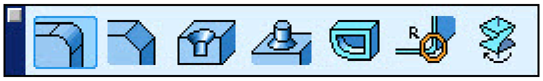
Before using this option, create a curve along the z axis for specifying the cross-section orientation.



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.

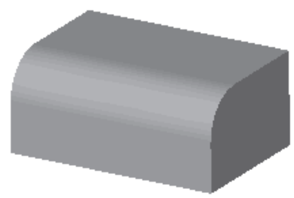


3. In the Message Line select the **Constant Blend** tool and the Radial option for the pull-down menu. The Message Line reads: *Radial Blend: Pick edges/faces to blend. [Ctrl=Advanced (Windows) or Option=Advanced (Macintosh), Shift=Extend]*
4. Press CTRL (Windows) or OPTION (Macintosh) to display the 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.

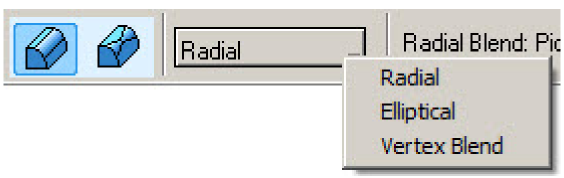
   The Message Line now reads: *Blend Solid: Pick curve to define cross section orientation*.
7. Select the curve.

   The blend is created.

Constant Blend Tool



When selecting this tool, its option pull-down menu appears.



There are three **Constant Blend** options: **Radial**, **Elliptical** and **Vertex Blend**.

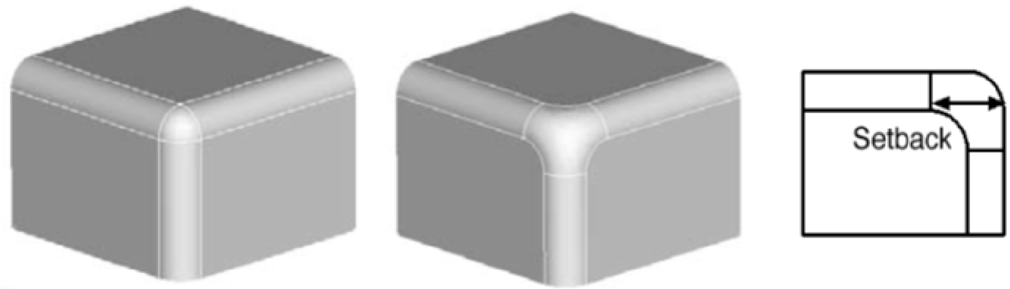
Important: (Windows users) When a **Constant Blend** option uses the CTRL and SHIFT keys to perform specific operations for the tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when blending.

Using the Radial Option

This option adds a blend with a constant radius along the selected edges and with a radius specified in the Status Line.

> ***Tip:*** *When 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.*

By selecting three or more edges, a smooth vertex blend or a setback blend can be created. 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.

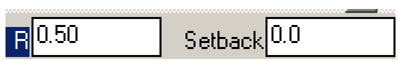


1. Select the **Blend** tool.
2. In the Message Line select the **Constant Blend** subtool and the Radial option from the pull-down menu. The Message Line reads: *Radial Blend: Pick edges/faces to blend. [Ctrl=Advanced (Windows) or Option=Advanced (Macintosh), Shift=Extend]*

   Set the advanced blend options by pressing CTRL (Windows) or OPTION Macintosh), if desired.
3. To create 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, the radius of the blend can be changed 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.



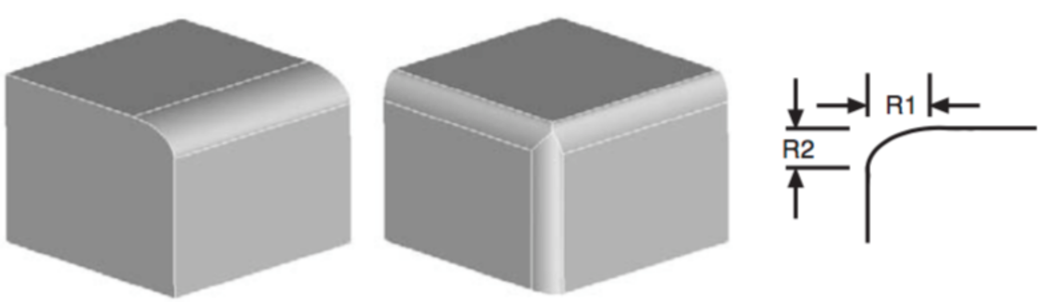
Geometric Characteristics

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.

The blend can be selected 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. The changes to the settings can be made on the Display and Attribute pages. See Design Explorer.

Using the Elliptical Option

This option adds a blend with a constant elliptical radius along the edges, based on the radii specified in the Status Line. When using this option the automatic chain select of Shared Edges feature does not function. 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 tapping CTRL (Windows) or OPTION (Macintosh), if desired.
3. Enter 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, the two radii of the blend can be changed in the Status Line to change the elliptical ration. Type a new value and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the major and minor axis radii for the blend.

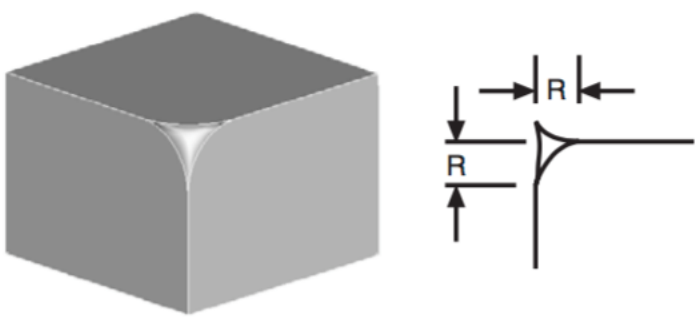
Geometric Characteristics

A constant elliptical blend is created by choosing a solid edge and specifying the 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.

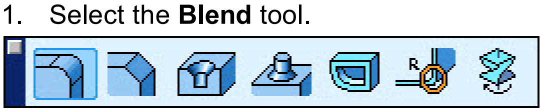
The blend can be selected using the Design Explorer. Double-clicking on the item in the Design Explorer or choosing Edit Parameters command from the Design Explorer menu displays the Edit Objects dialog box. Changes to the settings can be made on the Display and Attribute pages. See “Design Explorer” on page 34.

Using the Vertex Blend Option

This option adds a vertex blend at the intersection of three or more edges based on the radius specified in the Status Line. The left graphic shows an example of a vertex blend. The right graphic illustrates the radius variables.



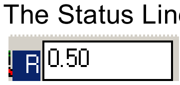
1. Select the **Blend** tool.



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. [SHIFT = Extend]*
3. Enter a radius in the Status Line data field.
4. Select the vertex for the blend.

While the blend is still selected the radius of the blend can be changed 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.

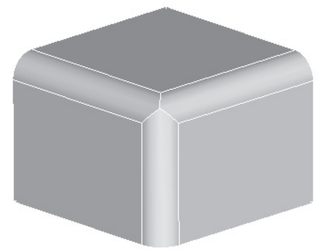
The blend can be selected using the Design Explorer. Double-clicking on the item in the Design Explorer or choosing Edit Parameters command from the Design Explorer menu displays the Edit Objects dialog box. Changes to the settings are made on the Display and Attribute pages. See “Design Explorer” on page 34.

Additional Examples

The **Constant Blend** tool creates a variety of blends. Here are some examples of advanced blends and the steps to create them.

Mitered Blend

A mitered blend is 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

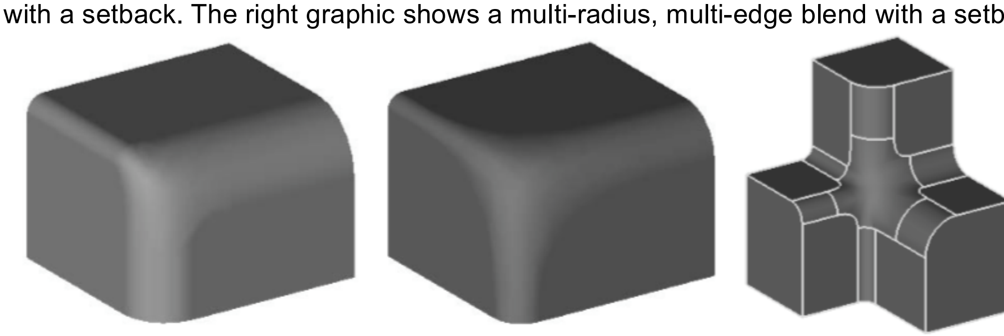
1. Select the **Blend** tool.
2. In the Message Line, select the **Constant Blend** tool and the **Radial** option from the pull-down menu. The Message Line reads: *Radial Blend: Pick edges/faces to blend. [Ctrl=Advanced (Windows) or Option=Advanced (Macintosh), Shift=Extend]*
3. Press CTRL (Windows) 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 the SHIFT key is held down, a smooth vertex blend will be created rather than a mitered blend.)

   To have 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 with and without 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. A setback to this blend can be added 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.



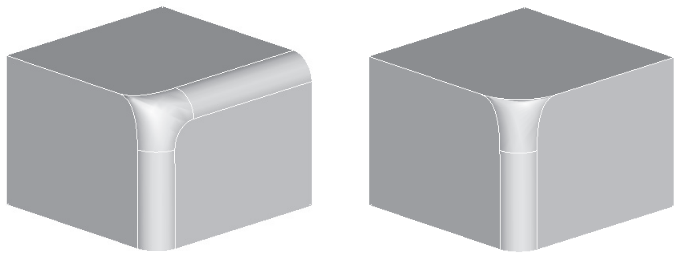
1. Select the object.
2. Choose ***Window>Design Explorer***.
3. Expand the history tree to display the fillet blend. (**Cobalt** and **Xenon** only)

4. Double-click on the fillet blend to display the Edit Objects dialog box.
5. Select the edge with the radius 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 the geometry updates.

Two-edge or One-edge Vertex Blend

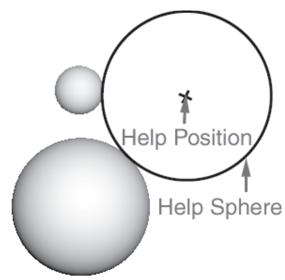
Create two- and one-edge vertex blends by first creating a constant radial blend. Using the Design Explorer, double-click on the blend to display the Edit Objects dialog box. For a two-edge vertex blend change the radius of one-edge to zero. For a one-edge vertex blend, change two radii to zero. The left graphic below shows the two-edge vertex blend and the right graphic a one-edge vertex blend.



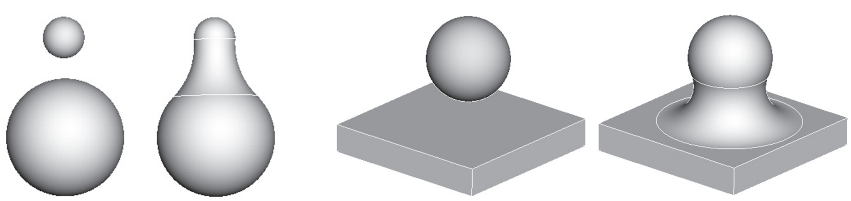
Disjoint Blend

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 it was necessary 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 the 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

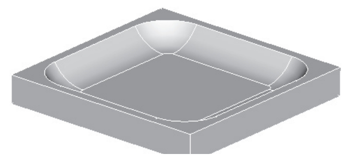
1. Select the **Blend** tool.
2. In the Message Line, select the **Constant Blend** tool and the **Radial** option from the pull-down menu. The Message Line reads: *Radial Blend: Pick edges/faces to blend. [Ctrl=Advanced (Windows) or Option=Advanced (Macintosh), Shift=Extend]*
3. Hold the SHIFT key down 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 the Designer Elements program specifies a radius for the blend that is greater than the distance from the edge to the face.

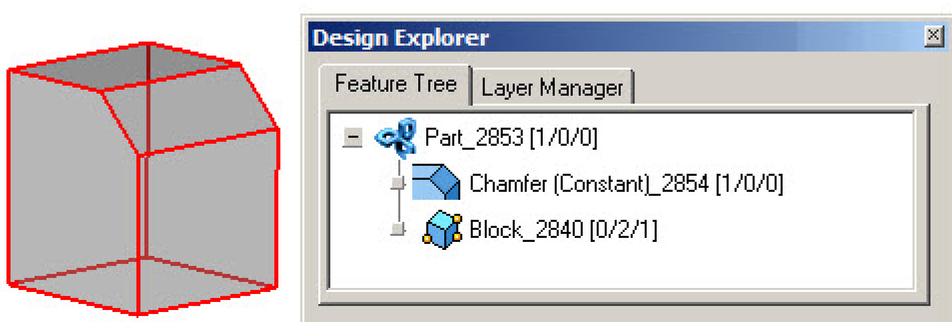
In the graphic here a one inch blend is applied using the inside top edge of the object and its inside bottom face. The user supplied help position is important in this blend. If the wrong location is chosen, the blend will not work. The one inch radius is larger than the distance from the edge to the face.



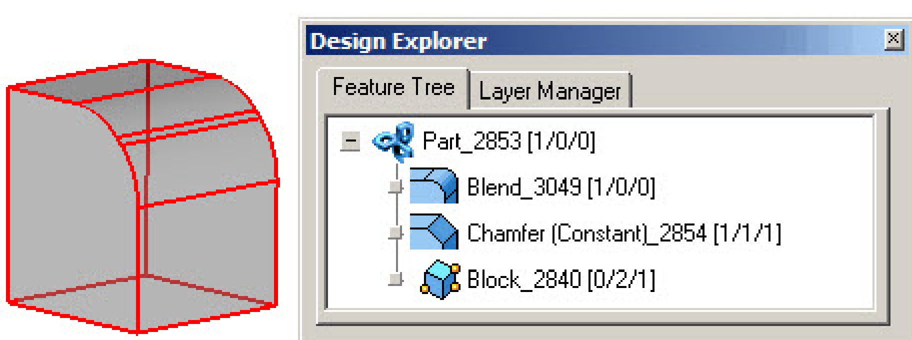
Disjoint Face-to-face Remote Blend

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 separated by the chamfer).

Before

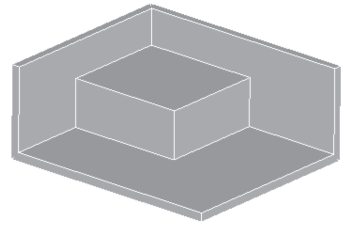


After



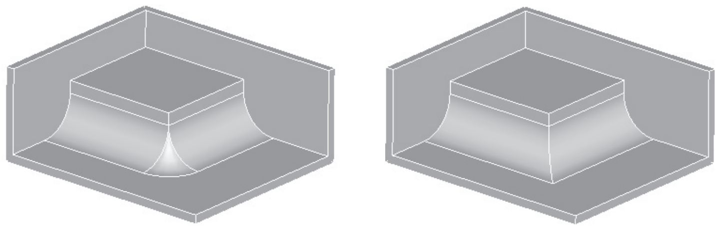
Notice in the Design Explorer, shown above, that the fillet blend does not replace the chamfer but follows the chamfer operation in the history tree.

Edge-to-face Roll Blend

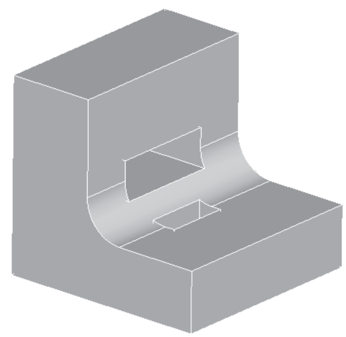


An edge-to-face roll blend is created by specifying a radius, selecting an edge and face to blend and choosing a help position. The graphic to the right shows the original object used for this blend.

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.



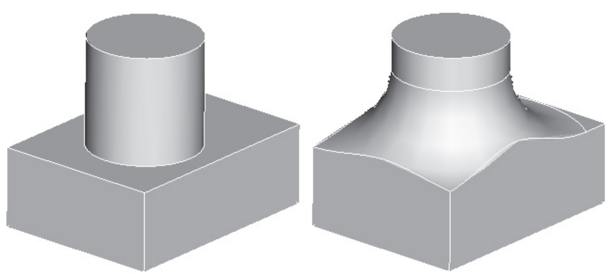
Blend with Cutouts



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 applying 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.

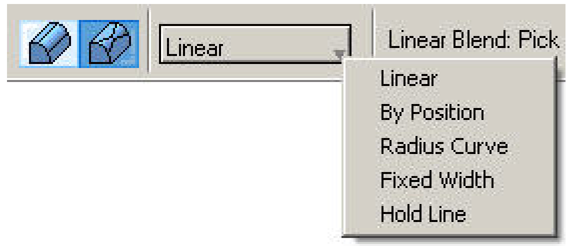


Creating an Extrapolation Blend

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/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



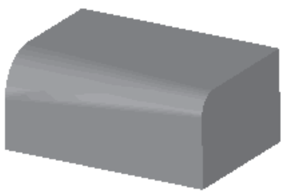
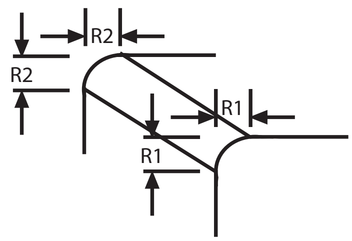
When selecting this tool its option pull-down menu appears.

There are five variable blend options: **Linear**, **By Position**, **Radius Curve**, **Fixed Width** and **Hold Line**.

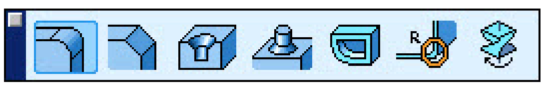
Important: (Windows users) When a **Variable Blend** option uses the CTRL and SHIFT keys to perform specific operations for the tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when blending.

Using the Linear Option

This option adds a blend with a tapered radius along the selected edges. 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]*
3. Select the edge along which to make the blend.

   The solid edge closest to the point of selection becomes R1.

While the blend is still selected the radii (R1 and R2) can be changed in the Message Line. Type the new values and press ENTER (Windows) or RETURN (Macintosh).

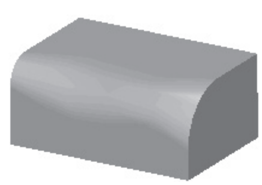
The Status Line contains R1 (beginning radius) and R2 (ending radius) for the linear blend.

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.

The blend can be selected 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. Changes to the settings can be made on the Display and Attribute pages. See Design Explorer.

Using the By Position Option



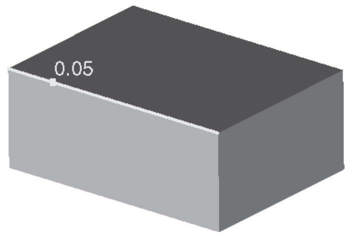
This option adds a blend with a varied radius along the selected edges. Specify the various radii in the Status Line.

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.

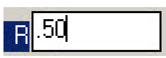
   As a radius is applied to a location the value appears on the edge.



6. Enter a new radius in the R field.
7. Specify the location for this radius along the edge.

   Continue following this procedure until all the desired radii have been entered. Double-click the last place radius point and the variable by position blend is created.

The Status Line contains the Radius for the blend.



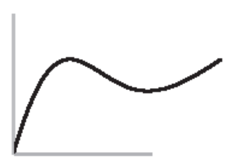
Geometric Characteristics

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 selecting 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.

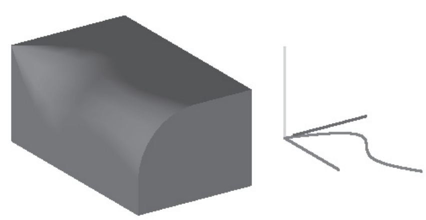
Select the blend using the Design Explorer. Double-clicking on the item in the Design Explorer or choosing Edit Parameters command from the Design Explorer menu displays the Edit Objects dialog box. Make changes to the settings using the Display and Attribute pages. “Design Explorer” on page 34.

Using the Radius Curve Option

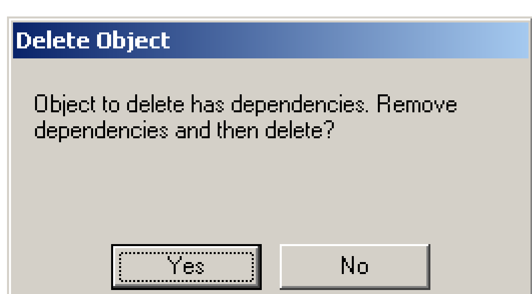
This option adds a blend based on a specified radius curve. Before using this option, create an open curve in the positive **xy** quadrant. This curve cannot cross over itself or backtrack. A circle or a loop would not make acceptable curves. The graphic here is an example of an acceptable curve.



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 percentage 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. It is also possible to 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. Attempting 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.

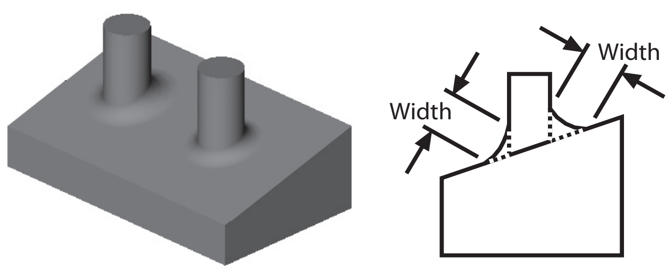
   There are no Status Line entries.

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. Changes to the Display and Attribute settings cannot be made through the Edit Objects dialog box. The blend radius cannot be changed 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

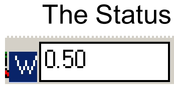
This option adds a blend with a fixed width along the selected edges adjusting the blend radius to maintain the width. This is important when placing a blend along the edge of two faces where the angle changes. Specify the width in the Status Line. In the graphic, the left boss has a constant radial 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/ 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.

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. Make changes to the settings on the Display and Attribute pages. "Design Explorer" on page 34 for more information.

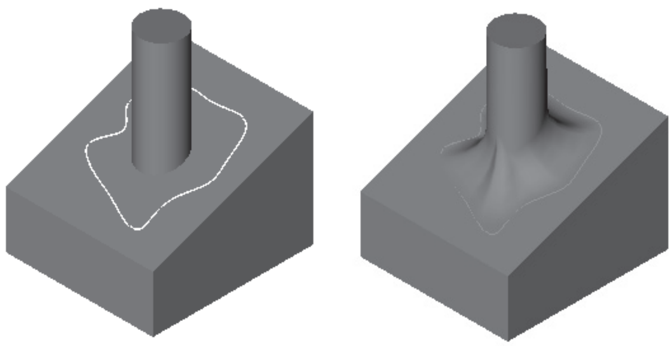
Using the Hold Line Option

This option adds a blend to the hold line curve specified, adjusting the blend radius as needed to fit the line.

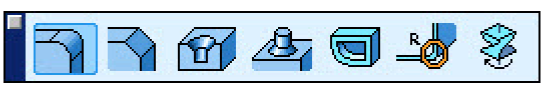
Hold Line Rules:

- 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/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.

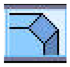
   A hold line blend is created.

   There are no Status Line entries.

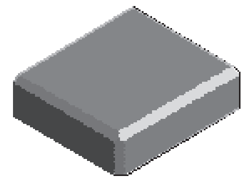
Geometric Characteristics

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. Changes are made to the Display and Attribute settings through the Edit Objects dialog box.

Chamfer Edge Tools

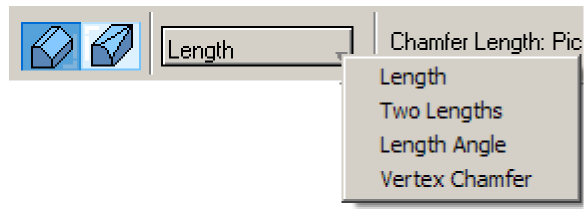


The **Chamfer Edge** tools create flat surfaces along a specified edge. The graphic here shows a chamfered solid along all four edges.



When the **Chamfer Edge** tool is selected, a subpalette appears in the Message Line containing two chamfer edge subtools: **Constant Length** and **Variable Length**.

Each of these tools includes a pull-down menu containing options for creating that type of chamfer. For the Constant Length tool these include:

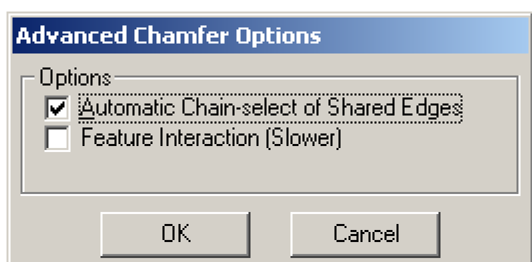


- **Constant Length Chamfer**
- **Two Lengths Chamfer**
- **Length Angle Chamfer**
- **Vertex Chamfer**

For the **Variable Length** tool they are:

- **Variable Lengths Chamfer**
- **Four Lengths Chamfer**
- **Lengths Angles Chamfer**
- **By Position Chamfer**
- **Fixed Width Chamfer**

Many of these options also have advanced capabilities that are accessed by tapping the CTRL (Windows) or OPTION (Mac) key. These advanced settings are outlined next.

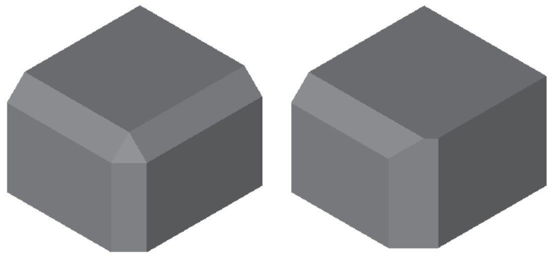


Advanced Chamfer Capabilities

The Advanced Chamfer Capabilities dialog box contains the following options:

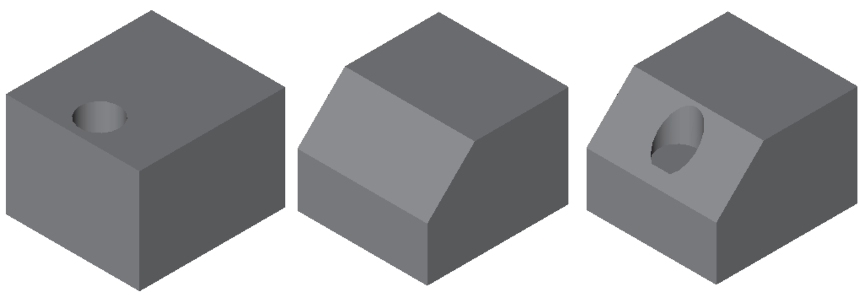
Automatic Chain-selectof Shared Edges

Checking this box chamfers 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 using the **Constant Length Chamfer** tool with the option selected. The right graphic shows two edges chamfered using the **Constant Length Chamfer** tool without the option selected.

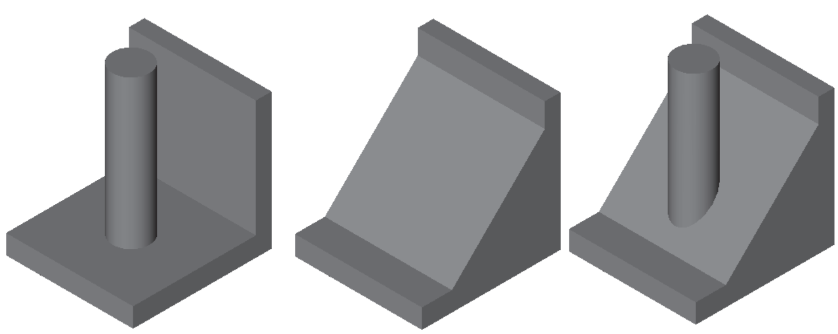


Feature Interaction (Slower)

Checking this box applies 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.



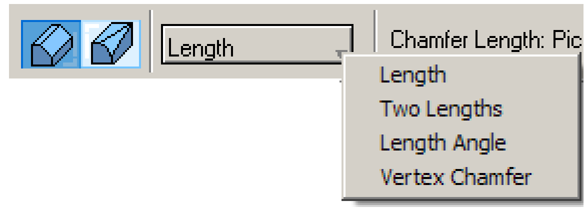
Without this checked for a protrusion the chamfer takes precedence over the protrusion. The left graphic below is the original part. The middle graphic is the part without the option selected. The right graphic is the part with the option selected.



Using this option results in a longer chamfering time.

Constant Length Chamfer Tool

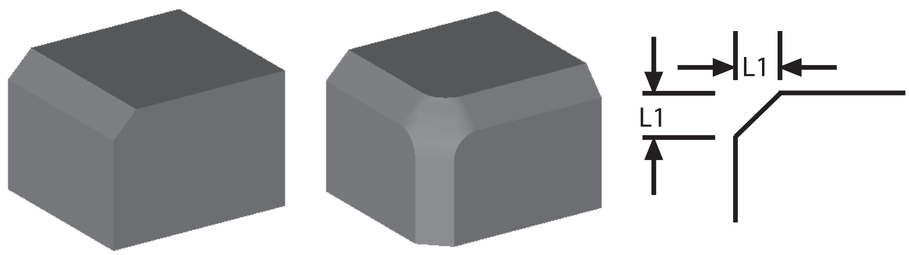
When this tool is selected, its option pull-down menu appears.



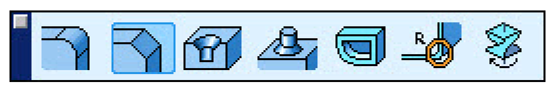
There are four constant length options: **Length**, **Two Lengths**, **Length Angle** and **Vertex Chamfer**.

Using the Length Option

This option adds a 45° chamfer based on the specified length to edge distance along the selected edges or faces. 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. (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.

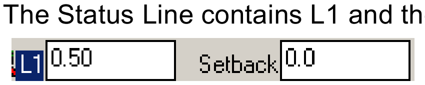


2. In the Message Line, select the **Constant Length Chamfer** tool and the **Length** option from the pull-down menu. The Message Line reads: *Chamfer Edge: Pick solid edges/faces to chamfer. [Ctrl = Advanced (Windows) or Option = Advanced (Macintosh)] [Shift = Extend]*
3. Enter a setback value in the Setback data field of the Status Line, if desired, for three or more intersecting edges.
4. Select the edges or faces of a part to chamfer. Use the Ambiguity box to specify the edge or a face if necessary.

   The chamfer is created.

While the chamfer is still selected, it is possible to change L1 and the setback length in the Status Line or the Edit Objects dialog box. Type the desired values in the data field and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains L1 and the Setback.

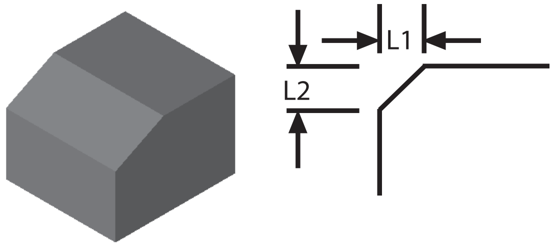


Geometric Characteristics

A constant length chamfer is created by choosing a solid edge or face 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.

Using the Two Lengths Option

Using this option adds a chamfer based on the specified length to edge distances along the selected edges or faces to create a constant length chamfer.



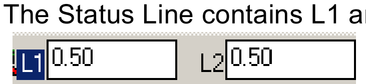
1. Select the **Chamfer Edge** tool.
2. In the Message Line, select the **Constant Length Chamfer** tool and the **Two Lengths** option from the pull-down menu. The Message Line reads: *Chamfer Edge: Pick solid edges/faces to chamfer. [Ctrl = Advanced (Windows) or Option = Advanced (Macintosh)] [Shift = Extend]*
3. Enter L1 and L2 values in the Status Line.
4. Select the edges or faces of a part to chamfer. Use the Ambiguity box to specify the edge or a face If necessary.

   The chamfer is created.

While the chamfer is still selected it is possible to change the L1 and L2 values in the Status Line or the Edit Objects dialog box. Type the desired values in the data field and press ENTER (Windows) or RETURN (Macintosh).

Swap chamfer values by tapping the Ctrl (Windows) or Option (Mac) key.

The Status Line contains L1 and L2.

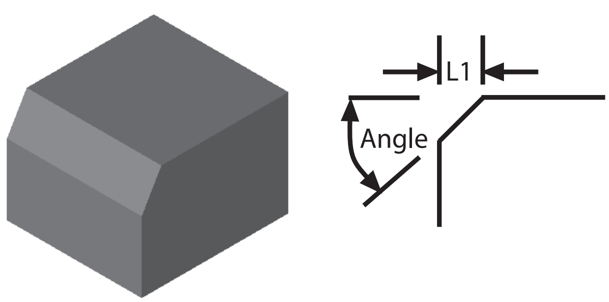


Geometric Characteristics

A constant two length chamfer is created by choosing a solid edge or face and specifying the lengths to the edge. It is made up of two characteristics according to Edit Objects: Length 1 and Length 2.

Using the Length and Angle Option

This option adds a chamfer at an angle and a constant length to the edge specified in the Status Line. The angle is measured from the tangent of the selected edge to the chamfered edge (see the right graphic).



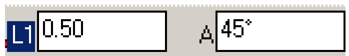
1. Select the **Chamfer Edge** tool.
2. In the Message Line, select the **Constant Length Chamfer** tool and the **Length and Angle** option from the pull-down menu. The Message Line reads: *Chamfer Length Angle: Pick solid edges/faces to chamfer. [Ctrl = Advanced (Windows) or Option = Advanced (Macintosh)] [Shift=Extend]*

3. Enter an L1 and angle in the Status Line data fields.
4. Select the edges or the faces of a part to chamfer. Use the Ambiguity box to specify the edge or a face when performing the operation.

While the chamfer is still selected it is possible to change the length and angle in the Status Line or Edit Objects dialog box as needed. Type the desired values in the data fields and press ENTER (Windows) or RETURN (Macintosh).

Swap chamfer values by tapping the CTRL (Windows) or OPTION (Mac) key.

The Status Line contains L1 and the A chamfer angle data fields.

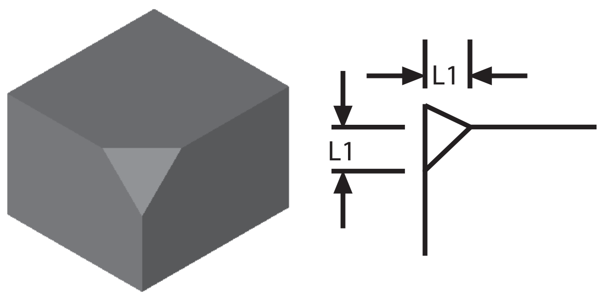


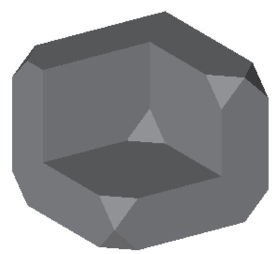
Geometric Characteristics

A constant length angle chamfer is created by choosing a solid edge or face 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.

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.



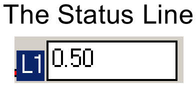


The faces next to the intersecting edges must be planar and have the same convexity or concavity. The graphic here shows convex and concave vertex chamfers.

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 Chamfer: Select vertex to* chamfer. [SHIFT = Extend].
3. Enter an L1 value in the Status Line data field.
4. Select the vertex to chamfer.

While the chamfer is still selected it is possible to change the length in the Status Line or Edit Objects dialog box. Type the desired value in the data field and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains L1 chamfer length.



Geometric Characteristics

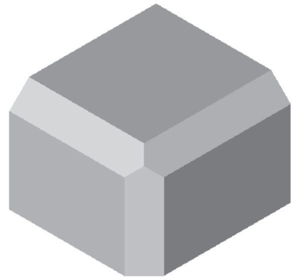
A vertex corner chamfer is created by choosing a vertex and specifying a length. It is made up of one characteristic according to Edit Objects: Radius 1.

Additional Examples

Here are a few chamfer variations using the constant chamfer tools.

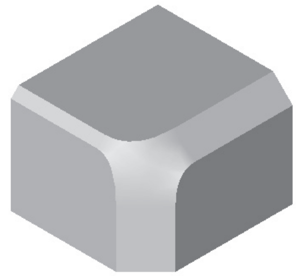
Multi-edge Chamfer with Mitered Corner

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.

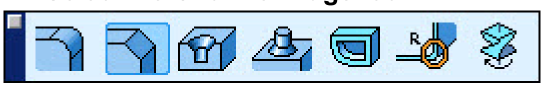


Multi-edge Chamfer with Varied Lengths and Setback

This chamfer is created using the **Length** option of the **Constant Length Chamfer** tool.

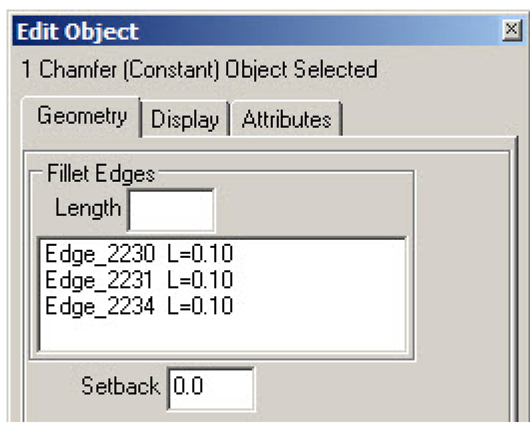


1. Select the **Chamfer Edge** tool.



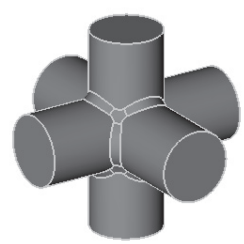
2. In the Message Line, select the **Constant Length Chamfer** tool and the **Length** option from the pull-down menu. The Message Line reads: *Chamfer Edge: Pick solid edges/faces to chamfer. [Ctrl = Advanced (Windows) or Option = Advanced (Macintosh)] [Shift=Extend]*
3. Enter a setback value in the 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

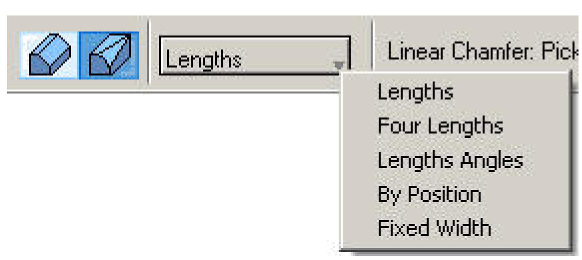
This Designer Elements program supports complex chamfering for intersecting objects with or without setbacks. The example here includes setbacks.



Variable Chamfer Edge Tool

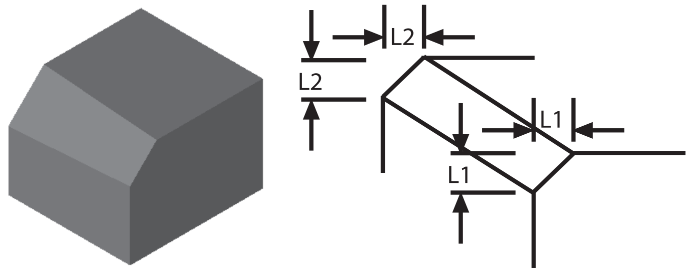
When this tool is selected its pull-down option menu appears.

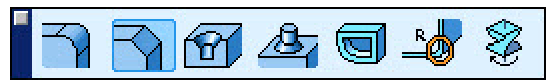
There are five variable length options: **Lengths**, **Four Lengths**, **Lengths Angles**, **By Position** and **Fixed Width**.



Using the Lengths Option

This option adds a chamfer tapered linearly along the edge from two specified setback lengths, L1 and L2.



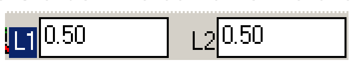


1. Select the **Chamfer Edge** tool.
2. In the Message Line, select the **Variable Length Chamfer** tool and the **Lengths** option from the pull-down menu. The Message Line reads: *Linear Chamfer: Pick edges/faces to linearly chamfer. (Pick edge near L1) [Ctrl = Advanced (Windows) or Option = Advanced (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 edges or the faces of a part to chamfer. Use the Ambiguity box to specify the edge or a face when performing the operation.

While the chamfer is still selected it is possible to change the setback lengths (L1 and L2) in the Status Line or Edit Objects dialog box. Type the desired values in the data fields and press ENTER (Windows) or RETURN (Macintosh).

Swap chamfer values by tapping the CTRL (Windows) or OPTION (Mac) key.

The Status Line contains the chamfer setback lengths, L1 and L2.

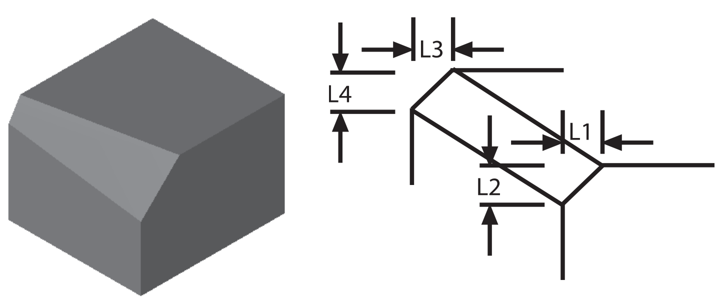


Geometric Characteristics

A variable length 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.

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 Length: 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 edges or the faces of a part to chamfer. Use the Ambiguity box to specify the edge or a face when performing the operation.

While the chamfer is still selected it is possible to change the setback lengths in the Status Line or the Edit Objects dialog box. Type the desired values in the data fields and press ENTER (Windows) or RETURN (Macintosh).

Swap chamfer values by tapping the Ctrl (Windows) or Option (Mac) key.

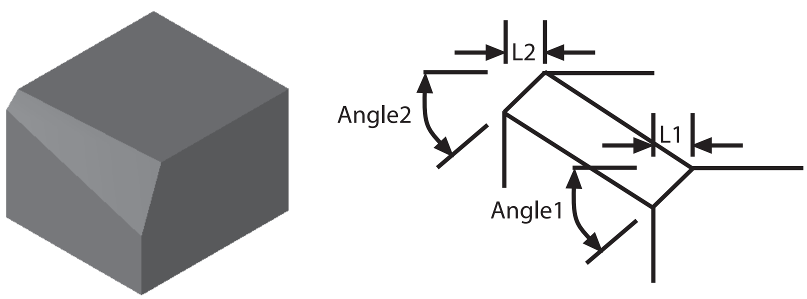
The Status Line contains the chamfer setback lengths, L1, L2, L3 and L4.

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.

Using the Variable Lengths and 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).

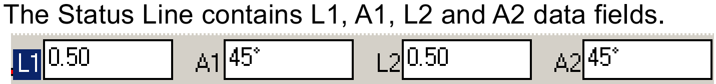


1. Select the **Chamfer Edge** tool.
2. In the Message Line, select the **Variable Length Chamfer** tool and the **Lengths and Angles** option from the pull-down menu. The Message Line reads: *Chamfer Length Angle: Pick solid edges/faces to chamfer [Ctrl = Advanced (Windows) or Option = Advanced (Macintosh)] [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 edges or the faces of a part to chamfer. Use the Ambiguity box to specify the edge or a face when performing the operation.

While the chamfer is still selected, it is possible to change the lengths and angles in the Status Line or the Edit Objects dialog box. Type the desired values in the data fields and press ENTER (Windows) or RETURN (Macintosh).

Swap chamfer values by tapping the Ctrl (Windows) or Option (Mac) key.

The Status Line contains L1, A1, L2 and A2 data fields.

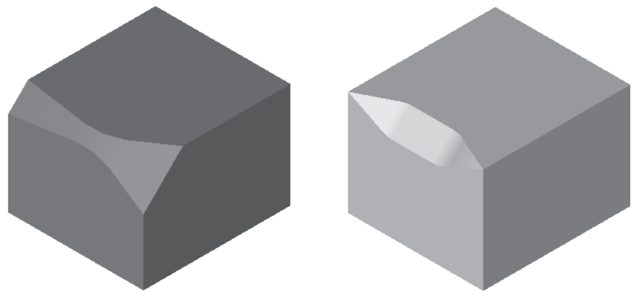


Geometric Characteristics

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.

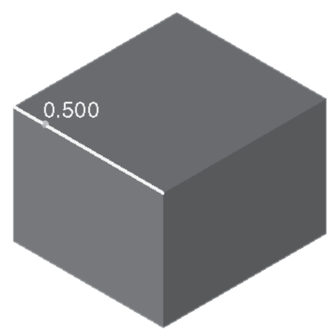
Using the Variable By Position Option

This option adds a chamfer with a varied lengths along the edges selected. Specify the lengths in the Status Line. To create a degenerative chamfer with this option specify 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, Double 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.

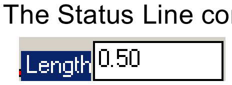
   When specifying the location the length appears in red at that point and all additional locations. The values remain until the last chamfer point double-clicked and the chamfer is created. The graphic illustrates this.



6. Enter a new length in the Length field.
7. Specify the location for this length along the edge.

   Continue following this procedure until all the desired lengths are entered. 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.

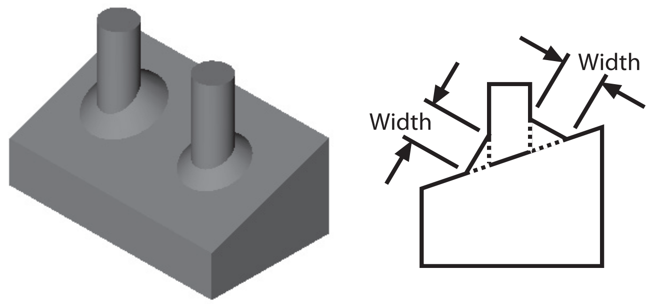


Geometric Characteristics

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 one of the items in the list is selected its Length displays in the Width data field and its percentage point location displays in the Param field.

Using the Fixed Width Option

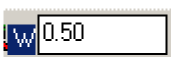
This option adds a chamfer with a fixed width along the selected edges, adjusting the chamfer length to maintain the width. This is important when placing a chamfer along of the edge of two faces where the angle changes. Specify the width in the Status Line. The left graphic shows a sloped block with 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/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 it is possible to change the width in the Status Line or the Edit Objects dialog box. Type the desired values in the data fields and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the width for the chamfer.

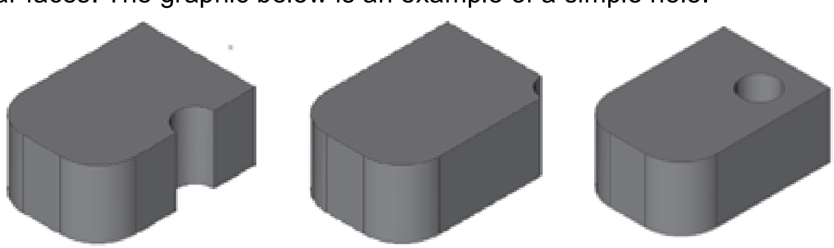


Geometric Characteristics

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 one of the items in the list is selected its width displays in the Width data field**.**

Hole Feature Tools

The **Hole Feature** tools model bolt holes found frequently in mechanical design parts. These 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.

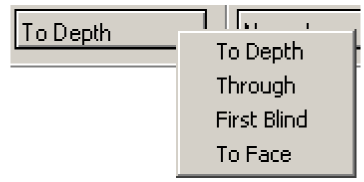


Holes are associative in Cobalt and Xenon. Modify them at any point in the design process.

When selecting the **Hole Feature** tools a subpalette appears containing three tools for creating holes: **Simple**, **Counter Bore** and **Counter Sink**.

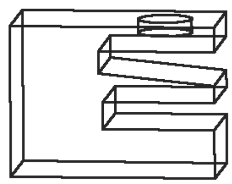
Through Types

The Message Line also contains a pull-down menu to choose a Through Type for the hole. There are four options: **Depth**, T**hrough**, **First Blind** and **To Face**.

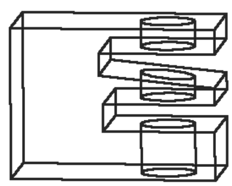


Tech Note: *To place a hole through an object that will intersect a blend, use the Depth option rather than the To Face option. If To Face is used, when the hole intersects the blend, a face change occurs and the hole ends.*

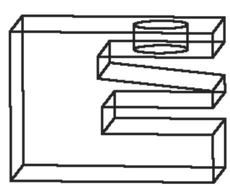
Depth Extends the depth of the hole to the value specified in the Status Line.



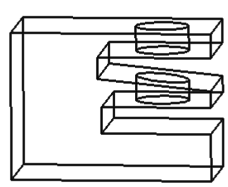
Through Extends a hole through the selected solid.



First Blind Extends a hole to the first open face.

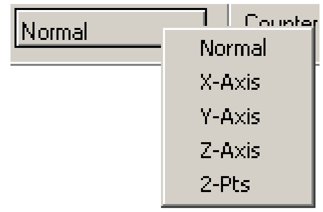


To Face Extends a hole to the selected face.



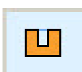
Hole Direction

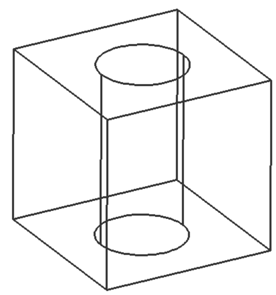
The Message Line contains a pull-down menu to specify a boss direction. There are five options: **Normal**, **X-Axis**, **Y-Axis**, **Z-Axis** and **2-Pts**.



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

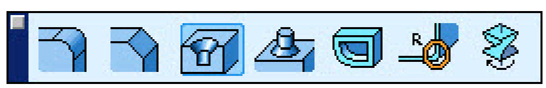




This tool creates straight, cylindrical holes through the selected solid. Specify the X, Y and Z center location, depth, diameter and draft angle of the hole in the Status Line.

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 to locate the hole.
5. Select the location for the center of the hole.
6. If the 2-Pts direction is selected, click two points to specify the direction.

   The hole is created.

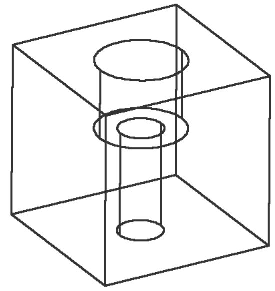
While the solid is still selected, the hole depth can be changed in the Status Line. Type the desired value in the data field and press ENTER (Windows) or RETURN (Macintosh).

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. For more information on the Design Explorer, see Edit Objects.

Counter Bore Tool



This tool creates a counter bore hole based on the values specified 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 the 2-Pts direction is selected, click two points to specify the direction.

   The bore is created.

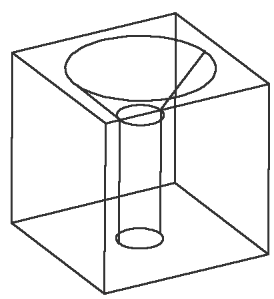
While the solid is still selected the depth, diameter and draft angle of the hole and the bore depth and radius can be changed in the Status Line. Type the desired values in the data fields and press ENTER (Windows) or RETURN (Macintosh).

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 Design Explorer for more information.

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 to locate the hole.

5. Select the location for the center of the hole.
6. If the 2-Pts direction is selected, click two points to specify the direction.

   The hole is created.

While the solid is still selected the depth, diameter and draft angle, sink radius and sink angle can be changed in the Status Line. Type the desired values in the data fields and press ENTER (Windows) or RETURN (Macintosh).

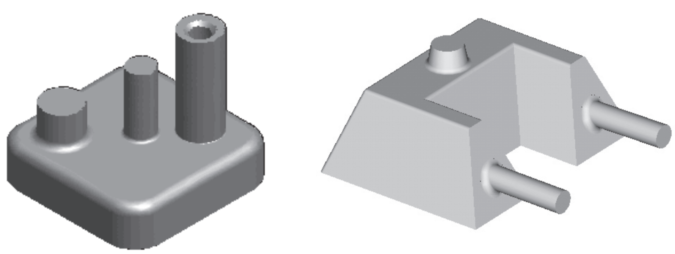
The Status Line contains the countersink's Depth, Diameter, Draft Angle, Sink Diameter and Sink Angle.

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. “Design Explorer” on page 34 for more information.

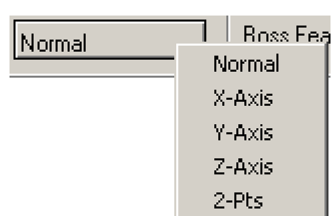
Boss Feature Tool

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. Specify the height, diameter and fillet radius in the Status Line.



Boss Direction

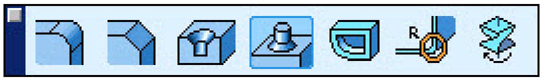
The Message Line contains a pull-down menu to specify a boss direction. There are five options: **Normal**, **X-Axis**, **Y-Axis**, **Z-Axis** and **2-Pts**.



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 the 2-Pts direction is selected, click two points to specify the direction.

   The boss is created.

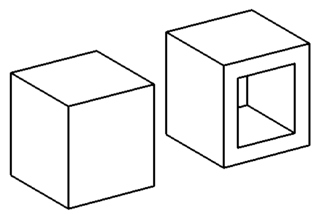
While the solid is still selected the height, diameter and fillet radius of the boss can be changed in the Status Line. Type the desired values in the data fields and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the boss' Height, Radius and Fillet Radius.

Geometric Characteristics

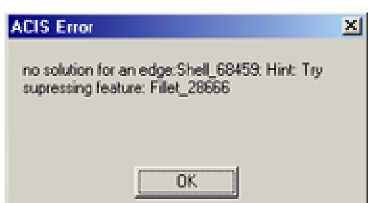
A boss is created by choosing the solid face and specifying the values for the cylinder. It is made up of these characteristics according to Edit Objects: Height, Diameter and Fillet Radius. To modify the boss characteristics, select the boss using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the boss name. See Design Explorer for more information.

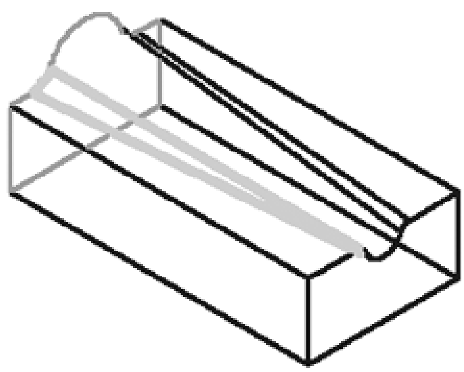
Shell Solid Tool



The **Shell Solid** tool hollows out a solid with a thickness specified.

This tool also includes an error detection routine. If the 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.



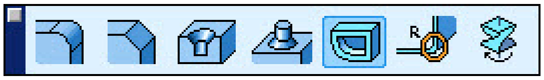


After shelling, the face thickness can be edited in the Edit Objects dialog box.

This tool also gives the option of keeping the core of the shelled object by using the CTRL (Windows) or SHIFT (Mac) key.

Using the Shell Solid Tool

1. Select the **Shell Solid** tool. The Message Line reads: *Shell Solid: Pick solid to shell [Ctrl = Keep Core (Windows) or Option = Keep Core (Macintosh)]*



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.

   Clicking in an area with no faces, the object will be hollowed out with no open faces.

While the solid is still selected, a new value may be entered in the offset field. Press ENTER (Windows) or RETURN (Macintosh) to accept the change.

The Status Line contains the offset or shell thickness.

> ***Tech Note:*** *Like all tools, set the thickness of the shell after the operation, using the Status Line. When working with small objects, it may be necessary to change the default thickness first to shell successfully.*

Geometric Characteristics

A shelled solid is created by choosing a solid, specifying an offset and the face(s) to 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 the operation is selected in the Design Explorer first and then display the Edit Object dialog box. See Design Explorer for more information.

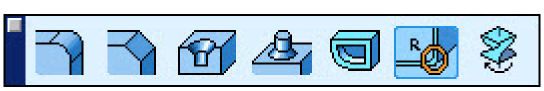
Bend Tool

The Bend tool creates a solid by bending a solid about an axis through a given radius.

There are three subtools in the Message Line: **Bend One End, Bend Center** and **Bend Along Path**.

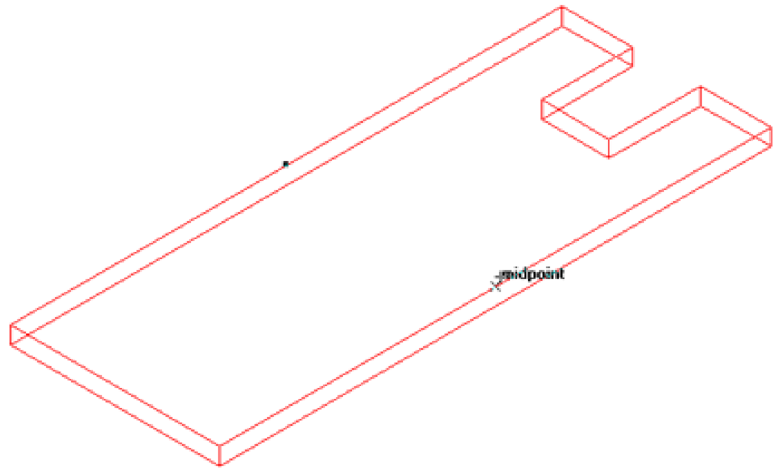
Using Bend One End

1. Click the **Bend Solid** tool. Select the **Bend One End** tool in the Message Line. The Message Line reads: *Bend One End: Pick Solid to Bend*.

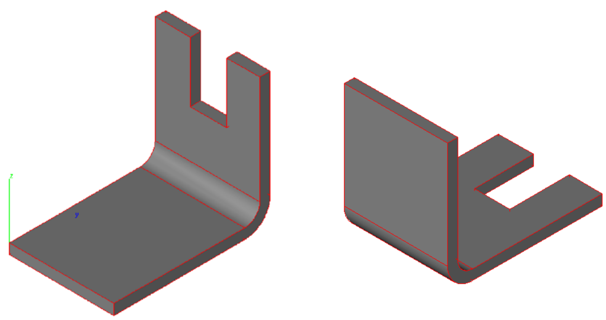


2. The Status Line contains the Bend Radius and Bend Angle fields. Enter the desired values for the bend solid. Tab between data fields.
3. Pick the solid to bend.

4. The message line now reads, *Bend One End: 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 clicking the two points, the solid is bent about the axis.



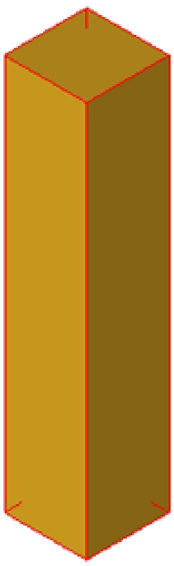
5. The Message Line now reads: *Bend One End: Pick Solid to Bend [Option = Flip Material]*. Press the CTRL (Windows) or OPTION (Mac) key to flip the end that is bent.



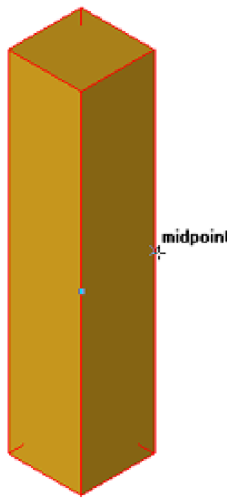
While the object is still selected, the bend radius and bend angle can be edited. Enter the new values and press ENTER (Windows) or RETURN (Macintosh) and the solid updates

Using Bend Center

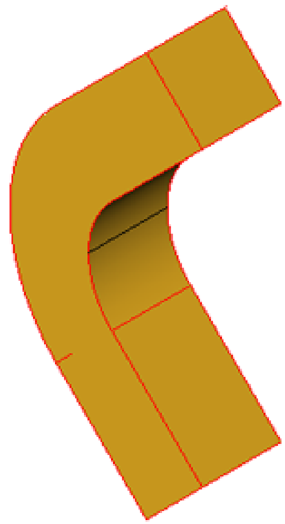
1. Click the **Bend Solid** tool. Select the **Bend Center** subtool in the Message Line. The Message Line reads: *Bend Center: Pick Solid to Bend*.
2. The Status Line contains the Bend Radius and Bend Angle fields. Enter the desired values for the bend solid. Tab between data fields.
3. Pick the solid to bend.
4. The message line now reads, *Bend Center: 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 clicking the two points, the solid is bent about the axis.
5. The Message Line now reads: *Bend Center: Pick Solid to Bend [Ctrl = Flip Material (Windows) or Option = Keep Material (Macintosh)]*. Press the CTRL (WIndows) or OPTION (Mac) key to flip the side of the material that was bent.



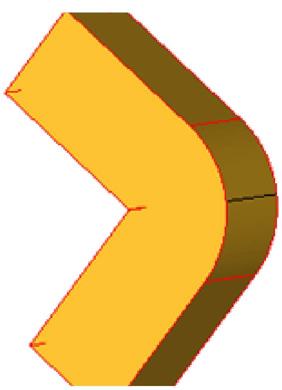
Solid to Bend



Bend Axis

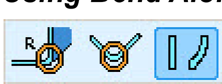


Resulting Bend



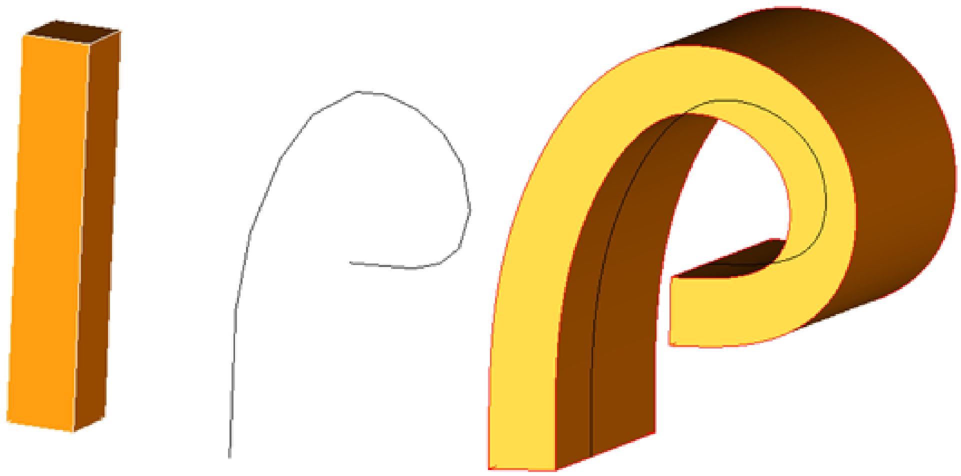
Flipped Material

Using Bend Along Path



1. Click the **Bend Solid** tool. Select the **Bend Along Path** subtool the Message Line. The Message Line reads: *Bend Along Path: Pick Solid to Bend*.
2. Pick the solid to bend.

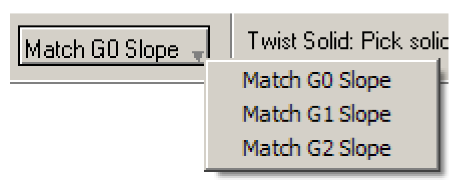
3. The message line now reads, *Bend Along Path: Pick curve to bend solid to.* Choose the curve.
4. The Message Line now reads, *Bend Solid: Enter two points for bend height axis*. Specify two points for the bend axis. The two points represent the point at which the bend will start.
5. The Message Line now reads, *Bend Solid: Enter two points for alignment axis*. Choose the points on the solid to aligned with the curve.



Geometric Characteristics

A bend is created by choosing a solid and either two points to bend around or a path to bend along. For the Bend One End and Bend Center it is made up of these characteristics according to Edit Objects: Radius and Angle. For the Bend Along a Path there are no geometric characteristics.

Twist Solid Tool



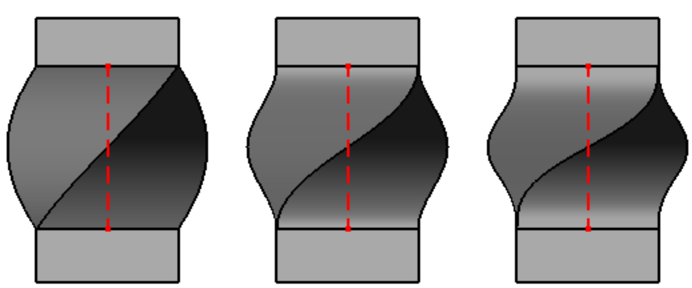
This tool adds angular twisting to a solid, surface

or curved object.

The Message Line contains a pull-down menu to specify a type of geometric continuity between twisted area of the object and non-twisted area. There are three options: **Match G0 Slope, Match G1 Slope and Match G2 Slope.**

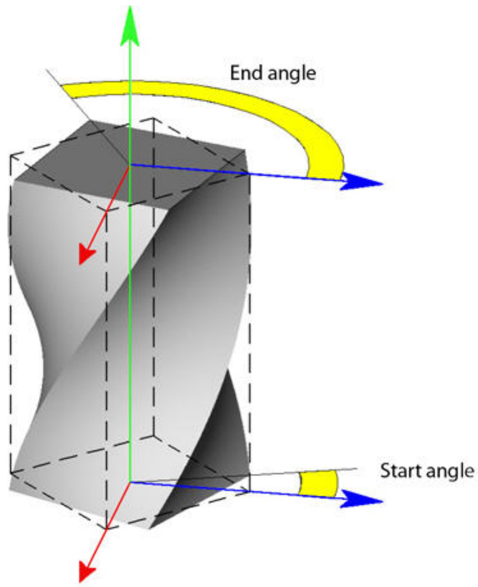
The **Match G0 Slope**, **Match G1 Slope**, and **Match G2 Slope** specify the orientations of the non-twisted portions of the body. If **Match G0 Slope** or **Match G1 Slope** is specified, the entire body is rotated about the twist axis by the given amount before the twist is performed. **Match G2 Slope** indicates that the portion of the body below the twist's starting region retains its orientation with respect to the current coordinate system.

The following images demonstrate the difference between G0, G1 and G2 modes.



G 0 **G 1** **G 2**

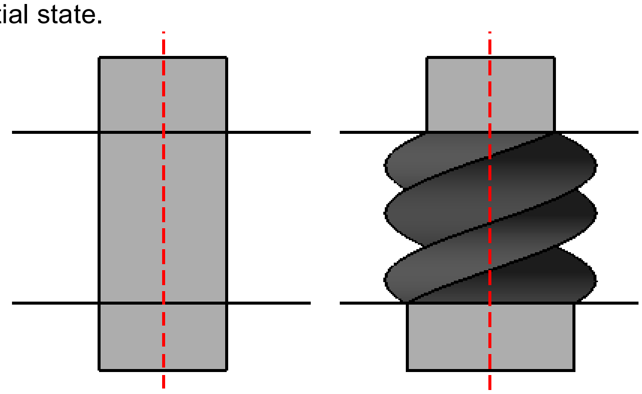
The difference between Start Angle and End Angle represents the amount, in degrees, that the twist region is warped about the axis. Both Start Angle and End Angle can be greater than or multiples of 360 degrees in order to obtain multiple turns within the twisting region.



Using the Twist Solid Tool

1. Select the **Twist** tool from **Solid Utility** tool palette. The Message Line reads: *Twist Solid: Pick solid to twist.*
2. The Status Line contains the Start Angle and End Angle fields. Enter the desired values for the twist. Tab between data fields.
3. Pick the solid object to twist. The Message Line reads: *Twist Solid: Enter two points for twist axis.*
4. Specify the start point for the region to twist.
5. Specify the end point for the region to twist.

Two planes set perpendicular to the twisting axis through the specified points define the twist region. The Start Angle and End Angle values define the angle of rotation of relative to the initial state.



Geometric Characteristics

A twist is created by picking the solid to twist and specifying the two points for the twist axis. It is made up of these characteristics according to Edit Objects: Start Angle, End Angle, Length, Width and Height. To modify the twist characteristics, select the twist using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the twist name. See Design Explorer for more information.

Solid Utility Tools

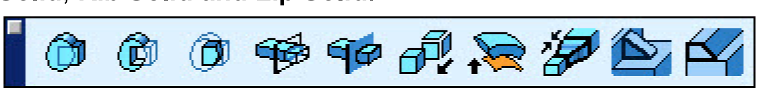
After creating a solid model it's often necessary to modify it. Cobalt, Xenon and Argon provide tools to accomplish this. Use Boolean routines to add solids to each other, subtract them or create a new solid from their intersections. Use other tools to split a solid in two, stitch several surfaces together 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.

Values for each tool can be entered in the Status Line to perform the editing operation either before or after the operation is completed. When entering the values after selecting the tool but before performing the operation, the first click in the drawing area automatically registers all Status Line values. When entering 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.

All editing operations conducted on a solid using Cobalt or Xenon are associative. In these two programs any modifications made to the parent will automatically alter all child objects that reference it.

The topics covered in this chapter include **Solid Editing** tools.

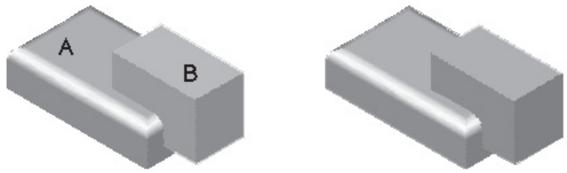
Cobalt, Xenon and Argon provide utilities tools located in the subpalette of the **Solids** tool palette. The utilities include the Boolean tools: **Union Solid**, **Subtract Solid** and **Intersect Solid**, plus **Trim Solid**, **Split Solid**, **Stitched Solid**, **Thicken Solid, Lofted Solid**, **Rib Solid and Lip Solid**.



Tech Note: *The add, subtract and intersect tools replace the two original solids with a new composite solid. The only way to recover the original solids after a boolean union or intersection is to undo the command or in Cobalt or Xenon suppress or remove the feature from the Feature Tree. Another technique for retaining the originals is to copy or instance them and Boolean the copies or instances.*

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.



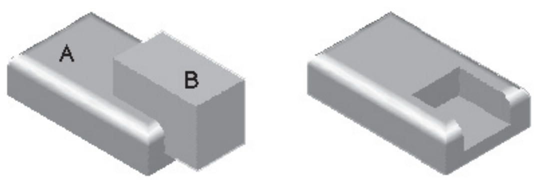
Using the Union Solid Tool

1. Select the **Union Solid** tool. The Message Line reads: *Union Solid: Pick solid to add to*.
2. Select the initial solid.
3. Select the solid(s) to add to the first solid. Hold down the SHIFT key before selecting 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.



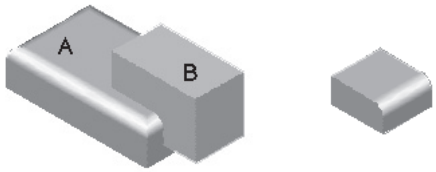
Using the Subtract Solid Tool

1. Select the **Subtract Solid** tool. The Message Line reads: *Subtract Solid: Pick solid to subtract from*.
2. Select the initial solid.
3. Select the solid(s) from which to subtract the first solid. Hold down the SHIFT key before selecting the first solid to select multiple solids Hold the CTRL key (Windows) or OPTION key (Mac) to retain the items subtracted.

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 **Intersect Solid** tool. The Message Line reads: *Intersect Solid: Pick two solids to intersect*.
2. Select the two solids to form the intersect solid.

There are no Status Line entries.

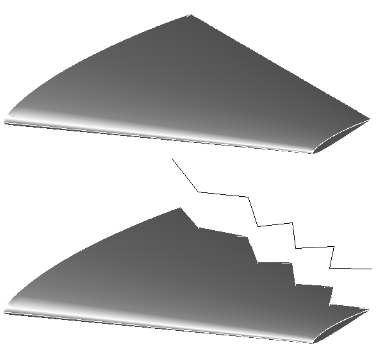
Geometric Characteristics

There are no characteristics listed in Edit Objects specific to a boolean solid.

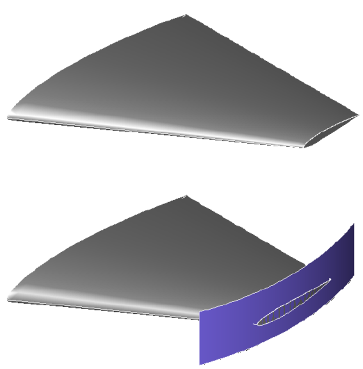
Trim Solid Tool

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 trim 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 the preference.

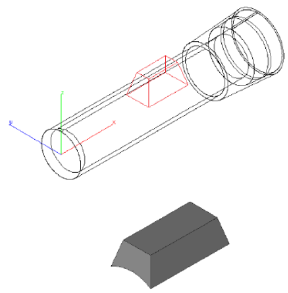
In the case of trimming a solid to a surface or other solid, the pull down option has no bearing on the results.



Trim Solid to Curve.



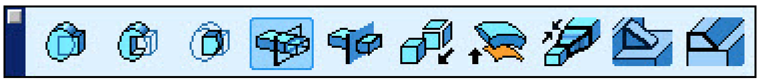
Trim Solid to Surface.



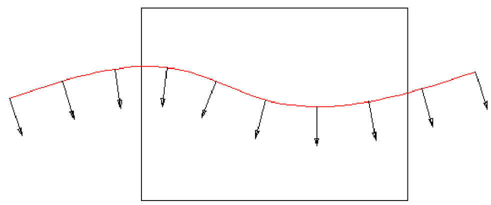
Trim Solid to Solid.

Using the Trim Solid Tool

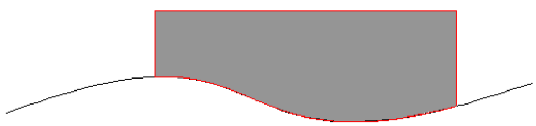
1. Click the **Trim Solid** tool. The Message Line reads: *Trim Solid: Pick curve or surface to trim solid*.



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 to change the material that is kept.

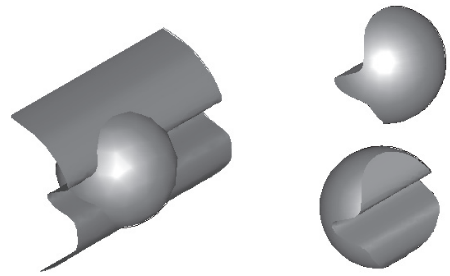
Note: Press the Control or Option key to change the direction of the material to be removed **after** the trim is completed. Do not hold the Control or Option keys during the trimming process.

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.

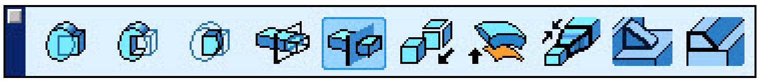
Split Solid Tool

The **Split Solid** tool divides a solid into two parts. Cobalt, Xenon and Argon use an infinite plane, surface, or solid as the splitting entity, so construct the splitting object before making the split.



Using the Split Solid Tool

1. Select the **Split Solid** tool. The Message Line reads: *Split Solid: Pick solid to split*.



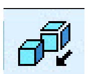
2. Select the solid. The Message Line reads: *Split Solid: Pick surface for splitting solid*.
3. Select the infinite plane or surface to split the solid.

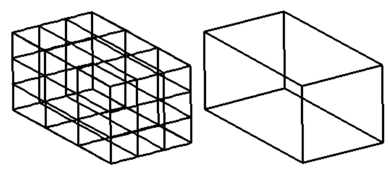
There are no Status Line entries.

Geometric Characteristics

There are no characteristics listed in Edit Objects specific to a split solid.

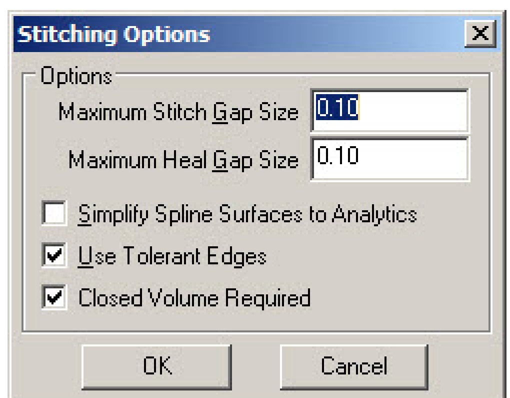
Stitched Solid Tool





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 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 — Sets the maximum size that the program will attempt to stitch to objects. Enter a value between 0 and 1.

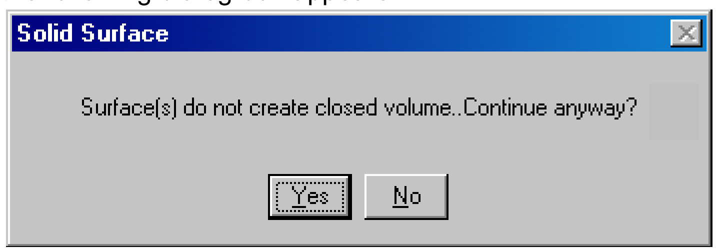
Example: If the gap size is .1 but the object has a gap of .2, the surfaces won't stitch.

Simplify Spline Surfaces to Analytics — 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, the program attempts to simplify the spline surfaces. If the 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 the file. By default, this option is not selected.

Use Tolerant Edges 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 the 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.

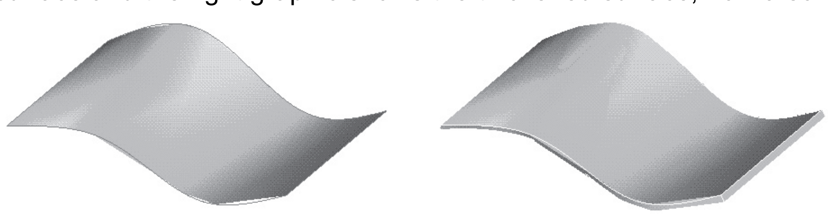
There are no Status Line entries.

Geometric Characteristics

There are no characteristics listed in Edit Objects specific to a stitched solid.

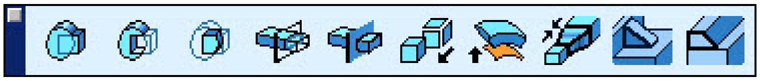
Thicken Solid Tool

The **Thicken Solid** tool thickens surfaces and solids. 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. The program thickens the selected object.

By pressing the CONTROL or OPTION key it is possible to toggle which side of the surface is thickened.

While the object is still selected, 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

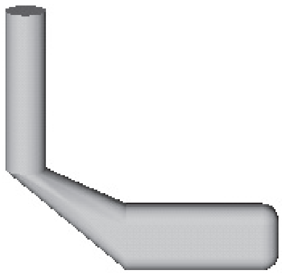
A thickened solid object is created by specifying a thickness and selecting the object. A thickened object is made up of the Thickness characteristic according to the Edit Objects dialog box. To modify the characteristics of the thickened object select the object using the Design Explorer (**Cobalt and Xenon only**) and choose ***Window>Edit Objects*** or double-click on the face name.

Display contains settings for Iso Lines and Silhouette. 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. There are three Silhouette settings, Smart, Off and On.

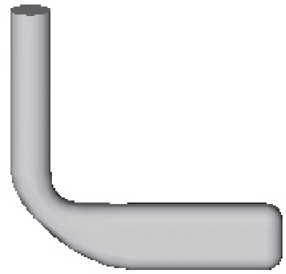
See Design Explorer for more information.

Lofted Solid Tool

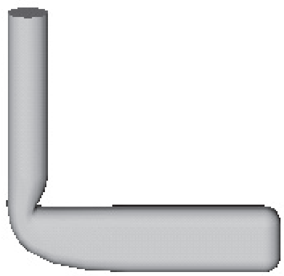
This tool creates a tangent solid between two neighboring solids using a specified bulge factor. The higher the bugle factor the more influential the slope of the tangent for the selected solid. A loft solid with a bulge factor of zero (0) creates a discontinuous loft. The graphics below illustrate the effects of the bulge factor.



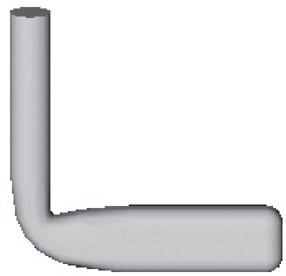
Bulge Factor = 0



Bulge Factor = 5



Bulge Factor 1 = 5; Bulge Factor 2 = 15 (Set in Edit Objects)

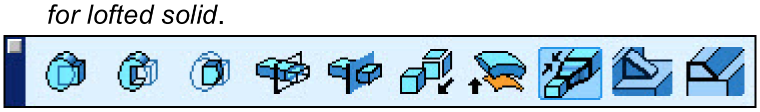


Bulge Factor 1 = 8; Bulge Factor 2 = 10 (Set in Edit Objects)

This tool does not support lofting if one of the solids is a sphere. If a sphere is selected no loft is created.

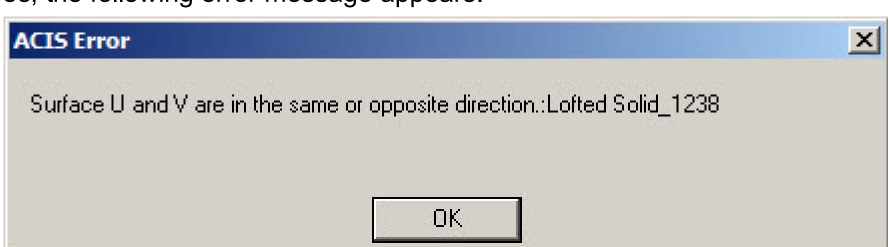
Using the Lofted Solid Tool

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, the following error message appears.



   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.

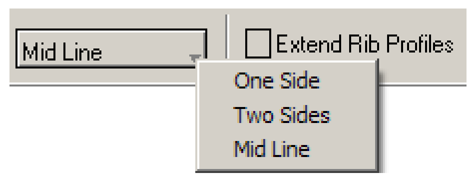
Geometric Characteristics

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

Cobalt's **Rib Solid** 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.

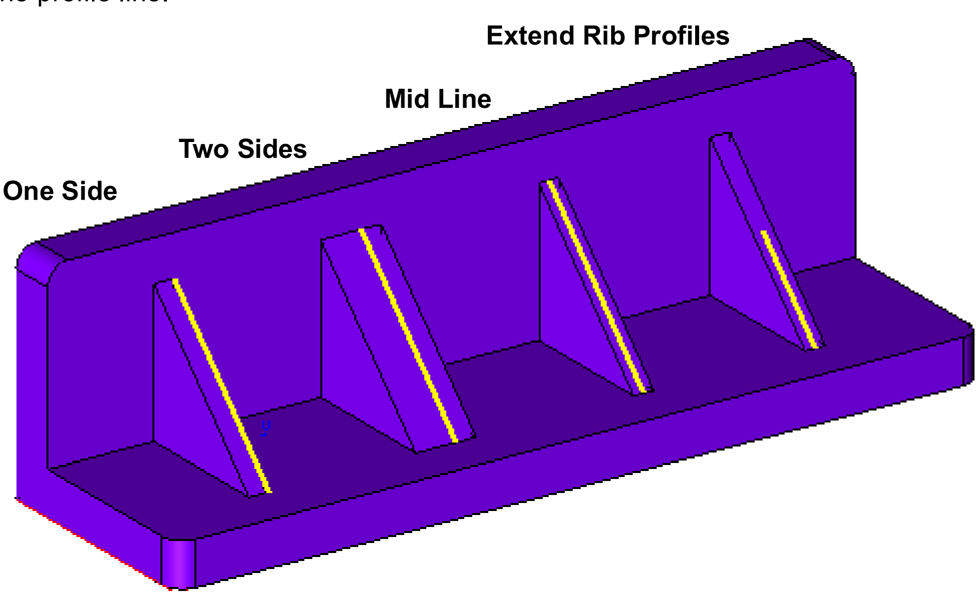
The Rib Solid Tool has three options that appear in the Message Line when the tool is selected:



Cobalt Only

One Side	Extrudes from one side or the other of the profile line using a negative or positive number to control both the thickness and the side.
Two Sides	Extrudes the distances specified for both directions from the profile line.
Mid Line	Extrudes an equal distance in both directions from the profile line.

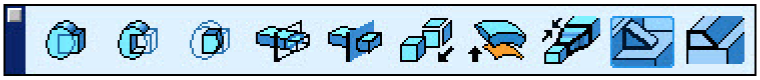
Check the **Extend Rib Profiles** box to automatically extend the rib to meet the corresponding faces of the related solid. If left unchecked the rib will only be as long as the profile line.



Using the Rib Solid Tool

Cobalt Only

1. Select the Rib Solid tool.
2. Specify the necessary option in the Message Line.
3. The Message Line reads: *Rib Feature: Pick solid for rib.*



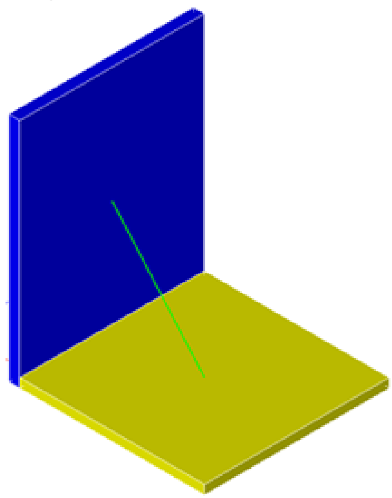
4. Select the solid to which to add the rib. The Message Line reads: *Rib Feature: Pick rib profile. [Shift = Extend]*

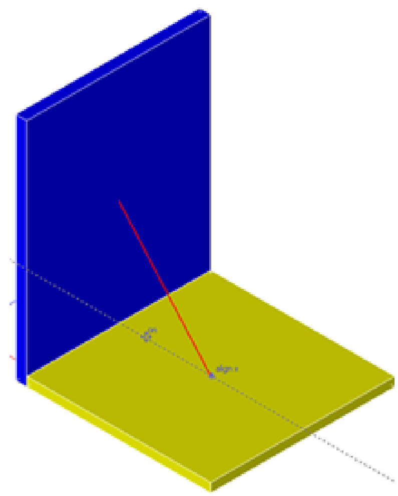
   After selecting the rib profile, specify the thickness of the rib. This is done in the Status Line entry box pictured here.

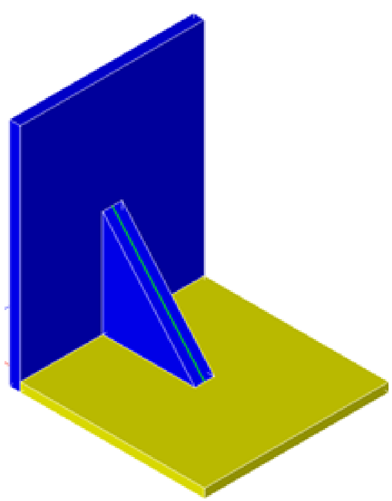
5. Once the rib profile is selected, the Message Line reads: *Rib Feature: Specify 2 points for rib direction.*

   For the tool to function correctly, the direction should run into the solid to which the rib feature is added.

To edit the thickness of the rib after placing it, go through the Design Explore and access the Edit Objects box associated with the rib. The examples below show the tool use progression.







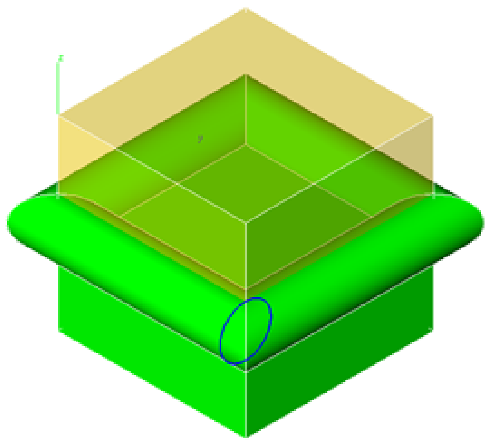
Geometric Characteristics

A rib solid is created by selecting a solid for rib, rib profile and two points for rib direction. It is made up of the following characteristics according to the Edit Objects dialog box: thickness, draft angle, rib type, direction. To display the Edit Objects dialog box for the rib solid only, double-click the ribbed solid item in the Design Explorer.

Lip Solid Tool

Cobalt Only

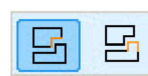
Cobalt's **Lip Solid** tool sweeps a profile about a collection of edges and either adds or subtracts the resultant sweep to or from the base solid. The given profile is automatically aligned perpendicular to the selected edge. In addition, a reference point is specified such that the profile is translated 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.



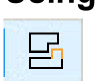
Lip Solid Add Feature Tool example.

Note: While picking 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.

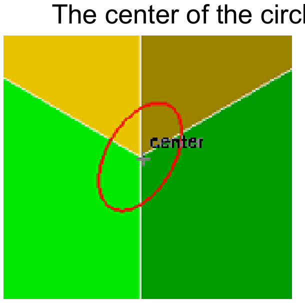


Using the Lip Solid Add Feature Tool



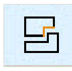
1. Select the Lip Solid tool. The Message Line reads: *Lip Feature Add: Pick curves for lip profile. [Shift = Extend]*
2. Select the curves for the lip profile. The Message Line reads: *Lip Feature: Specify profile origin*.

   The center of the circle is selected to be the origin in this example.



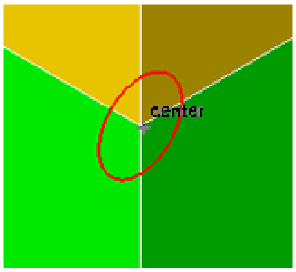
3. The Message Line reads: *Lip Feature Add: Pick edges for profile to sweep. [Shift = Extend]*
4. Pick the edges of the solid to which to add the lip feature. In the example above, the edges of the green block were chosen.

Using the Lip Solid Remove Feature Tool

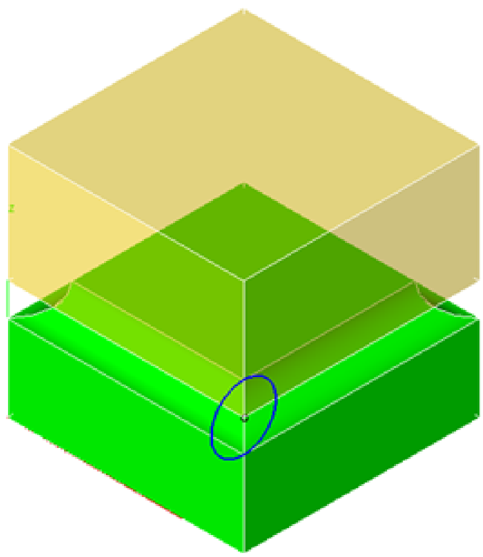


1. Select the Lip Solid tool. The Message Line reads: *Lip Feature Remove: Pick curves for lip profile. [Shift = Extend]*
2. Select the curves for the lip profile. The Message Line reads: *Lip Feature: Specify profile origin*.

   The center of the circle is selected to be the origin in this example.

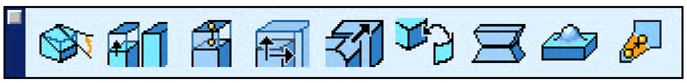


3. The Message Line reads: *Lip Feature Remove: Pick edges for profile to sweep. [Shift = Extend]*
4. Pick the edges of the solid from which the lip feature profile will be removed. In the example below, the edges of the green block were chosen.



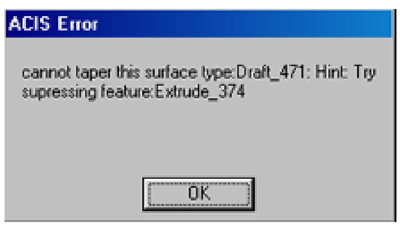
Note: The remove feature on both block edges cannot be used at the same time.

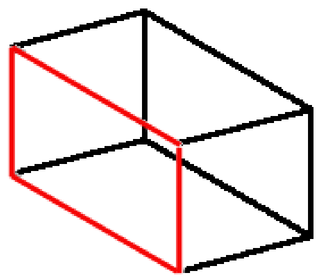
Solid Face Modification Tools



The **Solid Face Modeling** tools perform operations on faces independently of how the part was created. This means that parts from other 3D systems can be imported and edits to the geometry performed.

These tools also include an error detection routine. If the 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.





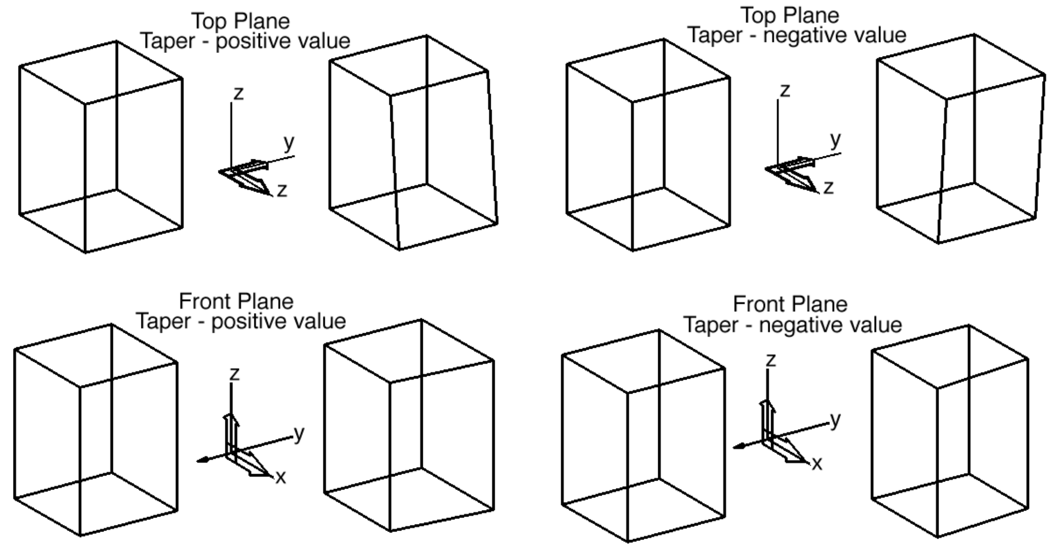
Draft Face Tool

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

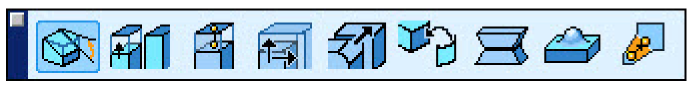
- The work plane cannot be parallel to the face or faces that are tapered. The work plane defines the pull-direction or direction of draw, the direction the part would eject from a mold.
- The neutral point is the position on the face or edge that remains fixed while the face tapers.

- 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

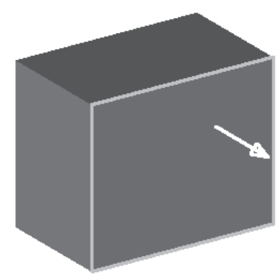
1. Select the **Draft Solid** tool. The Message Line reads: *Draft Solid: Pick reference face or edge for draft (click nothing for work plane]*.



2. Select the reference face, edge or work plane. This tells the program the pull direction of the normal for the taper.

   When selecting 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]*



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*.

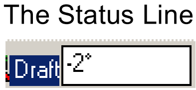
> ***Tech Note:*** *Make sure the chosen face is not normal to the z-axis. If it is, nothing will happen. Change the work plane and try again.*

4. Click a point on the screen to serve as the neutral position. The neutral position will remain constant.

   Cobalt, Xenon and Argon apply the taper or draft relative to a pull direction defined by the normal.

While the solid is selected, the Draft angle can be changed in the Status Line. Press ENTER (Windows) or RETURN (Macintosh) and the taper updates.

The Status Line contains the Draft angle.



Modifying Drafted Solids

Modify the draft angle of the solid by choosing the draft in the Design Explorer for the solid and displaying the Edit Objects dialog box. If there is more than one face drafted, specify a different draft for each one. Click Apply to accept the changes.

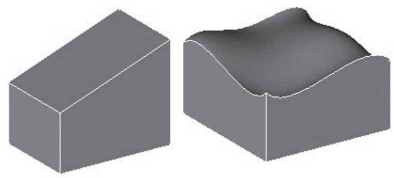
Geometric Characteristics

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 the operation is selected in the Design Explorer first and then the Edit Object dialog box is displayed. See “Edit Objects” on page 5-25 for more information on the Design Explorer.

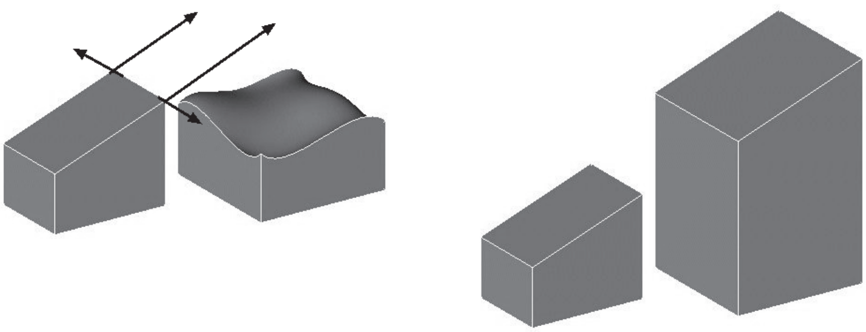
Match Face Tool

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.

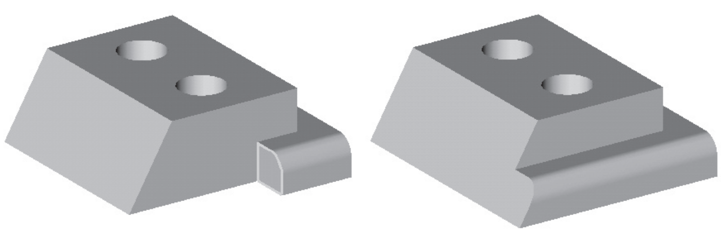
Planar Example



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.

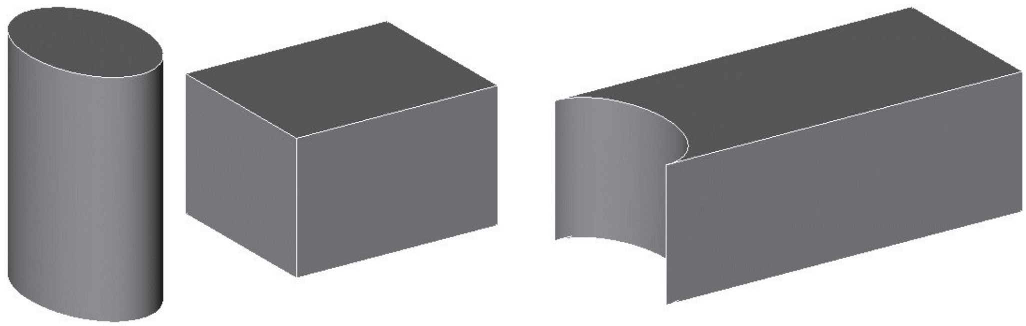


Here’s another example of the match face option. The left graphic shows a select face on a solid object. The right graphic shows the face matched to the sloped face.



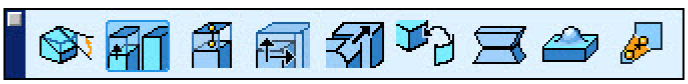
Elliptical Example

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 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 the selected face will 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.

The 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. There are three Silhouette settings: Smart, Off and On.

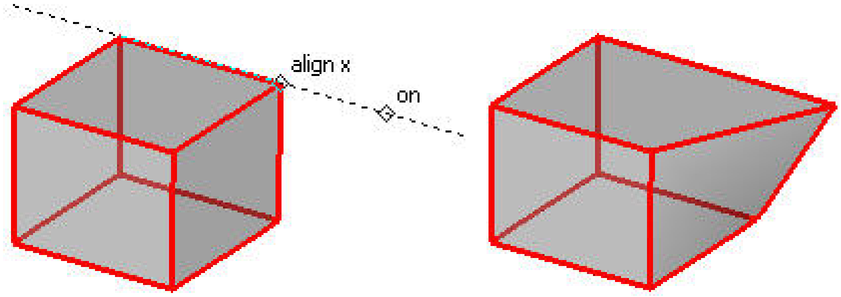
See Edit Objects for more information.

Move Solid Element Tool

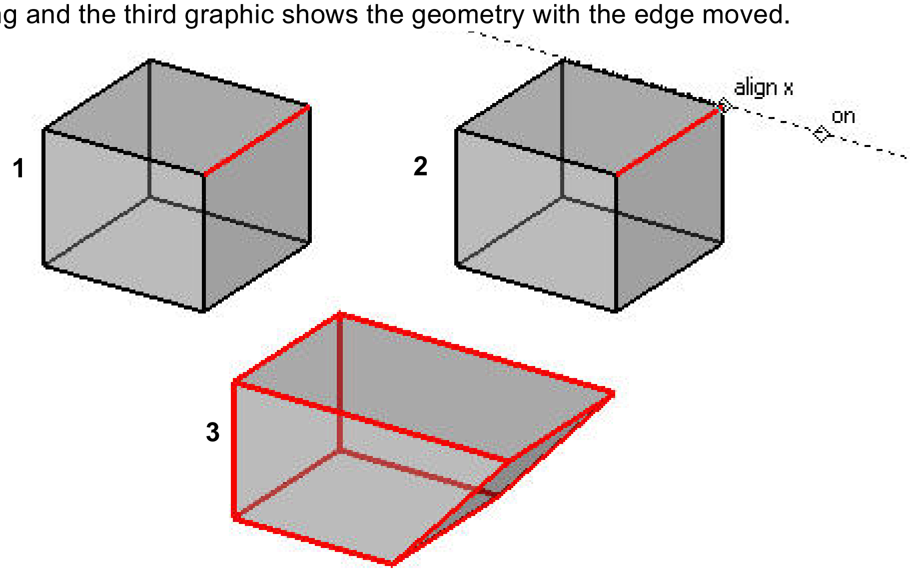
The **Move Solid Element** tool moves edge, vertex or the selected group of faces by a vector defined with the Drafting Assistant.

The Move Face tool has three options to specify moving a vertex, edge or face.

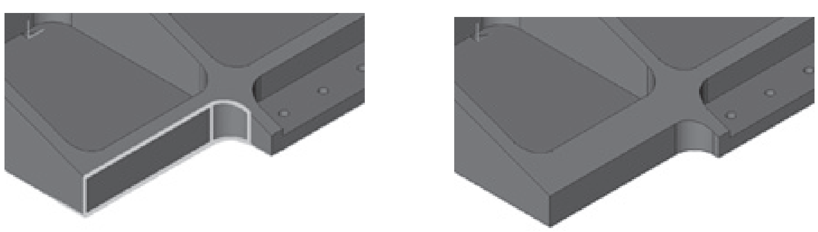
The graphics below show the operation of moving the vertex. The left graphic shows the original geometry with the selected geometry and the right graphic shows the geometry with the vertex moved.



The graphics below show the operation of moving the edge. The first graphic shows the original geometry with the selected edge, the second graphic shows the vector for moving and the third graphic shows the geometry with the edge moved.



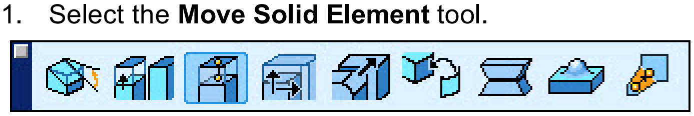
When moving more than one face, the selected faces cannot be perpendicular to the others. This tool is available for moving a flat face and fillet, or a face and a hole. The graphics here show just such an operation. The left graphic shows the original geometry with the selected faces and the right graphic shows the geometry with two faces moved.



Tech Note: *When using the Move Face option, do not move the whole face off a solid or an error message will occur.*

Using the Move Solid Element Tool

1. Select the **Move Solid Element** tool.



2. Select the desired option to move a vertex, edge or face. The Message Line reflects the option chosen. For Move Face it reads: *Move Face: Pick faces of a solid to move [Shift=Extend]*.
3. Select the face, vertex or edge of the solid to move. To select more than one, hold down the SHIFT key before selecting the first one.

   The Message Line now reads: *Move Face: Pick two points for move*. The points define a vector with DX, DY, DZ values. Either click two points or enter the desired values in the Status Line.
4. Select the two points to specify dX, dY, dZ values for the move. The order the points are selected determines whether the value is positive or negative.

   The selected face, edge or vector moves to the new location.

While the element is still selected, 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.

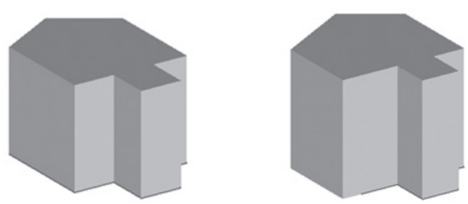
Geometric Characteristics

A move vertex, edge, face has distance and direction characteristics according to the Edit Objects dialog box. To modify the Geometry, Display and Attribute characteristics of the moved solid element, select the element using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the feature name.

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. There are three Silhouette settings, Smart, Off and On. See Edit Objects for more information.

Offset Face Tool

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.



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 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, new values can be entered in the data field in the Status Line. Press ENTER (Windows) or RETURN (Macintosh) to update the offset.

The Status Line contains the Offset value.

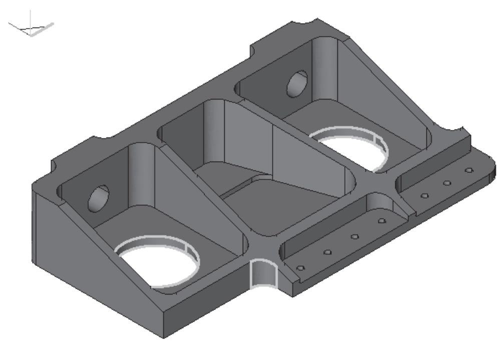
Geometric Characteristics

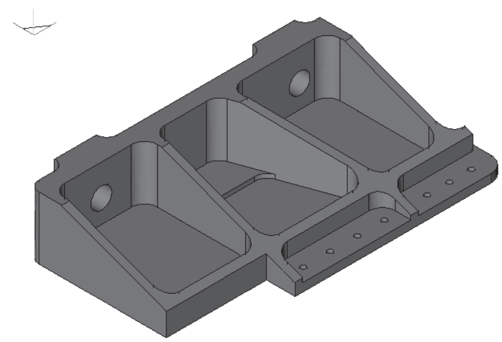
An offset face is defined by its offset value according to the Edit Objects dialog box. To modify the offset or the Display and Attribute characteristics of the offset face, select the face using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the face name.

Display contains settings for Iso Lines and Silhouette. 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. There are three Silhouette settings, Smart, Off and On. For more information see Edit Objects.

Remove Face Tool

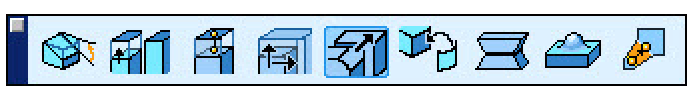
The **Remove Face** tool removes a face from a solid, extends the adjacent faces and relimits the solids as necessary. This is useful for removing holes and fillets from geometry imported without a history tree. The left graphic shows the original object. The right graphic shows the object with the holes and a fillet removed.





Using the Remove Face Tool

1. Select the **Remove Face** tool.

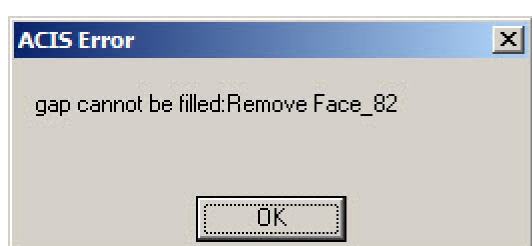


2. The Message Line reads: *Remove Face: Pick faces of a solid to remove. [Shift=Extend]*
3. Select the face of the solid to remove. To select more than one face be sure to hold down the SHIFT key before selecting the first face.

   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. A face of a cube cannot be removed if the cube has a blend; only the blend can be removed. Attempting to remove a face that will challenge the solid's integrity, the following error message appears.

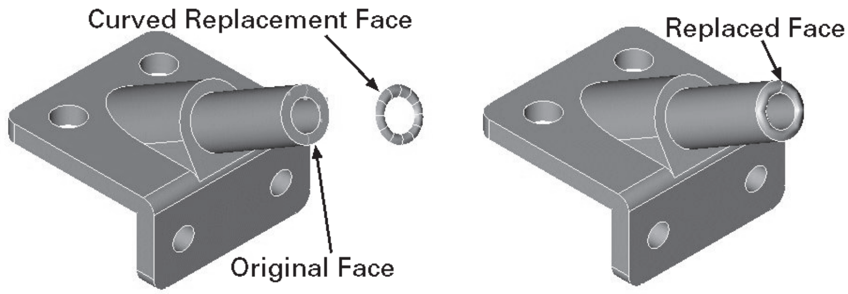


Geometric Characteristics

A removed face has no specific characteristics.

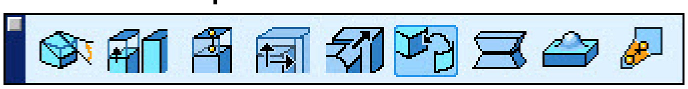
Replace Face Tool

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 to replace. To select more than one face be sure to hold down the SHIFT key before selecting the first face.

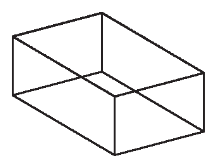
   The Message line now reads: *Replace Face: Pick new surface*.
4. Select the replacement face.

   The original face is replaced.

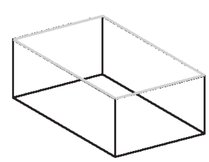
The Status Line contains no entries.

Replace Face Example

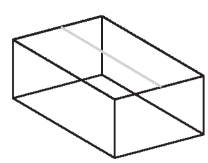
In the block shown here, the top face will be replaced with a warped surface.



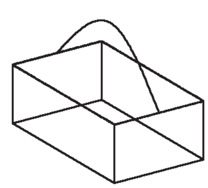
1. Select the **Rectangle** tool and trace over the top face of the block.



2. Select the **Through Point B-Spline** tool and create a three point spline from the midpoint of one edge to the midpoint of the opposite edge.

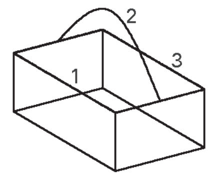


3. Using the **Selection** tool, select the center control point of the spline and drag it up along the z axis some distance.
4. Select the **Skin** tool.

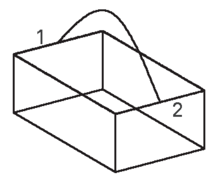


5. Select the **Guide Curves** option from the Message Line.

6. Select the skin curves consisting of the right edge curves, (curves 1, 2 and 3 in the graphic).

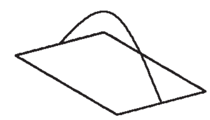


7. Select the guide curves consisting of the top and bottom curves (curves 1 and 2 in the graphic) to complete the skin surface.



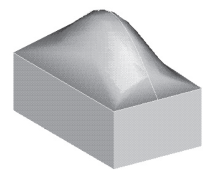
8. Select the *Replace Face* tool.

9. Select the top face of the block to replace.



10. Select the just created net surface to replace the face.

The top face of the solids is replaced.



Geometric Characteristics

A replaced 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.

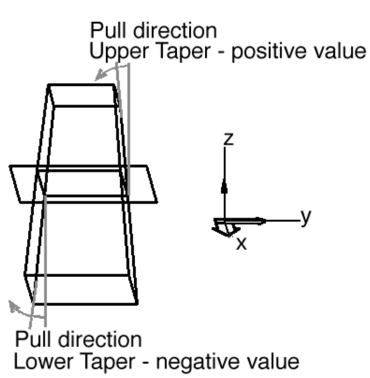
Display contains settings for Iso Lines and Silhouette. Display Iso Lines by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on the object. There are three Silhouette settings, Smart, Off and On. For more information see Edit Objects.

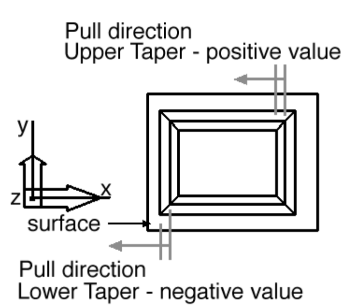
Parting Line Solid Tool

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 the curves must be created before using the tool.

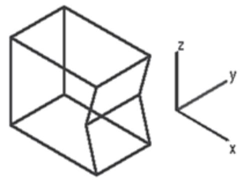
Rules

- The work plane cannot be parallel to the face or faces that are tapered. The work plane defines the pull-direction or direction of draw, the direction the part ejects from a mold.

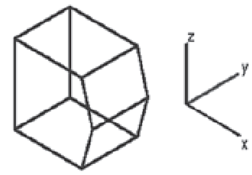




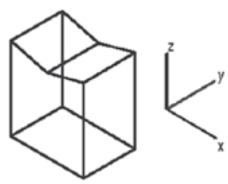
- The taper is an angle that the face rotates relative to the neutral point. The value can be negative or positive and the rotation occurs relative to the work plane. Typically, positive values add material and negative values remove material. The graphics here illustrate the relationship between that work plane and the taper angles.



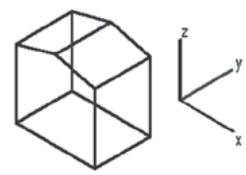
Top Plane
Upper Taper - positive value
Lower Taper - positive value



Top Plane
Upper Taper - negative angle
Lower Taper - negative angle



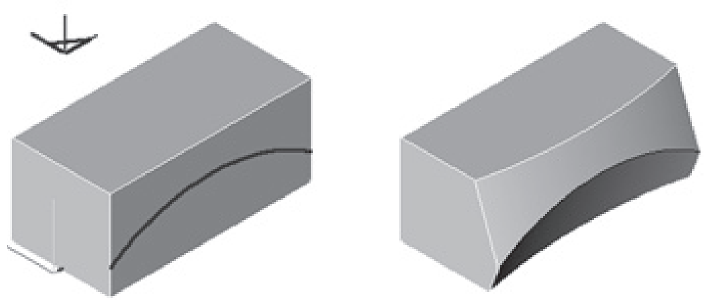
Side Plane
Upper Taper - positive value
Lower Taper - positive value



Side Plane
Upper Taper - negative angle
Lower Taper - negative angle

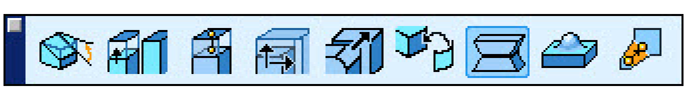
- If having difficulty placing a draft on a solid, try changing the work plane and attempting the draft again.

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

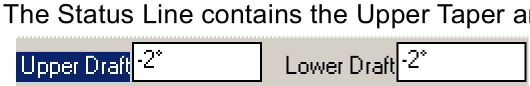
1. Select the **Parting Line** tool. The Message Line reads: *Parting Line Solid: Pick solid for parting line*.



2. Select the solid 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, 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

To modify the taper angles of the solid, choose the taper in the Design Explorer for the solid and display the Edit Objects dialog box. Modify the taper angle for the Upper and Lower Tapers. Click Apply to accept the changes.

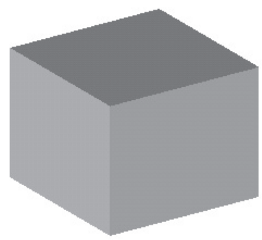
Geometric Characteristics

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 the operation is selected in the Design Explorer first and then the Edit Object dialog box is displayed. See Edit Objects for more information.

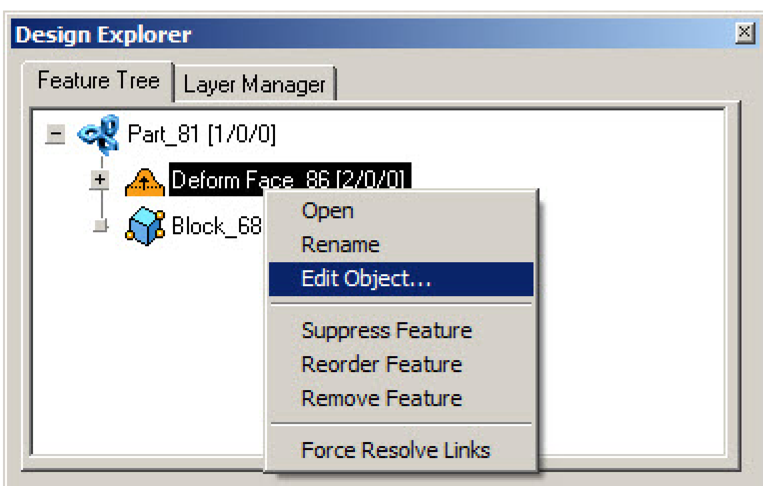
Deform Face Tool

The **Deform Face** tool deforms surfaces and solid faces. When selecting the **Deform Face** tool, a subpalette appears in the Message Line containing three options: **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.



If, after using the Deform Face tool and building the model, it is necessary to edit the deformed face, 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.

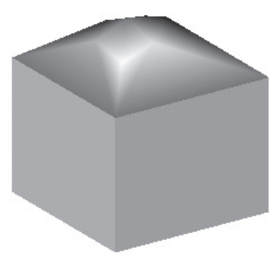
Bend Factor	The bend factor controls the deformable surface's resistance to bending. Deformable models with large bend values act like elastic beams attempting to distribute regions of bending over large areas and typically generate very fair shapes.
Resolution	The resolution slider controls the precision 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 only to pressure deformations, the first option icon. They do not apply to deformations to a point or curve. Be sure to use higher resolutions when using the below options to ensure 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 tangents of the face.
Shared Faces Tangent	Shared Face Tangent modifies all faces that share edges with the face being modified. The modification imposes tangencies at all shared edges.
Shared Faces Curvature	The Shared Face Curvature option modifies all faces that share edges with the face being modified. The modification imposes curvature continuity at all shared edges.
Shared Faces Tangent	The Shared Face Tangent Fixed option will only modifies the selected face to be tangent to all faces that share an edge with the selected face.
Shared Faces Curvature	The Shared Face Curvature Fixed option only modifies the selected face to be curvature continuous to all faces that share an edge with the selected face.

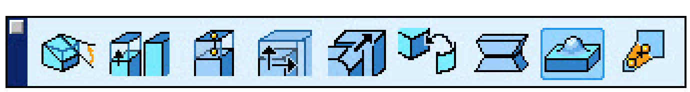
Deform Face with Gain Option

This option 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. A location for the deformation can not be specified. Use the **Deform Face to Point** option for that ability. The graphic shows the top face deformed using this option.



Using the Deform Face with Gain Tool

1. Select the **Deform Face** tool.



2. Select the **Deform Face with Gain** option 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 to deform.

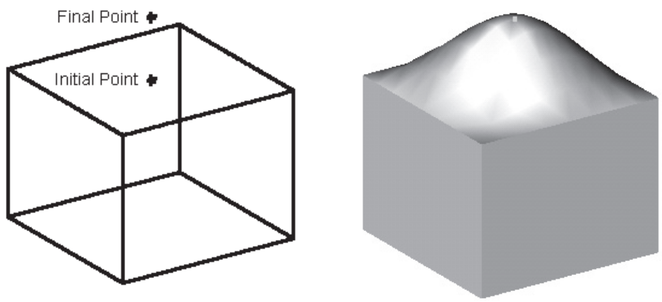
   The object is deformed. Render the object using any render option other than wireframe to view the deformation.

While the object is still selected, 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 Option

This option applies a pressure gain (negative or positive) to the selected surface or face passing through a specific point in space. 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 Option

If the deformation should pass through a point not already in the 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 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*.

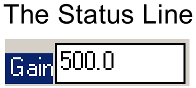
6. Click the final location for the face point where the deformation begins.

   The object deforms. Render the solid using any rendering option other than wireframe to view the deformation.

While the surface or face is still selected, enter a new value in the Gain data field and press ENTER (Windows) or RETURN (Macintosh) to change the gain.

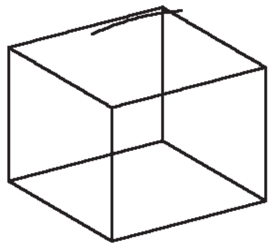
It is possible to move the point later by dragging it to a new location or using the Edit Objects dialog box when the point is selected. The deformed object automatically updates.

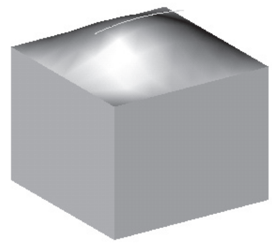
The Status Line contains the Gain value for the deformation.



Deform Face to Curve Option

This option applies a pressure gain (negative or positive) to the selected surface or face attached to a specified curve. Create the curve before using this option. 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 Option

If it is necessary for the deformation to use a curve not already in the drawing, create the curve before using this option.

1. Select the **Deform Face** tool.
2. Select the **Deform Face to Curve** option 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 to deform.

   The Message Line reads: *Deform Face: Pick curve constraint*.
5. Select the curve.

   The object is deformed. Render the object using any render option other than wireframe to view the deformation.

While the object is still selected, 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

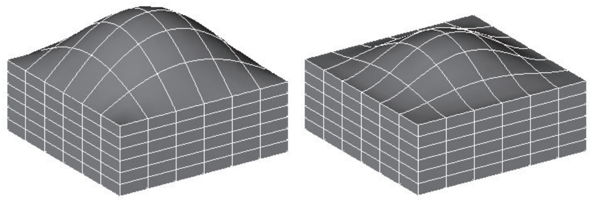
A deformable surface or face is created by selecting object and using pressure, a point or curve. This deformed surface or face is made up of the following characteristics according to the Edit Objects dialog box: Gain, Stretch Factor, Bend Factor and Resolution. The dialog box also contains the Keep Boundary Tangents option.

Gain	Displays the pressure applied to the surface or face. Enter a different gain and click Apply.
Stretch Factor	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 or use the slide to specify a factor.

Bend Factor Sets the surface or face resistance to bending. Higher values prevent sharp radical changes to the surface. Enter a value or use the slide to specify a factor.

Resolution 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 the surface. Enter a value or use the slide to specify a resolution.

Keep Boundary Tangents 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.

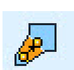


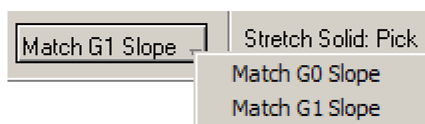
To modify the Display and Attribute characteristics of the deformed surface or face select it using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the item's name.

Display contains settings for Iso Lines and Silhouette. 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. There are three Silhouette settings, Smart, Off and On.

See Edit Objects for more information.

Stretch Solid Tool





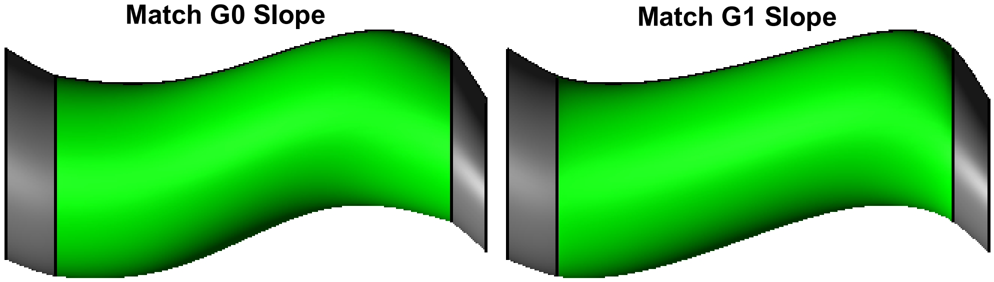
The **Stretch Solid** tool stretches a solid along an axis. When selecting the **Stretch Solid** tool, a drop down menu appears in the Message Line containing two options: **Match G0 Slope** and **Match G1 Slope.** These options specify the translations of the non-stretched portions of the body.

Match G0 Slope Indicates that the portion of the body below the stretch's starting region retains its position.

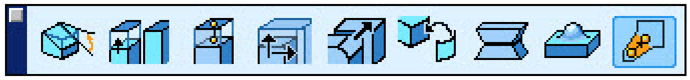
Match G1 Slope Translates the entire body along the stretch axis by the given distance before the stretch is performed.

Following images show the difference between G0 and G1 modes.

Match G0 Slope **Match G1 Slope**

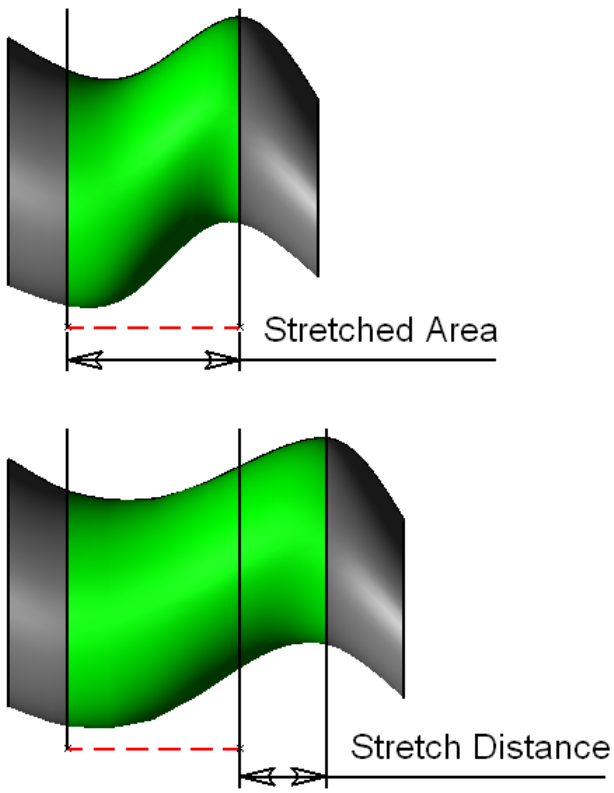


Using the Stretch Tool



1. Select the **Stretch** tool from **Solid Utility** tool palette. The Message Line reads: *Stretch Solid: Pick solid to stretch*.
2. Pick the solid object to stretch. The Message Line reads: *Stretch Solid: Enter two points for stretch region.*
3. The Status Line contains the Distance field. Enter the desired value for the stretch.
4. Specify the starting point for the region to stretch.
5. Specify the end point.

Two planes perpendicular to the stretch axis through the specified points define the region to stretch.

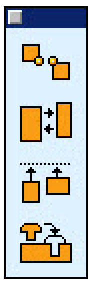


Geometric Characteristics

A stretched distance is created by picking the solid to stretch and entering two points for stretch region. This stretch is made up of the following characteristics according to the Edit Objects dialog box: Distance, Name, Resolution, Color, Control Points, Layer.

See Edit Objects for more information.

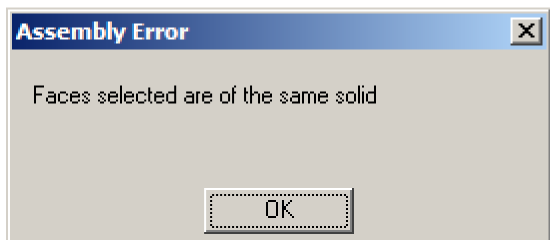
Assembly Modeling Tools



Cobalt and Xenon provide tools for doing assembly modeling. These tools are located in their own tool palette. Choose ***Window>Assembly*** to display the tool palette.

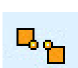
The **Assembly Modeling** tools include: **Assembly Connect**, **Assembly Mate**, **Assembly Align** and **Assembly Insert**.

Each tool performs a particular operation involving two solids. The following error appears if an operation is attempted on a single solid.

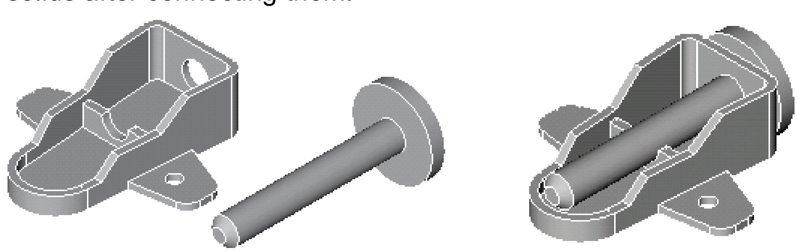


Tech Tip: *Mate or align the faces first, then set the connection points.*

Assembly Connect Tool

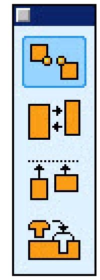


The **Assembly Connect** tool connects one solid to another at a specified connection point. The resulting part is associative. When moving one solid the other moves as well to maintain the connection. The left graphic shows two solid objects. The right graphic shows the solids after connecting them.



This tool supports connection points along object edges and on locations recognized by the Drafting Assistant such as *center*, *midpoint and endpoint*. It is not possible to relate more than one solid to another in a single command.

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 solid to be moved.

   The Message Line reads: *Assembly Connect: Pick solid to move to*.
4. Select a different solid to which the first solid will be moved.

   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 stationary solid where the first solid will be connected.

   The two solids are connected.

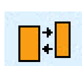
The Status Line contains no entries.

Geometric Characteristics

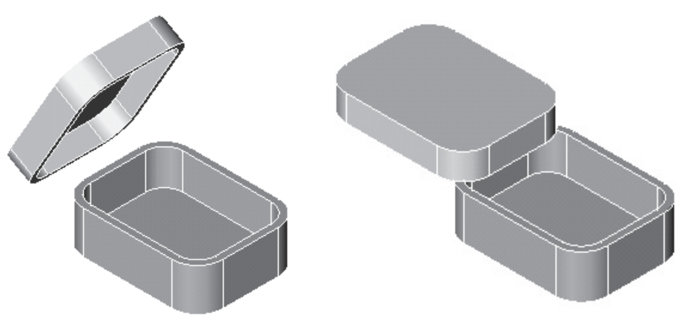
A connected assembly is made up of two solids. It does not include characteristics that are unique. To modify Display and Attribute characteristics of an assembly, select the object using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the face name. See Edit Objects for more information.

The Display tab of the Edit Object window 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. There are three Silhouette settings: *Smart, Off* and *On*.

Assembly Mate Tool

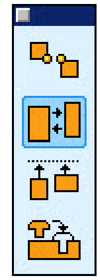


The **Assembly Mate** tool mates planar faces. Specify an offset distance between mated faces. The normals of the mated faces point in opposite directions. This tool only ensures that the selected faces are co-planar. To align the faces, use the **Assembly Align** tool. The left graphic shows the two solids. The right graphic shows the solids after mating them.



Tech Tip: *Don't be alarmed when using the Mate tool if the solid moves a long way from it's original position. The solids will be mated. Use the Connection tool to specify the exact connection and bring them back together.*

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 to mate.

   The Message Line now reads: *Pick solid face to mate with.*
3. Select the face of a different solid with which to mate the first.

   The two solids are now mated.
4. To offset the first solid 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.

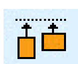
The Status Line contains the Offset data field.

Geometric Characteristics

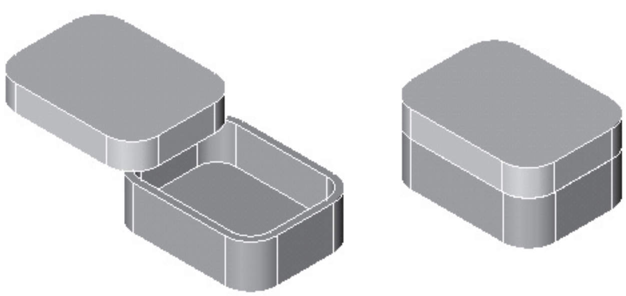
A mate assembly is made up of two mated solid objects and includes an Offset value according to the Edit Object dialog box. To modify the characteristics of a mate assembly, select the assembly using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the face name.

The Display tab of the Edit Object dialog window 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. There are three Silhouette settings: *Smart, Off* and *On*.

Assembly Align Tool

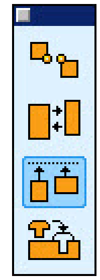


The **Assembly Align** tool aligns a face of two solid objects and supports cylindrical, spherical and planar faces. This tool reorients the parts so that the faces are aligned in the same direction. The left graphic shows the two solid. The right graphic shows the solids after aligning them.



Tech Tip: *Don't be alarmed when using the Mate tool if the solid moves a long way from it's original position. The solids will be mated. Use the Connection tool to specify the exact connection and bring them back together.*

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 to align.

   The Message Line now reads: *Assembly Align: Pick solid face to align with.*
3. Select the face of a different solid with which to align.

   The two solids are now aligned.

The Status Line contains no entries.

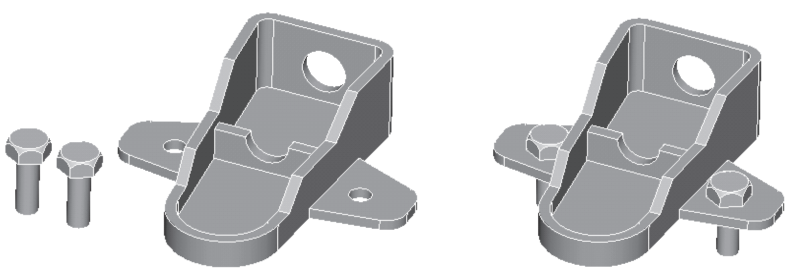
Geometric Characteristics

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 Edit Objects for more information.

The Display tab of the Edit Object window 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. There are three Silhouette settings: *Smart, Off* and *On*.

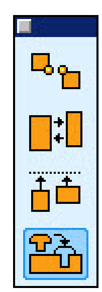
Assembly Insert Tool

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 to align.

   The Message Line now reads: *Assembly Insert: Pick solid face to align with.*
3. Select the face of a different solid with which to align the first.

   The Message Line reads: *Assembly Insert: Pick solid face to mate.*
4. Select the solid face to mate.

   The Message Line now reads, *Assembly Insert: Pick solid face to mate with.*
5. Select the face of a different solid with which to align the first.

   The first solid is now inserted into the second solid.

6. To offset the first solid 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 Edit Objects for more information.

The Display tab of the Edit Object window 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. There are three Silhouette settings: *Smart, Off* and *On*.

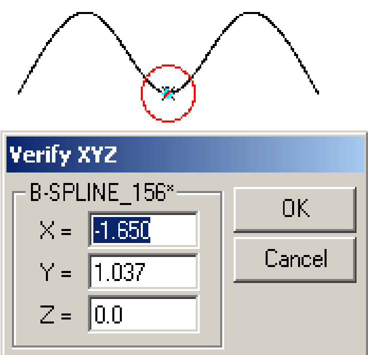
Object Properties & Verification

Geometric Properties

The Verify menu gives access to a variety of commands that provide information about the file or about specific objects.

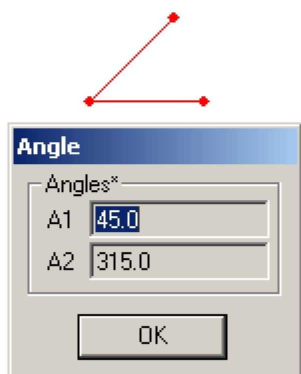
X, Y, Z Command

The *XYZ* command displays the model coordinate values of the selected point. A selected value may be edited and changed.



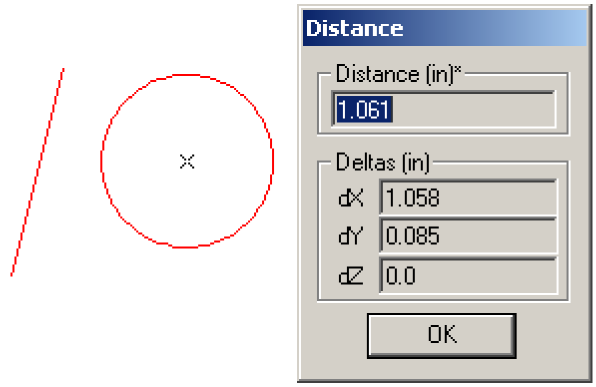
Angle 3 Points Command

The *Angle 3 Points* command calculates the angle formed by three chosen points.



Distance Point-to-Point Command

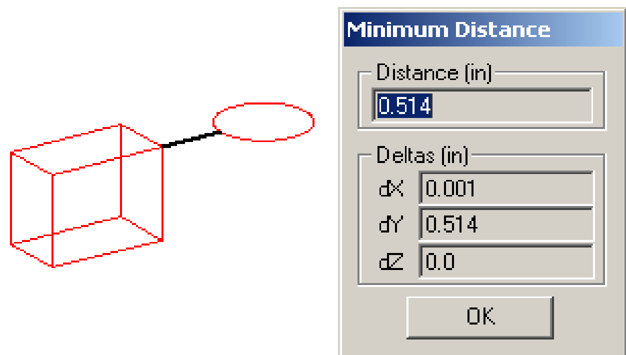
The Distance Point-to-Point command calculates the distance between two points taken from the Drafting Assistant.



Minimum Distance Command

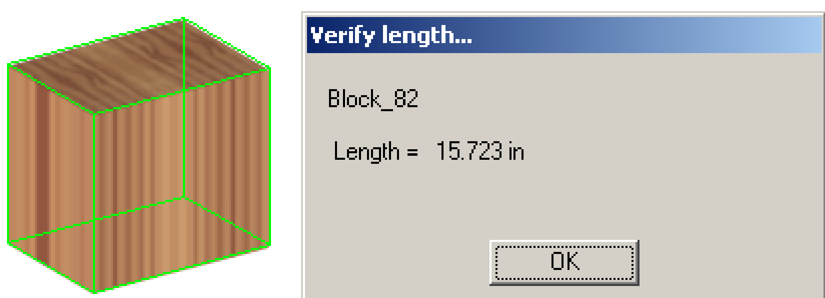
The *Minimum Distance* command calculates the smallest distance between two selected objects.

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.



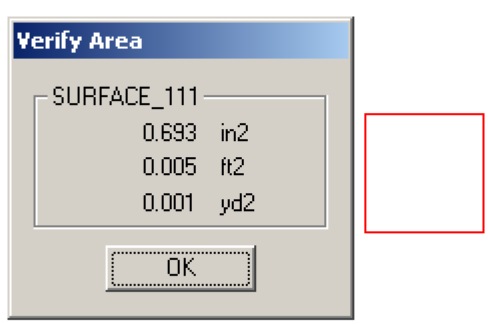
Length Command

This *Length* command calculates the length of a line, spline, arc, circle, conic, surface or solid. In the case of surfaces and solids the command adds all edge lengths, providing a perimeter length value as shown in the graphic here.



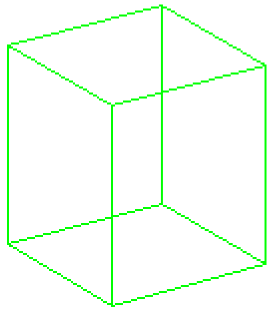
Area Command

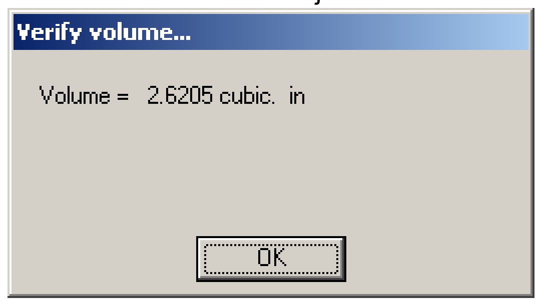
The *Area* command calculates the area of closed curves or a surface. These curves include basic objects such as circles and ellipses but also splines connected to create an enclosed area.



Volume Command

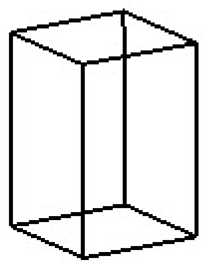
The *Volume* command calculates the volume of a closed object.

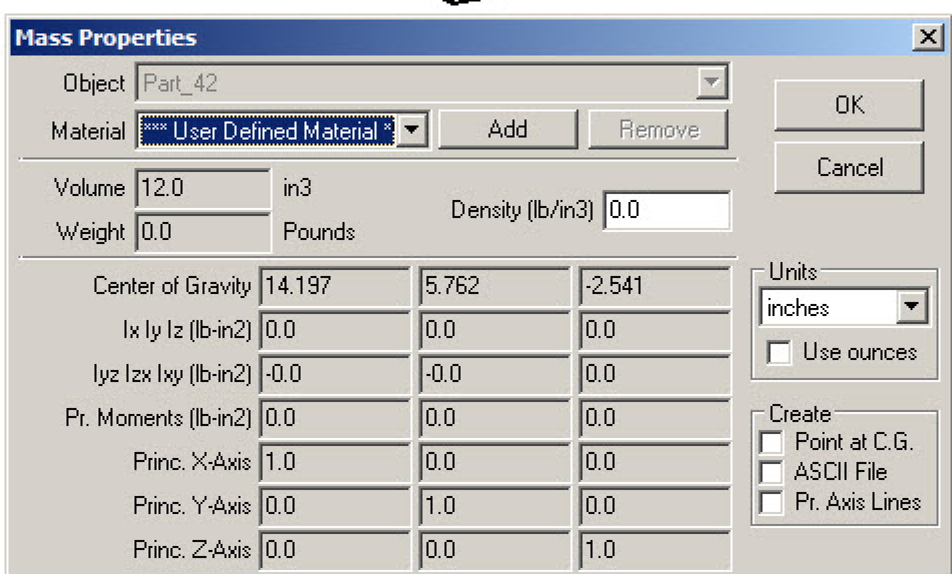




Properties Command

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 specifies a material type.

The menu includes a wide range of materials, many in multiple varieties, including cast iron, lead, steel, aluminum, brass, copper, zinc, nickel, magnesium, titanium, wood, ceramic, aggregate, plastic, composite and liquids.

Custom materials may also be defined by editing the materials.ini file. See the Ashlar-Vellum Support Center Knowledgebase for details. Select one of these materials and the associated density displays in the Density field. Enter your own values for the User-defined material.

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 exports the information contained in this dialog box to an ASCII file. When clicking OK to close the Properties dialog box, the *Save Document As* window appears asking for a name and location for the ASCII file.

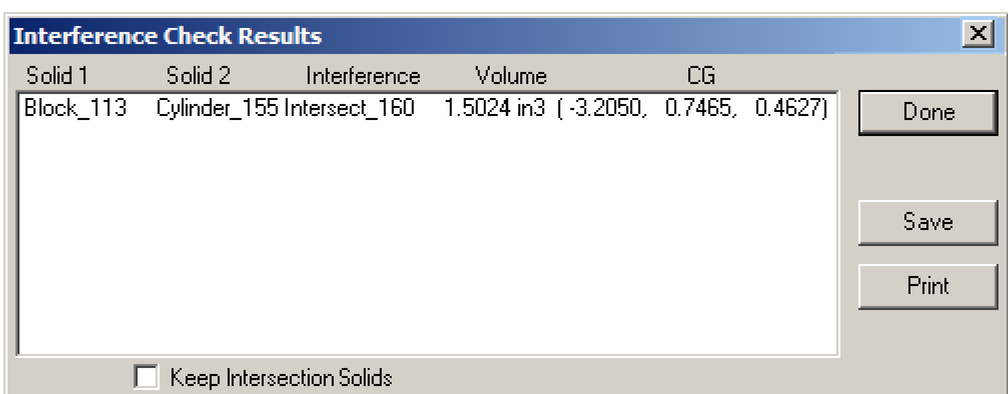
Point at C.G. - Clicking in this check box, places a point at the center of gravity for the selected object. Click OK to close the dialog box and a C. G. point appears. The style of the point is based on the selected style for the **Point** tool. See Point Tool for more information.

Pr. Axis Lines - Clicking in this check box, gives the axis direction vectors for the coordinate system where the products of inertia vanish.

Tech Note: *The material selected here is set separately from material hatching used for geometry in a section view.*
See Crosshatching for more information on crosshatching and Section View Tools for more information on section views.

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, the *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 C.G. (the center of gravity for the interference solid at the x, y and z location).

Keep Intersection Solids When checked, this option retains the intersection solids in the 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

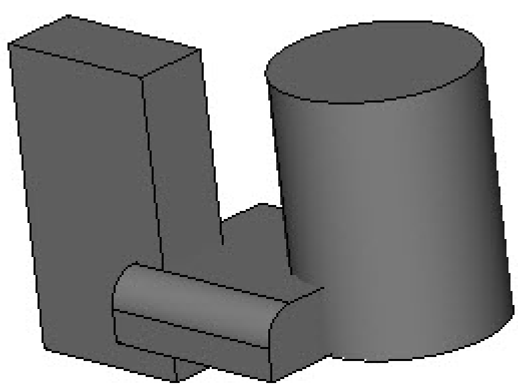
1. Display the model as a wireframe.
2. Select the solids on which to perform the check.
3. Choose ***Verify>Interference***.

   If an interference is encountered, the Interference Check Results dialog box appears listing the interference and displaying the interference solids in black on the model.
4. To retain these interference solids in the model, check the *Keep Intersection Solids* box.
5. Click Save or Print to save or print the log of the interference solids.
6. Click Done to close the dialog box.

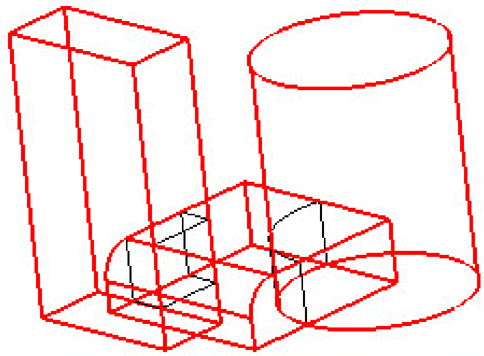
   If the *Keep Intersection Solids* box is checked, the model displays the intersection solids.

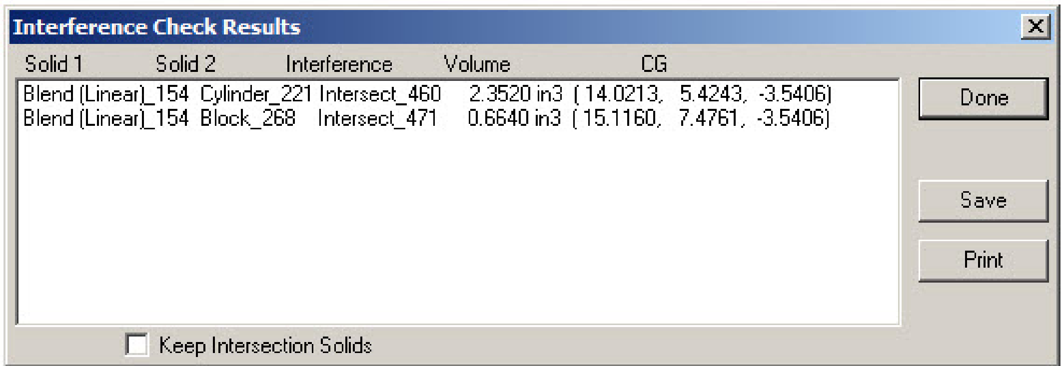
   Example:

   The model here displays three intersecting solids.

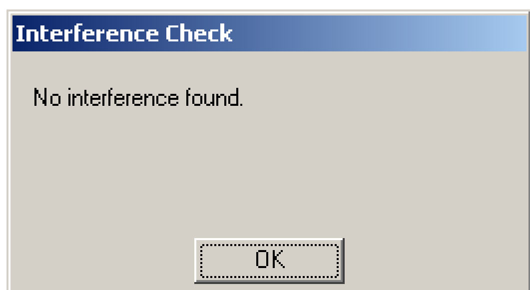


By choosing ***Verify>Interference*** on these selected models (shown below in wireframe), the Interference Check Results dialog box appears and the interference solids are shown in black.





If no interference is found the following dialog box appears.

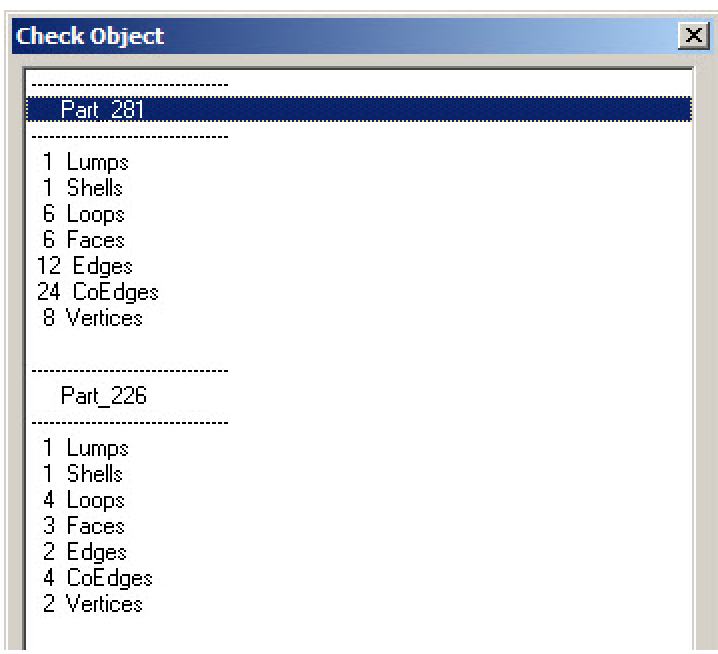


Object Counts Command

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 and 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 appropriate child-level entity, such as body has lump, face has edges, etc.
- Presence (non-NULL) and closure of back pointer from child to parent, for example 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 previous 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 discontinuities.
- Self intersecting surfaces.
- Analytic surfaces have valid definitions.
- Pcurve surface matches face surface (warning only if not equal since surface could be trimmed).
- Pcurve forms agrees with curve form, such as 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, such as have positive dot product. This also tests the following:
- Spline surface form is set correctly, such as surfaces closed in u report this. Checks the underlying 3D B-spline surface at 10 points along seam to verify form.
- 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).

One or more objects may be selected 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.

Surface Evaluation

Cobalt, Xenon and Argon all have methods to evaluate the smoothness of the surfaces. This is useful for identifying surface irregularities. The curvature evaluation is accomplished by placing the surface inside a brightly lit imaginary cylinder with longitudinal stripes. The cylinder stripes are reflected onto the surface to convey the surface smoothness. This is especially important when matching surfaces using the **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.

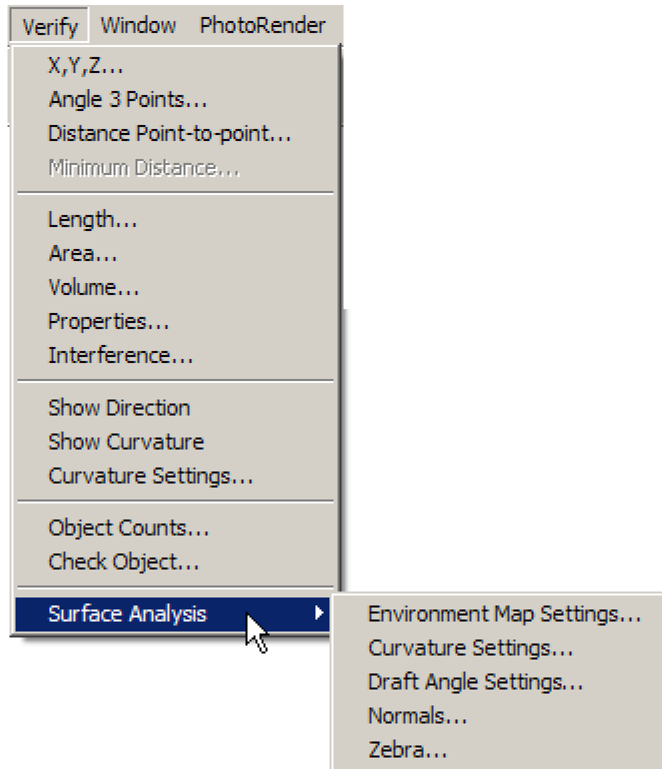
Evaluate the surfaces either through the Verify menu using the *Surface Analysis* commands or via the Display tab of the Edit Object window or through the Advanced Rendering feature.

Evaluation Through the Verify Menu and Edit Object Window

The *Surface Analysis* commands quickly evaluate the smoothness of one or more surfaces. All analysis colors are preset with these commands. To define your own settings, use the surface evaluation shader through the Advanced Rendering feature. See Surface Evaluation further ahead in this chapter.

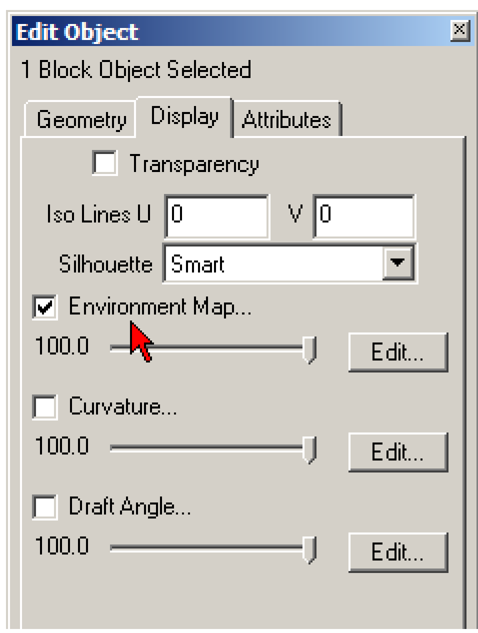
There are five surface analysis commands: *Environment Map*, *Curvature*, *Draft Angle, Normals* and *Zebra*.

The same commands are available on the Display tab of Edit Object Window.



Real-Time Environment Maps

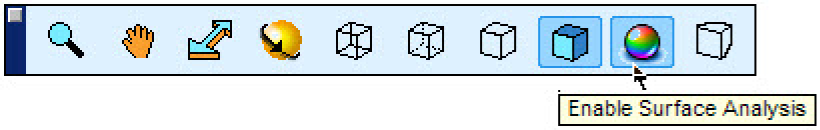
Cobalt, Xenon and Argon feature real-time environment mapping in Verify menu and the Edit Objects window.



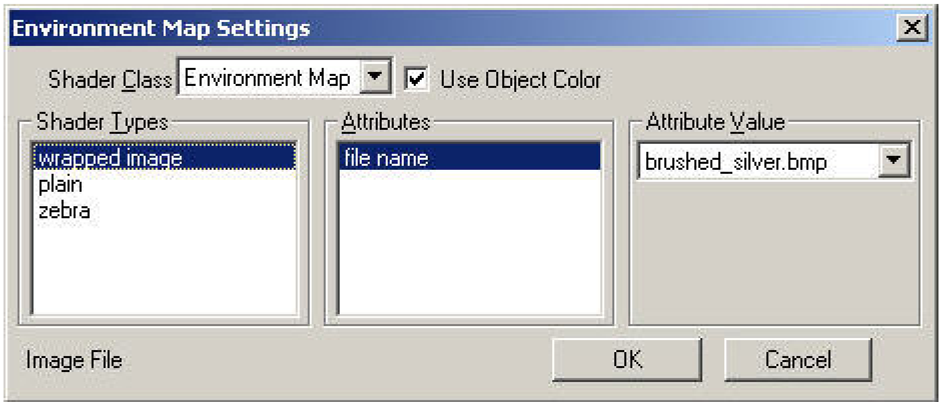
Environment maps provide interactive feedback on surface aesthetics. They are used to interactively evaluate curved surfaces in real-time. A real-time environment map is a way to simulate the reflections seen while examining a reflective model.

To use this feature:

1. Select an object for examination.
2. Activate the **Surface Analysis** icon in the **View** palette.

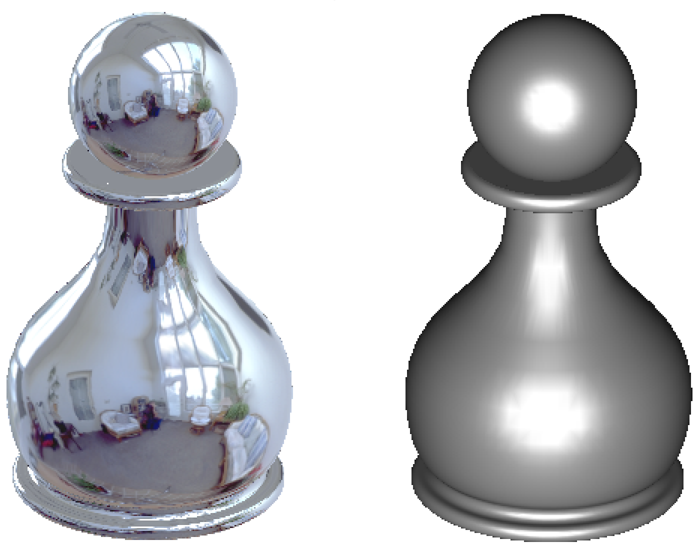


3. Choose ***Verify>Surface Analysis>Environment Map*** or check the Environment Map option in the Edit Objects dialogue box on the Display tab. To edit the environment map, press the Edit button. The Environment Map Settings dialog box opens.



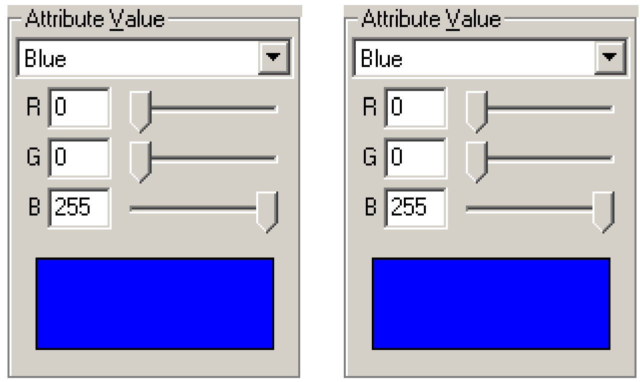
4. Choose any Shader Type and the corresponding attributes and values, then click OK.
5. When an object in the drawing area is selected and the Real-time Environment Map icon is clicked, one of the following Shader Types appear:

Wrapped Image The graphic on the right appears without an environment. The graphic on the left uses the real-time environment map to mirror an applied surrounding environment of a sun room.



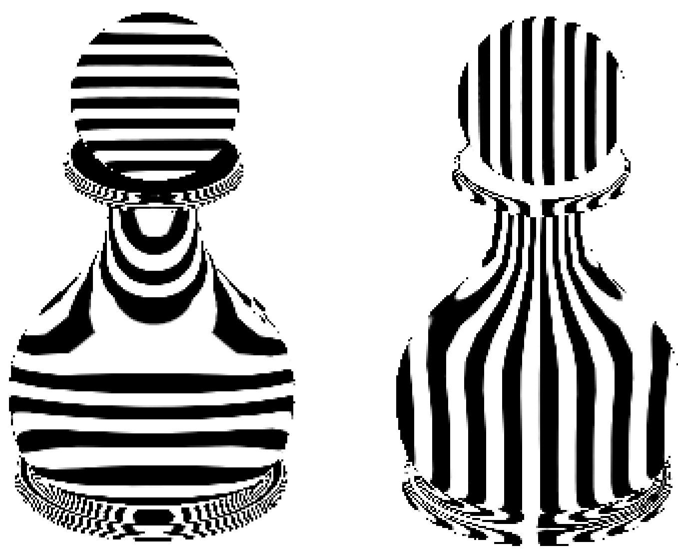
Plain

Applies a plain color to the object. Choose the color from the drop down menu and use the sliders to modify the color.



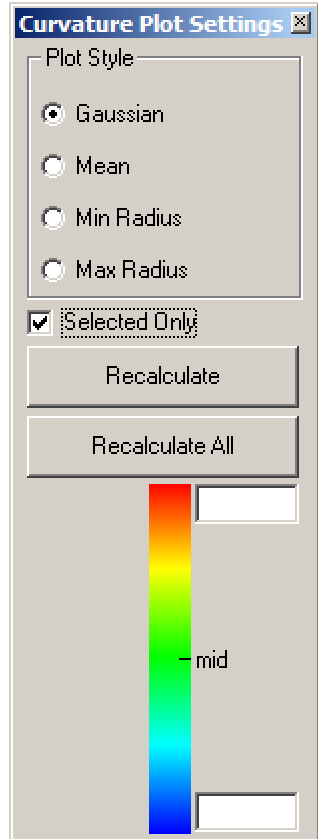
Zebra

Choose the Zebra thickness value and stripe orientation in the Attribute and Attribute Value fields. To make it easier to visualize the surface flow and transition between surfaces of tangent planes, use the Surface Match tool. After matching the planes the surfaces flow smoothly.



Curvature Command

This command creates a curvature plot of the surface. Whether select the command from the ***Verify>Surface Analysis>Curvature Settings,*** or on Display tab of the Edit Object window press the Edit button the Curvature Plot Settings dialog box appears containing: Analysis Styles, Selected Only check box, Recalculate and Recalculate All buttons, Histogram, Color Spectrum and Histogram Data Fields.



The dialog box contains the following elements:

Plot Style 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.

Selected Only Applies the settings only to selected objects.

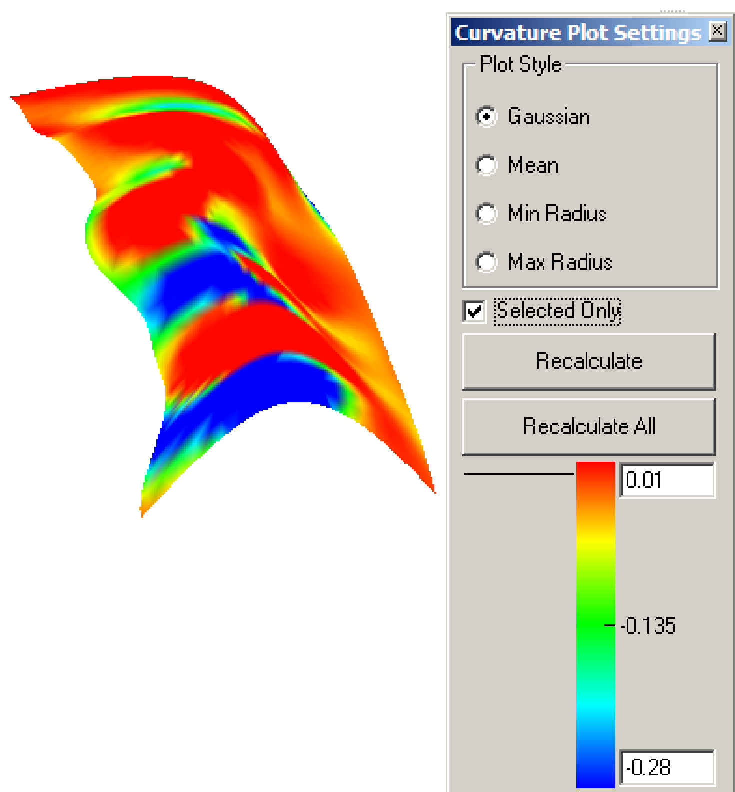
Recalculate Updates the surface curvature analyses for the plot style selected.

Recalculate All Updates the surface curvature analyses for all the plot styles.

Histogram Represents 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. The program calculates this histogram and displays it so that the entire graph fits into the dialog box area.

Color Spectrum Bar Displays the color spectrum used to create the histogram.

Histogram Data Fields Displays the maximum and minimum values used to calculate the histogram. When initially selecting the command, the program scans the surface(s) and sets the minimum and maximum fields for curvature. If different values are entered in the fields so 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.



Making changes to the values, the program waits for two seconds of non-action before recalculating the histogram, giving time to change the values before the image is rendered again.

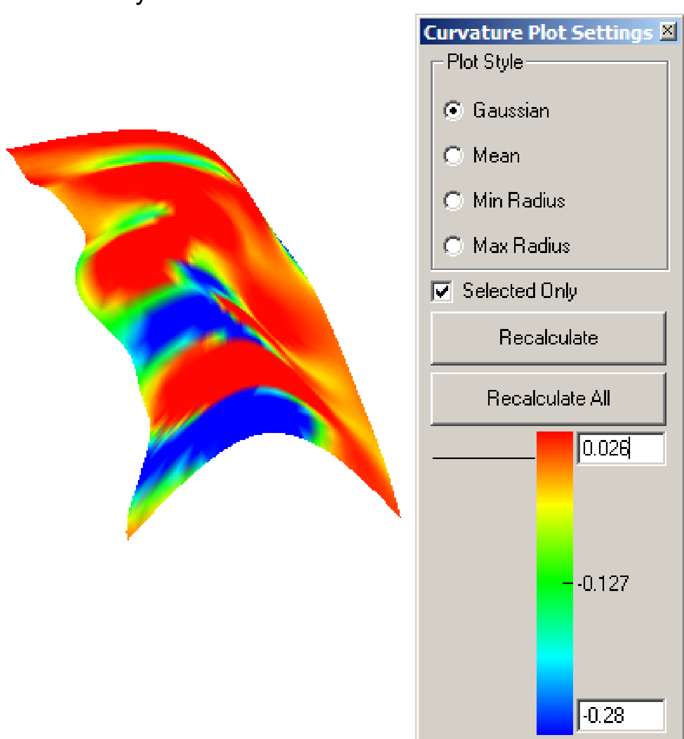
Return to the range initially displayed by clicking on the selected option again. (The option does not deselect when doing this.) Windows users can also reset the maximum and minimum values by clicking right mouse button near the respective end of the spectrum.

Tech Note: *The* **Split Surface** *tool will not work with grouped surfaces.*

Plot Styles

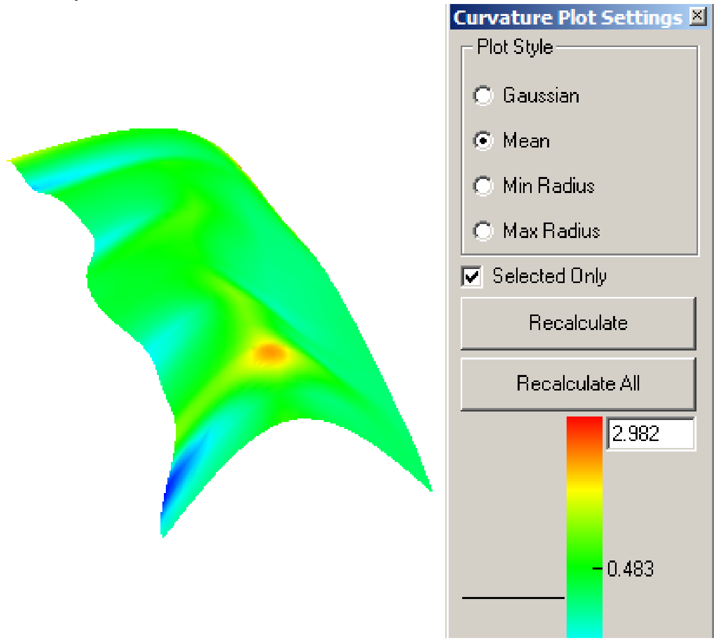
There are four curvature plot styles to choose from when selecting the *Curvature* command: *Gaussian*, *Mean*, *Min Radius* and *Max Radius*.

Gaussian Creates a Gaussian curvature plot on the 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.



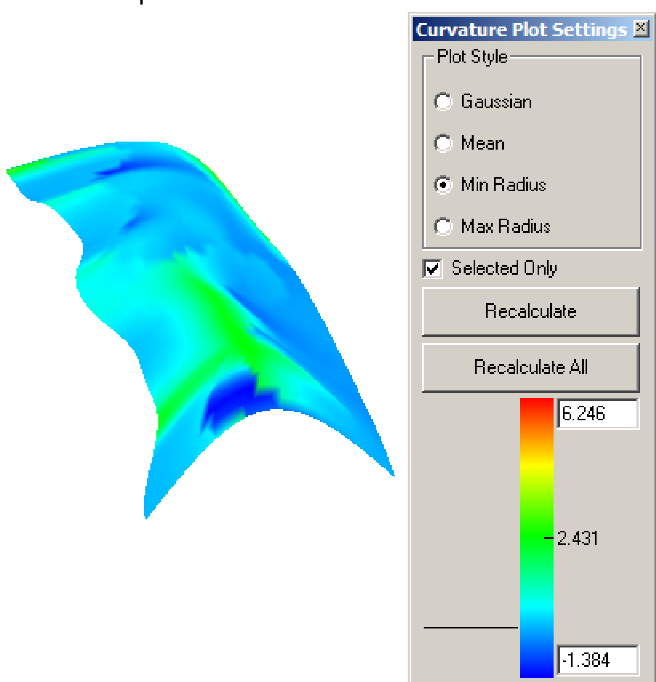
Mean

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.



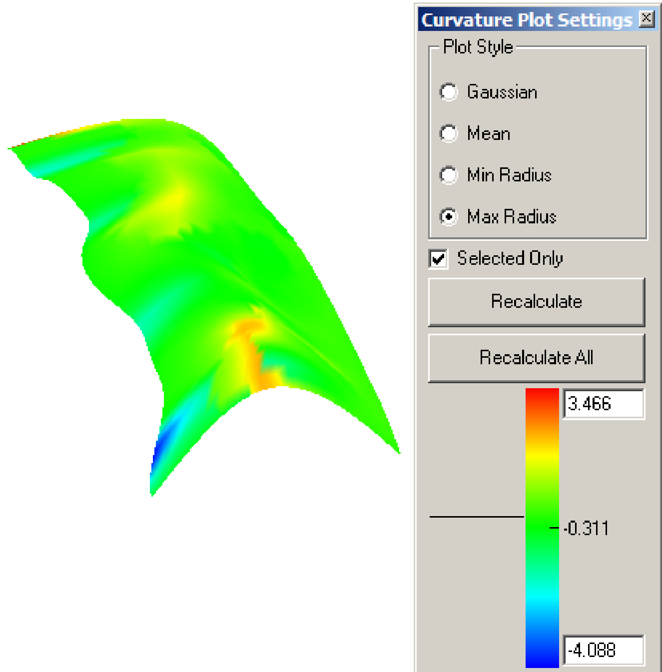
Min Radius

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

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.

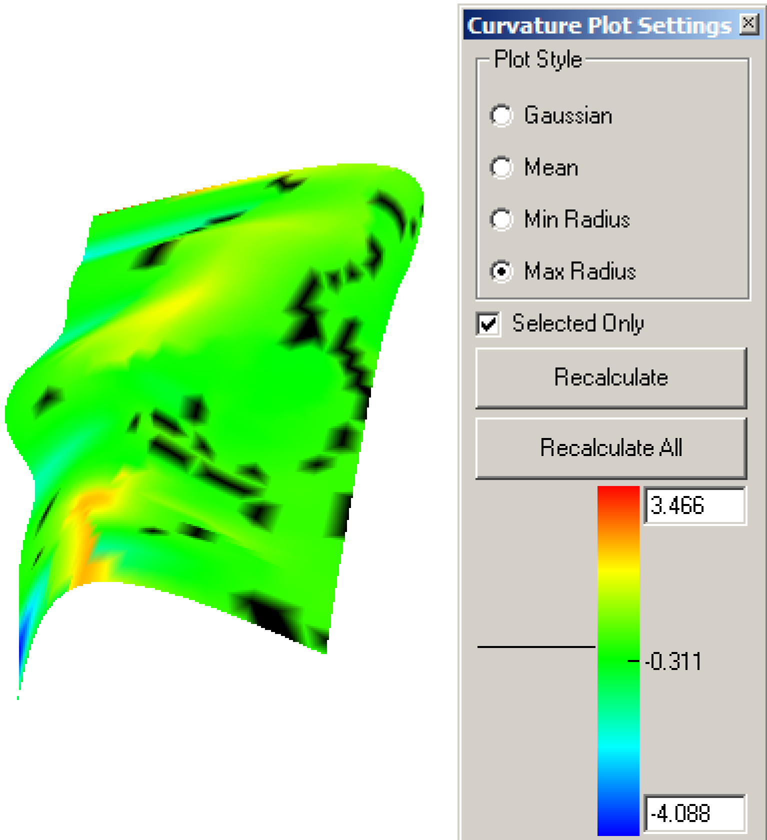


Using a Curvature Surface Analysis Command

1. Select the surface.
2. Choose ***Verify>Surface Analysis>Curvature Settings*** or double click the object to open the Edit Object window, click on the Display tab and press the Edit button.

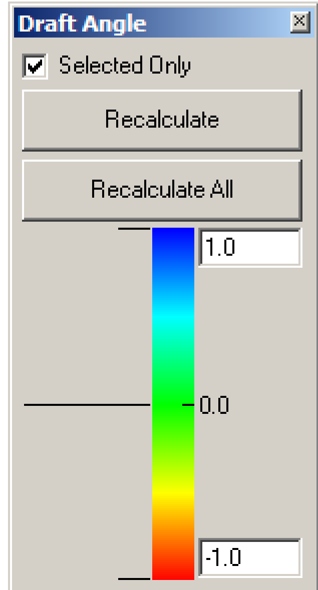
   The dialog box displays with the analysis. The geometry also displays the analysis.
3. Select a Plot Style if the desired style is not already selected.

   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 the 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 press the Recalculate button to update the surface curvature analyses for the plot style selected, or press the Recalculate All button to update the surface curvature analyses for all the plot styles along with the histogram and analysis.

Draft Angle



This command evaluates the drafts of an object for molding purposes. When selecting this command the Draft Angle dialog box appears. It is accessible through ***Verify>Surface Analysis>Draft Angle Settings*** or by checking the option on the Display tab of the Edit Object window and pressing the Edit button. When selecting this command the Draft Angle dialog box appears.

The dialog box contains the following elements:

Selected Only — Specifies whether to apply the settings only to the selected objects.

Recalculate — Updates the surface analyses.

Recalculate All — Updates the surface analyses.

Histogram — Represents the frequency of a draft angle using the color spectrum. The length of the bar represents the frequency.

The 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 — Display the maximum and minimum values used to calculate the histogram. When initially selecting the command the program scans the surface(s) and sets the min and max fields for the draft angle analysis.

When entering different values in the fields, so 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.

Making changes to the values, the program waits for two seconds of non-action before recalculating the histogram, giving time to change the values before the image is rendered again.

Return to the range initially displayed by clicking on the selected option again. (The option does not deselect when doing this.) Windows users can also reset the maximum and minimum values by right mouse clicking near the respective end of the spectrum.

Tech Tip: *The draft angle analysis is based on the relative angle between the normal to the surface and the normal to the current work plane.*

Using the Draft Angle Command

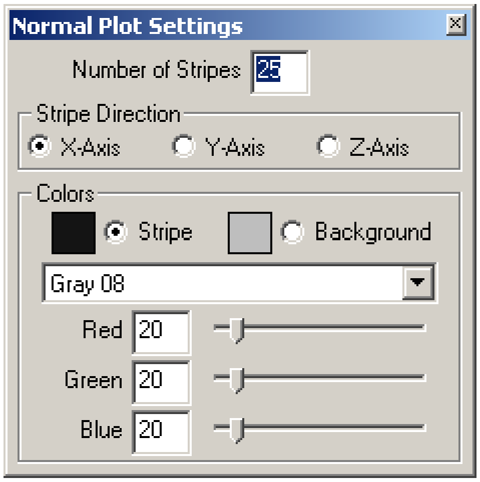
1. Select the object.
2. Choose ***Verify>Surface Analysis>Draft Angle*** or check the option on the Display page of the Edit Object dialog box and click the Edit button.

   The dialog box displays with the analysis. The 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 the 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

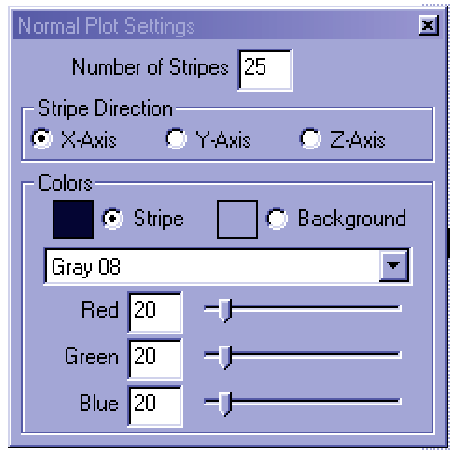
This command creates a normals plot of the surface. When selecting the command the Normal Plot Settings dialog box appears.



The dialog box contains the following elements:

Number of Stripes	Specifies the number stripes that appear on the surface.
Stripe Direction	Provides three direction options: X-Axis, Y-Axis and Z-Axis.
Colors	Specifies 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: sets the stripe color from the color pull-down menu or the RGB fields.
	Background Color option: sets the surface background color from the color pull-down menu or the RGB fields.

This graphic shows an example of a normal surface analysis.



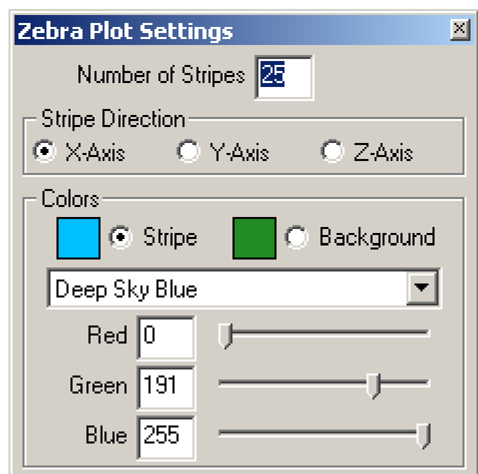
Using the Normal Command

1. Select the surface.
2. Choose ***Verify>Surface Analysis>Normals***.

   The dialog box displays with the analysis. The geometry also displays the analysis.

Zebra

This command creates a zebra plot of the surface. When selecting the command the Zebra Plot Settings dialog box appears.



The dialog box contains the following elements:

Number of Stripes Specifies the number stripes that appear on the surface.

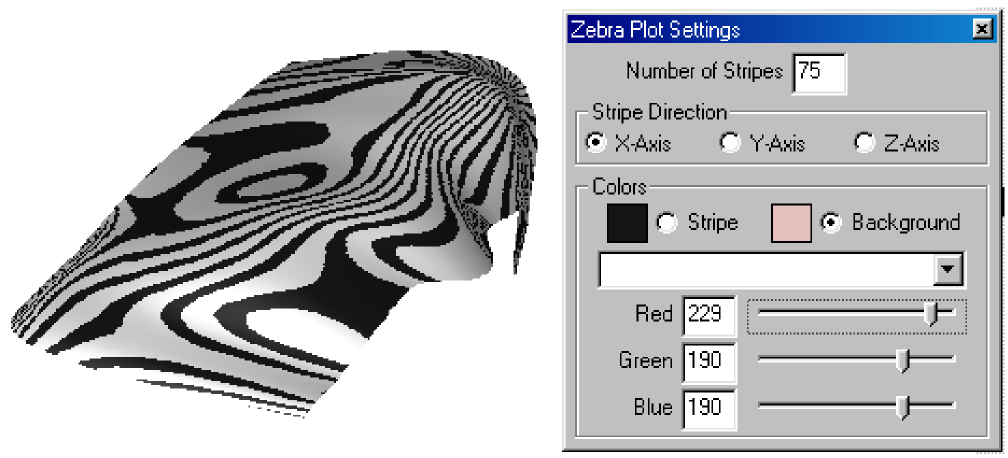
Stripe Direction Provides three direction options: X-Axis, Y-Axis and Z-Axis.

Colors Specifies 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: Sets the stripe color from the color pull-down menu or the RGB fields.

Background Color option: Sets the surface 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***.

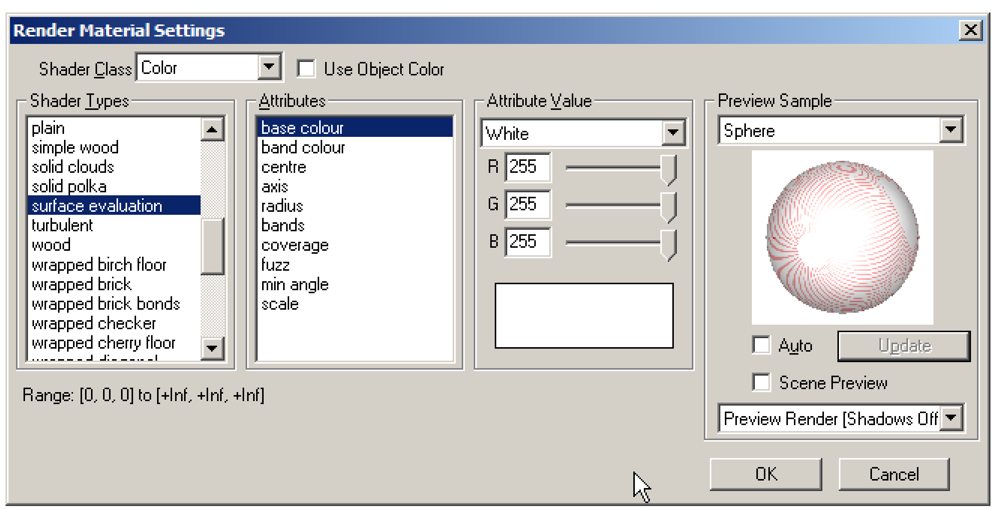
   The dialog box displays with the analysis. The geometry also displays the analysis.

Surface Evaluation Through the Advanced Rendering Feature

It is possible to evaluate the surfaces using user-defined attributes through the Advanced Rendering feature.

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. On the *Material* page of Edit Objects dialog box, click the Advanced button.
4. From the Color class choose the surface evaluation type. The following window displays:



The following list of attributes is available:

base color	Sets the base color of the cylinder used for the evaluation. This color is reflected onto the surface. If the surface color is dark and the base color is white, there will not be much change to the surface color when rendered.
band color	Sets the band or stripe color for the cylinder. This color is also reflected onto the surface. Use a color that is clearly visible to create the surface evaluation.
center	Sets the center of the evaluation cylinder. Click the desired location in the drawing and the numbers automatically enter into the X, Y and Z data fields.
axis	Specifies the cylinder axis (0 = X axis, 1 = Y axis, 2 = Z axis).
radius	Sets the cylinder radius.
bands	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	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.
scale	Controls the overall scale of the pattern produced and accurately accounts for changes to the scale at which the geometry is modeled.

5. Specify the value for each attribute.
6. Click the Update button to preview the surface evaluation in the preview window.

   Select the *Auto* option to make the preview window automatically update each time an attribute is changed.

   Select the *Scene Preview* option to view material setting changes in real time.
7. Click OK to save the changes and close the dialog box.

   Close the Edit Objects dialog box to have a clean screen when rendering.
8. Render the scene using one of the photorealistic commands in the PhotoRender menu.

   The rendered scene shows the surface evaluation.

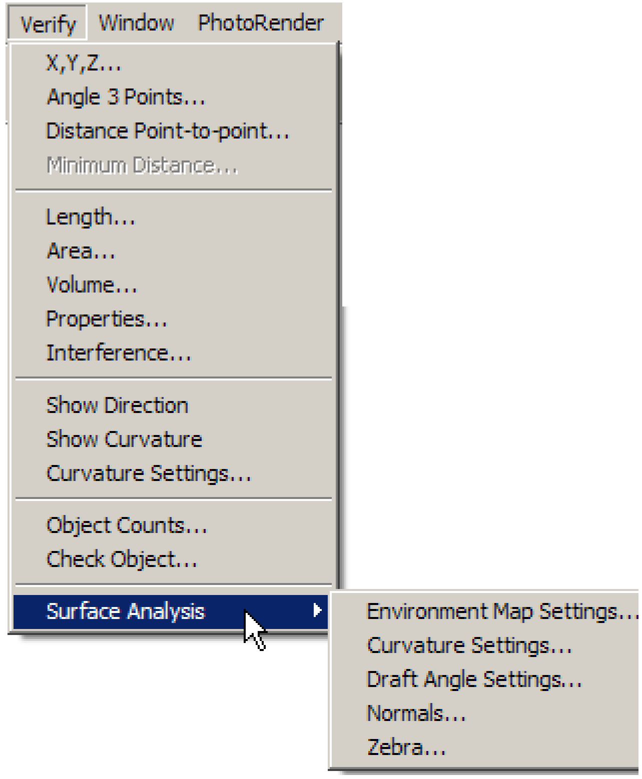
Draft Evaluation

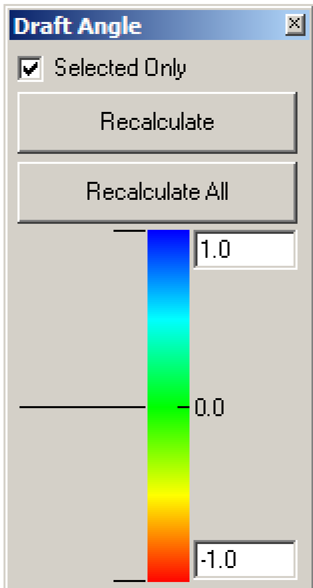
This Designer Elements program can evaluate the drafts of an object for molding purposes. This feature is available through the Verify menu, the Advanced Rendering capabilities and on Display tab of the Edit Objects window.

Draft Evaluation Through the Verify Menu or Edit Object Window

The *Draft Angle* in the Verify menu or on the Display page of the Edit Object window quickly evaluates the draft angles of a model.

All analysis colors are preset with this commands. To define custom settings, use the draft evaluation shader through the Advanced Rendering feature.





This command evaluates the drafts of an object.

The dialog box contains the following elements:

Selected Only Applies the settings only to selected objects.

Histogram Represents the frequency of a draft angle using the color spectrum. The length of the bar represents the frequency.

The program calculates this histogram and displays it so that the entire graph fits into the dialog box area.

Color Spectrum Bar Displays the color septum used to create the histogram.

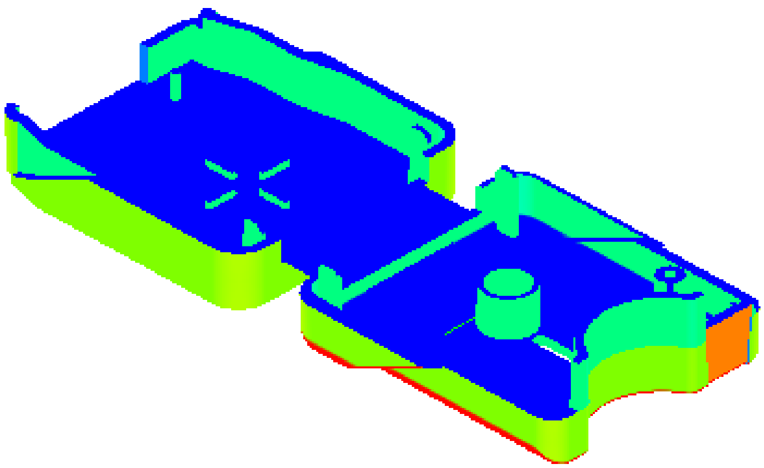
Histogram Data Fields Display the maximum and minimum values used to calculate the histogram. When initially selecting the command, the program scans the surface(s) and sets the min and max fields for the draft angle analysis.

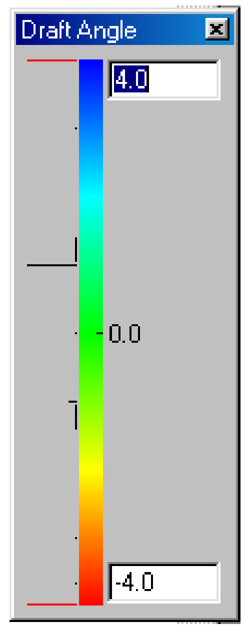
If different values are entered in the fields so that a large number of values fall outside of the specified range, a red line appears at the end(s) where the values fall. All values are still calculated, even though they are not displayed due to the specified range.

If changes are made to the values, the program waits for two seconds of non-action before recalculating the histogram, giving time to change the values before the image is rendered again.

Return to the range initially displayed by clicking on the selected option again. (The option does not deselect when finished.) 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.

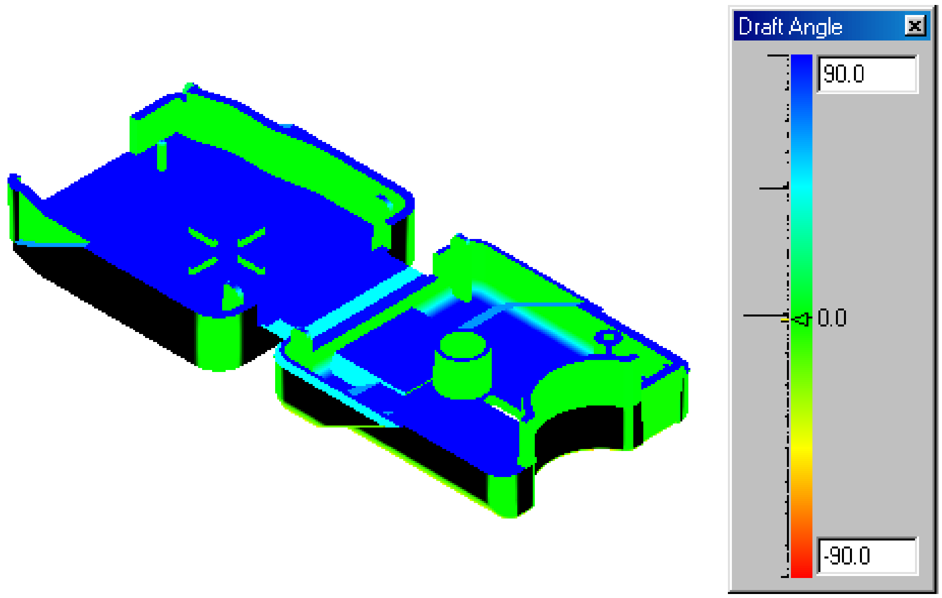




Using the Draft Angle Command

1. Select the object.
2. Choose ***Verify>Surface Analysis>Draft Angle*** or double click the object to open the Edit Object dialog box, check the option on the Display page and click the Edit button.

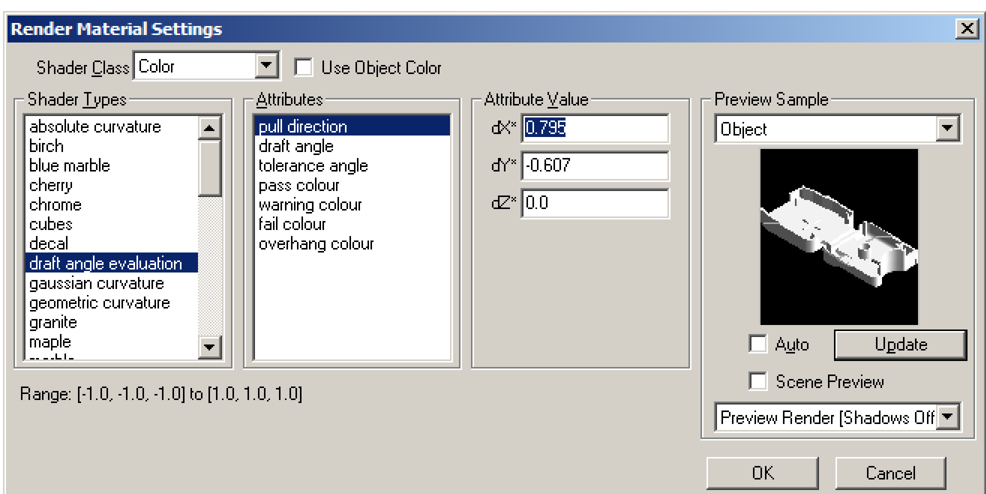
   The dialog box displays with the analysis. The geometry also displays the analysis.
3. To display a certain analysis/color area, place the pointer at the location over the color spectrum and click the mouse.



   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 the 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, it is possible to specify custom draft angle settings. When displaying the Render Material Settings dialog box, choose the draft angle evaluation type and the options appear.



The draft angle evaluation type contains the following attributes:

pull direction — 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 — 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 — Specifies the degree of 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.

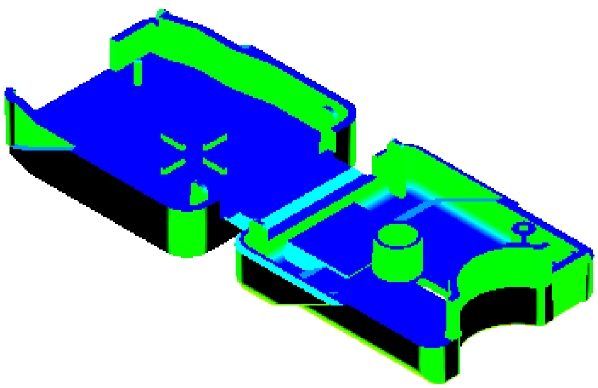
pass color — Specifies the color of the object for those areas where the draft angles are satisfactory. The surface normals fall between **90°+ draft angle + tolerance angle** and **180°** with the pull direction.

warning color — 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 — 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 — 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 below, depending upon whether you are viewing it in black and white or color, the pass color areas are blue or medium gray, the overhang color areas are black and the warning color areas are green or light gray.



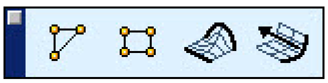
Using the Draft Evaluation Feature

1. Double click on the object to be evaluated 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 the object using one of the photo-realistic commands.

   The object displays using the specified draft evaluation colors and values.

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.

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 for rapid prototyping machines and for compatibility with programs that can only read meshes out of AutoCAD DXF and DWG 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 the Designer Elements program, the **Ruled Mesh** tool is the most powerful. This tool creates sections and “skins” 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.

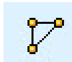
Values for each tool can be entered in the Status Line to define a mesh, either before or after creating the mesh. If the values are entered after selecting the tool but before creating the mesh, the first click in the drawing area automatically registers all Status Line values. If you enter values in 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.

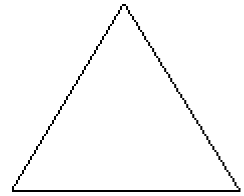
All three of the Designer Elements programs support four tools for creating mesh objects, **3-point Mesh, 4-point Mesh, Ruled Mesh and Extrude Mesh**. The topics explained in this chapter include:

- 3-point Mesh Tool
- 4-point Mesh Tool
- Ruled Mesh Tool
- Extrude Mesh Tool
- Mesh and Rendering
- Mesh Notations

To access the **Mesh** tools go to ***Window>Mesh***.

3-point Mesh Tool





The **3-point Mesh** tool creates a flat plate triangular object from three selected points.

Using the 3-point Mesh Tool

1. Select the tool. The Message Line reads: *3-point Mesh: Enter 3 locations for mesh*.

   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.

> ***Tech Note:*** *To select a mesh object immediately after creating it, choose the* **Selection** *tool. Otherwise, new mesh objects will continue being created.*

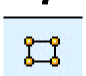
Geometric Characteristics

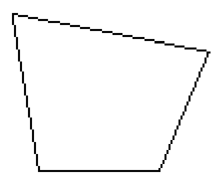
A three-point mesh is created by picking the three desired points. A mesh is made up of the following characteristic according to the Edit Objects dialog box: the Defining Point for the X, Y and Z location.

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.

4-point Mesh Tool





The **4-point Mesh** tool creates a flat plate quadrangular object from four selected points.

Using the 4-point Mesh Tool

1. Select the tool. The Message Line reads: *4-point Mesh: Enter 4 locations for mesh*.

   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.

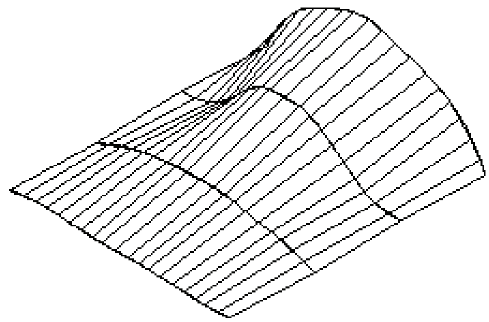
Geometric Characteristics

A four point 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.

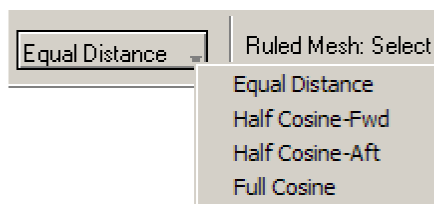
This information is listed in the Edit Objects dialog box under the Geometry tab. To display the dialog box, select the mesh and choose ***Window>Edit Objects*** or double-click on the mesh.

Ruled Mesh Tool



The **Ruled Mesh** tool creates an *M* by *N* mesh from a collection of curves. M is the number of curves selected to define the mesh boundary. The mesh is created from the order in which they are selected. N is number of segment lines displayed on the mesh. The graphic above shows a mesh curve with M = 4 (curves) and N = 20 (grid lines).

The **Ruled Mesh** tool provides several options that control mesh spacing. Pick them from the drop down list in the Message Line.



The options include:

Equal Distance Mesh points are spaced equally.

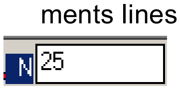
Half Cosine (Fwd) Mesh points are spaced more closely at the forward end of the mesh, according to a half cosine.

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: Ruled *Mesh: 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 a different number of segments is necessary, 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. To select more than two curves, hold down the SHIFT key before selecting the first curve.

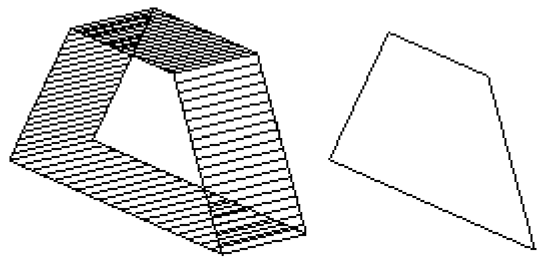
   Select the curves in the order that the mesh is created.

Geometric Characteristics

A ruled mesh 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.

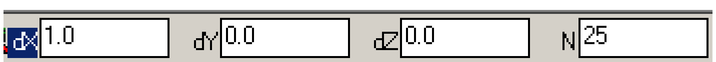
Extrude Mesh Tool



The **Mesh Extrude** tool sweeps a collection of curves along a vector, creating *N* mesh segments.

Using the Extrude Mesh Tool

1. Select the tool. The Message Line reads: *Extrude Mesh: Select objects to extrude [Shift=Extend]*.
2. The Status Line displays data fields 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. To select more than one curve, hold down the SHIFT key before selecting the first curve.
4. To enter a different value for the N segments than currently listed, select the field and type the new value. After the curves are selected, The Message Line reads: *Extrude Mesh: 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.

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. It is possible to display the mesh as Wireframe, Flat, Gouraud, Gouraud w/Edges, Phong, Phong w/Edges, Hidden and Hidden with Dimmed. See “Shade Options” for an explanation of these options.

Mesh Notations

The following notations will be helpful as for creating mesh objects.

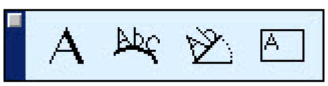
- Mesh objects are not surfaces. They are defined by nodes or 3D vertices.
- Since mesh objects are not surfaces, the **Surface Editing** tools, contained in the third set of tools within the **Surface** tool palette, cannot be used.
- It is possible to convert surfaces and solids into meshes and convert a closed mesh back into a solid.

Text

To add some basic annotations to the drawing, use one of the text tools. The text can be created and edited. Use the Text menu to set the characteristics such as font, text size, case and alignment. The topics discussed in this chapter include:

- Text Tools
- Text Commands
- Text Handling

Text Tools



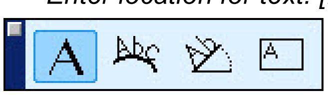
Cobalt, Xenon and Argon three tools for creating text: **Horizontal Text**, **Text Along a Curve** and **Text at an Angle**. The text tools in these programs do not use text boxes. Simply click the cursor at the location where the text will begin and start typing. Press ENTER (Windows) or RETURN (Macintosh) 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 specified location.

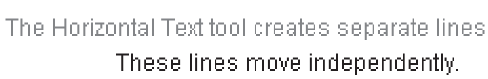
Using the Horizontal Text Tool

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. Press ENTER (Windows) or RETURN (Macintosh), to 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.



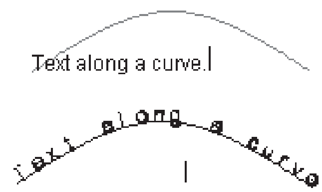
Text Along a Curve Tool

The **Text Along a Curve** tool creates text along the specified curve.

> ***Tech Note:*** *This tool may not support all of the 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 the text will appear, placing the points in the order that the text will be 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 just created curve. The cursor becomes a text cursor placed at the start point of the curve.
4. Enter the desired text. It displays horizontally.



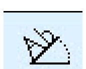
5. Press ENTER (Windows) or RETURN (Macintosh) and the text line molds to the curve.

A parent/child relationship exists between the curve and the text. Any change made 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, 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. The text can not be moved independently of the curve. To hide the curve with the text, change the curve color to blend with the background or place the curve on the construction layer. Then it still will be possible to edit the text appearance on screen by editing the curve.

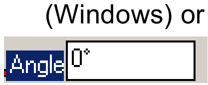
Text at an Angle Tool



The **Text at an Angle** tool creates text at the angle and specified location.

> ***Tip:*** *If the angle is entered in the Status Line after typing the text, 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.

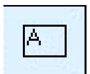


3. Place the cursor at the beginning location for the text and click. The cursor becomes a text cursor.
4. Enter the desired text. The text displays horizontally.
5. Press ENTER (Windows) or RETURN (Macintosh) and the text angle changes.

Geometric Characteristics

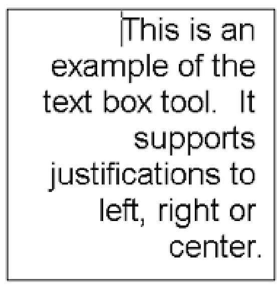
A text object is created by clicking the location of the start point of the text and typing the desired text. A text object is made up of the following characteristics according to the Edit Objects dialog box: Text and Angle. To display the dialog box, select the text and choose ***Window>Edit Objects*** or double-click on the text.

Bounded Box Text Tool



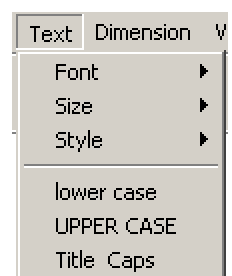
Bounded Box Text is defined by two diagonal points used to specify the width and height of a text box. Text is automatically wrapped to the width of the box. The text box also supports left, right and center justifications.

In addition, 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, copy a paragraph from Microsoft Word and paste it into a text box.



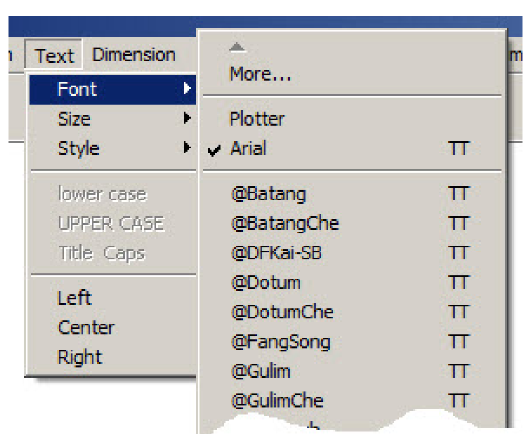
Text Commands

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 the computer and one Plotter font. Use the Plotter font whenever creating text for a drawing that is intended to be sent to a plotter, since plotting Postscript or TrueType fonts takes much more time and fonts may be substituted.

If text is selected and the font is changed, only the selected text is affected. If no text is selected and the font is changed, this sets the default font for future entries until another font is chosen.

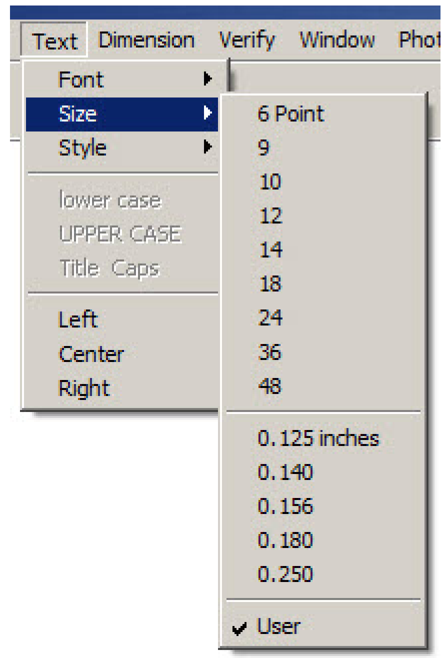
Special characters and accents are available as described in Appendix B.

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 text is selected and the size is changed, only the selected text is affected. If no text is selected and the text size is changed, the default size is set for future entries.

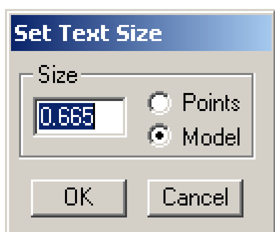
The size can be specified as either points (12 pt) or as units (.156 inches) as defined in the *Units* page of Preferences. If no unit is specified, the program picks the default unit.



Specifying a Non-standard Text Size as the Default Size

Choosing *User* from the Size submenu brings up a dialog box to specify the exact text size. Choose a point size or the units for the 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.

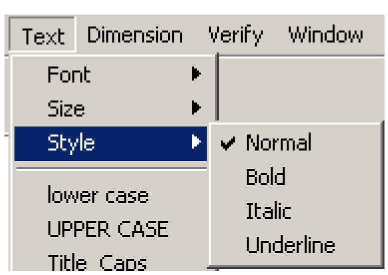
Choose *Model* to measure in the current units as set in *Units* page of Preferences.

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 specified stays in effect until another size is chosen.

Style

This command in the Text menu sets the style (such as **Bold** or Underline) for selected text or future text entries.

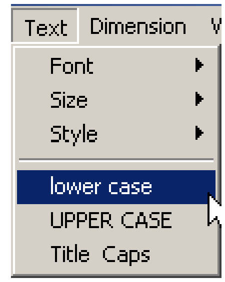


The specified style stays in effect until another default style is chosen. A check indicates the current style.

To change the style of one text line, select the text and choose ***Text>Style*** and necessary option. This does not affect the default style setting.

Case

Cobalt, Xenon and Argon all support 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.

Case cannot be set 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***.

   The text line changes to reflect the choice.

> ***Tip:*** *To change from UPPER CASE to Title Caps, first change the text to lower case and then change the case to Title Caps.*

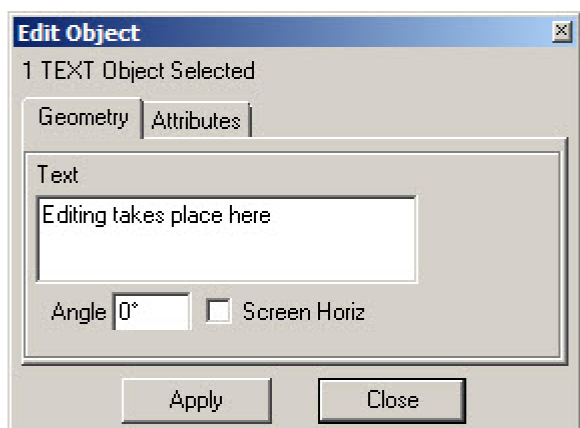
Text Handling

It is possible to perform various operations on the text like editing, moving etc.

Editing Text

To edit created text, 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 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

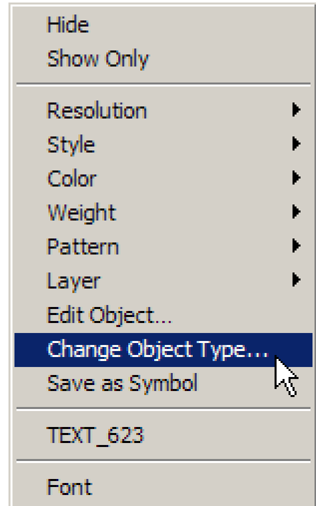
As seen in the picture above, the Edit Objects dialog box contains a check box for keeping text horizontal to the screen. With the box checked, the text will stay horizontal to the screen when using the Trackball to rotate the drawing. With the box unchecked, the text will stay horizontal to the plane it was created on. For example, if text is created in the top view and the view is switched to the trimetric, text will appear as if it were lying flat.

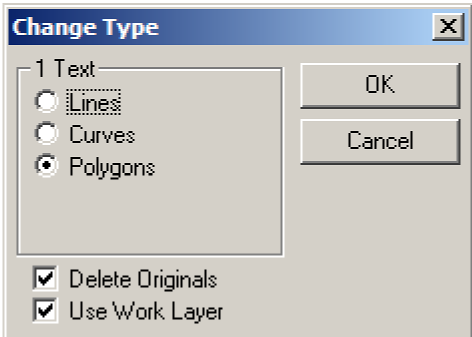
Convert Text into Lines, Curves, Poligons.

It is possible to change the object type of the text.

1. After the text is typed, choose **Select** tool.
2. Right click on the text and choose the Change Object Type option from the context menu.

   The Change Type dialog window appears.



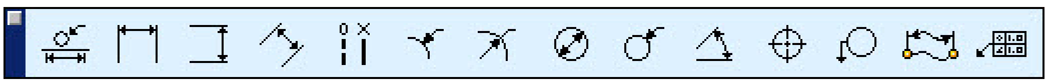


3. Choose the type to which to convert the text.
4. Uncheck the Delete Originals to leave the original text. In the picture below, the example on the left shows the result where original text was deleted, in the right example, the text was left. When the original text is not deleted, both the text and the new object can be selected and further edited with the Edit Object dialog..

5. Uncheck Use Work Layer box to be able to save the new object on any layer except the work layer.

Dimensions

This chapter explains how to use the dimension tools in Cobalt, Xenon or Argon. The



following topics are covered:

- Dimension Menu
- Dynamic Dimensions
- Dimension Tools
- Smart Dimension Tool
- Horizontal Dimension Tools
- Vertical Dimension Tools
- Oblique Dimension Tool
- Ordinate Dimension Tools
- Radial Arrow Out Dimension Tool
- Radial Arrow In Dimension Tool
- Diameter Arrow In Dimension Tool
- Diameter Arrow Out Dimension Tool
- Angular Dimension Tool
- Dimension Appearance
- Dimensions and Drawing Views

Dimension Menu

The Dimension menu contains commands for setting the format for dimensions. Before starting dimensioning, choose the Dimension tool from the Dimension tool palette.

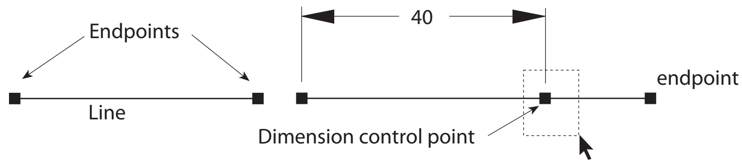
Like the other tool palettes, the **Dimension** palette is a floating tool palette. Select dimension tools in the same way as tools from the main tool palette. A few of the tools have subpalettes with subtools and options for those tools.

Dynamic Dimensions

Cobalt and Xenon's geometric dimensions are dynamic—when a change to the dimension control points is made, the dimension text changes also. This is not true if the # symbol is overwritten in the data field or Edit Objects with a set value/text string.

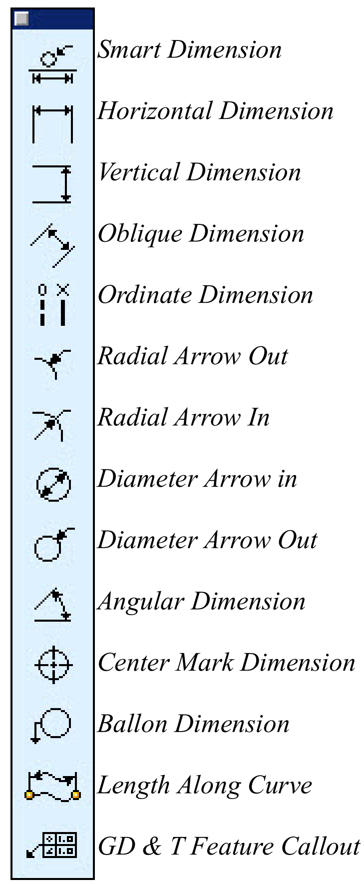
The dynamic nature of the dimension entities is a tremendous time saver in that not only do the numbers in the dimension update when the control points are moved, but the graphical characteristics will also change when a new dimension style is applied. The units can be changed from English to metric (in the Units page of Preferences) and every dimension on the drawing will reflect the change.

When extending 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 selecting the endpoint of the line the dimension control point is also selected.



Dimensions are dynamic relative to the points they measure. If the length of a line is changed 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



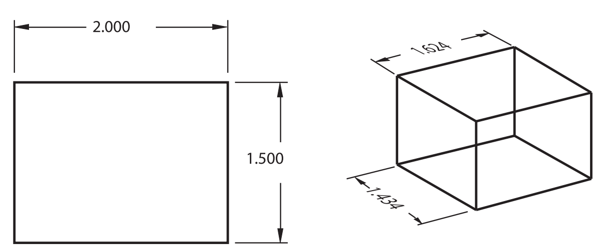
The Dimensions palette is located in the standard tools palette.

These tools can be used to measure either an object or the area between objects. Once an object (or space) is dimensioned, the dimensions update when you make changes to the geometry are made **(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 exiting 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 beginning 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. Working in the front plane and 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. Selecting 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 to measure.

   Dimensions automatically use the dimension pen style and current dimension text characteristics. To change the dimension pen color or weight, do so in the Dimension menu.
3. Move the dimension to a new location if it is not in the necessary location.

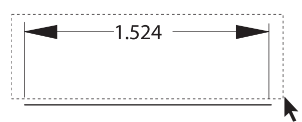
Moving the Dimension

After placing a dimension, it is possible to reposition it relative to the geometry. The steps differ depending on the tool. The dimension can be repositioned 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.

   To select more than one dimension drag a selection fence around them.



The text can also be repositioned along the dimension line by dragging it along the line.

For radial and diameter dimensions, 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 this symbol is deleted, the dimension will no longer be associative (that is, it won't update if the geometry it measures is changed).

It is also possible to add text before or after the # symbol which will display with the dimension text.

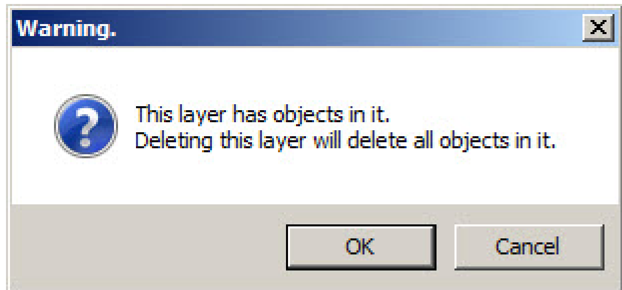
Geometric Characteristics

Dimension text is created automatically when an object is dimensioned and is made up of one characteristic according to Edit Objects, Text.

Automatically Placing Dimensions on a Separate Layer

Cobalt, Xenon and Argon automatically create a Dimension layer. Dimensions automatically go on that layer except in the case of dimensions placed in drawing views. These drawing view dimensions go on the Sheet View layer. For more information see Drawing Views.

If the Dimension layer is accidently deleted, the program automatically recreates the layer if the file is closed and opened. If it is necessary to delete the Dimension layer and there are objects on it, the program displays the following warning.



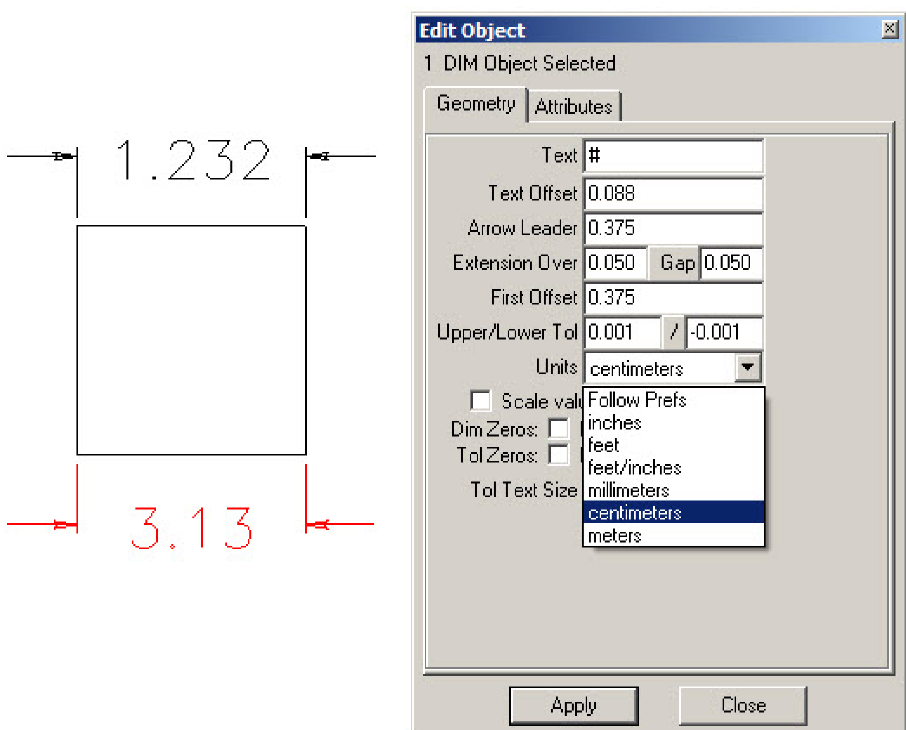
Dimensions and Edit Objects

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 the # symbol is deleted and another value is entered, the dimension is fixed and will not update if the units or the size of the geometry is changed.

Some of the dimensioning tools, such as the **Radial** tools, add a letter in the text status box which appears in the dimension itself.

Text can be added before or after the # symbol. Enter parentheses to enclose the resulting dimension in parentheses (#), or enter **# 2 Plcs** to display the dimension followed by the text 2 Plcs. When including the # symbol, the measurement of the geometry is used as well as the added text.

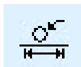
Edit Objects also includes dimension characteristics including a Units data field. This field sets the units for a selected dimension to those in Preferences or another settings. This dimensions a part with mixed units. The graphic below shows an example of mixed units with the Edit Objects dialog box.



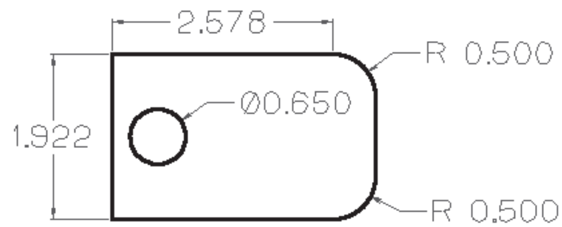
Dimensioning Surfaces and Solids

Surfaces and solids can be dimensioned 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



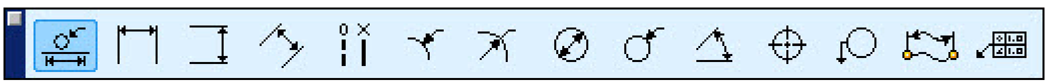
This tool in the **Dimension** tool palette quickly places horizontal, vertical, radial and diameter dimensions with a single click. These dimensions are associative to the curves. As the curve is changed, the dimension automatically updates (**Cobalt** and **Xenon** only). This tool is especially useful for drawing views.



Cobalt and Xenon Only

Using the Smart Dimension Option

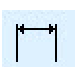
1. Select the tool from the **Dimension** tool palette. The Message Line reads: *Smart Dimension: Pick curve to dimension*.



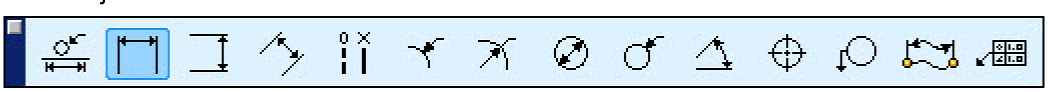
2. Select the curve.

   The dimension appears. You can drag the dimension to a new location. Dimension and extension lines automatically redraw.

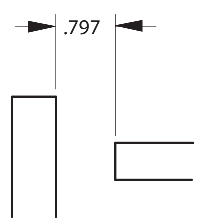
Horizontal Dimension Tools



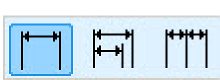
These tools dimension objects or spaces horizontally or measure the distances between linear objects.



When selecting the **Horizontal Dimension** tools, a subpalette appears in the Message Line containing three options, **Horizontal Dimension**, **Horizontal Base Line Dimension** and **Horizontal Chain Dimension**.



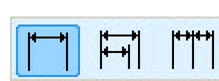
Horizontal Dimension Option



This option dimensions objects or spaces horizontally.

Using the Horizontal Dimension Option

1. Select the **Horizontal Dimension** option. 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.

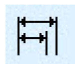
   The Message Line now reads, *Horizontal: Pick next dimension point*.

3. Click the right point.

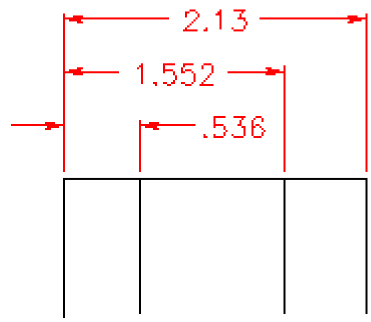
   The dimension appears. Drag it as necessary to a new location. Dimension and extension lines automatically redraw.

   Click the points in the opposite order to display the dimension below the objects.

Horizontal Base Line Dimension Option



This tool dimensions objects or spaces from a base point.



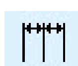
Using the Horizontal Base Line Dimension Option

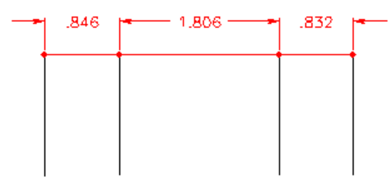
1. Select the **Horizontal Dimension** tools. Choose the **Horizontal Base Line Dimension** option. The Message Line reads: *Horizontal Base Line: Pick first dimension point*.
2. Click the base point of the geometry.

   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 to dimension. Each dimension can dragged to a new location. Dimension and extension lines automatically redraw.

   Click the points in the opposite order to display the dimension below the objects.

Horizontal Chain Dimension Option





This option dimensions objects or spaces from end to end, horizontally.

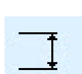
Using the Horizontal Chain Dimension Option

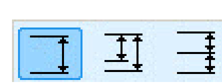
1. Select the **Horizontal Dimension** tools. Choose the **Horizontal Chain Dimension** option. The Message Line reads: *Horizontal Chain: Pick first dimension point*.
2. Click the first point on the geometry.

   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 to be dimensioned. Each dimension can be dragged to a new location. Dimension and extension lines automatically redraw.

   Click the points in the opposite order to display the dimension below the objects.

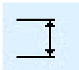
Vertical Dimension Tools

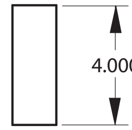




These tools measure an object or the vertical space or the distance between vertical objects. When the **Vertical Dimension** tools are selected, a subpalette appears in the Message Line containing three options: **Vertical Dimension, Vertical Base Line Dimension** and **Vertical Chain Dimension**.

Vertical Dimension Tools

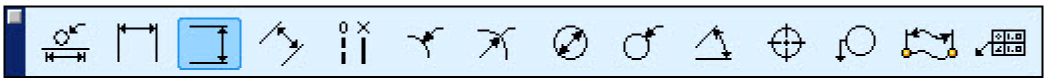




This tool dimensions an object or space vertically.

Using the Vertical Dimension Tools

1. Select the **Vertical Dimension** tool. The Message Line reads: *Vertical: Pick first dimension point*.



2. Click the top point of the geometry first.

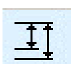
   The Message Line now reads, *Vertical: Pick next dimension point*.

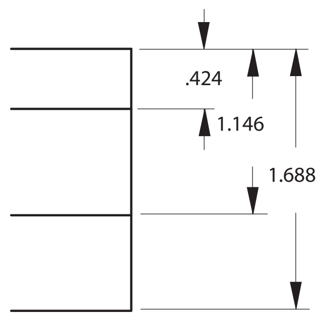
3. Click the bottom point.

   The dimension appears. Drag it to a new location. Dimension and extension lines automatically redraw.

   Click the points in the opposite order to display the dimension below the objects.

Vertical Base Line Dimension Option





This tool dimensions objects or spaces from a base point.

Using the Vertical Base Line Dimension Option

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.

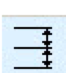
   The Message Line now reads, *Vertical Base Line: Pick next dimension point*.

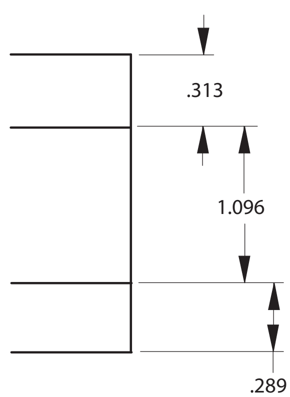
3. Click the point. The dimension appears.

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 to be dimensioned. Each dimension can be dragged to a new location. Dimension and extension lines automatically redraw.

   Click the points in the opposite order if to display the dimension below the objects.

Vertical Chain Dimension Option





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.*

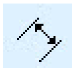
2. Click the first point on the geometry.

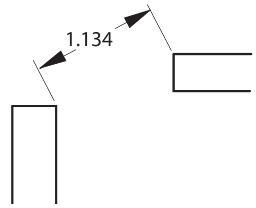
   The Message Line now reads, *Vertical Chain: Pick next dimension point*.

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 to be dimensioned. Each dimension can be dragged to a new location. Dimension and extension lines automatically redraw.

   Click the points in the opposite order to display the dimension below the objects.

Oblique Dimension Tool

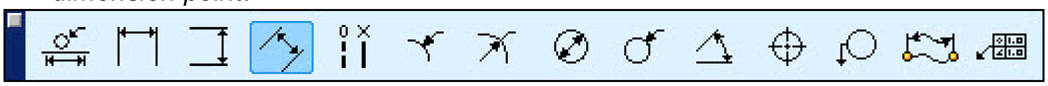




This tool measures space or objects parallel or point to point.

Using the Oblique Dimension Tool

1. Select the **Oblique Dimension** tool. The Message Line reads: *Oblique: Pick first dimension point*.



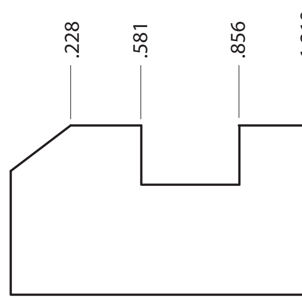
2. Click the left point of the geometry first.

   The Message Line now reads, *Oblique: Pick next dimension point*.
3. Click the right point.

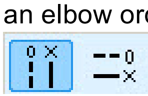
   The dimension appears. It can be dragged to a new location. Dimension and extension lines automatically redraw.

   Click the points in the opposite order to display the dimension below the objects.

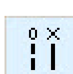
Ordinate Dimension Tools



This tool dimensions objects or spaces from a base point. When the **Ordinate Dimension Tool** is selected, a subpalette appears in the message line with two options: **Horizontal Ordinate Dimension** and **Vertical Ordinate Dimension**. Use the **Selection** tool to move, position or create an elbow ordinate.



Horizontal Ordinate Dimension Option



This option adds dimensions horizontally along a baseline.

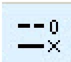
Using the Horizontal Ordinate Dimension Tool

1. Select the **Horizontal Ordinate Dimension** option 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 to be dimensioned. To display a dimension at the base point, click the base point after dimensioning all other points.

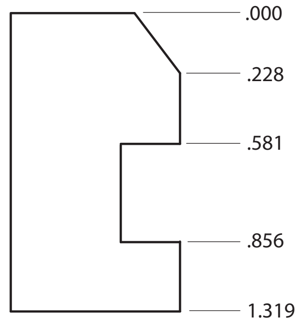
   Drag each dimension to a new location. Dimension and extension lines automatically redraw.

Vertical Ordinate Dimension Option



This option adds dimensions vertically along a line.

Using the Vertical Ordinate Dimension Option

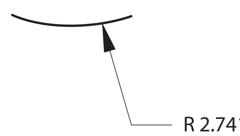


1. Select the **Horizontal Ordinate Dimension** option. The Message Line reads: *Vertical Ordinate: Pick first dimension point*.
2. Click the base point of the geometry.

   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 to be dimensioned. To display a dimension at the base point, click the base point after dimensioning all other points.

   Each dimension can be dragged to a new location. Dimension and extension lines automatically redraw.

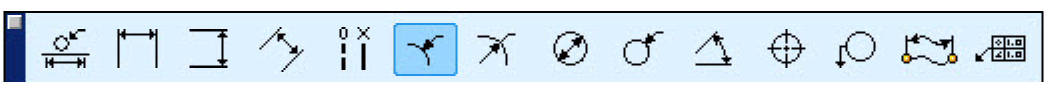
Radial Arrow Out Dimension Tool



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*.

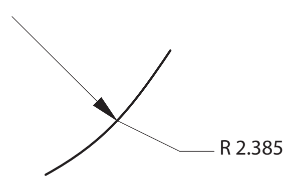


2. Click near the circle, arc or fillet to be to dimensioned.

   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 clicked location. 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 be used only on curves, not surfaces or solids.

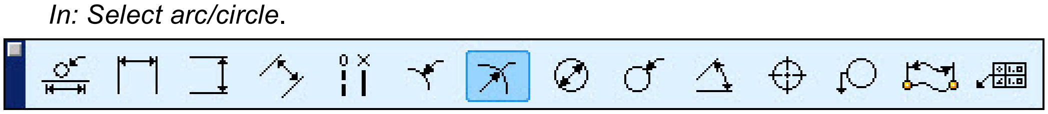
Radial Arrow In Dimension Tool



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*.

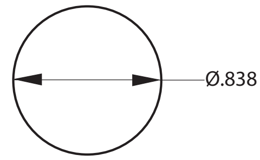


2. Click near the circle, arc or fillet to be dimensioned.

   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 clicked location. 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 be used only on curves, not surfaces or solids.

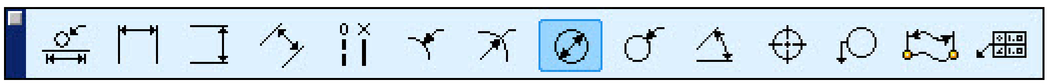
Diameter Arrow In Dimension Tool



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*.

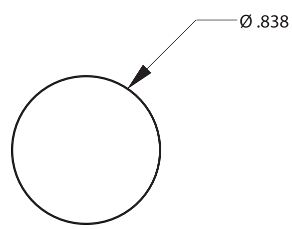


2. Click near the circle or arc to dimension.

   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 clicked location. 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 be used only on curves, not surfaces or solids.

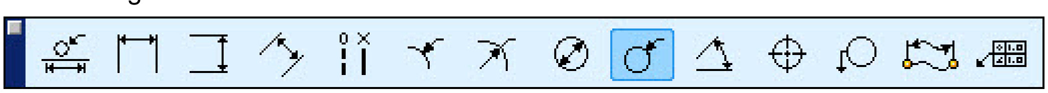
Diameter Arrow Out Dimension Tool



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*.

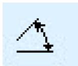


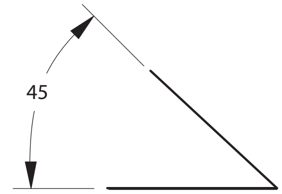
2. Click near the circle or arc to dimension.

   The dimension appears outside the object. When the dimension appears the leader line is placed at the nearest 15° increment from the clicked location. The dimension can be moved 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 be used only on curves, not surfaces or solids.

Angular Dimension Tool

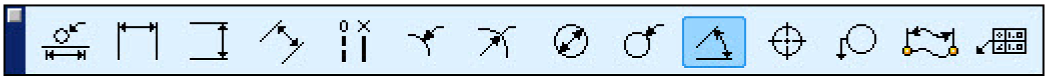




This tool measures the angle between two lines.

Using the Angular Dimension Tool

1. Select the **Angular Dimension** tool. The Message Line reads: *Angular: Pick first line*.



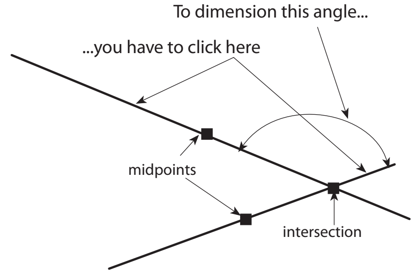
2. Click on the first line near the endpoint from which the angle will be measured.

   The Message Line now reads, *Angular: Pick second line*.

3. Click on the second line.

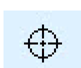
   The inside angle is measured between the lines. The program always measures the smaller angle between the selected lines.

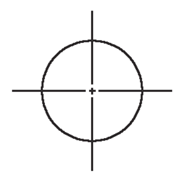
Dimensioning intersecting lines don't confuse the intersection with the midpoint of the lines.



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

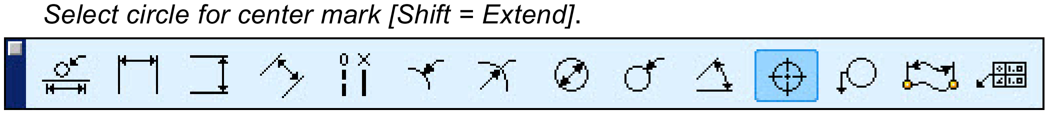




This tool creates a center line mark for circles and arcs. The center mark overlap can be defined in the Status Line or the Edit Objects dialog box. The overlap units are based on those in the Units page of Preferences.

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]*.

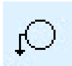


2. Select the circle or arc to dimension. Hold down the SHIFT key if to dimension more than one object at the same time.

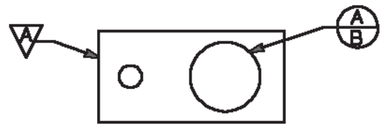
   The Axis overlap value can be changed in the Status Line. Type the value and press ENTER (Windows) or RETURN (Macintosh) and the overlap updates.

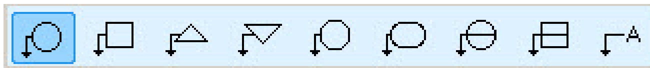
   The Status Line contains the Axis overlap data field.

Balloon Dimension Tools



These tools dimension objects with a callout balloons. There are nine balloon options 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 Tools

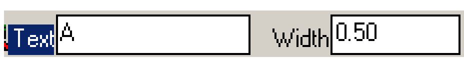
1. Select the **Balloon Dimension** tools.
2. Select the desired balloon shape. The Message Line reads: *Balloon: Enter 2 points for balloon dimension*.
3. Click the point on the geometry for the dimension arrow.
4. Click the second point to specify the location of the symbol.

   The balloon symbol appears.
5. Enter the text and frame width in the Status Line data fields. Press ENTER (Windows) or RETURN (Macintosh) to update the dimension.

   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 additional balloons are placed. Enter specific text in the balloon dimension, if necessary.

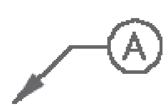
The Status Line contains Text and Width data fields.



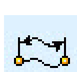
Geometric Characteristics

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

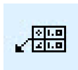


This tool dimensions curved objects such as splines.

Using the Length Along Curve Dimension Tool

1. Select the **Length 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.

GD&T Feature Control Tool



Selecting this icon from the Dimensioning Tools palette activates the new Geometric Dimensioning & Tolerancing (GD&T) GD&T tool in Cobalt. When the GD&T icon is selected, three options appear in the message line: Datum Feature, Datum Target and Feature Control Frame.

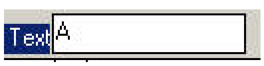
Using the Datum Feature Option

1. Select the **GD&T** icon.
2. Choose the **Datum Feature** option from the Message Line. The Message Line reads: *Datum Feature: Input 2 Points (Arrow and Shoulder).*

3. Click the point on the geometry for the dimension arrow.
4. Click the second point to specify the location of the symbol.

The datum feature appears.

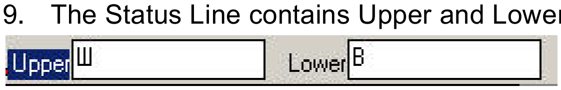
5. Enter the text in the Status Line data fields. Press ENTER (Windows) or RETURN (Macintosh) to update the dimension.
6. Move the dimension by choosing the **Selection** tool, selecting the dimension, placing the pointer over the end of the leader line, and when the pointer becomes the Move symbol, dragging the text to a new location.
7. Enter specific text in the dimension, as necessary.
8. The Status Line contains Text data fields.



Using the Datum Target Option

1. Select the **GD&T** icon.
2. Choose the **Datum Target** option from the Message Line. The Message Line reads: *Datum Target: Input 2 Points (Arrow and Shoulder).*

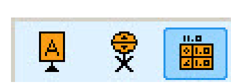
3. Click the point on the geometry for the dimension arrow.
4. Click the second point to specify the location of the symbol.
5. The datum target appears.
6. Enter data for the upper and lower section in the Status Line data fields. Press ENTER (Windows) or RETURN (Macintosh) to update the dimension.
7. 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.
8. Enter specific text in the dimension, as necessary.
9. The Status Line contains Upper and Lower data fields.



Using the Feature Control Frame

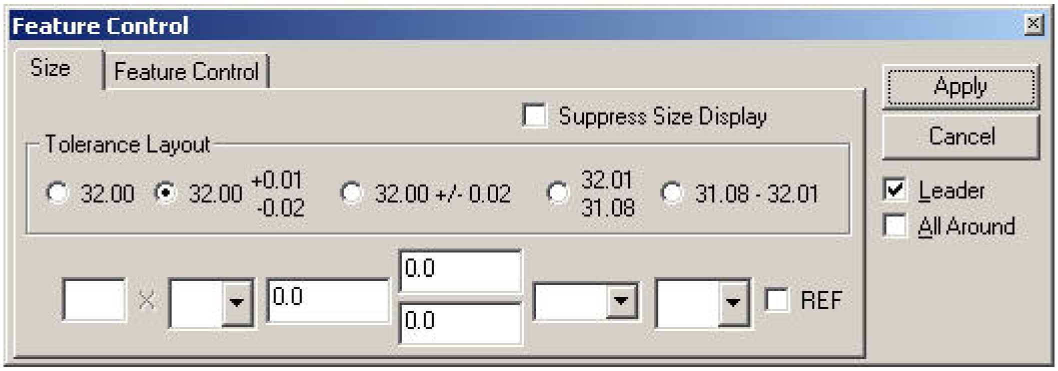
The option displays the Feature Control Frame to format data for the dimensions.

1. Select the **GD&T** tool icon.
2. Choose the **Feature Control Frame** from the Message Line. The Message Line reads: *Feature Control Frame: Input 2 Leader Points (Arrow and Shoulder).*



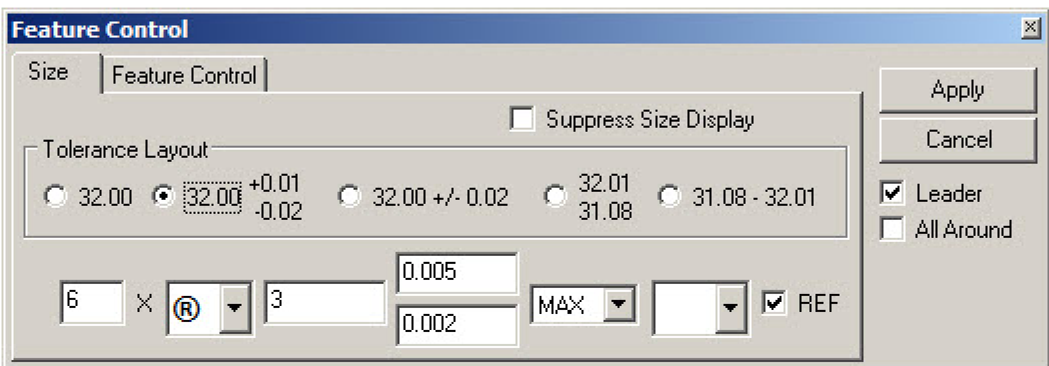
3. Click the point on the geometry for the dimension arrow.

4. Click the second point to specify the location of the symbol. The Feature Control dialog box appears. Fill in the dialog box using the parameters below.



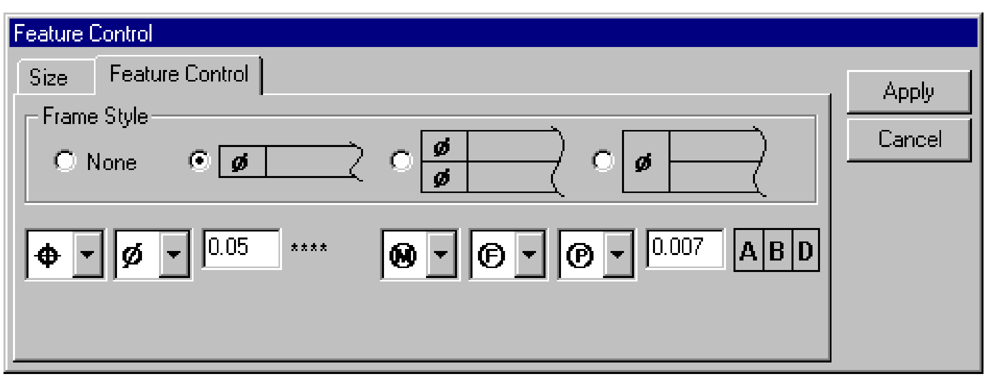
Size Tab

The Size tab defines 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.



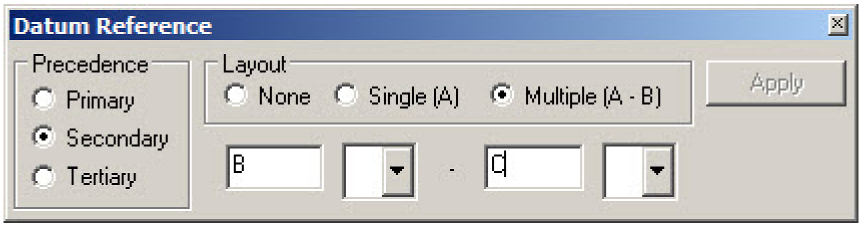
Feature Control

The Feature Control tab defines 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 two 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 (sub-region) 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 Dialog

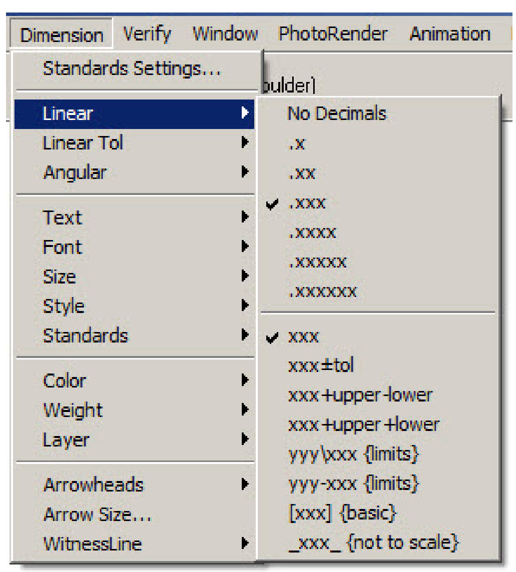
When appropriate for a feature, the Datum Reference Dialog associates existing datum planes (local coordinate origin) with the dimension. A maximum of 3 datums (Primary, Secondary, Tertiary) can be associated with a feature control frame. A datum must be fully defined in order to add another datum reference. 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

Cobalt, Xenon and Argon automatically create dimensions according to ANSI Y14.5, DIN, ISO or JIS standards. Many companies and individuals, however, have developed their own standards. The commands in the Dimension menu control virtually every aspect of the dimension appearance without having to construct dimensions manually and while retaining the Vellum's smart dimensions.

Settings made in the Dimension menu affect the currently selected dimension and all future dimensions.

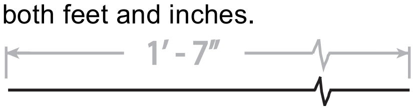
Linear



This command in the Dimension menu a sets the decimal precision.

All dimensions in this program are displayed in decimals regardless of the units set in Preferences.

If feet/inches units are set in Preferences, objects over one foot in length will display both feet and inches.



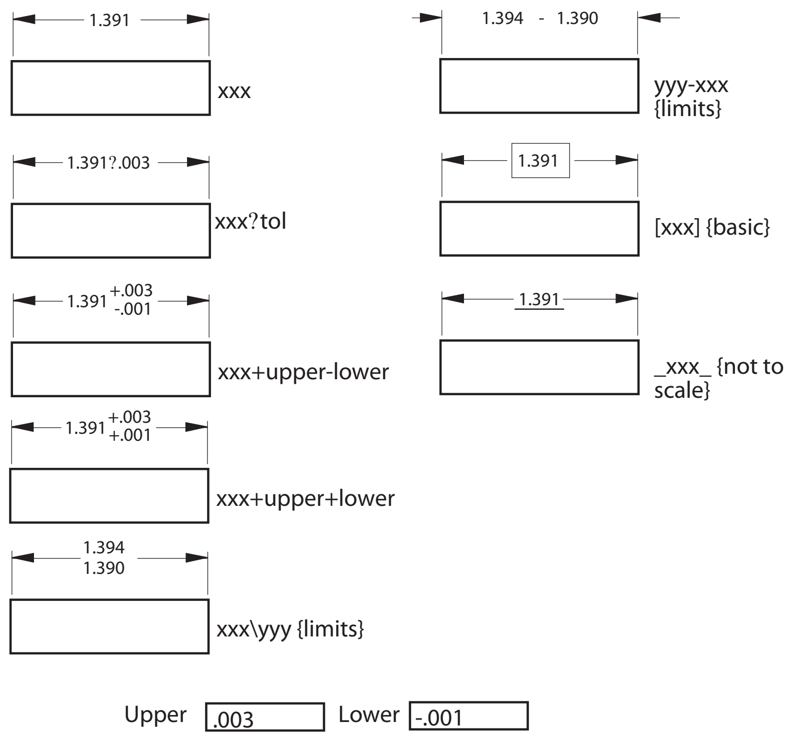
Cobalt, Xenon or Argon do not support dimensioning in fractions. The default is three decimal places.

Tech Note: *The number of decimals displayed with the dimensions is not affected by the number of decimals specified for units in Preferences.*

Linear Tolerance

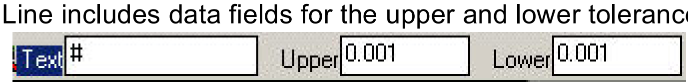
The Linear command in the Dimension menu sets the tolerance for the following **Linear** Dimension tools: **Horizontal Dimension** tool, **Vertical Dimension** tool, **Oblique Dimension** tool, **Angular Dimension** tool and **Smart Dimension** tool. Choose ***Dimension>Linear*** to select one of the seven tolerance options.

The following graphic shows the appearance of each tolerance option if the limits shown in the Status Line are entered.

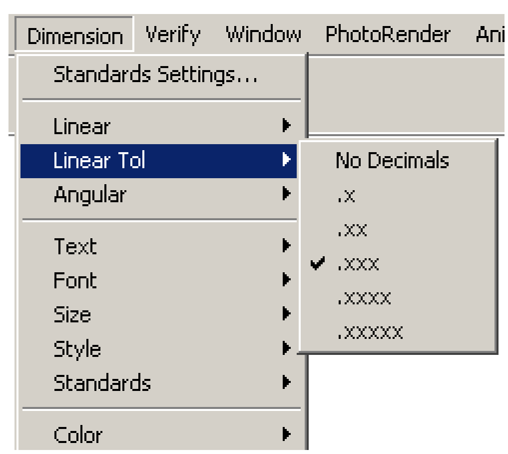


It is possible to use the Edit Objects box to override the + or - for either of these as needed.

When a **Dimension** tool is selected, such as the **Horizontal Dimension** tool, the Status Line includes data fields for the upper and lower tolerances, as shown here.



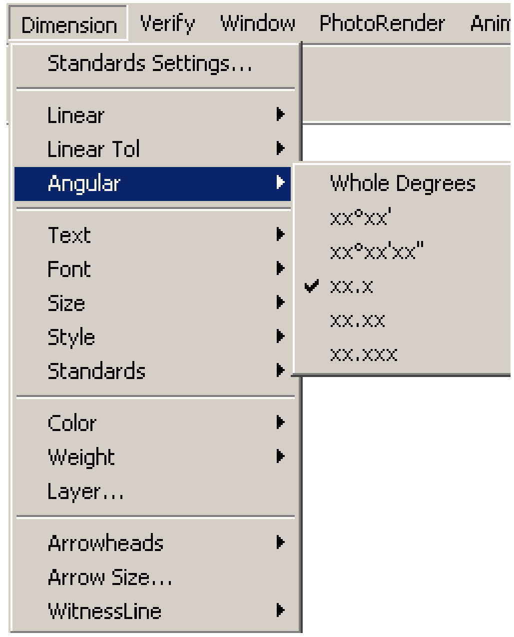
To set the number of decimal places for the linear tolerance, choose ***Dimension>Linear Tol***.



Angular

This command in the Dimension menu sets the format of Angular dimensions.

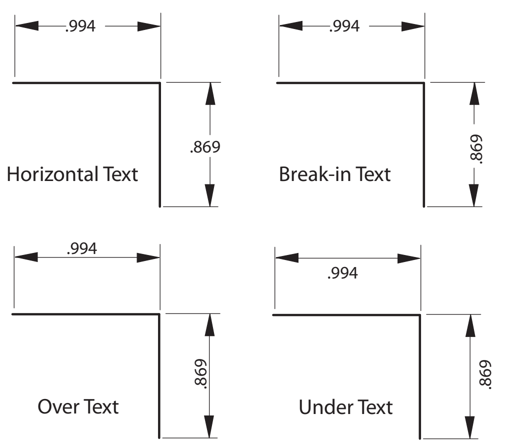
It is possible to set degrees, minutes and seconds for angular dimensions. The default format is degrees and minutes.

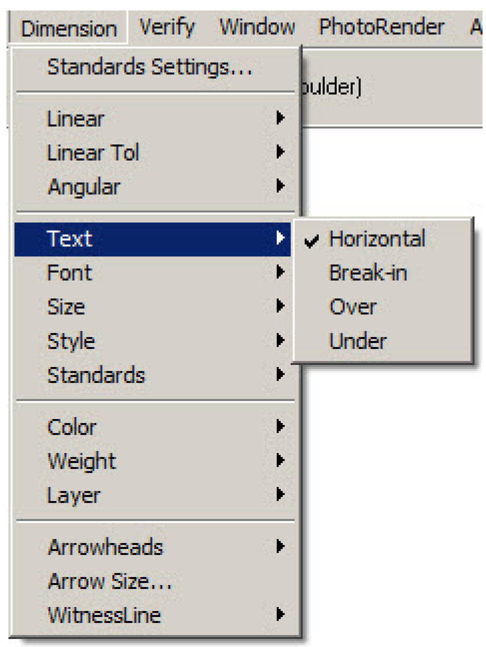


Text

This command in the Dimension menu specifies 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.

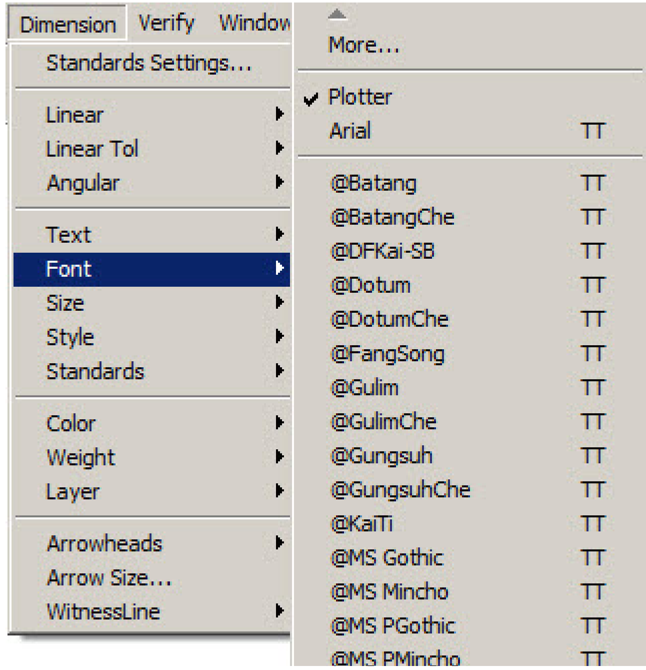




Font

This command in the Dimension menu picks the font for dimension text. The fonts in this list are the same as those available in the Font submenu from the Text menu.

The Font submenu in the Text menu has no effect on dimensions.



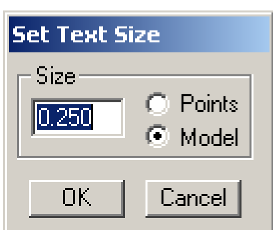
Tip: This command only affects dimensions.To change the font of other text objects, choose **Text>Font**.

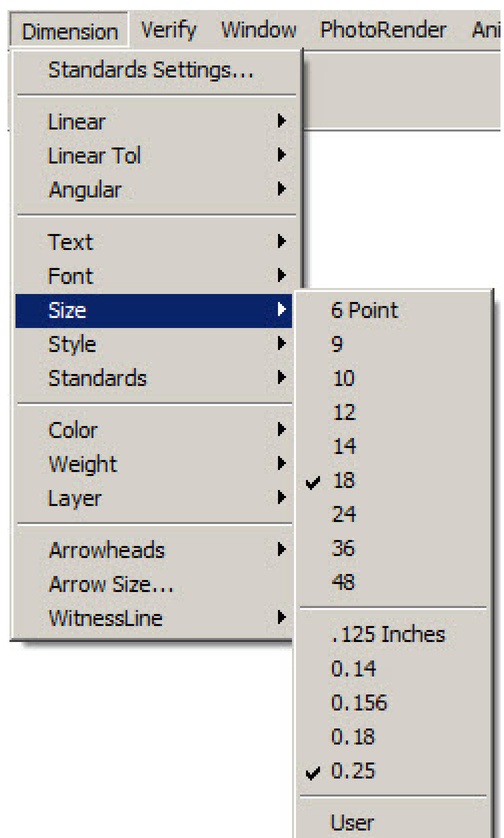
Size

This command in the Dimension menu picks 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.

The Size submenu from the Text menu has no effect on dimensions.

Selecting the User option specifies a non-standard font size for dimension text.



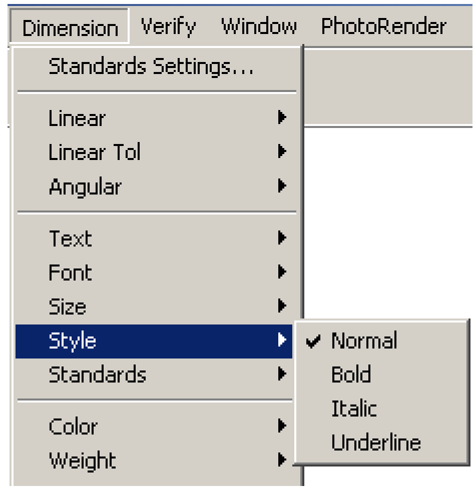


Selecting the Model option displays the text size in the Units specified in Preferences. Selecting the Points option, displays the point size of the text. Select the desired radio button, enter a value in the data field and click OK.

Tip: *This command only affects dimensions. To change the size of other text objects, use* Edit Objects *or the Size submenu from the Text menu.*

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.

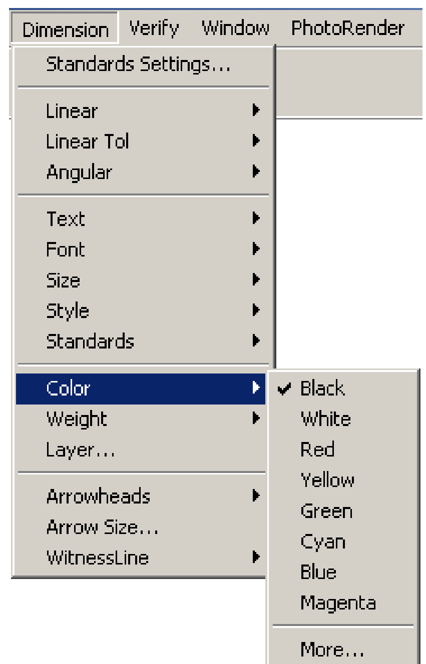


Tip: *This command only affects dimensions. To change the style of other text objects, use the Style submenu from the Text menu.*
The Style submenu from the Text menu has no effect on dimensions.

Color

This command in the Dimension menu specifies the color of the dimension. The default color is blue.

The Color submenu from the Pen menu has no effect on dimensions.

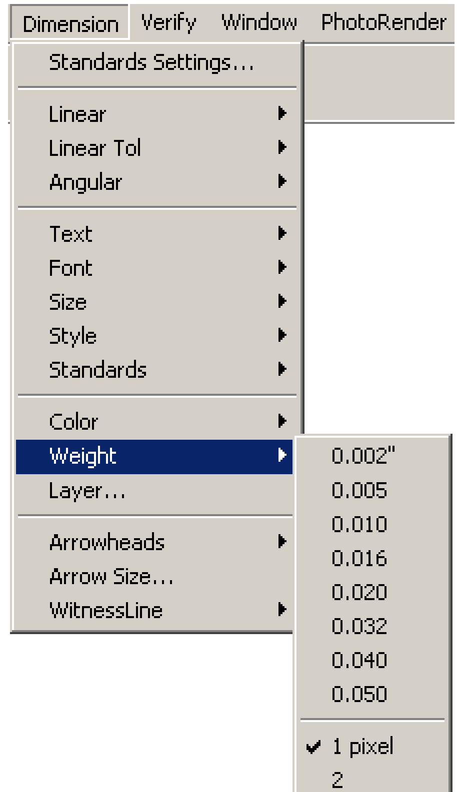


Tip: *This command only affects dimensions. To change the color of other objects, use Edit Objects or the Color submenu from the Pen menu.*

Weight

This command in the Dimension menu specifies 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.

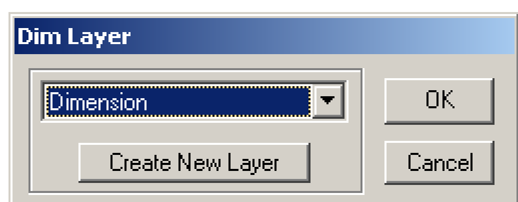


Tip: *This command only affects dimensions. To change the weight of other objects, use Edit Objects or the Weight submenu from the Pen menu. The Weight submenu from the Pen menu has no effect on dimensions.*

Layer

This command in the Dimension menu specifies 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 Drawing Views for information.

When selecting this command the following dialog box appears:



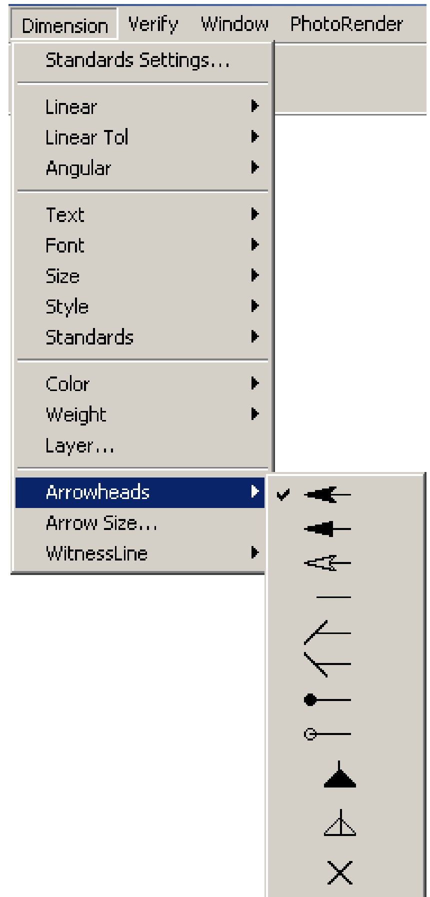
Dimensions can be placed on any visible layer. In the pull-down menu, select a layer for the dimensions. The default layer is the Dimension layer.

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.

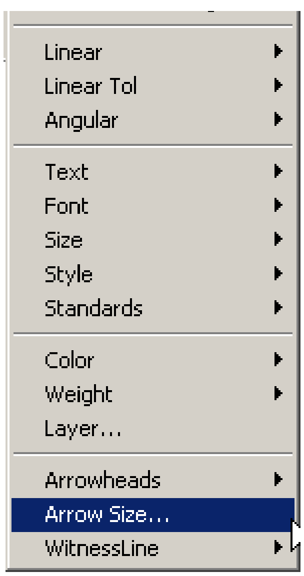
If the layer on which dimensions are to be placed is hidden or deleted, future dimensions will be placed on the current work layer.

Arrowheads

This command in the Dimension menu specifies the type of arrowhead used for dimensions.



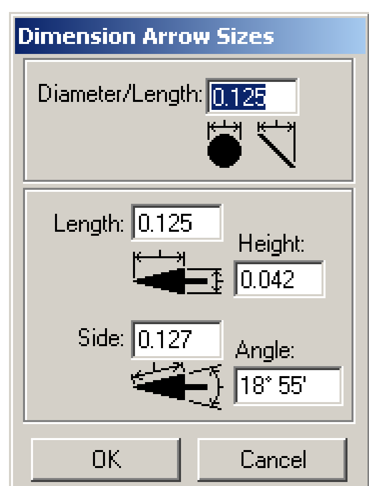
Tip: *This command only affects dimensions. To select the arrowhead type for lines or arcs, use the Arrowheads submenu from the Pen menu.*
The Arrowheads submenu from the Pen menu has no effect on dimensions.



Arrow Size Command

This command in the Dimension menu specifies the size of the selected arrowhead.

When selecting 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.

Changing any value in the Length, Height, Side, or Angle entry fields, the program will change the values in the other entry fields accordingly.

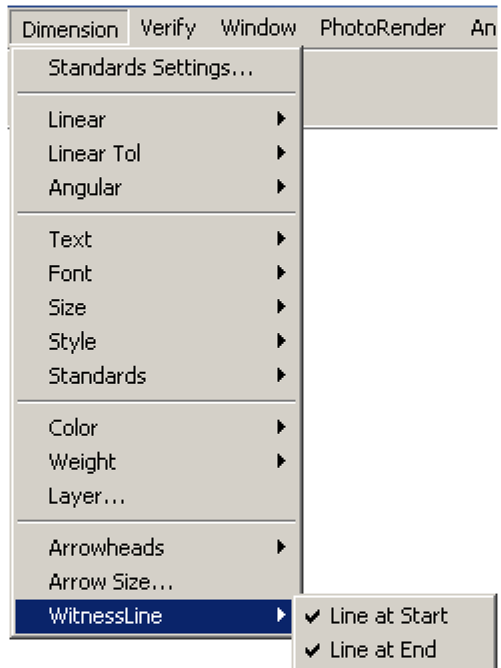
Witness Lines

This command in the Dimension menu lets specifies 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. Turn off one or more of the overlapping witness lines to 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

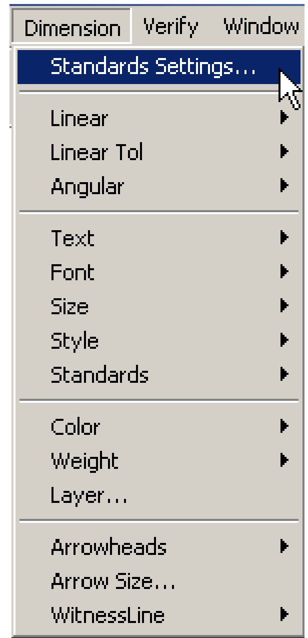
The dimension standards feature has been modified to manage and define the standards and specify which standard is used from the Standards submenu in the Dimension menu.

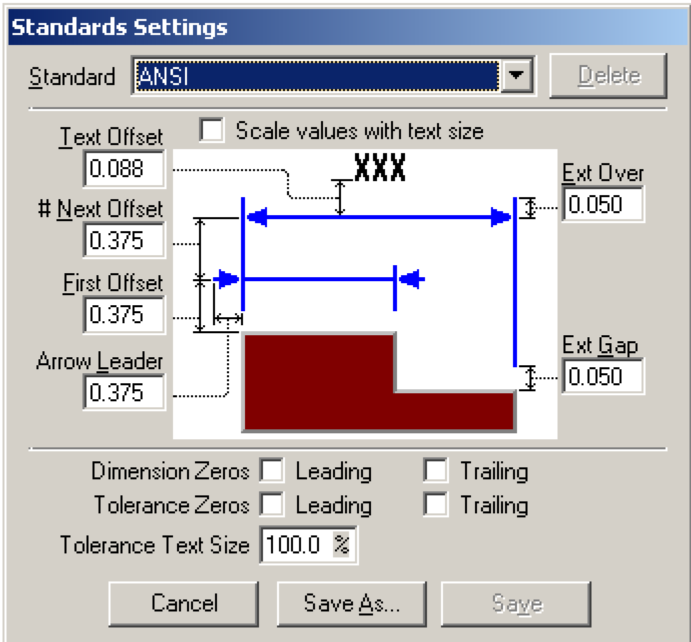
A user-defined standard contains all the current Dimension menu settings at the time the Standard Settings dialog box appears. This includes dimension text font, style and size. It does not include the Witness Line setting.

> ***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 sets the dimension standard. Cobalt, Xenon and Argon contain 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, user-defined standards can be created and saved 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.)

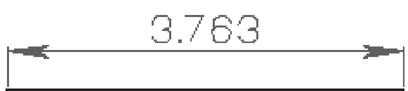
Cancel Click this button to close the dialog box without accepting any changes.

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

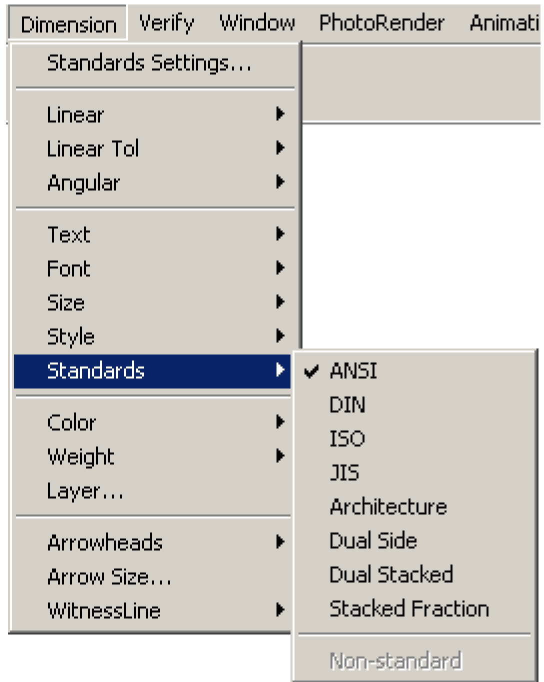
Dual Side Standard

Dual Stacked Standard

Stacked Fraction Standard

Setting the Standard

A Standards menu sets the standard as the default or change the standard for a selected dimension. If user-defined standards are created, they are also listed in the Standards menu.



Choosing a Standard

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, the default standard is set. If one or more dimensions are selected, the standard is changed for the selected dimensions.

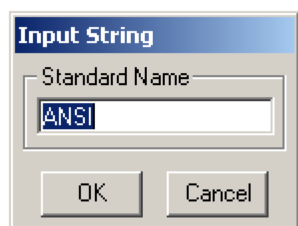
1. Choose ***Dimension>Standards***.
2. In the Standards submenu, choose one of the listed standards.
3. Click OK to accept the standard as the default and close the dialog box.

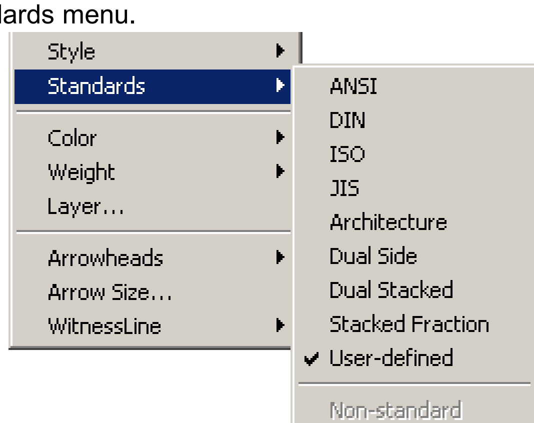
Creating a User-defined Standard

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.

   This user-defined standard can be later deleted 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 they are not seen, choose the Frame to Extent command in the Drawing View menu. Or drag the view edge manually to completely display the dimension. For information see Drawing Views.

Dimensions created in drawing views are placed on the Sheet View layer rather than the Dimension layer. This turns off all other layers including the Dimension layer when printing the sheet. This is helpful when dimensioning the original geometry and it is not necessary to print those dimensions.

Fill and Crosshatching

Cobalt Xenon and Argon apply fill to smart polygons and to apply crosshatching to the 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

Cobalt Xenon and Argon provide 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 highlight the polygons or separate overlapping polygons visually in the drawing. After creating the polygon, place a fill pattern and fill color in the polygon. Changes made 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.*

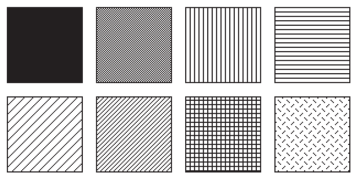
There are two fill commands in the Pen menu: *Fill Pattern* and *Fill Color*.

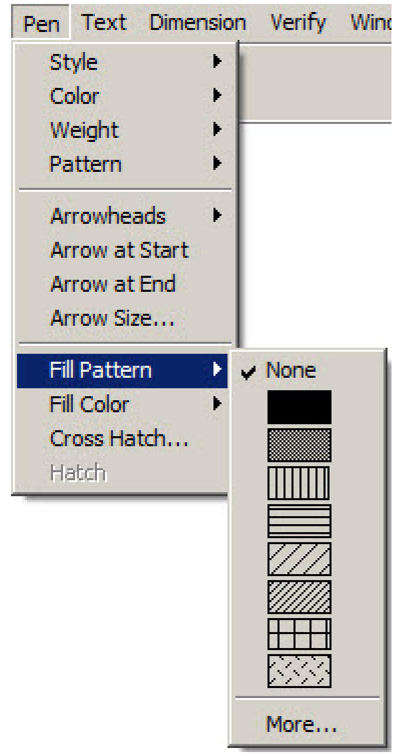
These commands work with each other. Using the *Fill Pattern* command, choose the pattern for the polygon. Using the *Fill Color* command, choose the pattern color.

Fill Pattern

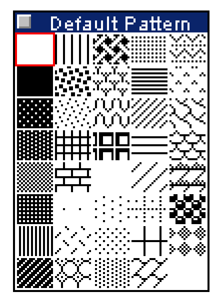
A wide variety fill patterns are available. Patterns appear parallel to the work plane and only in the view in which they were added. In other words, patterns are drawn correctly when viewed along the z-axis of the work plane.

When selecting the *Fill Pattern* command a submenu displays. There is the option to choose None (for no fill) or one of eight patterns (including solid fill) as shown here.

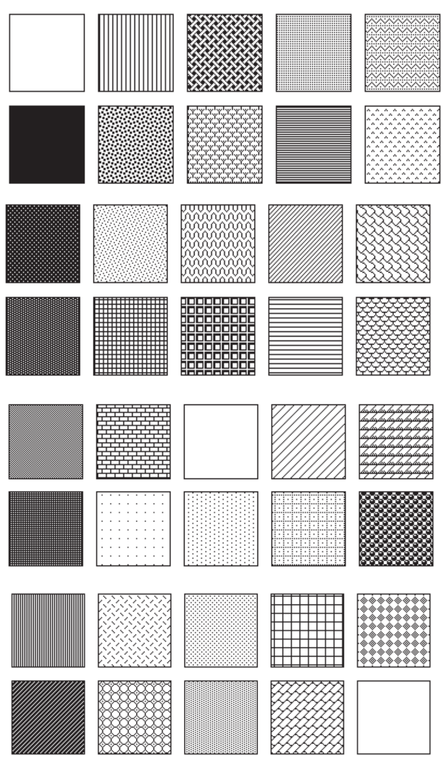




Selecting ***Pen>Fill Pattern>More***, the Patterns option box appears providing more patterns.

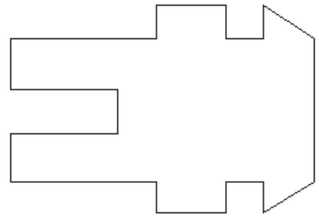


The patterns are enlarged below for clarity.



These patterns are not editable. They are predefined bitmaps. The default pattern is None.

Choosing a Fill Pattern

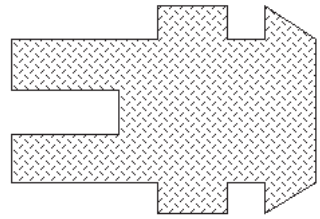


1. Choose the Selection tool.
2. Select the polygon to display with a fill pattern. Hold down the SHIFT key to select more than one polygon.

   One example is displayed here.
3. Choose ***Pen>Fill Pattern*** and one of the fill patterns.

   (Choose a fill pattern from the Patterns option box, for a selection box to appear around the pattern.)

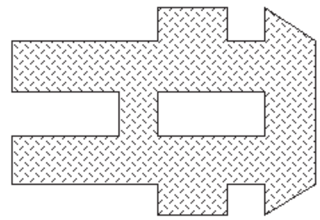
   The polygon is filled with the selected pattern.



Fill Patterns and Holes

Cobalt, Xenon and Argon do not support the placement of "holes" in polygons. A polygon, however, can be placed within another polygon and a solid white fill can be chosen to give the illusion of a hole.

Using the example above the following result is obtained.

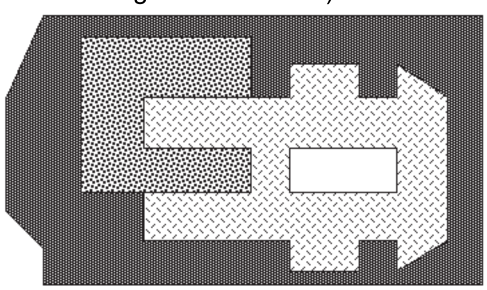


Overlapping Objects and Patterns

Since multiple objects with multiple patterns can be displayed in the drawing, some overlapping may occur.

The order of the overlapping is specified 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 Arrange Command on using this command).



Use the *Group* command in the Layout menu to keep the patterned objects in the specified order.

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 specified.

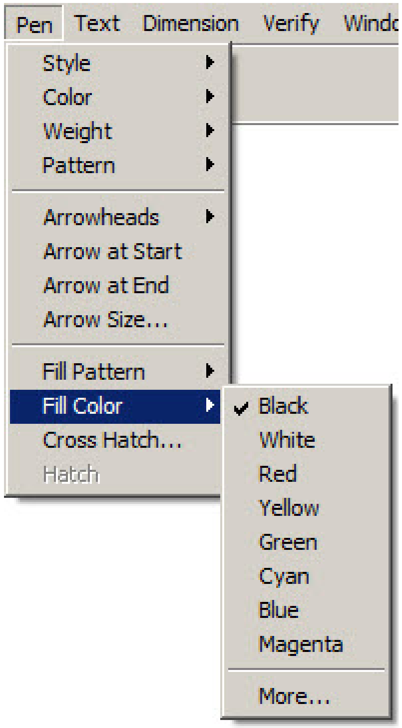
Removing the Fill Pattern

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.

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 Pen Settings).

A specific color can be chosen for all fill patterns. The default color is black.

Choosing a Fill Color

1. Choose the Selection tool.
2. Select the polygon to fill.

   Hold down the SHIFT key to select more than one polygon.
3. Select the fill color. The color displays.

   The new fill pattern color is displayed in the polygon.

It is not possible to 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 it is specified.

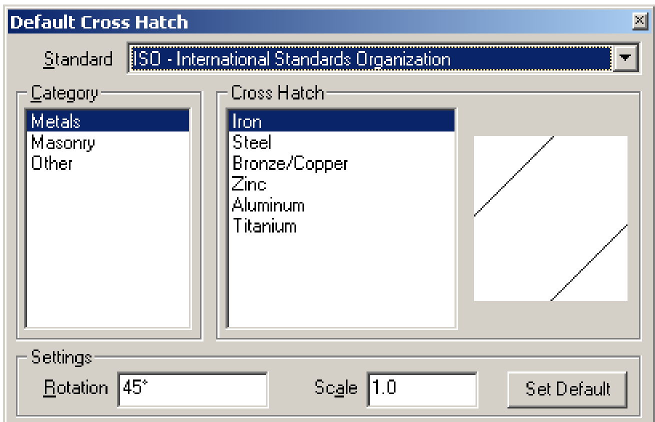
Crosshatching

This Designer Elements program can crosshatch any enclosed area in the drawing and then automatically update the crosshatching when the geometry is changed. Select a closed area containing a hole or cutout and the program accurately excludes the cutout area from the crosshatching. Crosshatching can also be applied to section cuts made through the 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 the geometry.

Cross Hatch Patterns

You have a wide variety of crosshatch patterns available to 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.

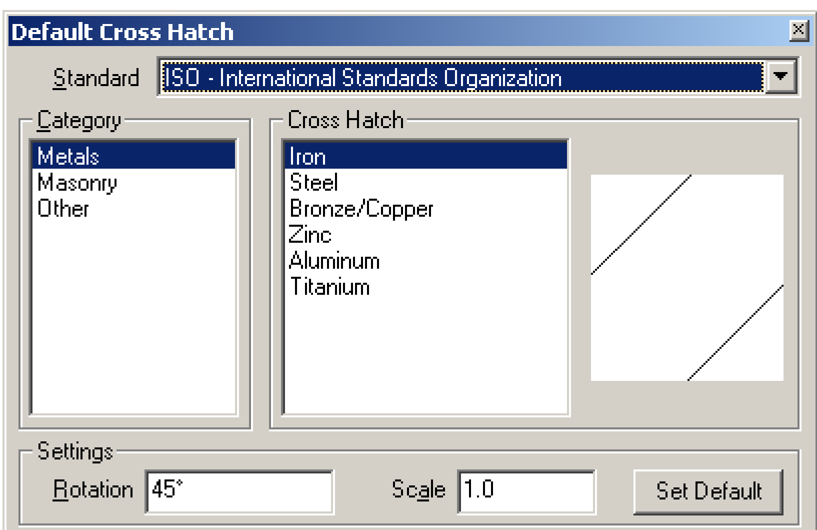
Scale (Settings) This field sets the scale of the crosshatch pattern.

Set Default (Default Cross Hatch dialog box) Click this button to set the default crosshatch pattern for the drawing.

Apply (Object Cross Hatch dialog box) Click this button to apply the crosshatch pattern to the selected object.

ISO Patterns

If the ISO standard is chosen, the ISO categories and crosshatch patterns display in the dialog box.



The categories for the ISO standard are:

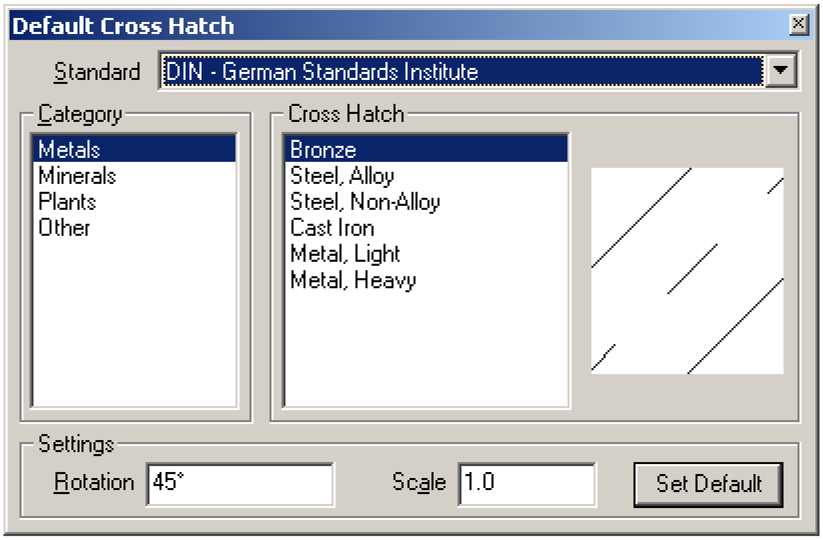
Metals The patterns in this category include: Iron, Steel, Bronze/ Copper, Zink, Aluminum and Titanium.

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 the DIN standard is chosen, 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).

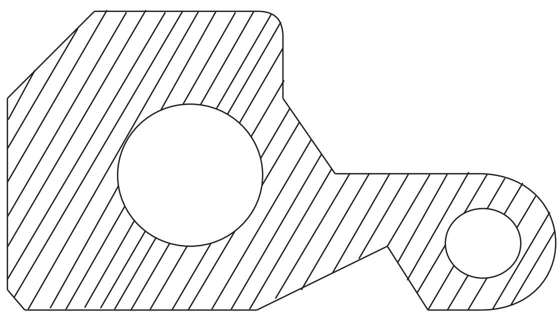
Minerals The patterns in this category include: Rock, Sand, Clay, Peat/ Humus Soil, Coal, Salt and Sandstone.

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 applies crosshatching to the geometry.



Setting the Default Hatch Patterns

When no objects are selected, 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 the *Hatch* command is chosen, in the Pen menu this default pattern is applied to the selected geometry.

Applying a Crosshatch Pattern for an Object

If one or more objects are selected, choose a specific crosshatching for them without changing the default.

1. Select the object to crosshatch.
2. To apply the default crosshatch, choose ***Pen>Hatch*** and the object is crosshatched.

   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 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. 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 to crosshatch.
2. Choose ***Pen>Hatch.***

   The selected object crosshatches with the default pattern.

Editing Hatching

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 use 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, it is possible to change the pattern, rotation and scale. In the same section, choose different hatch patterns for objects cut. See “Drawing Composition for Cobalt & Xenon” 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 the file size.

An instance is a copy of the original or master geometry. It is created when placing a symbol in the drawing or modifying that symbol by moving, rotating or scaling it.

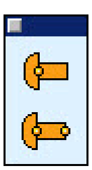
A symbol consists of a master symbol composed of geometry (text and dimensions are not supported) which is instanced into a drawing by clicking the placement location. Once a symbol is placed use the Transformation tools to move, rotate, scale or mirror it.

Symbols are created and organized through the Symbol Manager. Once the symbols are created use the Symbol tools to place them into the drawing.

The following topics are covered in this chapter:

- Symbol Palette
- Symbol Manager
- Creating Symbols
- Symbol Tools
- Symbol Example
- Transforming a Symbol
- Editing a Symbol
- Symbols and Rendering
- Symbols and Files

Symbol Palette

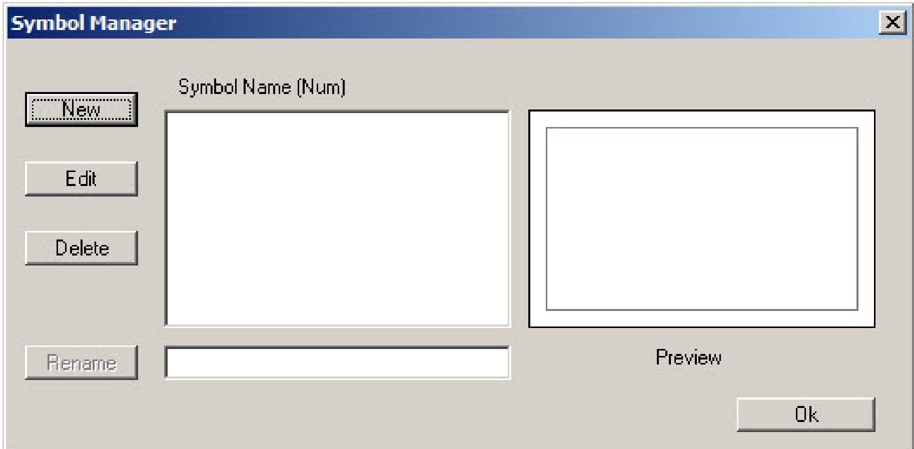


The symbol tools are contained in the Symbol tool palette. To display the palette choose ***Window>Symbols***.

The default status of the Symbol palette is closed and the default location is below the main tool palette. Like the other palettes, it is possible to save the status and location of the palette anywhere in the drawing area.

There are two tools for placing symbols into the drawing, Symbol 1 Point and Symbol 2 Point. Before using these tools, create the symbol through the Symbol Manager. Once the symbol is created, use one of these tools to place it into the drawing.

Symbol Manager



Use the Symbol Manager to create and organize symbols in the 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 — This lists the number of times a master symbol is instanced into the drawing.

Preview — The Preview panel displays a 2D preview of the selected symbol in window.

Rename — This renames any of the symbols in the drawing. When initially creating a symbol, Cobalt, Xenon or Argon automatically provide a name.

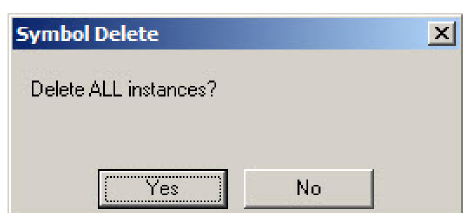
To change the symbol name, select the symbol name from the list and enter a new name in the *Rename* data field. The name updates as it is typed.

New — New opens the Edit Symbol drawing screen for creating a new symbol.

Edit — Edit opens the Edit Symbol drawing screen to edit a selected symbol in the *Symbol Name* list.

Delete — This deletes the selected symbol and all instances (copies) of the symbol in the drawing.

If there are instances of the symbol in the drawing when clicking this button, the following dialog box appears.

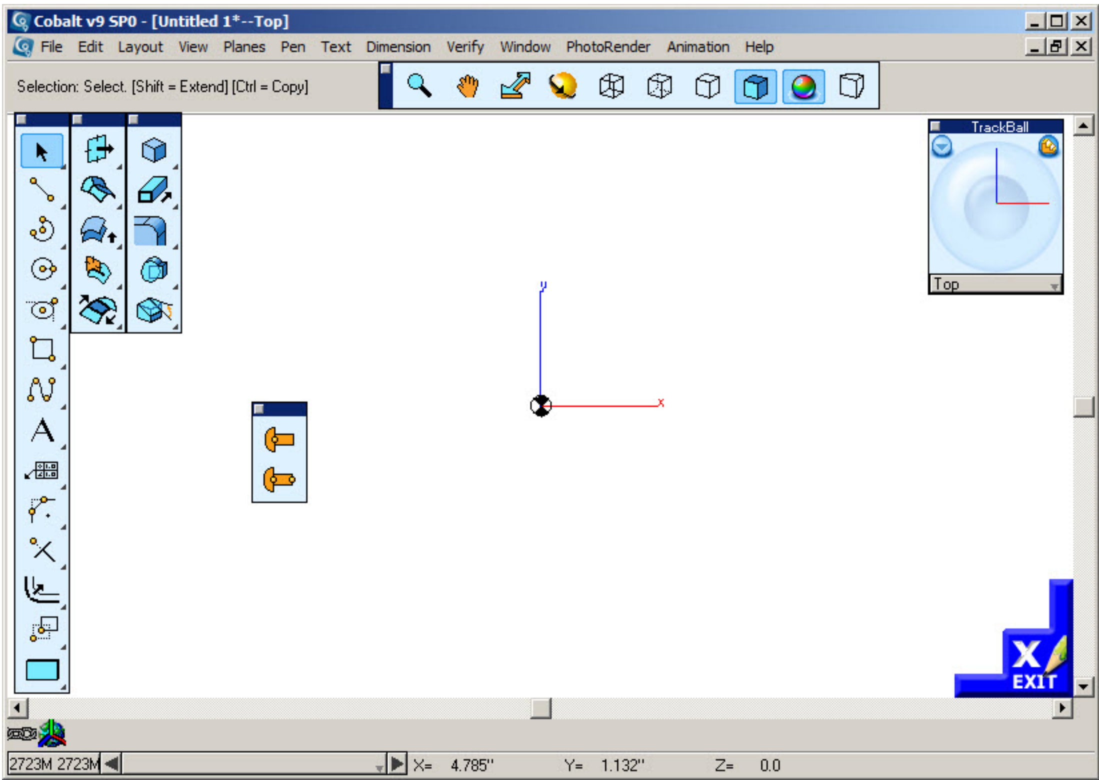


Click Yes to delete the symbol and all instances or No to close the box and return to the Symbol Manager.

Creating Symbols

Creating symbols is identical to creating any geometry. Symbols cannot contain text or dimensions. When creating a symbol, a master is created. Place copies or instances of this symbol into the 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.

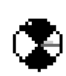


The screen is almost identical to the standard drawing area except for the EXIT Edit Symbol button and the Symbol Edit name in the Work Layer Indicator.

EXIT Edit Symbol — Click this button in the upper right corner of the drawing area when finishing creating the symbol geometry to return to the drawing area.

Symbol Edit — This name in the Work Layer Indicator designates 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 the symbol geometry using any of the tools and commands in the program.
5. When completing the symbol geometry, click the EXIT Edit Symbol button to return to the standard drawing area.

Follow these steps to create more symbols for the file.

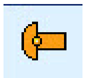
Like all other geometry, symbols are saved with the file. They do not become a part of a library for use with other files.

Cobalt, Xenon or Argon do not support text or dimensions with symbols.

Symbol Tools

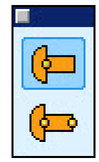
Two symbol tools are included for instancing symbols into the drawing, Symbol 1 Point and Symbol 2 Points.

Symbol 1 Point Tool



The Symbol 1 Point tool places a copy of the selected symbol at the specified location. The symbol x-axis is aligned with the x-axis of the work plane.

Using the Symbol 1 Point Tool



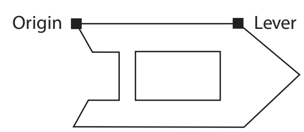
1. Select the Symbol 1 Point tool. The Message Line reads: *Symbol 1 Point: Enter location for symbol [Ctrl* (Windows) or *Option* (Macintosh) = *Symbol Manager]*.
2. Press the CTRL key (Windows) or OPTION key (Macintosh) to display the Symbol Manager.
3. Select the symbol to place in the drawing.
4. 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).

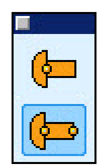
   The symbol is placed in the drawing. It is possible to place any amount of instances of the symbol.

Symbol 2 Points Tool

The Symbol 2 Points tool places a copy of the selected symbol at the origin and specified lever point. 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 to place in the drawing.
4. Click OK to close the Symbol Manager dialog box.

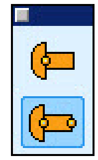
   The Message Line displays the name of the selected symbol.
5. Click the location for the symbol origin. The location is the symbol origin defined by the master symbol.
6. Click the lever point. The symbol is placed in the drawing.

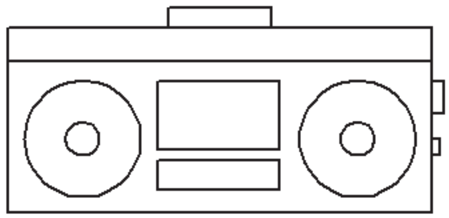
   It is possible to place any number of copies of the symbol in the drawing.

Symbol Example

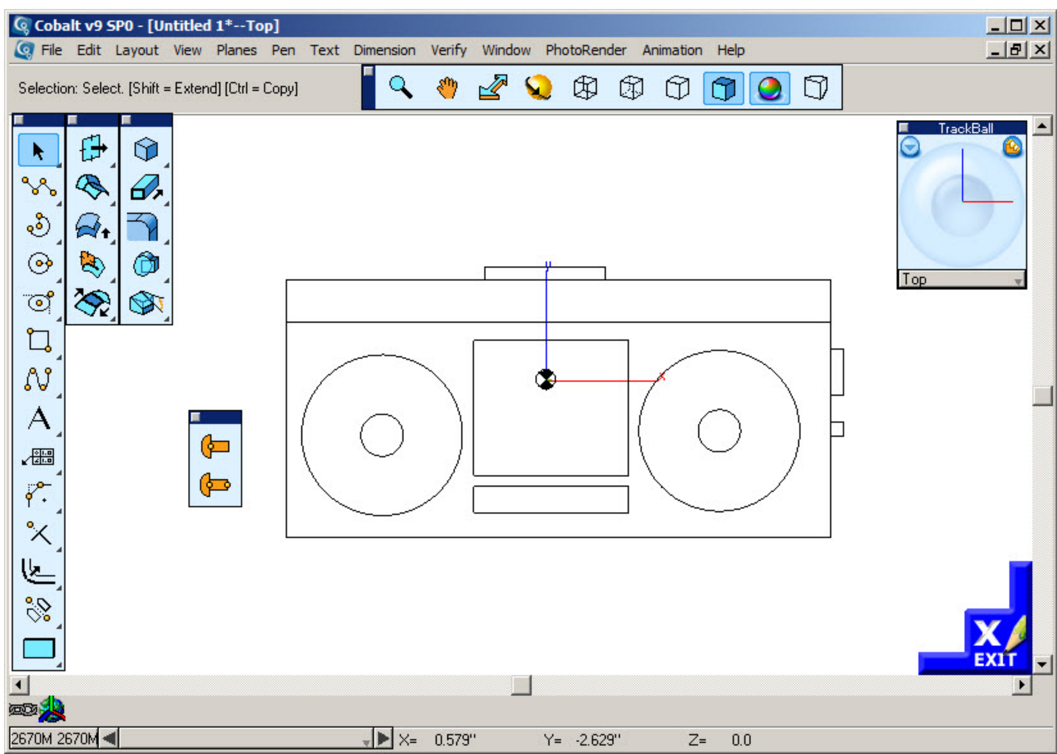
Try this simple example of creating and placing a symbol.

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.

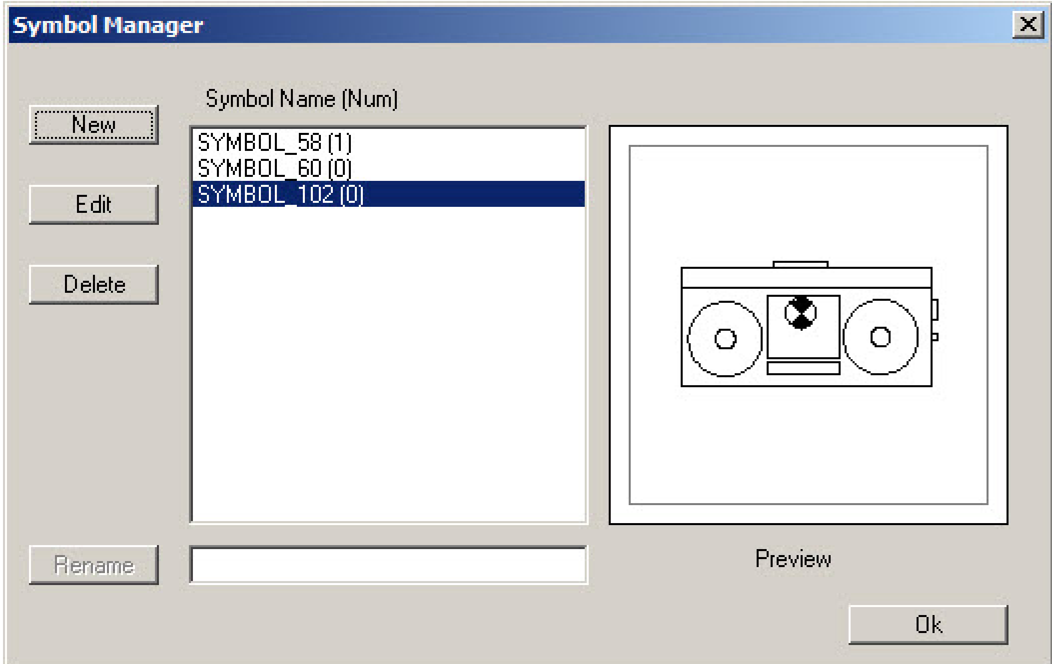




7. Once the symbol is completed, select the Symbol 2 Point tool and then click the EXIT Edit Symbol button in the lower right corner.



8. Press the CTRL key (Windows) or the OPTION key (Macintosh) to display the Symbol Manager.



   The symbol geometry is displayed in the *Preview* window with a default name in the *Symbol Name* list.

9. In the *Rename* field, which is already highlighted, type *BoomBox*. As it is typed, 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 the 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 the drawing. If a lever point is clicked 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 symbol was instanced once in the drawing.

    Using the Transformation tools it is possible now to perform numerous operations on the symbol.

Transforming a Symbol

Once a symbol is placed it is possible to use the Transformation tools to do some manipulation. Move, rotate, scale or mirror the symbols. Follow the steps for using these tools found in “Transforming Geometry” on page 1.

A symbol can be moved 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 the 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 to edit.
4. Click on the Edit button.

   The Symbol Edit screen displays.
5. Make the 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. To render a symbol, open the symbol through the Symbol Manager, copy the symbol and paste it into the drawing. Then the object, which is no longer a symbol, can be rendered.

Symbols and Files

Sometimes it is possible to have symbols 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 symbols for other files. It is possible, however, to copy and paste the geometry of one symbol opened in the Symbol Manager or one file into a new Symbol Edit screen of another file. This way it is not necessary to recreate the geometry.

Cobalt, Xenon and Argon do not support Symbol libraries.

Importing

Import drawing files into this program and then place that geometry within the Symbol Edit screen by copying and pasting. Make additions to the symbol in the Symbol Edit screen.

Cobalt, Xenon and Argon do not automatically support importing symbols from Graphite. Geometry from a symbol file can be imported into Cobalt, Xenon or Graphite copied and then pasted into the Symbol Edit screen as mentioned above.

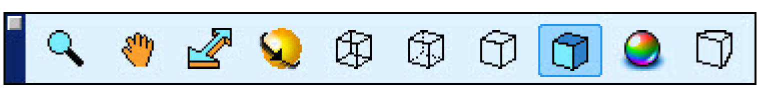
Viewing Geometry

In this Designer Elements program the geometry is created in one model space and can be viewed from many different angles and view scales as needed. A view describes the orientation of the eye position and direction toward the geometry.   A number of tools and commands are used to view the geometry. The following topics are covered:

- The View Palette
- Shade Options
- Zooming
- Panning a View
- View Displays
- View Rotation
- View Commands
- Views and Planes
- Views and Zoom Scale

The View Palette

The View palette of Cobalt, Xenon and Argon provides tools for various viewing options. They include the Zoom, Dynamic Pan, Dynamic Zoom, and Dynamic Rotate tools, all of which are described in detail further on in this chapter.



In addition to these tools, the palette also offers shortcuts to the following view options: Wireframe, Hidden Lines Dimmed, Hidden Lines Removed, Static Shade, Surface Analysis and Perspective.

Wireframe	Displays the model in wireframe.
Hidden Lines Dimmed	Displays the model in wireframe with hidden lines dimmed.
Hidden Lines Removed	Displays the model in wireframe with hidden lines removed.
Static Shade	Displays the model shaded with the last Static Shade setting designated in the Shade Options dialog box accessed by View>Shade Options, including Flat, Gouraud, Gouraud w/ Edges, Phong, Phong w/Edges.

Surface Analysis	Toggles the display of the real time surface analysis.
Perspective	Displays the model using perspective.

Shade Options

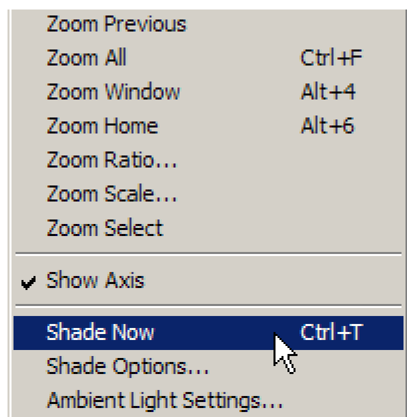
This Designer Elements program designs, displays and edits the models in a wide range of visualization modes. These include the static and dynamic options, wireframe, Flat shading, Gouraud shading, Phong shading, Hidden Line, Preview Render, Raytrace Render modes. Basic rendering is implemented under Open GL available through the video card. Do not 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 Model to Sheet Command for more information.*

Shade Now

CTRL+T (Windows); ⌘+T (Macintosh)

This *Shade Now* command in the View menu, displays the geometry using the *Render Now* setting selected in the Render Options dialog box (see the next section).



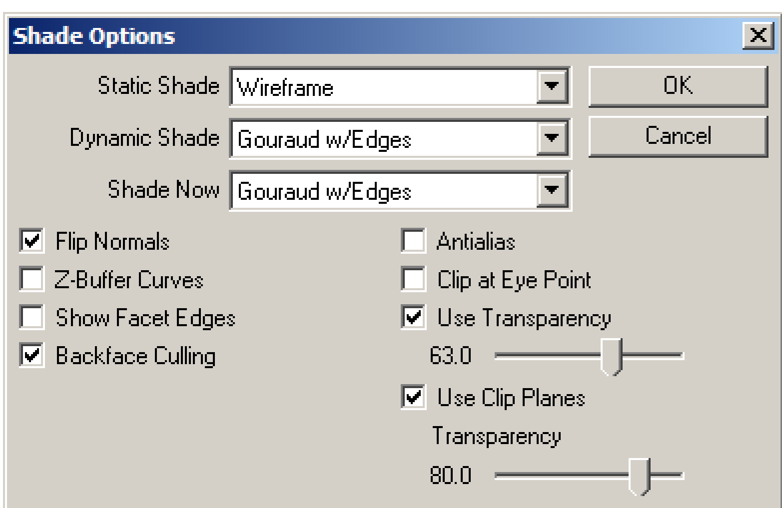
To use this command, choose ***View>Shade Now***.

All surface and solid geometry in the drawing renders.

Shade Options

This Designer Elements program provides a number of options for rendering the geometry.

When choosing ***View>Shade Options***, the following dialog box displays.



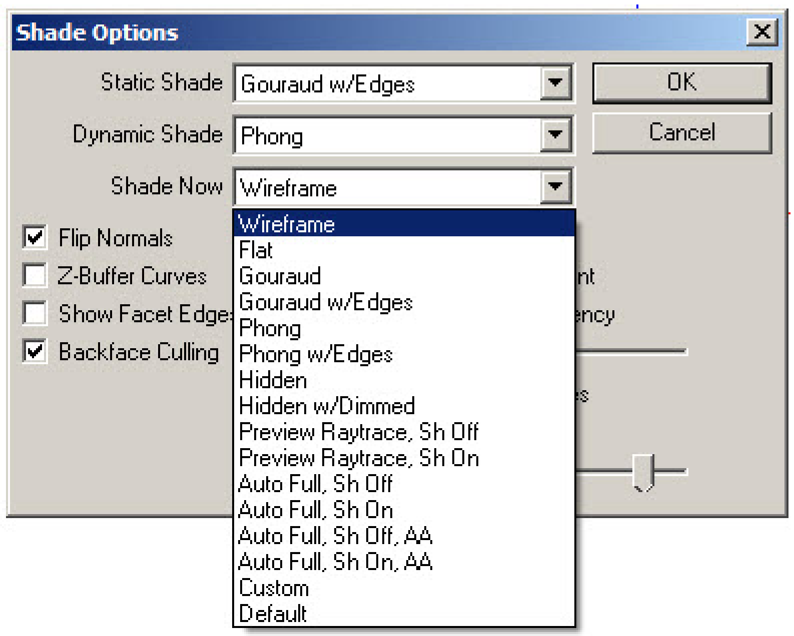
The dialog box includes the following rendering categories:

Static Render Defines how the geometry displays when the view is stationary. This Designer Elements program automatically repaints the object if this setting is changed and saved.

Dynamic Render Defines how the geometry displays as the view is rotated.

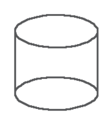
Render Now Defines how the geometry displays when choosing ***View>Shade Now***.

When clicking the arrow in a category pull-down menu appears.

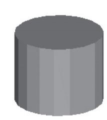


There are the following rendering options:

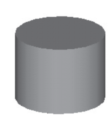
Wireframe Displays only the edges of the 3D model. This is the normal repaint display mode.



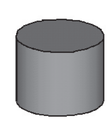
Flat Displays the geometry with a painter's algorithm using constant shading techniques.



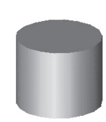
Gouraud Displays the geometry based on calculated light intensities at each vertex. This rendering method uses Open GL (Windows) or QuickDraw 3D (Macintosh).



Gouraud w/Edges Displays the model with face edge boundaries, silhouettes and isolines of a surface or solid on top of a rendered model. Specify the edge color in the *Display* page of Preferences (see Display).

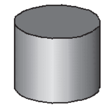


Phong Displays the geometry based on calculated light intensities at each pixel location.



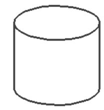
Phong w/Edges

Displays the model with face edge boundaries, silhouettes and isolines of a surface or solid on top of a rendered model. Specify the edge color in the *Display* page of Preferences (see Display).



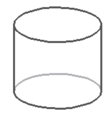
Hidden

Displays only the visible edges of the geometry. Since this is a rendering mode, any text in wireframe does not display. Text and wireframe objects do not display in this mode.



Hidden w/Dimmed

Displays the visible edges with the hidden edges of the geometry dimmed. Since this is a rendering mode, any text in wireframe does not display. Text and wireframe objects do not display in this mode.



Preview Render (shadows off)

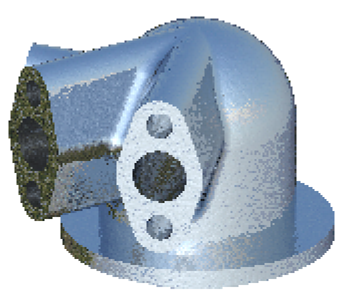
Creates a quality rendering of the scene. It renders most rapidly of all the commands since there are no ray tracing and shadow calculations.

Preview Render (shadows on)

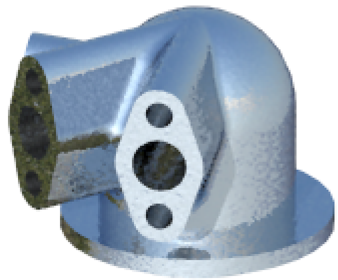
Creates a more realistic rendering than Preview Render (shadows off) with shadows. It renders quickly since there is no ray tracing.

Raytrace Render (shadows on/off)

Creates a high quality rendering of the scene. The processing time is extended with addition of the ray tracing operation and shadow calculations.



Raytrace Render (shadows on/off, anti-alias) Creates the highest quality rendering of the 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.



Custom Uses the custom render presets designated in the ***Advanced Settings>Photo-realistic Render Settings*** dialog window.

Default Uses the default factory render settings designated in the ***Advanced Settings>Photo-realistic Render Settings*** dialog window.

The dialog box also includes another render option setting:

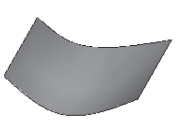
Flip Normals Flips the light normal on objects. The Flip Normals box is checked by default. This Designer Elements program automatically repaints the object if this setting is changed and saved.

When displaying 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 from another program.

Tech Note: *If the Dynamic Render mode is set to Phong with Edges, and the model vanishes, that's the result of light reflection. Adjust the light levels or location and render again.*

Flip Normal Example

If the following object is imported and rendered 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.

Choose ***Edit>Change Direction*** to flip the direction of the normals.

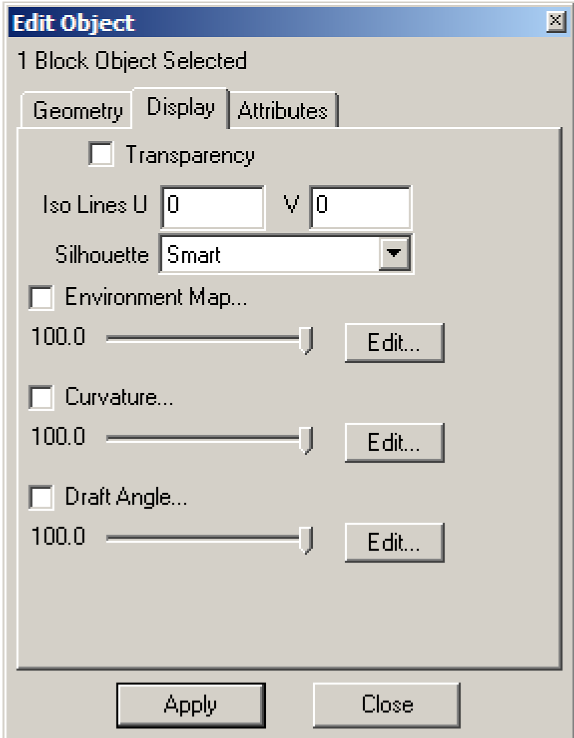
Setting the Shade Options

1. Choose ***View>Shade Options***. The dialog box displays.
2. Select the render options if necessary.
3. Click OK and the options are accepted. (Click Cancel to close the dialog box without saving the settings.)

If the *Flip Normal* or the *Static Render* settings are changed, this Designer Elements program automatically repaints.

These settings save as the default settings when exiting the program.

If Gouraud w/Edges or Phong w/ Edges are chosen and it is necessary to display the model with isolines or silhouette, double-click on the model to open the Edit Objects dialog box. In the *Display* page, specify the desired settings and click Apply.



Z-Buffer Curves Toggles 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 Toggles on or off the edges of the display facets. The facet density is determined by the entity resolution.

Use Clip Planes 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.

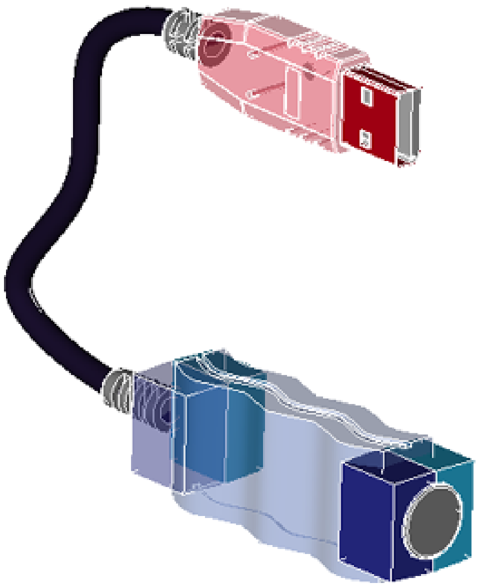
Backface Culling Enhances OpenGL performance. When this option is on, OpenGL ignores all facets from solids that have normals facing away from the viewing direction. Turn this feature off if there are hybrid solids that mix open surfaces with solids, otherwise the surface may appear invisible in OpenGL display modes.

Anti-Alias Tells the OpenGL drivers to enable anti-aliasing of edges.

Clip At Eye Point Toggles on or off clipping at the eye point. If off, clipping is automatically determined based on the extents of the given model.

Use Transparency 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

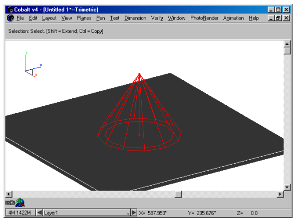
Because the rendering options deal with the display of the geometry, the rendering categories operate with each other and are affected by the view.

Shade Options Example

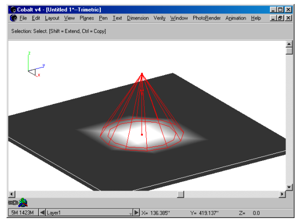
1. Create some geometry.
2. In the Shade Options dialog box, set *Render Now* to Hidden w/Dimmed and *Static Render* to Wireframe.
3. Click OK to save the setting and close the dialog box.
4. Choose ***View>Shade Now*** and the geometry is displayed in Hidden w/Dimmed view mode.
5. Rotate the view. The geometry is now displayed as wireframe because of the *Static Render* setting.

Lights & Facets

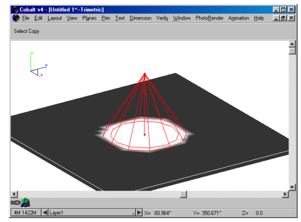
All facets are illuminated according to how much light falls on their vertices (the intersection of perpendicular isolines). If a light source is positioned over a large flat surface (such as a floor), it will appear that the source is not casting light. Set the object resolution to *Very Fine* or *Super Fine* to increase the facet density and enhance the lighting result. See “Change Resolution” on page 5 for more information on resolution.



Fine Resolution



Very Fine Resolution



Super Fine Resolution

Zooming

Cobalt, Xenon and Argon offer several ways to change the magnification of the 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 the geometry depending upon the selection and input. To zoom a particular area, use the *Stroke* feature or the Zoom tool, described later in this chapter.

Zoom All

CTRL+F (Windows); ⌘+F (Macintosh)

This command in the View menu zooms in or out to make all objects on the drawing fill the screen, regardless of the size of the objects.

Zoom In

CTRL+] (Windows); ⌘+] (Macintosh)

This command in the View menu zooms in to the screen center by a factor of two.

Zoom Out

CTRL+[(Windows); ⌘+[(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); ⌘+4 (Macintosh)

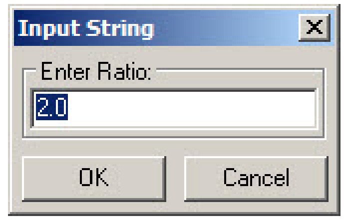
This command in the View menu drags a selection fence around the desired view window (from the upper left to the lower right).

Zoom Home

ALT+6 (Windows); ⌘+6(Macintosh)

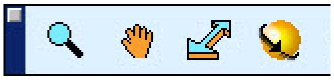
This command in the View menu adjusts the view scale so that the origin of the drawing (0, 0, 0) is centered on the screen.

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. The drawing scales to the value.

Zoom, Pan and Rotate Tools



The Zoom tools are located in the View tool palette.

Zoom Tool

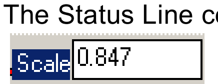
This tool zooms in or out by the specified factor. The default factor is two. It is possible also to specify a zoom scale (1 = full scale) in the Scale field of the Status Line. Zooming causes a visual rather than a physical change. Holding the CTRL (Windows) or Option (Mac) key changes the tool to zoom out.

> ***Tip:*** *Using the CTRL (Windows) or the OPTION (Macintosh) key, 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]*

   To get 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 or reduced by a factor of two. It is possible to drag a selection fence around an area so only that area is displayed.

The Status Line contains the Scale data field.

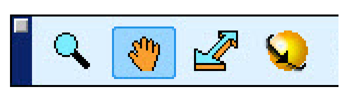


Pressing the CTRL (Windows) or the OPTION (Macintosh) key while using this tool causes it to change to the Zoom Out tool.

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.*

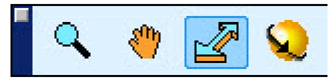
   The pointer becomes a hand icon. 
2. Place the pointer over the section of the screen to move and drag.

   When releasing 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 the stroke.

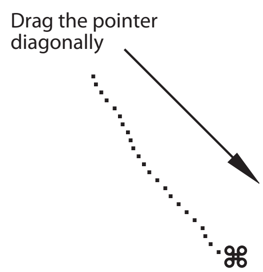
Stroke Zoom

Use stroke commands to zoom, magnify or reduce drawing. Stroke commands are useful because they don’t require you to get out of the tool being used in order to zoom. Hold down the SHIFT+CTRL keys (Windows) or the ⌘ key (Macintosh) and drag diagonally across the screen as described below. The pointer takes on the z shape when holding down the SHIFT+CTRL (Windows) or the ⌘ key (Macintosh) keys.

> ***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.*

This Designer Elements program remembers up to eight zoom strokes, enabling the return to previous zoom magnifications.

Using Stroke Zoom



Drag **Diagonally**	**Result**
Upper left to lower right	Zoom In: 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: 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, since all cases just give the previous magnification.

Zooming and Views

This Designer Elements program retains the last zoom scale that was set in a particular view. By zooming in a particular view, then changing views and zooming in the new view, the view scale in the previous view is not affected.

Panning a View

It is possible to move the view around to better see the geometry. Cobalt, Xenon and Argon provide the Dynamic Pan tool that enables doing this without using the scroll bars. Simply hold down the space bar, click on the geometry and drag to the desired location.

View Displays

There are two view types in this program: pre-defined and user-defined. Use these to set the view orientation of the geometry.

Pre-defined Views

Cobalt, Xenon and Argon provide five standard views: Side, Front, Top, Isometric and Trimetric. Each view is defined by Eye Point locations on the X, Y and Z axes or the Azimuth and Elevation. Changing the values in either Eye Point or Azimuth/Elevation automatically changes the other fields.

These values can be changed as desired. In ***File>Preferences>General***, 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 Point 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 Point 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 Point values are: X = 0.0, Y = -500.0, Z = 0.0. The Azimuth value is 0.0. The Elevation is 90.0.

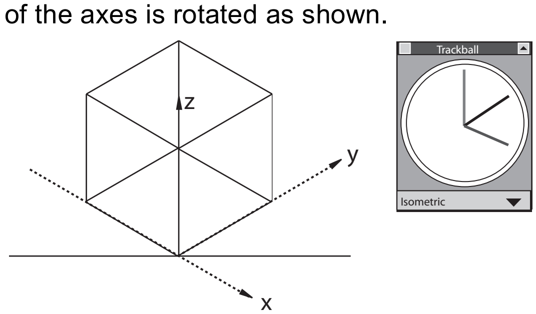
Aerospace: The view of the y, z plane. The Eye Point values are: X = -500.0, Y = 0.0, Z = 0.0. The Azimuth value is 0.0. The Elevation is 90.0.

Top

Default & Aerospace: The view of the x, y plane. The Eye Point values are: X = 0.0, Y = 0.0, Z = 500.0. The Azimuth value is 0.0. The Elevation is 0.0.

Isometric

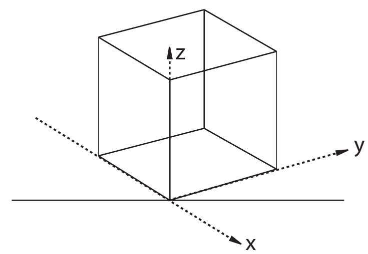
The view of the axes is rotated as shown.

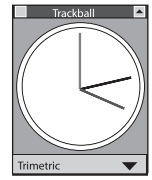


Default: The Eye Point values are: X = 500.0, Y = -500.0, Z = 500.0. The Azimuth value is -45.0. The Elevation is 54.736.

Aerospace: The Eye Point 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 Point 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 Point values are: X = -382.176, Y = -256.20, Z = 195.712. The Azimuth value is 33.837. The Elevation is 66.957.

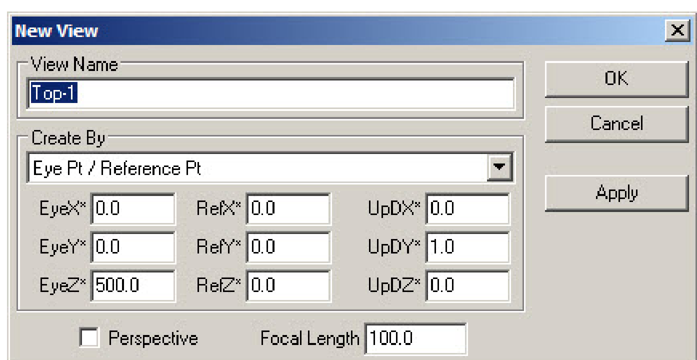
User-defined Views

There is the ability to define any new view using the *New View* command or the Trackball. There is also 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, specifies a new view based on a current view or entirely independent of any of the available views.



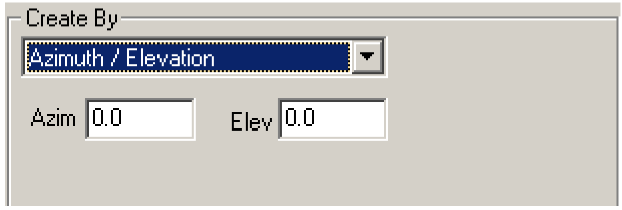
Choosing the *New View* command brings up the dialog box and contains the following options:

View Name Contains the name of the current view.

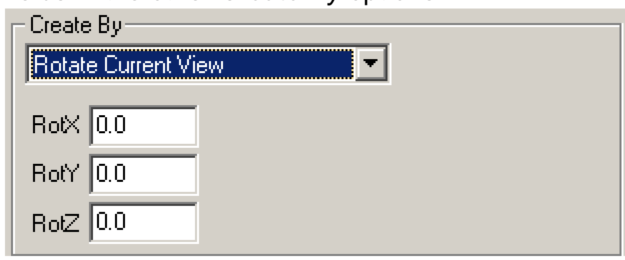
Create By Specifies the new view; *Eye Pt/Reference Pt, Azimuth/ Elevation* and *Rotate Current View.*

Eye Pt/Reference Pt - creates a view based on 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 the ability to click the location in the drawing area and have the values automatically entered in the fields.

Azimuth/Elevation - specifies the angle of the azimuth and the elevation of the eye with respect to the view.



Rotate Current View - rotates the current view by a specified amount. The values entered in these fields affect the related fields in the other *Create By* options.



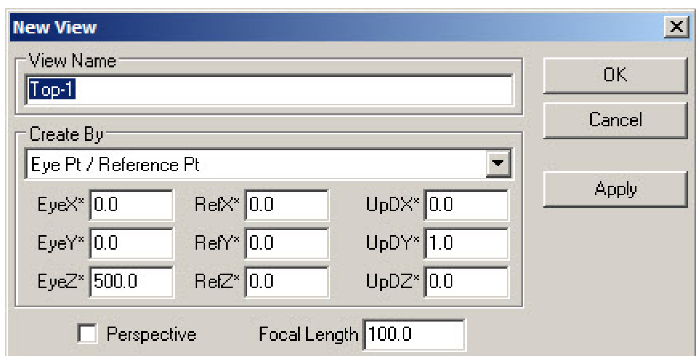
OK Saves the new view and closes the dialog box.

Cancel Closes 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 the view. Enter the appropriate values according to the *Create By* option chosen.

   For the *Eye Pt/Reference Pt* method, click the appropriate points in the drawing and the values will be entered automatically into the data fields.
4. Click OK. The dialog box closes and the new view is defined. (Click Cancel to close the dialog box without saving the view.)

Once a new view is created, select it by choosing ***View>User View*** and the view name or choosing the view in the Trackball pull-down menu.

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).

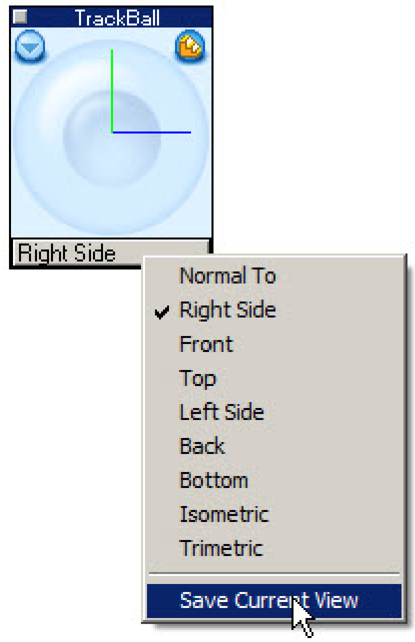
Trackball - Save Current View

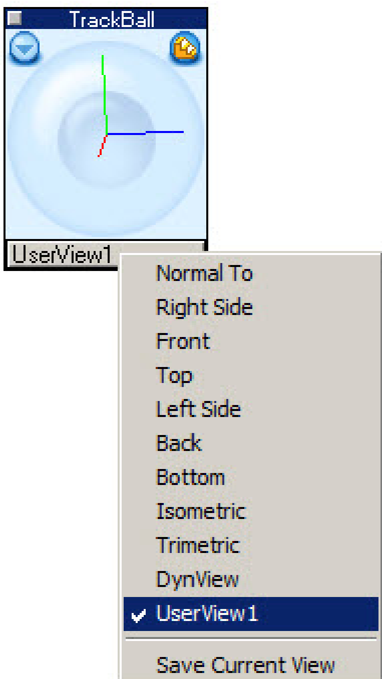
Define a view using the Trackball command, *Save Current View*, at the bottom of the Trackball pull-down menu.

> ***Tip:*** *When changing a view of the geometry but making no changes to the 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 the view with the file.*

1. Display the Trackball.
2. Rotate the 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 displaying the pull-down menu again, the new view is listed as *User View1*.





   See the addition of a *DynView*, which is the current non-standard view.

All views defined in this way are numbered sequentially. Rename these views by choosing ***View>Modify View***.

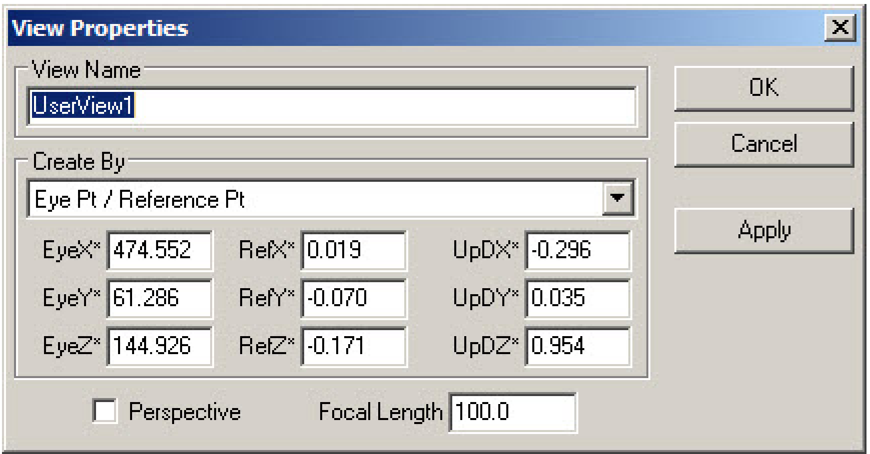
Modifying a View

It is possible to modify only user-defined views. A standard view cannot be modified. An attempt to do so creates a duplicate of the view except for the changes made. This new view becomes a user-defined view which can be renamed as desired. Choose ***View>User View***, to display the modified view.

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 to change. The Modify View dialog box displays.

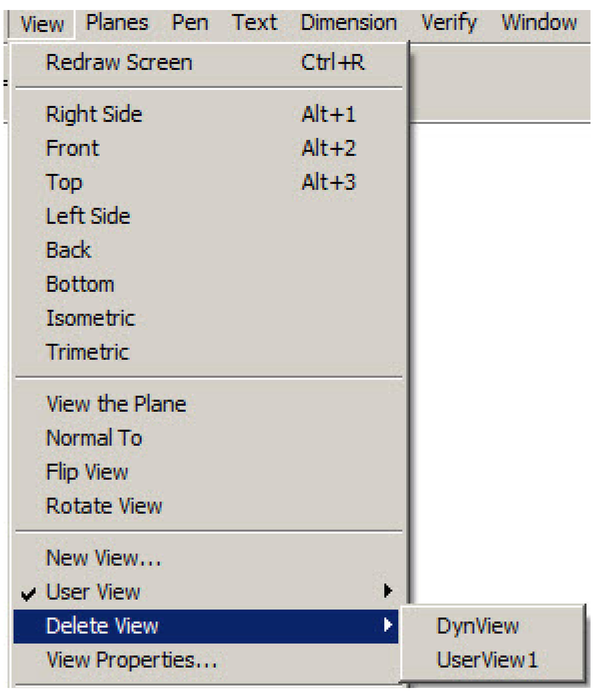


   This dialog box is identical to the New View dialog box except for its title.

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

Delete any DynViews or user-defined views using the *Delete* command in the View menu. Choose ***View>Delete View*** and the view to remove.



The current view or the standard views cannot be deleted. This command is unavailable if there are no user-defined views.

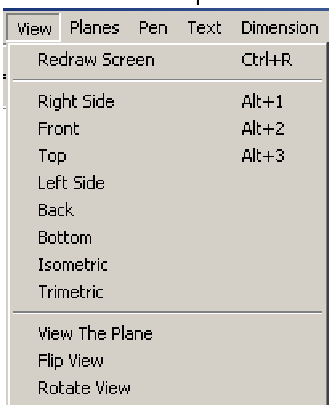
View Rotation

Views can be chosen 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

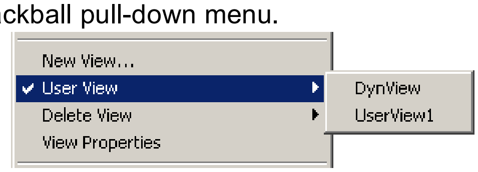
To use any of the five standard views mentioned earlier choose one in the View menu or in the Trackball pull-down menu.



See Trackball for more information on using it.

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

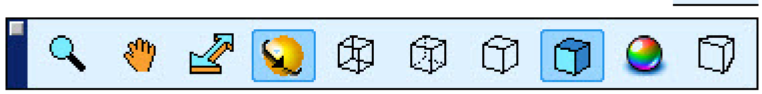
Choose undefined views by using the Trackball to rotate the view to a new orientation.

Notice that when rotating the view with this method the view name changes to DynView. This view has been added to the Trackball pull-down menu and is a temporary view that will change as the view is rotated using the Trackball. One advantage of this feature is that it is not necessary to define this view but it will still be available until the next Trackball rotation.

Dynamic Rotate Tool

The Dynamic Rotate tool, located in the View tool palette, rotates the 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 the cursor to change the view.

   or

   Enter values in the Status Line to rotate the view a specified angle.

   Press ENTER (Windows) or RETURN (Macintosh) and the view changes.

Using the Shift and Arrow keys

In any view hold down the SHIFT key and use the arrow keys to rotate the view.

View Commands

Redraw Screen

CTRL+R (Windows); ⌘+R (Macintosh)

This command in the View menu refreshes the screen. Sometimes when making changes to the 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: 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: press ESC or the ⌘ (Command) key to stop the redrawing of the screen.

View the Plane

This command located in the View menu, changes the view to the current work plane.

Show/Hide

This command in the Window menu helps manage the display of objects in the drawing.

Choose which objects display at a given time. When choosing ***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 the drawing.

Using the Invert Command

Select on the *Invert* command to display all objects currently hidden and hide all objects currently displayed.

Using the Show Only Command

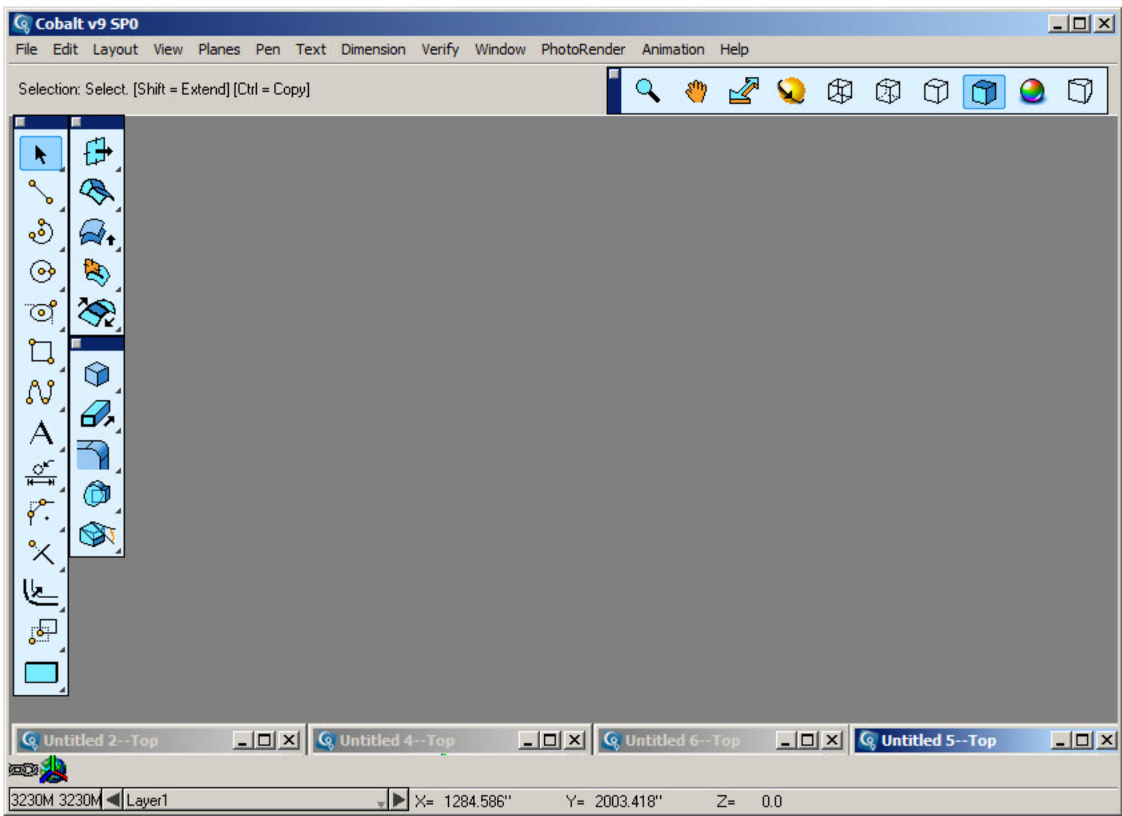
1. Select the *Show Only* command. The Message Line reads: *Select entities to show only. [Shift = Extend]*
2. Select the object to display. Hold down the SHIFT key to select more than one object.

   All other objects in the drawing are hidden.

Tile

In some cases, multiple files are open at the same time. Rather than having to switch back and forth between them, they may be displayed all at once. In the Windows menu choose *Tile Vertically* or *Tile Horizontally* (Windows) or Tile (Macintosh).

Arrange Icons (Windows only)



On occasion many open files may be reduced to their title bars but arranged haphazardly around the screen.

Choose ***Window>Arrange Icons*** and the files will be neatly arranged in the lower left corner of the screen.

Open File Windows

The bottom of the Window menu shows the names of all the open program files. To bring a different document to the top choose it from the list.

Views and Planes

To view geometry, choose a particular view orientation as discussed in this chapter. To create geometry, choose a particular plane on which to draw.

To eliminate the potential confusion between work planes and view, the Front, Side and Top views are tied to their respective planes. For example, choose the Top view, the Top plane is selected in the Plane menu and all other planes are grayed out. For more information see Planes.

Views and Zoom Scale

Cobalt, Xenon and Argon link the view to the zoom scale. Each view remembers the last zoom scale set in that view. 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

Think of visible layers as transparent pages and hidden layers as invisible pages. Use layers to show and hide various components of the drawing. They are particularly useful in helping to view and print complex drawings. For example, when dimensioning a part, the dimensions can be placed on a separate layer which can be displayed or not, as required. Layers enable printing different versions of the same document.

- Hide the dimension layer to present a design to a planning team and show the dimensions when presenting the drawing to engineers.
- Hide certain drawing components when printing or plotting. For example, hide the construction layer so that construction lines and geometry don't print but remain in the drawing ready for further design changes.
- Construct different layouts using one layer as the basis. For example, use one layer to define the basic shape of an object and then use other layers to try different methods of detailing.

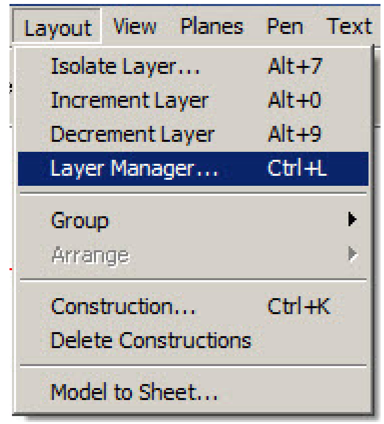
Cobalt, Xenon and Argon each provide up to *32,700* layers in the drawing. Layers must be visible to select objects on them. Layers do not have an orientation or origin.

The topics explained in this chapter include:

- Layer Manager
- Creating New Layers
- Creating New Layers
- Renaming Layers
- Deleting Layers
- Hiding Layers
- Showing Layers
- Locking Layers
- Unlocking Layers
- Making a Layer the Active Work Layer
- Layers and Pen Color
- Layers and Pen Style
- Layers and Copying/Pasting Objects

Layer Manager

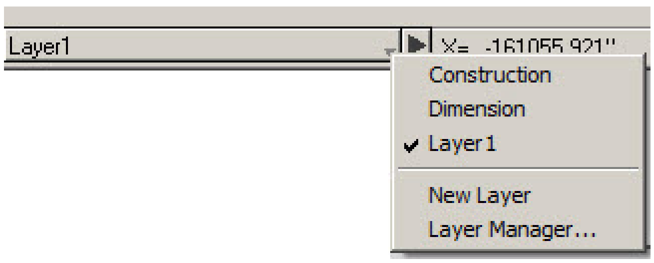
CTRL+L (Windows); ⌘+L (Macintosh)



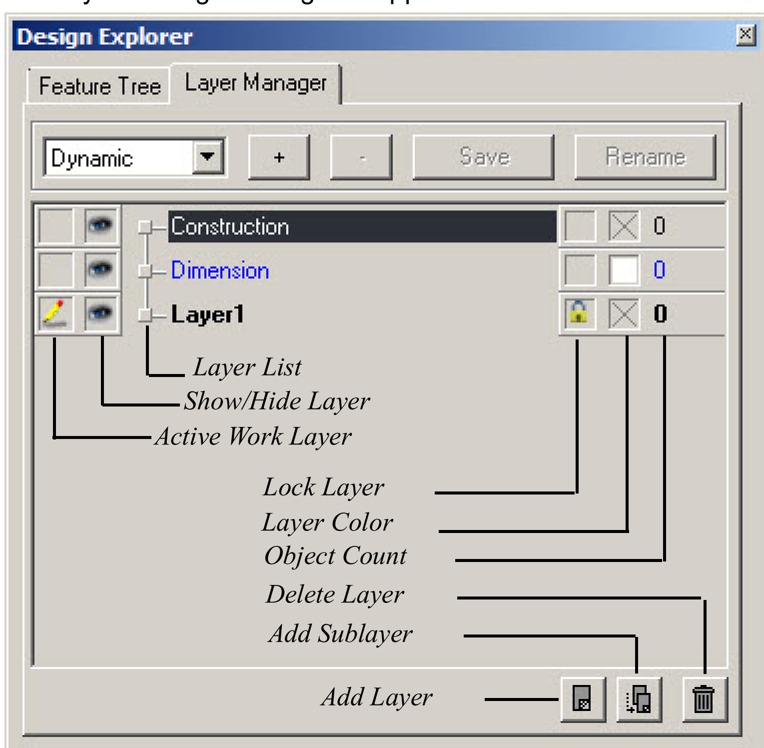
The Layer Manager creates, deletes, hides, displays and renames layers, as well as sets layer specifications. The work layer is the active layer on which geometry is created. There are three ways to open the Layer Manager.

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 Sets the active work layer. To change the active work layer move the pencil icon to the layer to be added to geometry.

Layer List Displays all layers in the file.

Show/Hide Layer Shows whether a layer is visible or hidden. The active work layer cannot be hidden.

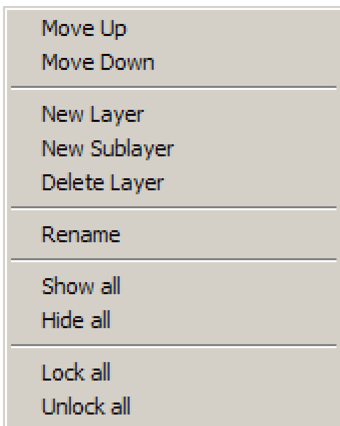
If the Show Layer icon displays, a layer is visible. 

Layer Lock If this column contains a Lock icon next to a layer, the layer is locked. Objects on this layer cannot be selected or modified. 

Layer Color 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 Displays the number of objects on the layer.

By right clicking on a layer the following menu pops up:



Move Up Moves the selected layer up one position in the dialog box.

Move Down Moves the selected layer down one position in the dialog box.

New Layer Adds a 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 Adds a sublayer to the selected layer. These layers are controlled 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 Deletes the selected layer. The active work layer cannot be deleted.This option also has a button at the bottom of the Layer Manager dialog box. 

Rename Renames the highlighted layer. Also it is possible to click twice on a layer name, once to select the layer and the second time to rename it.

Show All Shows all layers.

Hide All Hides all layers except the active work layer.

Lock All Locks all layers including the active work layer.

Unlock All Unlocks all layers.

Default Layers

When opening a new drawing the default layers include: *Construction*, *Dimension* and *Layer 1*.

Construction — Automatically accepts all Construction lines created with stroke commands or with the Construction dialog box. If this layer is accidentally deleted, it automatically recreates when a construction line is created.

Dimension — Automatically accepts dimensions on the layer selected from the Layer list of the Dimension menu. The default layer is the Dimension layer. Generally, this layer should be reserved for dimensions.

Layer 1 — 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.

Tech Note: *Dimensions placed in drawing views go on the Sheet View layer. See Drawing Composition for Cobalt & Xenon for more information.*

Any geometry or text that is visible can be edited regardless of its layer. To make some geometry unselectable but still visible use the *Selection Mask* in the Window menu or lock the layer containing the geometry. Also the *Selection Mask* specifies the selectability of layers.

Creating New Layers

There are three ways to create a new layer: using the Layer Manager, the Work Layer Indicator or the *Isolate Layer* command.

Using the Layer Manager

1. Display the Layer Manager dialog box.
2. Click the new layer button. A new layer is added to the layer list.

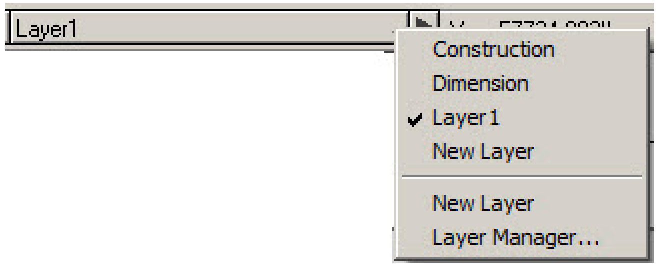
   

   Name the layer by clicking twice on the layer name, once to select the layer and the second time to rename it. Cobalt, Xenon and Argon each support up to 32,700 layers.
3. The new layer is added.

Tip: *If the Selection Mask is open and a new layer is added, the Selection Mask automatically updates.*

Using the Work Layer Indicator

1. Click on the Work Layer Indicator to display the pop-up menu.



2. Choose the *New Layer* command.

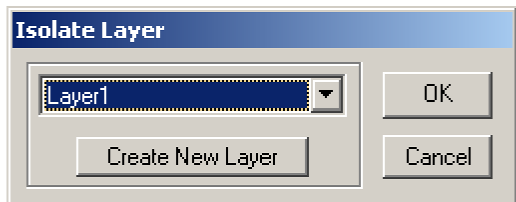
   A new layer is created and automatically set as the active work layer.

Using the Isolate Layer Command

Alt+7 (Windows); ⌘+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.

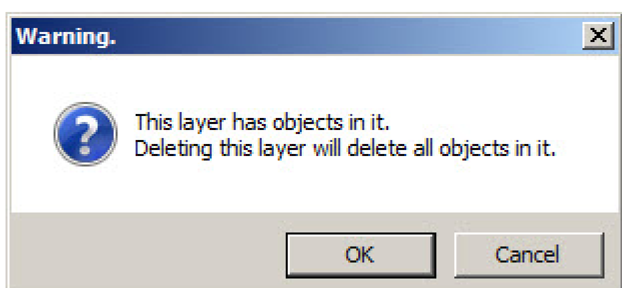
Renaming Layers

1. Display the Layer Manager dialog box.
2. Click twice on the layer 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.

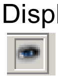


Hiding Layers

Hide one layer at a time or all layers but the active work layer.

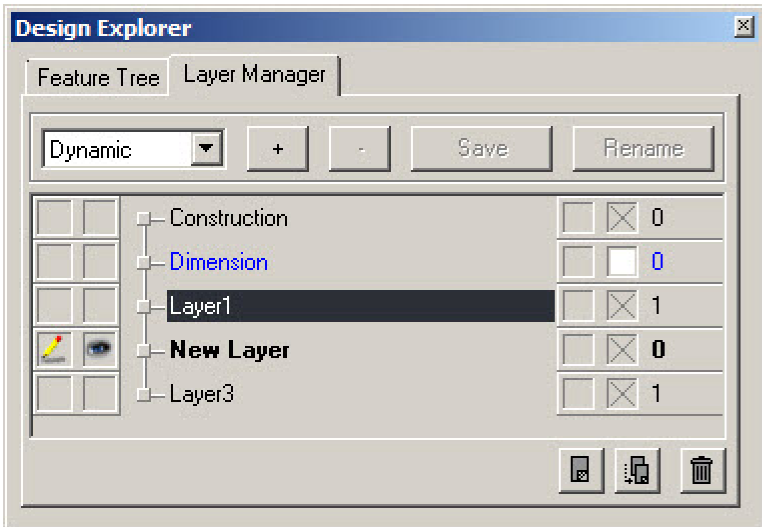
Hiding One Layer

1. Display the Layer Manager dialog box.



2. Click on the show icon (the eye) to the right of the layer name to turn hide.

The Show 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. Close the Layer Manager dialog box.

Notes:

- The active work layer cannot be hidden.
- If a layer is hidden and *Select All* and delete are chosen, the objects on the hidden layer are not deleted.

Showing Layers

It is possible to turn on one or all layers in the drawing.

Showing One Layer

1. Display the Layer Manager dialog box.
2. Click the box to the left of the layer name to turn on Show.

   A Show 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

There are 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.

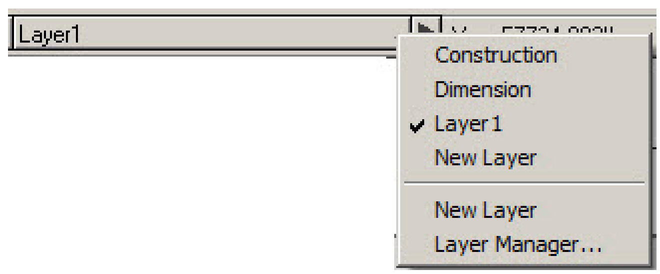
Using the Layer Manager

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 to make the active work layer.

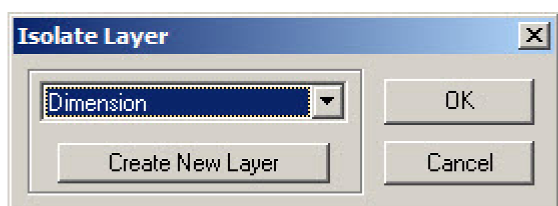
   The selected layer becomes the active work layer.

Using the Isolate Layer Command

Alt+7 (Windows); ⌘+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); ⌘+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. This command cannot be used if the active work layer is the last layer in the list.

Decrement Layer

ALT+9 (Windows); ⌘+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. This command cannot be used if the current work layer is the first layer in the list.

Layers and Pen Color

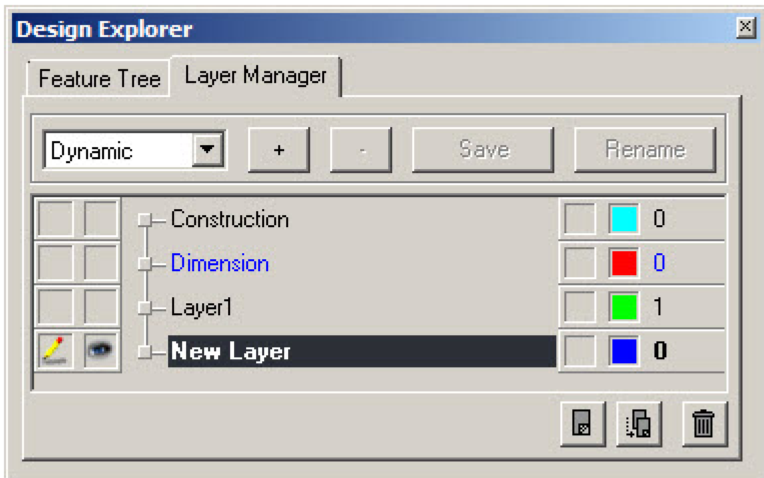
It is possible to override the color for all objects on a layer.

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

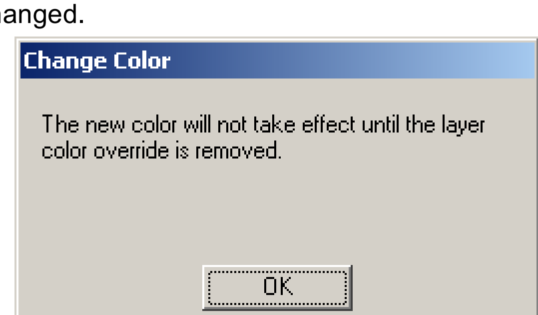
1. Display the Layer Manager tab of the Design Explorer dialog box if it is not already displayed.
2. Click in the *Layer Color* column next to the desired layer. A color patch appears displaying the first color available, white.



3. Click on the color patch to advance to the next patch color.
4. For a list of available colors, to designate a custom color, or to remove colors right click (or CTRL+click on a single button mouse) on the column to display the color menu.
5. Click OK to close the dialog box and save the change.

Layer Color Warning

The program displays a warning if the color of an object on a layer with a specified color is attempted to be changed.



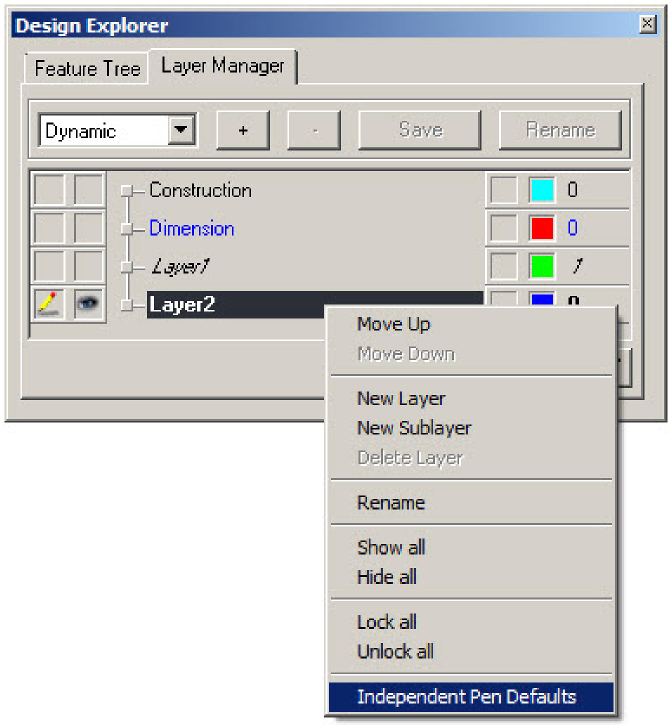
As the box indicates, the color cannot be changed unless the color override for that layer is removed.

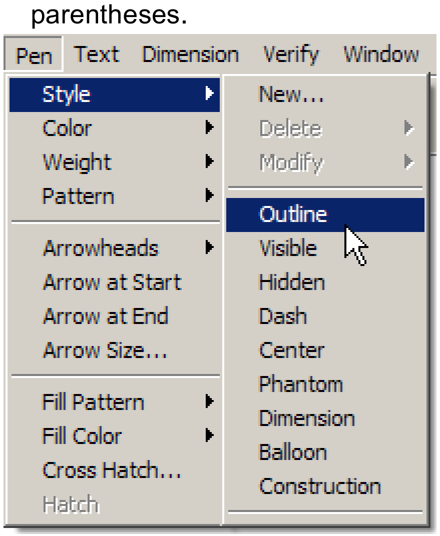
Layers and Pen Style

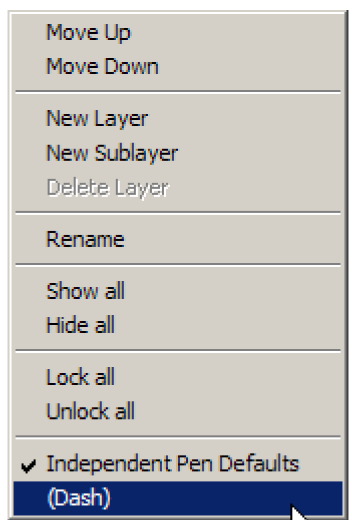
Line styles can be assigned to layers. The selected pen style will be the new default for that layer when active, for all future geometry, until another style is selected. A different pen style can be set for each layer as desired. It can be used only when the layer is active.

Using the Pen Style Override

1. Display the Layer Manager tab of the Design Explorer dialog box if it is not already displayed.
2. Click on the layer to activate. Right click on the layer to modify the line style.
3. Choose the Independent Pen Defaults option from the context menu. All layers that are overridden appear in italics in the Design Explorer.
4. Go to ***Pen>Style*** and choose one of the patterns available.
5. The Independent Pen Defaults option becomes checked in the right click menu and the style name is indicated below in the parentheses.







Layers and Copying/Pasting Objects

When objects are copied 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 geometry can be created. Cobalt, Xenon and Argon provide 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. Move the work plane as desired by creating a customer plane 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 Surface Primitive Tools for more information.*

As you learn to use this Designer Elements program, attempt to use the Z-Drafting Assistant without moving the work plane. Even though the Z-Drafting Assistant does a lot of the work, 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.

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 its 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.

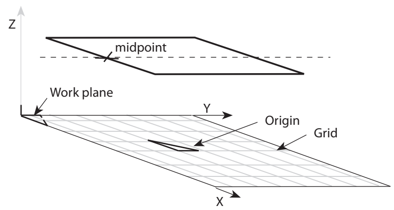
The topics explained in this chapter include:

- Work Plane
- Work Plane and Views
- Work Plane and World Coordinates

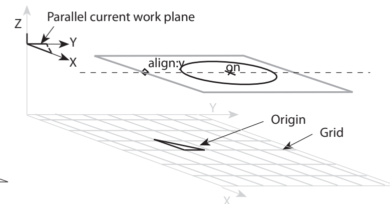
Work Plane

The work plane is an important feature of any CAD program. In conventional CAD programs it is necessary to specify both the orientation of the work plane and its exact location along the z-axis.

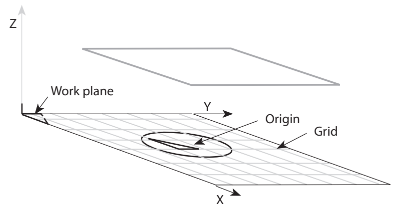
In this Designer Elements program, specify only the orientation of the work plane. Once that is done, all parallel planes act equally as the current work plane (e.g. the Drafting Assistant identifies the location of the work plane automatically).



When identifying one point of an object with the Drafting Assistant...



...the new object geometry is created on the plane of the identified object.



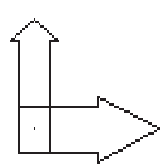
If an object is not identified, the new object geometry is placed onto the work plane at the origin.

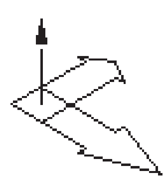
Another way to explain the relationship presented in the graphics above is to remember the role of the Drafting Assistant. For new geometry to be placed on the same plane as another piece of geometry either create a user-defined work plane at that location or brush over that “old” geometry to wake up one of its control points like midpoint or endpoint. The program will then retain this plane information as the new geometry is created.

If no object exists or no geometry is referenced for plane information, the 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 the 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

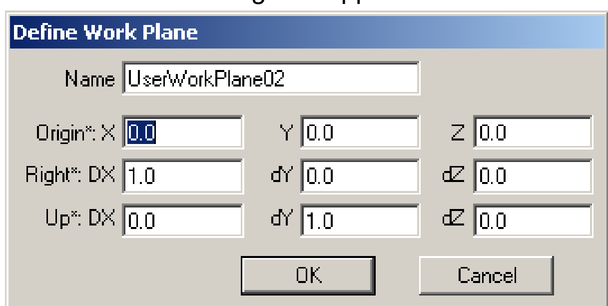
The Work Plane Manager, accessed through the coordinate system icon in the Status Line, is a menu of commands that defines, deletes, displays the properties of and saves the current work plane. It also shows or hides the work plane.

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***) defines a new work plane. An unlimited number of planes can be defined. When choosing this command, the Define Work Plane dialog box appears.



The dialog box contains the following items.

Name	Displays the work plane name. If a unique name is not entered, 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 the ability to specify locations by clicking in the drawing area. The values can be entered 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. The locations can also be specified by clicking in the drawing area.
3. Enter in name for the work plane in the Name data field.

   If a name is not entered, 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.

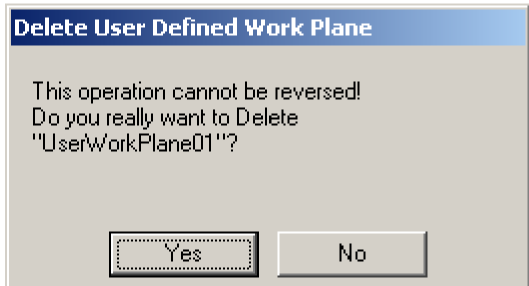
   Notice that the new work plane is listed and is the current work plane. It is possible now to operate in the user-defined coordinate system.

Referral: For information see Coordinate System Axis.

Delete Command

Use this command, located in the Planes Menu (***Planes>Delete Work Plane***) to delete a user-defined work plane. It is not possible to delete the GlobalWorkPlane or DynWorkPlane items.

1. Select the user-defined work plane 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*.

This plane can be renamed by selecting the name and choosing the *Properties* command in the Work Plane manager.

Show Work Plane Command

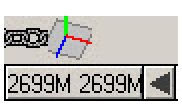
This is the same command available though the Planes menu. Choosing the command displays the work plane icon in the drawing.

Work Plane Identification

Determine if the current work plane is global or user-defined through the coordinate system/work plane icon displayed to the left side of the Status Line.

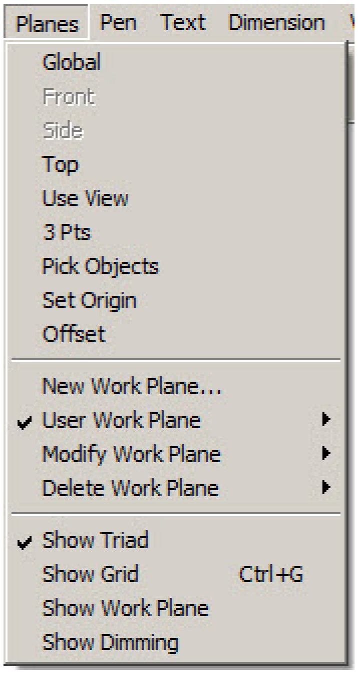
The Global Work Plane icon is a miniature version of the axis icon.

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 there are commands for setting the work plane. A work plane may be defined other than the standard planes. For example, to work on an angled face, reorient the work plane.



It is possible to 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, the work plane manager menu also can be used to define and set the work plane.

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 another work plane is set. Use the *New* command in the work plane manager to define and save a work plane with the file. If objects are created on a temporary work plane, 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.)

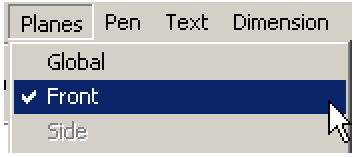
Tech Note: *Being in the Front, Side or Top view, the work plane is already set to the respective plane and the other two predefined planes in the Plane menu are unavailable. Temporary planes created with the User View, 3 Pts, Pick Objects and Define commands are not saved for future use.*

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 the 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.

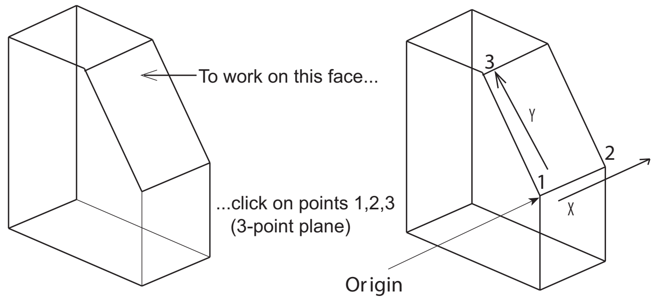
Choosing the Use View Command

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 moving from view to view while drafting.

Choosing the 3 Pts Command

This command sets a temporary work plane from the three specified points. This command is especially helpful to create non-parallel planes.



Referral: *See Infinite Plane Tool for information on using it.*

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 sets the work plane based on selected objects. This includes selecting an infinite plane icon.

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 set as the work plane.

   The work plane changes to the plane of the objects. Choose ***Plane>Show Work Plane*** to verify that the new plane is selected.

Use this tool to set the work plane to the face of a solid object.

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.*

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 pressing the short cut key associated with work plane snapping, the following behavior will be:

- Work plane origin moves to Drafting Assistant snap location. Aligns to the Top plane.

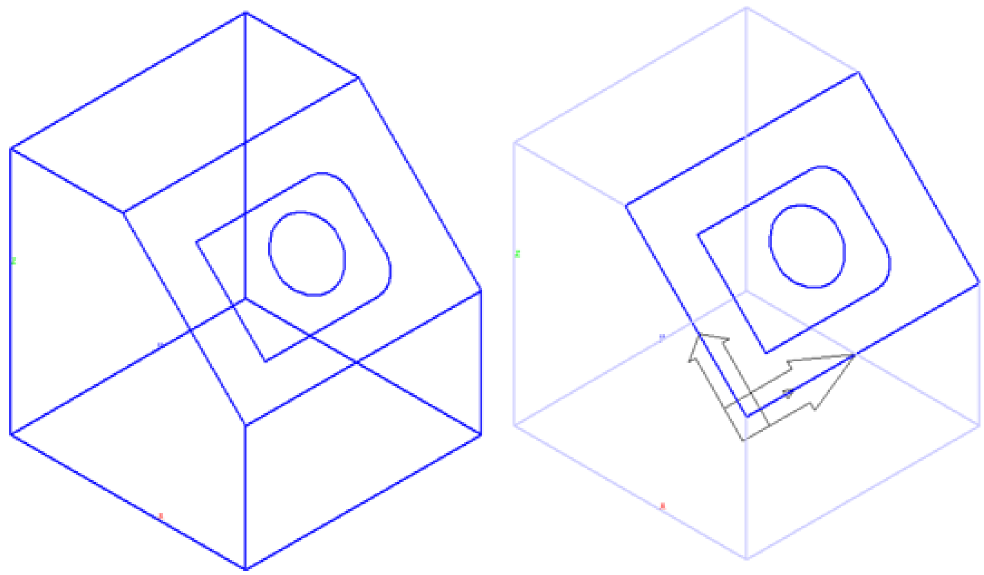
- Pressing C a second time changes from Top to Side work plane orientation.
- A third time changes from Side to Front work plane orientation.
- The fourth time C is pressed changes from Front to a user-defined work plane orientation.
- The fifth time C is pressed changes from back to top plane.

If the face snap options is on, pressing the short cut key will automatically align with respect to the face normal.

Note: an object must be selected in order for the work plane snapping short cut to work.

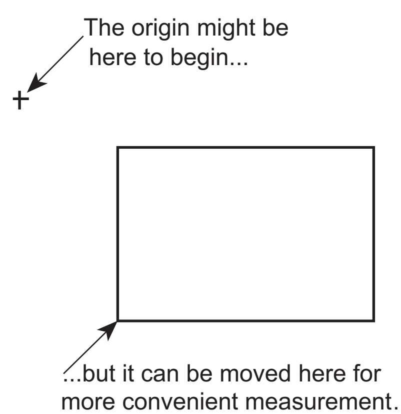
Work Plane Dimming

This menu command provides a means to dim objects that do not lie in the work plane.



Moving the Origin

Occasionally, it may be necessary simply to move the origin of the work plane. This is especially useful for measuring distances. For example, 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.

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

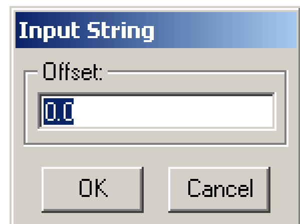
Using this Designer Elements program, the work plane normal may be moved a certain distance to the current work plane so that new geometry will snap to it. The *Offset* command in the Planes menu gives this ability.

> ***Tip:*** *Use the 3 Pts command in the Planes menu if the new plane is not parallel.*

Using the Offset Command

1. Display the work plane icon. (This is not required but it will help to 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

To create geometry, choose a particular plane on which to draw. To view geometry in this Designer Elements program, choose a particular view orientation.

To eliminate the potential confusion between work planes and views Cobalt, Xenon and Argon has tied the Front, Side and Top views to their respective planes. For example, if the Top view is chosen, the Top plane is selected in the Plane menu as the work plane and all other planes are unavailable.

For more information about Views see “Viewing Geometry”.

View the Plane

This command in the View menu changes the view to the current work plane.

Work Plane and World Coordinates

Cobalt, Xenon and Argon use the world coordinate system (as opposed to the User Coordinate System) for defining planes.

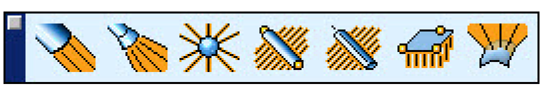
Lighting

Cobalt, Xenon and Argon provide default lights to display the objects. But excellent photo-realistic rendering requires more than just the defaults. That's why these Designer Elements programs provide directional, ambient and sun light sources with geometric pattern and color temperatures. This chapter describes how to use each of these including:

- Directional Lighting
- Ambient Lighting
- Default Lighting
- User-defined Lighting Layouts
- Photo-realistic Sunlight for Stills and Animations
- Lights & Facets

> ***Tech Note:*** *For more artistic elements, refer to Appendix E, Photo-realism Fundamentals.*

Directional Lighting

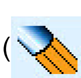
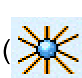
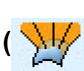


The **Distant Light** source, **Spot Light** source and **Point Light** source are directional sources. Location and direction can be specified for each source. Place these lights in the drawing by using the **Light** tool palette.

These light sources also support shadows when working within the advanced rendering environment.

Light Palette

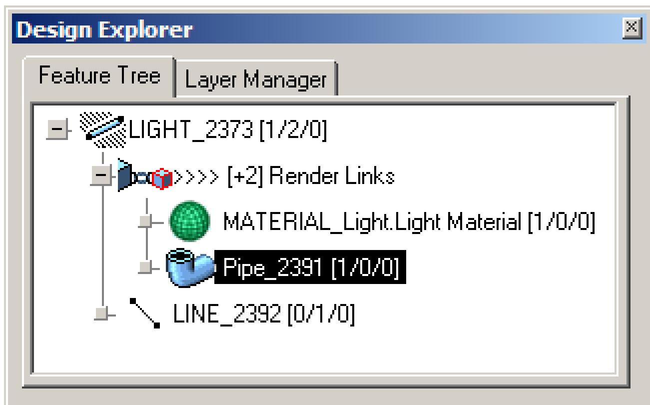
The **Light** tool palette does not automatically display when launching this Designer Elements program. To display the palette choose ***Window>Lights***.

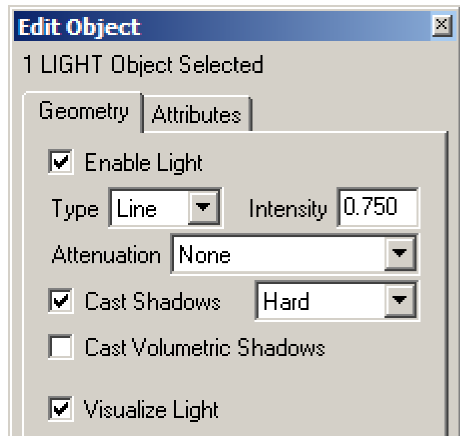
The **Light** tool palette includes the **Distant Light** (), **Spot Light** (), **Point Light** (), **Linear Tube Light** (), **Tube Light From Curve (****)**, **Rectangular Area Light (****)**, **Area Light from Selected Surface (****)**. Light sources placed using this palette contribute light color and intensity to all objects that lie within their influence and are used to define the light.

Light sources appear in the 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 unless Visualize Light option is checked in the Edit Objects dialog box to show the light source while rendering.

Activate the **Create Solid Body** option in the Message Line to be able to edit the source of light through the Edit Object dialog box.

The solid body of the light appears in the Design Explorer and can be edited on the Geometry tab of the Edit Object dialog box.

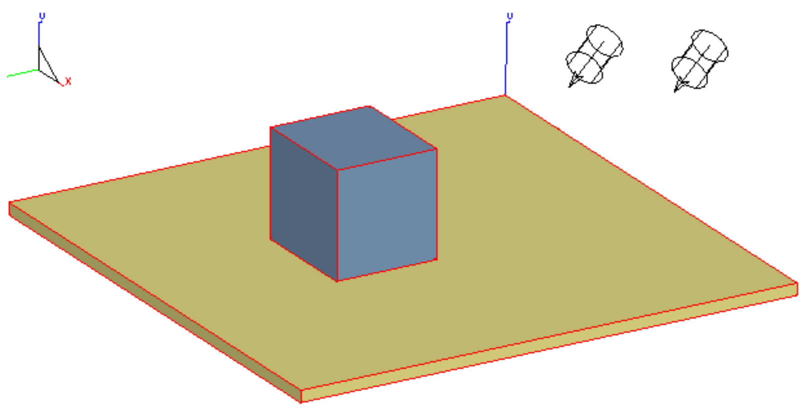




Light sources can be hidden using the *Show/Hide* command or moved to a separate layer and hidden. However both actions turn off the light.

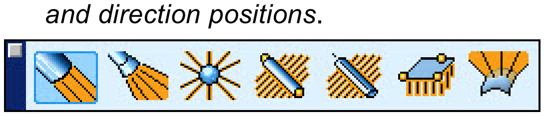
Distant Light Source

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 the 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. Adding lights, 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.

Flip Direction button changes the direction of the light source for 180 degrees.

The Enable Light check box turns off a light while retaining the intensity, direction and location settings. When checked, the light is on. The default setting is on.

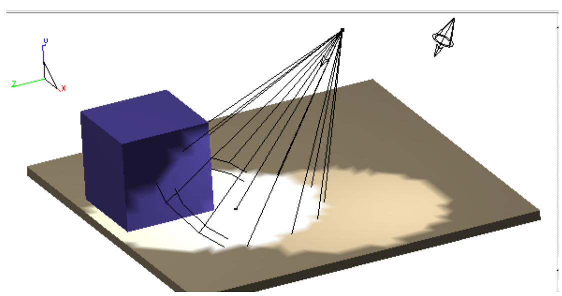
The Cast Volumetric Shadows check box enables simulation of a full range of effects occurring in a participating medium, i.e. attenuation within the medium, light filtration through a colored medium and first order light scattering inside the medium with volumetric shadows.

The 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, there is the option of setting the type of shadow casting. There are 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 the scene with Preview Render (shadows on) or Raytrace Render (shadows on, Anti-Alias). See Photo-realistic Rendering Commands.

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 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 the 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 lights are added, 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 and Flip Direction button.

Flip Direction button changes the direction of the light source for 180 degrees.

The Enable Light check box turns 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 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, there is the option of setting the type of shadow casting. There are four shadow type options: hard, medium, soft and blurry.

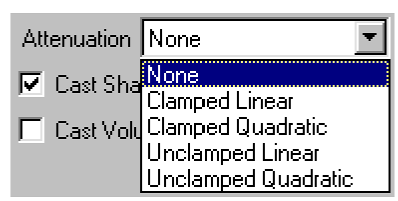
The ability to control shadow casting is especially valuable in a scene containing multiple light sources. Too many shadows can render a scene confusing. This check box setting only applies when rendering the scene with Preview Render (shadows on) or Raytrace Render (shadows on, Anti-Alias). See Photo-realistic Rendering Commands.

Slide check box enables the associated drop down for browsing the image which will be used for shining through.

For Attenuation Settings see further.

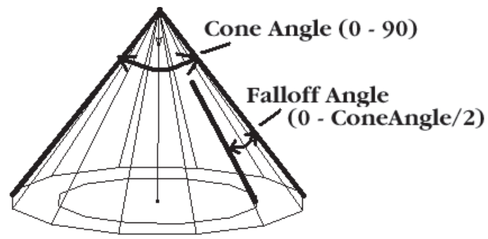
Attenuation Settings

Attenuation controls how quickly the light intensity diminishes with the distance from the light. The following attenuation options are available:



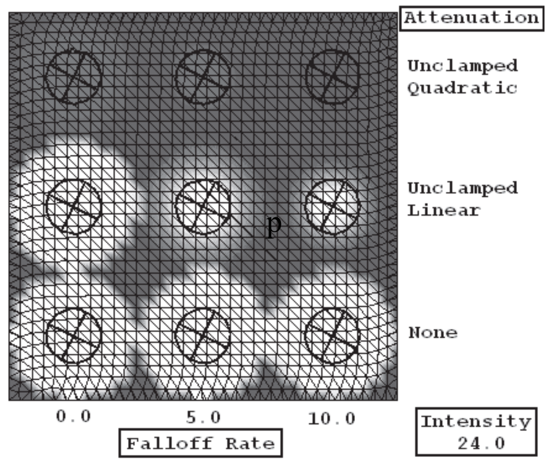
None	Light intensity does not change with distance.
Clamped Linear	Light intensity diminishes according to the following formula, *Intensity /(distance+1)*.
Clamped Quadratic	Light intensity diminishes according to the following formula, $Intensity\ /(distance^2+1)$.
Unclamped Linear	Light intensity 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.

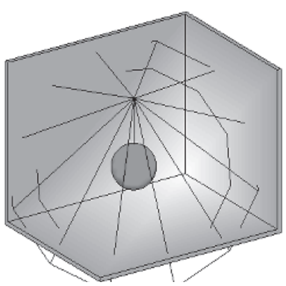
Falloff Angle controls the sharpness of the spot light’s edge. The smaller the angle is the sharper the edge. The light intensity diminishes from full intensity at the inner cone to zero at the outer cone. The value for the falloff angle ranges from zero (0) degrees to *Cone Angle/2* degrees.

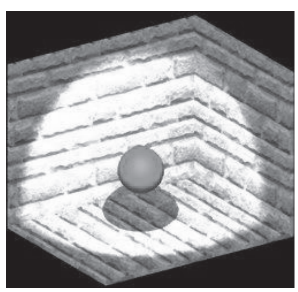


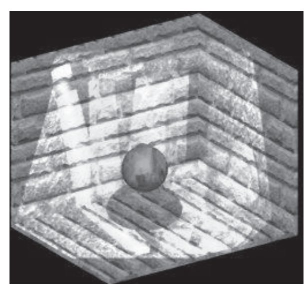
Falloff Rate controls how light is distributed within the spot light’s cone. The intensity diminishes from the center line of the cone outward with the cosine of the angle raised to the power of the falloff rate. A falloff rate of zero (0) results in uniform light within the cone. Higher values produce 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.

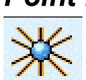

The scene

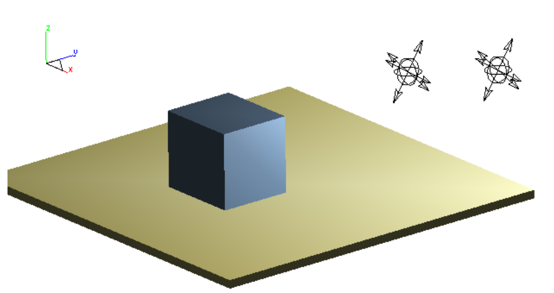

Without slide image


With slide image

The light source color is accessible through the Attributes page.

Point Light Source

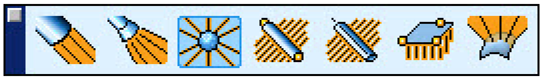




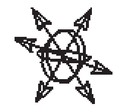
A point light illuminates a scene with light emanating in all directions. A candle or a table lamp is a type of point light.

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 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 the 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 lights are added, 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.

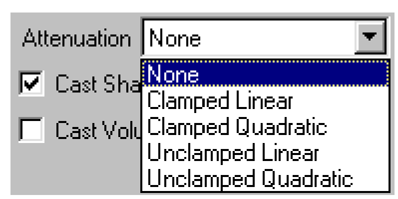
The Enable Light check box turns off a light while retaining the intensity, direction and location settings.

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, there is also the option of setting the type of shadow casting. There are four shadow type options: hard, medium, soft and blurry. This check box setting only applies when rendering the scene with Preview Render (shadows on) or Raytrace Render (shadows on, Anti-Alias). See “Photo-realistic Rendering Commands” in this chapter.

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.

Attenuation Settings

Attenuation controls how quickly the light intensity diminishes with the distance from the light. There are 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*.

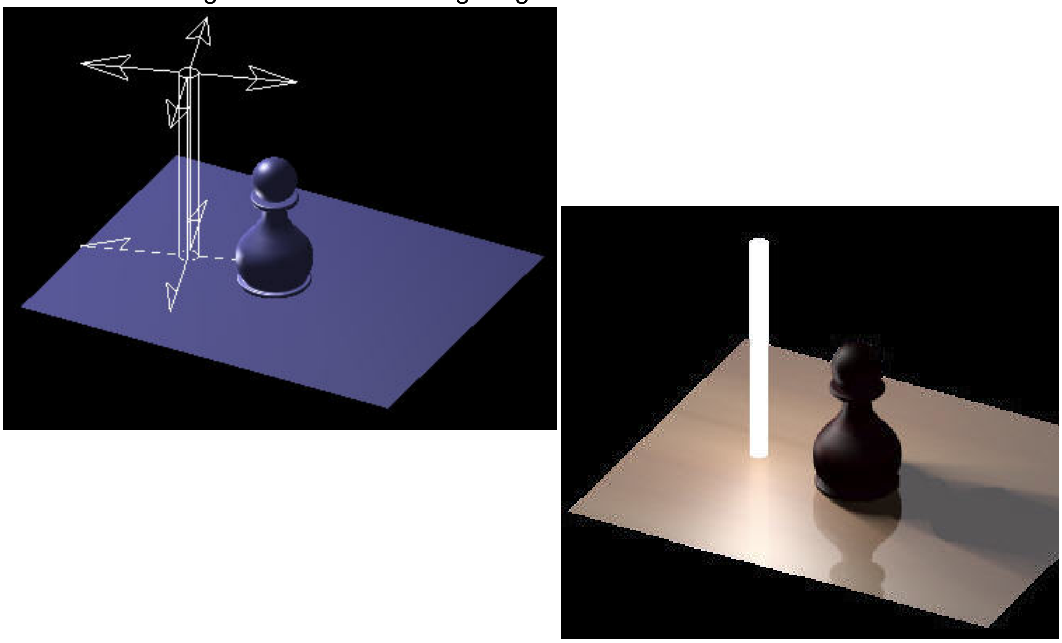
Unclamped Quadratic Light intensity diminishes according to the following formula, $Intensity / distance^2$.

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.

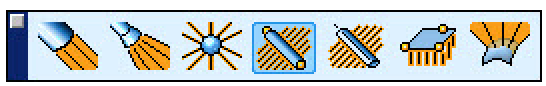
The light source color is accessible through the Attributes page.

Linear Tube Light

Use linear tube lights for fluorescent lighting effects.



Using the Linear Tube Light

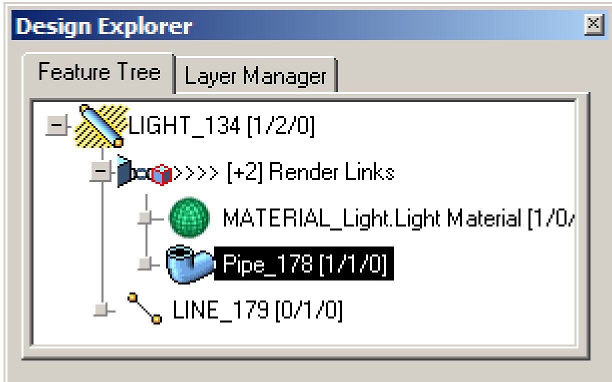


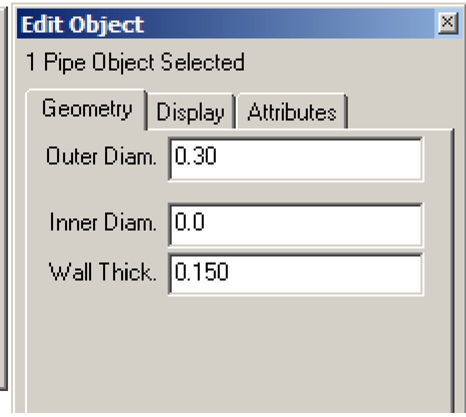
1. Select the **Linear Tube Light** tool.
2. Activate the **Create Solid Body** option in the Message Line to create a pipe body that presents source of light. When the light is visualised it looks like on the right picture above.

   The Message Line reads: *Pick first point of linear tube light.*
3. Following the prompts in the Message Line, pick two points to specify the beginning and the end of the tube.
4. In the Edit Objects dialog box, check the Visualize Light option to show the light source while rendering.

The solid body of the light appears as a pipe in the Design Explorer and can be edited on the Geometry tab of the Edit Object dialog box.





The linear tube light symbol appears on the screen.

5. Render the scene.

Geometric Characteristics

According to the Edit Objects Geometry page, linear tube light source is made up of the following characteristics: Type, Intensity, Attenuation, Cast Volumetric Shadows. It also includes the Enable Light check box, Cast Shadows check box and its associated pull-down menu and Visualize Light check box.

The Enable Light check box turns off a light while retaining the intensity, direction and location settings.

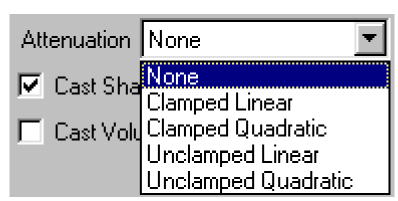
The Cast Volumetric Shadows check box enables simulation of a full range of effects occurring in a participating medium, i.e. attenuation within the medium, light filtration through a colored medium and first order light scattering inside the medium with volumetric shadows.

The 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, there is the option of setting the type of shadow casting. There are four shadow type options: hard, medium, soft and blurry. This check box setting only applies when rendering the scene with Preview Render (shadows on) or Raytrace Render (shadows on, Anti-Alias). See Photo-realistic Rendering Commands.

Visualize Light check box makes the rendered light source look like a fluorescent lamp. Works only when Create Solid Body option was enabled while creating the light source.

Attenuation Settings

Attenuation controls how quickly the light intensity diminishes with the distance from the light. There are 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* $/(distance^2+1)$.

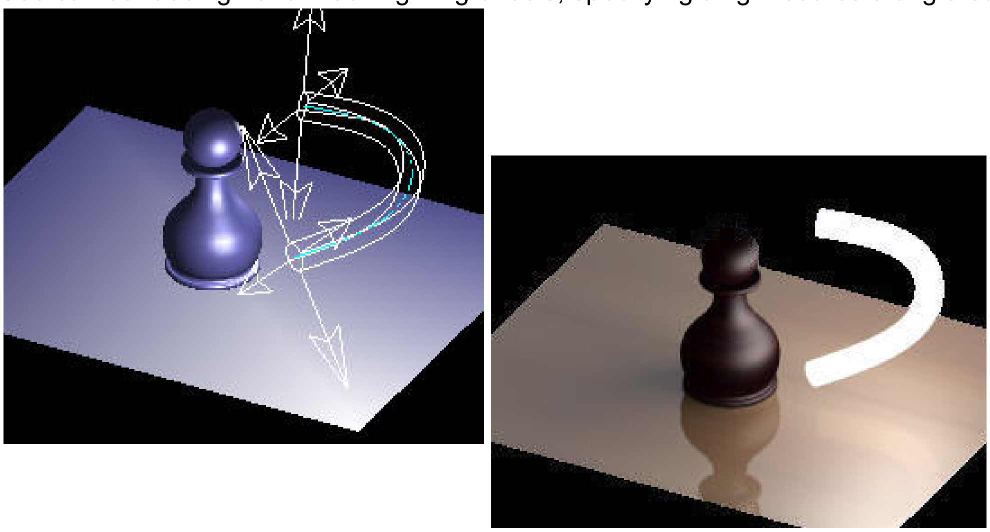
Unclamped Linear Light intensity diminishes according to the following formula, *Intensity /distance*.

Unclamped Quadratic	Light intensity diminishes according to the following formula, *Intensity /distance*^{2}.

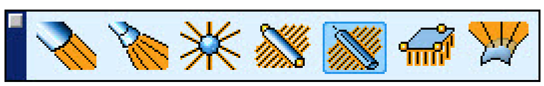
Use clamped attenuations for situations were the distance from a light source to an object is less than 1” (25.4 mm). Clamped attenuations do not increase

Tube Light From Curve

Use curved tube lights for neon lighting effects, specifying a light source along a curve.



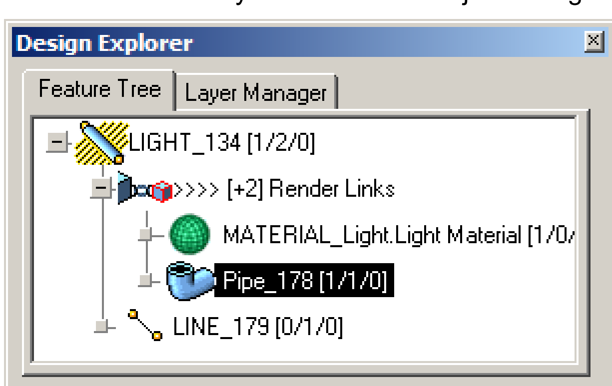
1. **Using the Tube Light From Curve**

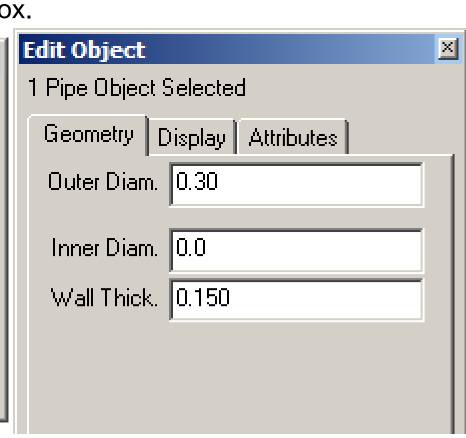


1. Create a curve such as an arc, circle or spline, along which to create a light source.
2. Choose **Tube Light from Curve** in the **Lights** tool palette.
3. Activate the **Create Solid Body** option in the Message Line to create a pipe body that presents source of light. When the light is visualised it looks like on the right picture above.

4. Following the prompts in the Message Line, pick the curve created.
5. In the Edit Objects dialog box, check the Visualize Light option to show the light source while rendering.

   The solid body of the light appears as a pipe in the Design Explorer and can be edited on the Geometry tab of the Edit Object dialog box.





The symbol for this light source appears on the screen.

6. Render the scene.

Geometric Characteristics

According to the Edit Objects Geometry page, tube light from curve source is made up of the following characteristics: Type, Intensity, Attenuation, Cast Volumetric Shadows. It also includes the Enable Light check box, Cast Shadows check box and its associated pull-down menu and Visualize Light check box.

The Enable Light check box turns off a light while retaining the intensity, direction and location settings.

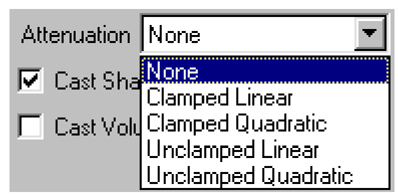
The Cast Volumetric Shadows check box enables simulation of a full range of effects occurring in a participating medium, i.e. attenuation within the medium, light filtration through a colored medium and first order light scattering inside the medium with volumetric shadows.

The 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, there is the option of setting the type of shadow casting. There are four shadow type options: hard, medium, soft and blurry. This check box setting only applies when rendering the scene with Preview Render (shadows on) or Raytrace Render (shadows on, Anti-Alias). See Photo-realistic Rendering Commands.

Visualize Light check box makes the rendered light source look like a neon lamp. Works only when Create Solid Body option was enabled while creating the light source.

Attenuation Settings

Attenuation controls how quickly the light intensity diminishes with the distance from the light. There are the following attenuation options:

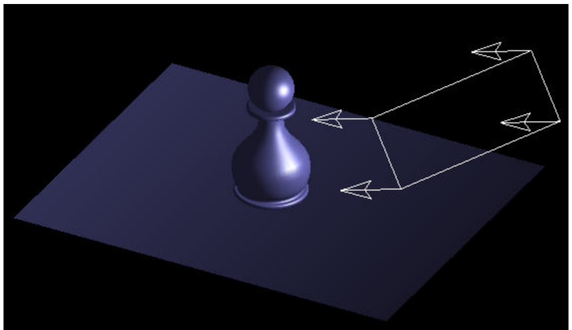


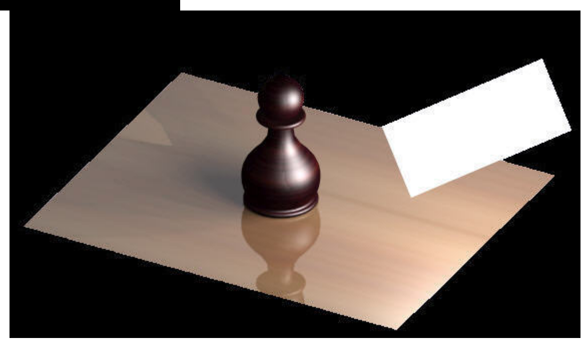
None	Light intensity does not change with distance.
Clamped Linear	Light intensity diminishes according to the following formula, *Intensity /(distance+1)*.
Clamped Quadratic	Light intensity diminishes according to the following formula, *Intensity* $/(distance^2+1)$.
Unclamped Linear	Light intensity 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

Rectangular Area Light

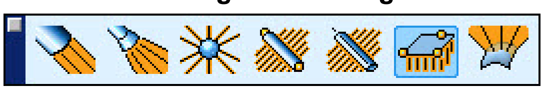
Area lights create a lovely diffused light. Simply create a rectangle and have it glow as a light source. Use these lights in stills and animations for improved realism for photo-realistic renderings.





Using the Rectangular Area Light

1. Select **Rectangular Area Light** tool from **Lights** palette in the Window menu.



2. Following the prompts in the Message Line, click three points to build the rectangular area to be the light source.
3. In the Edit Objects dialog box, check the Visualize Light option to show the light source while rendering.

   The rectangular light symbol appears on the screen.
4. Render the scene.

Geometric Characteristics

According to the Edit Objects Geometry page, rectangular area light source is made up of the following characteristics: Type, Intensity, Attenuation, Cast Volumetric Shadows. It also includes the Enable Light check box, Cast Shadows check box and its associated pull-down menu, Visualize Light check box and Flip Direction button.

Flip Direction button changes the direction of the light source for 180 degrees.

The Enable Light check box turns off a light while retaining the intensity, direction and location settings.

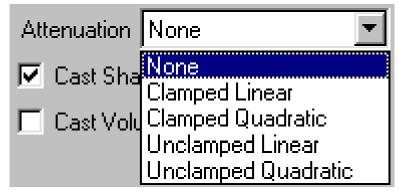
The Cast Volumetric Shadows check box enables simulation of a full range of effects occurring in a participating medium, i.e. attenuation within the medium, light filtration through a colored medium and first order light scattering inside the medium with volumetric shadows.

The 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, there is the option of setting the type of shadow casting. There are four shadow type options: hard, medium, soft and blurry. This check box setting only applies when rendering the scene with Preview Render (shadows on) or Raytrace Render (shadows on, Anti-Alias). See Photo-realistic Rendering Commands.

Visualize Light check box makes the rendered light source look like a rectangular neon lamp when the scene is rendered.

Attenuation Settings

Attenuation controls how quickly the light intensity diminishes with the distance from the light. There are 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* $/(distance^2+1)$.

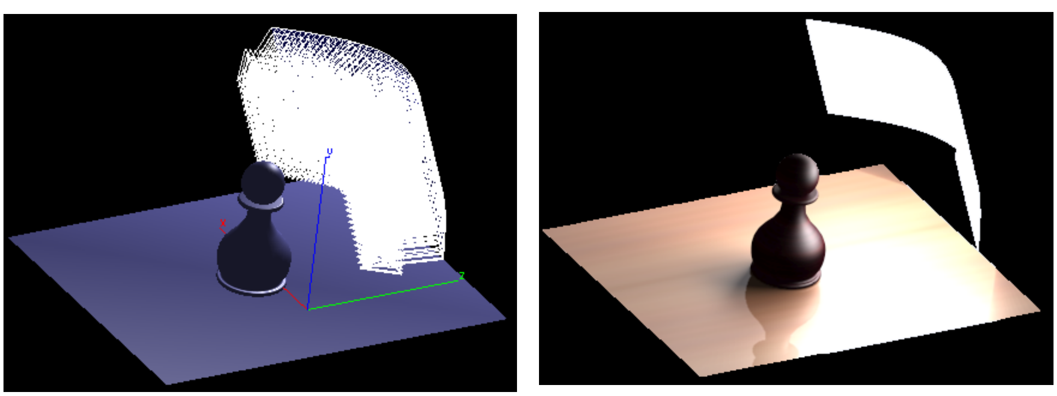
Unclamped Linear Light intensity diminishes according to the following formula, *Intensity /distance*.

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

Area Light from Selected Surface

This kind of light emanates from a glowing surface achieving a perfect photo-realistic effect for stills and animations.



Using the Area Light From Selected Surface

1. Create any surface object.
2. Choose **Area Light from Selected Surface** in the **Lights** tool palette.

3. Following the prompts in the Message Line, pick the surface.
4. In the Edit Objects dialog box, check the Visualize Light option to show the light source while rendering.

The area light source symbol appears.

5. Render the scene.

Geometric Characteristics

According to the Edit Objects Geometry page, area light from selected surface light source is made up of the following characteristics: Type, Intensity, Attenuation, Cast Volumetric Shadows. It also includes the Enable Light check box, Cast Shadows check box and its associated pull-down menu, Visualize Light check box and Flip Direction button.

Flip Direction button changes the direction of the light source for 180 degrees.

The Enable Light check box turns off a light while retaining the intensity, direction and location settings.

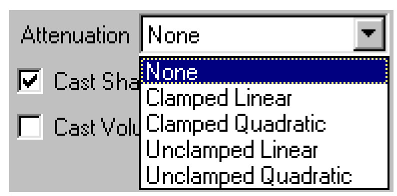
The Cast Volumetric Shadows check box enables simulation of a full range of effects occurring in a participating medium, i.e. attenuation within the medium, light filtration through a colored medium and first order light scattering inside the medium with volumetric shadows.

The 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, there is the option of setting the type of shadow casting. There are four shadow type options: hard, medium, soft and blurry. This check box setting only applies when rendering the scene with Preview Render (shadows on) or Raytrace Render (shadows on, Anti-Alias). See Photo-realistic Rendering Commands.

Visualize Light check box makes the rendered light source look like a rectangular neon lamp when the scene is rendered.

Attenuation Settings

Attenuation controls how quickly the light intensity diminishes with the distance from the light. There are the following attenuation options:



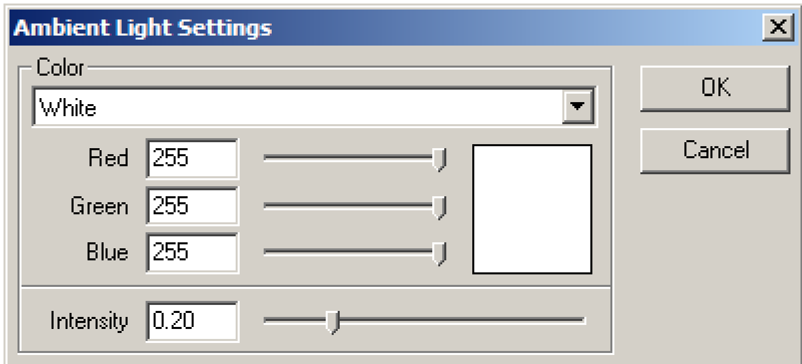
None	Light intensity does not change with distance.
Clamped Linear	Light intensity diminishes according to the following formula, *Intensity /(distance+1)*.
Clamped Quadratic	Light intensity diminishes according to the following formula, $Intensity/(distance^2+1)$.
Unclamped Linear	Light intensity diminishes 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

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. 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. Either enter the value in the data field or use the slide to set the intensity. Zero (0) is off.

After choosing the ambient settings, click OK. When saving the file, the ambient light setting is also saved.

Modifying the Lights

As designing the geometry and rendering it, view it under various lighting conditions, either by changing the layout or editing the light source.

Modifying Ambient Light

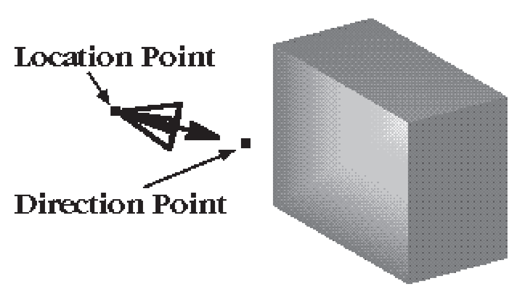
Ambient light can be modified through the Ambient Light Setting dialog box in the View menu. Adjust the color and intensity level and click OK to save the changes. Choose ***PhotoRender>Render*** to display the geometry with the new settings.

Modifying Directional Lights

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. It is possible to 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

Perform comprehensive editing on directional light sources through the Edit Objects dialog box. The options available depend on the light source.

Geometry Tab

Type	Choose from the three lights available: Distant, Spot and Point.
Intensity	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 turns 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	Enter a specific name for the source.
Resolution	This characteristic does not apply to light sources.
Color	Specify any color for the light source. The pull-down menu offers the standard colors and the *More* option, which displays the color palette when selected.
Control Pts	Display or hide the control points of the light source.
Layer	Place a light source on any layer. The menu displays all available layers.
Locked	Lock the light source to prevent modification.

Click Apply to accept the changes and then Close to exit the dialog box. Choose ***PhotoRender>Render*** to display the 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 the geometry when using the *Render* command. The default lighting is based on the viewer's eye location and direction to provide a high quality rendering of the geometry. The default light source cannot be modified. However, specify own ambient light with the default lighting. When placing own light sources, the default light source deactivates.

User-defined Lighting Layouts

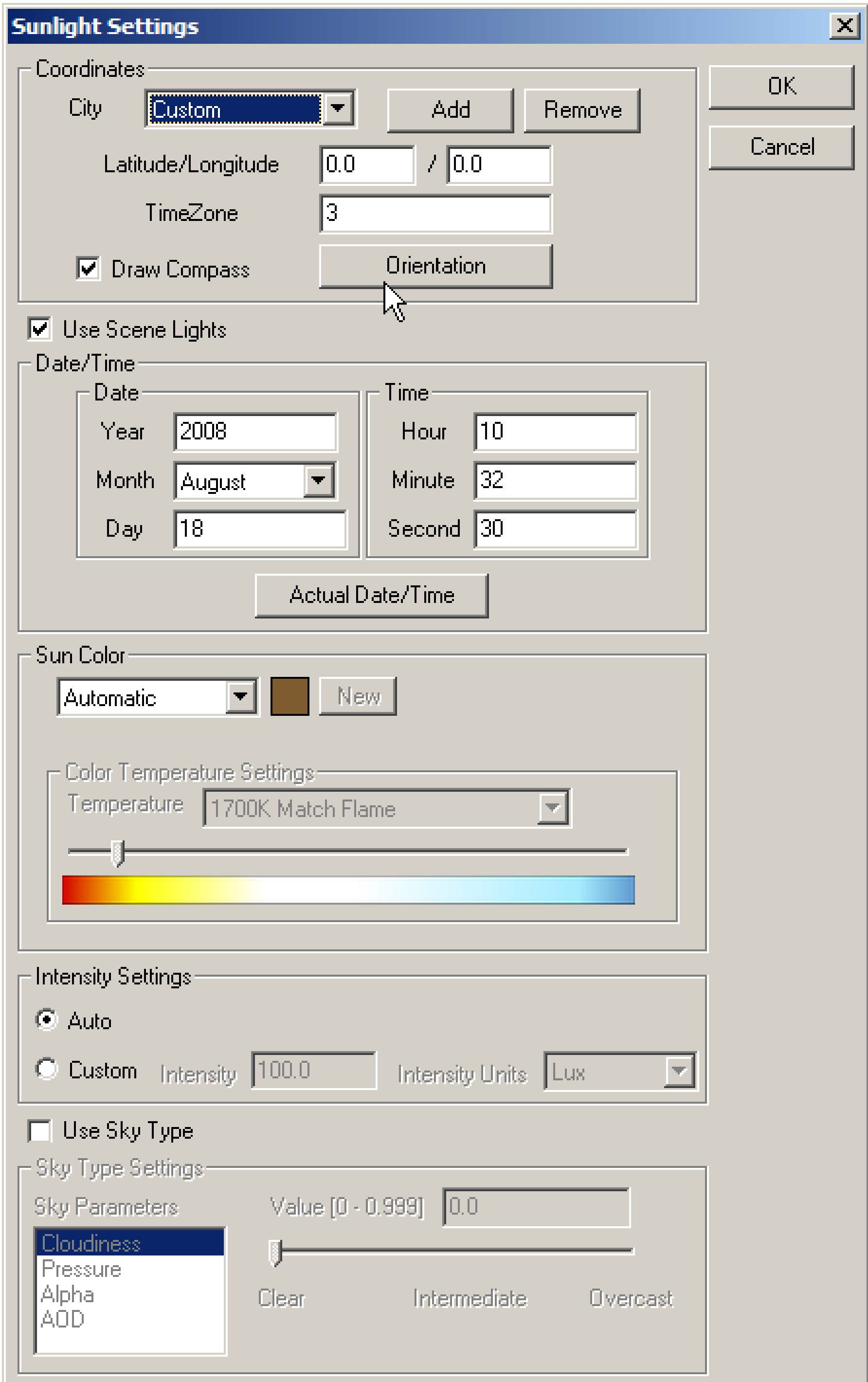
Create own lighting layout using that layout for other drawings by exporting the lights to a stand-alone Designer Elements program file. This lighting file can then be imported into other files. To export a lighting set, do the following:

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.

Now it is possible to import this lighting into other Designer Elements program files.

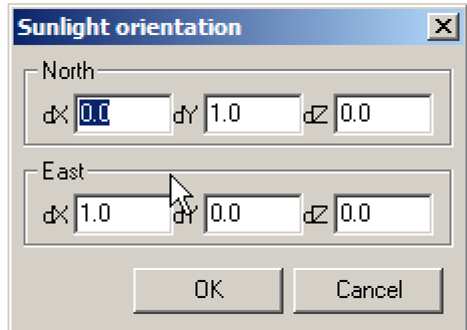
Photo-realistic Sunlight for Stills and Animations

Cobalt, Xenon and Argon v8 feature photo-realistic sunlight settings for stills and animations using location, date and time. This makes it easy to do light and shadow studies for architectural designs.

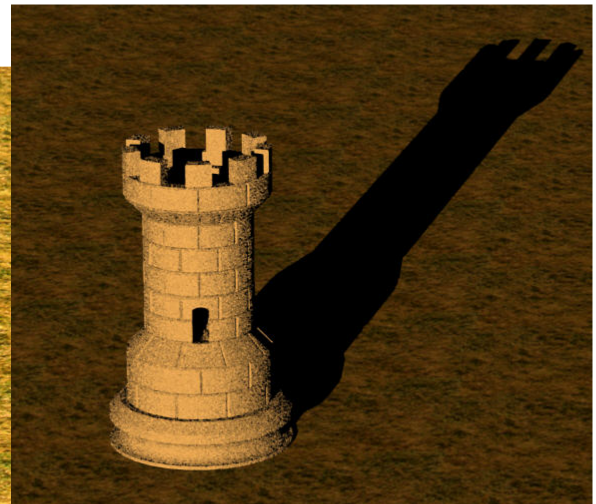


1. Select ***PhotoRender>Edit Sunlight***. The Sunlight Settings dialog box appears.
2. Enter a city from the list for the longitude and latitude to show automatically, or click Add to include a custom location using longitude and latitude, and the time zone plus or minus from Greenwich Mean Time.

3. Click Orientation and in the Sunlight Orientation box set the position of the sunlight by North and East vectors.
4. Enter the desired date and time. This specifies the intensity of the sunlight automatically.



This rendering is set for 1 pm.



This rendering is set for 6 pm.

When set to automatic mode, the sunlight color is determined by the coordinates and the date and time. In the Custom mode, it is possible to set any plain color. In the By Temperature mode, the sunlight color is determined by the color temperature. The temperature rate can be varied within 150 K and 20,000 K.

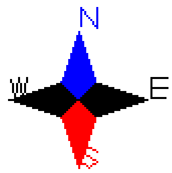
Intensity Settings specify the exact amount of the sunlight that illuminates the square unit of the area. Auto uses the parameters specified in the Coordinates, Date/Time, and Sun Color. The Custom option provides the Intensity Units drop down menu with Lux, Kilolux and Footcandle options where Lux = Lumen/m2, Kilolux = 1000 Lux, Footcandle = Lumen/in2.

Sky Type Settings designate the Sky Parameters which also influence the sunlight intensity.

Cloudiness	Is set between 0 and 0.999. Either use the slider or input field to specify the level of cloudiness between Clear, Intermediate and Overcast. The greater the value the more overcast the sky.
Pressure	Set it to Low, Normal or High with the slider or input the value between 0 and 2500 in the field.
Alpha	Specifies the type of the locality. Use the slider to choose between Mountain, Rural and Maritime locality or set the value between 0 and 2 in the input field.
AOD	Specifies the type of the atmosphere. Use the slider to choose between No Aerosol, Clear and Dusty atmosphere or enter the value between 0 and 1 in the input field.

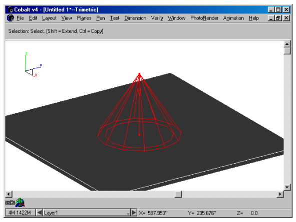
The rendering results will reflect the correct position of the sun at that time and season of the year.

If the Draw Compass option is checked, the compass symbol appears in the left bottom corner of the screen. The compass symbol sets the orientation of the scene.

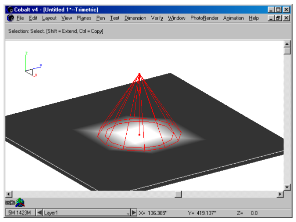


Lights & Facets

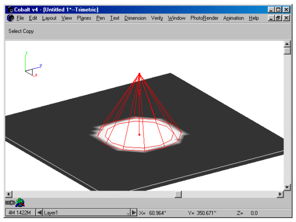
All facets are illuminated according to how much light falls on their vertices (the intersection of perpendicular isolines). If a light source is positioned over a large flat surface (such as a floor), it will appear that the source is not casting light. Set the object resolution to *Very Fine* or *Super Fine* to increase the facet density and enhance the lighting result. See Change Resolution for more information on resolution.



Fine Resolution



Very Fine Resolution



Super Fine Resolution

Display Materials, Decals & Environments

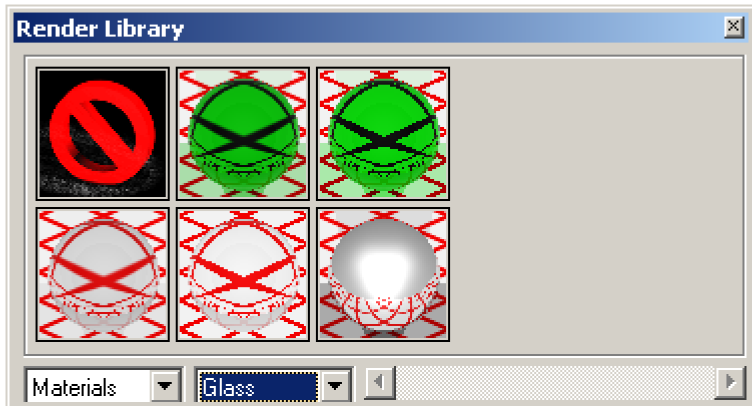
Cobalt, Xenon and Argon come with a library of materials, decals, foregrounds and backgrounds to make rendering easier. There is also a tool for implementing photo-realistic environment maps that reflect the surrounding world on an object's surface. This chapter discusses:

- Render Library
- Editing the Rendered Scene
- Photo-realistic Environment Maps for Stills and Animations

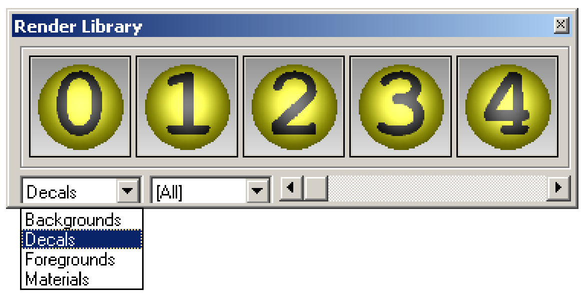
> ***Tip:*** *Additional packages for materials, decals and environments are available for separate purchase from Ashlar-Vellum. Please see the website at www.ashlar.com.*

Render Library

This Designer Elements program comes with an extensive render library containing materials and decals that can be applied to objects and background and foreground properties that can be applied to the scene. This library is contained in the **Render Library** tool palette. Choose ***Window>Render Library*** to display the tool palette.



Resize the palette horizontally or vertically by dragging its edges or corners.



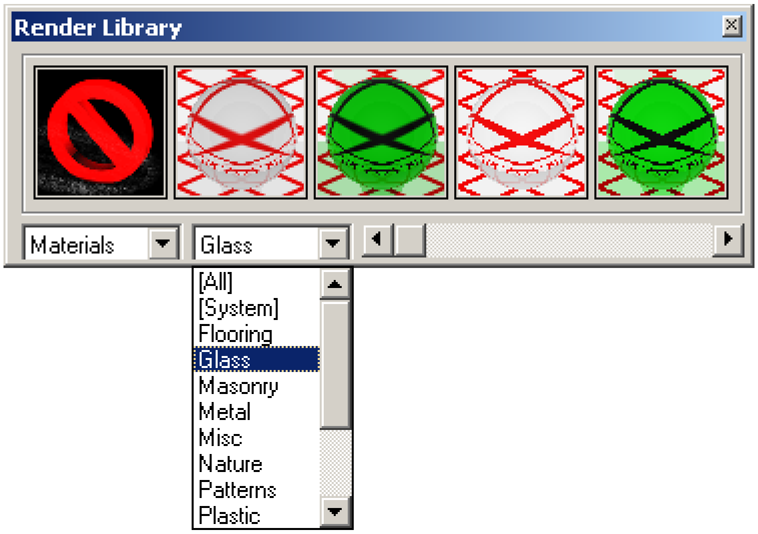
The palette contains the preview area, library type menu, category menu and scroll bar. The type menu controls which library displays on the palette. The Decals library type is shown in the graphic here.

Render Library Items

The render library menu contains four render libraries: Backgrounds, Decals, Foregrounds and Materials. Select any library by clicking on it or using the up or down arrow keys on the 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. Select any category within the category menu by clicking on it or using the up or down arrow keys on the 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 the objects regardless of their pen color. However, for 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 placing 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 to apply to the model. A material defines the surface color, transparency, reflectivity and roughness properties of a model. it is possible to 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.
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.

__Tech Note:__ 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.

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.

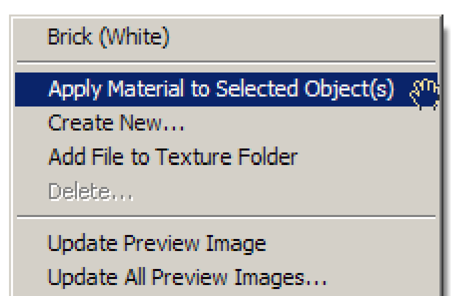
Application Symbol

4. Continue dragging to the object to which 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, then right click the mouse button.

__Tip:__ To apply different materials to different faces of a solid see Material Editing

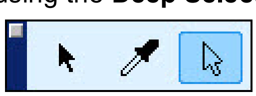
Choose Apply Material to the Seleted Object(s) from the context menu.



5. Apply more images to the objects as desired. Then render the drawing using one of the commands in the Render menu.

Material per Face

Different materials may be applied to each face using the **Deep Select** tool.



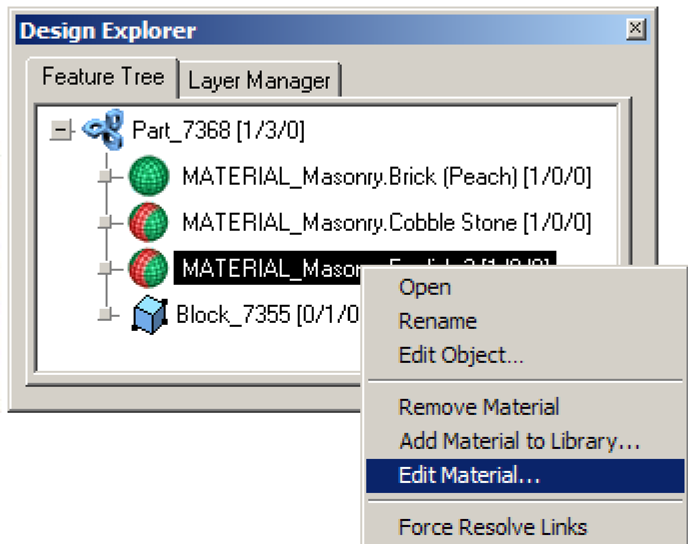
To apply a material to a face:

1. Select **Deep Select** tool, choose the face and designate the material from the ***Window>Render Library*** menu.

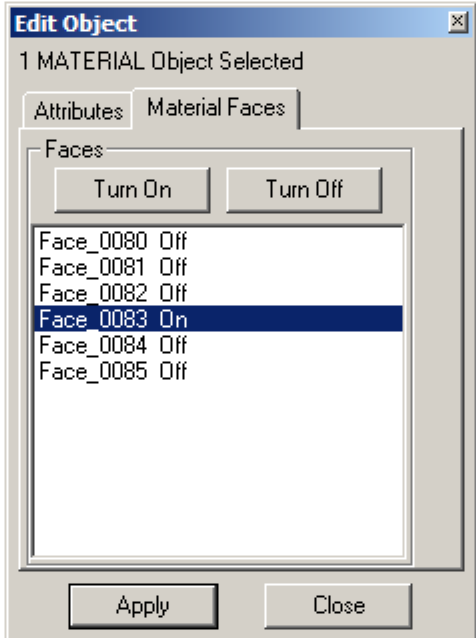
2. Right-click on the material and choose Apply to Solid Face(s).

3. Alternatively, choose the material from the Render Library menu, right-click it and select Apply to Solid Face(s) from the context menu. Then choose the face of the solid. The material is applied. Render the object to see the changes.

The material for each face appears in the Design Explorer as a ball  and can be edited.



Materials per Face are also editable through the Edit Objects box. Turn on or off the material under the Material Faces tab.



Geometric Characteristics

When applying 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.

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.

One material can be replaced with another by simply dragging the new material image to the desired object. It is also possible to 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 Material Editing for more information.

Using the Decals Library

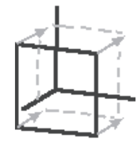
The Decals library contains pre-defined decals that can be applied to the model. A decal overrides the local surface color properties of an object to which it is applied.

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.

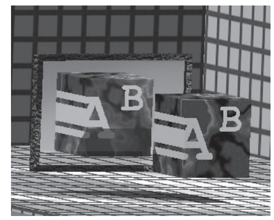
The categories in the Decals library include Misc and Digits.

Planar Wrap

This mode projects artwork along the decal's normal vector and appears on the surface at the specified location. The graphic shows a planar decal symbol. The gray arrows indicate the normal direction. The area defined by the artwork and the normals (the dotted lines) is the planar wedge.



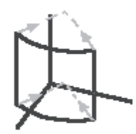
The graphic shows planar wrapping for one object using two decals. One decal extends beyond the surface edge and wraps along the other adjacent surface as defined by the planar wedge. The object's reflection appears in the mirror behind it.



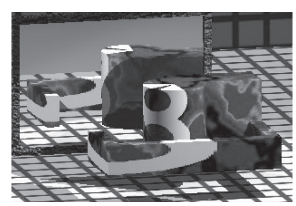
Cylindrical Wrap

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.

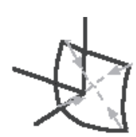


The graphic here shows cylindrical wrapping for one object composed of a cylinder, bracket and block. The decal extends beyond the edge of the cylinder and bends toward the center of the cylindrical wedge, wrapping around the object. The object's reflection appears in the mirror behind it.

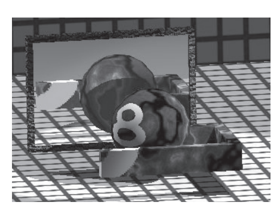


Spherical Wrap

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.



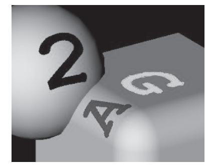
The graphic shows spherically wrapping for one object composed of a sphere and a bracket.



Misc Render images include: Eroded. When an object is rendered with this decal, the object material appears to be wearing away.

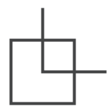
Digits Render images include the numbers 0 through 9.

This graphic is another example of the decal wrap modes.



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 it is to be applied. Use the axes included with the symbol for alignment. Release the mouse button.

Apply more decals to the objects as desired. Then render the drawing using one of the commands in the Render menu.

> ***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.*

Removing or Replacing Decals

To remove a decal from an object select the decal and delete it like any other object. 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 a decal is applied, 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 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.

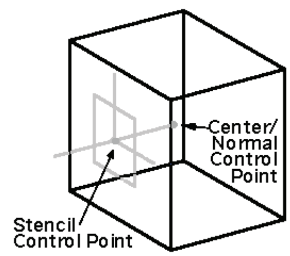
When this box is left unchecked, the wrap mode is set using the Wrap Mode pull-down menu.

One possible application of this is 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. Select Lock Normal again and click Apply, the decal snaps back to the normal location.

Radius

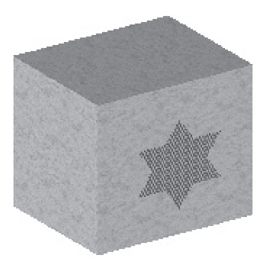
(Cylindrical and Spherical wrap only) Specify a radius for the center point or the stencil control of the decal, different from the default value listed.

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

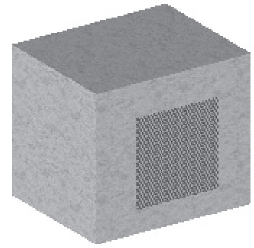
This menu controls how a decal is applied on an object's surface. There are two options, *Stencil* and *Factor*.

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. To use own stencils in this Designer Elements program, place them in this folder.

When creating 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: Creating a stencil that is 100% red (RGB), only the artwork color projected onto the object is used. If the red percentage is lowered, some of the underlying material color mixes in. If black and red is used in the 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.

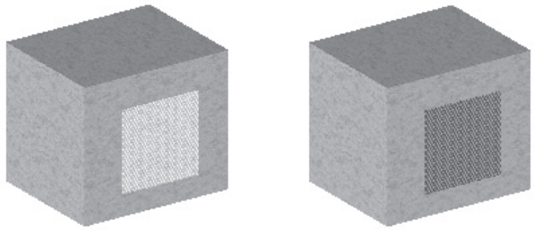
In equation form, this combination translates into: *SurfaceColor=ArtWorkColor*RedVal+MaterialColor*(1.0-RedVal)*.

This formula shows how this Designer Elements program references the stencil file.

Factor slide

(Appears when Factor Masking is selected) This slide sets 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.

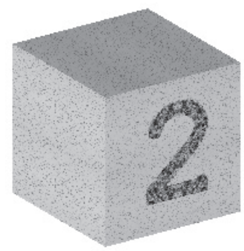
When the slider is set to the far left, representing 0.0, the decal color is set by the object's material. When the slider is set to the far right, representing 1.0, the decal color is used. The left graphic below has a 0.0 factor setting. The right graphic has a 1.0 factor setting.



In equation form the combination translates into: *Surface=ArtWorkColor*MixFactor+MaterialColor*(1.0-MixFactor)*.

Logo

This menu sets the source for the decal's artwork. There are 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. 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

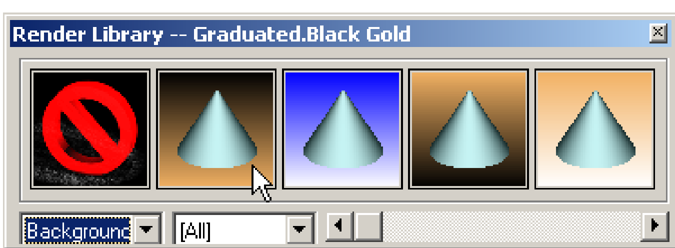
(Appears when Image is selected from the Logo menu.) This pull-down menu lists the images available for decal artwork. These images are located in the Textures folder within the Photo Render folder. To use own images place them in this folder. See User-defined Images for more information.

Using the Backgrounds and Foregrounds Libraries

Backgrounds and foregrounds can be used to set the overall scene. Only one foreground and one background can be used per rendered drawing.

Backgrounds Library

The Backgrounds library contains pre-defined background effects that can be applied to the model. Backgrounds control the appearance of those regions of a scene that do not contain objects.



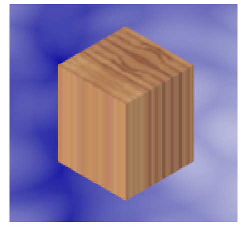
The categories in the Backgrounds library include Images, Misc and Nature.

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).

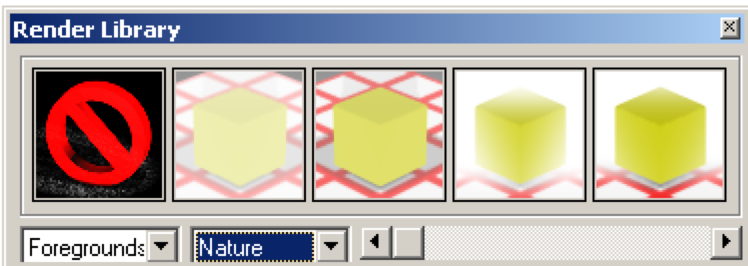
Nature Render images include: Clouds.

The graphic shown is an example of a cloud background.



Foregrounds Library

The Foregrounds library contains pre-defined foreground effects that can be applied to the model. Foregrounds effect the way the space between the eye point and the scene objects alter the rendering results.

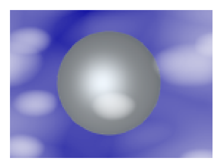


The categories in the Foregrounds library include Misc and Nature.

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.

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.

User-defined Images

Own material and decal images can be used in this Designer Elements program when used with the Edit Objects dialog box. This ability is useful to have specific materials or decals unique to your company and industry. On possible use is the application of company logos to objects in the 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 Custom Decals

To apply own decals, create own decal stencils and artwork. Since the artwork projects through the stencil, any bitmap image is acceptable. Create the artwork image using any graphic program that supports bitmaps.

To create a stencil do the following:

1. Create the shape of the stencil.
2. Color the stencil using a percentage of red depending on the desired stencil affect.

   Set the Red value (RGB color) to 255 or 100% red 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 only the material’s color must be added when the artwork is projected.

Applying User-defined Material Images

1. Create 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 the object.
4. Double-click on the object to display the Edit Objects dialog box.
5. From the Texture file menu select the image created.
6. Click Apply and render the scene. Your own material is displayed on the object.

Applying User-defined Decal Images

1. Create your own bitmap images, both the decal stencil and the decal artwork.

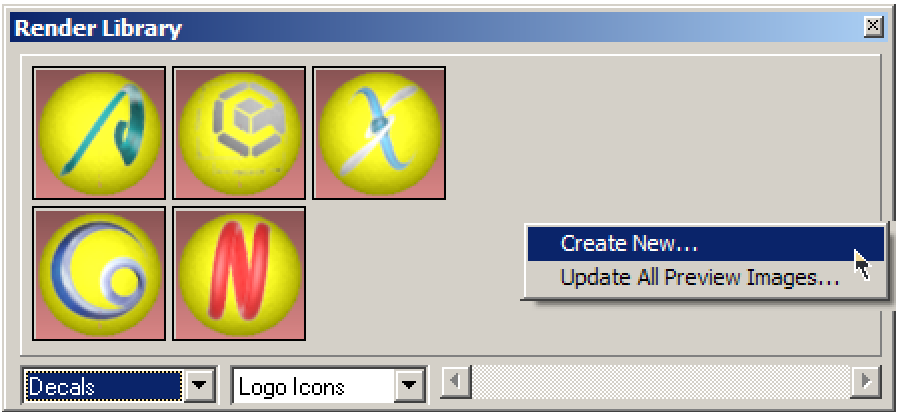
   Make sure to follow the stencil and artwork formulas as directed earlier in this chapter.
2. Place the stencils and image in the Textures folder located within the PhotoRender folder stored with the program.
3. Double-click on the decal to display the Edit Objects dialog box.
4. From the Stencil menu in Edit Objects select the stencil image created.
5. In the Logo menu choose the Image option.
6. From the Image menu, select the artwork image created.
7. Click Apply and render the scene.

   Your own decal is displayed in the scene.

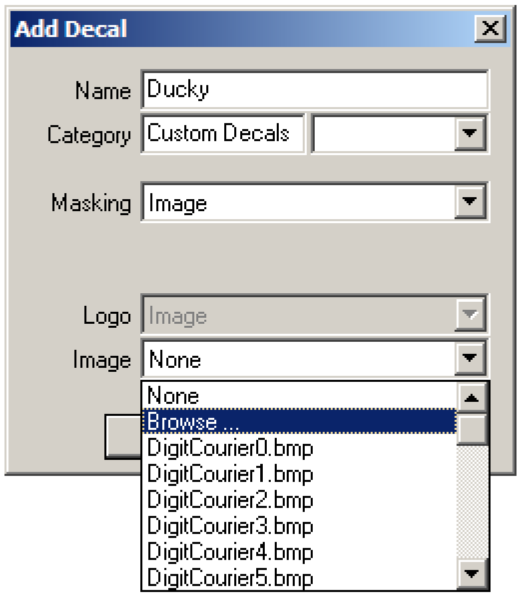
Custom Decals From Alpha Images

It is possible to create custom decals. To accomplish this:

1. Go to ***Window>Render Library***.



2. Right-click in the gray space of the Render Library dialog box to bring up the context menu and choose the Create New option. The Add Decal dialog box appears.

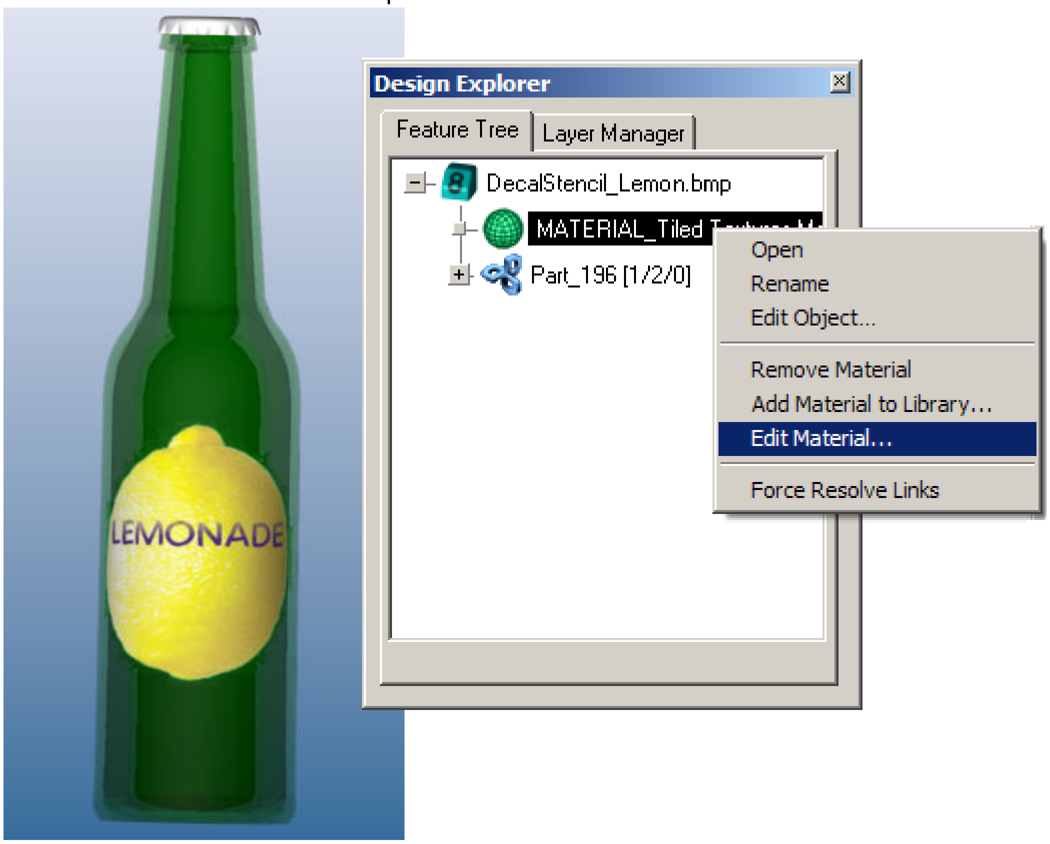


3. Fill in the Name and Category fields or choose the Category from the drop down menu on the right.
4. Choose Image from the Masking drop down menu.
5. Specify the image file name from the Image drop down menu or browse the system.

   To browse the image, navigate to the file location in the Open window, select the file and press Open. The system confirms that the file has been successfully copied. Click OK in the window. Also click OK in the Add Decal dialog.
6. The custom decal is created with the desired file. The newly created decal can be used in the same way as all the other decals from the library.

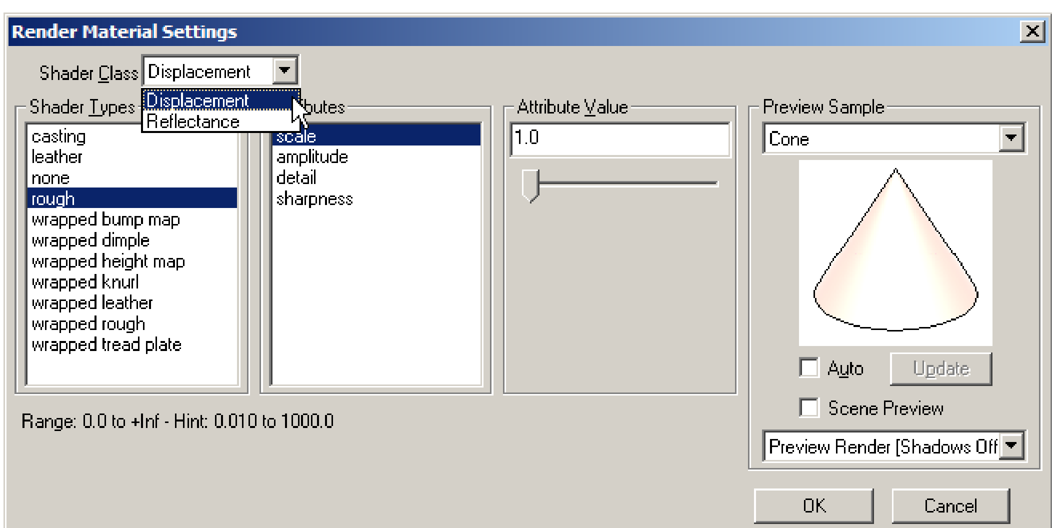
Decal Material Options

Materials can be applied to decals as well as to the objects. The picture shows a label on a bottle, which was applied as a decal, and a material was applied to the decal. So the decal has its own material options.



To edit the decal material, right click on the Material in the Design Explorer's Feature Tree.

Displacement and Reflectance options are available for decal materials in the Render Material Settings.



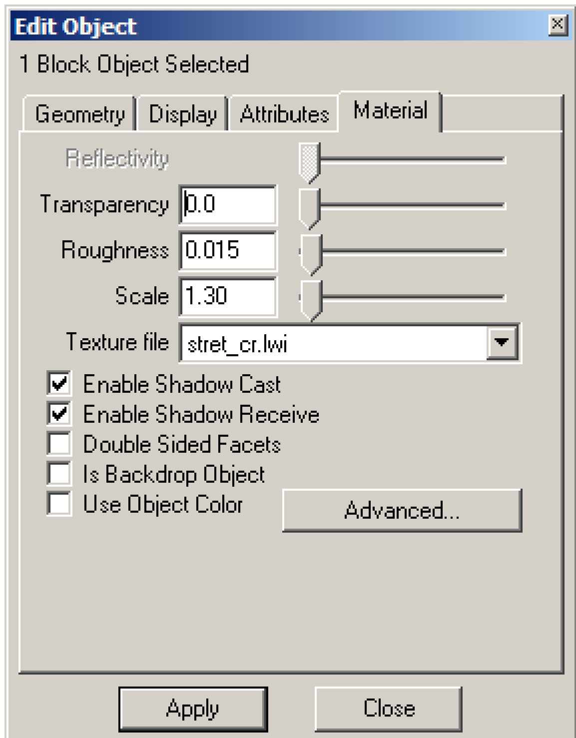
Editing the Rendered Scene

This Designer Elements program now gives the access to Lightworks shader technology. Instead of one shader, there are now five different shaders to render the scene. Change the object materials or the background and foreground images.

Once materials are applied to the objects and a background or foreground are applied if desired, it is possible to edit them.

Material Editing

Cobalt, Argon and Xenon provides a wealth of pre-defined render materials. Sometimes though, it is necessary to edit these materials or perhaps start from scratch to create your own. 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.



***Tip:** When using materials such as polished metal setting the reflectivity to a value less than one may produce a better rendering.*

The Material page of the Edit Objects dialog box include the following characteristics or options:

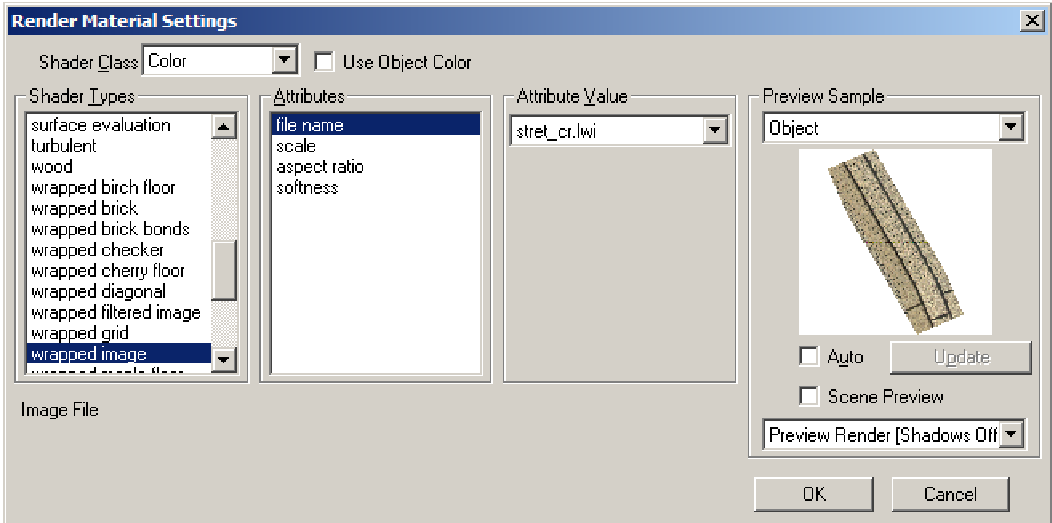
Reflectivity This field sets the material’s reflectivity. Values can be entered between zero (0) and one (1). Entering a zero in the field renders a flat finish. Entering a one in the field renders a mirrored finish.

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 specifies 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 specifies 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 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 optimizes 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 clicking 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. There are 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 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 sets 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.

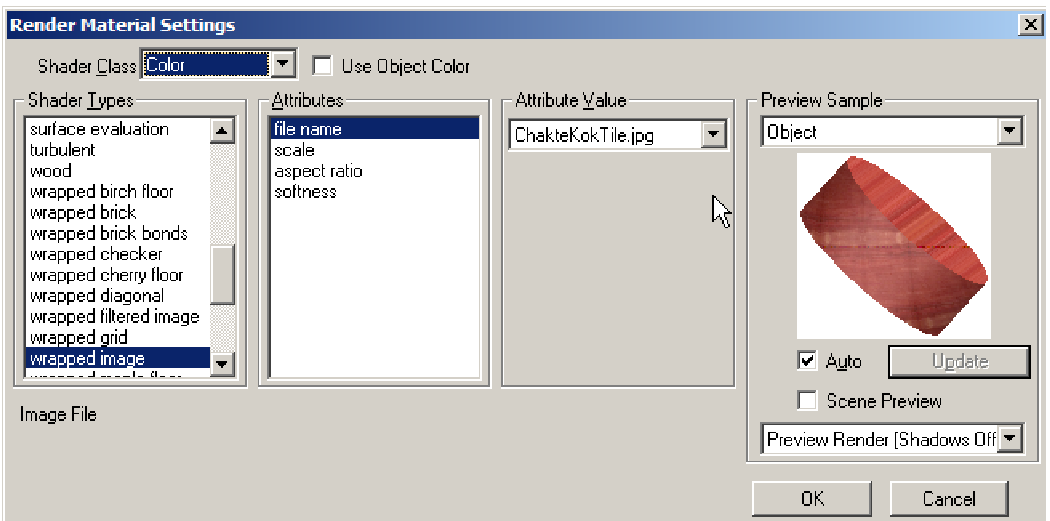
The Update button controls the manual refresh of the preview window. Click the button to refresh the preview window image after making a material setting change.

Range This area, at the lower left edge of the dialog box, displays the value range for the selected attribute.

Auto Check this to view material setting changes display in real time right away after the settings are changed.

Update Updates the view of the object. If Scene Preview is unchecked, the material changes are displayed in Render Material Settings dialogue box, if Scene Preview is checked the changes display in the main scene.

Scene Preview Real time photo-realistic material changes display in the Render Material Settings dialogue box if the Scene Preview box is not checked.



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 specifying 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.

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 Geometric Properties for information of surface and draft evaluation. Also see Surface Evaluation and 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 defines 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 F: Shader Attribute Definitions.

Tech Note: *Enter values in the attribute value field and change to another shade type, those values are not retained. To experiment with various settings, record the values before changing shader types.*

Displacement Class

Use this class to define the roughness of an object. The displacement shader types specifies 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 F: Shader Attribute Definitions.

> ***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 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 defines the decal and base color reflectance.

All shader attributes are defined in Appendix F: Shader Attribute Definitions.

Transparency Class

Use this class to define the transparency of the object. This shader types 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 F: Shader Attribute Definitions.

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 specifies 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 F: Shader Attribute Definitions.

Using the Render Material Settings Dialog Box to Edit Your Object

1. Double click on the object being modified to open the Edit Objects dialog box.

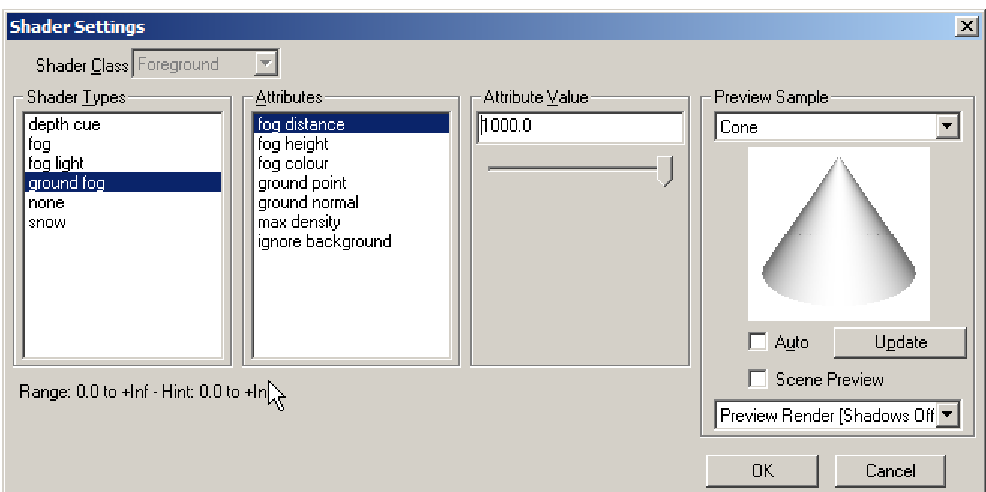
   Remember that a material must already have been applied to the object.
2. Select the Material page.
3. Click the Advanced button to display the Render Material Settings dialog box.
4. Select the desired settings for each shader class, as desired.
5. Preview the object using the Update button or the *Auto* option.
6. Click OK, when the settings are satisfactory, to close the dialog box and save the settings.

   Another object can be selected for editing 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 Eye Dropper Tool for information on the **Eye Dropper** tool.
7. Render the scene with one of the photorealistic commands. The objects now appear based on the settings specified for each object.

   See Photo-realistic Rendering Commands for information on using them.

Background and Foreground Editing

To edit the background or foreground appearance already applied to the scene, choose ***PhotoRender>Edit Background*** or ***Foreground***. (If a background or foreground is not applied to the 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 Material Editing 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 specified color.

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 mixes 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 the 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, having 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 the 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.

Photo-realistic Environment Maps for Stills and Animations

A shiny object in a real world scene will exhibit reflections on its surface. These relate to a phenomenon called specular reflection, which occurs when light energy on the surface is reflected immediately, without being absorbed by the material. There are two categories of specular reflections. The first is known as incoherent reflection and manifests itself as highlights. Highlights are specular reflections of light sources and appear as bright regions when they are reflected in a surface. The second category is known as coherent reflection, and appear as mirror-like reflections of other objects within the scene. An extreme example of this is a wall mirror, but the same property is exhibited by many types of surfaces, such as machined metals, chrome, and polished floors and tables.

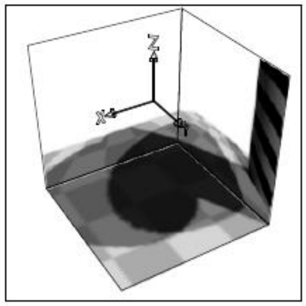
In computer graphics, reflection mapping is an efficient method of simulating a complex mirroring surface using a predefined texture image. The texture is used to store the image of the environment surrounding the rendered object. There are several ways of storing the surrounding environment. The most common ones store a single texture containing the image of the surrounding as reflected on a mirrored ball.

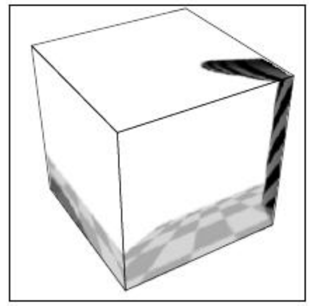
Environment mapping is a method of quickly calculating reflections. It is similar to ray tracing, however, ray tracing is much more accurate since everything in the room must be modeled. Environment mapping, on the other hand, substitutes a picture so that items are reflected on the model for visual effect. Photo-realistic environment maps are a handy tool when the details of the reflection are less important.

Referral: *Real-time environment maps, as opposed to photo-realistic ones, are also available for surface analysis of objects. See Surface Evaluation for details.*

Seven photo-realistic environment map styles are included in Cobalt, Xenon and Argon. They include:

Cube A cubic environment map at infinite distance with six independent images.

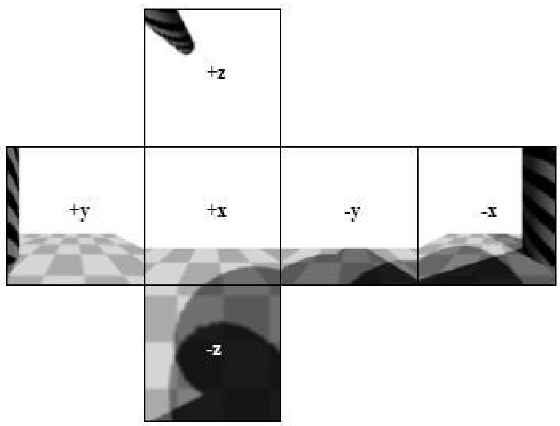




Fixed Cube A cubic environment map at a fixed distance and location from the model, composed of six images.

Cross A cubic environment map at infinite distance using one composite image assembled in a photo editing program.

Fixed Cross A cubic environment map at fixed distance and location using one composited image.



An Unfolded Environment Cube

Spherical A spherical environment map at infinite distance using one fisheye image of 180°.

Latlong A panoramic environment map at infinite distance using a 360° panoramic image.

Using a Photo-realistic Environment Map

There are two ways to apply a photo-realistic environment map.

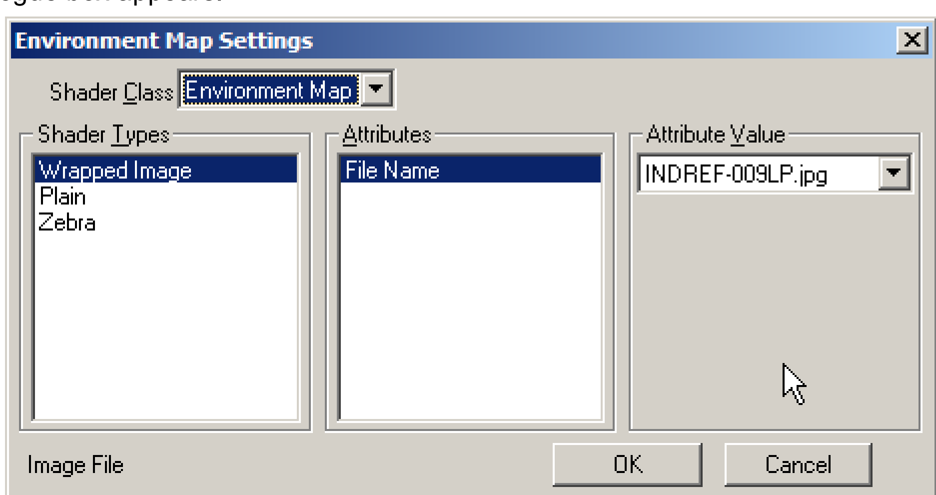
To apply a reflection map to a rendering:

1. Go to ***Window>Render Library*** and select Environment Map from the drop down menu of the Render Library dialogue box. Then specify the type of the environment map.
2. The system queries whether or not to apply an environment background.
3. Choose Yes for the environment box to appear around the object that reflects it.

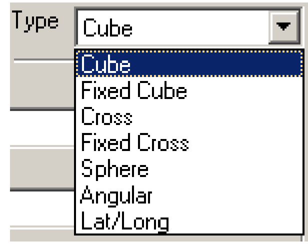
   Choose No for the object to reflect the surrounding environment while the environment stays invisible.
4. Check the Use Environment Map option in the PhotoRender Menu.
5. Apply the environment map by choosing one of the first nine options from the Photo-Render Menu.

Alternatively:

1. Use the ***PhotoRender>Edit Photorender Environment Map*** command. The dia-logue box appears.



2. Choose the type of the environment map from the drop down menu.

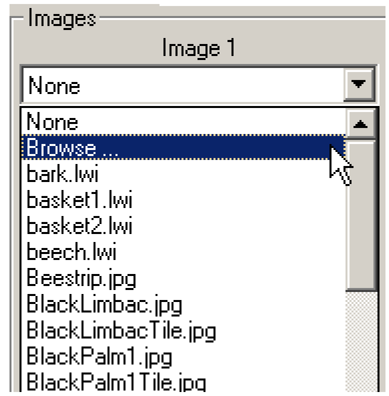


3. Select the image(s) for the environment map.

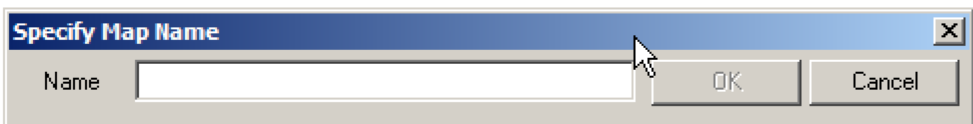
   For most environment map types only one image can be specified.

   If Cube is chosen, all the six image drop downs can be specified. Use the Browse option in the drop down menu to locate your own image.

   If Fixed Cube is chosen, all the six images, plus the center point and size of the environment map can be specified. Check Auto to let the program set the environment map parameters.



4. Click the Add to Render Library button to apply. Specify the name of the map to add to the render library and click OK.



After designating environment map settings click OK.

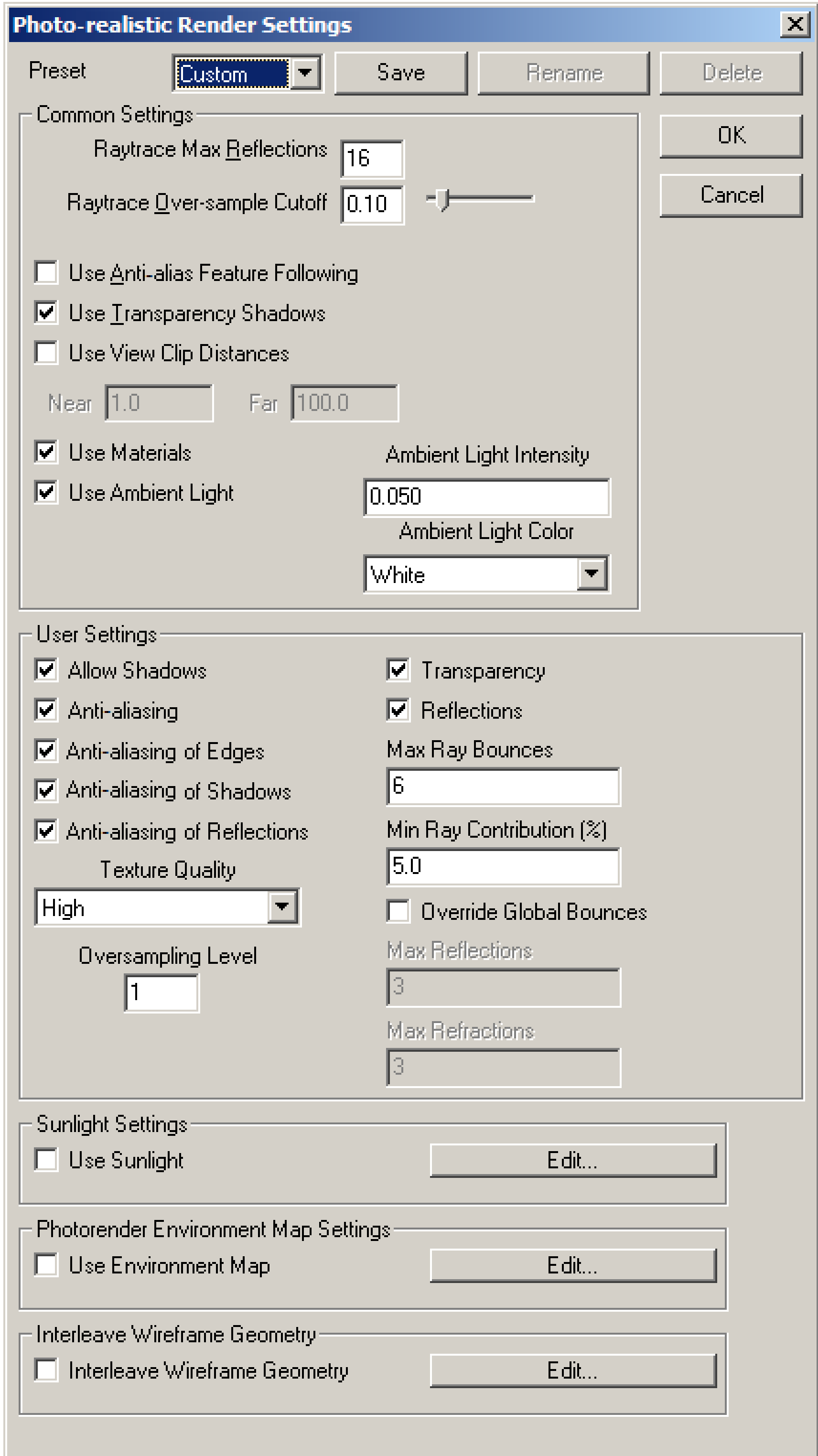
Photo-realistic Images

Cobalt, Xenon and Argon provide advanced rendering tools that control the definition and generation of photo-realistic images, including ray-tracing. 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. Discussions in this chapter include:

- Advanced Rendering Settings
- Interleaved Wireframe Geometry
- Raytrace Rendering

Advanced Rendering Settings

Use ***PhotoRender>Advanced Settings*** to bring up the Photo-realistic Render Settings dialog box to designate custom settings. User Settings are used only when selecting Render (User Settings) from the PhotoRender menu. Common Settings and Sunlight settings influence rendering when any other option from the PhotoRender menu is chosen.



Advanced settings include:

Allow Shadows	Enables or disables shadows.
Anti-aliasing	Enables or disables all forms of anti-aliasing. Checking this box accesses the advanced anti-aliasing settings including those for edges, shadows, reflections and texture quality. The default value is unchecked, meaning that no anti-aliasing is performed.
Anti-aliasing of Edges	Anti-aliases the edges of geometry when rendered. The default is unchecked. Note that if the overall Anti-aliasing box is not checked, then checking this will have no effect.
Anti-aliasing of Shadows	Anti-aliases the edges of shadows when rendered. Setting this provides the most complete form of anti-aliasing, including the edges of reflected or refracted geometry. Note that checking this will override all other anti-aliasing settings (except the overall Anti-aliasing control).
Anti-aliasing of Reflections	Anti-aliases reflected and refracted textures when rendered. The default value is unchecked. Note that if the overall Anti-aliasing box is unchecked, then modifying this will have no effect.
Texture Quality	When set to high, anti-aliasing is enabled for textures and material shaders which support anti-aliasing. The default setting is low, which means that no texture anti-aliasing takes place.
Oversampling Level	Sets the Oversampling Level which applies very simple anti-aliasing to the rendering if required. Setting a value greater than 1 results in over-sampling of each pixel. The color of each pixel is sampled many times and then an average color calculated. There is a noticeable effect on rendering speed if this value is increased. The valid range is 1 to 16, the default being 1 with no oversampling performed. Note that if the Anti-aliasing box is unchecked, then modifying this value will have no effect. Also if Anti-alias Shadows is on, it overrides this setting.
Transparency	Enables or disables transparency.
Reflections	Enables or disables ray-traced reflections.
Max Ray Bounces	Controls the number of ray trace bounces used during rendering. The smaller the number the faster ray tracing will be for complex scenes. If the value is set to be too low then effects, such as multiple reflections, may not be rendered correctly. Ashlar-Vellum recommends starting with the default setting of 6 and adjusting it from there.
Min Ray Contribution	Controls the minimum contribution made by a secondary ray (reflected, refracted or alpha ray) before it is ignored. A value of 5 means that if the traced ray would contribute less than 5 percent to the total illumination at the pixel of interest then its contribution is ignored.

Override Global Bounces Enables or disables the following two controls. If checked it allows control of maximum reflections and refractions. If left unchecked then all ray depths are the same as Max Ray Bounces.

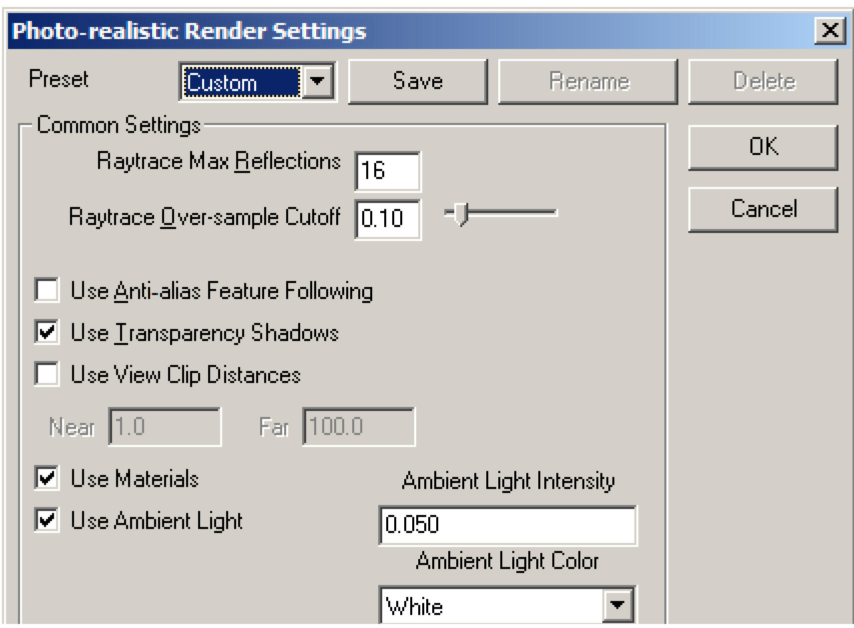
Max Reflections Controls how many levels of reflection are considered during renders. The lower the number of reflections, the faster ray tracing will be for complex scenes. If the value is set to be too low then any multiple reflections may not be rendered correctly.

Max Refractions Is the same as Max Reflections, but applies to refracted or alpha rays.

Tech Note: *The pre-set options accessed from the PhotoRender menu use a value of 10 for ray bounces, reflections and refractions.*

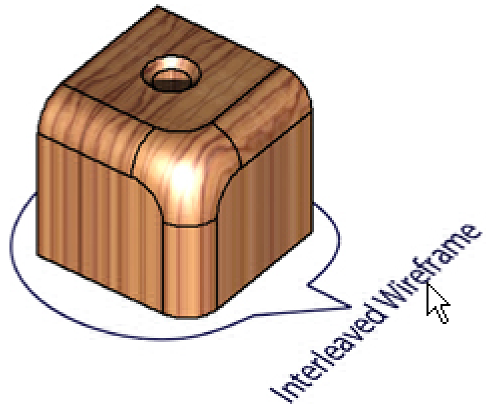
Custom Render Presets

Advanced Settings can be changed as necessary. Go to ***PhotoRender>Advanced Settings***. In the Advanced Settings dialog box modify the settings as desired, press the Save button, then type in a new name for this group of settings.



The new settings will be available in the Preset drop down menu of the Photo-realistic Render Settings dialog and also in the Shade Now option in the Shade Options dialog. The objects can be rendered only in the Render [User Settings] mode of the PhotoRender menu.

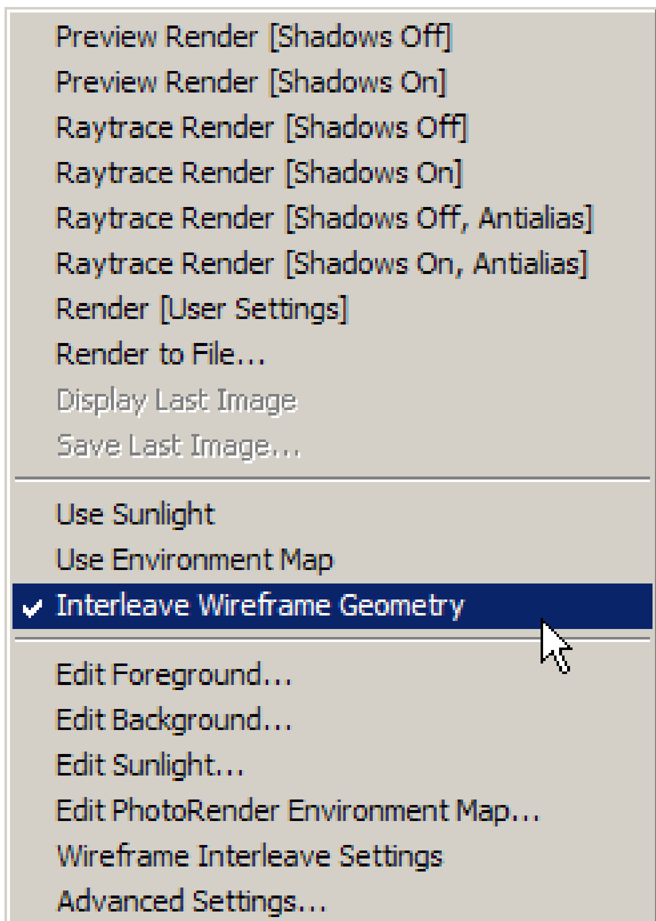
Interleaved Wireframe Geometry

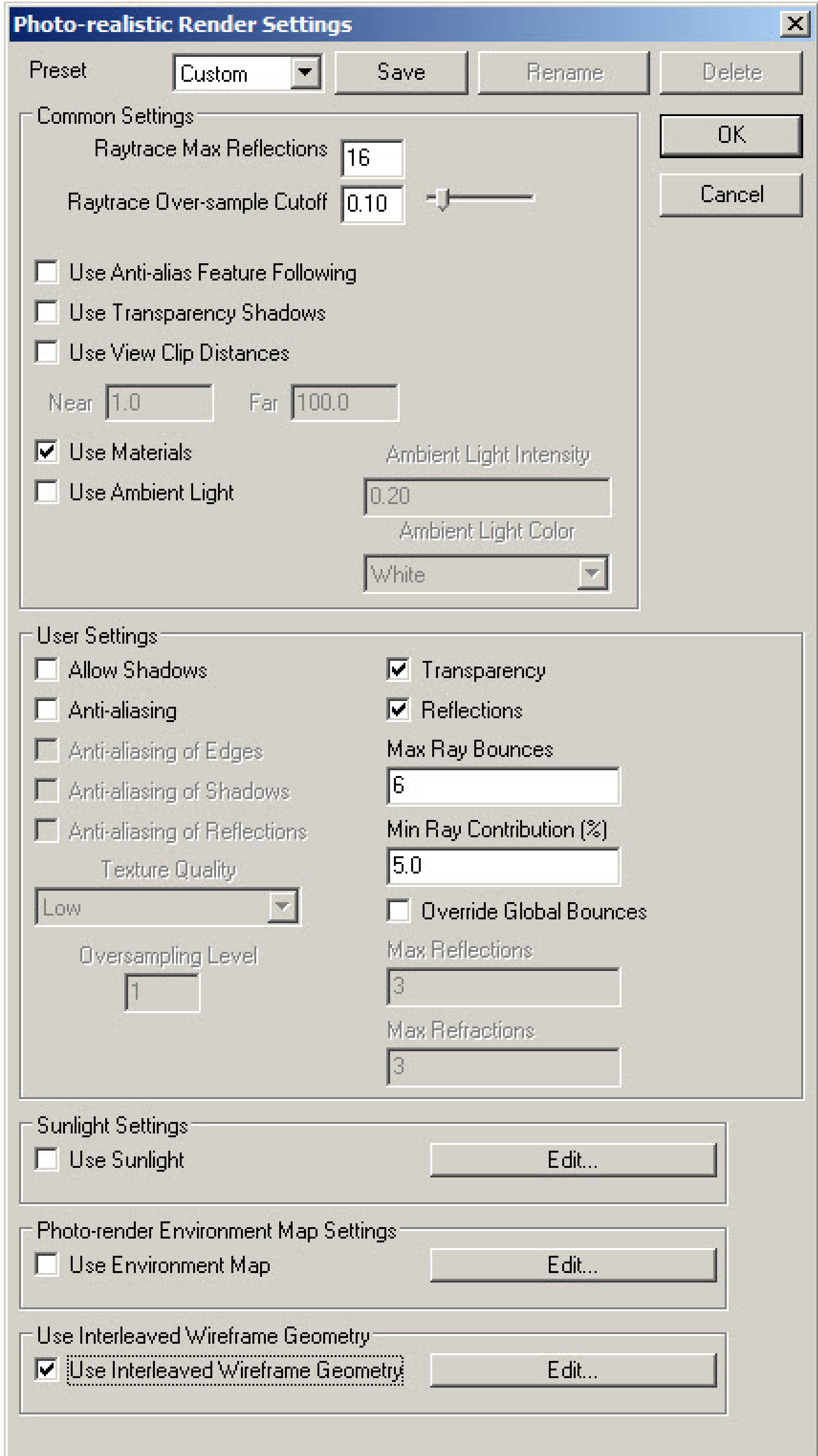


It is possible to interleave 3D wireframe geometry in photo-renderings including all types of edges, curves, dimensions and text.

To enable interleaved wireframe geometry:

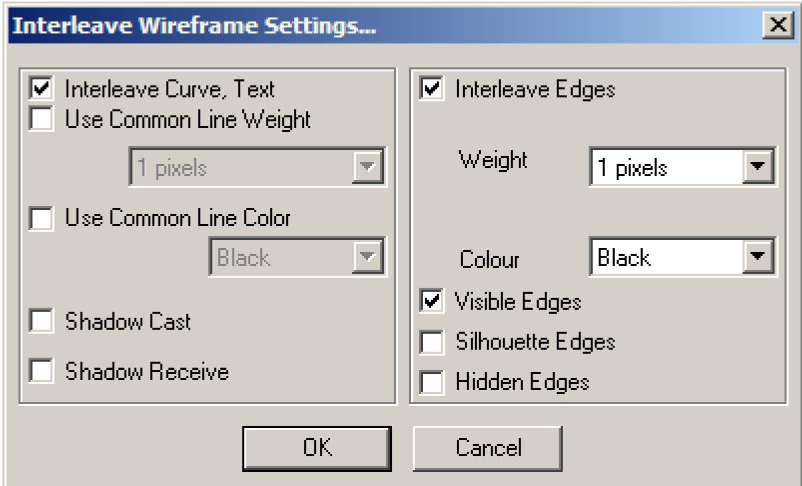
1. Check the Use Interleave Wireframe Geometry option in the PhotoRender menu or in the Photorealistic Render Settings window.





2. Select the Edit Interleaved Wireframe Geometry Settings option in the PhotoRender menu or press the Edit button next to the Interleave Wireframe Geometry option in the Photo-realistic Render Settings dialog.

3. The Interleave Wireframe Settings dialog box appears.



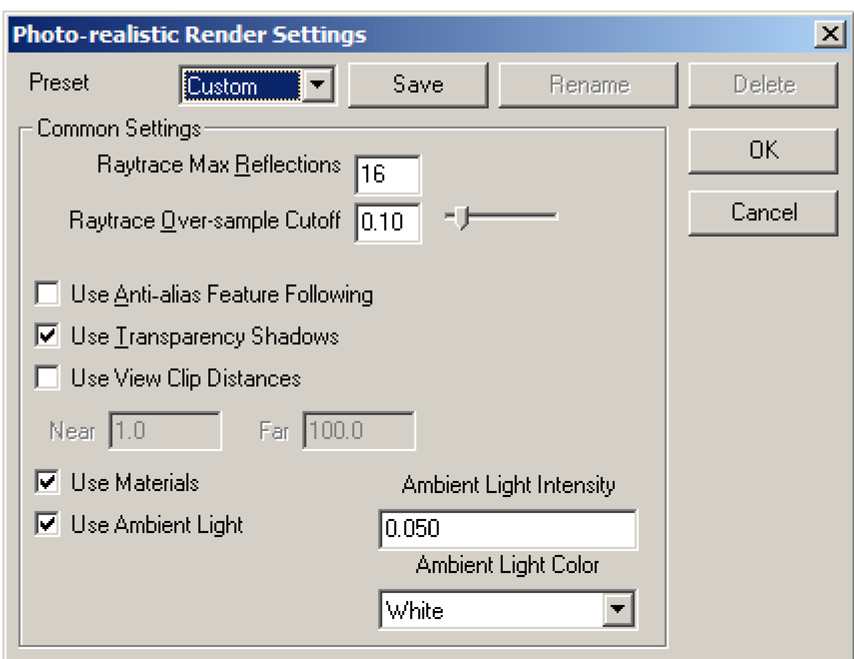
In this window separately toggle On/Off rendering for wireframe geometry and object edges (Render Wireframe and Render Edges check-boxes). Also set color and weight properties for wireframe rendering. If the Use Common Line Weight option is checked then PhotoRender uses the selected weight for all wireframe objects rendering. Otherwise PhotoRender uses object weight for rendering. This box also sets rendering color options by checking the Use Common Line Color check-box and specifying a color.

Raytrace Rendering

This Designer Elements program photo-realistically renders the geometry. Specify rendering settings and choose among various commands before rendering the scene.

Photo-realistic Rendering Settings

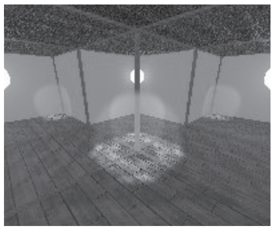
Change the low-level behavior of the rendering engine through the Photo-realistic Render Settings dialog box. Choose ***PhotoRender>Advanced Settings*** to display the dialog box.



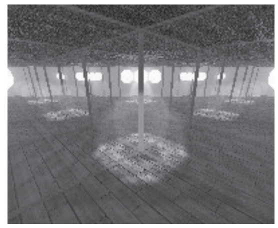
The dialog box contains the following options:

Raytrace Max Reflections

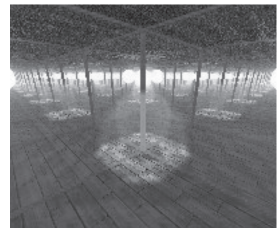
Controls the maximum number of bounces a ray travels. Once a ray has reached the specified limit no further color calculations occur for that ray. The images of a mirror-walled room, shown here, demonstrate the effect of changes in the value.



Max Reflections = 2



Max Reflections = 6



Max Reflections = 16

Raytrace Over-Sample Cutoff

Controls the threshold for adaptive image over-sampling. When the rendering operation encounters a significant color change in the 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.

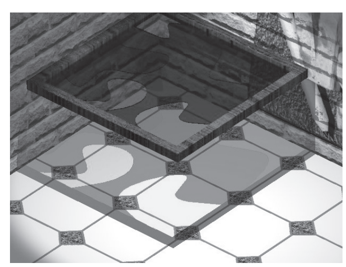
Use Anti-Alias Feature Following

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

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.

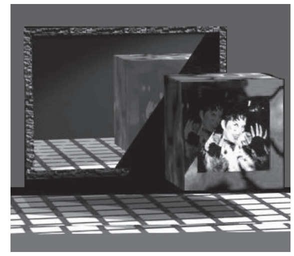


Use View Clip Distances

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. 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.

Use Materials Toggles On/Off the material applied to each object.

Use Ambient Light Enables ambient light and allows setting its intensity and color.

OK Saves the new settings and closes the dialog box.

Cancel Exits the dialog box without saving the settings.

Defaults Returns the settings to the factory state.

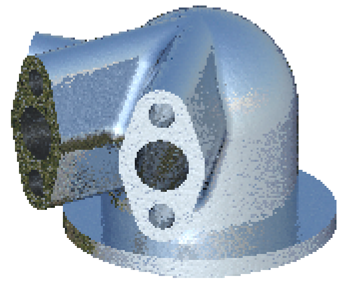
Photo-realistic Rendering Commands

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)*.

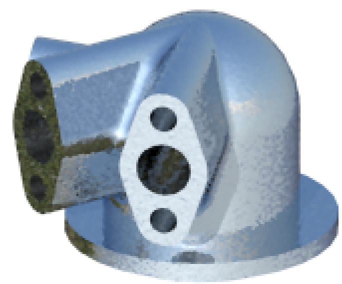
Preview Render (shadows off) Creates a quality rendering of the scene. It renders most rapidly of all the commands since there are no ray tracing and shadow calculations.

Preview Render (shadows on) Creates a more realistic rendering than Preview Render (shadows off) with shadows. It renders quickly since there is no ray tracing.

Raytrace Render (shadows on/off)	Creates a high quality rendering of the scene. The processing time is extended with addition of the ray tracing operation and shadow calculations.



Raytrace Render (shadows on/off, anti-alias)	Creates the highest quality rendering of the 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.

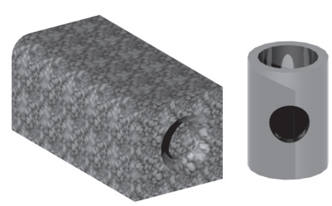
The *DefaultVSMFile* entry in the *[RenderOptions]* section of the *Render.ini* file controls the default material. If the *DefaultVSMFile* entry is not found, the material specified by the *MiscPlain* material is used.

For more information on shader types, see the Material Editing.

Rendering the Geometry

Cobalt, Xenon and Argon all render the entire scene or a specified area.

Rendering the Entire Scene

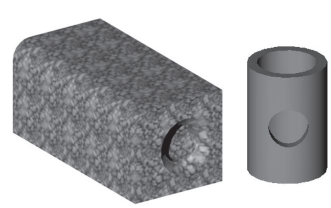


This Designer Elements program renders the entire scene with one command, showing how the applied materials appear on the objects. The graphic is rendered using the *Raytrace Render (shadows on, Anti-Alias).*

1. Apply materials to the objects in the scene.
2. Choose ***PhotoRender>*** and the command.

   The scene renders.

Rendering an Area



When applying materials to the objects, it is possible to see the results from one specific area of the scene. The area render feature gives that ability. Using the CTRL (Windows) or OPTION (Macintosh) key, specify an area to render. In the graphic only the area around the left object is rendered.

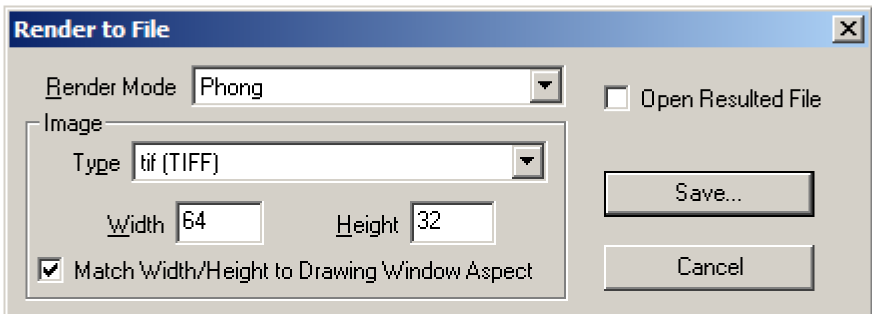
1. Apply materials to objects in the scene.
2. Hold down the CTRL (Windows) or OPTION (Macintosh) key and from the Photo-Render menu, select one of the photorealistic commands.

   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.

Render to File

In addition to the commands for rendering the scene to a window, render the image to a file. Choose ***PhotoRender>Render to File*** to display the dialog box.



Tech Note: *This Designer Elements program does not support directly printing a photo-realistic image. Use the Render to File command on the image, open the file in a graphics application and print.*

The dialog box contains the following options:

Render Mode Specifies the render mode used in creating the file. Phong and Gouraun shading options can be used when rendering to a file.

Image Type This section provides 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 Specifies the image size in pixels. To have 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 Specifies 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, enter any value.

Open Resulting File Opens the resulting file when checked.

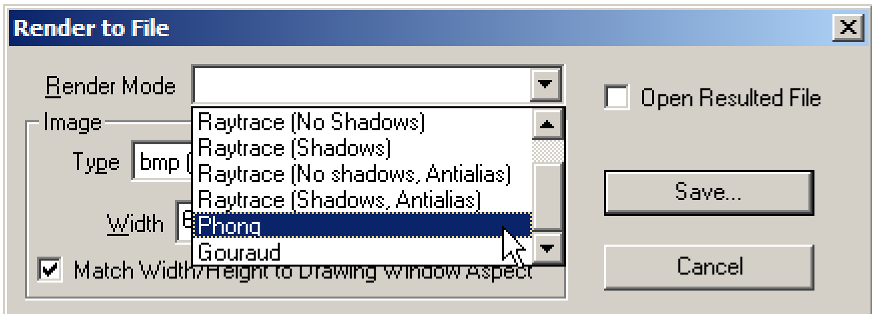
Save Saves the file and closes the dialog box.

Cancel Exits the dialog box without creating a file.

Creating Image Files

This Designer Elements program creates publication quality images with higher resolution images than produced on a typical computer monitor.

1. Create the drawing and apply the render materials to the objects.
2. Choose ***PhotoRender>Render to File***.
3. Choose the render mode from the pull-down menu. Phong and Gouraud shading options can be used when rendering to a file.
4. Choose the image file format.



5. Enter the desired width and height values. If the Match Width/Height to Drawing Window Aspect box is checked, only enter a value in one field.
6. Check Open Resulted File if necessary to view it at once.
7. Click Save to create the file.

Display Last Image Command

Use this command in the Render menu to display the previously rendered scene without having to render the scene again. This command is helpful if a dialog box appears or the mouse is inadvertently clicked and the rendered image is lost. This command saves time if the scene is rendered with one of the photorealistic commands.

Note: If the geometry is rotated after rendering the scene and then this command is used, the image that appears reflects the last rendering and not the current drawing orientation.

Rendering and Multi-processors

If your computer uses multi-processors and an image is rendered with one of the photo-realistic commands, this Designer Elements program will make use of those processors to separate the image into pieces and thus improve the rendering time.

Animation

Cobalt, Xenon and Argon can create five types of movies. The animation features are available through the Animation menu.

Of the five movies types, the first three, *Walk-through, Fly-by* and *Paths,* specify a path for the camera movement through a scene and require that QuickTime be installed on the Windows or Macintosh computer. The last two, *Object VR* and *Panoramic VR,* 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 *Path* animations. They cannot be used for *Object VR* and *Panoramic VR* animations.

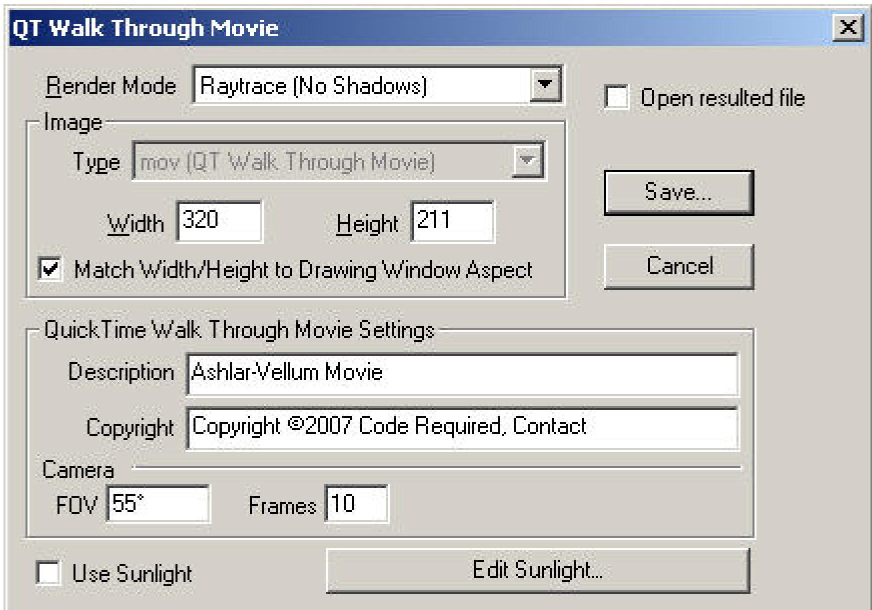
QuickTime Movies with Camera Movement

The *Walk-through*, *Fly-by* and *Path* movies define the movement of the camera using a curve. Proceeding 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 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 sets the render mode for the movie. There are eight options: Preview (shadows off), Preview (shadows on), Raytrace (No Shadows) and Raytrace (Shadows), Raytrace (No Shadows, Anti-alias), Raytrace (Shadows on, Anti-Alias), Phong and Gourund.

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 the animation type is chosen.

Image Width/Height
These data fields specify the image size in pixels. To have 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, any value can be entered.

Description
This field contains the movie description which displays when the movie is viewed. Enter the movie description.

Copyright
This field contains the movie copyright which displays when the movie is viewed. Enter the 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 to be generated for the movie. The default number is 10.

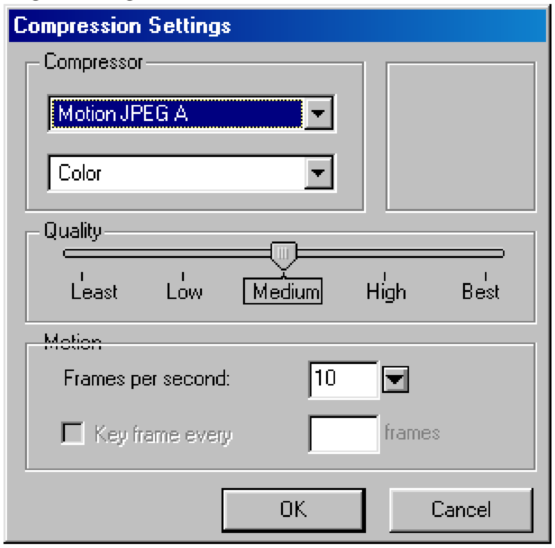
Save
Click this button to save the file and close the dialog box.

Use Sunlight Check this box to include sunlight settings.

Edit Sunlight Click here for the Sunlight Settings dialog box. See “Photo-realistic Sunlight for Stills and Animations” on Lighting for more information about sunlight settings.

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 sets 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 A color option can be chosen from the pull-down menu. The options vary according to the compressor type chosen.

Quality This slider sets the movie quality.

Motion This section includes the *Frames per second* data field and the *key frame every x frames* check box/data field.

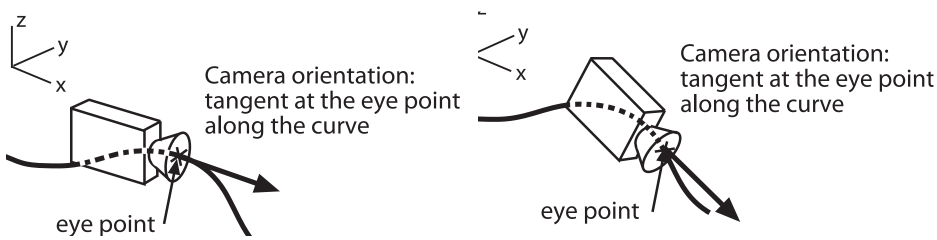
Frames per second: Enter the number of frames per second or use the pull-down menu to specify the number.

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 it is necessary later to sync sound to the movie track, a key frame provides some control. For example: to 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 syncs a sound track to be 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.

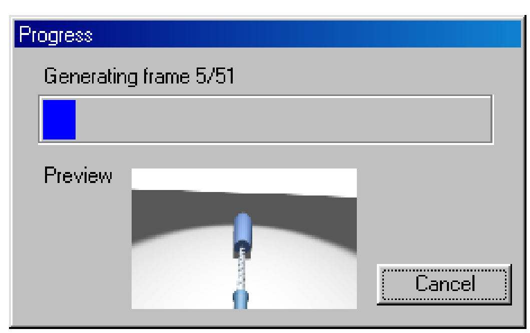


Each movie frame is taken at a particular location on the curve based on the number of frames. If a curve slopes down, the camera follows the tangent and slopes down as well, resulting in the eye looking down at the angle specified by the tangent. The up vector of the camera is always the z axis.

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 the settings.
5. Click Save. The Save As dialog box appears.
6. Type in the movie name and navigate to the location where to save the file.
7. Click Save. The Compression Settings dialog box appears.
8. Specify the settings for the compressor.
9. Click OK. A progress dialog box appears providing a preview movie window and information on the number of frames generated.

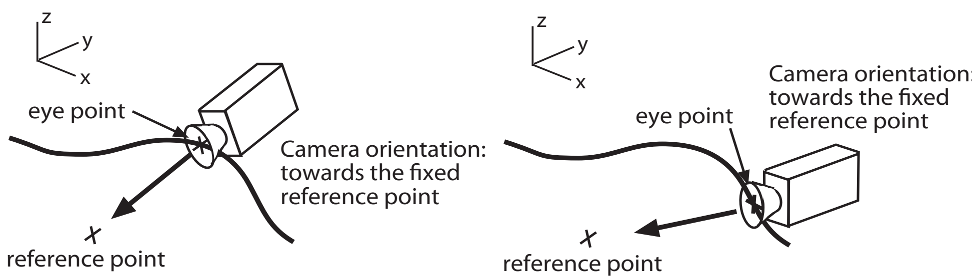
   

   The Message Line provides information on the pass, time elapsed and time remaining to generate the movie.

   The operation can be clicked to end before it's complete. The program will finish generating the current frame and write out the movie at that frame. Once the movie is generated it is not possible to add more frames to it. The dialog box disappears when the movie generation is complete.
10. It is possible now to 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 specified. 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 the settings.
6. Click Save. The Save As dialog box appears.
7. Type in the movie name and navigate to the location where to save the file.
8. Click Save. The Compression Setting dialog box appears.
9. Click OK. A progress dialog box appears providing a preview movie window and information on the number of frames generated.

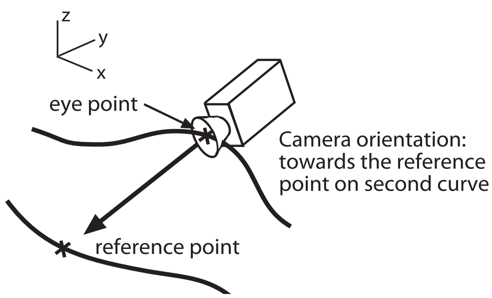
   The Message Line provides information on the pass, time elapsed and time remaining to generate the movie.

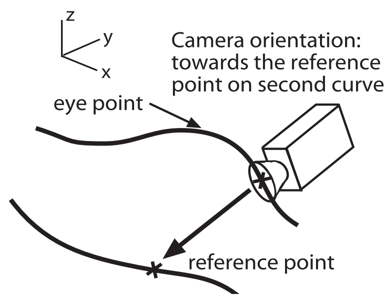
   It is possible to click to end the operation before it's complete. The program will finish generating the current frame and write out the movie at that frame. Once the movie is generated it is not possible to add more frames to it. The dialog box disappears when the movie generation is complete.
10. It is possible now to view the movie using QuickTime player.

Path Animation

This animation command creates a path 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 Path 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 the settings.
6. Click Save. The Save document As dialog box appears.
7. Type in the movie name and navigate to the location where to save the file.
8. Click Save. The Compression Setting dialog box appears.
9. Click OK. A progress dialog box appears providing a preview movie window and information on the number of frames generated.

   The Message Line provides information on the pass, time elapsed and time remaining to generate the movie.

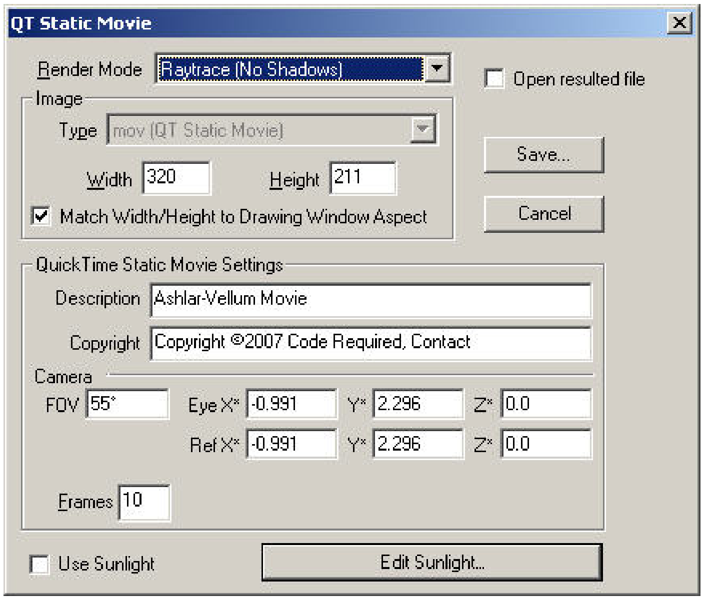
   It is possible to click to end the operation before it's complete. The program will finish generating the current frame and write out the movie at that frame. Once the movie is generated it is not possible to add more frames to it. The dialog box disappears when the movie generation is complete.
10. It is possible now to view the movie using QuickTime player.

Backgrounds and Movies

Put in a background for *Walk-through*, *Fly-by* and *Paths* animations if necessary.

1. Create the 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 the background.
6. Choose ***PhotoRender>Edit Background***.
7. Select image in the Shader Types list.
8. From the Attribute Value pull-down menu, choose the image.
9. Click OK and render the scene.

Static Animation



QuickTime VR Movies

This Designer Elements program creates two types of VR movies: Object and Panoramic.

QuickTime Object Movie

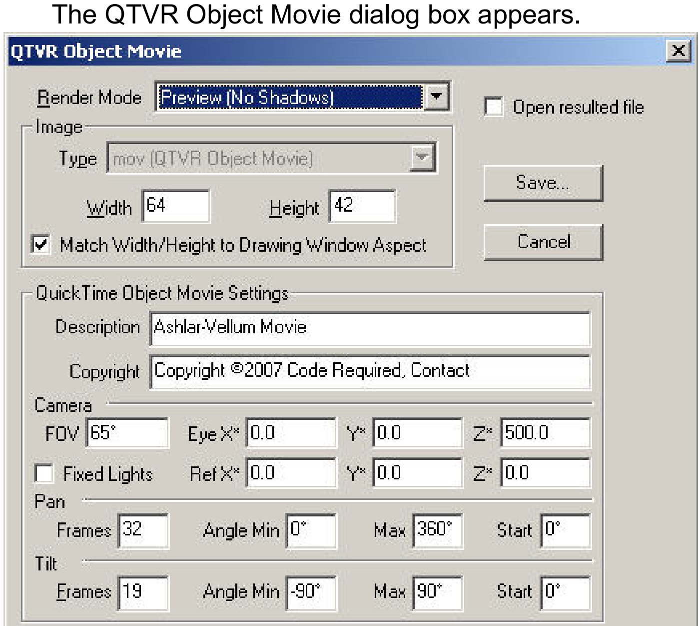
An object movie keeps the observation point fixed as the eye point is moved at a fixed distance about the observation point. This gives the visual effect of moving completely around an object on the surface of an invisible sphere.

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 the drawing and apply the render materials to the objects.
2. Choose ***Animation>Object VR***.

   The QTVR Object Movie dialog box appears.



The dialog box includes these additional options:

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 the ability to specify locations by clicking in the drawing area. The values can be entered manually.

3. Enter the desired values.
4. Click Save to create the movie.

QuickTime VR Panoramic Movie

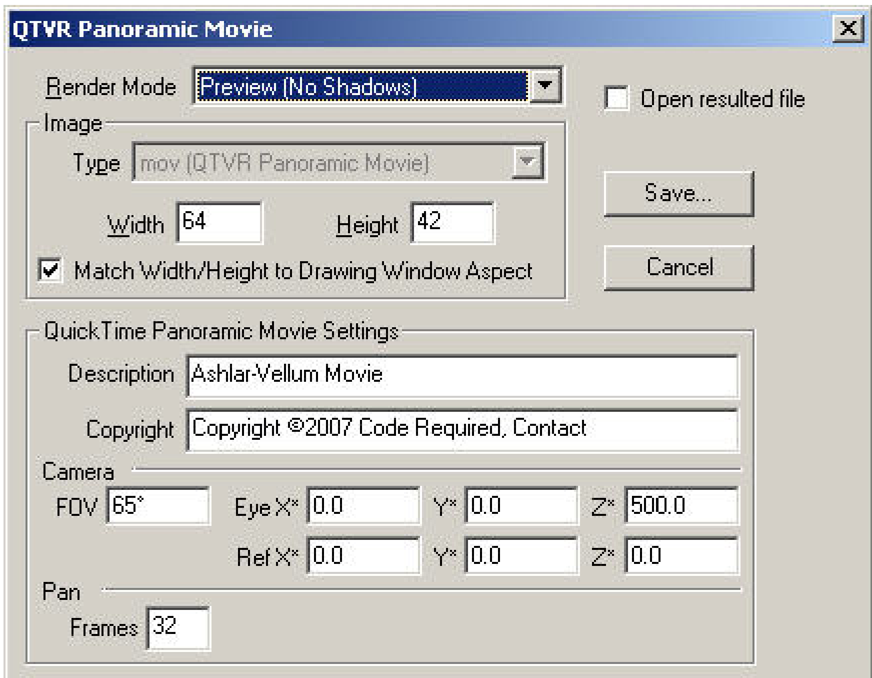
A panoramic movie keeps the eye point fixed as the observation point is rotated 360°. This gives the visual effect of turning 360° in place.

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 the drawing and apply the render materials to the objects.
2. Choose ***Animation>Panoramic VR***.

   The QTVR Panoramic Movie dialog box appears.



   The dialog box includes these additional options:

Pan — The Frames field sets the number images to be 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.

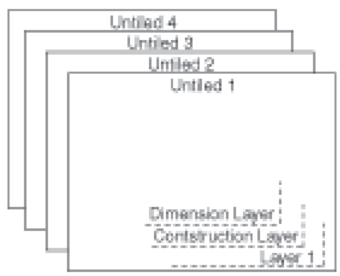
   The asterisk (*) next to the field indicates the ability to specify locations by clicking in the drawing area. The values can be entered manually.

3. Enter the desired description and values.
4. Click Save to create the movie.

File Management

Whenever either Cobalt, Xenon or Argon is opened or *New* from the File menu is chosen, a new document appears. Multiple files can be opened as desired.

Each file supports the layers feature. This feature adds a great deal of flexibility to the documents. Layers display 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 for sharing files with other people.

The following topics are covered in this chapter.

- Using Documents
- Files and Preferences
- Files and Locking
- File Size & Deleting
- Uninstalling This 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 it is saved. Saving a drawing is explained later in this chapter.

File Use and Operating System Short Cuts

Cobalt, Xenon and Argon take advantage of some features of the operating systems when working with files.

Windows

The program starts and opens files from within the *Windows Explorer* or other folders. It also supports the *Drag and Drop* functionality.

Windows Explorer and other Folders

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.

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 the program is not running, the program launches first, before the files open.

Drag and Drop

Cobalt, Xenon and Argon have the ability to manipulate files by dragging and dropping them.

Drag and drop files into this program to open the files.

- By dragging and dropping one or more files into an open drawing, they are merged into the drawing.
- By dragging and dropping one or more files into this Designer Elements program when no files are open, each opens individually.
- By holding down the SHIFT key before dropping multiple files into the program when no files are open, files are merged into a single new drawing.
- By dragging and dropping one or more files onto the program desktop icon and the files open individually with the program open.

Macintosh

Double-click on the program file to launch the program and open the file.

- By dragging and dropping one or more files into an open drawing, they are merged into the drawing.
- By holding down the SHIFT key before dropping multiple files into this program when no files are open, files are merged into a single new drawing.
- By dragging and dropping one or more files onto the Designer Elements program desktop icon. 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*.

New

CTRL+N (Windows); ⌘+N (Macintosh)

This command in the File menu creates a 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 more than one new document is opened, the subsequent documents are numbered sequentially until they are named by saving.

> ***Referral:*** *Use the* ***File>Import*** *command to use drawings of other format types, such as DXF, IGES, or SAT. See Importing for more information.*

Open

CTRL+O (Windows); ⌘+O (Macintosh)

This command in the File menu opens an existing Cobalt, Xenon or Argon document. Graphite files can also be opened using this command. A progress bar appears as the file is opening. This is especially helpful for large files.

The document appears in the drawing area maintaining the same settings as they are saved the last time.

> ***Tech Note:*** *To open a Cobalt, Xenon or Argon 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 backward compatible.*

The dialog box specifies the document and changes folders, if necessary.

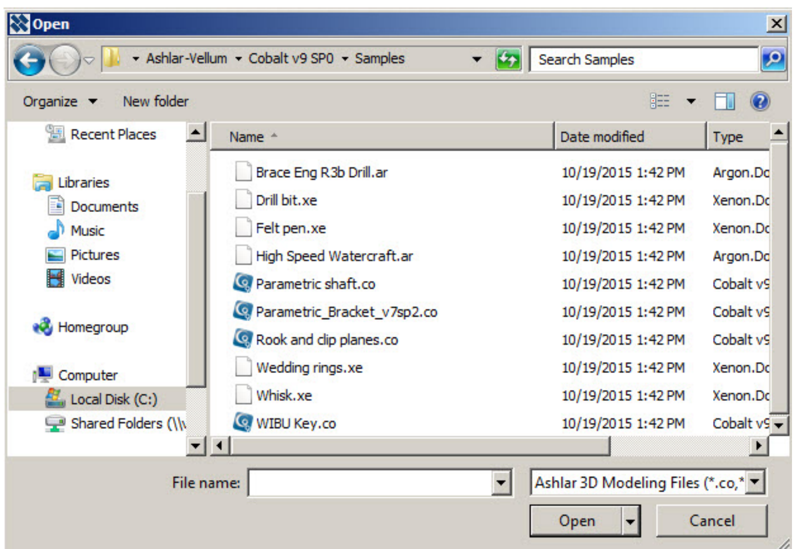
Windows - If the *Open as read only* box is checked, the file can be opened and printed but not altered. Macintosh - If a file is locked, it can be opened and printed but not altered. Unlock a file by selecting the file and choosing Get Info (Macintosh standard).

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 to open.
3. Click the File name to open in the list box.
4. Click OK.

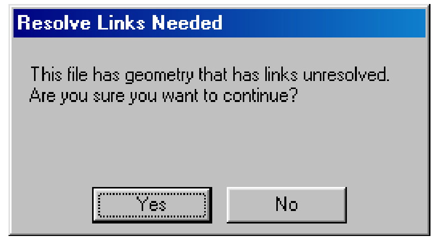
Recent File List

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 program.

The number of files displayed depends on the number selected in the Filing page of the Preferences dialog box. Up to 20 files can be displayed 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 program will ask you to locate the file by providing the standard Open dialog box.

Saving a Drawing



Save a drawing by choosing ***File>Save*** or ***Save As***. The file is stored on the computer in the specified folder.

Attempting to save a file containing links that are not resolved, the following warning appears:

Save

CTRL+S (Windows); ⌘+S (Macintosh)

This command in the File menu saves the current document to its original folder. To save it to a different folder or with a different name, choose *Save As*. If *Save* is chosen and the document has not been saved previously, the Save As dialog box appears automatically to name the document and specify the folder in which to save it.

> ***Tech Note:*** *When attempting to save a file with the name of a locked file, a warning message will appear.*
> *If objects are deleted 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.*

If no geometrical changes are made in the file, the *Save* command is not available. Changing an object's layer is not recognized as a geometrical change. If the document have been named and saved before, a brief message appears when *Save as* chosen. The program pauses while it updates the information.

Note: It is necessary to save frequently. It is important to save before performing any intricate, multistep procedure. That way, if the result is not exactly what is desired, it is possible to abandon the file by closing it without saving.

Save As

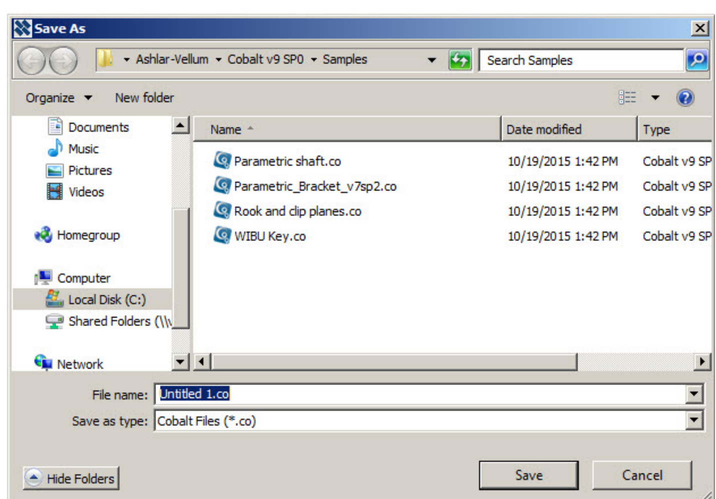
This command in the File menu saves the current document. A dialog box appears to name the current document, give it a different name, or save it to a different directory. Use the *Save As* command to make a backup of a document.

(Windows) - It is possible to save a file as Read only. 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 to use in the File name box.
4. Click OK.

   The appropriate filename extension for your software is automatically appended to the filename (.co, .xe or .ar).

Tech Note: *Macintosh users: Although the .co, .xe, ar, and .vs extension is not required for the file name, it is included to ease file transfer between Macintosh and Windows.*

Making a Backup

Make a backup of the work in case there are many changes and it is necessary to go back to the original version. Choose ***File>Save As*** and save the file with another name.

Auto Save

This command is found under ***File>Preferences>Filing*** and directs the program to save a backup copy of the work periodically. If your computer crashes for any reason, the work done up to the last Auto Save will be recoverable.

There is the option to save after a certain number of commands or minutes. It is also possible to specify the number of backup files created before reusing a backup file name. (See Auto Save for more information on setting up or using the Auto Save feature.) For Windows, files opening with the READ ONLY attribute are also autosaved.

Files saved using this feature are placed in the Backup folder within the program folder.

If *Auto Save* is set to on in Preferences but geometry is not modified 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); ⌘+F4 (Macintosh)

This command in the File menu closes the current program document. If other program documents are open, they remain open when the current document is closed. If changes have been made since the file was last saved, a dialog box appears. Close the document with or without saving the changes.

For Windows, 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 - ⌘+Q (Macintosh)

This command in the File menu closes this Designer Elements program. If there were changes since the last saving, a dialog box opens. If more than one document is open, an alert message opens to save unsaved documents.

Files and Preferences

In the *Filing* page of the Preferences dialog box, choose settings for this Designer Elements' documents. 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 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. Or save the file under the same name. While saving, the system prompts to confirm the overwriting of the locked file.*

File Size & Deleting

When objects in this Designer Elements program are deleted, the object data is still retained until a second delete operation is performed. If the 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 uninstalling this Designer Elements program, there are some folders and files left in User domains (e.g. C:\Documents and Settings\/CURRENT_USER/\Application Data\Ashlar-Vellum\Modeling V8) and Local domain (e.g. C:\Documents and Settings\All Users\Application Data\Ashlar-Vellum\Modeling v8). Those folders do not uninstall automatically since they were created after installation.

Importing & Exporting

There are various reasons to use the import and export functions. It may be necessary to bring a document in from another application to work on in this Designer Elements program. It may be necessary to save a document in a format other than this Designer Elements program format for use with another application. This Designer Elements program offers a wide variety of options to accomplish this.

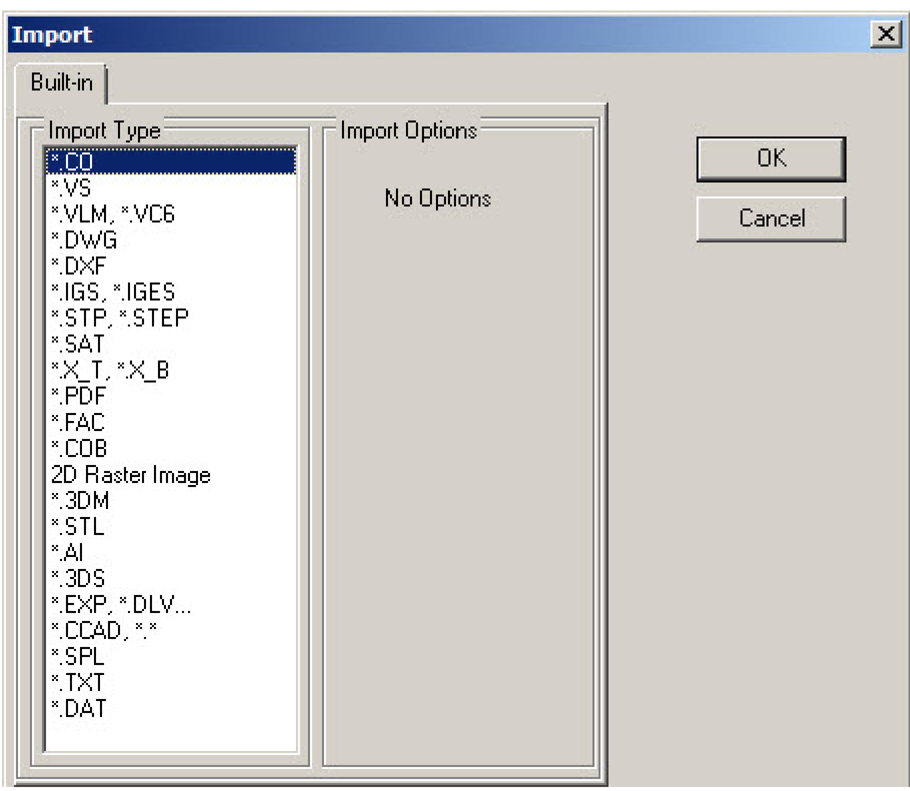
The following topics are covered in this chapter:

- Importing
- Exporting
- Batch Convert Tool

Importing

Cobalt, Xenon or Argon import many different kinds of files. Some types include import options specific to the format.

When the *Import* command is chosen, 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

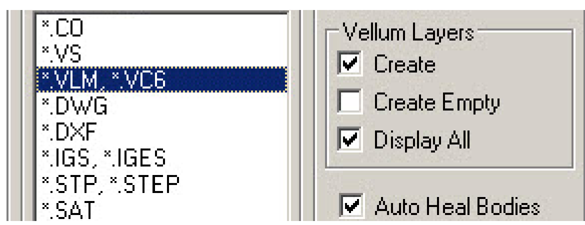
This Designer Elements program imports these file formats: Cobalt (including Xenon and Argon), Vellum Solids, Graphite/Vellum 3D, DWG, DXF, IGES, STEP, ACIS SAT, Parasolid (Windows only), Facet, Truespace COB, 2D Raster Image, Rhino 3DM, STL, Adobe Illustrator, 3D Studio, Catia v4, Spline, Text, Grid Surface, PICT (Macintosh only) , and ClarisCAD.

Vellum Solids (VS)

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, import the file using this format.

Graphite/Vellum (VC6, VLM)

Selecting this type imports files created in Graphite/Vellum 2D and 3D. There are 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, the program creates/imports the layers in the Graphite/Vellum 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, the layers that contain no geometry are also created. If this box is not checked, layers that contain no geometry are deleted.

Display All - When checked, all objects on all layers are displayed. If this box is not checked, the import file determines which layers display.

Auto Heal Bodies When checked, auto-healing is applied 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

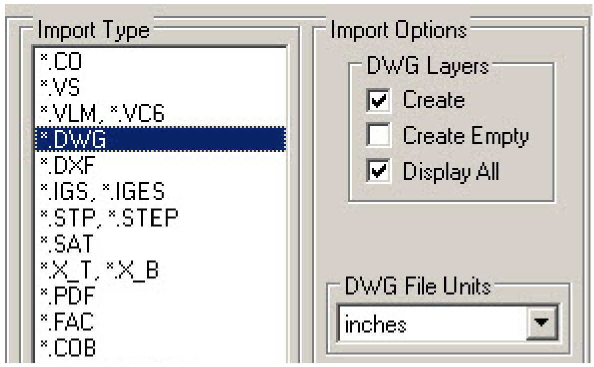
- The translator does not support the following entities: work plane, fill, bitmap, balloon, hatch, sheet and view.
- ACIS does not support skewed ellipses. Only ellipses created with the **2-Point Center Ellipse** tool and the **Opposite-Corner Ellipse** tool in Vellum 3D can be imported into Cobalt, Xenon or Argon.
- Cobalt, Xenon and Argon do not support models (model space). When importing a file with geometry in different models, each model is placed on its own layer.
- When importing a file it is necessary 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 R2008.

This translator imports horizontal, vertical, diameter and radial dimensions in addition to geometry.

DWG includes three options: *DWG Layers*, and *DWG Units*.



DWG Layers Includes three check boxes, *Create*, *Create Empty* and *Display All*.

Create - When checked, the layers in the DWG file are created/ imported and geometry placed on the respective layers. If this box is not checked, all geometry are placed on the work layer.

Create Empty - When checked, layers that contain no geometry are created. If this box is not checked, layers that contain no geometry are deleted.

Display All - When checked, all objects on all layers are displayed. 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.

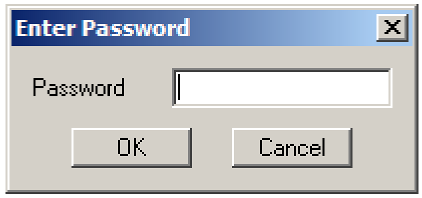
Importing a DWG file that contains entities that are not supported, a box appears warning that the entities were not read.

Opening Password Protected DWG Files

The password protection in AutoCad files is supported in Cobalt, Xenon and Argon under the Windows operating system only. To open a password protected file:

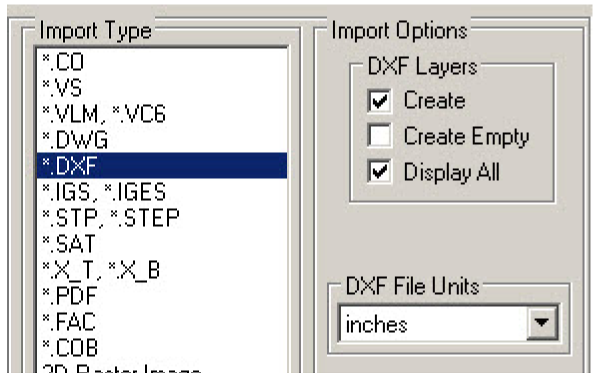
1. Go to ***File>Impor***t.
2. Choose DWG or DXF formats in the Import dialog box and click OK.
3. Navigate to the place where the file is located with the Open window and click Open button.

4. Input the password in the Enter Password Dialog box and click OK.



5. The drawing opens in the Cobalt window.

DXF



Selecting this imports DXF (AutoCAD’s Data eXchange Format) files. The translator will read DXF files up to and including version R2013.

This translator imports horizontal, vertical, diameter and radial dimensions in addition to geometry.

DXF includes two options: *DXF Layers*, and *DXF Units*.

DXF Layers This section includes three check boxes: *Create*, *Create Empty* and *Display All*.

Create - When checked, the layers in the DXF file are created/ imported and geometry is placed on the respective layers. If this box is not checked, all geometry is placed on the work layer.

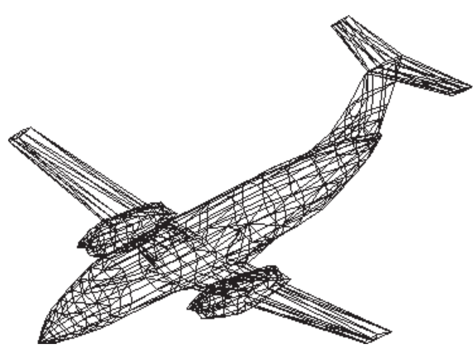
Create Empty - When checked, layers that contain no geometry are created. If this box is not checked, layers that contain no geometry are deleted.

Display All - When checked, all objects on all layers are displayed. 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. Cobalt, Xenon and Argon convert these surfaces to 3-Point mesh elements. This graphic shows an example of a DXF facet file that was imported into the program.



DXF Notations

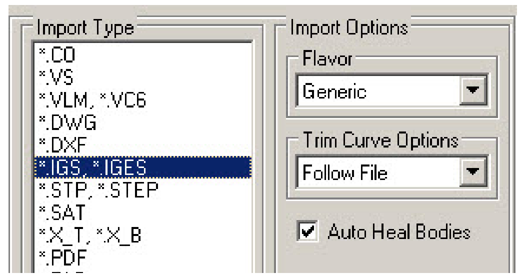
The DXF translator does not support the following entities: Shape, Ole2frame, MLine, Leader, MText, ViewPort, Tolerance, Proxy, Hatch and Image.

Importing a DXF file that contains entities that are not supported, a warning box will appear saying that the entities were not read.

Tech Note: *There is almost no reason to use an IGES file format for importing from SolidWorks. Use ACIS SAT, Parasolids or STEP for 3D, or DXF for 2D. Use IGES with the SolidWorks flavor only if importing from a SolidWorks version prior to 2000. For any version after 2000 use one of the formats above or, if absolutely necessary, IGES Generic.*

IGES

Selecting this imports IGES files created by various CAD programs. IGES includes three options: *Flavor*, *Trim Curve Options and Auto Heal Bodies*.



Tip: *IGES is best known in the industry as "I guess."*

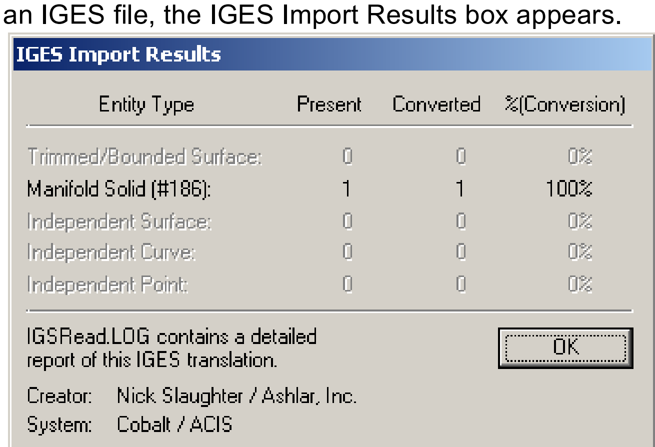
This translator also supports importing horizontal, vertical, diameter and radial dimensions.

Flavor This section includes a pull-down menu to choose to import three different IGES flavors: *Generic*, *AutoCAD* or *SolidWorks*. Use *Generic* if specific IGES flavor is not listed.

Auto Heal Bodies When checked, auto-healing is applied to imported surfaces. Auto-healing attempts to find collections of surfaces that define closed volumes and convert them to solids.

Trim Curve Option Is set appropriately for the flavor chosen but can be overridden using this option.

When importing an IGES file, the IGES Import Results box appears.



The box contains the following items:

Entity Type Lists five entity types: Trimmed/Bounded Surface, Manifold Solid (#186), Independent Surface, Independent Curve and Independent Point.

Present Displays the number of a particular entity in the file.

Converted Displays the number of entities for the type that converted.

%(Conversion) Lists the percentage conversion of the particular entity.

Creator Information 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

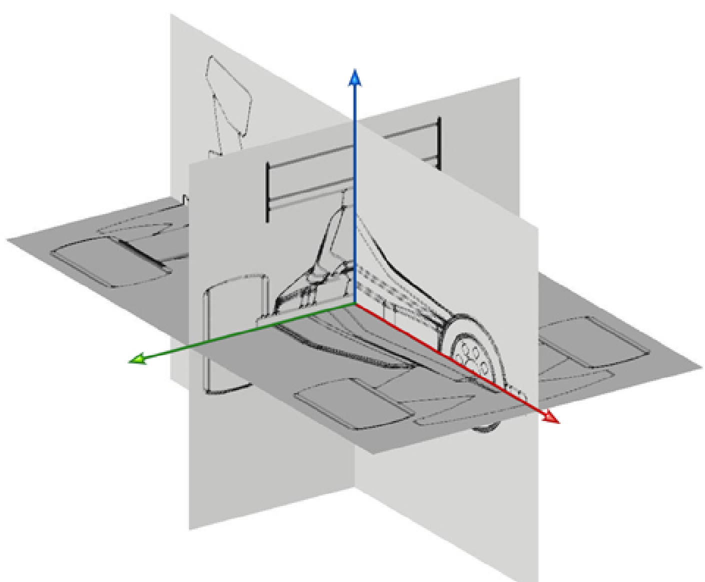
The graphic shows a facet file after it was imported into this Designer Elements program. Each shade represents a material to be analyzed by the signature predication code. The canopy and raydome are transparent, so they are not included in the signature model.

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.

Bitmap Images (2D Raster Images)

Hand sketches, photographs and other 2D images can be imported and displayed in 3D. This makes it significantly easier to trace over the sketches to create a 3D model. These images can be oriented in any direction and viewed from any perspective regardless of the work plane. The bitmap comes in on the current work plane. Use the Trackball to view the images from all directions.



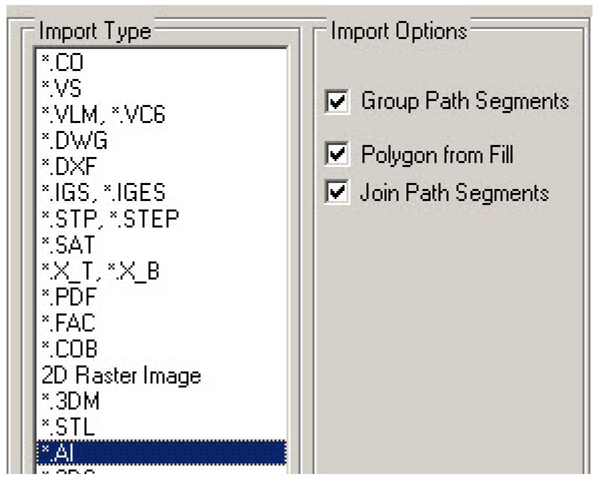
Importing 2D raster images provides the following options:

Import the Image Center Sets the center of the target file at the origin of the drawing area.

To the Work Plane Origin Imports the image to the work plane origin.

Adobe Illustrator

Selecting this type imports Adobe Illustrator version 5.0 through 8.0 files. Adobe Illustrator includes three options: *Group Path Segments, Polygon from Fill* and *Join Path Segments*.



Group Path Segments Determines how paths are imported. When selected, separate segments from a multi-span path are grouped.

Polygon from Fill 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.

Join Path Segments Attempts to merge adjoining path segments into as few segments as possible without changing the shape.

This translator also supports importing text.

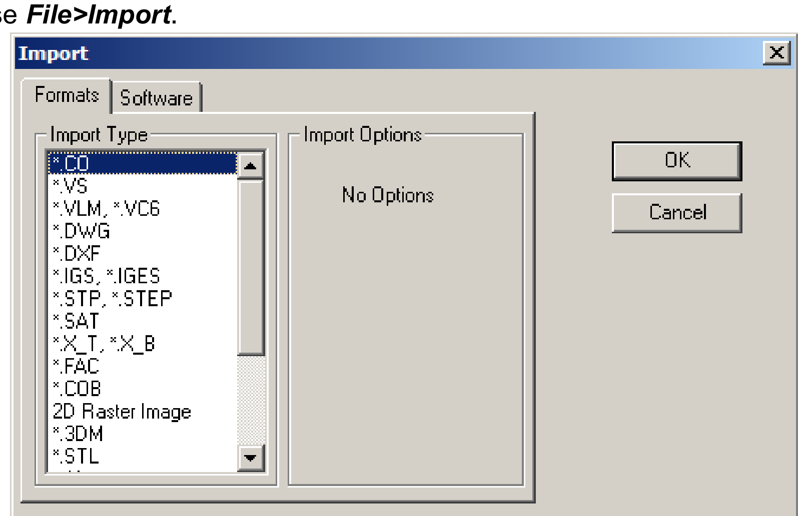
Spline

Selecting this type imports ASCII Spline files created by various CAD programs. There are no options for this import type.

Import Command

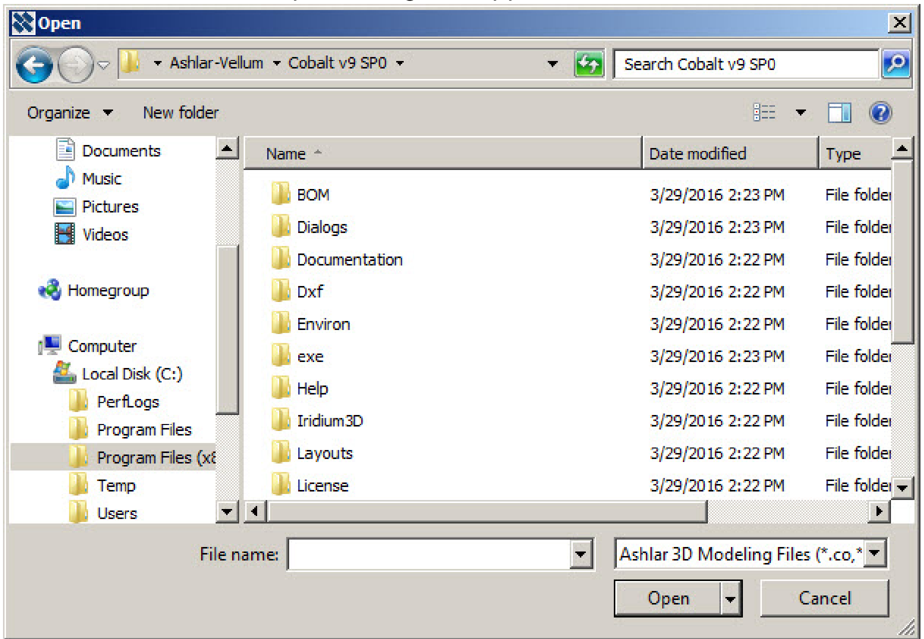
This command in the File menu imports a document and places it in the current file.

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 to import.

   (Windows users: If the *Open as read-only* box is checked, the original file can be viewed but changes cannot be saved under the same file name.)
5. Click Open to import the file.

The file appears in the drawing area. To save the file in its original format after editing, choose the *Export* command from the File menu.

Importing Splines

When importing a text file that contains the coordinates of a spline, Cobalt, Xenon or Argon 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.

   The program begins creating the spline.

Importing Tips and Notations

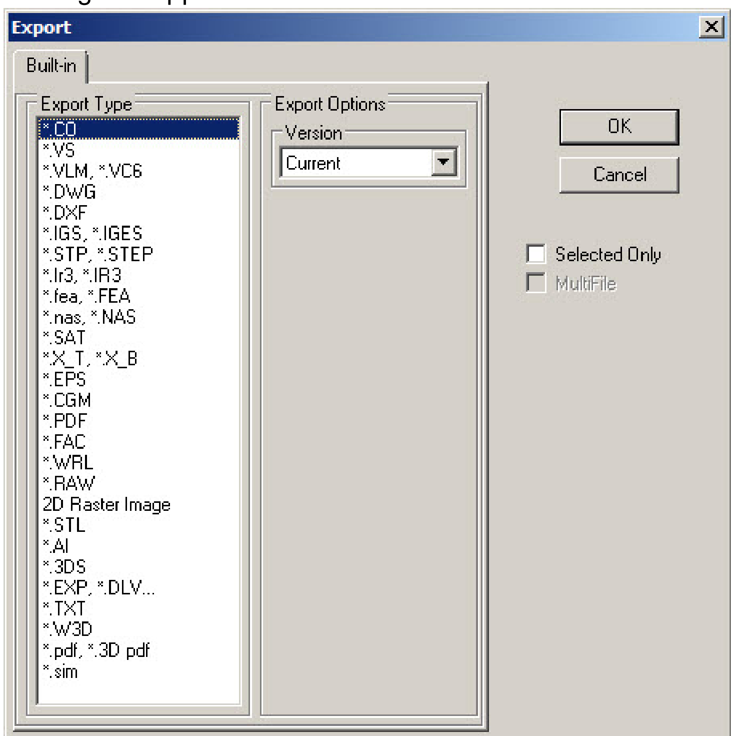
These tips and notations will help to successfully import files.

- **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** - Cobalt, Xenon and Argon do 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.

- 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 trying to import a file, Cobalt, Xenon or Argon create a file log in the program folder. For example, if a file is imported through the IGES translator and an error occurs, the file IGSRead is created.

Exporting

This Designer Elements program exports to many different kinds of files formats. Some types include export options specific to the format. When choosing the *Export* command, the dialog box appears.



The dialog box contains these elements:

Formats/Software Tabs Simplify the selection of the correct format for file translation. Click the tabs to switch between the lists of formats or software. The Formats tab contains a list of specific file types. The Software tab allows the program name to be chosen instead of a cryptic file extension.

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 When checked, only selected geometry is exported.

MultiFile When checked creates a file for each object in the drawing, using the same name as in the Design Explorer. This option is useful when exporting files to products that do not handle multiple objects within one file such as Pro/E and SolidWorks.

End of Line Specifies an end of line structure for the exported file. There are three options: Mac (LF), PC (CR/LF) and Unix (CR).

OK — Click this button to close the Export dialog box and display 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 (including Xenon and Argon), Vellum Solids, Graphite, DWG, DXF, IGES, STEP, ACIS SAT, Parasolid, EPS, CGM, Facet, VRML, RAW, STL, Adobe Illustrator, Catia v4, Text, Macrmedia, PICT (Macintosh only).

Vellum Solids (VS)

Selecting this type exports files as a Vellum Solids file. There are four options: VS 2000, VS 2000 SP1, VS 99 SP1, VS99 and VS 98.

VS 2000 — Exports Vellum Solids 2000 Service Pack 0 files.

VS2000 SP1 — Exports Vellum Solids 2000 Service Pack 1 files.

VS 98 — Exports the Vellum Solids 2000 file as a Vellum Solids 98 file.

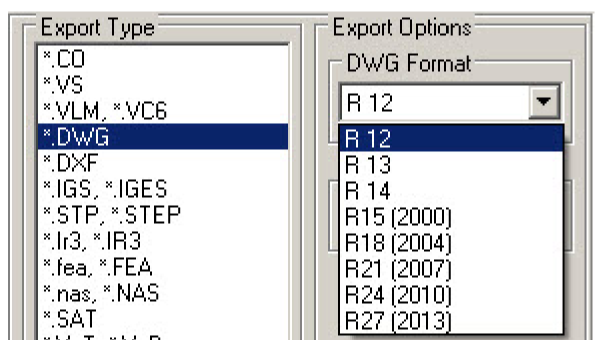
VS 99 — Exports Vellum Solids 99 files.

VS 99 SP1 — Exports Vellum Solids 99 Service Pack 1 files.

Graphite/Vellum 3D (VC6 and VLM)

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

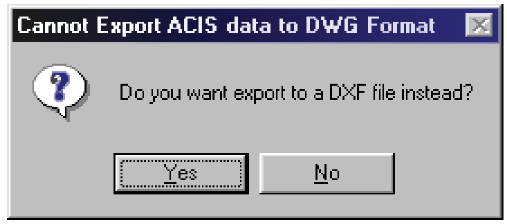


Selecting this type exports DWG files compatible with AutoCAD and other programs that support the DWG file format. (DWG is the binary version of DXF.) DWG includes several options.

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 (_).

> ***Tech Note:*** *For maximum compatibility among DWG versions of 2000 or later, use the Version 2000 option since no new data was added to the later file formats and those with different versions will have trouble reading the file.*



ACIS data cannot be exported using the DWG translator. When attempting 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 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 Exports the file as an R13 file. Ellipses, splines and ACIS curves are supported. Conics are converted into polylines.

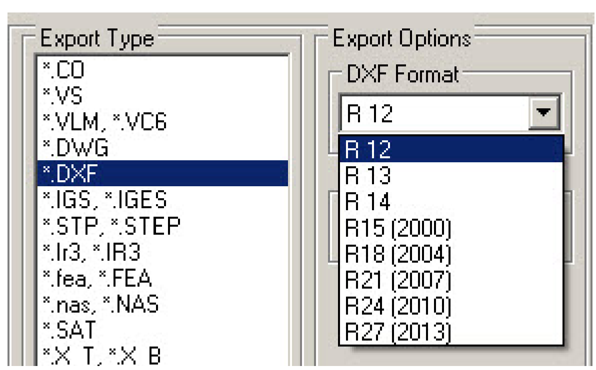
Version 14 Exports the file as an R14 file. Ellipses, splines and ACIS curves are supported. Conics are converted into polylines.

Version 2000 Exports the file as an R2000 file. This was the last time the file type was extended by Autodesk relevant to Ashlar-Vellum products. For maximum compatibility among users with Version 2000 and later, use this option.

Versions above 2000 Provided for convenience. This option will prevent users of earlier Autodesk products from opening these files. For best compatibility use the Version 2000 option.

Choose the DWG option based on the translator version supported by the program into which to import the file.

DXF



Selecting this type exports DXF files compatible with AutoCAD. DXF includes several options.

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.

> ***Tech Note:*** *For maximum compatibility among DXF versions of 2000 or later, use the Version 2000 option since no new data was added to the later file formats and those with different versions will have trouble reading the file.*

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 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 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 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 Exports the file as an R2000 file. This was the last time the file type was extended by Autodesk relevant to Ashlar-Vellum products. For maximum compatibility among users with Version 2000 and later, use this option.

Versions above 2000 Provided for convenience. This option will prevent users of earlier Autodesk products from opening these files. For best compatibility use the Version 2000 option.

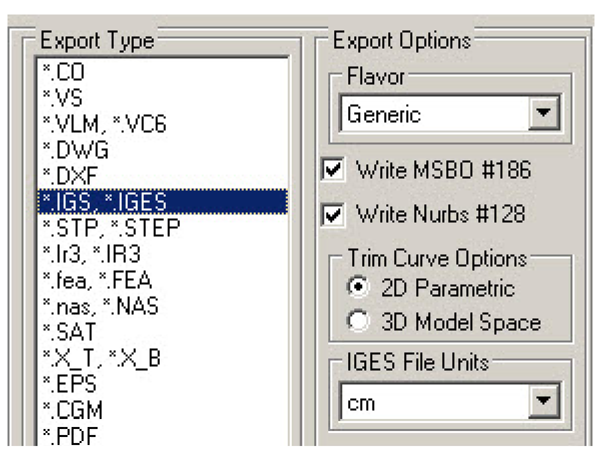
Choose the DXF option based on the translator version supported by the program into which it is necessary to import the file and the end of line setting.

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.



Tip: *IGES is best known in the industry as "I guess."*

Tech Note: *There is almost no reason to use an IGES file format for importing from SolidWorks. Use ACIS SAT, Parasolids or STEP for 3D, or DXF for 2D. Use IGES with the SolidWorks flavor only if importing from a SolidWorks version prior to 2000. For any version after 2000 use one of the formats above or, if absolutely necessary, IGES Generic.*

Flavor

Includes a pull-down menu 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 a specific IGES flavor is not listed.

Write MSBO #186

When checked, exports 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

When checked, exports solids using this nurb surface type.

Trimming Curve Prefs

Includes two check boxes *2D Parametric* and *3D Model Space*. Choose one option.

2D Parametric - Exports 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.

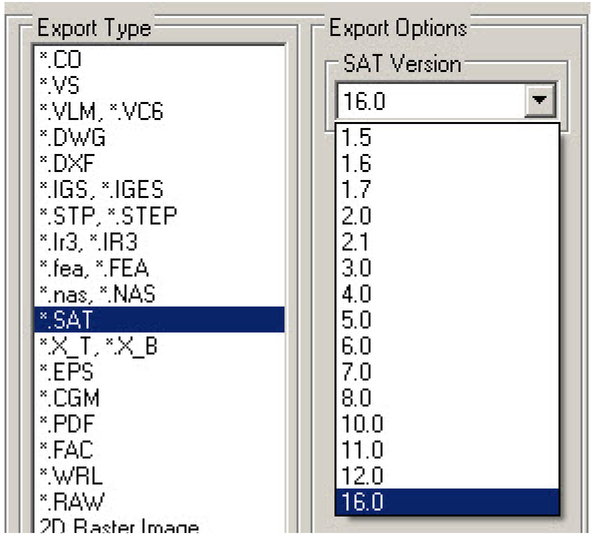
3D Model Space - Exports the actual 3D trim curve in the model space.

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.

It is also possible to choose the end of line structure (Mac (LF), PC (CR/LF) or Unix (CR)) for the file from the End of Line pull-down menu. There are no other options for this export type.

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 in the pull-down menu.

This translator supports exporting curves, surfaces, solids and grouped objects. It does not support exporting layers.

EPS

Selecting this type exports an EPS (encapsulated postscript) file. There are no options for this type.

> ***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.*

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.

PDF

Ashlar-Vellum’s built-in PDF writer has several advantages over printing to PDF. It is possible to create multi-sheet vector PDFs drawn from multiple models in the same file and embed the source files or translations directly in the PDF.

Everything for an entire project can be emailed, displayed and archived. Use Adobe Acrobat to further enhance the file, inserting and replacing pages, commenting and marking changes or tracking revisions.

Tech Note: *PDF page size is controlled by the print layout settings and paper selection.*

Exporting to *.pdf,*provides the following options:

Attach Source File Includes the source file with the PDF.

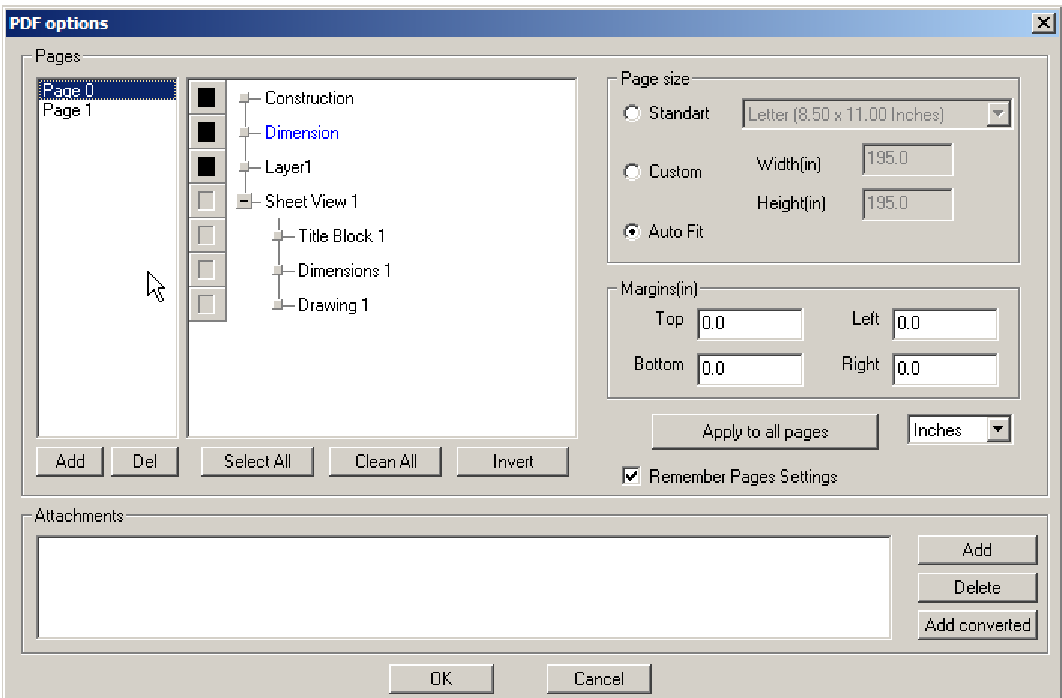
Open Resulted File Opens the new file automatically after export.

Selected Only Exports only those entities selected.

Multi File Creates one entity per file for later importing into other types of software.

Advanced Brings up the following display:

.



In the PDF Options dialog box it is possible to set the following:

Page Selects which layers will appear on which pages of the PDF. Add and delete pages as necessary. If layers are not specifically designated, all visible layers are exported to one page of the PDF by default.

+ Adds pages to the target file.

- Deletes pages.

Select All Selects all the layers for the current page.

Clean All Deselects all the layers for the current page.

Invert Inverts the selected or layers on the current page.

Page Size Designates page size. Can be *Standard* or *Custom.*

Standard enables the drop down menu to select one of the predefined standard formats.

Custom enables the Width and Height fields for setting the custom page size. The allowed PDF size range is from 0.04167x0.04167 inches, 1.05833x1.05833 millimeters or 3x3 points up to 200x200 inches, 5080x5080 millimeters or 14400x14400 points.

Auto Fit Expands the page size so that all entities fit on the page.

Margins Designates margin settings. For standard and custom page sizes the margins are measured inwards from the strictly defined page size. Geometry is inscribed within the remaining area.

Apply to All Pages Applies settings designated for current page get applied to all pages of the document.

Units Drop down menu for setting the units for the width and height of the pages.

Remember Pages Settings Saves the current page, size, and margins settings with the Cobalt, Xenon or Argon file for later exporting.

Add Attaches additional files to the PDF. Click Add then navigate to the file.

Add Converted Exports the same file in any other format and attaches it to the PDF.

OK Closes the Advanced settings box.

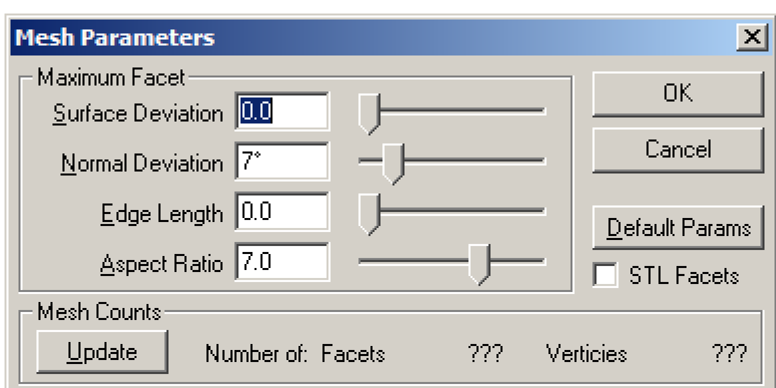
When the PDF is opened in a viewer such as Adobe Reader or Acrobat, each page is shown as designated.

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.



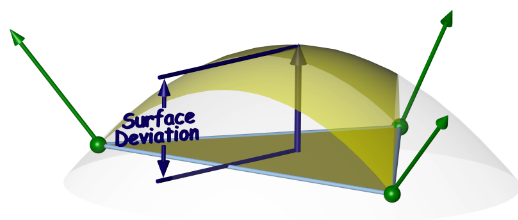
During the conversion, vertex points are distributed on the surface or solid. These vertices are then grouped into 3-sided and 4-sided facets. The conversion is deemed acceptable when the generated vertices and facets satisfy the settings. The 5 available settings are Surface Deviation, Normal Deviation, 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.

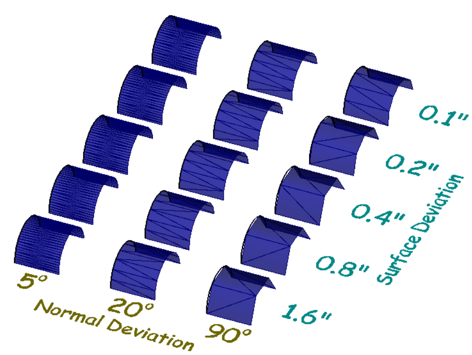
Keep in mind that the settings are used by the faceting algorithms if possible. It is often not possible to satisfy all settings simultaneously. In this situation, the algorithm decides which settings to “loosen.”

The Mesh Parameters dialog box contains the following options:

Surface Deviation 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 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 Controls the maximum allowed edge length of any given facet.

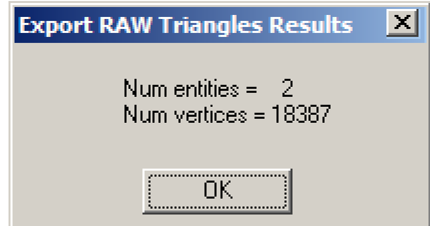
Aspect Ratio Controls the maximum allowed aspect ratio of any given facet.

STL Facets Forces 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 the 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.

The raw file can be viewed 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 exporting an Illustrator file, this Designer Elements program automatically displays it with an Illustrator icon. Double-click the file to launch Adobe Illustrator.

> ***Tip:*** *For best results use the PDF export option for Adobe Illustrator since Adobe Illustrator natively supports PDF.*

Text

Selecting this option will export the text contained in the file along with a list of objects contained in the file.

Catia v4 EXP or DLV

Selecting this type exports a CATIA v4 file. There are no options for this type.

Macromedia

Selecting this type exports a Macromedia file. Choose the Geometry Quality and there is the option of creation a log file if the box is checked.

Bitmap Images (2D Raster Images)

The dialog box for exporting 2D raster images provides the following options:

View Area	Exports the current view only. If left unchecked the software will Zoom All and capture everything visible.
Ignore Background	Tells the software to ignore any background color or gradient.

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 specified format.

1. Choose ***File>Export***.

   The Export dialog box appears.

2. Select the export type and its options. 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 a name and location for the exported file.

   Windows: The type of file being exported 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

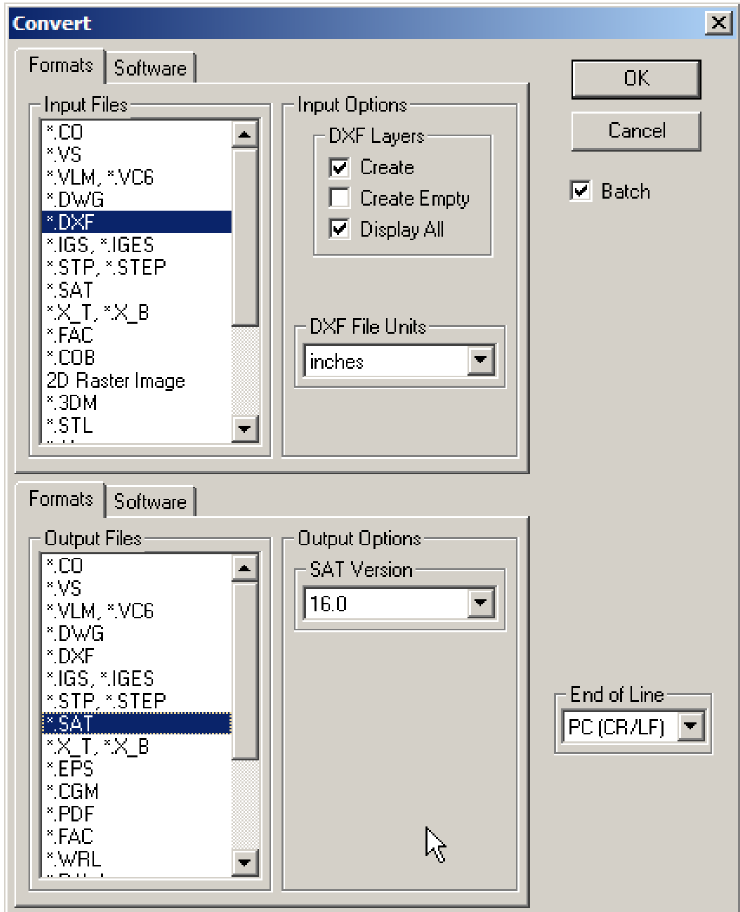
These tips and notations will help to successfully export files.

- When exporting files from Cobalt, Xenon or Argon 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 for the target application. Be sure to check whether there are any tips or notations for a successful translation in this program or the target application.
- In general, it is best to show all layers before exporting from this Designer Elements program or from another program that will be imported into this program, to know what is being exported.
- AutoCAD does not like the & symbol in the layers table of the DXF file. Remove the symbol from the 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.

Batch Convert Tool

The *Convert* command in the File menu converts entire folders of files from one type to another by automating the import or export process. To use the command:

1. Create two new computer folders. One will host all of the source files that will be converted. The other will receive the converted files.
2. Place all files to be converted in one batch in one of the folders created in step 1.
3. Choose ***File>Conver***t.
4. In the Convert dialog box indicate the input and output file types along with any options available. Activate the Batch check box and click OK.



5. In the first dialog box navigate to the new folder containing the files to convert. Click OK.
6. In the second dialog box navigate to the destination folder and click OK.

Drawing Composition for Cobalt & Xenon

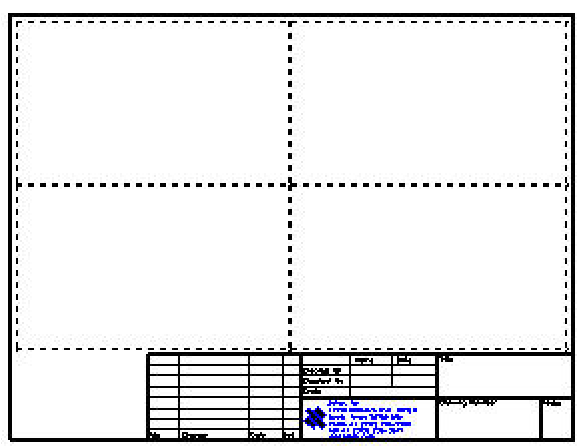
Cobalt and Xenon Only

Cobalt and Xenon quickly create 2D drawings from one or more entities. The 2D drawing views are associative in Cobalt and Xenon. Changes made to the objects automatically appear in the 2D drawings. In Argon, however, the 2D drawing views are snap shots without associativity. Instructions for creating drawings in Argon are discussed in the next chapter. The topics covered in this chapter for Cobalt and Xenon include:

- Model to Sheet Command
- Drawing Views
- Sheet Tools
- Unfolding Views
- Editing a Drawing View
- Layout Templates
- Printing the Layout

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 Cobalt or Xenon creates 2D geometry from the selected 3D wireframe, surface and solid models and places them in drawing views on a new layer, called *Sheet View*. The original 3D geometry remains.



Tip: *Turn on all layers and zoom out, to see the original geometry in the Top view.*

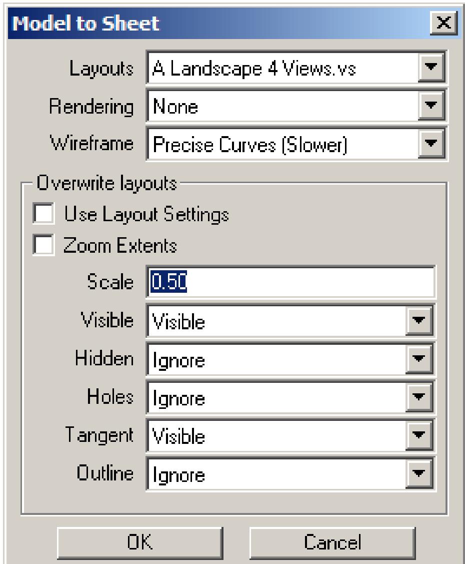
Warning: Do not use this command with curves or groups.

If the geometry contains objects or drawing border lines using the background color, the program automatically changes it to the foreground color. This capability is especially helpful when the background color is black.

See Colors 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 Sets the layout for displaying the geometry. These layouts are predefined and located in the Layout folder within the same folder as the application. The layouts contain drawing views and may contain title blocks and drawing size boundaries depending on the layout selected.

Create or modify any of the templates based on the drawing layout preferences. See Layout Templates for more information.

Rendering Represents *Flat, Gouraud* and Phong shading options and Photo-rendered, Photo-rendered (no materia) rendering options to make the object look photorealistic in the model to sheet.

Wireframe Sets how hidden edges display in the final drawing. There are *Precise Curves (Slower)*, and *Polylines (Faster)* settings.

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 Shade Options for information on the render modes.*

Use Layout Settings The dialog box has 5 combination boxes that change the pen style when doing a Model to Sheet. It is possible to have a layout with predefined pen and scale settings for each view and retain the values by checking the Use Layout Settings check box. When this box is checked, the pen style options are grayed out.

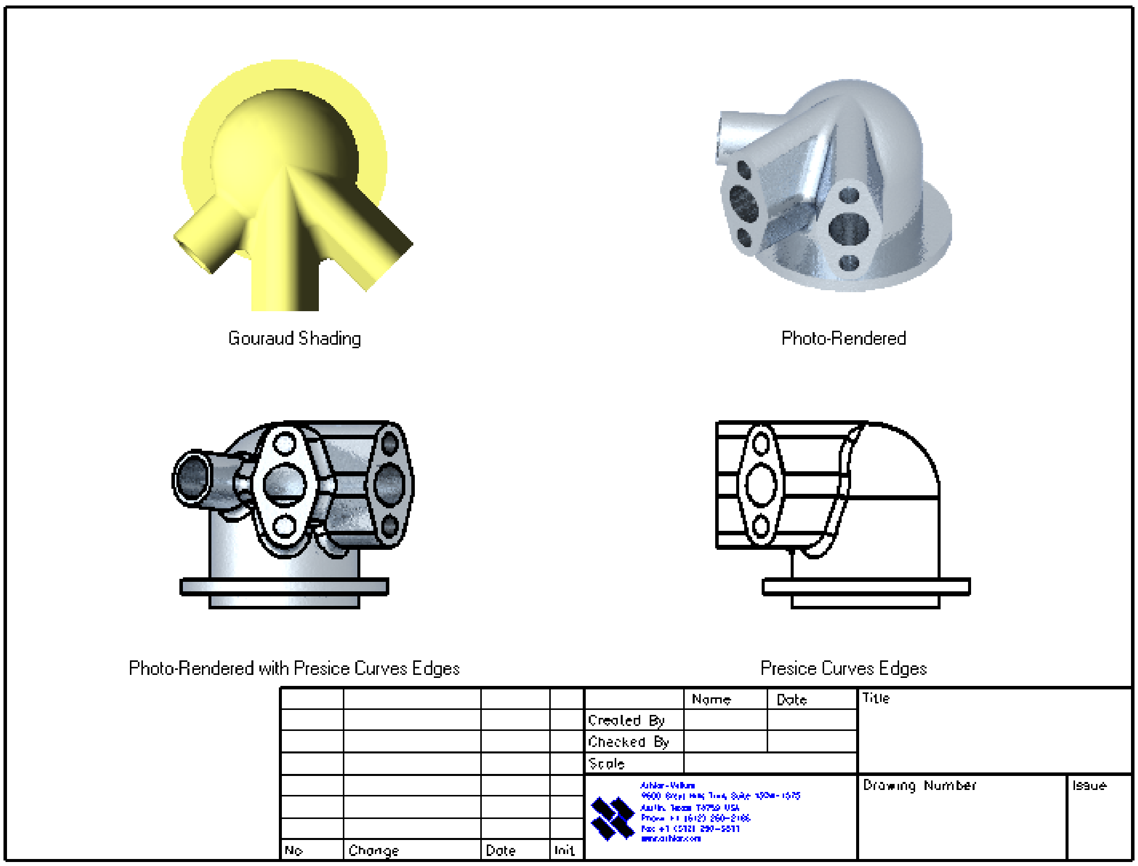
Zoom Extents Changes the scale of the drawing view so the objects within it fill the drawing view. This command operates similarly to the *Zoom All* command except that it applies to a drawing view.

Scale Sets the scale of the drawing.

Pen Styles Refer to the Draw View Properties section of this chapter for details.

OK, Cancel Clicking OK completes the *Model to Sheet* command and displays the geometry based on the specified settings. Clicking Cancel, cancels the command and closes the dialog box.

Below is a sheet with four views



Any number of photo-rendered views can be included in engineering drawings.

Using the Model to Sheet Command

1. Select the models to display on the sheet.
2. Choose ***Layout>Model To Sheet***.

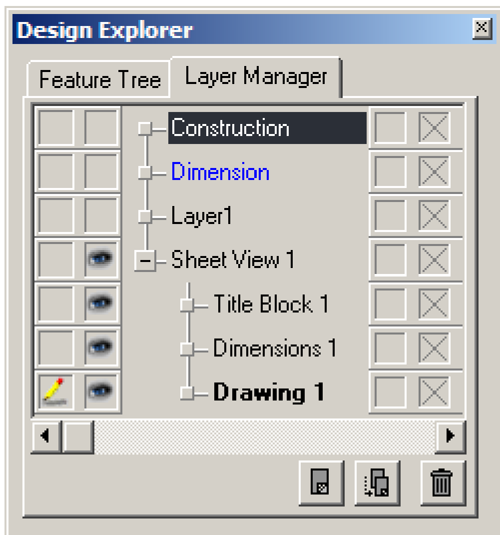
   The Model to Sheet dialog box displays.

3. Choose the *Layout, Rendering and Wirefrme* options from their respective pull-down lists.
4. Enter a scale in the *Scale* field.
5. Click OK.

   2D geometry is created from the 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 the geometry are placed.

   Referral: *Drawing Views are explained later in this chapter.*

   If a layout is selected 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 number of new 2D objects on that layer.



   The Sheet View layer is now the work layer and all other layers are turned off.

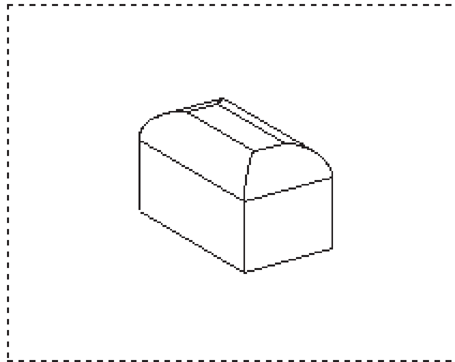
Undoing the Model to Sheet Command

If the *Model to Sheet* command is inadvertently selected or later it is necessary to remove the sheet, use the *Undo* command or the Layer Manager.

- Using *Undo*: Choose ***Edit>Undo*** (CTRL+Z (Windows) or ⌘+Z (Macintosh). The *Undo* command can be used 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. The view is still set to top. Change the view and zoom scale to view the geometry.

Drawing Views

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.



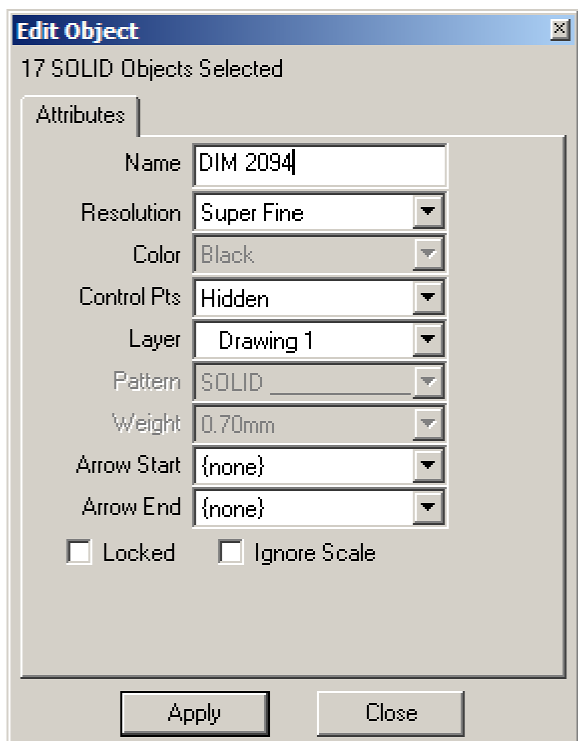
Tech Note: *Drawing views operate differently from those in Vellum 3D. You are not looking at a 3D model in a drawing view but rather 2D wireframe object created from the 3D model. The trackball cannot be used to rotate the view orientation of a specific drawing view or zoom in the drawing view. Use the Change View command and Properties command in the Drawing View menu to make adjustments. See the Drawing View Menu.*

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.

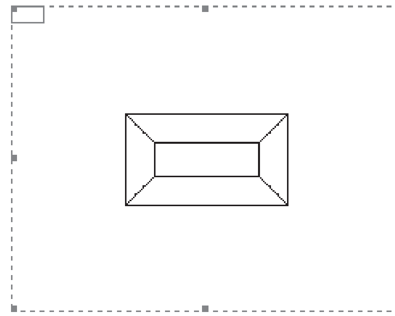
Spacing for crosshatching, dimensions, text, arrow size and line font in drawing views can now be set relative to the sheet through the Edit Objects dialog box, independent of the scale set in the drawing view. This sets any scale for the view without concerning how the item will appear.

Just double-click the item to display the Edit Objects dialog box. On the Attributes page, select the Ignore Scale option. Click Close or Apply to update the selected item.



Activating Views

Once a drawing is created, activate individual views by clicking in the rectangular region surrounding the view. When activated, the view boundaries highlight in red.

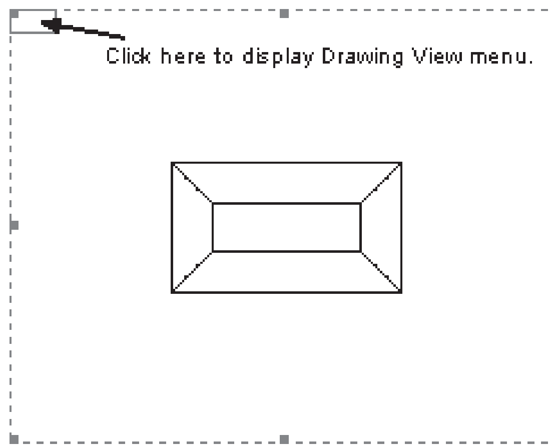


You cannot activate drawing views if the sheet is displayed in any view other than top. All geometry in the view becomes available for selection when the view is activated. The Drafting Assistant recognizes all snap points in the view. Click outside the area to deactivate the view.

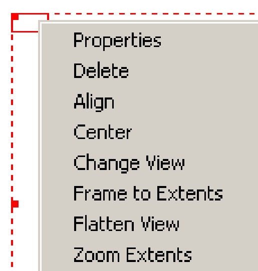
Tech Note: *The Drafting Assistant only recognizes geometry within an active view.*

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.

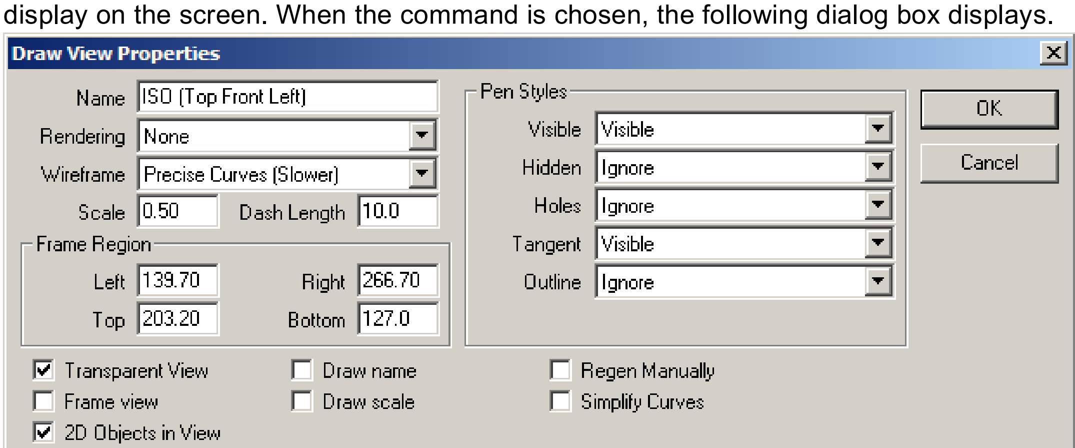


The view menu can also be displayed by clicking the right mouse button (Windows) or by holding down the CTRL key and clicking the mouse button (Macintosh).

The View menu contains the following commands: *Properties*, *Delete*, *Align*, *Center*, *Change View*, F*rame to Extents* and *Flatten View*.

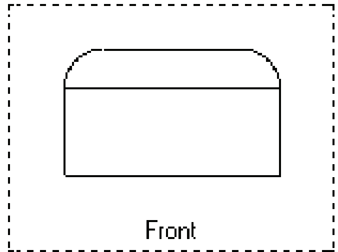
Properties Command

The *Properties* command accesses to a set of properties that affect a drawing view's display on the screen. When the command is chosen, the following dialog box displays.



The dialog box contains the following options:

Name Specifies the name that appears in the drawing view when the *Draw name* check box in the dialog box is activated.



Rendering Provides *Flat, Gouraud* and Phong shading options and Photo-rendered, Photo-rendered (no materia) rendering options to make the object look photorealistic in the model to sheet.

Wireframe Sets the hidden edges display in the final drawing. There are *Precise Curves (Slower)*, and *Polylines (Faster)* settings.

Scale Sets the scale of the drawing.

Dash Length Sets the length of dashes when using either the *Dash All Hidden (precise)* or *Dash Only Holes (precise)* edge display options.

Frame 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).

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 different pen styles can be explored for all the drawn views. Any change to the master pen style will automatically update all edges that use that pen style.

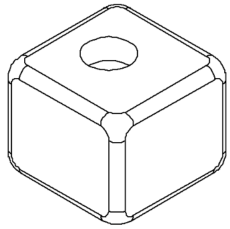
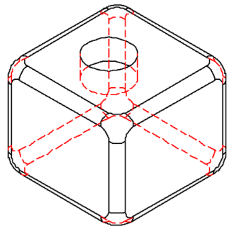
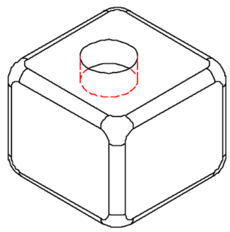
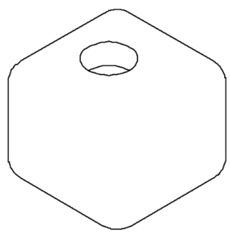
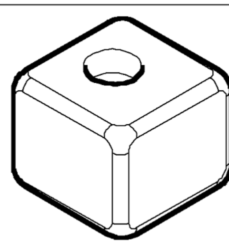
For example, suppose it is necessary the visible lines printed in a E size drawing at a heavier weight than an A size drawing. In this case just change the Visible pen weight to the desired weight and print.

Note: It is also possible to select "Use Object Color" which does not use a pen style but always follows the object color used to create the draw view.

Visible	Applied to edges that are not hidden. This excludes edges that are classified as tangent or outline.
Hidden	Applied to edges that are occluded.
Holes	The holes edge format is applied to edges that are hidden and cylindrical.
Tangent	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.

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 Shade Options for information on the render modes.*

The following picture gives examples of how changing these parameters affect the 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 Styles 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	

Transparent View Controls whether the drawing view background is clear or opaque. When checked, 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 Controls the display of the drawing view frame. When selected, the view frame is visible.

2D Objects in View 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.

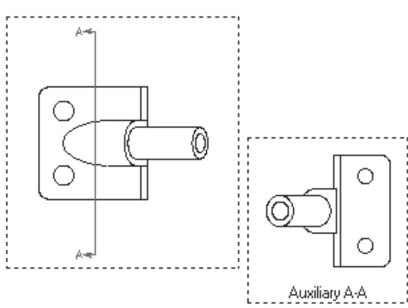
Draw Name	Controls the display of the text in the Name field. When selected, the text displays at the lower border of the frame, centered between the left and right edges.
Draw Scale	Adds a scale to the layout as a text element with the view names.
Regen Manually	Controls whether the view regenerates or updates automatically after making changes to the geometry. This control is helpful if a file requires a long regeneration time. With this option selected, a series of changes can be made without the delay of regeneration. To regenerate one particular drawing view, toggle this option between on and off. This option relates to the **Manual View Regeneration** tool in the **Sheet** tools palette which regenerates all views on the sheet.
Simplify Curves	Attempts 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	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 Fill and Crosshatching.
Area Hatch Only	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.

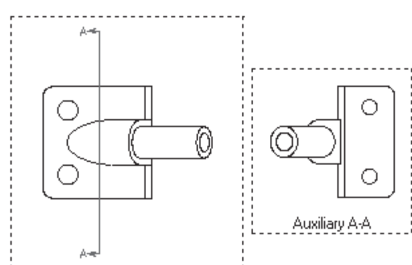
Delete Command

The *Delete* command in the Drawing View menu removes the drawing view and its contents from the drawing. The original model remains.

Align Command

The *Align* command in the Drawing View menu functions like a tool with the Message Line which contains steps for its use. This command aligns an active view with another selected view. Use the command on section views, general drawing views and auxiliary views. The left graphics here show, the top view of a CAM part with its associated auxiliary view. The right graphics show the same two views aligned.





1. Select the view to align.

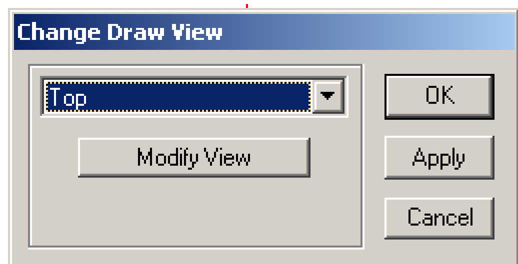
2. In the Drawing View menu, choose the *Align* command. The Message Line reads: *Align View: Pick parent view to align with*.
3. Select the parent view. The two views align.

Center Command

The *Center* command centers the objects within the frame boundaries. This is especially helpful if the drawing view is moved or the view orientation of the geometry is changed.

Change View Command

The *Change View* command changes the view orientation of the selected drawing view. Change the view to one of the pre-defined or user-defined views or modify the view by specifying the Eye/Reference point, Azimuth/Elevation or rotation values.

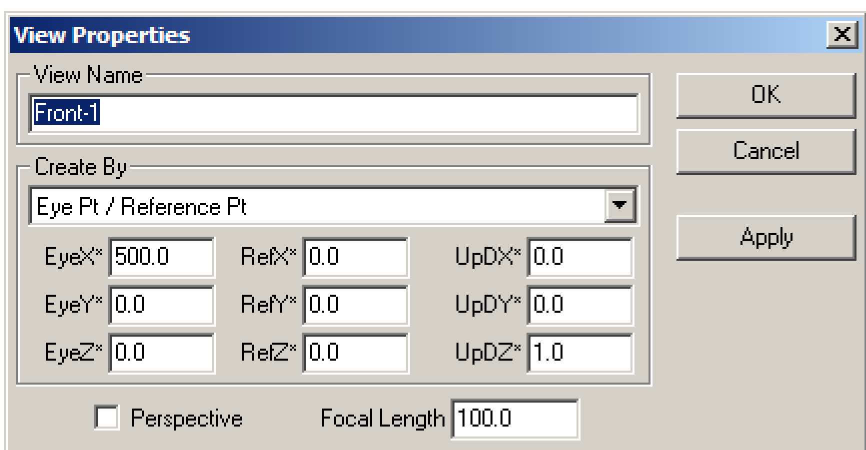


When this command is chosen, the Change Aux View dialog box appears.

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 Clicking this button brings up the View Properties dialog box.



Change the Eye/Reference point, Azimuth/Elevation or rotation values for the view. It is also possible to name the view. If a name entered and to display it in the drawing view, it is necessary also to 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.

Perspective Enables perspective so that the size of an object's dimensions along the line of sight are relatively shorter than dimensions across the line of sight.

Focal Length Sets the convergence or divergence of perspective. The naked eye sees at about 50 which gives noticeable perspective. Shorter focal lengths increase perspective with a wide-angle effect. Longer focal lengths shorten perspective. Focal Length is only applicable when Perspective is checked.

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.

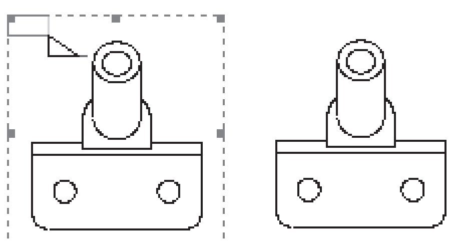
Flatten View Command

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 the 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 an object is dimensioned in a view that was 1.25 inches and shown at a scale of 2, the flattened dimension would still read 1.25. The actual length, however, is 2.50.

Important: Be sure to mark the drawings “Not to Scale” if flattening a view with any scale other than 1 so that people read the dimension value rather than measuring the object.

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.



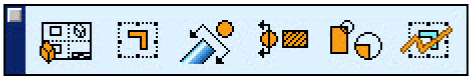
If a detail view is flattened, the detail view boundary is converted into a circle with a phantom pen pattern and flattened onto the sheet with the geometry.

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 the drawing is created, some changes can be made to the drawing to better illustrate certain aspects of the model. Tools are provided to create new views, modify a selected view and add auxiliary views, section views and details views to the 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.

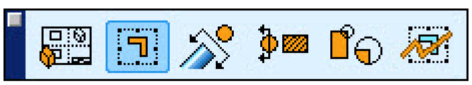


Drawing View Tool

This tool creates empty drawing views. Use this tool to add drawing views to the current layout already containing views or create customized layouts.

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 the pointer is moved 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

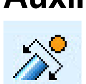
Cobalt or Xenon add geometry (associative to the original 3D geometry) to empty views. When changes are made to the geometry all associative geometry updates.

1. Turn on the layers containing the 3D geometry to place in the new view.
2. Select the 3D geometry.

   If geometry is selected from an existing drawing view, only 2D wireframe geometry would be copied.
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



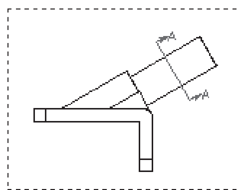
This tool creates auxiliary views from a referenced drawing view. Auxiliary views dynamically align to that view and are associative to both the original view and the auxiliary dimension. Change the geometry and the auxiliary view automatically updates.

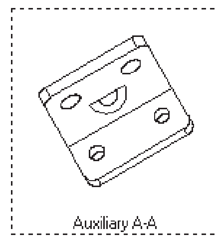
Move the dimension location in the original view and again the auxiliary view reflects the changes. Auxiliary dimension lines appear in the original view. These dimensions are placed on the Sheet View layer. The Auxiliary tool can only be used when a drawing view is present.

By default, auxiliary labels are alphabetical. If more than one auxiliary view is placed in the drawing, the auxiliary view label automatically increments to the next letter. If the file containing auxiliary views is closed and later it is reopened and another auxiliary view is added, the label increments to the next letter based on the last auxiliary label in the file. Any label text can be entered if desired. The user-defined label text, however does not automatically increment.

> ***Tech Note:*** *The font type, size or style of the dimension label cannot be changed. Use the* **Text** *tool to create custom labels.*

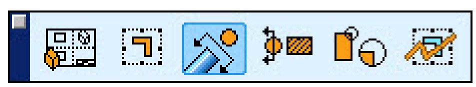
The left graphic here shows the referenced view with auxiliary view dimension lines. The right graphic shows the auxiliary view.





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 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 the pointer is moved, an outline of the frame appears.

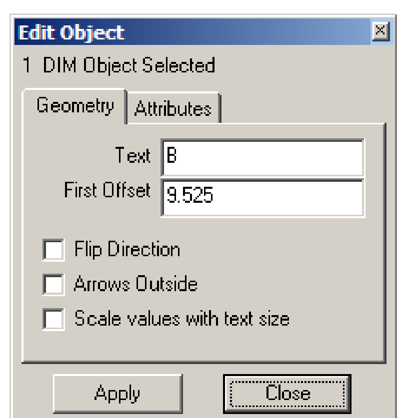
   The auxiliary view displays with a view label at the specified location. The auxiliary view dimension appears in the original view.

   To delete the view later, delete both the view and the dimension line in the original view. The auxiliary labels may need to be adjusted to accommodate the deletion.

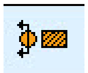
Changing the Direction or Label of the Auxiliary View

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.

The view label can be changed in this dialog box.



Section View Tools



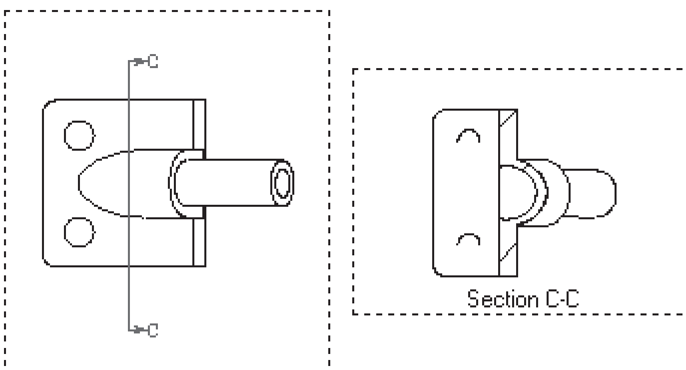
These tools create horizontal, vertical and 2 point section views. Section view geometry is associative to the geometry in the view and the section dimension. Change the geometry and the section view automatically updates. Move the dimension location in the original view and again the section view reflects the changes. Section view dimensions are placed on the Sheet View layer.

The **Section** tools specify whether to see only 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.

By default, section labels are alphabetical. If more than one section view is placed in the drawing, the section view label automatically increments to the next letter. If the file containing section views is closed and later reopened and another section view is added, the label increments to the next letter, based on the last section label in the file. Any label text can be entered if desired, however, user-defined label text does not automatically increment.

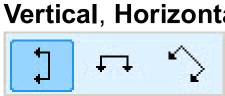
> ***Tech Note:*** *The font type, size or style of the dimension label cannot be changed. Use the* **Text** *tool to create own labels.*

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.

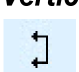


> ***Tech Note:*** *Solids cannot be pasted into section views. To add a solid, add it to the original view.*

When the **Section View** tool is selected, a subpalette appears containing three tools: **Vertical**, **Horizontal** and **2 Pt**.



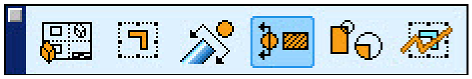
Vertical Section View Tool



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 the geometry for the section.

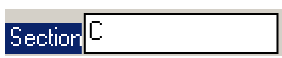
   The Message Line reads: *Drag window to final position*.

5. Drag the view to a new location and click the mouse button. Notice that as the pointer is moved, an outline of the frame appears.

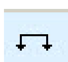
   The section view displays with the view label shown in the Status Line. To have a different label, enter it in the Section data field and press ENTER (Windows) or RETURN (Macintosh). The section view dimension appears in the original view.

   To delete the view later, delete both the view and the dimension line in the original view. It may be necessary to adjust the section labels to accommodate the deletion.

   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 the geometry for the section.

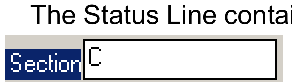
   The Message Line reads: *Drag window to final position*.

5. Drag the view to a new location and click the mouse button. Notice that as the pointer is moved, an outline of the frame appears.

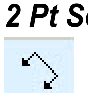
   The section view displays with the view label shown in the Status Line. To have a different label, enter it in the Section data field and press ENTER (Windows) or RETURN (Macintosh). The section view dimension appears in the original view.

   To delete the view later, delete both the view and the dimension line in the original view. It may be necessary to adjust the section labels to accommodate the deletion.

   The Status Line contains the Section view label.



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.

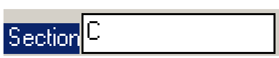
   The Message Line now reads: *Section View: Pick start and end of section orientation*.
3. Click two points on the geometry to indicate start and end points for the section cutting line.

   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 the pointer is moved, an outline of the frame appears.

   The section view displays with the view label shown in the Status Line. To have 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. Adjust the section labels to accommodate the deletion.

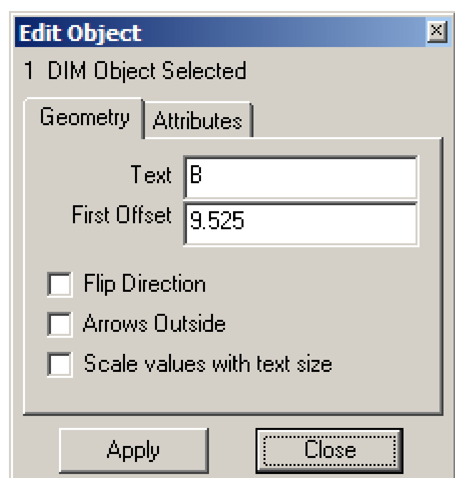
   To delete the view later, delete both the view and the dimension line in the original view.

   The Status Line contains the Section view label.



Changing the Direction or Label of a Section View

To change the section view direction opposite to that indicated by the view dimension, select the dimension and choose ***Window>Edit Objects***. On the *Geometry* page, check the Flip Direction option and click Apply. The direction of the view changes.



It is also possible to change the view label in the dialog box.

Sections and Crosshatching

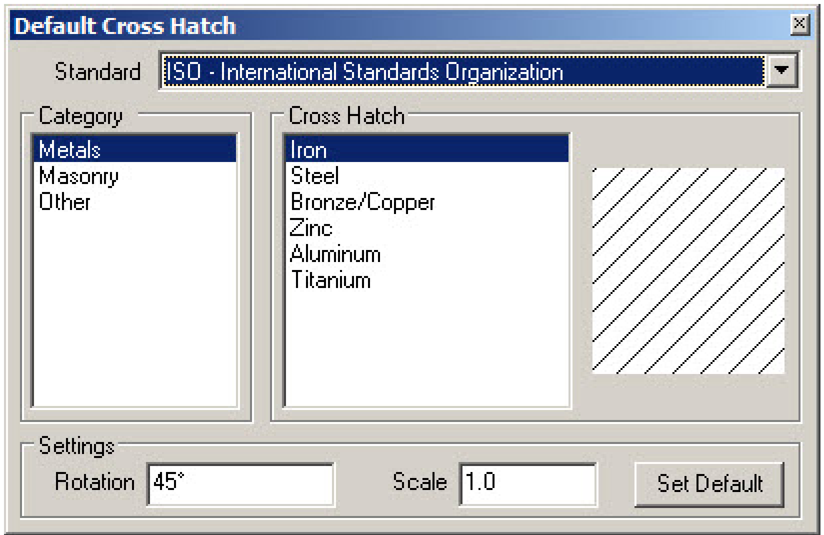
By default, section views automatically contain crosshatching. Remove the crosshatching or change the hatch pattern for a selected view through the Cross Hatch dialog box.

> ***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 the Editing Objects chapter for more information on Mass Properties.*

It is also possible to set the default hatch pattern through this dialog box. When nothing is selected, choose ***Pen>Cross Hatch***. See Fill and Crosshatching chapter for more information on the dialog box and the hatches available.

If there is more than one object cut in the section, define separate hatch patterns.

1. In the section view, select the hatch pattern.

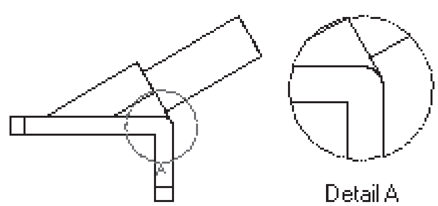


2. Choose ***Pen>Cross Hatch***. The following dialog box displays.
3. Select a category and Cross Hatch pattern from their respective lists.
4. Set the hatch rotation angle and scale.
5. Click Apply to change the selected hatch.

   If you are not satisfied with the hatch pattern, choose another. Change the hatch patterns of any other section geometry in the view or in other views if necessary.
6. Click the Close button (Windows) or the Close box (Macintosh) to exit the dialog box when finished.

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.



By default, detailed dimensions are alphabetical. If more than one detail view is placed in the drawing, the detail view dimension and label automatically increment to the next letter. If the file containing detail views is closed and later reopened and another detail view is added, the dimension and label increment to the next letter, based on the last detail label in the file. Label text can be entered as desired, however, user-defined label text does not automatically increment. The graphic above shows a drawing view and an associated detail view.

> ***Tech Note:*** *The font type, size or style of the dimension label cannot be changed. Use the* **Text** *tool to create own labels.*

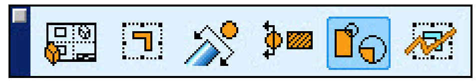
Once a detail view is created, the detail view dimension can be moved in the original view from its default location, shown as A in the left graphic above. Select the label, choose ***Edit>Show Points*** and drag the control point/detail dimension to another location. The detail view label, shown as Detail A above, cannot be moved.

This tool can only be used when a drawing view is present.

> ***Tech Note:*** *Solids cannot be pasted into section views. To add a solid, 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 the geometry for a detail view.

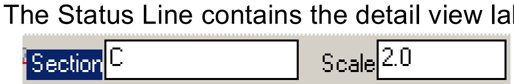
   The Message Line reads: *Detail View: Pick detail edge pt.*
4. Click the detail edge point.

   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 the pointer is moved an outline of the frame appears.

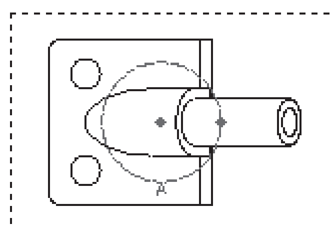
   The detail view displays with the view label and the scale shown in the Status Line. To have a different label or scale, enter the data in the appropriate data field and press ENTER (Windows) or RETURN (Macintosh). The detail view dimension appears in the original view.

   To delete the view later, delete both the view and the dimension line in the original view. Adjust the detail labels to accommodate the deletion.

The Status Line contains the detail view label and Scale.



Resizing the Detail View Dimension



It is possible to change the diameter of the detail view dimension. Select the detail dimension and choose ***Edit>Show Points***. Two control points appear defining the center and edge of the dimension.

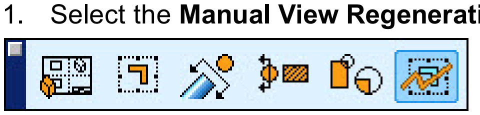
Select one of the control points and drag it to a new location.

Manual View Regeneration Tool

This tool regenerates all drawing views on the sheet. This is useful if the *Manually Regen* option in a Draw View Properties dialog box is selected for one or more views and changes are made to them.

Using the Manual View Regeneration Tool

1. Select the **Manual View Regeneration** tool.



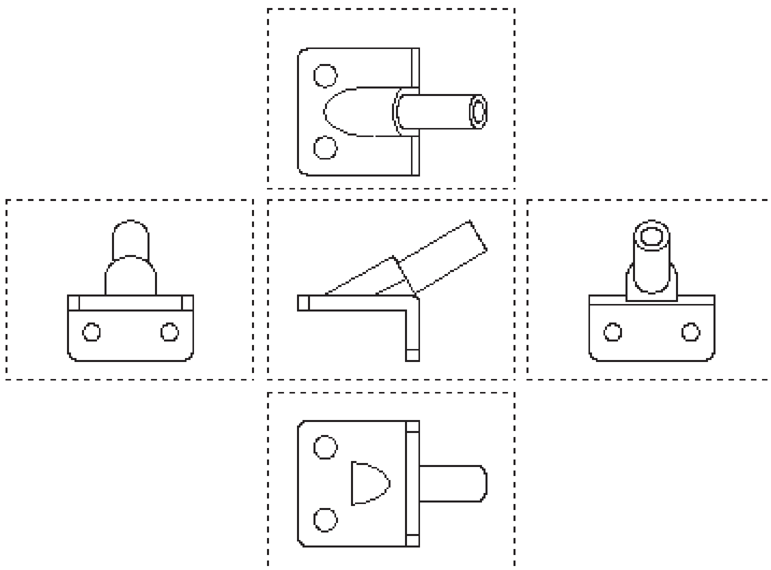
The Draw Views dialog box appears displaying the number of views that need to be regenerated.

2. Click Yes to regenerate all views.

   Click No to close the dialog box without regenerating the views.

Unfolding Views

New views can be quickly created 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 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

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.

Creating Objects in a View

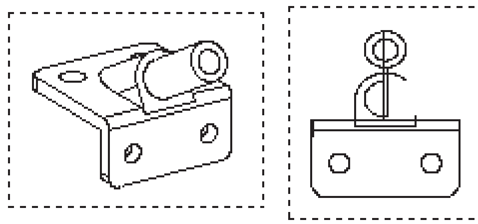
Once a view is activated, create additional wireframe and solid objects in the view. Since the Drafting Assistant functions inside the view, place the objects relative to the other objects already present.

Dimensions and text can be placed 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

Objects copied from another location can be pasted into a selected view. When pasting the objects, the contents of the paste buffer is examined and inserted 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. It is possible to change the view later and the geometry will display correctly. If 2D objects are copied 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

It is possible to edit 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

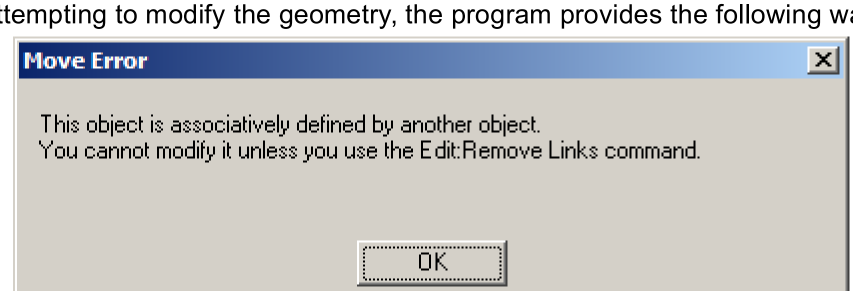
Since an object placed in the drawing views are true 2D wireframe, assign a different line pattern, color, weight or layer to the entities making up the object. To change a line characteristic, select the line in the view. In the Pen menu, choose a new pattern, color or weight. Line characteristics can be changed 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

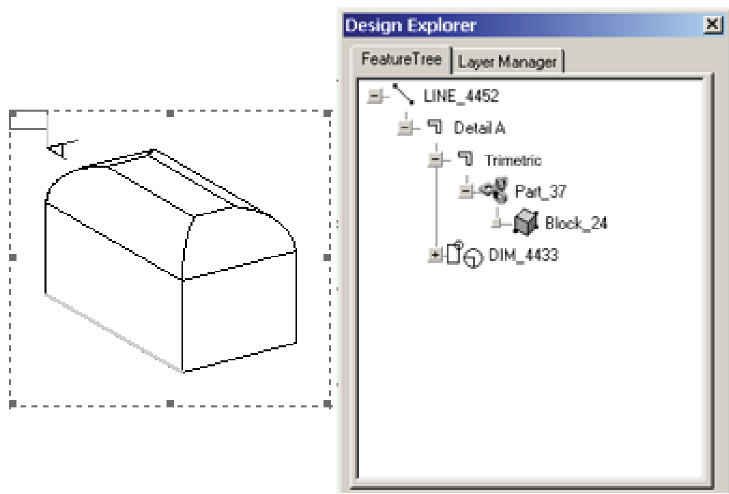
Because the 2D geometry created from the 3D model is composed of individual curves, the length or the location of the curve cannot be modified by dragging a control point like in a normal model. If this were possible, the associativity of the model would be destroyed. For example, a line in a model may represent the visible edge of a cylinder which is not actually present in the 3D model. If this was changed, a cylinder would not exist longer.

When attempting to modify the geometry, the program provides the following warning.



Given this, there are 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 as a reminder reminding that this command deletes all associative relationships and that this operation cannot be undone. 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.

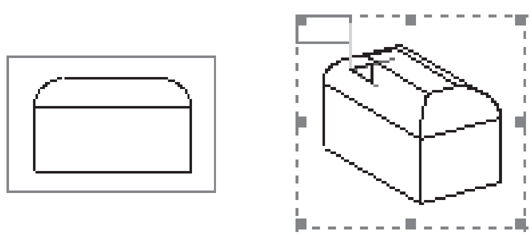


Editing View Characteristics

Changing the Scale of a View

There are two methods for changing the scale within a view:

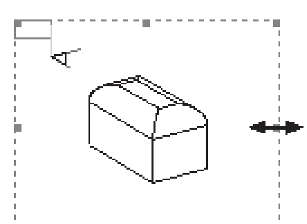
- Choose the *Properties* command in the Drawing View menu and change the scale value within the dialog box.
- Use the Selection Make and the Design Explorer to select the views to modify, then change them in the Edit Objects dialog box.



Resizing a Drawing View

There are 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 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 so that they can't be seen, choose the *Frame to Extent* command in the Drawing View menu. The view edge can be dragged manually to completely display the dimension. See Dimensions for information on the standard dimension tools.

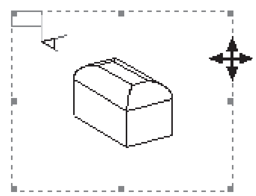
All dimensions placed in drawing views go on the Sheet View layer rather than the Dimension layer. This enables turning off all other layers and still prints the sheet with views containing dimensions.

Changing or Deleting View

The view orientation, scale and view properties can be changed through the Drawing View menu for each view. If the view orientation of a drawing view is changed and all of the geometry in the view cannot be seen, choose the *Center* command or *Frame to Extents* command in the Drawing View menu. See the Drawing Views section earlier in this chapter for more information.

It is possible to delete a view through the Drawing View menu. All drawing views can be deleted 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



There are two methods for moving a drawing view:

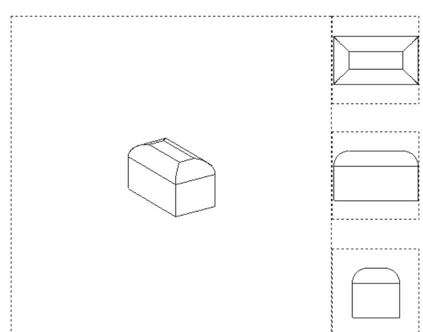
- Dragging the View: Select the view. Place the pointer over a view edge, not a control point. The pointer becomes the move symbol. Select the top or bottom view edge to drag the view vertically. Select the left or right view edge to drag the view horizontally.
- Using the *Properties* command in the Drawing View menu

Drawing Views and the Edit Menu

If you notice that the Edit menu name is red while working in a sheet view, the drawing contains some unresolved links. This occurs when geometry is moved or any other change is made. Choose ***Edit>Resolve Links***. The Edit menu name becomes black again.

Layout Templates

A default set of layout templates is displayed for each localization setting. The set of templates to controlled through the language selection in the Preferences section. To change the default set, go to ***File>Preferences>Localization*** and select the desired language set.



Creating User-defined Layout Templates

Create custome templates from scratch or by modifying an existing template.

Modifying an Existing Template

All templates provided are in the Designer Elements program files and can be opened like any other file.

1. Open one of the files in the Layouts folder.

2. Make any changes. It is possible to adjust the format, add text to the title block, and add or remove drawing views.
3. Save the file under another name in the desired Layouts folder.

   Select this template from the Layout pull-down menu in the Model to Sheet dialog box.

Creating a New Template

Create a new template from scratch using a standard 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.

   This template can be selected from the Layout pull-down menu in the Model to Sheet dialog box.

> ***Tech Note:*** *There are three possible locations for Layouts:*
> *1 In the Applications folder*
> *2 In the All Users folder*
> *3 In the Current Users folder*
> *See the knowlegebase in the Support Center of www.ashlar.com for information specific to your operating system and version of the software.*

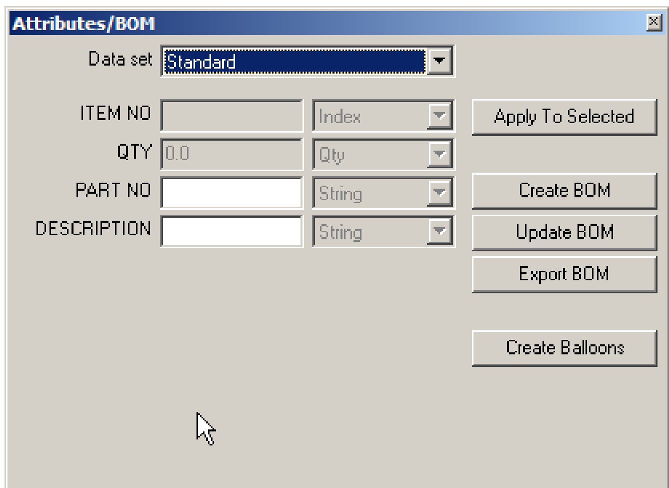
Printing the Layout

To print the layout with the views, turn off all layers other than the Sheet View layer. Make sure the *Page Setup* (Windows) or *Print Setup* (Macintosh) matches the layout.

Attributes and Bill of Materials

Cobalt and Xenon Only

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 templates and five button options.

The predefined templates are:

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.

Material Assigns a material to an entity.

Perimeter Calculates the 2D perimeter for curves.

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.

Volume Calculates the volume of a solid.

Weight Calculates the weight of a solid using the material assigned in 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, column 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

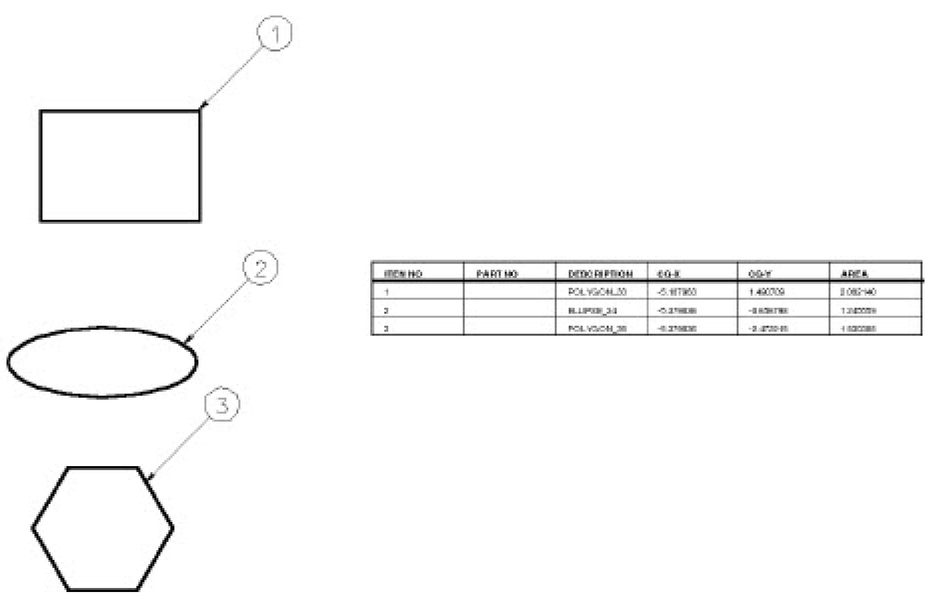
ITEM NO	PART NO	DESCRIPTION	CG-X	CG-Y	AREA
3		POLYGON_35	-5.379836	-2.472215	1.530388
2		ELLIPSE_34	-5.379836	-0.656798	1.245559
1		POLYGON_33	-5.187963	1.490709	2.082140

Graphic 2 equals Item Numbers Up

Update BOM This option updates the BOM table and the corresponding balloons.

Export BOM This option creates a tab separated text file that can be opened with any text editor as well as imported into Microsoft Excell or other similar spreadsheet programs.

Create Balloons This option adds balloon item dimensions to geometry. This option requires an item number type to be used with the attribute. All of the supplied attribute files have item numbers.



Tip: *The Bill of Materials function does not incorporate the use of Copy/Cut and Paste.*

BOM User Defined Templates

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 contain two columns, the first column contains the attribute definition string, and the second column the attribute type. Commas separate 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.

Page Setup and Printing

The geometry is created using the wireframe, surface and solids tools, then various operations are conducted on it such as extruding, sweeping and blending, and finally some basic annotation are added to the drawing.

All during the design process it was not necessary to designate a scale or paper size. Now to print a hard copy it is necessary to set up the page according to a scale, paper format, orientation and the printer specifications.

This Designer Elements program prints and plots on most printers and plotters supported by your computer. Follow the manufacturer's instructions for installing and setting up the printer or plotter, and to set up the page size as needed.

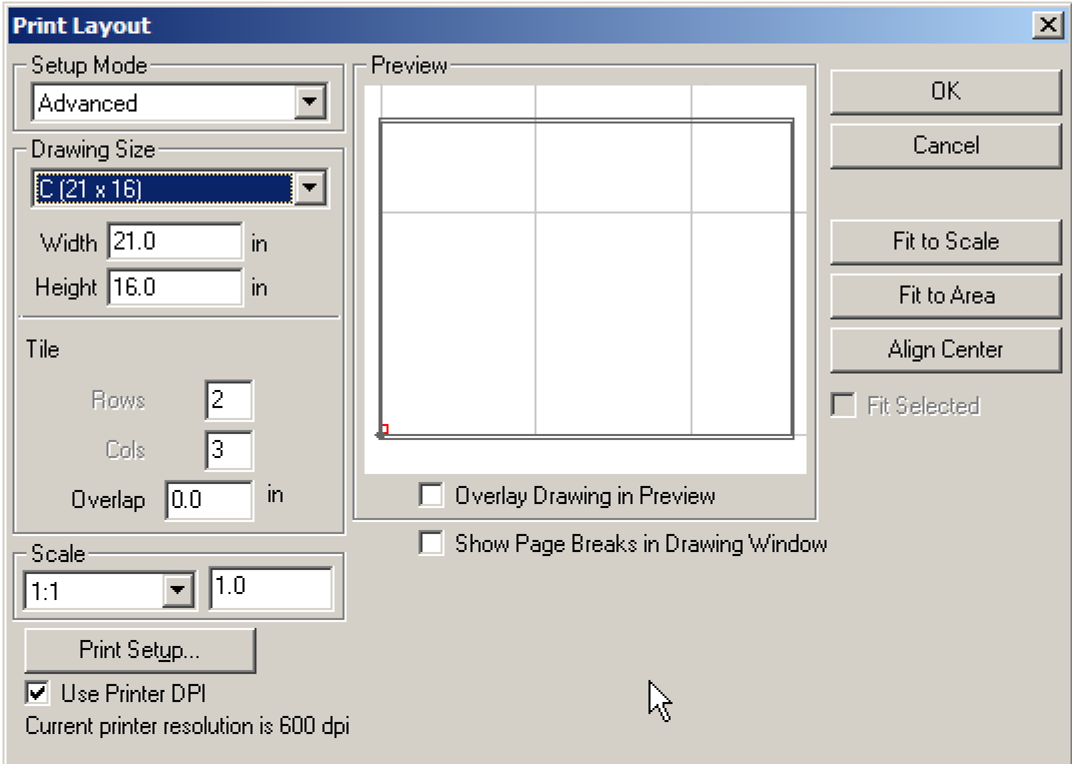
The following topics are covered:

- Page Layout Command
- Printing a Drawing
- Print Window

Page Layout Command

The *Print Layout* command, located in the File menu, displays the dialog box to set the page size, scale and other options.

Selecting the command the Print Layout 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 the last time Apply was clicked and close the dialog box.

The Drawing Size dialog box includes the following sections: *Setup Mode*, *Drawing Size, Scale, Preview* and *Utility Controls*.

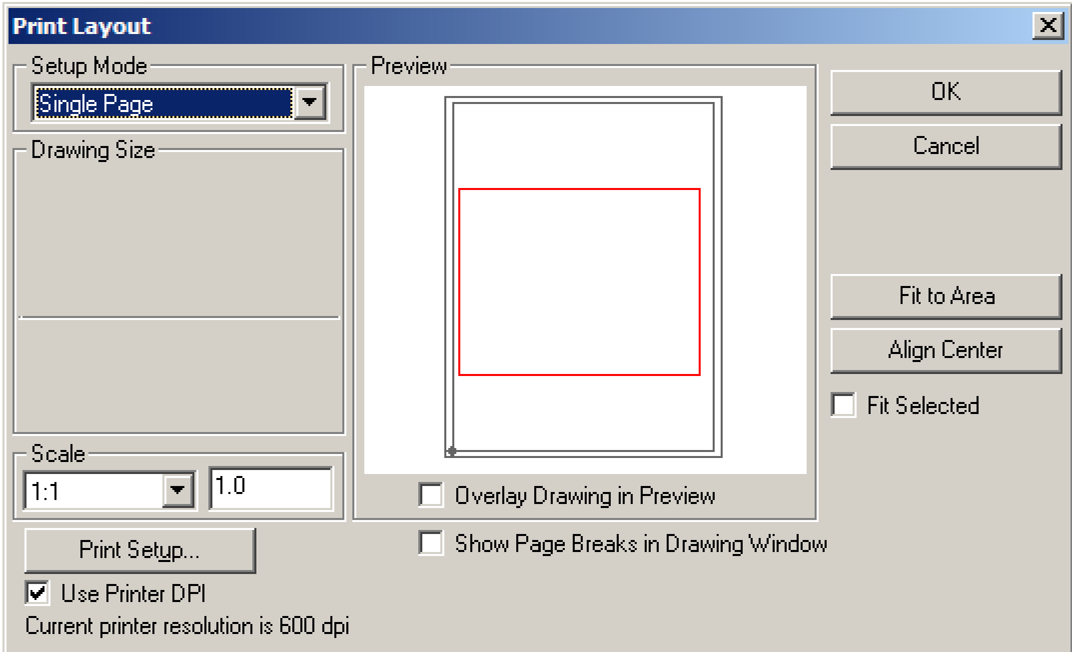
Setup Mode Section

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. Change 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 for 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.

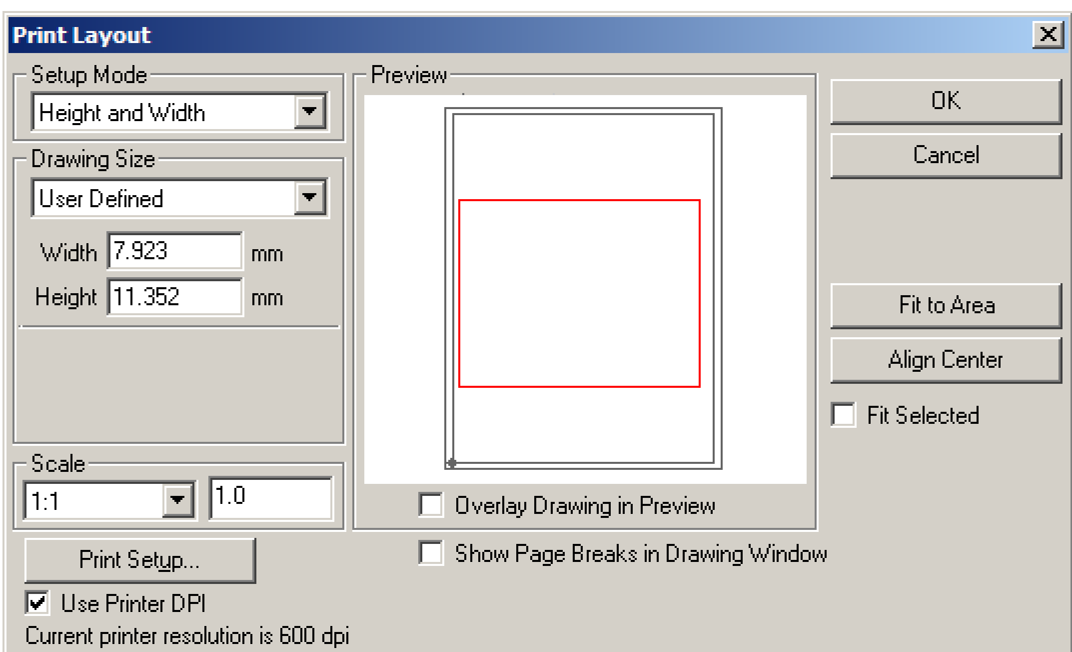


Set the plot scale by making a selection from the Scale drop down list or by changing the value in the Scale data field.

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.

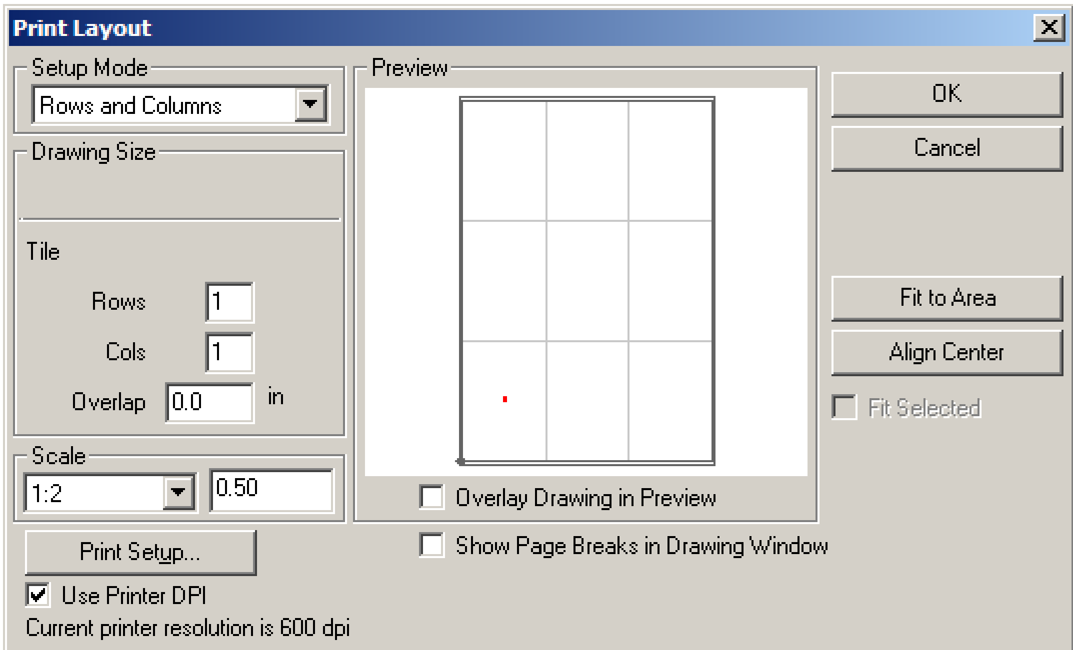


For drawing sizes larger than what the printing device allows, the drawing is tiled and can later be assembled into the large format plot. The *Preview* window displays the page tile edges within the drawing frame automatically.

Specify a standard drawing size or a custom drawing size. Set the scale in the *Scale* section or automatically compute the scale using the Fit to Area button.

Rows and Columns Mode

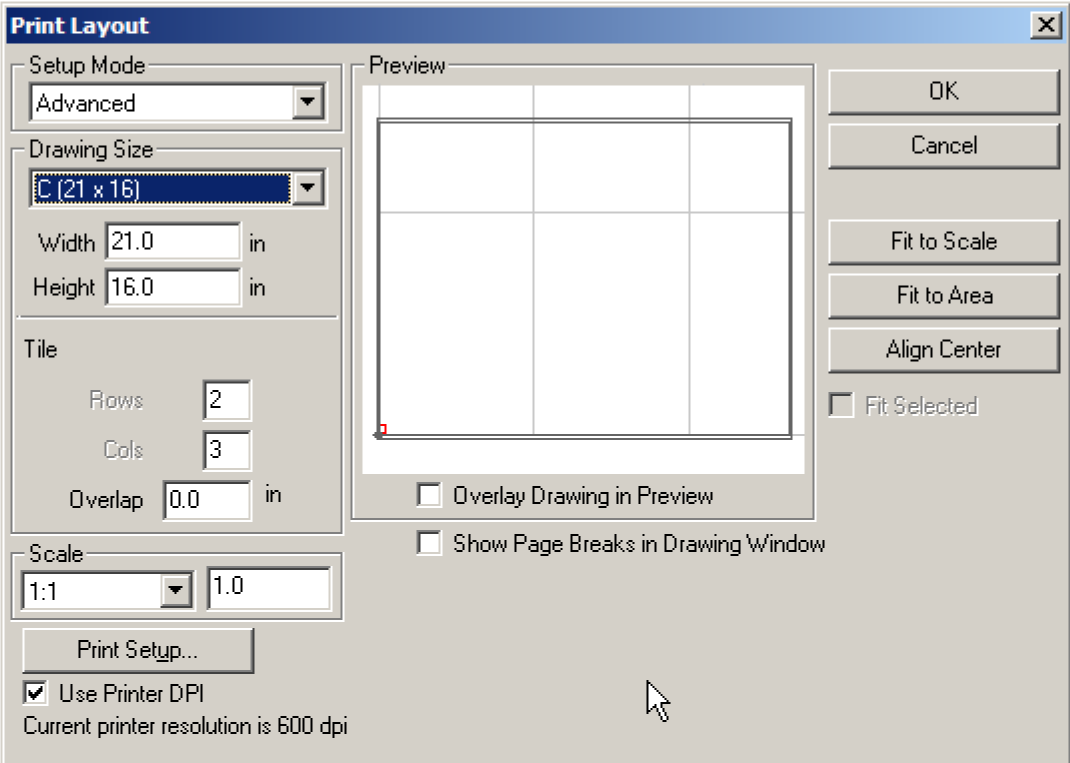
The *Rows and Columns* mode is used to force whole pages to be used for tiled plots. Unlike the *Height and Width* mode, this mode will use all the printable area available for the plot, however, the plot will not be a standard size. Selecting this mode displays the *Tile* area of the *Drawing Size* section.



Specify the tile rows and columns by changing the values in the *Rows* and *Cols* data fields. The Overlap data field controls how tile pages will overlap. The overlap region helps align the pages when assembling the final plot.

Advanced Mode

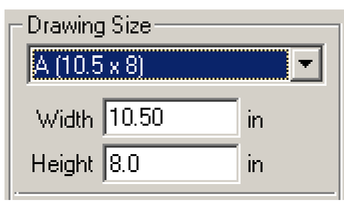
The *Advanced* mode provides access to all height, width, row and column plot settings. This mode permits complete control over all aspects of tiling. Selecting this mode displays the entire Drawing Size section.



Specify any of the listed elements.

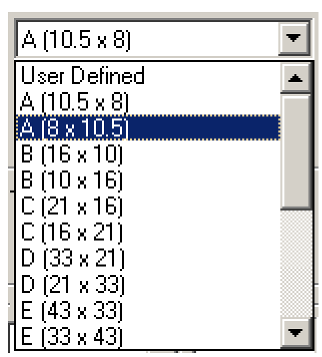
Drawing Size Section

This section contains the drawing format sizes available and the height and width of the selected format size.



The format size field includes a pull-down menu listing all of the formats and their sizes.

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.



***Tech Note:** When choosing a drawing format size, be sure to set the page orientation for the 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 the current page orientation.*

The units (inches or mm) are determined by the preference setting. When selecting one of these predefined formats, the size is displayed in the *Width* and *Height* fields.

It is also possible to set the drawing size by selecting the *User Defined* option in the list. Choosing this option, 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 program folder. This file can be edited but keep in mind it may change or be overwritten by future installations (updates) to the program. Before editing the file, save the original version under another name in order to have a copy in case it is necessary to return to the default sizes.

If the page size is larger than the size supported by the 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 the size, determine what formats the printer or plotter supports.

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 values are entered controls the drawing dimension, represented by the activated field name. The associated field name is unavailable.

Setting the Format Size

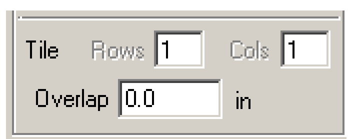
1. Choose ***File>Print Layout***.
2. Choose the *Height and Width* mode from the pull-down menu.
3. Display the pull-down menu for the drawing format size.
4. Select the desired size.

   The size is displayed in the *Width* and *Height* fields.

   If the *User Defined* format size is selected, enter the size in the *Width* and *Height* fields.

Page Tiling

The *Tile* area of the Drawing Size section sets up the file so a larger drawing can be printed in tiled sheets.



This area includes these elements:

Rows

Represent the number used to print the file, based on the size supported by the printer driver and the drawing scale. A value automatically appears in this field when a drawing size larger than that which the printer supports is selected.

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 the drawing.

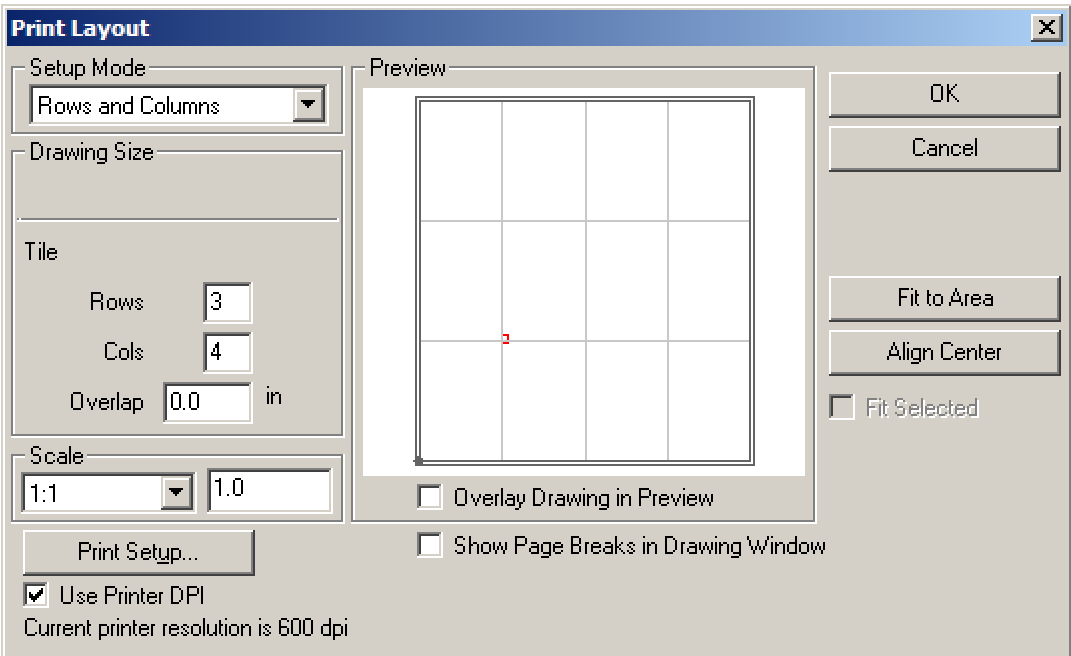
Cols

Represent the number of columns used to print the file, based on the size supported by the printer driver and the height of the drawing. A value is automatically entered in this field when a drawing size is selected.

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 the drawing.

Overlap When tiling, 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 is used to align the tiles when joining the pages.

This graphic shows an example of tiling with three rows and four columns.



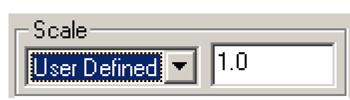
Tiling operates independently of the page orientation.

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 the preferences setting.

Scale

This section specifies the scale of the drawing. Select a standard scale from the pull-down list or set the scale in the data field.



Scale Options

The pull-down list provides 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.

To specify your own scale, enter the value in the data field. The scale name changes to User Defined, regardless of the scale entered.

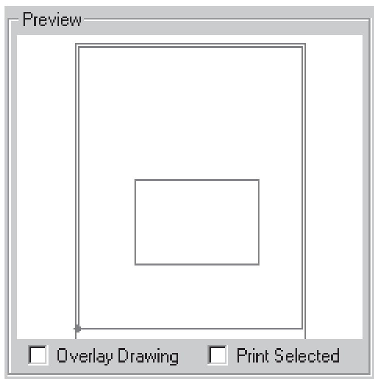
The drawing scales displayed in the pull-down list are contained in the *DrawSize.ini* file in the Environ folder within the program folder. This file can be edited, but keep in mind it may change or be overwritten by future installations (updates) to the program. Before editing the file, save the original version under another name in order to have a copy in case it is necessary to return to the default scales.

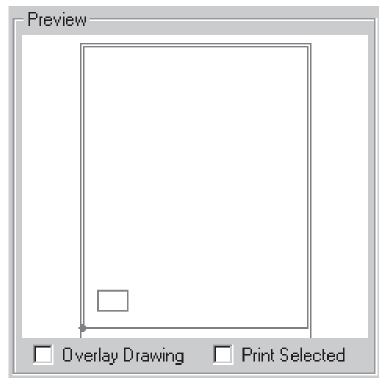
Setting the Scale

1. Choose ***File>Print Layout***.
2. Display the pull-down list for the scale.
3. Select the desired scale. The scale appears in the edit field.

   If the User Defined scale option is selected, 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.





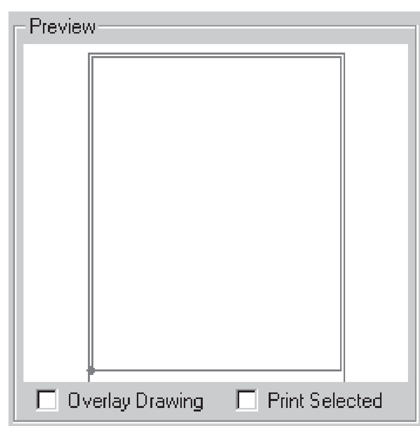
Scaling does not change the actual dimensions of the part. Verify this by selecting an object and choosing ***Window>Edit Objects***.

Preview Section

To assist in choosing the correct format size for the drawing, the 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 the drawing size is changed, the *Preview* window still displays the last drawing size selected until Apply is clicked.

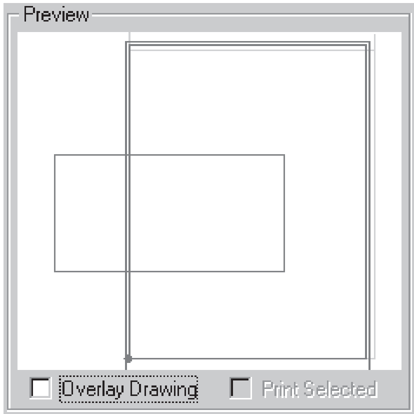
Only objects or part of objects that lie within the page bounds are printed.

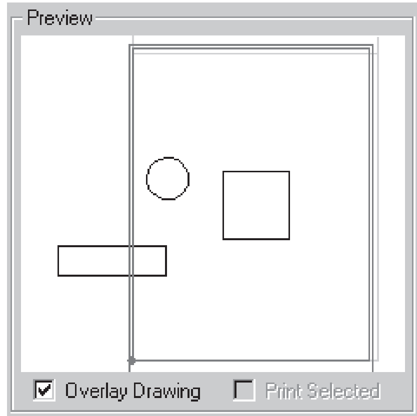
The *Preview* window displays a rectangle, representing the drawing or the actual geometry. This display is determined by the *Overlay Drawing* setting. See the next section for more details.

The Axis displays in the *Preview* window only if it's displayed in the drawing area. The Axis does not print.

Overlay Drawing

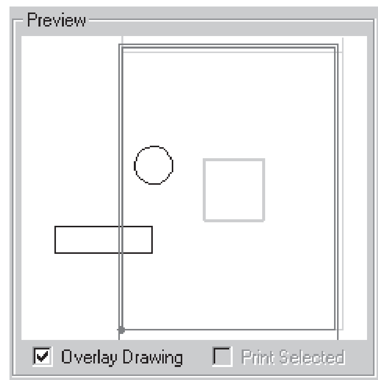
This check box determines how the geometry appears in the *Preview* window. When the box is not checked, a red rectangle displays, representing the drawing area used by the objects (the graphic on the left below). When the box is checked, the actual geometry displays (the right graphic).





Print Selected

This check box specifies 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.



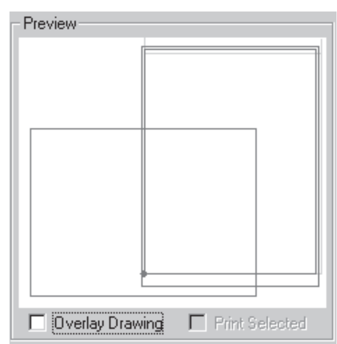
In the graphic, all of the geometry displays. If the *Overlay Drawing* check box is deactivated, only the selected rectangle displays in the *Preview* window.

This check box only becomes available when geometry is selected.

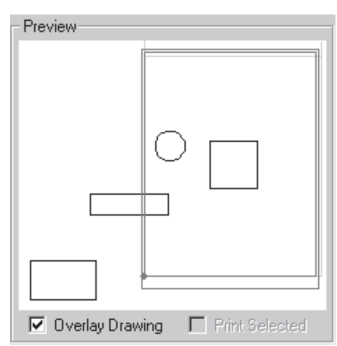
Preview Section Example

An example of using the option in this area might clarify how the *Preview* window and check boxes interrelate.

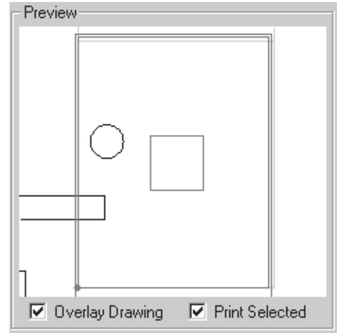
1. Draw some geometry.
2. Choose ***File>Print Layout***. The Drawing Size dialog box displays. The *Preview* window shows a red rectangle representing the object area.



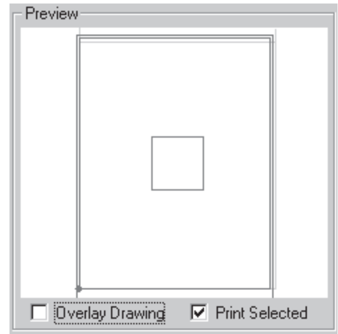
3. Check the *Overlay Drawing* box to display the actual geometry.
4. Click OK in the dialog box to close it.
5. Select an object within the page bounds.
6. Choose ***File>Print Layout*** to display the dialog box again.



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.

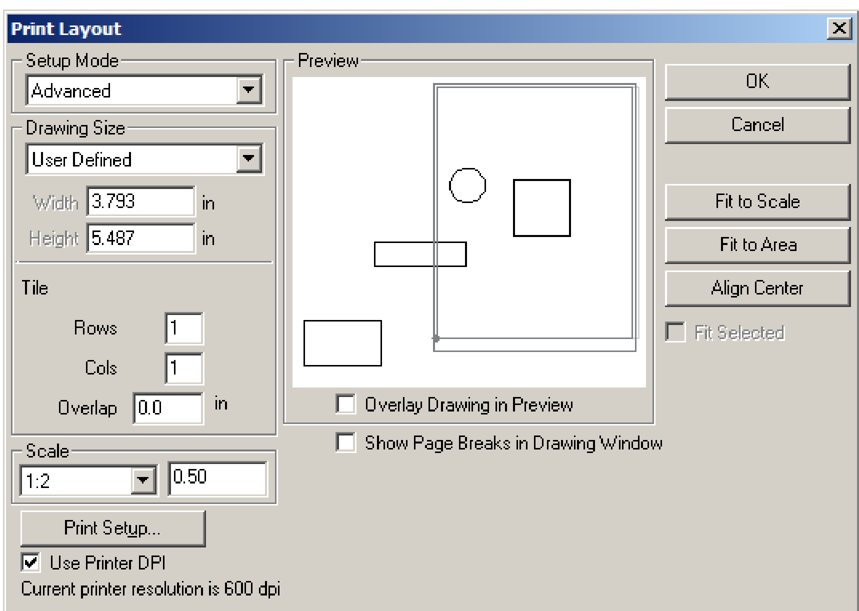


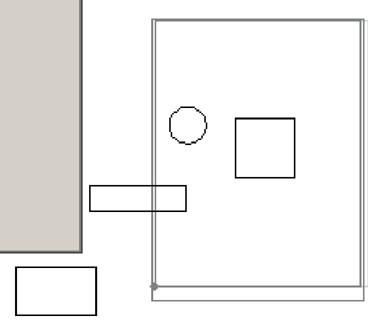
Utility Controls

The Drawing Size dialog box contains utility controls for setting up the 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 specifies whether to display the page breaks/boundaries in the drawing area. When this box is checked, the page bounds display.

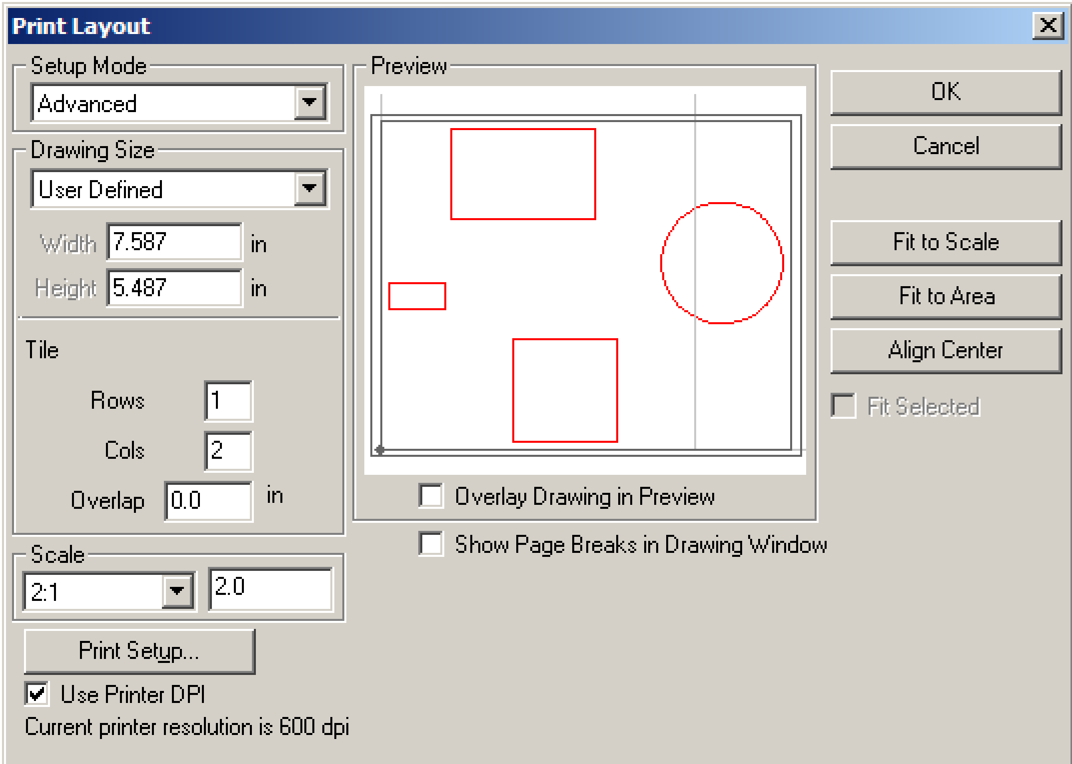




When OK is clicked and the dialog box is closed, the page boundaries can be moved. Place the cursor over the marker at the lower left corner of the page boundaries. The cursor becomes the move symbol (shown to the right). Drag the page boundaries to the new location. See the Moving the Print Boundaries section at the end of this chapter for more information.

Fit to Scale

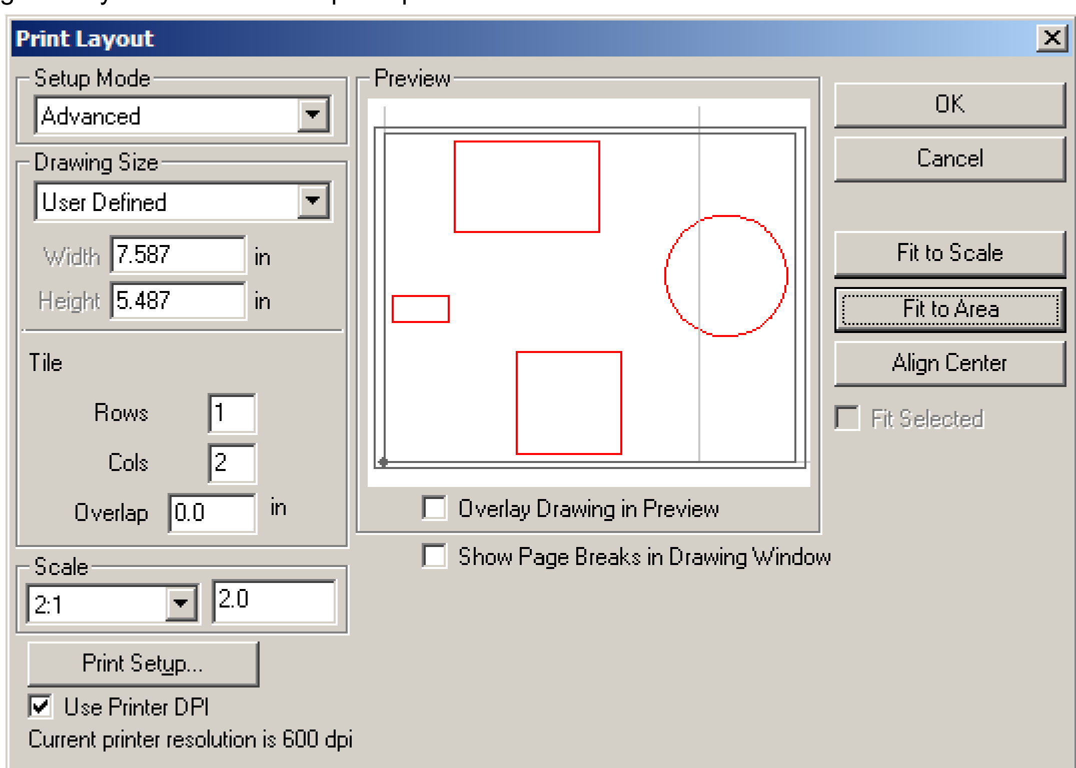
Clicking this button uses the current scale value and automatically changes the height, width and page boundaries to fit the geometry.



This button only displays with the *Advanced* mode.

Fit to Area

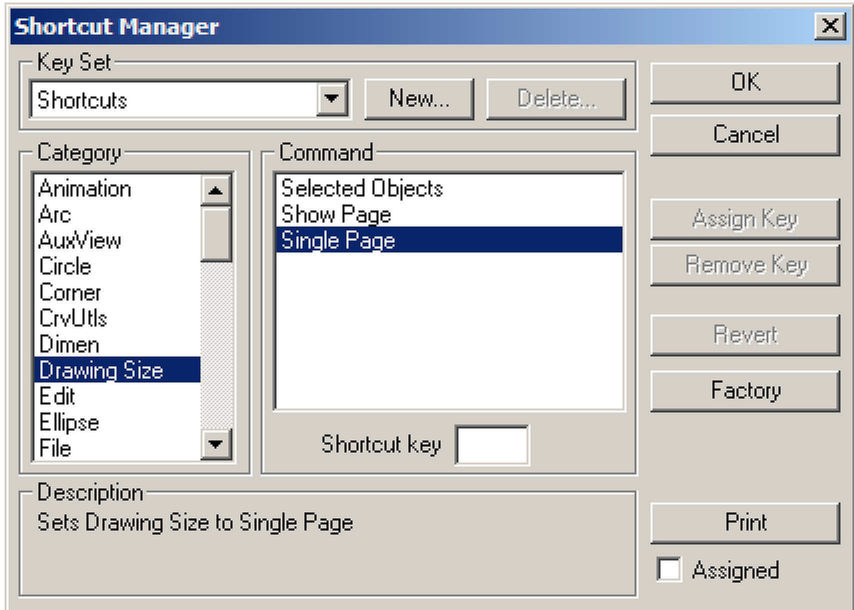
Clicking this button automatically computes the scale and page boundaries to fit the geometry to the entire tiled plot's printable area.



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, is in the Drawing Size category in the Short Cut Manager.



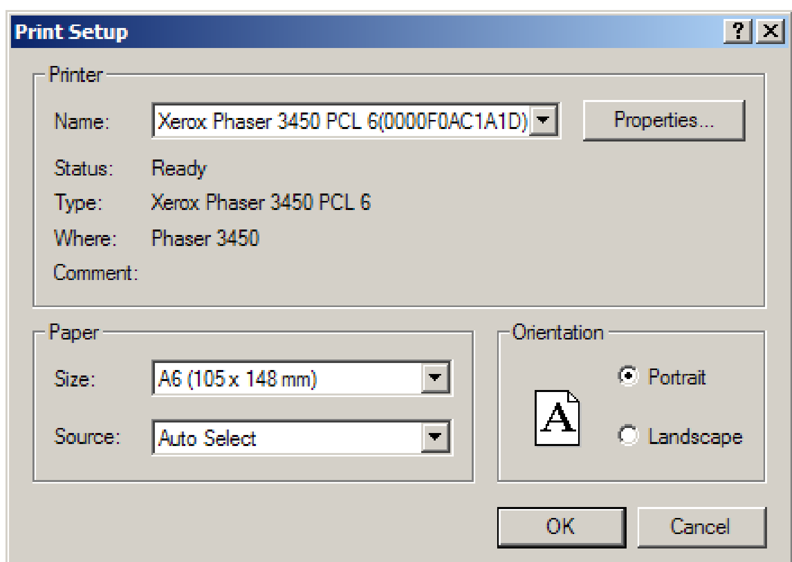
Using this key automatically chooses the *Single Page* mode, activates the *Fit to Area* function and *Shows Page Breaks*. This short cut may be assigned to any key.

Printing a Drawing

After setting up the page it is ready to be printed. Choose ***File>Print Setup*** (Windows) or ***File>Page Setup*** (Macintosh).

Print Setup (Windows); Page Setup (Macintosh)

Choosing this command in the File menu displays the printer setup window.



Choose the necessary settings for paper size and page orientation to agree with the settings in the Drawing Size dialog box. Click OK to save settings. See the printer manual for information about setting the printer options.

Print Command

CTRL+P (Windows); ⌘+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 the drawing to the appropriate size and reposition the print/plot region.

It is possible to specify tiling (printing on several pages to be pasted together) by choosing ***File>Print Layout>Advanced***.

Print to a File

It is also possible to print to a file rather than to a plotter or printer. This way, a plotter does not need to be attached to the computer. Someone else can plot the drawing without having Cobalt, Xenon or Argon on the plotter's computer. The type of plotter chosen when setting up the page determines the format of the plot file.

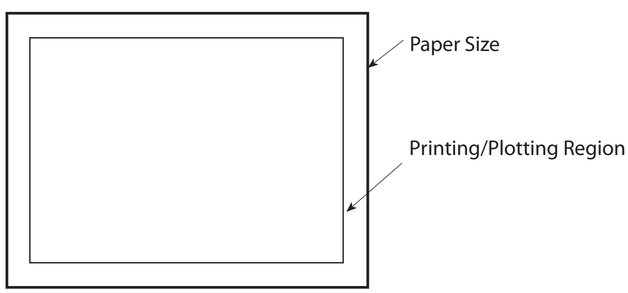
If a PostScript printer is chosen, the file format is Encapsulated PostScript. Use the HPGL language when selecting Hewlett Packard plotters. The computer that finally plots the file must have an application compatible with the file format of the printer or plotter.

Plotter Font

When using a plotter, specify the Plotter font for the text and dimensions on the drawing. It is also possible to generate special characters and accents as described in "Appendix B: Special Characters" on page 1.

Printing/Plotting Region

When ***File>Print*** is chosen, only the geometry within the page boundaries prints. View those boundaries by choosing ***File>Print Layout***. If the 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

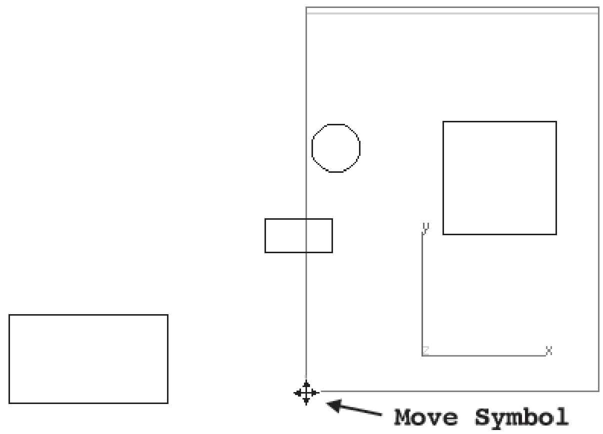
The size of this region is based on the paper size and the printer or plotter driver currently selected.

Moving the Print Boundaries

If the geometry is not contained within the page boundaries the page boundaries can be moved.

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 the 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.



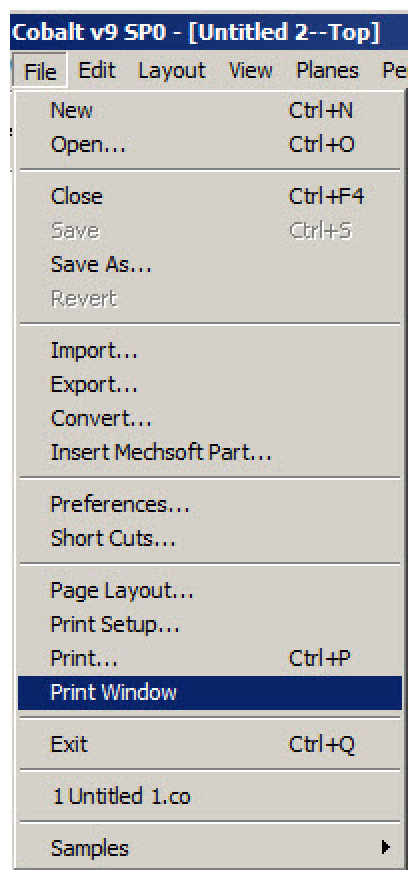
Printing and Rendering

Cobalt, Xenon and Argon print both wireframe and rendered geometry. To print rendered geometry, set the *Static Render* option in the Render Options dialog box to the desired mode.

If the printer supports color, *Static Render* must be set to Flat or Gouraud.

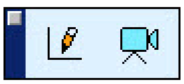
Print Window

The *Print Window* command in the File menu copies the image within the drawing screen and sends it to the printer.



Dimensionally Constrained Parametrics

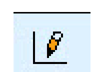
Cobalt Only



Cobalt supports 2D profile dimensionally constrained parametrics. This feature provides a mechanism for dimensioning entities to define distances and angles in order to establish geometric constraints between curves.

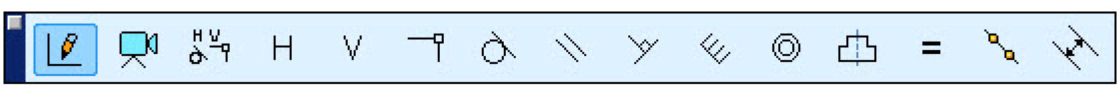
To access the **Constraint** tools and enter a 2D sketch mode go to the Window menu and choose *Constraints*.

Sketch Mode



To enter the 2D sketch mode to use dimensionally constrained parametrics, first choose the **Sketch** tool.

Clicking on this tool expands the tool palette, showing the tools available while in sketch mode.



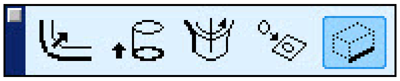
By entering the sketch mode, the application knows to adjust the user interface in such a manner that creating 2D sketches is intuitive. The sketch mode does the following automatically when creating a new sketch or modifying an existing sketch:

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 geometry is created and modified. It can be turned off in sketch mode through the popup menu activated with a right mouse click (Windows) or CTRL + mouse click (Macintosh).
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. Turn on Dimming. This feature is useful when working on faces of solids where edges not in the sketch plane are dimmed. In addition, if a sketch on the face of a

solid is modified, the part is rolled back to the point where the sketch was originally created.

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 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 the original geometry is moved, 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 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.

Exiting Sketch Mode

Exiting the sketch will return Cobalt to the settings before the sketch was entered. One other advantage of the sketch mode is that it temporarily suspends the regeneration of a dependent feature. This means that while in sketch mode it is possible to perform a series of operations such as adding and removing curves to the sketch that would otherwise invalidate downstream operations. When exiting the sketch mode, however, it is necessary to resolve a valid profile so that dependent features can then be updated.

Sketch Mode Tools

The tools available in sketch mode are listed below.

Sketch Tool

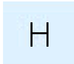
The **Sketch** tool creates or modifies an existing sketch.

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



This tool adds a horizontal constraint to a line. 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.

Pick the tool. The Message Line reads: *Horizontal Constraint: Pick line for horizontal constraint* **[Ctrl = Share X position** (Windows) or *Option = Share X position* (Macintosh)] *[Shift = Extend]*.

When applying a horizontal constraint, pressing the CTRL key (Windows) or OPTION key (Mac) aligns the geometry horizontally to a specific point on another line. The two chosen points will have the same x value.

Vertical Constraint Tool

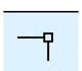
This tool adds a vertical constraint to a line.

Select one or more curves to apply a vertical constraint. Vertical is defined by the work plane y-axis.

Pick the tool. The Message Line reads: *Vertical Constraint: Pick line for vertical constraint [Ctrl = Share Y position* (Windows) or *Option = Share Y position* (Macintosh)] *[Shift = Extend]*

When applying a vertical constraint, pressing the CTRL key (Windows) or OPTION key (Mac) aligns 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

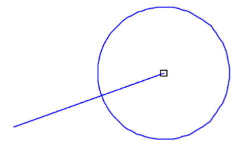
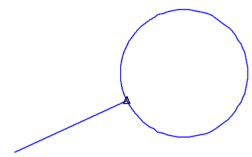
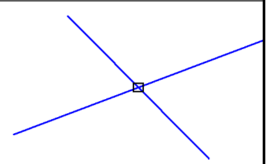


This tool adds a coincident constraint between two object positions.

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, midpoints, 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 and a fixed constraint is shown as a small rectangle about the two shared points.

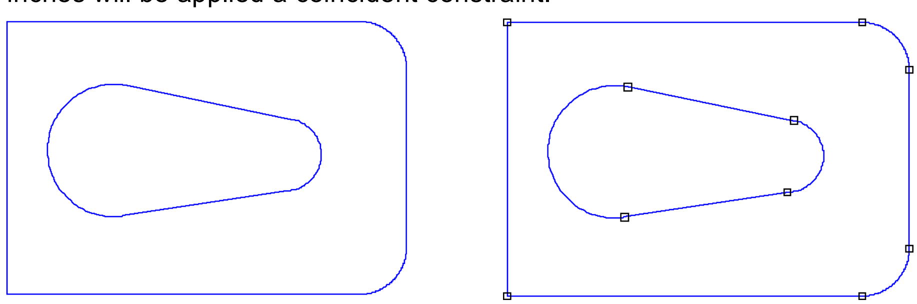
Some examples of coincident constraints:

End Point/End point	
End Point/Center Point	
Point On/End Point	
Midpoint/Midpoint	

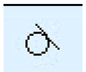
The **Coincident Constraint** tool provides several methods for attaching relationships. The first method is to follow the Message Line prompts:

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 draw a selection box around 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.



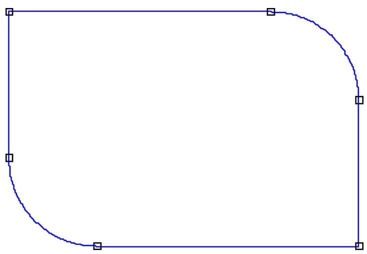
In addition, the **Coincident Constraint** tool allows for pre-selections. If any curves are pre-selected before clicking the tool palette icon, coincident relations are automatically applied to the selected objects.



Tangent Constraint Tool

This tool creates a tangent constraint between two or more curves.

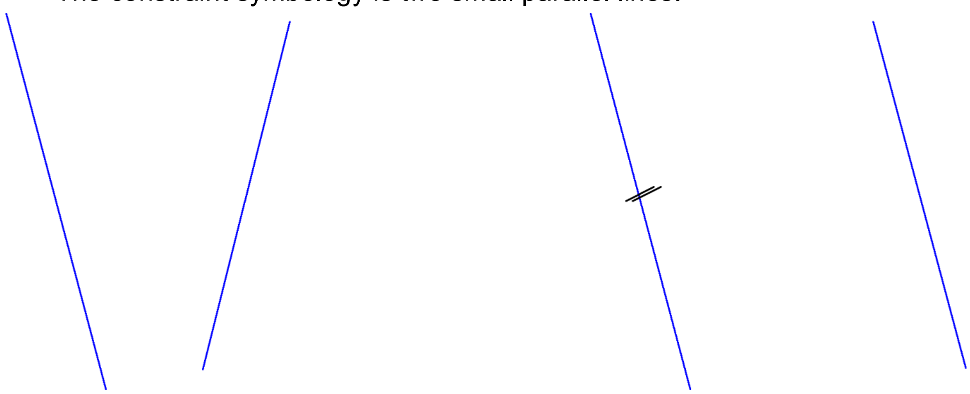
To use the **Tangent Constraint** tool simply select the curves to apply a tangent constraint. When selecting more than two curves, the tangent constraint tool only applies a constraint between curves that are already tangent within 1 degree. This constraint draws a circle to represent the existence of a tangent constraint.



Parallel Constraint Tool

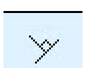
The **Parallel Constraint** tool adds a parallel constraint between two lines.

Only lines can be used in this tool; arcs, circles, ellipses, splines are ignored. The constraint symbology is two small parallel lines.



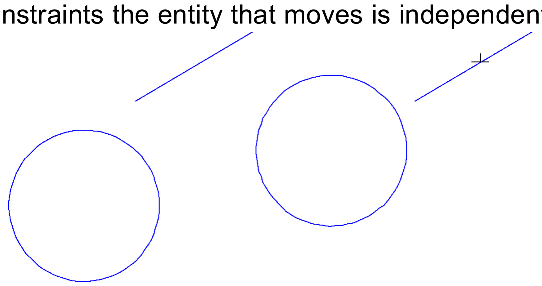
Note: The solver determines which line to move based on a set of rules involving the curve type, other curve relations and minimization of geometry movement. Therefore the order in which the curves are picked is irrelevant as to which curve is actually moved.

Perpendicular Constraint Tool

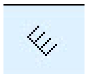


The **Perpendicular Constraint** tool creates a 90° angle between a line and another curve.

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.

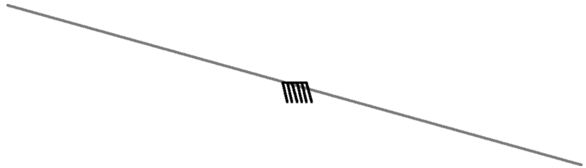


Fixed Constraint Tool

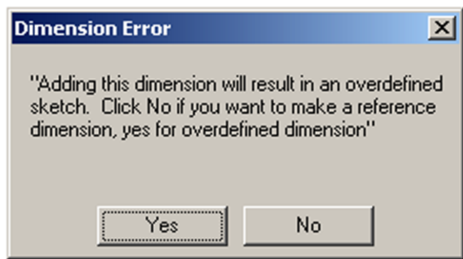


The **Fixed Constraint** tool locks the entity from being moved by the constraint solver.

It is possible to move the entity with the **Move** or **Transform** tools. A fixed object uses the fixed display color which is by default gray and whose symbol consists of a collection of slanted lines.



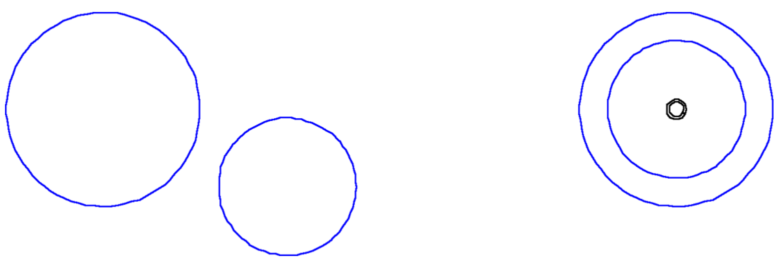
Note: Attempting to dimension a fixed entity will result in the display of the dimension error dialog box.



This error message implies that the position, direction, and length are fixed due to the fix constraint previously applied.

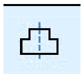
Concentric Constraint Tool

The **Concentric Constraint** tool creates a constraint that forces circles to share the same center point.

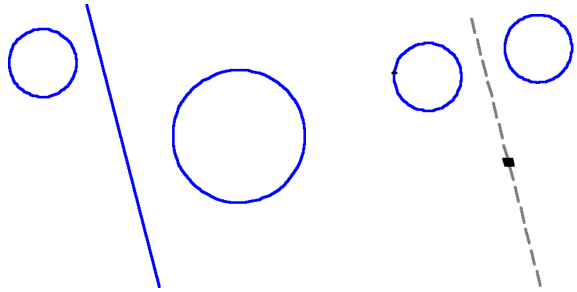


This concentric tool recognizes circles and points as valid selectable entities.

Symmetric Constraint Tool

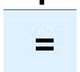


This tool creates a symmetric constraint between entities of asimilar type and symmetry line.

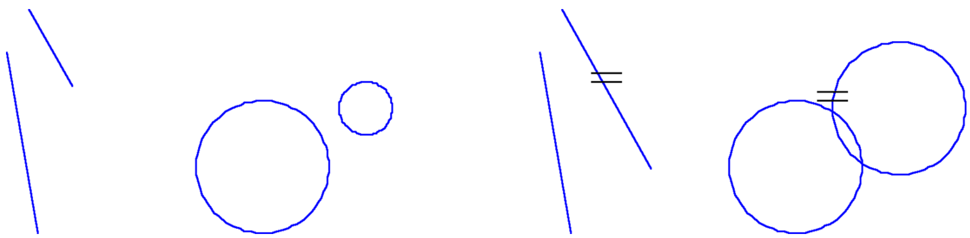


Note: Use the mirror tool in the transformation tool palette while in the sketch mode to automaticaly apply mirror constraints at the time the mirror operation is performed.

Equal Constraint Tool

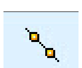


The **Equal Constraint** tool applies an equal distant or radius constraint between two entities.

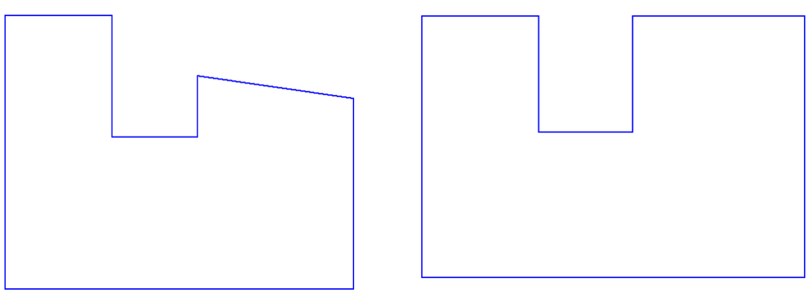


This tool works only with lines or circles. When selecting two lines, the lengths are forced to be the same for the two entities. In the case of circles, the same radius value is applied between the selected circles. As with many constraints, this operation is independent of which curve is selected first due to the method in which the solver finds solutions.

Colinear Constraint Tool

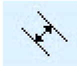


The **Colinear Constraint** makes two lines colinear.

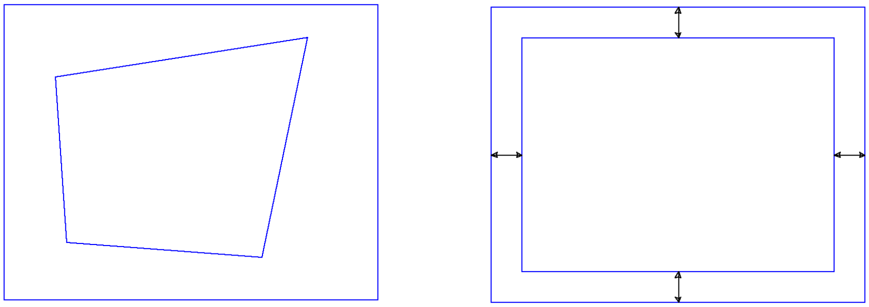


Colinear implies that the two resulting lines have the same direction and lie within the same line. It does not imply anything regarding their lengths.

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, use the edit objects dialog to change an existing offset value. Click on the constraint symbol to display this edit page.

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.

Constraint Animation Tool

The **Animation** tool will animate a sketch by modifying a dimension value through a range of values. To use the tool, select a dimension that was used within a sketch and then choose the **Constraint Animation** tool.

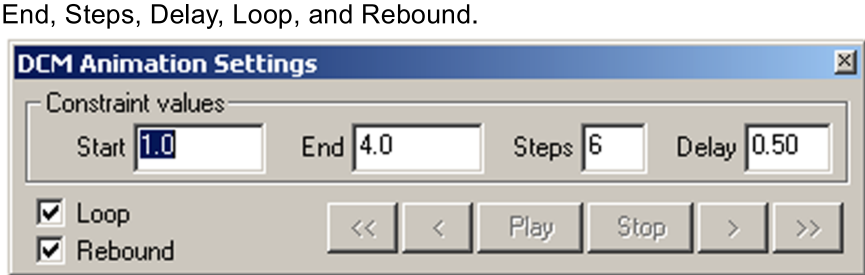
This tool is always visible in **Constraint** palette and can be accessed either within sketch mode or from outside. See the next section on Dimensionally Constrained Animation Controller for more details on this tool.

Note: If a sketch is animated while in sketch mode, only the sketch is updated. If a sketch is animated outside of the sketch mode, dependent surfaces and solids update accordingly.

Dimensional Constraint Animation Controller

Dimensional Constraint Animation Settings

The Dimensional Constraint 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 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 the steps are 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 transit from start to end in the specified number of steps, and then continue by “rebounding” from End to Start in the same number of steps.

Running the Animation

Once the settings are made, select the dimension to animate. The animation will start automatically. Control the execution of the animation using the VCR style buttons in the dialog box.

Deleting Constraints

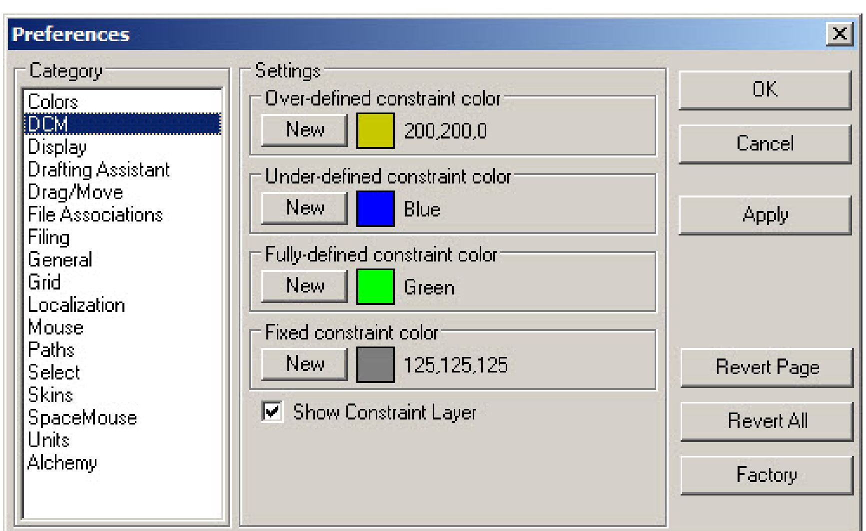
To delete any constraint, simply select the constraint symbol and press the DELETE key or use *Cut* from the Edit menu.

Verifying Constraint Relations

To confirm the entities used in a constraint relationship, simply move the 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.

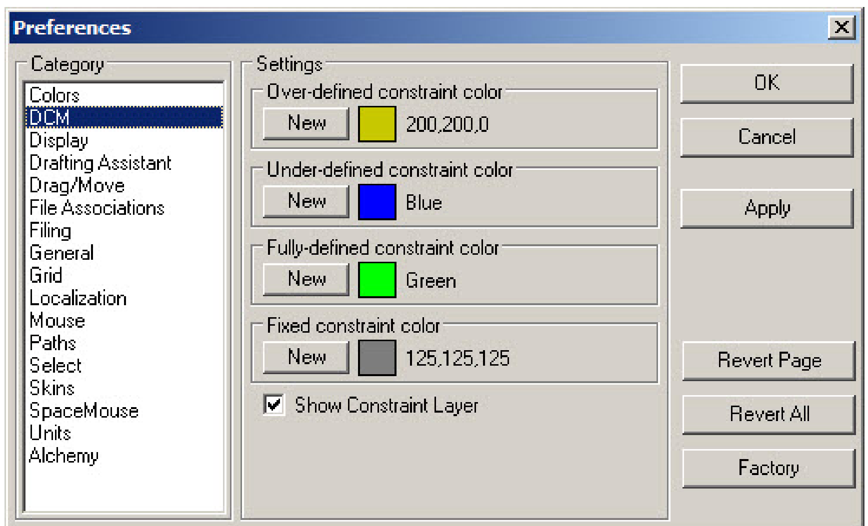
Over-Defined Constraint Color Any entity in a sketch that has too many constraints or creates an ambiguity for the solver will display in the Over-defined Constraint color. It is best to resolve any over-defined sketch entities before exiting a sketch.

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.



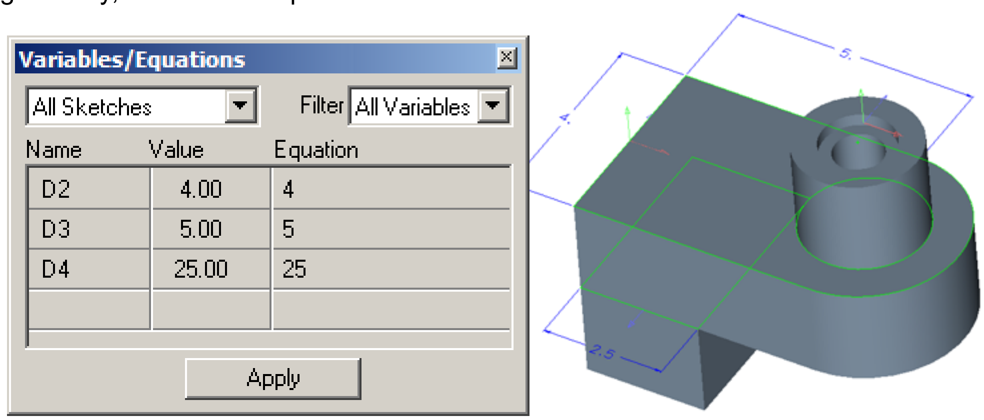
Exiting Sketch Mode

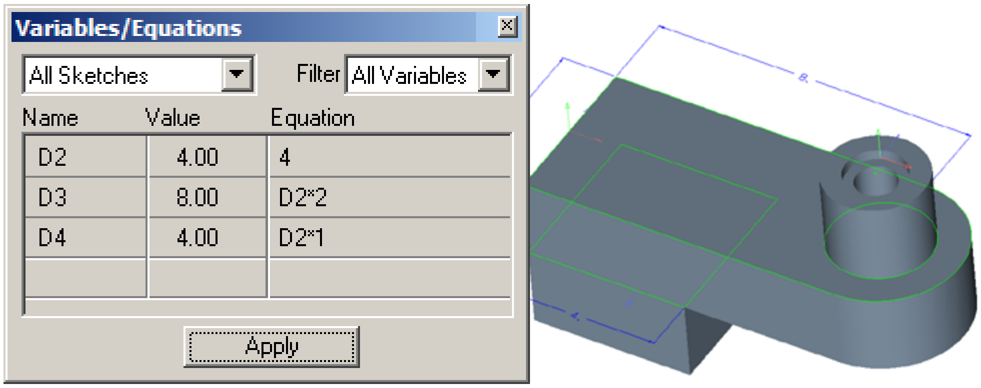
Once a sketch is created and constrained, to exit Sketch Mode simply click the blue Exit icon in the lower right-hand corner of the drawing window:

Variables and Equations

Cobalt Only

Cobalt supports assigning variables and equations to dimensions applied while in the constraints sketch mode. Having geometry that has been constrained using equations is an easy way to manipulate the geometry, see the examples below.





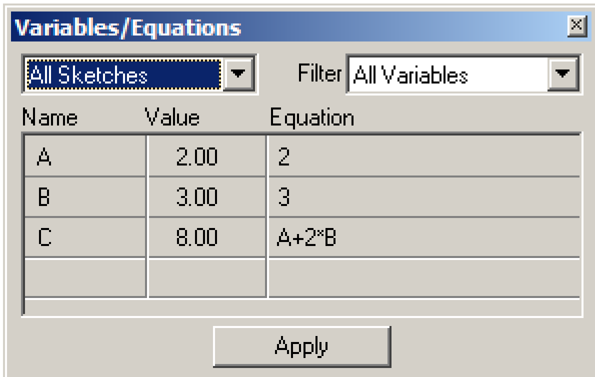
Using Equations & Variables

Once the parametrically constrained part is completed, edit the dimensions applied to the part. This can be done using the equations dialog box or through the Edit Objects box. Both methods are described below. To edit dimensions or change the names in either dialog box, or through the Edit Objects box. Both methods are described below. To edit dimensions or change the names in either dialog box, click on the field and type in the new value.

Using the Variables/Equations Dialog Box

To open the Equations dialog box go to the Window menu and choose Equations.

The Variables/Equations dialog box supports all kinds of mathematical expressions. To use one dimension as a reference to another 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: Mathematical Operators.



The Variables/Equations dialog contains the following options:

All Sketches This pull down menu contains the list of every sketch within the file.

Filter Sorts by the type of dimensions shown in the dialog box.

Name Displays the name of the dimension.

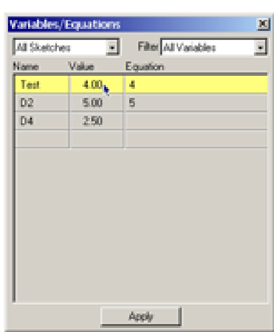
Value Shows the current value for each dimension.

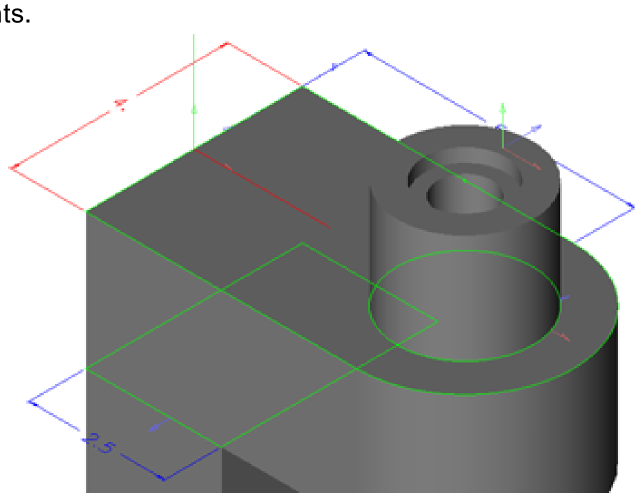
Equation Displays the mathematical expression that will define the dimension.

Highlighting Dimensions in the Sketch

Use the following steps to highlight the dimension on the screen.

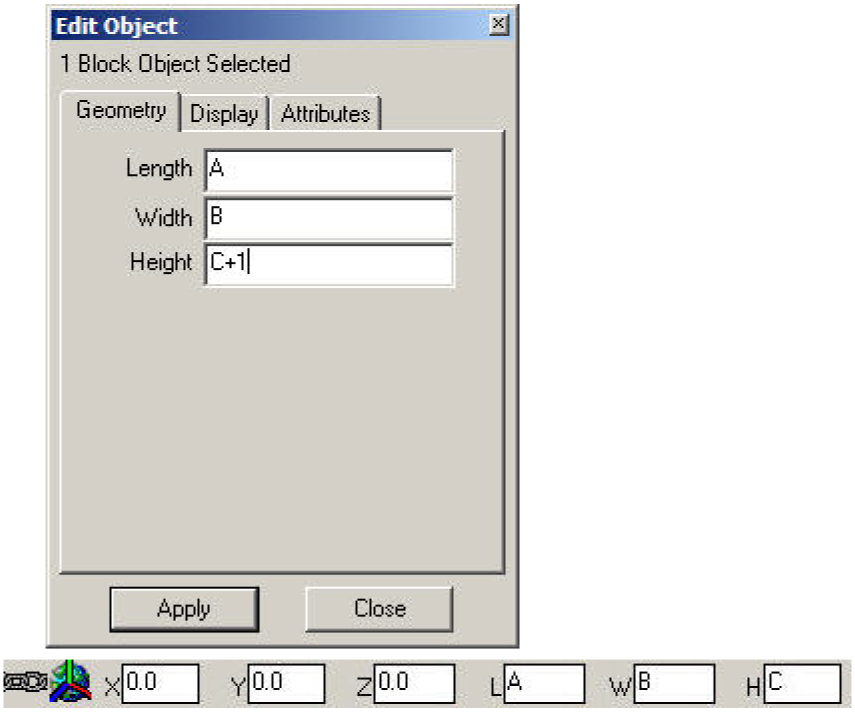
1. Move the cursor to the Dimension Value field to be highlighted.
2. Click and hold down the mouse button.
3. The dimension highlights.





Using the Edit Objects Box for Variables & Equations

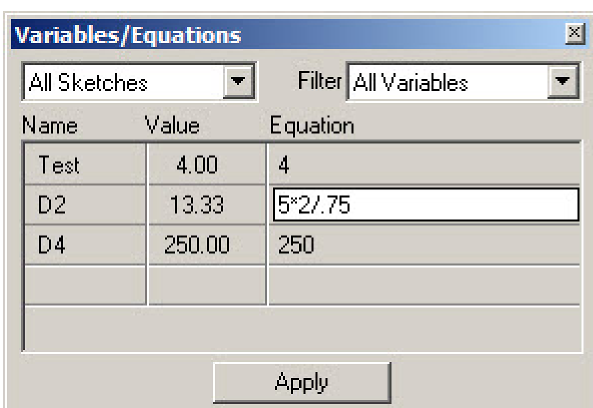
Variables and equations can be referenced within Cobalt in the Edit Object box and in the Status Line.



Defining Equations

Using mathematical expression or operators, change or resolve the size of the part.

1. Click in the equation field of the dimension to be changed.
2. Add a mathematical expression, such as D1*2/3.
3. Click Apply for the changes to take place.



Note: The *Undo* command (CTRL+Z (Windows) or COMMAND + Z (Macintosh)) cannot be used for expressions applied in the equations dialog box. Repeat the steps above to change the dimension.

Nested Equations

Equations can reference other equations within Cobalt for more robust equation-driven parametrics. The results of one equation can be used in other equations.

Conditional Equations

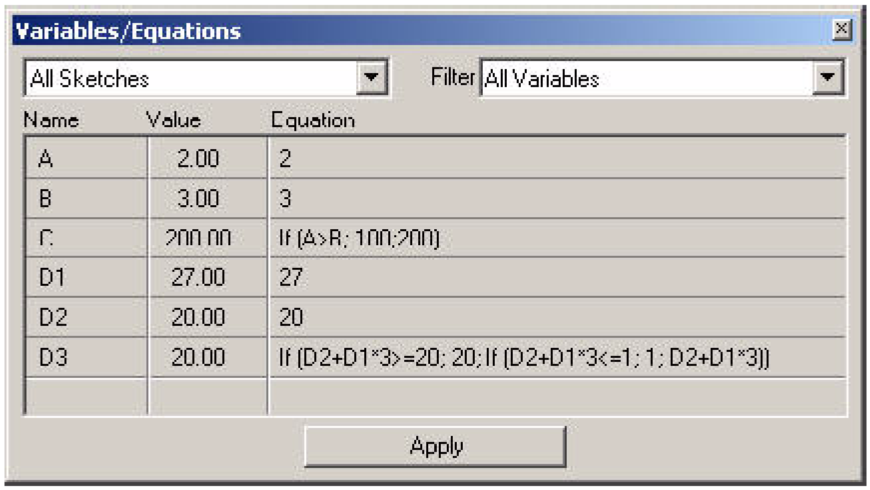
Cobalt’s parametrics contain equations for more complex and intelligent geometric constraints. This enables adding if-then-else statements to the parametric constraints bounding the upper and lower limits of geometry.

For example, a sketch has the dimensions: D2+D1x3.

If the maximum constraint of these dimensions is 20 and the minimum constraint is 1, then a conditional equation could be written:

If (D1x3>=20;20; If (D1x3<=1;1;D1x3))

This bounds the dimension at 1 and 20.



Syntax

Conditional equations include:

IF (condition; exp1; exp2)

This means that if [condition] is true (is not equal to zero) then expression exp1 will be used, else exp2.

Logical Operations

Logical operations include AND, OR, NOT, <, >, <=, >=, and =.

Results of logical operations are 1.0 or 0.0.

Priority of Operations

1.( )

2.NOT, unary +, unary -

3.^ (power)

4.*, /, %

5.+, -

6.<, >, <=, >=

7.=

8.AND

9.OR

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, shortcut 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 preferences are 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.

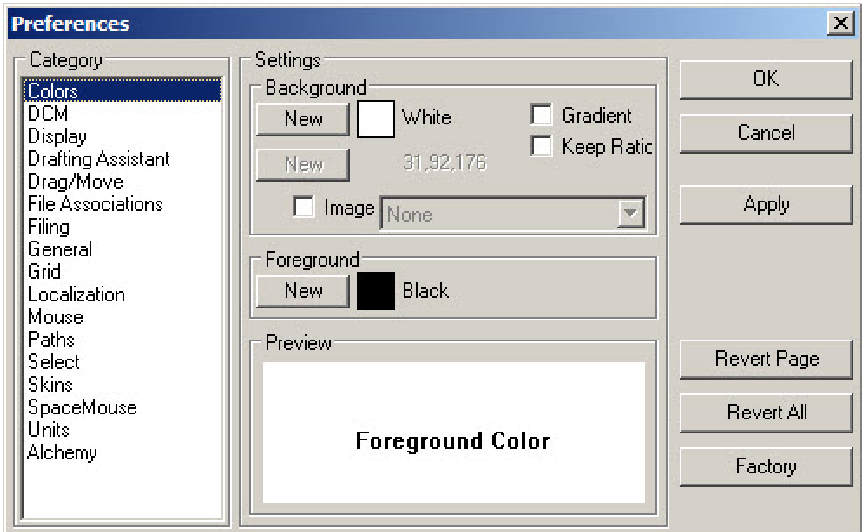
This chapter covers the following topics:

- Preferences
- 3Dconnexion SpaceMouse Information
- Pen and Dimension Preferences
- Shortcuts

Preferences

Cobalt, Xenon and Argon all save preferences for a particular session. The programs, however, do not support saving preferences with a specific file.

When more than one file is open during a particular 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 specified in this session and closes the Preferences dialog box.

Cancel — Closes the dialog box without saving all changes.

Apply — Applies the change instantly and leaves the dialog box upon for further selections.

Revert Page — Undoes changes made to the current preference group.

Revert All — Undoes changes made to any preference group.

Factory — Resets the program to the actual factory defaults for all Preferences, not just the current Preference pane.

The *Category* list includes:

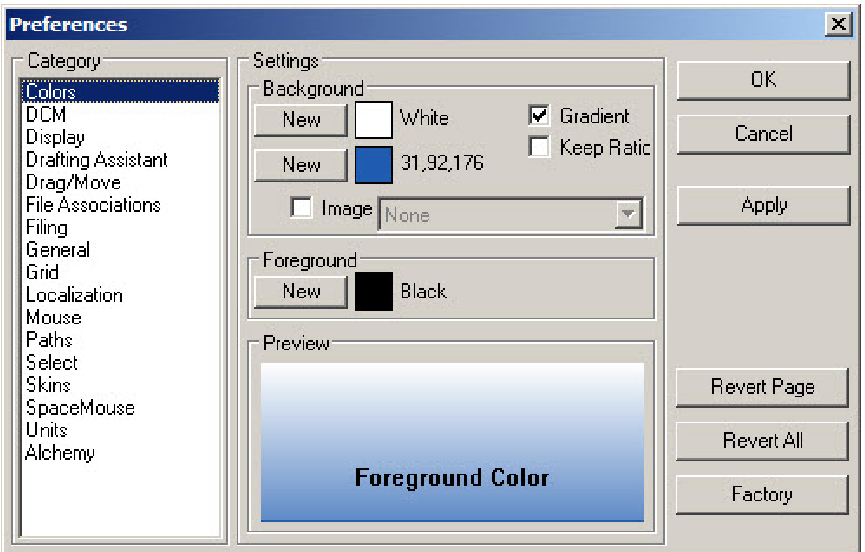
- Colors
- Dimensional Constraint Manager (DCM) (Cobalt Only)
- Display
-
- File Associations
- Filing
- General
- Grid
- Localization
- Drag/Move

- Paths
- Select
- Skins
- SpaceMouse
- 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 sets the background color for the drawing area. The current colors are displayed in the window. The standard color names or the color values display to the right of the view window. To change the background:

1. Click New. The color palette is displayed.
2. Choose the background colors.
3. For a solid color background uncheck the Gradient box.
4. To use an image as a background, check the Image box and select the desired one from the drop down list or browse to select it from the system. Click OK.

Foreground

This option sets the foreground color for the drawing area (specifically the location indicator and the indicator separator lines). The current color is displayed in the window. The standard color names or the color values display to the right of the view window. To change the foreground:

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.

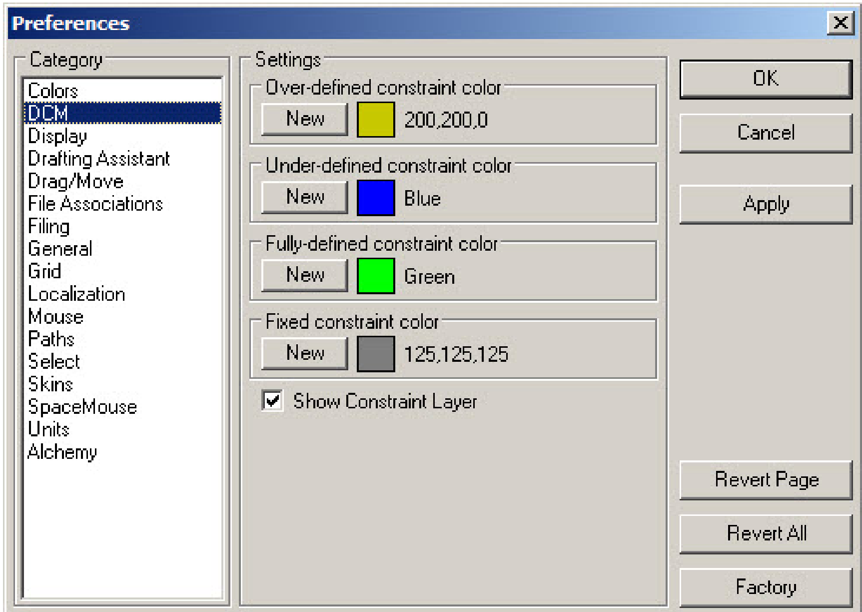
Preview

The Preview window displays the background and foreground choices.

Dimensional Constraint Manager (DCM) (Cobalt Only)

Cobalt Only

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



Over-defined Constraint Color

Set this field to the desired color for over-defined constraints.

Under-defined Constraint Color

Set the field to the desired color for under-defined constraints.

Fully-defined Constraint Color

Set this field to the desired color for a sketch that is fully defined.

Fixed Constraint Color

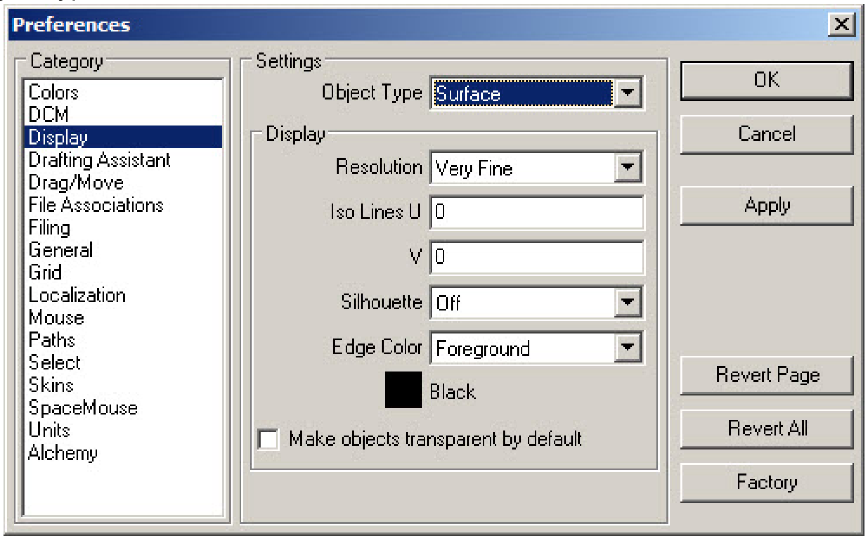
Set this field to the desired color for fixed constraints.

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.

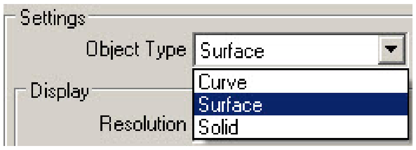
Display

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



Object Type

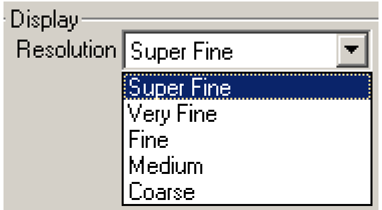
This sets the appearance of curves, surfaces and solids. Each object type offers different display options.



Display

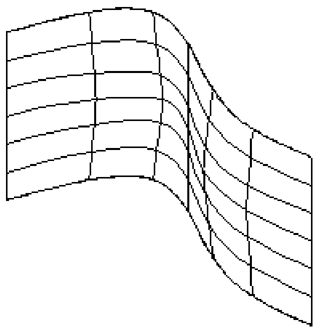
Choose an option for each object type (curve, surface and solid). Display options vary according to the object type and may include: *Resolution*, *Iso Lines*, *Silhouette* and *Edge Color.*

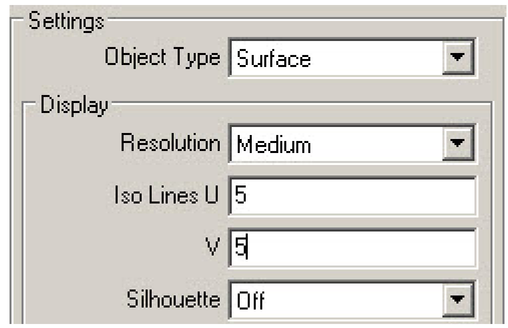
Resolution (Available for all object types.) Controls how accurately an object's curves appear. It is possible to set the curve resolution to Coarse, Medium, Fine, Very Fine and Super Fine. An object with a Coarse resolution draws quickly but may be visually less appealing. An object with a Fine resolution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



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.

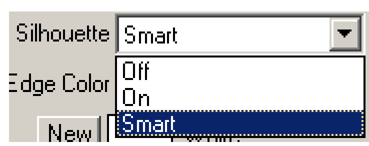
The left graphic below shows a surface with both U and V Iso lines set to five (5).





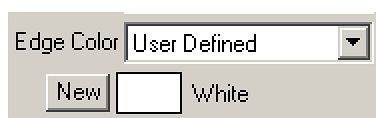
Silhouette

(Available for Surface and Solid object types.) This 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 sets the edge color of rendered mesh objects separate from the entity itself.

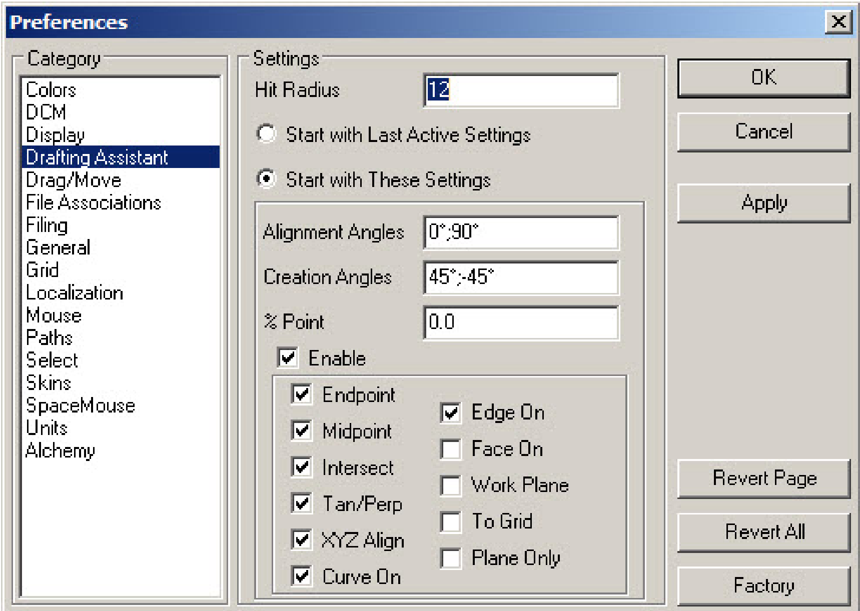


Select one of four options from the pull-down menu, Foreground, Background, Entity and User Defined. To specify a User Defined color, click on New, choose a color in the palette and click OK. The new color is displayed in the Edge Color window with its RGB values.

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.*

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 the mouse is clicked.

Alignment Angles

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

To automatically display a dynamic construction line through a point, move the pointer to the point to activate it. (A diamond appears.) Up to eight points are active. When the ninth point is activated, the first one deactivates.

Creation Angles

These define angles for the Drafting Assistant's dynamic construction lines, displayed while creating geometry. (These lines are not part of the list of lines generated from the eight active points.) The defaults are **+**45**°** and **-**45**°**. Separate values by semicolons.

% Point

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

Drag/Move

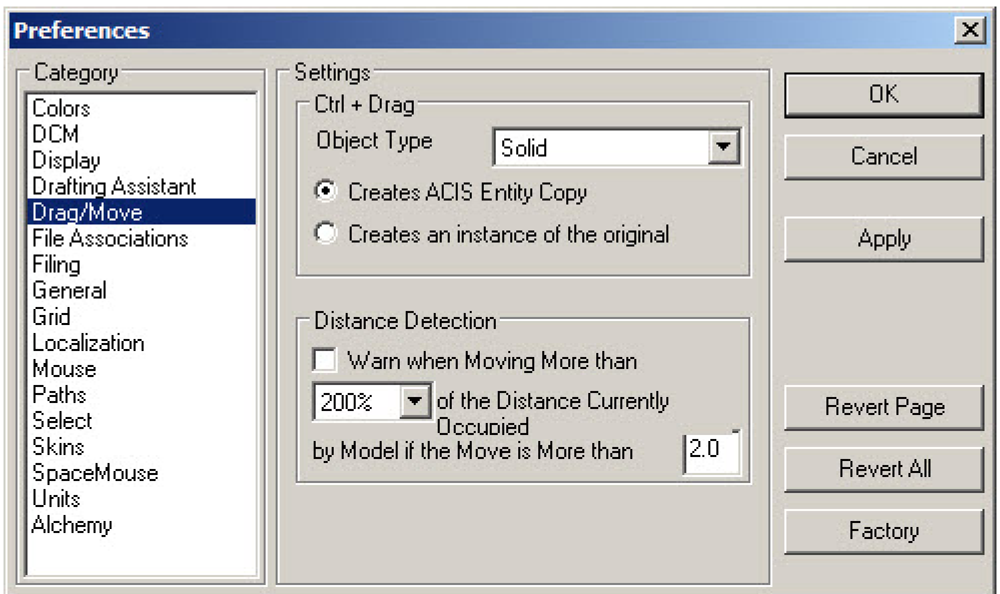
CTRL+Drag or OPTION+Drag

This dialogue box controls what happens when selecting a part and copying it by dragging while holding the CTRL key on Windows or the OPTION key on Mac.

In the Object Type pul-down it is possible to designate different behaviors for both surfaces and solids.

Create ACIS Entity Copies This creates copies of the original without any history.

Creates Instances of the Original These share the original part's history. Any changes made to the original, flow through to the instance part.

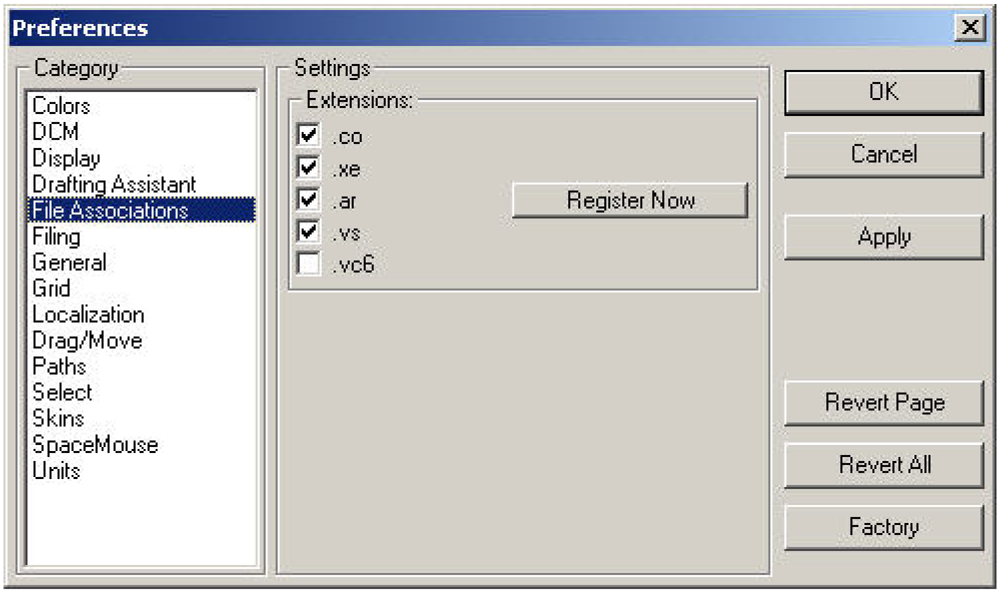


Distance Detection

Check this to display a notice if an object is being moved more than the range specified.

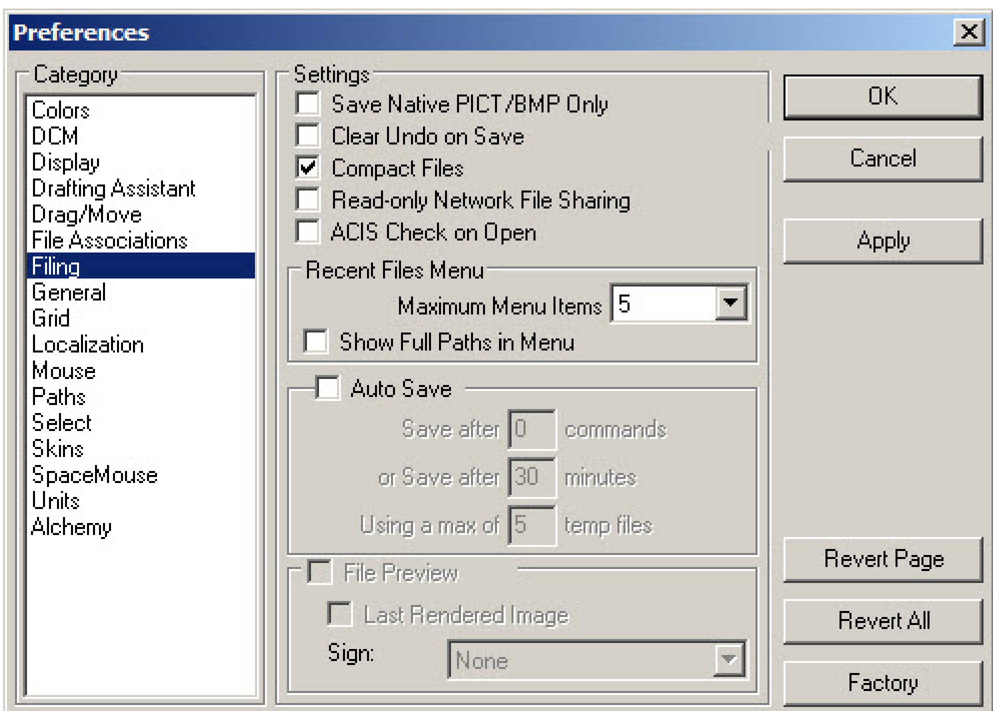
File Associations

(Windows only.) This re-establishes the association between this application and these file types.



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 disables multiple platform picture support. Saving only the native format will reduce file size.

Clear Undo on Save

Activating this option flushes 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.

ACIS Check on Open

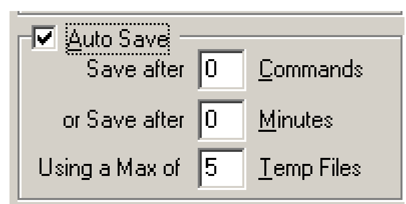
Check this for the application to check every entity containing ACIS data upon opening. If errors, such as unresolved links, are found, the program attempts to fix them, taking longer to open. For faster open times, uncheck this box and forgo the ACIS data checks.

Recent Files

This option sets 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.

Maximum Menu Items	The drop down list specifies 0 to 20 file names. The default number is five.
Show Full 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 the 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 program folder. Backup files are numbered sequentially. If the file is not yet saved, the backup will be named untitled.

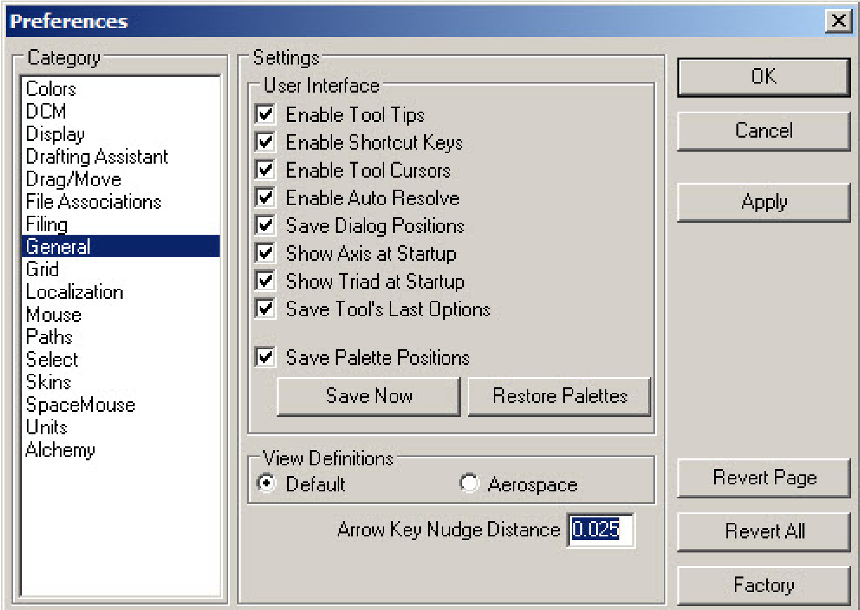
If auto save is on but no geometry has been modified in the drawing since the period that the last auto save operation occurred, auto save does not activate.

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 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 Shortcut Keys Enables setting the keyboard short cut keys through ***File>Shortcuts***.

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 Resolve 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.

Show Axis at Startup Shows the coordinate axis when the program launches.

Show Triad at Startup Shows the coordinate triad when the program launches.

Save Tool’s Last Options For the tools with multiple options, the last option chosen saves as the default.

Save Palette Positions Saves the tool palette positions and displays its status when exiting. The next time the program launches, the palettes display in their previous positions.

Save Now Immediately saves tool palette positions, pen color, fill color, dialog box locations and display status.

Restore Palettes Returns all palettes back to their factory default position. If palettes are out of the drawing area, use ***Windows>Organize Palettes*** to show them in the drawing area.

View Definitions

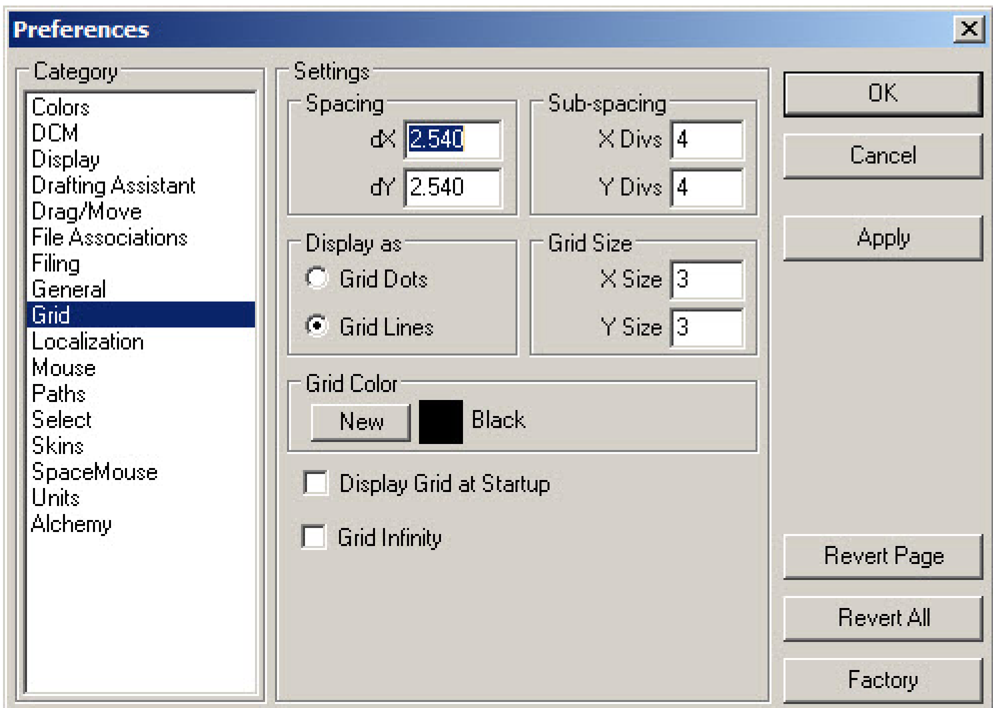
Cobalt, Xenon and Argon all support two different common view definitions, *Default* and *Aerospace*. Select the *Default* option to use view definitions commonly used for mechanical drafting. Select the *Aerospace* option to use view definitions commonly used for aerospace lofting.

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, startup 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

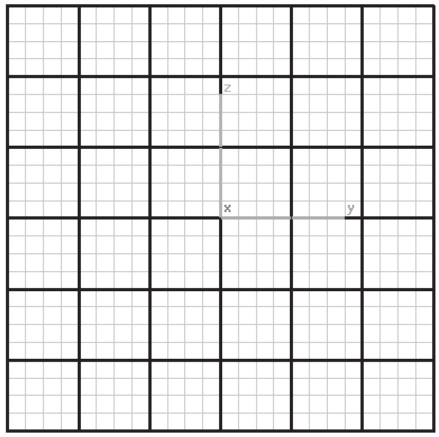
X Divs and *Y Divs* specify the number divisions for the grid to which geometry snaps in the drawing when the Snap to Grid command is activated.

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 the created in any plane. For example, by changing 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 at startup of this Designer Elements program.

Snap to Grid at Startup

Check this box to activate the snapping function at program startup.

Grid Color

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 use of language and decimals versus commas in this Designer Elements program.

Use Commas as Decimal

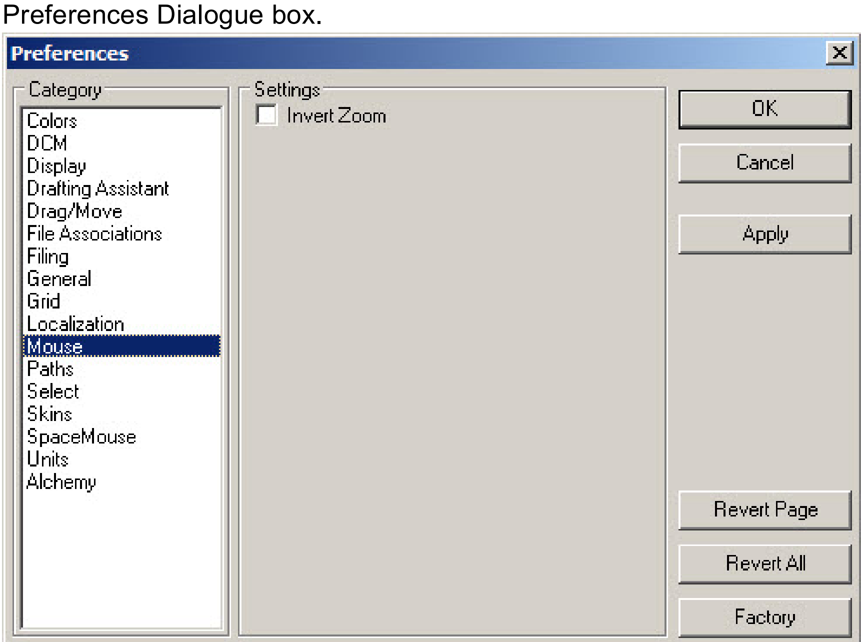
Checking the *Use Comma as Decimal* option allows international users to display numbers according to their numerical standards.

Language

The drop down menu provides the list of languages to choose. There is an Auto command which chooses the language of the preferences of your computer.

Mouse

Invert Zoom of your mouse by checking the *InvertZoom* option on this page of the Preferences Dialogue box.



Paths

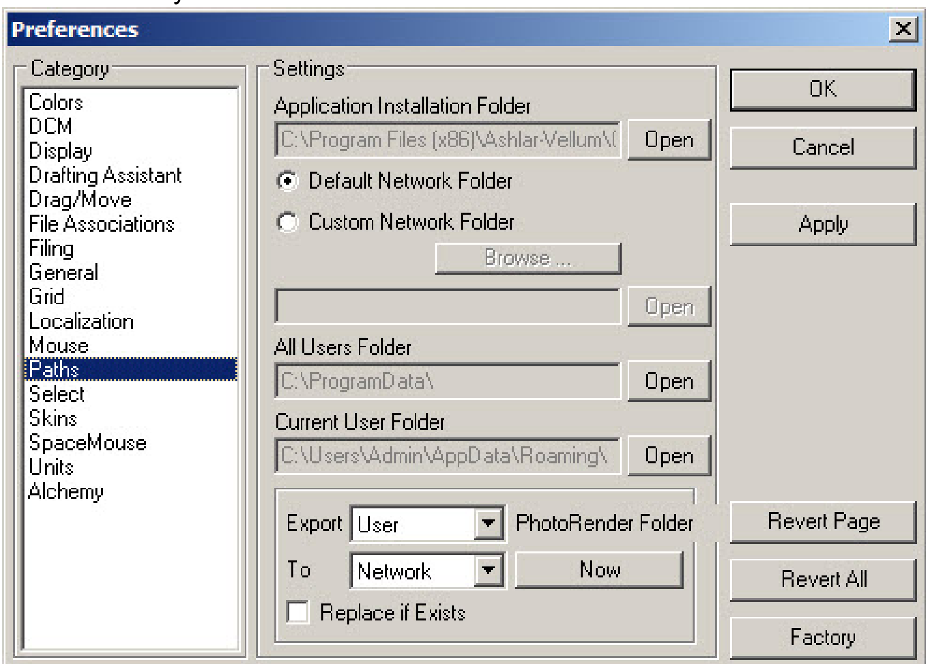
This page is used mostly by system administrators and Ashlar-Vellum tech support specialists.

It is useful for workgroups to set up a custom network folder for the PhotoRender, BOM and Layout libraries.

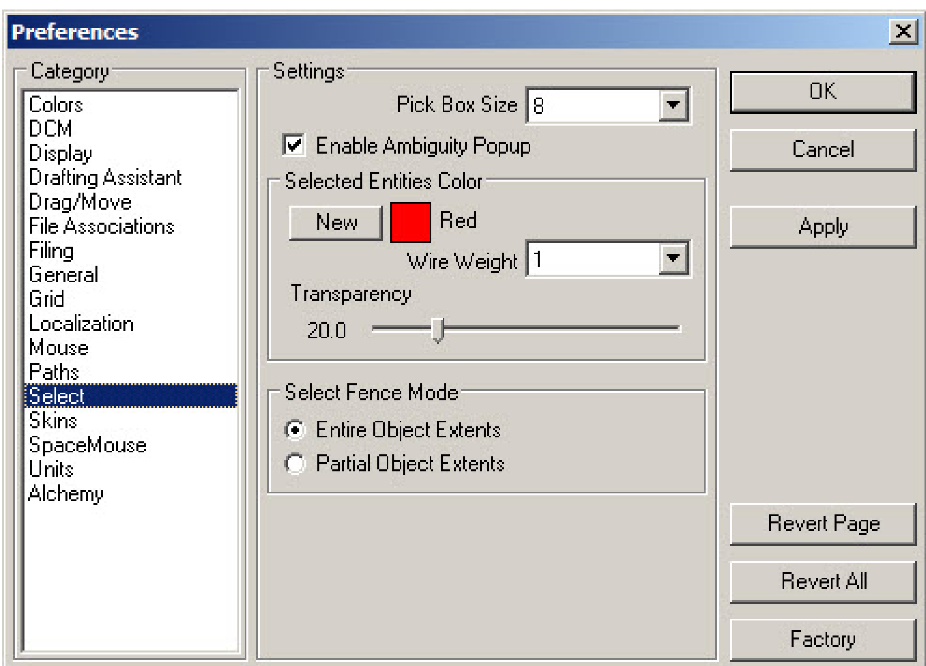
It is also a useful tool for allowing others in the group to have access to common data. For example, if a user creates a new material or adds a texture, this can be exported to the Custom Network Folder for use by others without copying it to each local machine.

Tech Note: *Write-permission is required for destination directories.*

The Application Installation Folder, Default Network Folder (if used by default by the operating system), All Users Folder, and Current User Folder show the automatically generated paths. Custom Network Folder is created by the system administrator and can be used by all the users of the local network.



Select



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

Pick Box Size

When an object is selected, 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. To specify the area use 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 the 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 sets 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.

Wire Weight

Select a curve and the line weight of that curve is increased according to the setting in the drop down box. The weight is specified in pixels. Choose between zero, which does not increase the weight when selected, and four, which increases the selected line weight by four pixels.

Transparency

Select a surface or solid object and the surface or solid becomes transparent. The Transparency slider controls how transparent the object becomes. A value of zero means no transparency is added to the selected object. A value of 100 means the selected object becomes fully transparent, showing only the edges.

Selected Points

When a point is selected the entire curve will highlight, however, the edits will only affect the selected point.

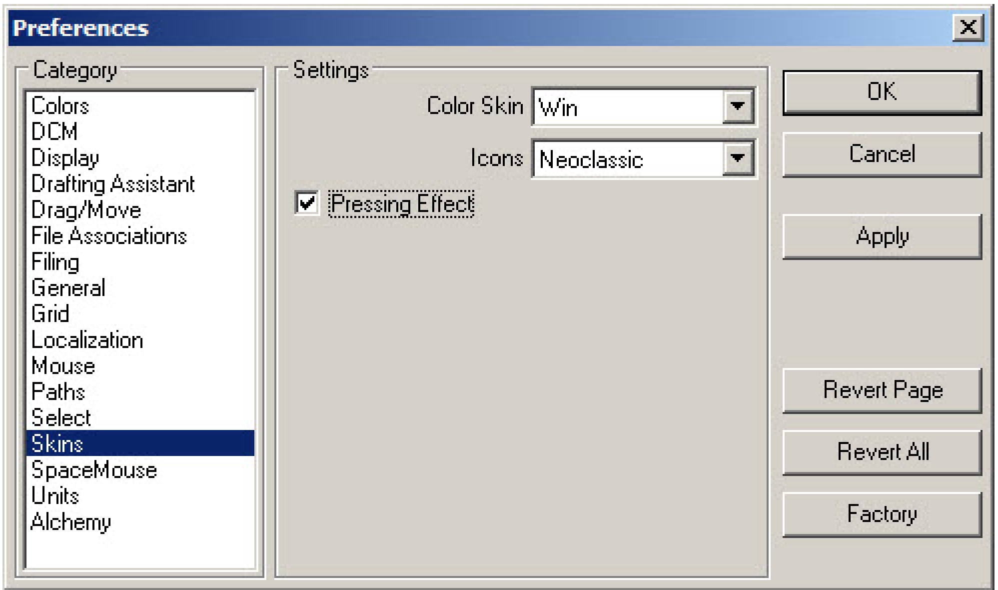
Select Fence Mode

The program supports two modes 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.

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 the selection fence covers a control point for a circle, only the center point is selected rather than the entire circle.

Skins

Choose the look of the user interface using the Skins page of the Preference settings.



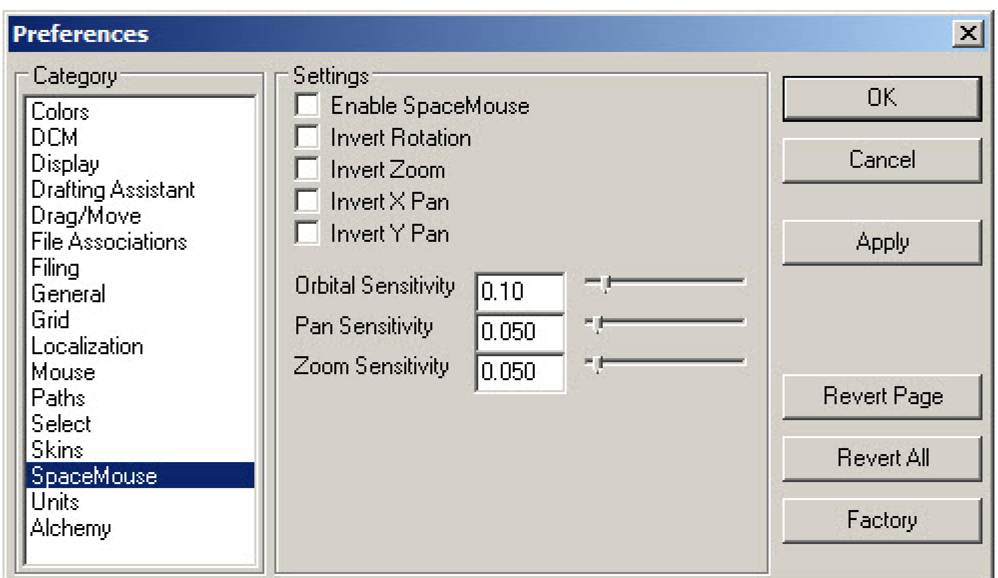
Color Skin Select the color theme desired.

Icons Select the icon style. The default is Modern.

Pressing Effect Check this box for the selected icon to appear as a pressed button.

SpaceMouse

The SpaceMouse Preferences control the interaction for all 3Dconnexion motion control devices including the SpaceMouse, SpacePilot, SpaceBall and Space Traveler. The intuitive reaction of individual users to the controller's operation varies widely, so optional settings are provided.



Enable SpaceMouse Check this box to enable interaction between Ashlar-Vellum programs and the SpaceMouse.

Invert Rotation Check this box to invert the SpaceMouse rotation vector, altering the default rotation behavior of the scene with respect to the SpaceMouse motion controller.

Invert Zoom Check here to invert the SpaceMouse zoom vector, changing the behavior of the motion controller with respect to the scene.

Invert X Pan When set, the behavior of the SpaceMouse X motion component vector is inverted with respect to the scene.

Invert Y Pan When checked, the behavior of the SpaceMouse Y motion component vector is inverted with respect to the scene.

Orbital Sensitivity The Orbital Sensitivity slider adjusts the sensitivity of the programs to the orbital input from the SpaceMouse motion controller. Zero disables the orbital input. Selecting a larger value increases sensitivity, requiring less physical motion on the motion controller to achieve scene changes.

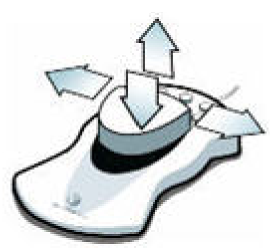
Pan Sensitivity The Pan Sensitivity slider determines how sensitive the program is to the pan input from the SpaceMouse motion controller. Zero disables the pan input (X and Y). A larger value increases sensitivity, requiring less physical motion on the SpaceMouse to achieve scene changes.

Zoom Sensitivity The Zoom Sensitivity slider lets you set how sensitive the program is to the zoom input from the SpaceMouse motion controller. Zero disables the zoom input. Selecting a larger value increases sensitivity for physical motion of the SpaceMouse to achieve scene changes.

3Dconnexion SpaceMouse Information

Working with the Motion Controller in 3D Applications

Pan Pull up and push down to move the model vertically.



Move left and right, keeping the controller head parallel with the work surface.

Rotate

Tilt the controller head left and right or forward and backward to rotate accordingly (X-Z axis).

Zoom

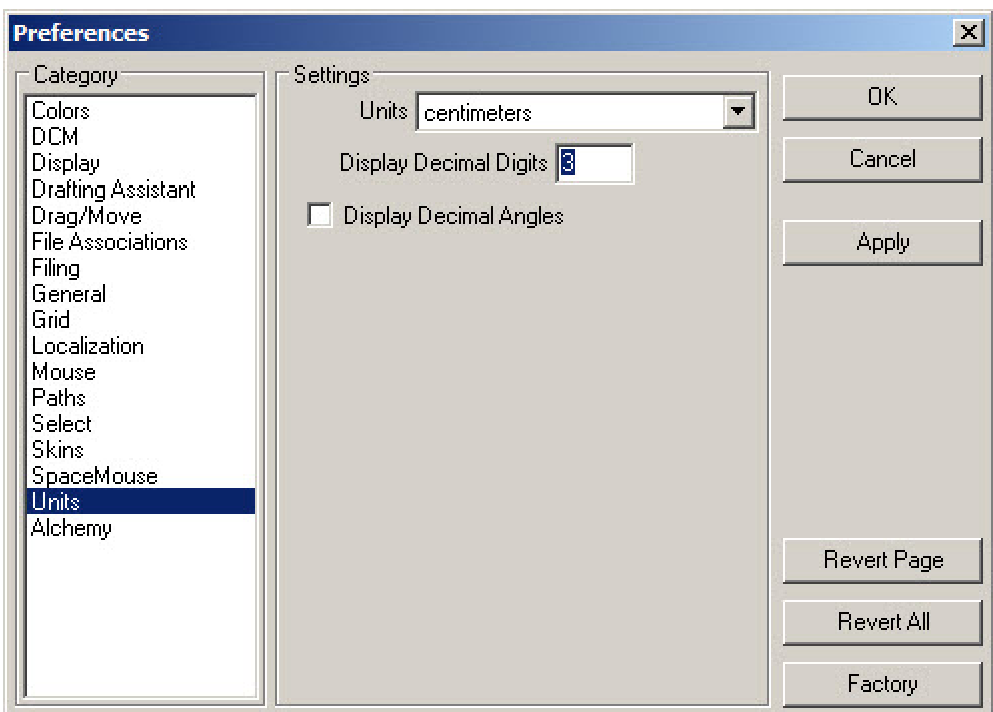
Push forward or pull backwards keeping the controller head parallel with the work surface.

Rotate

Twist clockwise and counterclockwise to rotate the model accordingly (Y axis).

Units

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



***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.*

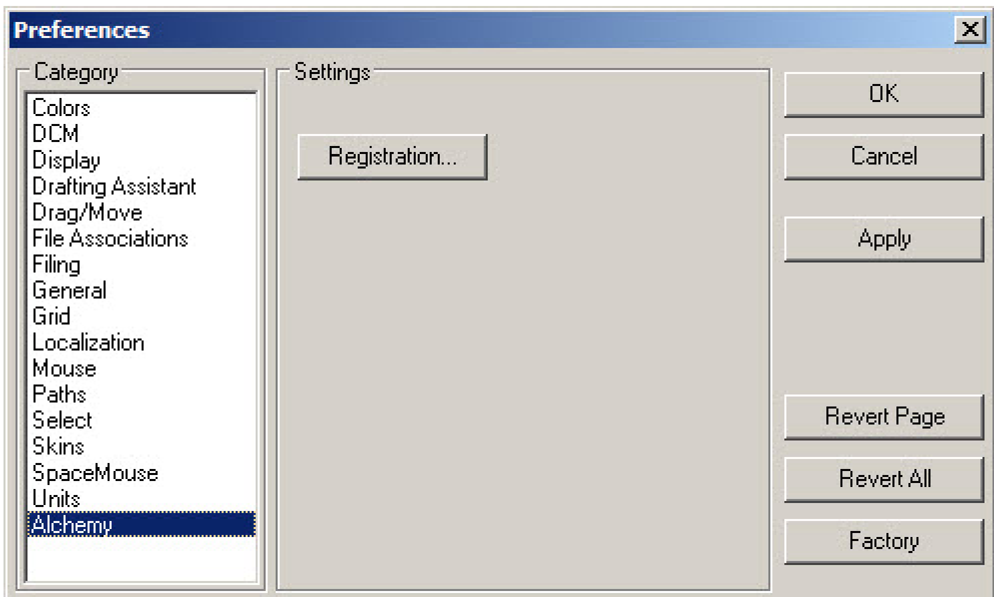
Units can be set to inches, feet, feet/inches, millimeters, centimeters and meters.

> ***Tech Note:*** *If feet/inches are chosen for units, it is fine to enter a combination of feet and units in the Status Line and Edit Objects dialog box. These values, however, will convert to decimals after they are applied to the geometry. For example: A value of 1"3" converts to 1.25 feet.*

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

Alchemy

This page of the Preferences dialogue box allows registering your Alchemy license.



Changing the Preference Settings with the Preferences Command

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

Changing the Preference Settings Manually

1. Quit the program.
2. Open the Preferences file, *prefs.ini,* in a text editor.
3. Make the desired change to the characteristics.
4. Save and close the file.

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

The preferences are set.

Tech Note: *The location of the prefs.ini file varies with the program, the version, the platform and the version of the operating system. It can be found under User, Workgroup, System (Mac only) and/or Program. For specific information on your version try the knowledgebase in the Support Center on the website at www.ashlar.com.*

Preferences and Object Creation

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

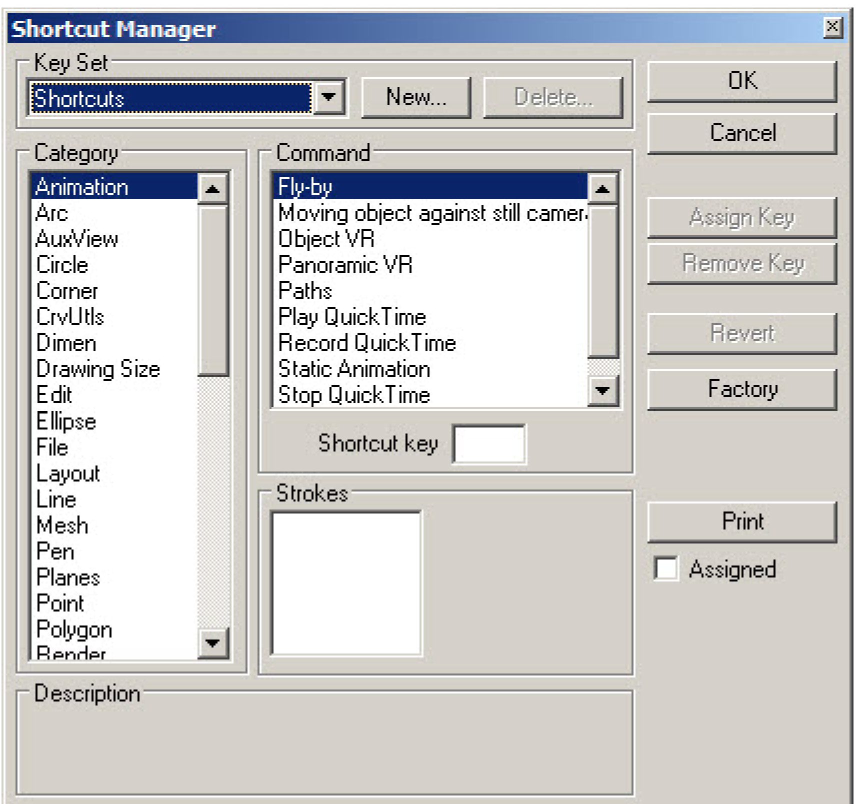
Pen and Dimension Preferences

When Cobalt, Xenon or Argon are launched, the default pen, text and dimension characteristics can be set. Without anything selected, specify the settings. It is not necessary 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.

Shortcuts

The *Shortcuts* command in the File menu creates keyboard combinations, providing alternative ways to invoke commands. It is possible to create shortcuts for activating tools and commands, setting the work plane, changing layers, switching between wireframe and render modes and more. If a shortcut to a tool is assigned, the short cut displays with the tool tip when the pointer is moved over the tool.

Shortcuts are organized by the *Shortcut Manager*.



The Shortcut Manager dialog box includes the following elements:

Category Contains all available operations in this 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. 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 Shortcut Manager include:

OK Saves the new Shortcuts and closes the Shortcut Manager.

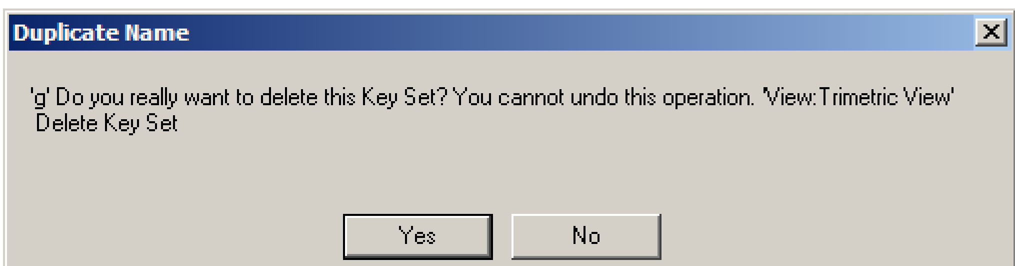
Cancel Closes the Shortcut Manager without saving any changes.

Print Prints a hard copy of the current shortcut key assignments. The factory default Shortcuts are included in Appendix C: ShortCuts.

Assigned If this box is checked, when printing the Shortcut list, only assigned keys are included.

Assign Key Assigns the Shortcut key combination for future use.

If the shortcut key is already assigned, a warning box displays asking to confirm its reassignment.



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

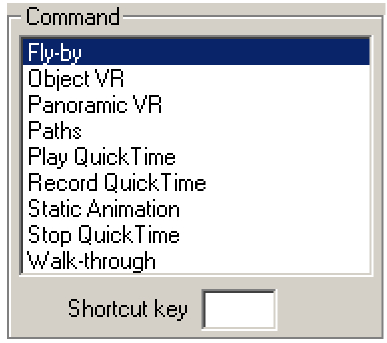
Remove Key Removes the key combination from the selected command.

Revert Undoes any changes made to any groups.

Factory Resets all groups to the factory settings.

Creating a New Shortcut

1. Choose ***File>Shortcuts***.
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. The key combination displays next to the selected command.
5. Continue entering all key combinations as desired.
6. Click OK to save the new shortcut keys and close the dialog box.



Appendix A: Mathematical Operators

The data fields in Cobalt, Xenon and Argon 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	are accepted in any of the following forms: 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
p	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

It is possible to use special characters and accents which are available with the operating system and region keyboard for the computer. Usually these characters are described in an appendix of the operating system's 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 necessary character.

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*

Macintosh:

To access international and mathematical symbols on the Mac, activate the Keyboard Viewer and Character Palette under the Apple System Preferences/International. A flag icon appears in the menu bar. Click it for a menu with options to show and hide these two tools.

> ***Tech Note:*** *Numeric values must be entered with an enhanced 101 keyboard using the separate number pad.*

In addition, the OPTION key can be used to generate international accents in the current font, including the 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)
±	*OPTION+SHIFT =*

Appendix C: ShortCuts

These are the short cut keys currently programmed in this Cobalt, Xenon and Argon. There are many short cuts which can be programed with specific keys. Choose ***File>Short Cuts*** to display the Short Cut Manager. See 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 the 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 the 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 to drag a selection fence to specify the zoom area.

Appendix D: Spline Text Files

It is possible to create text files for importing spline data into Cobalt, Xenon or Argon.

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 the 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

   Specifing decimal coordinates are allowed as well:

1.33	1.1	0
2.4	2.5	3.5678

   Press ENTER (Windows) or RETURN (Macintosh) after the last coordinate. If not, Cobalt, Xenon or Argon will not import the coordinates specified in the last line.
3. Save the file as Text only and import into Cobalt, Xenon or Argon using the Spline Import format to create the spline.

Appendix E: Photo-realism Fundamentals

Photo-realistic rendering borrows terminology and technology from many disciplines. From photography to painting, there is often more going on behind the scene than the viewer realizes. To make an interesting composition, Cobalt, Xenon and Argon provide an extensive library of materials, lights and decals for use in a scene, but making that scene come to life will require some fundamental techniques.

In this lesson are some general guidelines to help make your renderings look real. These include:

Creating the Environment

- Modeling Tips for More Realism
- Creating Floor Surfaces and Backdrops
- How to View the Scene
- Composition: Arranging Objects in the Scene
- Making Reflective Objects Look Real

Lighting the Environment

- The 3-point Light Setup
- Specific Lighting Situations
- Lighting Color
- Shadows

Adding Realistic Materials

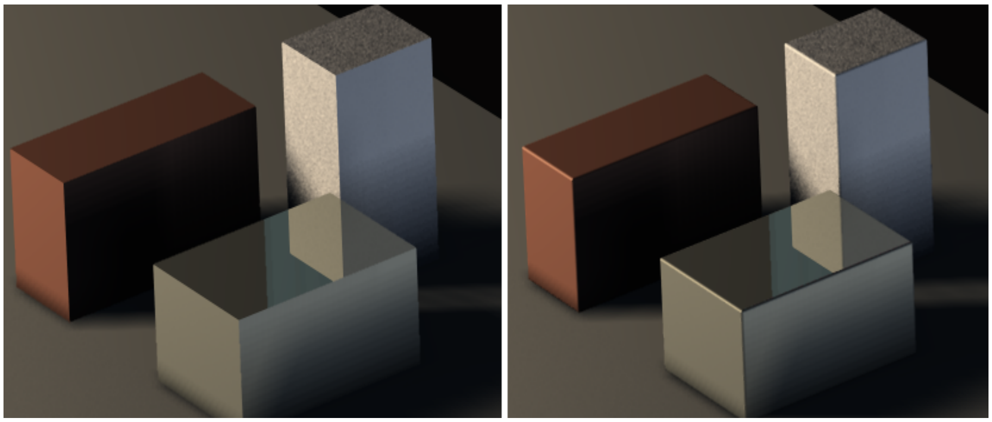
- Basics of Materials
- Types of Plastic
- How to Make Metal
- How to Make Clear Materials
- Image-based Textures
- Practice Makes Perfect

Creating the Environment

Modeling Tips for More Realism

Round off Edges

In the real world, there are almost no sharp edges. Putting tiny blends on all edges can add subtle realism to a rendering.



Set Object Resolution to Super Fine

Choose ***Edit>Change Resolution...*** and select Super Fine from the window, or right mouse click an object, choose ***Resolution>Super Fine*** from the flyout menu.

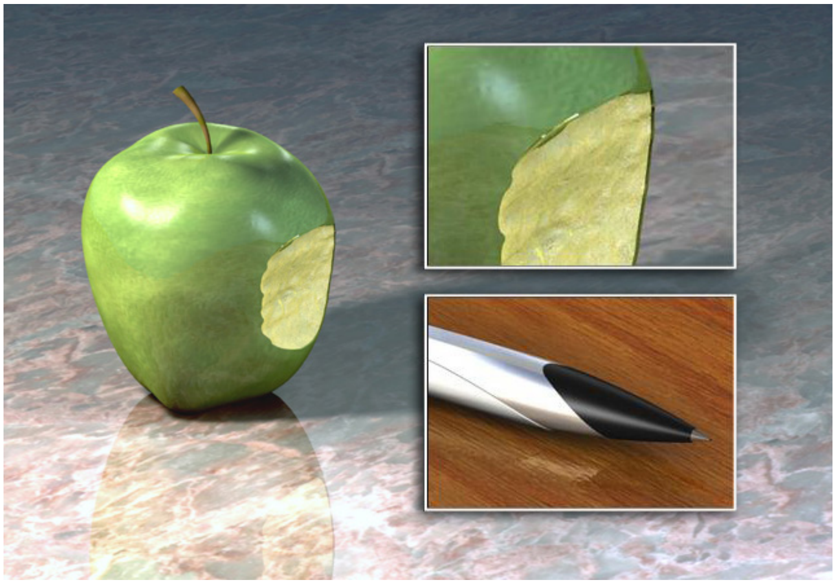
Use Surfaces for More Texturing Control

Sometimes different materials need to be applied to the same object. To obtain this do one of the following:

1. Use the **Deep Select** tool to select one face of a solid and apply necessary material to it.
2. Choose ***Edit>Change Object Type...*** and change the object to surfaces. This allows you to place different materials on each different surface.

Model as Much Detail as Possible

Nothing in the real world is as perfect as a computer can make it. It's the subtle imperfections in things that make us know they're real. Model the smallest details like the piece of apple skin hanging off the edge of the bite area or the ink in the tip of the pen below.



Creating Floor Surfaces and Backdrops

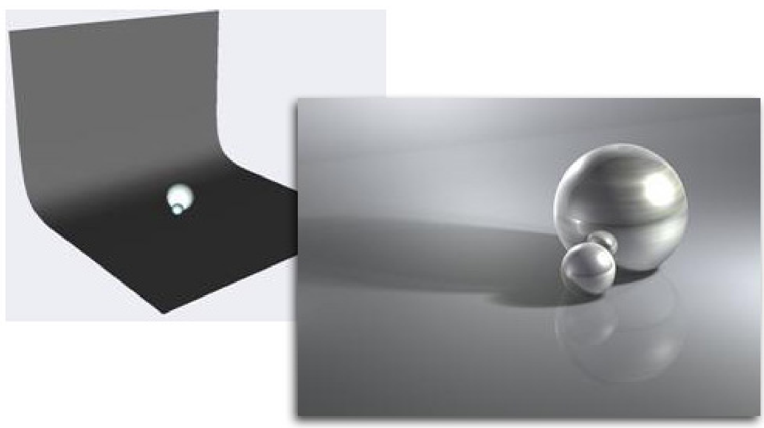
Placing objects in a scene or on a simple backdrop can add dimension and scale to an otherwise flat rendering. Whether it's a simple floor or a detailed room, objects will seem more realistic if they are placed in a suitable environment.

Simple Floor

A simple floor is a nice way to add a shadow to an object. The human eye is used to seeing shadows. Including them in a rendering adds realism.

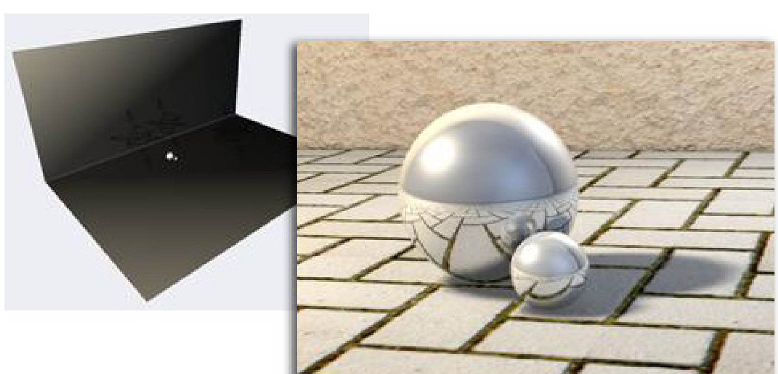
Curved Surface

A curved surface under objects more accurately simulates a photographic studio's technique of putting a large roll of paper under an object. This gives a smooth background with no seam between the floor and the rear wall.



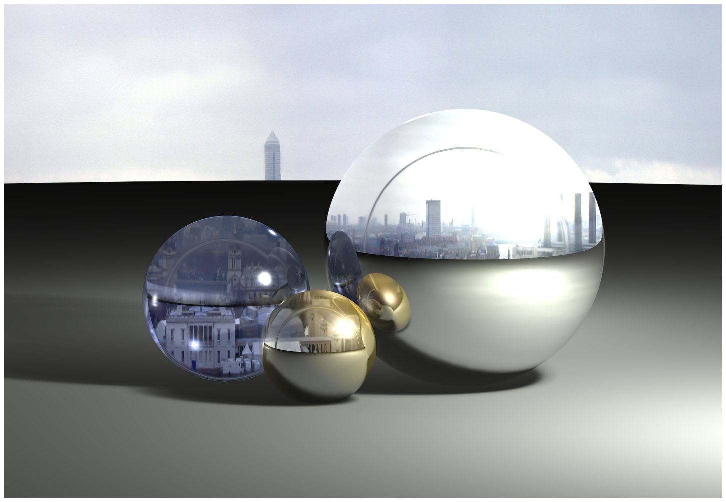
Floor and Wall

Sometimes a distinct separation between the floor and the wall is needed. Two or more surfaces can be used to simulate this.



Background Image vs. Sky Dome Sphere

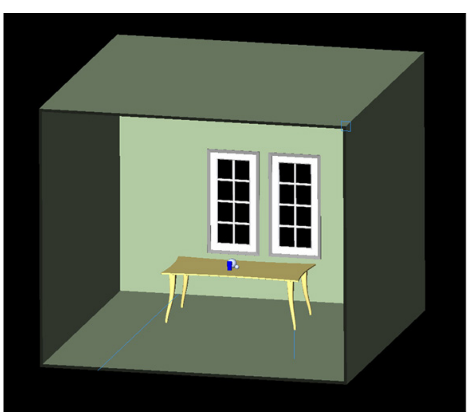
Cobalt, Xenon and Argon allow you to place an image in the ***Photo Render>Texture*** folder, then from the ***PhotoRender>Edit Background...*** place an image using the Image Shader Type. This will work for most situations, but the image is always placed to the back of the current view as if shown from a projector. Therefore, reflections in mirrored or glass objects will not reflect as they would in the real world.



To achieve a realistic reflection, the image needs to wrap around the scene. This Sky Dome is created by placing a solid primitive sphere around the scene. Highlight the sphere, go to ***Edit>Change Object Type...*** and change the object to surfaces. Then place the backdrop as a decal or a wrapped image onto the sphere surface.


Detailed Room

Build a detailed room around the objects to make them appear as they would in the real world. Notice the reflections show all of the walls in the room.




How to View the Scene

View Angles - Wide Angle vs. Close-up

Choosing a wide angle will allow the viewer to get a feel for the environment in which the object resides. If you don't have a lot of detail in the wide angle view, this can detract from the object you are showing. A close-up view will show more detail in the objects.

View Angles - High Angle vs. Low Angle

Viewing the objects from a high angle will give a viewer the feeling of looking down into objects. This will also give the perception that the objects may be small. Viewing down low, however, will give the feeling that the objects are large or it can also give a grand feeling to a scene.

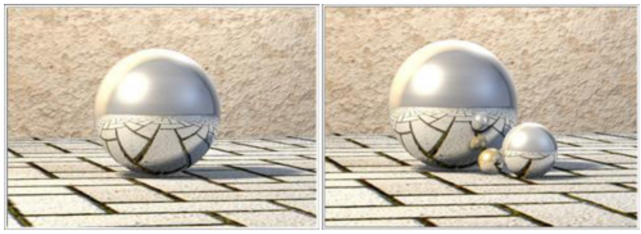
Which Side of the Object to Light

Which side of your object do you want to be lit, or conversely, which side of the object do you want the shadow to be on?

Placing the key light toward the wrong face of an object can hide the most important features. Make sure you place your objects and lighting so the most important feature is prevalent in your scene and that the shadow falls in a manner so as to give the object depth and realism.

What's the Focal Point?

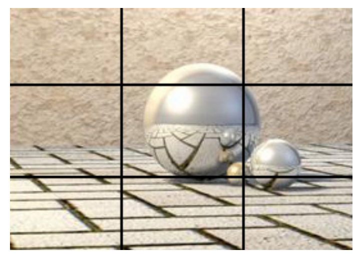
Adding secondary objects to the scene can help to enhance the realism of the main object. It can be smaller objects that make the main object look bigger, or a floor with tiles to give a sense of scale. Make sure that the main object is what a viewer concentrates on when looking at your rendering. Too many additional objects can become distracting and lure the focus away from the subject.



Composition: Arranging Objects in the Scene

The Rule of Thirds

A general guideline when setting up a scene is to imagine the frame being divided up into thirds. Placing the subject along one of the vertical lines will give your scene a sense of motion. It will make the image a little more interesting to the viewer.



Compositional Balance

While the Rule of Thirds is often very helpful in making a scene interesting, it can often make a scene feel unbalanced. Adding an object to a scene that is subdued in color and does not draw your attention away from the subject can be just what's needed to bring a scene together.

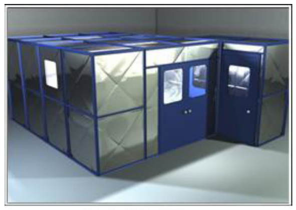
Perspective: High-Angle vs. Low-Angle

Perspective angle combined with the view-angle is what gives a feeling of scale. A high-angle perspective with a narrow field of view will show objects close to their natural look. A low-angle view with a wide field of view will exaggerate the size of a model. It also help so move in real close.

Perspective: Inside vs. Outside

The field of view can be changed to simulate whether you are inside or outside of a room. The field of view is how wide the viewing angle is.

Change the perspective focal length by using ***View>View Properites*** or clicking the Perspective icon in the View palette..



Outside View:
Focal Length = 400

Inside View:
Focal Length = 120

Making Reflective Objects Look Real

Floor and Additional Objects

Metal or glass objects are reflective by nature. In order to look realistic they require something to reflect. If you have a simple object and apply a chrome material to it, then render the scene, it won't look like chrome at all.

In the following images we see the original object with a chrome material applied. When rendered, it only reflects the black background, therefore it doesn't look like chrome. In the second image, a white floor has been added. The chrome looks a little better, but you still don't get the feeling that it's chrome. Finally, some additional objects have been added to the scene. It becomes obvious now that the object is a chrome part.

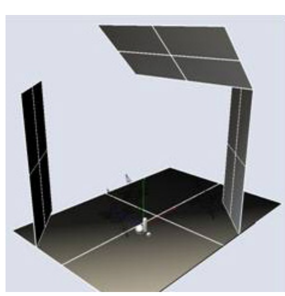
Chrome Material Applied

Floor Added

Additional Objects Added

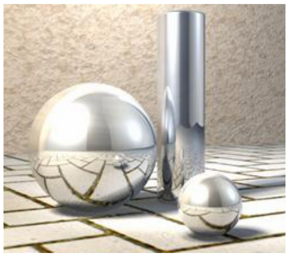
Off-screen Objects

Adding objects to the scene that will only be seen in the reflections can further enhance the realism. Shown below is a scene where some additional surfaced have been added outside the view. They have been given a white material so they will show in the reflective surfaces of the subject pieces.



Texturing

Adding a textured surface to the floor and walls can also enhance the look of reflective objects.



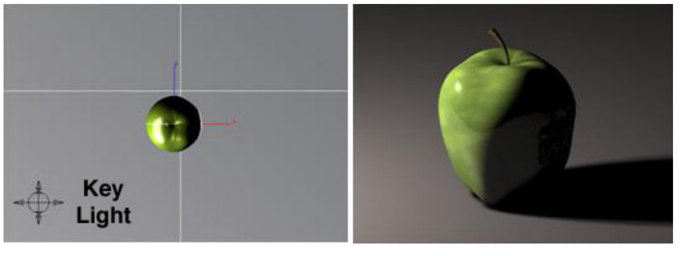
Lighting the Environment

The 3-point Light Setup

One of the most popular lighting setups in both photography and 3D rendering is the 3-point lighting setup. Three-point lighting is favorable in most scenes because it adds depth to the subject matter and offers more control with less lights in the scene.

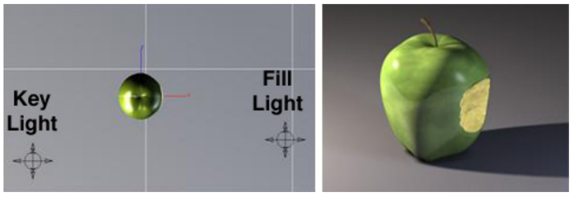
The Key Light

This light is the main light in a scene. It is responsible for the majority of the light on the subject and for defining the main shadow.



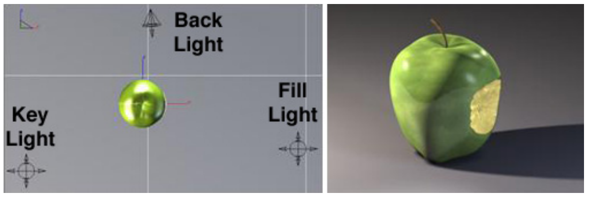
The Fill Light

The Fill light, as it's name implies, serves to fill in the darker area of the object not lighted by the Key light. This light helps to soften the bold shadow created by the Key light.



The Back Light or Rim Light

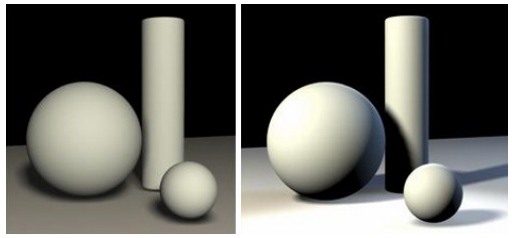
This light is used to help separate the object from the background. It will add a rim of light to the edge of the object. It is most often placed behind the object.



Specific Lighting Situations

Lighting Creates 3D Form and Depth

Lighting a scene properly can add depth indicating to the viewer the true shape of the objects. Proper light placement will highlight edges and cast shadows to show the true form in the scene.

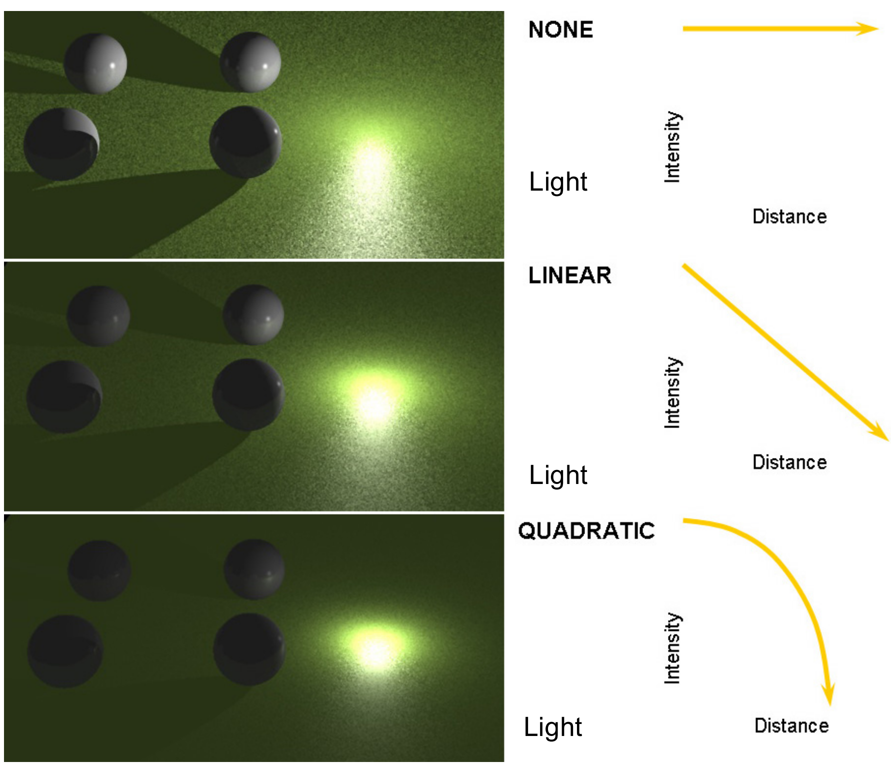


Lighting can Draw Attention and Create Interest

Making the background darker will draw attention to the objects in the foreground. Again, proper light placement and light intensity is the key to drawing the viewers interest where you want it.

Real Light Decreases Intensity with Distance

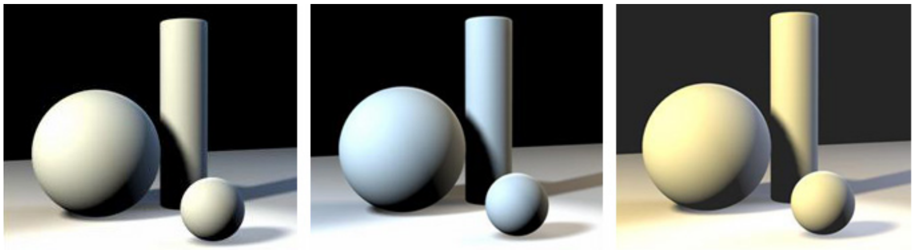
Computer generated lights are often set so the intensity does not diminish based on the light's distance from the subject. It is possible to adjust this setting in Cobalt, Xenon or Argon to be more realistic if necessary. When placing a light, with it still selected, use the Attenuation pulldown in the Edit Objects dialog box. There will be five options: None, Clamped Linear, Clamped Quadratic, Unclamped Linear and Unclamped Quadratic.



Lighting Color

Lighting has Color

Create a mood with the color of the lights. You can make a scene appear as though it's shot in the warm summer sun or in the midst of a cold winter day. You can also simulate different light types with color. For instance, use a light that produces the color of a candle flame, an incandescent light or a fluorescent light.



Color Temperature

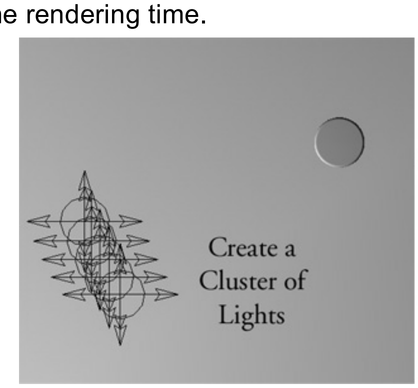
Light color is often described as its color temperature, which is measured in degrees Kelvin. The following is a general guideline for common light sources and their respective RGB values for use in Cobalt, Xenon and Argon.

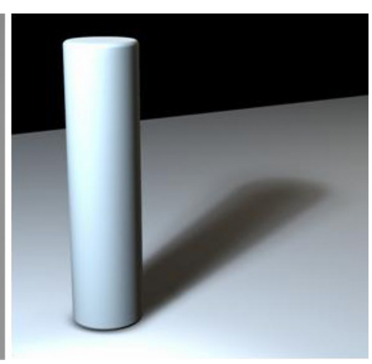
Type of Light	Degrees Kelvin	RGB Equivalent
Match Flame	1700K	R 255, G 95, B 7
Candle Flame	1850K	R 255, G 105, B 11
Sun: At Sunrise or Sunset	2000K	R 255, G 116, B 16
Household Bulb	2500K	R 255, G 146, B 41
Fluorescent Light	4500K	R 255, G 229, B 183
Sun: At Noon	5000K	R 255, G 243, B 219
Daylight (Sun + Sky)	5500K	R 255, G 255, B 255
Sun: clouds/haze	6000K	R 225, G 235, B 255
Sky: Overcast	6500K	R 203, G 219, B 255
Outdoor Shade Area	7500K	R 172, G 196, B 255
Sky: Partly Cloudy	10000K	R 136, G 165, B 255

Shadows

Shadows - Hard vs. Soft

When a light is selected in a scene, its shadow type can be changed in the Edit Objects Dialog box. A hard shadow will make the scene look like a bright, sunny day. A blurry shadow may simulate an overcast day. Bear in mind that this technique will also increase the rendering time.



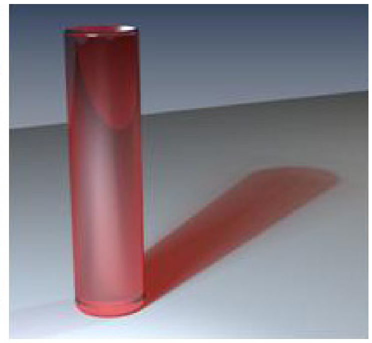


Transparent Shadows

If you have a transparent object in the scene, have the software render a transparent shadow. To do this, open ***PhotoRender>Advanced Settings...*** and click the Use Transparent Shadows check box. When rendering, the shadows will appear as if the light was traveling through the object. If the object is colored, the shadow will retain the color.

Glass Object
Transparent
Shadows Off

Glass Object
Transparent
Shadows On

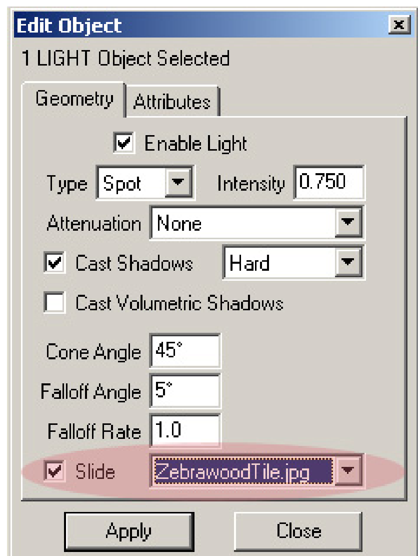


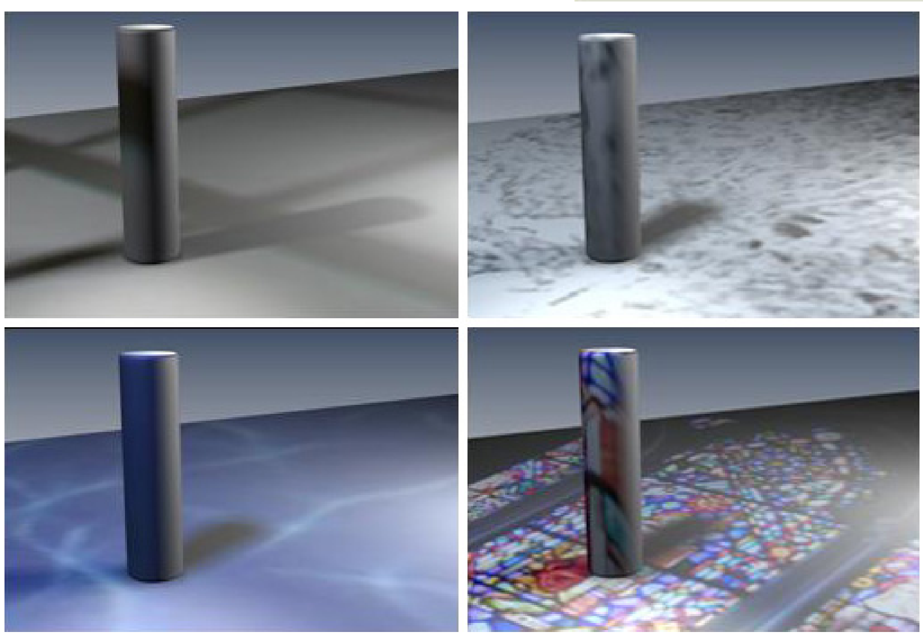
Colored Glass Object
Transparent Shadows On

Shadows with Gel Images

It is possible to have the light cast an image to simulate light shining through a tree, bouncing off water or even through a stained glass window. This only works on a Spot light, not the Point or Distant light.

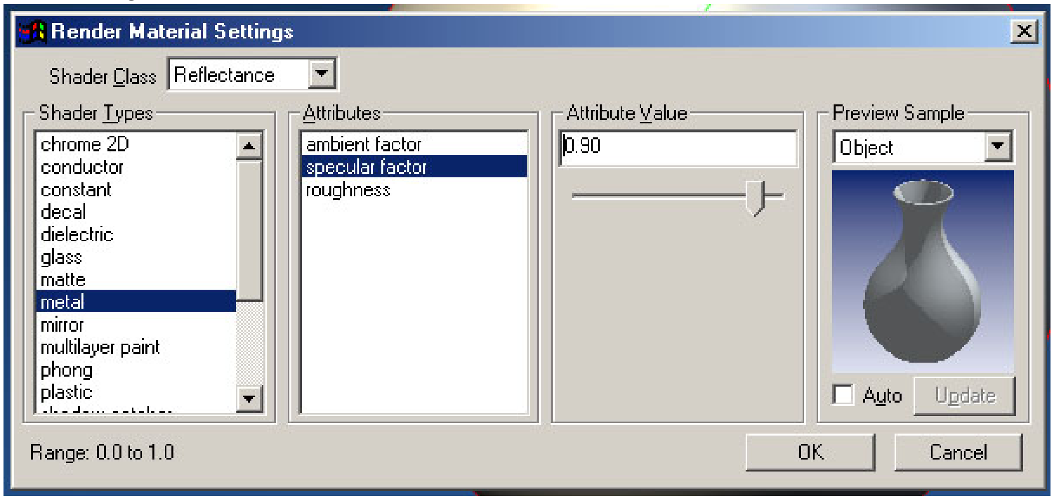
Place a spot light, then open the Edit Objects Dialog box. Enable the Slide checkbox then choose an image. The image will be projected just like a slide in a slide projector.





Adding Realistic Materials

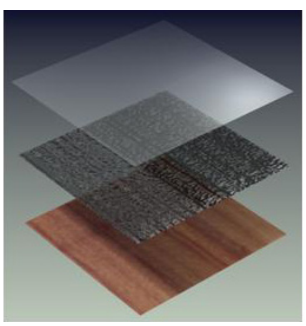
Cobalt, Xenon and Argon provide a wealth of pre-defined render materials. Sometimes though, you'll need to edit these materials or perhaps start from scratch to create your own. To edit a material, use the Render Material Settings dialogue box. Bring up this box by right mouse clicking an object that has a material applied and choosing Edit Material or by selecting the object, opening the Edit Object Dialog box, choosing the Material tab and cicking the Advanced button.

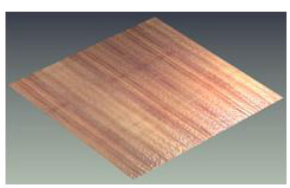


Basics of Materials

Material Categories Like Additive Layers

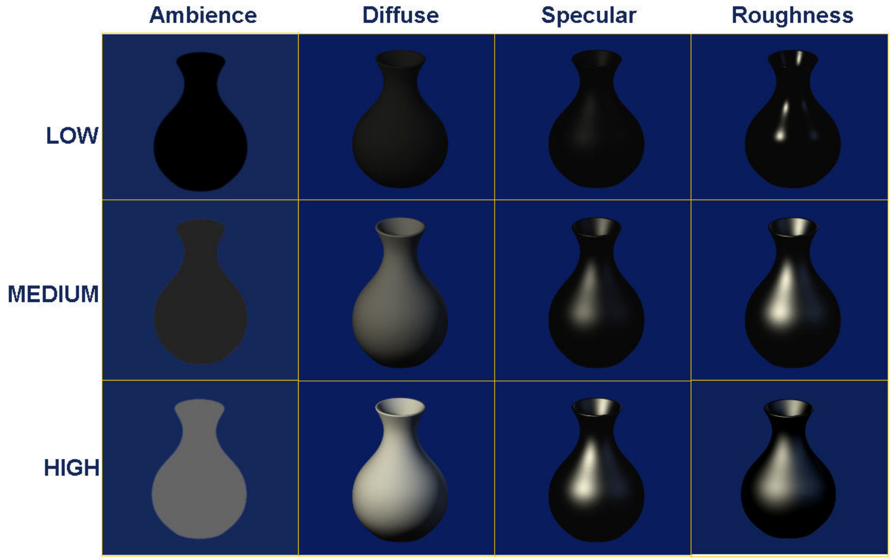
Each material may have several different layers that make it look real. A shiny wooden floor will consist of a layer that contains the image of the wood, a layer that makes the wood look rough and a layer that makes it shiny. Each of these layers, or shader classes, has it's own area in the Render Material Settings Shader Class pull-down box.



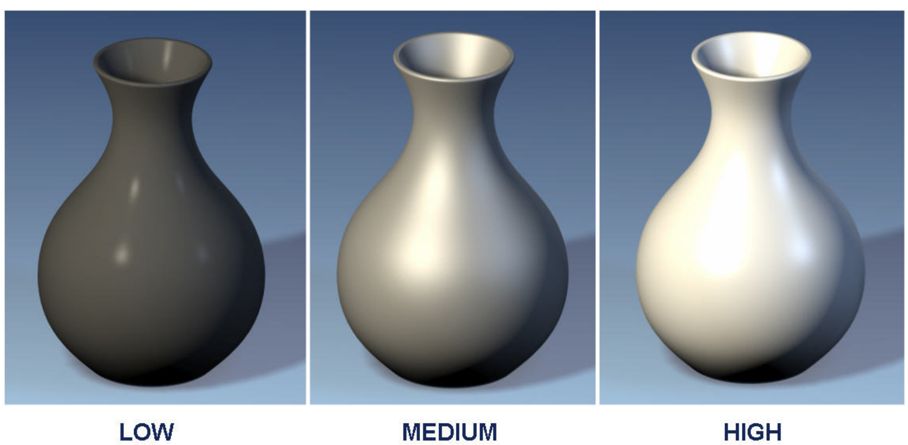


The Reflectance Category and What It Does

The Reflectance category designates a material as matte or shiny, reflective or dull. Here are some images that show the different categories.



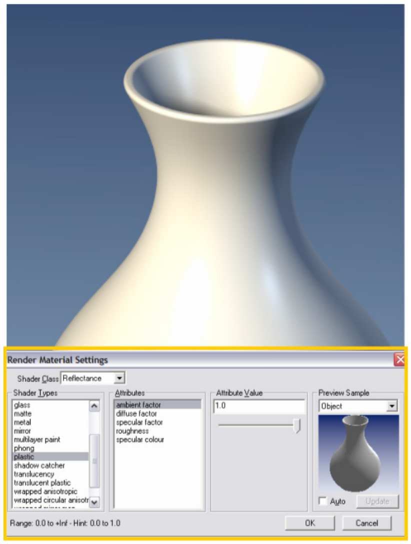
Combined Reflectance Attributes

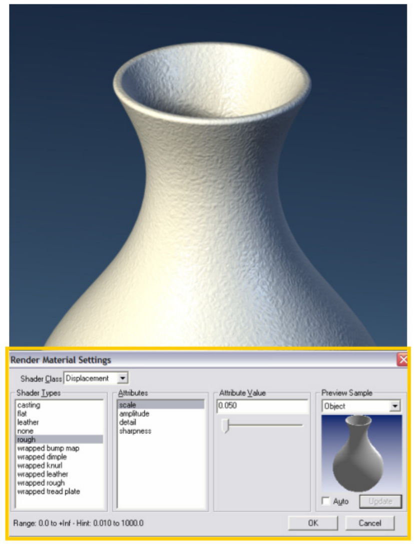


Types of Plastic

Smooth Plastic and Rough Plastic

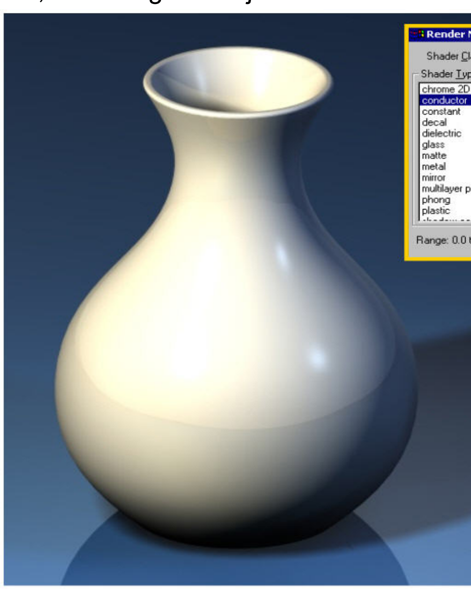
Note the Reflectance and Displacement Shader Class settings to make plastic smooth or rough.

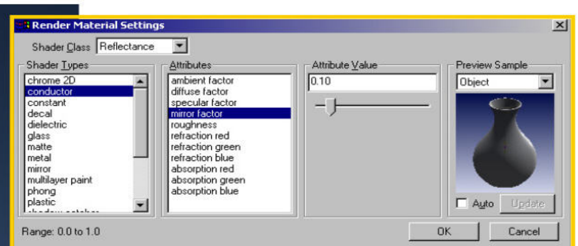




Reflective Plastic

In the Reflectance Shader Class, switch the Shader Type to conductor and experiment with the mirror factor Attribute setting. Note an extremely high value will become mirror-like, reflecting the objects environment.





Mirror Factor

How to Make Metal

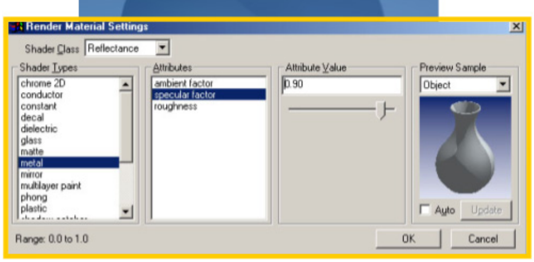
Basic Metal

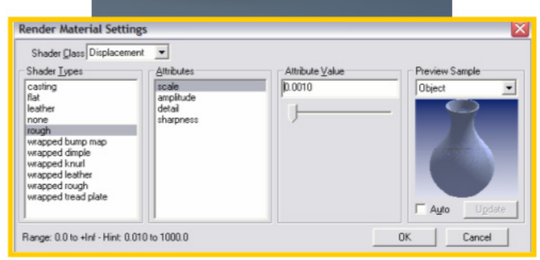
Some basic settings for making Aluminum, Gold or Brass appear in the *Reflectance* Shader Class *Conductor* Shader Type.

Basic Aluminum
Ambience (.20)
Diffuse (1.0)
Specular (.60)
Mirror (.70)
Roughness (.02)

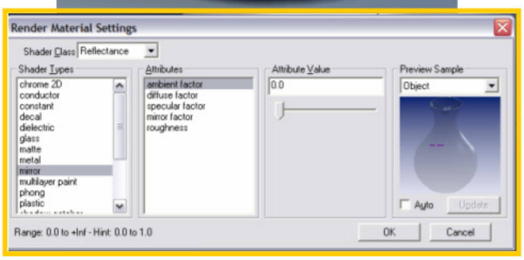
Gold or Brass
Ambience (.20)
Diffuse (1.0)
Specular (.60)
Mirror (.70)
Roughness (.02)

Raw or Anodized





Chrome



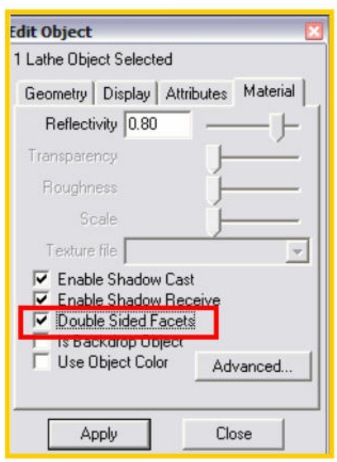
How to Make Clear Materials

Glass

Index Of Refraction For Common Materials

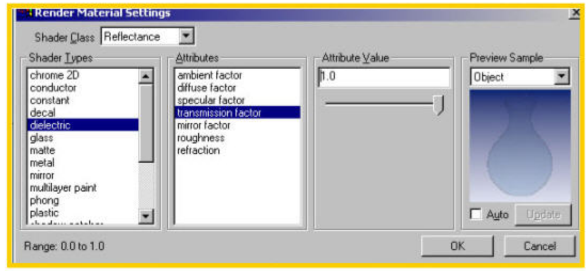
Vacuum - 1.0	**Ice - 1.309**
Air - 1.0003	**Emerald - 1.576**
Water - 1.33	**Ruby - 1.76**
Glass - 1.5174	**Sapphire - 1.76**
Crystal Glass - 2.0	**Opal - 1.45**
Acrylic - 1.50	**Iodine Crystal - 3.34**
Clear Plastic - 1.46	**Cubic Zirconia - 2.17**
Quartz - 1.544	**Diamond - 2.417**

<u>Basic Glass</u>
Transmission (.93)
Refraction (1.5174)
Mirror (.15)
Specular (1.0)
Roughness (.003)



<u>Crystal Glass</u>
Transmission (1.0)
Refraction (2.0)
Mirror (.4)
Specular (2.5)
Roughness (.002)

Clear Plastic



Acrylic
Transmission (1.0)
Refraction (1.4)
Mirror (.8)
Specular (1.0)
Roughness (.005)
Diffuse (1.0)
Ambience (1.0)

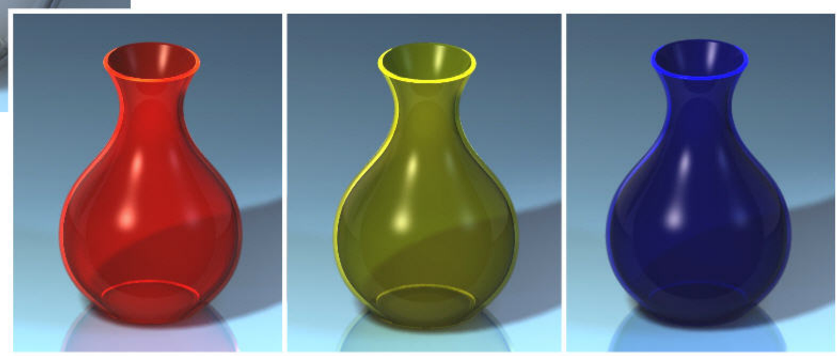


Image-based Textures

Brushed Metal

Wood.

Marble and Stone

Practice Makes Perfect

Since computer-based rendering is based in so many disciplines, including photography, cinematography, illustration and painting, it's not something that's easy to master. There are many books on these subjects that will apply to your rendering efforts. Remember to experiment with settings and note the changes. Practice with little test scenes. If you see something interesting in your environment, or a picture that you are fond of, try to simulate it in Cobalt, Xenon and Argon. The tools are there, now it's up to you to master the craft.

Appendix F: 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. There are 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 the space can be set.
decal transparency	Sets the transparency from clear to opaque.

amplitude	Sets the magnitude of an attribute relative to another.
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 being evaluated. 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.

amplitude	Sets the magnitude of an attribute relative to another.
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 the space can be set.
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 the shader can be set.
shader	Sets the shader.

absorption	Sets the amount of light absorbed.
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.

coverage	Sets the degree that an attribute covers the object with the associated shader.
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. There are 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.

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.
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.

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.
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.

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.
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 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 attempting to select one object among objects so the desired object can be chosen.
Anchor	This point defines the direction when placing a distant light source in the 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 changes the display of overlapping objects in the 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.

Accelerators	Keyboard Equivalents that invoke commands rather than using the mouse to choose from menus.
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 the 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 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 includes 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.

Accelerators Keyboard Equivalents that invoke commands rather than using the mouse to choose from menus.

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. 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 are used for exact alignment. The Drafting Assistant creates dynamic, temporary construction lines. It is also possible to 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.

Accelerators	Keyboard Equivalents that invoke commands rather than using the mouse to choose from menus.
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 to chose a color from the crayon box.
Custom Colors	(Windows only) This button in the color display defines 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 othere value or choice is not specified.
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 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.

Accelerators	Keyboard Equivalents that invoke commands rather than using the mouse to choose from menus.
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 provides 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 to be exact. When the pointer is close enough to display feedback, the Drafting Assistant locks onto the exact location.
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 the geometry in the selected render mode as it is rotated.
DWG	AutoCAD's native file format.

Accelerators Keyboard Equivalents that invoke commands rather than using the mouse to choose from menus.

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 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 the 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 the object with a painter's algorithm using constant shading techniques.

Flavor The types of IGES files this Designer Elements program can import and export.

Accelerators	Keyboard Equivalents that invoke commands rather than using the mouse to choose from menus.
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 the 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 the 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 the geometry such that only visible edges are displayed.
Hidden w/Dimmed	Renders the 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.

Accelerators	Keyboard Equivalents that invoke commands rather than using the mouse to choose from menus.
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. Creating 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 a master symbol is created, an instance occurs when the symbol is placed in the drawing. An instance is associative to the original geometry. Any change made to the original is reflected in the instance. If the object is copied and pasted, 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.

Accelerators Keyboard Equivalents that invoke commands rather than using the mouse to choose from menus.

Layer Analogous to transparent media used in conventional manual drafting. Parts can be constructed on several layers which can be made visible or invisible.

Line Pattern Manager Through this dialog box, 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 the geometry.

Accelerators	Keyboard Equivalents that invoke commands rather than using the mouse to choose from menus.
Model Point	The point that can be specified 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. The origin can be changed 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.

Accelerators	Keyboard Equivalents that invoke commands rather than using the mouse to choose from menus.
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 the geometry based on calculated light intensities at each pixel location.
Phong w/Edges Shading	Renders the 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 pixel area defined in Preferences that selects 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.

Accelerators	Keyboard Equivalents that invoke commands rather than using the mouse to choose from menus.
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 reverses the action of the **Undo** command.
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 the geometry in a specified mode.
Render Now	The render option activated when the *Render* command is chosen. Set the mode in the Render Options dialog box.
Render Options	The dialog box that sets the shading mode for the geometry.
Resolution	The appearance of surfaces and solids when rendered. There are 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	1. 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. 2. A memory aid in which the thumb of the right hand points in the direction of the positive axis of rotation. The fingers curve in the direction of rotation.
SAT	A file format for ACIS based programs.

Accelerators	Keyboard Equivalents that invoke commands rather than using the mouse to choose from menus.
Section View	A view created by making a section cut through the geometry displayed in a drawing view.
Selection Fence	A bounding box that temporarily appears as the **Selection** tool cursor is dragged around an area in the drawing.
Setback	The distance that a blend extends from three or more intersecting edges.
Short Cuts	Key combinations for performing Designer Elements program operations to customize the 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 sets the specifications for the Drafting Assistant.
Sphere Trackball	The standard trackball that rotates 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 the geometry will display when the view is stationary.
Status Line	The line at the bottom of the drawing area in which the specifications for the geometry being created can be entered.

Accelerators	Keyboard Equivalents that invoke commands rather than using the mouse to choose from menus.
Stencil	An image that defines the regions through which artwork projects onto an object.
Step Trackball	The trackball that rotates 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 of twisting a solid when sweeping a profile to a solid.

Accelerators	Keyboard Equivalents that invoke commands rather than using the mouse to choose from menus.
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. An infinite number of actions can be undone 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 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 magnifies or reduces an image.

Index

Symbols

% point 3-6, 49-7

Numerics

1 point
    block primitive 23-4
    cone primitive 23-8
    cylinder primitive 23-6
    prism primitive 23-12
    pyramid primitive 23-14
    rotate 8-3
    sphere primitive 23-2
    symbol 34-4
    torus primitive 23-10
2 points
    block primitive 23-5
    center ellipse 12-2
    cone primitive 23-9
    conic 12-5
    cylinder primitive 23-7
    mirror 8-8
    prism primitive 23-13
    pyramid primitive 23-15
    section view 44-16
    sphere primitive 23-2
    symbol 34-4
    torus primitive 23-11
2-entity
    chamfer 15-3
    fillet 15-2
3 points
    arc 11-2
    center ellipse 12-3
    circle 11-5
    conic 12-6
    mirror 8-9
    rotate 8-4
    work plane command 37-6
3-Corner Ellipse tool 12-4
3-Entity Fillet tool 15-2
3-point light setup E-9
4-Point Conic tool 12-7

A

ACIS SAT
    export 43-14
    import 43-6
add
    boolean surface 20-9
    union solid 26-2
Add Spline Control Point tool 14-7
Adobe Illustrator
    export 43-18
    import 43-7
align 5-9
    assembly 28-3
    drawing view 44-10
Align Objects tool 8-21
alignment angles 3-6, 49-7
along curve
    Bend solid 25-33
ambient lighting 38-14
ambiguity popup 4-6, 49-16
analysis
    surface 29-9
analysis (surface) 29-10, 29-22
Angle 3 Pts command 29-1
angles (construction line) 3-9
angular
    chamfer 15-3
    dimension 32-11
Angular Dimension tool 32-16
animation 41-1
    Fly By 41-5
    Paths 41-5
    VR Object 41-7
    VR Panoramic 41-9
    Walk Through 41-4
ANSI dimension standard 32-23, 32-24
anti-alias 40-8
Apply button 2-19
Arbitrary Polygon tool 13-7
    show frame 13-7
Arc tools 11-1
    3-Point 11-2
    Center-Point 11-1
    Tangent-Point 11-3
Architecture dimension standard 32-23, 32-24
Area command 29-2
area render 40-10
arrange
    command 5-8
    icons 35-18
arrays. See duplicate
arrow key (nudge distance) 49-12
arrowheads
    dimension 32-21
    pen 10-8
aspect ratio (mesh) 5-16, 5-18
Assembly Modeling tools 28-1
    align 28-3
    Connect 28-1
    Insert 28-4
    Mate 28-2
    palette 28-1
associativity
    dimension 32-1
    features 25-1
    parent/child 1-3
    solid 22-5
    surfaces 16-4
asterisks 2-19
Attenuation E-10
attribute (shader) 39-19

Attributes and Bill of Materials 45-1
- Apply To Selected 45-2
- Area 45-1
- Area 2D 45-1
- Create Balloons 45-2
- Create BOM 45-2
- Export BOM 45-2
- Material 45-1
- Perimeter 45-1
- Price 45-1
- Standard 45-1
- Stock Size 45-1
- Update BOM 45-2
- User Defined Templates 45-3
- Vendor No. 45-1
- Volume 45-1
- Weight 45-1

auto
- decal wrap 39-10
- hatch 44-10
- heal bodies 43-2
- layers 20-2
- save 49-10

Auxiliary View tool 44-13

Axis
- Bend solid 25-31

axis 2-2, 2-14
- show/hide 6-10

azimuth (view) 35-13

B

backdrop E-4

backgrounds
- applying 39-13
- drawing window color 49-3
- editing 39-24
- library 39-2, 39-12
- settings 39-25
- types 39-25

backup files 42-5

Balloon Dimension tools 32-12

base line dimension
- horizontal 32-6
- vertical 32-7

bend (deform face) 27-15

Bend Solid Tool 25-31

Bend Solid tool
- Along Axis 25-31
- Along Curve 25-33
- Center 25-33

Bezier Spline tool 14-4

blend (constant)
- disjoint 25-9
- edge to face roll 25-11
- elliptical 25-6
- mitered 25-8
- multi-radius 25-8
- radial 25-5
- vertex 25-7, 25-9

blend (variable)
- by position 25-12
- fixed width 25-14
- hold line 25-15
- law curve 25-13
- linear 25-12

blend options (advanced)
- chain select edges 25-3
- cross section 25-4
- feature interaction 25-3

Blend tools
- Constant 25-5, 25-16
- param 25-13, 25-24
- Variable 25-12

Block Primitive tools 23-4
- 1 Point 23-4
- 2 Point 23-5
- Diagonals 23-5

BOM User Defined Templates 45-3
- Area 45-3
- CG-X 45-3
- CG-Y 45-3
- CG-Z 45-3
- Float 45-3
- Index 45-3
- Integer 45-3
- Material 45-3
- Name 45-3
- Perimeter 45-3
- Qty 45-3
- String 45-3
- Sum 45-3
- Volume 45-3
- Weight 45-3

Boolean Solid tools 26-2
- Intersect 26-3
- Subtract 26-2
- union 26-2

Boolean Surface tools
- add 20-9
- Intersect 20-9
- Subtract 20-9

Boss Feature tool 25-29
- direction 25-29

branch (Design Explorer)
- collapse 5-34
- expand 5-34

bulge factor (lofted solid) 26-8

by position blend 25-12

by position variable chamfer 25-24

C

C2 continuous splines 14-4

camera (movie) 41-2, 41-8, 41-9

case (text) 31-5

Catia v4
- export 43-18

Center
- Bend solid 25-33

center
  center mark dimension 32-11
  Drafting Assistant snap 3-2
  Drawing View command 44-11
center of gravity 29-4
Center/Axis Rectangle tool 13-3
Center/Corner Rectangle tool 13-2
center-point
  arc 11-1
  circle 11-4
CGM 43-14
chain (selection) 4-11
chain dimension
  horizontal 32-6
  vertical 32-7
chain select edges blend
  option 25-3
chain select edges chamfer 25-17
chamfer (constant)
  length 25-18
  length angle 25-19
  mitered 25-21
  two lengths 25-19
  vertex 25-20
chamfer (variable)
  by position 25-24
  fixed width 25-24
  four lengths 25-22
  lengths 25-22
  lengths angles 25-23
Chamfer Edge tools
  Variable 25-21
chamfer options (advanced)
  chain select edges 25-17
  feature interaction 25-17
Chamfer tools 15-1
  2-Entity 15-3
  Angular 15-3
change
  curves to lines 5-13
  curves to splines 5-13, 5-14
  dimension to lines 5-19
  direction 5-11
  drawing view 44-11
  layer 5-6
  object type 5-12
  preferences 49-20
  resolution 5-4
  solid to curves 5-17
  solid to mesh 5-17
  solid to surfaces 5-17
  solids to solids 5-19
  surface to curves 5-14
  surface to mesh 5-15
  surface to surfaces 5-15
  text to curves 5-14
Change Object Type E-3
check box 2-18
Check Object command 29-7
Circle tools 11-3
  3-Point 11-5
  Center-Point 11-4
  Opposite-Point 11-5
  Tangent-Point 11-6
Circumscribed Polygon tool 13-5
  scribe type 13-6
  show frame 13-6
Clear Materials E-17
clipboard 5-3
Close 24-17
close file 42-5
collapse (Design Explorer)
  branch 5-34
  item 5-34
color
  ambient light 38-14
  background 49-3
  current pen 10-3
  default dimension 49-21
  default pen 49-21
  define 10-3
  dimension 32-19
  fill 25-1, 25-4
  foreground 49-3
  grid 49-13
  layer 36-3, 36-8
  lighting 38-15
  name 10-4
  palette 10-3
  pen 10-2
  preferences 49-3
  selection 4-3
  shader class 39-21
  texture shader 39-21
Color Temperature E-11
compact files 49-10
Compositional Balance E-7
compressor (movie) 41-3
Cone Primitive tools 23-8
  1 Point 23-8
  2 Point 23-9
  Diagonal 23-9
Conic tools 12-5
  2-Point 12-5
  3-Point 12-6
  4-Point 12-7
  rho 12-5
  shoulder point 12-5
  slope control point 12-5
Connect Assembly tool 28-1
Connect Curve tool
  Join 15-8
  move curve point 15-8
Connect Surface tool 20-10
Connected Lines tool 11-3
constant blend

disjoint 25-9
edge to face roll 25-11
elliptical 25-6
extrapolation 25-11
mitered 25-8
multi-radius 25-8
one edge vertex 25-9
radial 25-5
two edge vertex 25-9
vertex 25-7
with cutouts 25-11
Constant Blend tool 25-5, 25-16
constant chamfer
length 25-18
length angle 25-19
mitered 25-21
two lengths 25-19
vertex 25-20
construction geometry 3-10
construction line
angles 3-8, 3-9
construction command 3-9
delete 3-11
Drafting Assistant 3-6
offset 3-9
parallel 3-10
pattern 10-8
permanent 3-8
control points
adding to spline 14-7
remove from spline 14-7
selecting 4-8
show 4-7
slope 12-5
spline 14-2, 14-3
control vertex modification 16-5
insert knot 21-6
convert
files (see export)
objects (see change object type)
text to curves 5-14
coordinate system 2-2
choosing 6-6
global 6-5
user-defined 6-5
Coordinate System Axis 2-14
copy
along path 8-9
bitmap 5-2
command 5-2
object characteristics 4-14
objects 8-23
objects/layer 36-9
pict 5-2
Status Line entries 2-13, 6-3
Corner Trim tool 15-7
count (layer objects) 36-3
Counter Bore tool 25-28
Counter Sink tool 25-28
Cover Surface tool 18-9
n-sided 18-9
cross section (blend) 25-4
crosshatching 25-4
auto hatch 44-10
command 25-6
DIN patterns 25-6
edit 25-7
ISO patterns 25-5
patterns 25-4
section cuts 25-7
section views 44-17
curvature plot 29-13
curve check
protruded feature 24-13
solid profiles 24-17
Curve Utility tools 15-9
Explode Edge 15-16
Extrude Curve 15-16
Offset Curve 15-9
Project Curve 15-13
Revolve Curve 15-12
Curve/Surface Intersection tool 20-7
Curve/Surface Projection tool 20-4
cut
command 5-1
solids 22-7
surface 16-4
Cutout Feature tool 24-11
Cylinder Primitive tools 23-6
1 Point 23-6
2 Point 23-7
Diagonals 23-7
cylindrical wrap (decals) 39-8

D

dcm
fixed constraint
preferences 49-4
fully defined constraint
preferences 49-4
over defined
preferences 49-4
over defined constraint
preferences 49-4
preferences 49-4
under defined constraint
preferences 49-4
DCM Animation Settings 47-8
Delay 47-9
End 47-9
Loop 47-9
Rebound 47-9
Running the Animation 47-9
Start 47-9
Steps 47-9
decals 39-2, 39-7
applying 39-9

auto wrap 39-10
creating 39-14
cylindrical wrap 39-8
images 39-12, 39-14
lock normal 39-10
masking 39-10
planar wrap 39-8
reflectance 39-23
rotation 39-9
shader 39-21
spherical wrap 39-8
stencil 39-11
user-defined 39-14
decimal places 2-13
decrement layer 36-7
default layers 36-4
define
color 10-3
defining spline point 14-3
work plane 37-3
definitions G-1
Deform Face tool
bend 27-15
boundary tangents 27-15
gain 27-12
resolution 27-15
stretch 27-14
to curve 27-14
to point 27-13
degree (elevate surface) 16-5
delete
construction lines 3-11
drawing view 44-10
geometry 5-3
layer 36-3, 36-5
lighting 38-16
view 35-15
work plane 37-4
deselecting 4-11
Design Explorer 5-26
change item name 5-30
close window 5-34
collapse branch 5-34
collapse item 5-34
edit parameter 5-34
expand branch 5-34
expand item 5-34
move feature 5-33
remove feature 5-34
reorder feature 5-33
suppress feature 5-32
unsuppress feature 5-30, 5-32
Designer Elements window 2-1
axis 2-2, 2-14
coordinate system axis 2-2, 2-14
drawing area 2-2, 2-15
location indicator 2-2, 2-12
menu bar 2-2, 2-15
Message Line 2-2, 2-12, 6-6
pointer 2-2
scroll bars 2-2, 2-15
Status Line 2-2, 2-12, 6-2
title bar 2-2
tool palette 2-2
tool tip 2-2
Triad 2-2, 2-14
work layer indicator 2-2, 2-15
Detail View tool 44-18
diagonal primitive
block 23-5
cone 23-9
cylinder 23-7
prism 23-13
pyramid 23-15
sphere 23-2
torus 23-11
Diagonal Rectangle tool 13-2
dialog box 2-18
Apply button 2-19
asterisks 2-19
check box 2-18
list/entry box 2-19
Option button 2-18
Diameter Arrow Dimension tool
In 32-10
Out 32-10
dimension
angular appearance 32-17
arrowheads 32-21
associative 32-1
color 32-19
default settings 49-21
drawing view 32-25, 44-5
edit objects 32-4
font 32-18
layer 32-4, 32-20
linear appearance 32-15
menu 2-16, 32-1, 32-25
move 32-3
offset 32-23
palette 32-2
size 32-18
standards 32-22
style 32-19
text 32-17
tolerances 32-16
units 32-4
weight 32-20
witness lines 32-22
work plane 32-2
dimension standard
ANSI 32-23, 32-24
Architecture 32-23, 32-24
choosing 32-25
creating 32-25
DIN 32-23, 32-24

Dual side 32-23, 32-24
Dual stacked 32-23, 32-24
ISO 32-23, 32-24
JIS 32-23, 32-24
Stacked fraction 32-23, 32-24
Dimension tools
Angular 32-11, 32-16
Balloon 32-12
Center Mark 32-11
Diameter 32-10
Horizontal 32-5, 32-16
Oblique 32-8, 32-16
Radial 32-9
Smart 32-16
Vertical 32-6, 32-16
DIN crosshatching 25-6
DIN dimension standard 32-23, 32-24
direction
change 5-11
disjoint blend 25-9
edge to face 25-10
face to face 25-10
displacement
see also roughness
shader 39-22
wrapped 39-22
display
edit objects 5-21
isolines 49-5
preferences 49-5
resolution 49-5
silhouette 49-6
Display Last Image command 40-12
Distance Pt-Pt command 29-2
Divide at Location tool 15-7
Divide tool 15-6
documentation 1-1
margin notes 1-1
style conventions 1-2
Draft 27-1
draft angle
extrude solid 24-7
lathed solid 24-2
draft angle analysis 29-19
draft angle evaluation
advanced rendering 29-26
Verify menu 29-24
draft face
local face modeling 27-1
Draft Surface tool 19-4
draft. See also taper
Drafting Assistant 3-1
% point 3-6, 49-7
alignment angles 3-6, 49-7
axis locking 3-3
construction lines 3-6
creation angles 3-6, 49-7
customizing 3-4
hit radius 3-5, 49-7
permanent construction lines 3-8
preferences 3-5, 49-6
Snap command 3-4
snap options 3-4
snap points 3-2
snapping to geometry 3-1
solid 22-3
drawing
area 2-2, 2-15
Drafting Assistant 3-1
full scale 6-11
object creation methods 6-1
Status Line 6-2
techniques 6-1
drawing composition 44-1
drawing view 44-5
Model to Sheet command 44-1
Sheet tools 44-23
unfold views 44-20, 44-23
drawing size
format 46-6
page breaks 46-10
page tiling 46-6
scale 46-7
short cut 46-12
drawing view 44-5
activating 44-6
align 44-10
center 44-11
change 44-11
delete 44-10
dimension 32-25
edit objects 44-20
edit view 44-22
flatten view 44-12
frame to extents 44-12
menu 44-6
properties 44-6
right mouse button 2-11
scale 5-22, 44-5
tool 44-13
Dual side dimension standard 32-23, 32-24
Dual stacked dimension standard 32-23, 32-24
duplicate
linear 8-10
polar 8-13
DWG
export 43-10
import 43-3
DXF
export 43-11
facet files 43-4
import 43-4
dynamic
Pan tool 35-9
render 35-3

Rotate tool 35-16
Zoom tool 35-9

E

edge
color 49-6
tolerant (stitched solid) 26-6
edge to face blend 25-10
edge to face roll blend 25-11
edit
background shader 39-24
crosshatching 25-7
drawing view 44-22
drawing view objects 44-20
foreground shader 39-24
lights 38-15
menu 2-15
parameter 5-34
pattern 10-7
render material 39-17
symbols 34-3, 34-7
text 31-5
edit a material E-13
edit objects
attributes 5-22
command 5-19
dimension 32-4
display 5-21
entering values 5-20
geometry 5-20
selecting 4-5
editing commands 5-1
Editing tools
Chamfer 15-1
Divide at Location 15-7
general 15-1
Join 15-8
solid 22-3
surface 20-1
trim 15-4
Elevate Curve tool 14-10
elevate degree (surface) 16-5
elevation (view) 35-13
Ellipse tools 12-1
2-Point Center 12-2
3-Corner 12-4
3-Point Center 12-3
modify ellipse 12-5
Opposite-Corner 12-3
Ellipsoid Primitive tool 23-16
1 Point 23-16
2 Point 23-16
By diagonals 23-17
elliptical blend 25-6
endpoint 3-2
entire object extents (selection) 4-3
Entity Color 49-16
EPS export 43-14
erase geometry 5-3
escape key 6-12
evaluation
surface 29-9, 29-10, 29-22
exit 42-5
expand (Design Explorer)
branch 5-34
item 5-34
Expand/Shrink tool
differential 8-5
uniform 8-5
Explode Edge tool 15-16
export 43-9
ACIS SAT 43-14
Adobe Illustrator 43-18
Catia v4 43-18
CGM 43-14
command 43-18
DWG 43-10
DXF 43-11
end of line 43-9
EPS 43-14
Facet 43-16
Graphite/Vellum 3D 43-10
IGES 43-14
Macromedia 43-18
options 43-9
pict 43-18
ProE/Granite 43-18
Raw 43-18
solids 22-7
STL 43-16
surfaces 16-4
tips 43-19
type 43-9
Vellum Draft 43-10
Vellum Solids 43-10
VRML 43-17
expression parsing 2-13
extrude
curve 15-16
mesh 30-4
solid 24-4
Eye Dropper tool 4-14

F

face
draft 27-1
match 27-3
move 27-4
offset 27-6
remove 27-7
replace 27-8
solids 22-1
face to face blend 25-10
facet
export 43-16
import 43-6
Fair Spline tool 14-11
feature

move 5-33
remove 5-34
reorder 5-33
suppress 5-32
unsuppress 5-30, 5-32
feature interaction
blend 25-3
chamfer 25-17
Feature tools 22-3
associative 25-1
Blends 25-2
Boss 25-29
Cutout 24-11
Holes 25-25
Protruded 24-12
file 43-7
auto save 42-5, 49-10
backup 42-5
close 42-5
compact 49-10
exit 42-5
lock 42-6
management 42-1
name 42-3
native picture formats 49-9
new 42-2
open 42-3
preferences 42-5, 49-9
recent list 42-3, 49-10
revert 42-5
save 42-4
short cuts 42-1
size 42-6
fill
color 25-1, 25-4
holes 25-3
overlapping objects 25-3
pattern 25-1
Fillet Surface tool 19-5
Fillet tools
2-Entity 15-2
3-Entity 15-2
fixed width
variable blend 25-14
variable chamfer 25-24
flat render 35-3
flatten view 44-12
Fly By animation 41-5
Focal Point E-7
font 31-3
dimension 32-18
plotter 46-13
font (default) 49-21
foregrounds
applying 39-13
color 49-3
editing 39-24
library 39-2, 39-13
settings 39-25
types 39-25
format
drawing size 46-6
four lengths variable chamfer 25-22
Frame to Extents command 44-12
front view 35-11
full scale 6-11

G

G1 surface 21-2
G2 surface 21-3
gain (deform face) 27-12, 27-14
Gaussian plot 29-15
global
coordinate system 6-5
work plane 6-6
Glossary G-1
Gouraud render 35-3
Graphite/Vellum 3D
export 43-10
grid
color 49-13
preferences 49-12
Show command 2-15
Snap command 2-15, 6-11
spacing 49-12
group
command 5-7
objects 5-7
selection 5-7

H

Hatch command 25-7
Helix Curve
creating a helix 14-6
Helix Curve Tool 14-3
Helix Curve tool 14-6
pitch 14-6
hidden
lines 44-2, 44-7
render 35-4
hide
axis 6-10
command 35-17
layer 36-5
points 6-11
Triad 6-10
work plane 6-11
histogram 29-14
hit radius 3-5, 49-7
hold line blend 25-15
Hole Feature tools 25-25
Counter Bore 25-28
Counter Sink 25-28
direction 25-27
Simple 25-27
Through Types 25-26
Horizontal Dimension tools 32-5
Base Line 32-6

- Chain 32-6
- Horizontal 32-16
- Ordinate 32-8

Horizontal Section View tool 44-16
Horizontal Text tool 31-1
hot spot 2-11
HPGL 46-13

I

IGES
- export 43-14
- import 43-5

image (shader) 39-21, 39-22, 39-23
Image Based Textures E-18
import 43-1
- ACIS SAT 43-6
- Adobe Illustrator 43-7
- auto heal bodies 43-2
- command 43-7
- DWG 43-3
- DXF 43-4
- facet 43-6
- IGES 43-5
- options 43-2
- pict 43-6
- spline points 43-7
- symbols 34-7
- tips 43-8
- type 43-2
- Vellum 3D 43-2
- Vellum Solids 43-2

increment layer 36-7
inertia (moment of) 29-4
Infinite Plane tool 17-1
- 3 Pts In Plane 17-2
- Along Curve 17-2
- Clipping Planes 17-4
- Normal Location 17-2
- X, Y, Z Station 17-2

influence (spline) 14-4
Inscribed Polygon tool 13-4
- scribe type 13-5
- show frame 13-5

insert knot 21-6
Insert tool (assembly) 28-4
instance 1-2
- solid 22-5

intensity (lighting) 38-3
Interference command 29-4
intersect
- curve/solid 20-8
- curve/surface 20-8
- plane/curve 20-3
- plane/solid 20-4
- plane/surface 20-3
- solids 26-3
- surface/solid 20-7
- surface/surface 20-7
- surfaces 20-9

Invert command 35-18
ISO crosshatching 25-5
ISO dimension standard 32-23, 32-24
isolines 5-5, 49-5
isometric view 35-11
isopram lines 5-5, 49-5
item (Design Explorer)
- collapse 5-34
- expand 5-34

J

JIS dimension standard 32-23, 32-24
Join tool 15-8

K

key
- arrow nudge distance 49-12
- escape 6-12
- short cut 49-21, 49-23

L

Lathed Solid tool 24-2
law curve blend 25-13
layer 36-1
- auto 20-2
- change 5-6
- color 36-3, 36-8
- copying/pasting 36-9
- creating 36-4
- decrement 36-7
- default 36-4
- delete 36-3, 36-5
- dimension 32-4, 32-20
- hide 36-5
- increment 36-7
- isolate 36-5, 36-7
- lock 36-3, 36-6
- manager 36-2
- new 36-3
- rename 36-4, 36-5
- show 36-6
- unlock 36-6
- work 36-2, 36-7

layout 44-2
- creating 44-23
- lighting 38-16
- menu 2-16
- modify 44-23
- printing 44-24
- templates 44-23

length angle constant chamfer 25-19
Length command 29-2
lengths angles variable chamfer 25-23
library
- Backgrounds 39-2, 39-12
- Decals 39-2, 39-7
- Foregrounds 39-2, 39-13
- Materials 39-2
- Render 39-1

Light tools
- Point 38-5

Spot 38-3
light's distance E-10
lighting 38-1
ambient 38-14
attenuation 38-4
color 38-14, 38-15
delete 38-16
directional 38-1
edit 38-15
enable 38-3
intensity 38-3
layouts 38-16
modify 38-14
move 38-14
palette 38-1
render 35-8, 38-19
shadows 38-3, 38-4, 38-6, 38-8, 38-10, 38-11, 38-13
Line Pattern manager 10-6
Line tools 11-1
Connected Lines 11-3
Parallel Line 11-4
Point 11-5
Single Line 11-1
linear blend 25-12
linear duplicate 8-10
lines (modifying) 11-6
links 5-25
remove 5-26
list/entry box 2-19
Local Face Modeling tool
draft 27-1
match face 27-3
Modifying Taper 27-6
move face 27-4
offset face 27-6
remove face 27-7
replace face 27-8
Local Face Modeling tools 27-1
Local Surface Tools
elevate degree 16-5
insert knot 21-6
untrim 16-5
Local Surface tools 21-1, 29-1
Match Surface 21-1
Rebuild Surface 21-4
localization (preferences) 49-14
location indicator 2-2, 2-12
lock
files 42-6
layer 36-3, 36-6
normal (decals) 39-10
objects 5-8
Lofted 19-2
Lofted Solid tool 26-8
bulge factor 26-8
error 26-8
Lofted Surface tool 19-2
Lop Solid tool 26-11

M

Macintosh
page setup 46-12
Macromedia
export 43-18
margin notes 1-1
masking (decals) 39-10
match face 27-3
Match Surface tool 21-1
G1 slope 21-2
G2 slope 21-3
Mate tool (assembly) 28-2
material
crosshatching 25-5
Properties command 29-3
Render library 39-2
user-defined 39-14
material Categories E-13
material editing 39-17
mathematical operators A-1
menu 1-1
choosing commands 1-1, 2-16
dimension 2-16
displaying 2-16
drawing view 44-6
Edit 2-15
Layout 2-16
Pen 2-16
PhotoRender 2-16
Planes 2-16
submenu 2-17
Text 2-16
toggling commands 2-19
Trackball 6-9
Verify 2-16
View 2-16
Window 2-16
menu bar 2-2
mesh
aspect ratio 5-16, 5-18
count 5-16, 5-18
normal deviation 5-16, 5-18
smoothing 5-18
surface deviation 5-16, 5-18
Mesh tools
3 Pts 30-2
4 Pts 30-2
Curve 30-3
extrude 30-4
rendering 30-5
spacing 30-3
Message Line 2-2, 2-12, 6-6
Metal E-16
midpoint 3-2
Minimum Distance command 29-2
Mirror tool 8-7
2 Pts 8-8

3 Pts 8-9
XY plane 8-8
ZX plane 8-9
ZY plane 8-8
mitered blend 25-8
mitered constant chamfer 25-21
Model to Sheet command 44-1
layouts 44-2
undoing 44-4
modeling
solid 22-1
surface 16-1
wireframe 7-1
Modify Slope tool 14-8
explicit 14-8
free 14-8
reference 14-8
reverse 14-8
modifying
ellipses 12-5
layout 44-23
lighting 38-14
lines 11-6
parting line solid 27-11
tapered solid 27-6
view 35-14
moment of inertia 29-4
mouse 1-3
arrow pointer 1-3
click 1-3
double-click 1-3
drag 1-3
pointer 1-3
right button 2-5
selection arrow 1-3
smart pointer 2-11
Move 27-5
move
dimension 32-3
face 27-4
feature 5-33
light 38-14
objects 8-21
origin 37-7
text 31-5
tools 8-2
movie
Fly By 41-5
Paths 41-5
VR Object 41-7
VR Panoramic 41-9
Walk Through 41-4
multi-radius blend 25-8

N

n# sections 18-5
normal deviation (mesh) 5-16, 5-18
normals plot 29-20
n-sided 18-9
nurb splines 14-1

O

Object Counts command 29-7
object resolution E-2
object type (preferences) 49-5
objects 35-18
align 5-9
area 29-2
arrange 5-8
change type 5-12
check 29-7
copy 8-23
count 29-7, 36-3
crosshatching 25-7
dimensions 32-3
direction 5-11
drawing view 44-20
edit 5-19
group 5-7
hide 35-17
interference 29-4
length 29-2
lock 5-8
move 8-21
properties 29-3
selecting 4-1
show 35-17
show All 35-17
sizing 8-24
transparency 40-8
ungroup 5-8
unlock 5-8
volume 29-3
Oblique Dimension tool 32-8, 32-16
Off Screen Objects E-8
offset
construction lines 3-9
curve 15-9
dimension 32-23
linear duplicate 8-11
solid face 27-6
Surface tool 19-1
work plane 37-8
Offset Curve tool
by point 15-11
to value 15-10
one edge vertex blend 25-9
open file 42-3
Opposite-Corner Ellipse tool 12-3
Opposite-Point Circle tool 11-5
Option button 2-18
ordinate dimension
horizontal 32-8
orientation (blend) 25-4
origin
move 37-7
set 37-7

P

page breaks (drawing) 46-10
page setup 45-1
page tiling 46-6
palette 2-2
    Assembly Modeling 28-1
    Color 10-3
    Dimension 32-1
    floating 2-3
    Lighting 38-1
    Render library 39-2
    selecting a tool 2-3
    Sheet tools 44-23
    Solid 22-2
    subpalette 2-4
    Surface 16-2
    Symbols 34-1
    tear away 2-5
    Wireframe 2-4
pan
    view 35-11
pan (movie) 41-8, 41-9
Parallel Dimension tool
    see Oblique Dimension tool
Parallel Line tool 11-4
param (blend) 25-13
param (chamfer) 25-24
Parametric Constraints 47-1
    Sketch Mode 47-1
    Sketch Mode Exiting 47-2
parent/child 1-3
partial objects extents (selection) 4-3
Parting Line Solid tool 27-9
    modifying 27-11
    pull-direction 27-9
    taper (draft) 27-10
paste
    command 5-2
    objects/layer 36-9
    solids 22-7
    Status Line entries 2-13, 6-3
    surfaces 16-4
Paths animation 41-5
pattern
    choosing pen 10-8
    construction line 10-8
    creating 10-6
    crosshatching 25-4
    DIN crosshatching 25-6
    editing 10-7
    fill 25-1
    ISO crosshatching 25-5
    manager 10-6
    new 10-6
    pen 10-5
pen
    arrowheads 10-8
    characteristics 10-1
    color 10-2
    default settings 49-21
    menu 2-16
    pattern 10-5
    pattern manager 10-6
    settings 10-1
    style 10-1
    weight 10-4
permanent construction lines 3-8
perpendicular 3-3
photorealistic E-1
    commands 40-9
    rendering 40-7
    settings 40-7
pick box 4-2, 49-16
Pick Objects command (plane) 37-6
pict
    export 43-18
    import 43-6
pitch (helix) 14-6
pixels (pen weight) 10-4
planar wrap (decals) 39-8
plane 37-1
    3 Pt 37-6
    pick objects 37-6
    views 35-19
    work 37-2
plane (work) 6-6
Plane/Surface Intersection tool 17-3, 20-1
Planes menu 2-16
Plotter fonts 46-13
point operations
    selectable points 4-9
    selecting 4-7
    show/hide 4-7, 6-11
Point tools 11-5
    Along a Curve 11-5
    Along a Surface 11-6
    At XYZ Location 11-5
    Light 38-5
    marker 11-5
pointer 1-3, 2-2
    location indicator 2-12
    smart 2-11
    solids 20-1
polar duplicate 8-13
Polygon from Curves tool 13-8
    show frame 13-8
Polygon tools
    Arbitrary 13-7
    Circumscribed 13-5
    Inscribed 13-4
    Polygon from Curves 13-8
    Rectangle 13-2
    Single Line 13-1
    Smart Polygon 13-1
PostScript 46-13
pre-defined views 35-11

preferences 49-2
- changing settings 49-20
- color 49-3
- command 49-20
- dcm 49-4
  - fixed constraint 49-4
  - fully defined constraint 49-4
  - over defined 49-4
  - over defined constraint 49-4
  - under defined constraint 49-4
- display 49-5
- Drafting Assistant 3-5, 49-6
- file 42-5
- filing 49-9
- general 49-11
- grid 49-12
- localization 49-14
- object type 49-5
- resolution 49-5
- selection 4-2, 49-15
- short cut 49-21
- units 49-19
- user interface 49-11
- view definitions 49-12

preview sample (shader) 39-19

primitive (ellipsoid) 23-16

Primitive tools 22-2
- Block 23-4
- Cone 23-8
- Cylinder 23-6
- Prism 23-12
- Pyramid 23-14
- Slab 23-3
- Sphere 23-1
- Torus 23-10

print 46-1
- boundaries 46-13
- command 46-13
- layout 44-24
- region 46-13
- render 46-14
- setup 46-12
- to file 46-13
- Window command 46-14

Print Layout Command 46-1

Prism Primitive tools 23-12
- 1 Point 23-12
- 2 Point 23-13
- Diagonals 23-13

ProE/Granite
- export 43-18

Profile Solid tools
- curve checks 24-17
- extrude 24-4
- Lathed 24-2
- Skinned 24-14
- Swept 24-9

Profile tools 22-2

project
- curve 15-13
- curve/solid 20-6
- curve/surface 20-6

Project Curve tool
- plane options 15-14
- projection options 15-15

Properties command 29-3
- center of gravity 29-4
- density 29-4
- material 29-3
- moment of inertia 29-4
- volume 29-4
- weight 29-4

Properties command (planes) 37-4

Protruded Feature tool 24-12
- curve check 24-13
- termination types 24-12

pull-direction (parting line solid) 27-9

Pyramid Primitive tools 23-14
- 1 Point 23-14
- 2 Point 23-15
- diagonals 23-15

Q

QuickTime movie
- compressor 41-3
- Fly By 41-5
- Paths 41-5
- VR Object 41-7
- VR Panoramic 41-9
- Walk Through 41-4
- with camera movement 41-1

R

Radial Arrow Dimension tool
- In 32-9
- Out 32-9

radial blend 25-5

ratio (elliptical blend) 25-6

Raw export 43-18

ray cube (background) 39-25

ray trace
- commands 35-3, 40-10
- reflections 40-8
- shader 39-22

Rebuild Surface tool 21-4

recent file list 42-3, 49-10

Rectangle tools 13-2
- Center/Axis 13-3
- Center/Corner 13-2
- Diagonal 13-2

redo 5-3

redraw screen 35-17

reflectance shader 39-22
- decals 39-23
- ray trace 39-22
- wrapped 39-23

Reflective Objects E-8

reflectivity 5-23, 39-17

Relimit tool 15-5
remove
- control point (spline) 14-7
- face 27-7
- feature 5-34
- links 5-26

rename
- rename (Design Explorer) 5-30

render
- anti-alias 40-8
- applying materials 39-5
- Backgrounds library 39-2, 39-12
- copying characteristics 4-14
- Decals library 39-2
- display last image 40-12
- Eye Dropper tool 4-14
- Foregrounds library 39-2, 39-13
- image files 40-12
- library 39-1
- lighting 35-8, 38-1, 38-19
- Materials library 39-2
- menu 2-16
- mesh 30-5
- photorealistic 40-7, 40-9
- print 46-14
- ray trace 40-8
- reflectivity 5-23, 39-17
- roughness 5-23, 39-18
- shade options 35-2
- shader 39-19
- shadows 5-23, 39-18
- textures 5-23, 39-18
- to file 40-11
- transparency 5-23, 39-17, 40-8

render area 40-10
render options
- dynamic 35-3
- flat 35-3
- Gouraud 35-3
- hidden 35-4
- static 35-3
- view rotations 35-7

reorder feature 5-33
replace face 27-8
resolution
- change 5-4
- deform face 27-15
- isolines 5-5
- preferences 49-5
- silhouettes 5-5
- solid 22-4
- surface 16-3
- wireframe 7-3

revert file 42-5
Revolve Curve tool 15-12
Revolved Surface tool
- About a Curve 18-11
- About Two Points 18-11

rho (conic) 12-5
Rib Solid tool 26-9
right mouse button 2-5
- color 2-8
- copy as instance 2-10
- copy part history 2-10
- drawing view 2-11
- edit objects 2-9
- hide all 2-7
- hide/show only 2-8
- layer 2-9
- object characteristics 2-10
- object name 2-9
- over an object 2-8
- over no object 2-6
- planes 2-6
- resolution 2-8
- select all 2-6
- show all 2-7
- transparency 2-10
- view 2-6

right-hand rule 2-14
rotate
- decals 39-9
- dynamic (view) 35-16
- view 35-13, 35-16

Rotate tools 8-3
- 1 Pt 8-3
- 3 Pts 8-4

roughness
- Edit Objects 5-23, 39-18
- see also displacement

Rule of Thirds E-7
rules
- Design Explorer 5-35
- grouping 5-7
- local face modeling 27-1
- relimit 15-5
- right-hand rule 2-14
- sweep surface 18-15

S

SAT
- export 43-14
- import 43-6

save 42-4
- auto 42-5, 49-10
- save as 42-4

scale
- differential 8-5
- drawing size 46-7
- drawing view 44-5
- full 6-11
- ignore 5-22
- ignore (drawing view) 44-5
- material 5-23, 39-18
- sheet 44-5
- uniform 8-5

screen horizontal

text 31-5
scribe type
circumscribed polygon 13-6
inscribed polygon 13-5
scroll bars 2-2, 2-15
Section 1
Create the Environment E-2, E-9
Section 2
Lighting the Environment E-9
Section 3, Make the Environment Come Alive E-13
Section View tools 44-15
2 Pts 44-16
crosshatching 25-7, 44-17
Horizontal 44-16
Vertical 44-15
Segment tool 15-7
selected points 49-16
selection
ambiguity popup 4-6, 49-16
chain 4-11
color 4-3
control point 4-8
deselecting 4-11
edit objects 4-5
entity color 49-16
fence 4-2, 49-16
Group command 5-7
hollow arrow 20-1
objects 4-1, 4-4, 4-9
pick box 4-2
points 4-7
preferences 4-2, 49-15
select all 4-9
selected points 49-16
selection mask 4-9
tool 2-3, 4-4
transparency 4-4, 49-16
wire weight 4-3, 49-16
selection transparency 49-16
set origin 37-7
shade now 35-2
command
render
shade now 35-2
Shade Options 35-2
shader
attribute value 39-19
attributes 39-19, F-1
backgrounds 39-25
class 39-19
foregrounds 39-25
preview sample 39-19
texture projection 39-23
types 39-19
shader class 39-19
color 39-21
displacement 39-22
reflectance 39-22
texture space 39-23
transparency 39-23
shader types 39-19
shadow casting (lighting) 38-3, 38-4, 38-6, 38-8, 38-10, 38-11, 38-13
Shadows E-11
Shadows with Gel Images E-12
Sheet tools 44-23
Auxiliary View 44-13
Detail View 44-18
drawing view 44-13
Model to Sheet command 44-1
Section View 44-15
short cut
creating 49-23
file 42-1
list C-1
preferences 49-21
view rotation 6-10
short cut (tool tip) 49-21
shoulder point (conic) 12-5
show
axis 6-10
command 35-17
control point 4-7
Dimension palette 32-2
layers 36-6
points 6-11
Show All command 35-17
Show Only command 35-18
silhouettes 5-5
Triad 6-10
work plane 6-11, 37-2
show frame
arbitrary polygon 13-7
circumscribed polygon 13-6
inscribed polygon 13-5
polygon from curves 13-8
show work plane 37-4
side view 35-11
silhouettes 5-5, 49-6
Simple Hole tool 25-27
Simple Trim tool 15-4
Simplify object
change 5-19
edit 5-19
simplify object
command 5-19
simplify surfaces (stitched solid) 26-5
Single Line tool 11-1
size
dimension 32-18
objects 8-24
text 31-4
Sketch Mode 47-1
Animating Dimensions Tool 47-8
Auto Constraints Tool 47-2

- Coincident Constraint Tool 47-3
- Colinear Constraint Tool 47-7
- Concentric Constraint Tool 47-6
- Constraint Preferences 47-9
- Deleting Constraints 47-9
- Equal Constraint Tool 47-7
- Exiting 47-2
- Fixed Constraint Tool 47-6
- Horizontal Constraint Tool 47-3
- Offset Constraint Tool 47-8
- Parallel Constraint Tool 47-5
- Perpendicular Constraint Tool 47-5
- Sketch Tool 47-2
- Symmetric Constraint Tool 47-7
- Tangent Constraint Tool 47-5
- Verifying Constraint Relations 47-9
- Vertical Constraint Tool 47-3

sketch spline
- creating a spline 14-5

Sketch Spline tool 14-5
Skin Solid tool
- guides 24-17

Skin Surface tool
- basic 18-6
- closed skin 18-6
- guides 18-7
- n# sections 18-5
- standard 18-5

Skinned Solid tool 24-14
- basic 24-16

Slab Primitive tool 23-3
slope (conic) 12-5
smart
- hot spot (pointer) 2-11
- pointer 2-11
- polygon 13-1

Smart Dimension tool 32-16
Snap command 3-4
solid
- associativity 22-5
- cutting/pasting 22-7
- display edges 22-4
- Drafting Assistant 22-3
- exporting 22-7
- instances 22-5
- modeling 22-1
- resolution 22-4
- tools 22-2

Solid Editing tools
- Assembly Modeling 28-1
- Boolean 26-2
- Chamfer Edge 25-16
- Features 22-3

solid modeling
- Ellipsoid Primitive tool 23-16
- error messages 22-7

Solid Utility tools 26-1
- Intersect Solid 26-3
- Lip Solid 26-11
- Local Face Modeling Tools 27-1
- Lofted Solid 26-8
- Parting Line Solid 27-9
- Rib Solid 26-9
- Split Solid 26-5
- Stitched 26-5
- Subtract Solid 26-2
- Thicken Solid 26-6
- Union Solid 26-2

Solids Creation tools
- Primitives 22-2
- Profiles 22-2

special characters B-1
Sphere Primitive tools 23-1
- 1 Point 23-2
- 2 Point 23-2
- Diagonals 23-2

spherical wrap (decals) 39-8
spline
- control point 14-2, 14-3
- creating text files D-1
- import points 43-7

Spline tools 14-1
- Add Control Point 14-7
- Bezier 14-4
- Elevate Curve 14-10
- Fair Spline 14-11
- Helix Curve 14-6
- influence 14-4
- Modify Slope 14-8
- nurbs 14-1
- Remove Control Point 14-7
- Sketch Spline 14-5
- Through-Points B-Spline 14-2
- Vector 14-3

Split Solid tool 26-5
Spot Light tool 38-3
- cone angle 38-5
- falloff angle 38-5
- falloff rate 38-5

Stacked fraction dimension standard 32-23, 32-24
static render 35-3
Status Line 2-2, 2-12, 6-2
- copying/pasting entries 2-13, 6-3
- expression parsing 2-13
- units 6-3
- world coordinate system 2-13

stencil (decal) 39-11
Stitched Solid tool 26-5
- gap size 26-5
- options 26-5
- simplify surfaces 26-5
- tolerant edges 26-6

STL export 43-16
stretch (deform face) 27-14
stroke construction lines 3-8

stroke zoom 35-10
style
  dimension 32-19
  pen 10-1
  text 31-4
submenu 1-1, 2-17
subtract
  solid 26-2
  surface 20-9
suppress feature 5-32
surface
  associativity 16-4
  cutting/pasting 16-4
  Editing tools 20-1
  evaluation 29-9
  exporting 16-4
  Mesh tools 30-1
  modeling 16-1
  resolution 16-3
  tools 16-2
surface analysis
  advanced rendering 29-22
  curvature plot 29-13
  draft angle 29-19, 29-24
  Gaussian 29-15
  histogram 29-14
  normals plot 29-20
  Verify menu 29-10
  zebra plot 29-21
Surface Creation tools
  Cover Surface 18-9
  Draft Surface 19-4
  Fillet Surface 19-5
  Infinite Plane 17-1
  Lofted Surface 19-2
  Offset Surface 19-1
  Sweep Surface 18-12
  Tube Surface 18-15
surface deviation (mesh) 5-16, 5-18
Surface Editing tools
  Boolean Surface 20-8
  Connect Surface 20-10
  Curve/Surface Intersection 20-7
  Curve/Surface Projection 20-4
  Plane/Surface Intersection 20-1
  Split Surface 20-10
  surface evaluation 29-9
  Surface/Surface Intersection 20-7
  untrim 16-5
Surface From Curves Tools 18-1
  Net Surface Tool 18-1
  Surface Tools 17-10, 18-1
Surface from Curves Tools
  Net Surface 18-1
Surface/Surface Intersection tool 20-7
Sweep Surface tool
  Along 1 Rail 18-13
  Along 2 Rails 18-14
Sweep Surface tool Along 2 Points 18-12
Swept Solid tool 24-9
symbol 34-1
  1 Point tool 34-4
  2 Point tool 34-4
  creating 34-3
  edit 34-3, 34-7
  files 34-7
  importing 34-7
  manager 34-2
  palette 34-1
  transforming 34-6

T

tangent 3-3
Tangent Surface tool
  bulge factor 19-3
tangent-point
  arc 11-3
  circle 11-6
taper
  parting line solid 27-10
text
  case 31-5
  convert to curves 5-14
  default settings 49-21
  dimension font 32-18
  dimension size 32-18
  dimension style 32-19
  dimension text 32-17
  drawing view 44-5
  editing 31-5
  font 31-3
  menu 2-16
  move 31-5
  screen horizontal 31-5
  size 31-4
  style 31-4
Text tools 31-1
  Along a curve 31-2
  At an angle 31-2
  box 31-3
  Horizontal 31-1
texture (shader) 39-21
texture projection shader 39-23
texture space shader 39-23
textures 5-23, 39-18
Thicken Solid tool 26-6
Through-Points B-Spline tool 14-2
tile
  page 46-6
tilt (movie) 41-8
tips
  exporting 43-19
  import 43-8
  tool 2-2
title bar 2-2
toggling commands 2-19
tolerances (dimension) 32-16

tool tip 2-2
short cut 49-21
tools
Arc 11-1
Circle 11-1
Conic 12-1
Ellipse 12-1
Line 11-1
Mesh 30-1
Polygon 13-1
Rectangle 13-2
Selection 4-4
solid modeling 22-2
Solid Utility 22-3
surface modeling 16-2
wireframe 7-2
top view 35-11
Torus Primitive tools 23-10
1 Point 23-10
2 Point 23-11
Diagonals 23-11
Trackball 6-6
Axis Locking 6-6
rotation options 6-7, 6-8
save view 35-14
Sphere 6-7
Step 6-7
view menu 6-9
Transformation tools 8-1
Align 8-21
Copy Along Path 8-9
Linear Duplicate 8-10
Mirror 8-7
Move 8-2
Polar Duplicate 8-13
Rotate 8-3
transparency
Edit Objects 5-23, 39-17
selection 4-4
shader 39-23
shadows 40-8
wrapped 39-23
Transparent Shadows E-12
Triad 2-2, 2-14
show/hide 6-10
trim operations
surface/curve 20-11
surface/solid 20-11
surface/surface 20-11
Trim Solid Tool 26-3
Trim tools 15-4
Corner 15-7
Divide 15-6
Divide at Location 15-7
Relimit 15-5
Segment 15-7
Simple Trim 15-4
Surface 20-10
untrim surface 16-5
trimetric view 35-12
Tube Surface tool 18-15
turbulent (shader) 39-21
twist angle (swept solid) 24-9
two edge vertex blend 25-9
two lengths constant chamfer 25-19

U

u/v lines 5-5, 49-5
undo 5-3
unfold view 44-20, 44-23
ungroup 5-8
uninstalling 42-6
units
mixed 32-4
pen weight 10-4
precision 2-13
preferences 49-19
setting 32-4
Status Line 6-3
unlock
layers 36-6
objects 5-8
unsuppress feature 5-30, 5-32
untrim surface 16-5
Use View command (plane) 37-5
user interface 49-11
user-defined
coordinate system 6-5
standard 32-25
work plane 37-3
user-defined views 35-12
Utility tools (solid) 22-3
uv shader 39-23

V

variable blend
by position 25-12
fixed width 25-14
hold line 25-15
law curve 25-13
linear 25-12
Variable Blend tool 25-12
variable chamfer
by position 25-24
fixed width 25-24
four lengths 25-22
lengths 25-22
lengths angles 25-23
Variable Chamfer Edge tool 25-21
variable lengths chamfer 25-22
Variables and Equations 48-1
Adding Equations 48-3
Highlighting Dimensions in the Sketch 48-2
Using Equations 48-1
Vector Spline tool 14-3
Vellum 3D
import 43-2

Vellum Draft export 43-10
Vellum Solids
- export 43-10
- import 43-2

Verify commands
- Angle 3 Pts 29-1
- Area 29-2
- Check Object 29-7
- Distance Pt-Pt 29-2
- Interference 29-4
- Length 29-2
- Minimum Distance 29-2
- Object Counts 29-7
- Properties 29-3
- Volume 29-3
- X,Y,Z 29-1

Verify menu 2-16, 29-1
vertex blend 25-7
vertex chamfer 25-20
Vertical Dimension tools 32-7
- Base Line 32-7
- Chain 32-7
- Vertical 32-16

Vertical Section View tool 44-15
view
- azimuth 35-13
- delete 35-15
- drawing 44-5
- dynamic rotate 35-16
- elevation 35-13
- menu 2-16
- Modify command 35-14
- New View command 35-12
- pan 35-11
- planes 35-19
- pre-defined 35-11
- preferences 49-12
- redraw screen 35-17
- rotate 35-13, 35-16
- rotation short cut 6-10
- show/hide 35-17
- Trackball 35-14
- Trackball rotation 6-7, 6-8
- unfold 44-20, 44-23
- user-defined 35-12
- view the plane 35-17
- work plane 37-8
- zoom 35-10, 35-19

View Angles E-6
Views pop-up menu 6-10
Volume command 29-3
VR Object movie 41-7
VR Panoramic movie 41-9
VRML export 43-17

W

Walk Through animation 41-4
weight
- dimension 32-20
- pen 10-4

Window menu 2-16
Windows
- print setup 46-12
- uninstalling 42-6

wire weight 49-16
- selection 4-3

wireframe
- Drafting Assistant 7-3
- modeling 7-1
- resolution 7-3
- tools 7-2

witness lines 32-22
work layer 36-2, 36-7
- indicator 2-2, 2-15, 36-5, 36-7

work plane 37-2
- 3 Pts 37-6
- define 37-3
- delete 37-4
- dimensions 32-2
- dimming 37-7
- global 6-6
- new 37-3
- offset 37-8
- origin 37-7
- pick objects 37-6
- properties 37-4
- save current 37-4
- setting 37-5
- show 37-2, 37-4
- show/hide 6-11
- snapping 37-6
- use view 37-5
- user-defined 37-3
- view the plane 37-8
- views 37-8
- world coordinate system 37-8

world coordinate system
- Status Line 2-13
- work plane 37-8

wrap (decal)
- auto 39-10
- cylindrical 39-8
- planar 39-8
- spherical 39-8

wrapped
- displacement 39-22
- image 39-21, 39-22, 39-23
- reflectance 39-23
- texture 39-21
- transparency 39-23

X

X,Y,Z command 29-1

Z

zebra plot 29-21
zoom
- all 35-8
- commands 35-8

dynamic 35-9
home 35-8
in 35-8
    out 35-9
out 35-8
previous 35-8
ratio 35-9
stroke 35-10
tools 35-9
views 35-10, 35-19
window 35-8