

Punch! Shark FX User's Manual

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Punch! Shark FX User's Manual

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Part 1

Getting Started

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Documentation

This User's Manual is organized for both the beginner and the experienced user. It is designed as an introduction to the tools for new users, and as a reference guide to be used long after you have mastered the basics of the program. To accomplish these goals, this manual is organized into seven parts, each part covering a specific group of tools and operations.

Part 1: Getting Started

The seven chapters associated with this section help you get your product properly installed, teach you the basics of the application environment, and prepare you with a solid foundation for creating and manipulating curves, surfaces, and solid models.

Part 2: Menu Commands

This section describes the commands available from the Menu Bar including File, Edit, View, Pen, Text, and Dimension.

Part 3: Drawing Tools

Introduces the designer to tools for curve and wireframe modeling. This section covers object creation and editing of points, lines, arcs, circles, ellipses, conics, and splines.

Part 4: Surface Modeling

Surface modeling is a powerful tool for describing complex shapes that span areas. This section covers the NURBS surface capabilities of the product.

Part 5: Solid Modeling

Solid modeling tools define shapes that define volumes. This section covers creating solids from primitives, profiles, Booleans, and features.

Part 6: Drawing Composition

After you've created a model, you may need to create a 2D drawing from the 3D model. This section covers the Model To Sheet command and the Sheet Tool palette.

Part 7: Rendering

As you're creating, there are a variety of ways to view your design, from 2D wireframe and transparent views to fully rendered 3D views, including shadows. This section covers the rendering and animation tools available to enhance the design process.

Using This Manual

To get the most out of this manual you should be familiar with the basics of using a computer. For example, you should know how to start up your computer and make menu selections, have some familiarity with using a mouse, and be able to manipulate windows. If not, please refer to your computer's owner guide. In addition, you should be familiar with some of the terms used throughout this manual regarding use of the mouse:

Mouse Command	What you do:
----------------------	---------------------

Press	Press and hold down the mouse button.
Click	Quickly press and release the mouse button.
Point and click	Position the cursor and then click the mouse button.
Double-click	Click the mouse button twice in rapid succession.
Drag	Move the mouse while holding down the button.
Control-click	Press the control key and the mouse button. If you have a two button mouse, press the right button (Right-click).

Installation

System Requirements

This software product runs on Macintosh® or Windows® computers that satisfy the following requirements:

Windows

Recommended Requirements

- Microsoft® Windows® XP Home, Windows® XP Professional, Windows® Vista™ or Windows® 7
- Intel® Pentium® or AMD® Athlon™ class processor
- 4 GB of hard disk space
- 1 GB of RAM
- Mouse Pointing Device (wheel button recommended)
- DVD-ROM Drive

Macintosh

Recommended Requirements

- Intel® Mac, Macintosh® OS 10.4 or higher
- 4 GB of hard disk space
- 1GB RAM or greater
- Mouse Pointing Device (wheel button recommended)
- DVD-ROM Drive

Installation

The application consists of a collection of files and folders extracted from the installer. The installer will decompress the following files and folders:

Punch! Shark FX (Mac)

Application bundle used to launch the program.

EXE (PC Only)

The Exe folder contains the executable and shared libraries needed to run the application.

BackUp

Folder where files are placed when using the Backup options for the application. This folder is located under your user directory.

BOM

ASCII files that serve as templates for Bill of Materials.

DXF

This folder contains a data file used in support of DWG and DXF translations.

Environ

The Environ folder contains a collection of preference files that are used to customize the look and feel of the application. This contains user settings for colors, line fonts, cross hatching, and keyboard short cuts.

Templates

This folder contains a collection of templates that are used to support the Model To Sheet tool.

Strings

The strings folder contains information needed to localize the application for different languages.

Symbols

The Symbols folder contains a collection of files displayed under the Symbol tab of Concept Explorer.

Launching Punch! Shark FX

After installing the software onto your hard disk start the application by:

- 1 Double-clicking the Punch! Shark FX folder to open it.
- 2 Double-clicking the Punch! Shark FX application icon.

Punch! Shark FX will launch and be ready to create your first model.

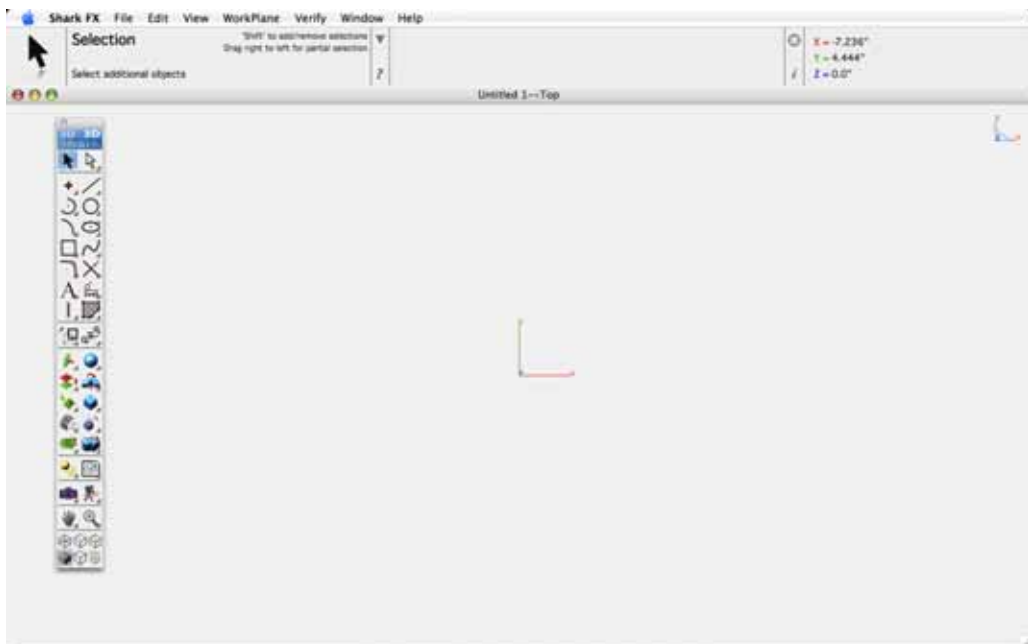
First Look

This chapter provides a description of the application menu and tool palettes. Descriptions include:

- Startup Screen
- Menu Bar
- Prompt Window
- Main Tool Palette
- Mouse Button Functions

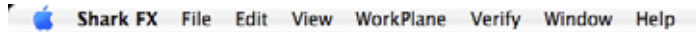
Startup Screen

The default startup screen for the application consists of the Menu Bar, Main Tool Palette, Prompt, and Data Entry Window.

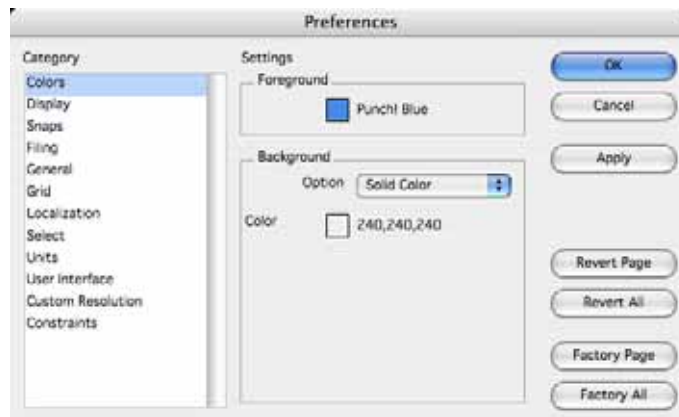
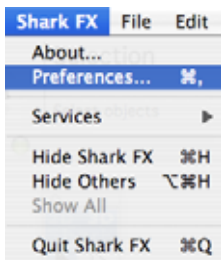


Menu Bar

The Menu Bar contains a list of pull-down menus such as filing, editing, and viewing. Some of the pull-down menus contain “hierarchical” submenus. A right pointing arrow in the menu identifies hierarchical menus. To access a pull-down menu simply press the mouse, move it to the desired option, and release the mouse button. To access options in a hierarchical submenu move the mouse to the right and the submenu will appear. Continue holding the mouse until you are over the desired submenu option.



Menus with three dots (...) indicate that a dialog box will appear upon selection.



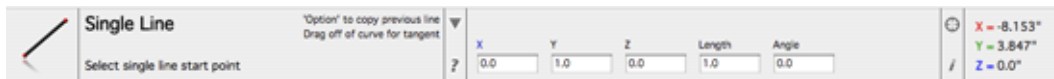
The Menu Bar contains the following items. These items, from left to right, are:

- File** Commands for opening, saving, importing, and printing files.
- Edit** Commands for copy, paste, cut, layer, resolution, and direction.
- View** Commands for changing the view orientation of your file.
- WorkPlane** Commands for creating, setting, and modifying the work plane.
- Verify** Verify geometry properties associated with an object. Properties include position, angles, length, areas, and mass properties.
- Window** Commands for displaying tool palettes and windows such as Concept Explorer, Rendering, Snaps, and Object Masking.
- Help** Provides access to online help resources.

Prompt Window

The Prompt Window displays a brief description of what the application is expecting you to do for the current command. In addition to the prompt string, it displays a large detailed tool icon, the tool name,

tool hints, and help information. To the right of the prompt strings you will find the data entry fields, and to the far right are the screen coordinates of the cursor.



Additional Tool Options

Some tools contain additional options. The presence of these options is indicated by an arrow in the large icon, left of the tool name. By default the additional tool palette will display just below the prompt. If the additional tool palette has been closed, it will remain closed until you click on the arrow located in the large image icon. Drag to the left to keep the palette open permanently.



Tool Hints

To the right of the tool name in the prompt window are Tool Hints. Tool Hints display additional information regarding the current command. Hints may contain tool options controlled by hitting the Control or Apple keys as well as tips for using the tool.

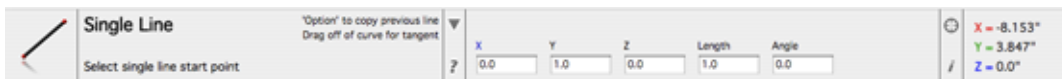


In the above prompt window, the tool hints provide the following additional information regarding the Selection tool:

- Pressing the Shift key will add or subtract entities to the selection.
- Pressing the Option key will copy selected objects if clicked and dragged.
- Selecting from right to left will select any entity that crosses the select box.

Data Entry Fields

Whenever you select a tool from the Tool palette, the Data Entry Fields appear in the center of the Prompt Window. The data entry fields display numerical values related to the current tool. For example, when the Single Line tool is selected, the Data Entry Fields contain values for the X, Y, and Z coordinates of the beginning point and the Length and Angle values for the line.



You can use the Data Entry Fields in three ways:

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

Whenever you construct an object, the data entry field containing the specification that you are most likely to change is preselected (the box is highlighted). For example, when you draw a line with the Single Line tool, the Length box is highlighted so you can enter a value for the length. You do not have to move your cursor to activate entries into the pre-selected field. Just type in a value from the keyboard and it will enter it into the highlighted entry field. Press the Return key to accept the entry.

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

- Press the Tab key to activate and cycle the selection highlight through the status boxes from left to right.
- Click inside the status box and the pointer allows you to edit text.
- Double-click inside the status box to select the entire contents of the box.
- Click the box label to select the entire contents of the box.

In addition, selected fields support cutting and pasting into and out of other fields.

Expression Parsing

The Data Entry Fields boxes also accept various mathematical, trigonometric, and exponential operators. Position the cursor in the text field and type in the additional operation. The expression parser handles decimals, integers, and fractions, all with units. Equation parsing is also supported in all dialog box fields that are looking for numeric inputs. The expression syntax is as follows:

```

expression:    value [binaryop value]
value:         [+|-]{literal | param | const | macro | [(expression)]}
binaryop:      {+ | - | * | / | ^ | % }
literal:       {integer[unitspec] | float[unitspec] | fraction }
float:         {integer{.,}integer | {.,}integer}[{e|E}[+|-]integer]
fraction:     [integer{unitspec} [integer{unitspec} ...][simplefrac]
simplefrac:    [integer][ integer/integer[unitspec]]
unitspec:     {y | yd | yds | yards | f | ft | feet | ' | i | in | inch | inchs | inches | " | millimeters |
millimeter | mm | centimeters | centimeter | cm | decimeters | decimeter | dm |
meters | meter | m | pts | pt | p}
param:        alphas[integer]
const:        {pi}
  
```

<i>macro:</i>	<i>macroname(expression)</i>
<i>macroname:</i>	{ sin cos tan asin acos atan sinr cosr tanr asinr acosr atanr abs ceiling ceil dtor exp floor factorial fact log ln neg round rtod rnd sqrt truncate trunc sqr cube odd even sign }
<i>integer:</i>	{0..9}{{0..9}...}
<i>alphas:</i>	{a..z A..Z}{{a..z A..Z}...}
where,	
[X]	Optional component where X is a list of components separated by a “ ” char
{X}	Mandatory component where X is a list of components separated by a “ ” char
X...	X is repeated any number of times
X..Y	Component shorthand for all ASCII chars from X to Y inclusive

Examples of supported expressions are listed below:

<u>Operator</u>	<u>Example</u>	<u>Operator</u>	<u>Example</u>
Addition	3+.450	Arctangent	atan(1.0)
Subtraction	3-.500	Log Base 10	log(7.25)
Multiplication	3*.725	Natural Log	ln(8.5)
Division	3/5.25	Remove	truncate(6.125)
Square Root	sqrt(8.75)	Absolute Value	abs(-47+16)
Parenthetical	3/(5*2/4)	Smallest Larger	ceiling(5.25)
Scientific	4e-3	Largest Smaller	floor(12.75)
Exponentiation	exp(2.7)	Negative Value	neg(1.12)
Sine of Angle	sin(15)	Round	round(1.12)
Cosine of Angle	cos(30)	Fraction	1 1/2 + 3/4

PrecisionPoint

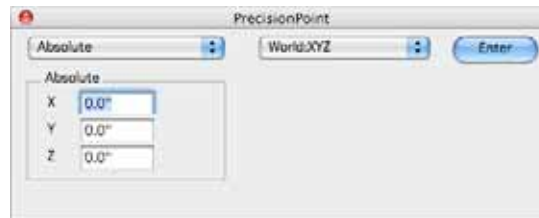
The PrecisionPoint icon appears to the right of the Data Entry fields. The PrecisionPoint tool provides a means to rapidly enter explicit data to create and modify geometry. Options include data entry using absolute XYZ coordinates, relative coordinates, distance along entity, and percent along entity. This is a helpful option for creating entries according to specific coordinates or moving existing entities in certain positions.

Methods

- World:XYZ
- Work Plane:XY
- Polar
- Relative Polar

Absolute

Coordinates that are entered are relative to the origin (0,0).



Using the Absolute option, you can specify the coordinate system:

- World XYZ - Based on XYZ coordinates.
- Work Plane XY - Based on XY coordinates.
- Polar - The entity's coordinates are based on a specified distance and angle, from the origin.

Relative

Coordinates that are entered are relative to the last point picked. The last point data will be displayed, and you have the option to specify different coordinates by clicking at a new position or entering new values.

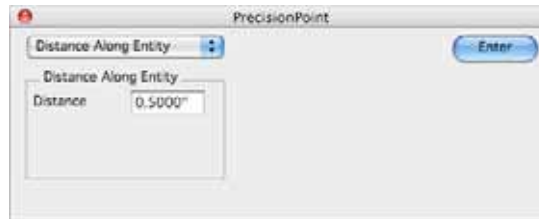


Using the Relative option, you can specify the coordinate system:

- World XYZ - Based on XYZ coordinates.
- Work Plane XY - Based on XY coordinates.
- Polar - The entity's coordinates are based on a specified distance and angle, from the last point.
- Relative Polar - Specify the two points from which you want an entity placed, then enter the distance and angle.

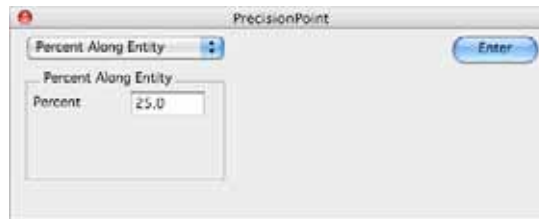
Distance Along Entity

Allows you to specify the distance from an existing point that an entity is placed. As you move your cursor along an existing entity, it snaps to the distance that has been specified.



Percent Along Entity

Allows you to specify at what percentage of an existing entity to draw or place a point. As you move your cursor along an existing entity, it snaps to the percentage that has been specified.

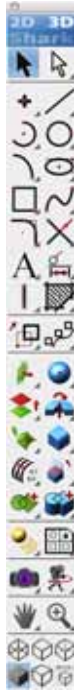


Inspector

The Inspector icon activates the Inspector. For information regarding the Inspector, see the chapter titled "Inspector", which begins on page 35.









Main Tool Palette











The start up screen displays the Main Tool Palette in the upper left corner of the window. This tool palette is your primary interface to the wide variety of commands available in the application. These commands include wireframe, surface, solid, and tools to modify. From left to right, top to bottom, an overview description of these tools follows.













Main Tool Palette

General Purpose/Wireframe Tools







-  **Select** – Select an object to modify.
-  **Deep Select** – Select and modify objects within groups or control vertices.
-  **Point** – Tools for creating points.
-  **Line** – Tools for creating lines.
-  **Arc** – Tools for creating arcs.
-  **Circle** – Tools for creating circles.
-  **Conic** – Tools for creating conics.
-  **Ellipse** – Tools for creating ellipses.

-  **Polygon** – Tools for creating polygons.
-  **Spline** – Create and modify spline entity tools.
-  **Fillet/Chamfer** – Fillet and chamfer curve tools.
-  **Trim** – Trim, relimit, divide, join curve utilities.
-  **Text** – Create horizontal, along curve, and at angle text.
-  **Dimension** – Tools for annotating your drawing with dimensions.
-  **2D Architectural** – Tools for drawing walls, windows, doors, and attached groups to walls.
-  **Region Hatch & Fill** – Tools to hatch and fill an inside region.
-  **Transforms** – Tools for translating, rotating, mirroring, scaling objects.
-  **Advanced Transforms** – Tools for transforming objects as arrays, paths, and connections.







Surface and Solid Modeling Tools

-  **Surface From Curves** – Creates surface from curves or position (planar).
-  **Solid Primitives** – Creates simple solids such as blocks, slabs, cones, and cylinders.
-  **Surface From Surface** – References other surfaces to creates surfaces.
-  **Solid From Profiles** – Creates solids from curve profiles.
-  **Surface Utilities** – General purpose surface utilities.
-  **Solid Features** – Creates feature based solids such as fillets and chamfers.
-  **Surface Local Operations** – Tools that operate on a surface.
-  **Solid Local Operations** – Tools that operate on a face.
-  **Advanced Surface Utilities** – Additional surface utilities such as adding or subtracting surfaces.
-  **Solid Utilities** – Additional solid utilities such as adding or subtracting solids.

More Tools

-  **Lights** – Creates light objects for rendering.
-  **Model To Sheet** – Makes 2D drawings from 3D objects.
-  **PhotoRender** – Tools for creating photorealistic renderings.
-  **Animation** – Fly by, walk through and object-based animations.
-  **Dynamic Zoom** – Dynamically zoom, rotate or pan a view.
-  **View** – Zoom up or down to specified user area.

Display Toggles

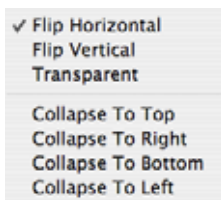
-  **Wireframe** – Displays model in wireframe line mode.
-  **Hidden** – Displays model in hidden line mode.
-  **Hidden Dim** – Displays hidden lines lighter than visible lines.
-  **Shaded** – Displays model in Gouraud or Phong OpenGL.
-  **Perspective** – Turns perspective on or off.
-  **View Props** – Displays a dialog that allows you to change specific settings for the current view.

Floating Palettes

This software product uses floating tool palettes. Floating tool palettes contain options that you frequently use to create drawings and designs. To select an option in a floating tool palette simply press and release the mouse over the option.



If you Control-click (right-click) in the title bar of a floating tool palette you will get a menu of options that further customize the floating tool palette.



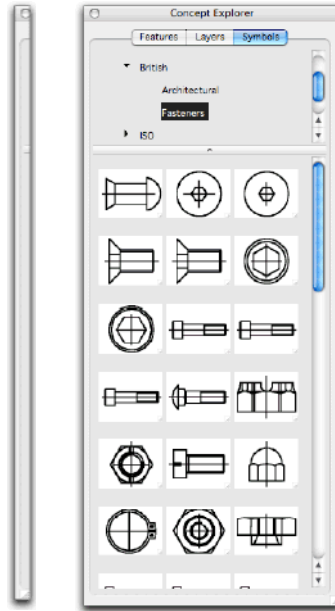
Flip Horizontal and *Flip Vertical* will change the tool palette between a left to right palette and a top to bottom palette.

Transparent hides the lines of the palette so only the tools appear. This feature is available for Mac only.



Collapse to Top, *Collapse to Right*, *Collapse to Bottom* or *Collapse to Left* will collapse the palette when the cursor is not over the tool palette.

Not only does collapsing work with tool palettes, but you can use it with other dialogs as well, including Concept Explorer. Auto collapse and expand is a great way to efficiently use screen area.



Tools with a right pointing arrow (located at the bottom right corner of the tool icon) indicate that a sub palette exists for that tool option. Sub palettes are a collection of tools that are similar in function. A sub palette can be made visible by clicking on the arrow portion of the icon. A sub palette will disappear once you have made your selection and release the mouse button.

A sub palette can be made to stay permanently by dragging your mouse beyond the far right extents of the palette and “tearing” it away from its parent.

To remove any palettes from the screen simply press your mouse button in the upper left corner of the window.

Tear Away Menus

Many of the floating tool palettes have additional options. More options are available if the icon displays a right pointing arrow in the lower right hand corner of the icon. To access these options, press the mouse button on or near the arrow. As you continue to hold the mouse down, a sub tool palette will appear with additional options. Move the mouse to the desired sub tool palette option and release the mouse button. If you move the mouse beyond the right extents of the palette, the menu will “tear” away from the parent menu and stay permanently on the screen (or until you close the palette).

Tip: Select the *Shark FX:Preferences* command and click OK to save all of the currently displayed tear away menus and their associated screen locations. This will store the locations in the preference file so that you don’t have to relocate menus to your preference each time you start the application.

Tool Tips

As you move the cursor over any icon, a brief description of the corresponding command is displayed within the prompt window (overlays the x, y, z display). Tool tips can be turned on or off through the General Preferences setting, on the Shark FX menu.

Tip of the Day

The Tip of the Day dialog box displays useful tips, reminders, and hints about getting the most out of the application. The Tip of the Day dialog appears upon launching the application. Select the next button to display additional tips. Check the “Show tips at startup” button to have the tips dialog box display every time the program is launched. You can also show the tips dialog box from the Help menu.

The tip of the day dialog reads strings from the tips.ini file located in the Environ folder. You can customize the tips by modifying the contents of this file.

Mouse Button Functions

The mouse provides feedback to the software regarding a variety of functions. Although the three button mouse is optional, it is highly recommended. The functions are listed below:

Left Button

- Click to select objects
- Click to specify locations and xy coordinates
- Press space bar and click to dynamically pan the drawing

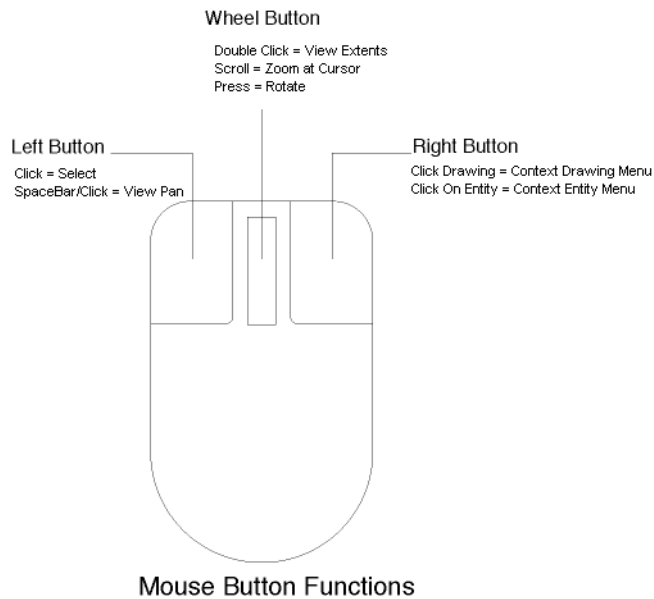
Right Button (Optional)

- Click to terminate sequence of points (splines & multi-lines)
- Click within drawing window for window context sensitive menu
- Click on entity for entity context sensitive menu

Note: For one button mouse, you can emulate the right button with a Control Left Button click

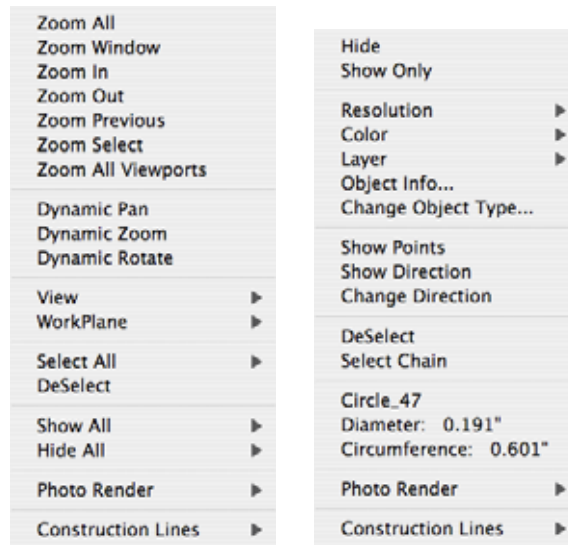
Wheel Button (Optional)

- Scroll forward and backward to zoom in/out at cursor location
- Double click to zoom extents
- Press and hold to rotate view orientation



Context Sensitive Menus

Right-clicking (Control-clicking) in the drawing area of the Application window will display the Drawing Context sensitive pop-up menu. This pop-up menu contains commands for zooming, panning, selecting, hiding, and creating construction lines. Right-clicking on an entity within the Application window will display the Entity Context sensitive pop-up menu. This menu contains commands for hiding, changing resolution, color, and layers. Additionally, it will show geometry data relative to the entity selected, such as lengths or diameters.

*Drawing Menu**Entity Menu**Context Sensitive Menus*

2D/3D Tool Palettes

The top of the main tool palette contains an icon that toggles the user interface between 2D and 3D. Clicking on the Shark icon to illuminate 3D will change all tool palettes, dialog boxes, and menus to include an expanded set of 3D functionality. Clicking on the Shark icon to illuminate 2D will likewise change all user interface components to be an exclusive 2D tool set.

You can toggle between 2D and 3D at any time. If you are in the 3D mode and have surfaces or solids, you can still Go to 2D without compromising your 3D data. Switching to 2D will temporarily hide your 3D data and automatically restore it when you decide to switch back to 3D.



2D/3D User Interface Palettes

Snaps

The intuitive Snap tool is one of the features that makes this application unique among computer aided design products. The Snap tool intelligently infers positions from geometry and alignments between geometry.

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

In addition to the precision allowed by Snap tools, a unique mechanism is enabled called the LogiCursor.

Snapping onto Geometry

When the pointer is in the drawing area, it has a snap point function. The snap point locks onto specific points on existing objects as you move the pointer near them. The Snap tool tells you when the snap point is **on** an object and if the snap is locked onto an object point such as *center*, *endpoint*, *intersection*, *midpoint*, *quadrant* or *vertex*.

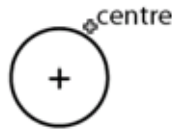
on

Projects the pointer location onto an object.



center

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



endpoint

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



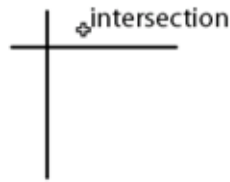
midpoint

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



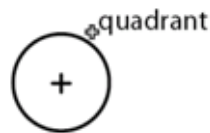
intersection

The intersection of two lines or curves (both permanent lines and the Snap tool's dynamic construction lines).



quadrant

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



vertex

The vertices of an ellipse, spline or dimension point.

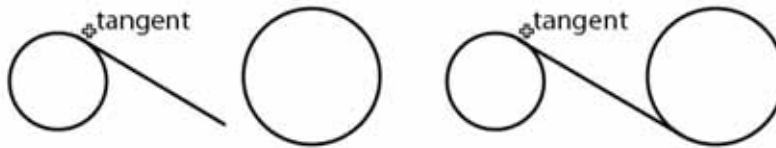


Using Tangents and Perpendiculars

With the Single Line tool enabled, if you click a point on an arc or circle and drag the pointer away at about a 45 degree angle, the Snap tool locks onto the tangent. If you drag away at a 90-degree angle the Snap tool locks onto a perpendicular.

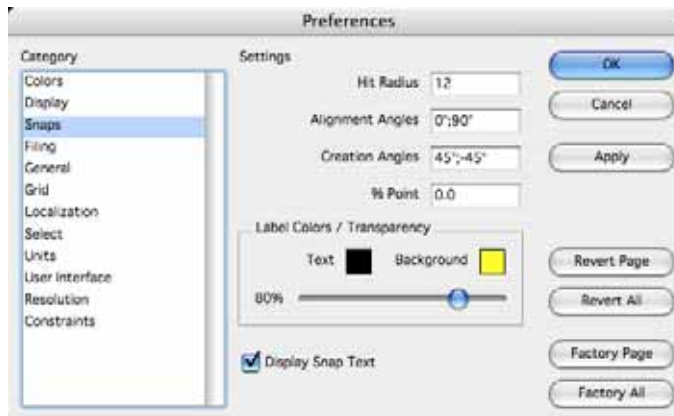


If you continue holding down the mouse button, the line remains tangent *or* perpendicular while you drag the ending point around the object. This is a very useful feature if, for example, you want to create a line from and tangent to an existing circle to the tangent point of another circle. Once a line is tangent to the circle, it can be dragged to the tangent point on the other circle, with the tangency maintained at both ends.



Snap Settings

Snap settings are changed in the Preferences option located in the File pull-down menu. The dialog box allows you to change the hit radius, alignment, angles, and other settings.



Hit Radius

This setting determines the detection distance in pixels. When the pointer is within the specified Hit Radius, the Snap tool notations are displayed and the object is selected when you click the mouse.

Alignment Angles

These angles define the dynamic construction lines that the Snap tool automatically uses. If you want to change the orientation of your drawing, you can change these specifications. For example, you could set these angles to 30, 90, and 15 for an isometric drawing. The defaults are 0 (horizontal) and 90 (vertical). Use a semicolon to separate the values. To display a temporary construction line through a point, move the pointer to the point to activate it (a diamond appears); construction lines automatically display through the active point. You can have as many as eight active points. When you activate the ninth point, the first one in the series is deactivated.

Additional Creation Angles

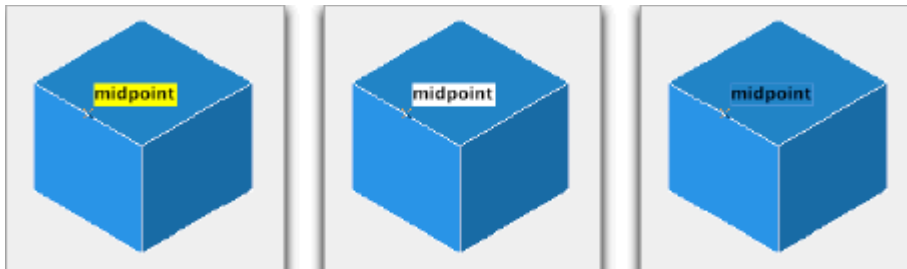
These lines are used by the Snap tool only when you are creating geometry and they are not part of the list of lines generated from the eight active points. The defaults are 45 and -45. Use a semicolon to separate the values.

% point

The divisions of a curve for Snap tool notations. If you want to divide the line into quarters, use a 25 specification. The default is 50, which shows the midpoint of lines. For example, entering 25 instructs the Snap tool to tell you when the pointer is 25% of the distance along a line.

Label Colors/Transparency (Mac Only)

The Preferences:Snaps dialog box contains settings for the user to control several attributes associated with snap text. These attributes include the Snap Text Color, Snap Text Background Color, and Background Transparency. The transparent slider set at 100% is fully see through to the drawing window.



(a) Yellow

(b) White

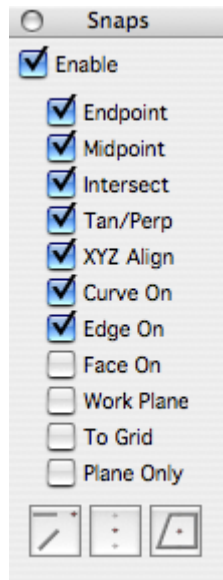
(c) Transparent

Example Snap Background Settings

Snap Filters

The Snap filters enable/disable specific snap checks. By default the surface and solid edge/face projections are turned off. The options remain on/off until changed again through the dialog box.

Selecting the Apparent Intersection Snap, Midpoint Snap, Centroid Snap or Relative Snap disables the other Snap options.



For more information on Snaps filters, see “Snaps” on page 159

Snap Locking

Pressing the x, y or z key will lock a snap along an axis alignment. Snap locking takes the current cursor location and projects to the x, y or z alignment axis. Once you press a lock key, additional wake ups are ignored, fixing the lock to the alignment at the time the key was hit. Snap locking does allow you to reference other points that are then precisely projected unto the alignment axis.

LogiCursor

The LogiCursor allows you to align the cursor with an existing entity, for precise placement. Unlike snaps, which distinguish point on a surface, the LogiCursor displays intersections along an axis, tangent, perpendicular, and 45-degree angle.

Object Selection & Editing

The Select or Arrow tool is used to select and edit objects. Typically, commands in the Menu Bar require you first select an object before picking the command. In addition, the Select tool provides a means to edit an object either through grips or by parameter editing. This chapter covers the following topics:

- Select Metaphors
- Single Pick and Box Pick Selections
- Select Markers
- Select Masking and Ambiguity
- Grip Editing
- Parameter Editing
- Deep Select

Select Metaphors

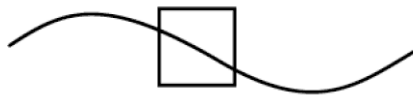
Punch! Shark FX utilizes two techniques for selecting objects. The first technique uses a noun/verb metaphor. It's convenient to think of the noun/verb metaphor as selecting the object first and then some action to perform on the object. For example, you can select an object, change the color, edit object parameters, and copy/paste it to a new location, all by having only selected the object once. All commands in the Punch! Shark FX Menu Bar support the noun/verb metaphor.

The second technique uses the reverse, a verb/noun method. With verb/noun you select the command and then the object upon which the action is performed. For example, to trim a collection of curves to another, you select the trim command, then identify the curves to trim and finally the curve to trim to. Commands in the tool palette typically support either metaphor.

Single Pick Select

The single pick box is an “invisible” box centered about the cursor tip. The default size of the box is 8 by 8 pixels. This can be changed by the user in the *File:Preferences:Select* dialog box (*Shark FX:Preferences* for Mac). The pick box requires that the user move the cursor over the desired object and press and release the mouse button. The mouse is in the same position for press and release mouse clicks. Objects that pass through the pick box are selected.

Invisible Pick Box Extents



Pick Window Select

The pick window is a rubber banding marquee window that selects objects that are contained within the window. To use a pick window, move your mouse to the upper left corner of the object(s) you want to select, press and hold the mouse button down, then drag to the lower right corner and release the button. There are two methods for determining if an object is to be selected. The first and default method requires the entire object be contained within the box window. The second method requires only that a portion of the object be contained. Use the *File:Preferences:Select* dialog to set this behavior. Objects can be added or removed by holding down the Shift key while clicking on objects.

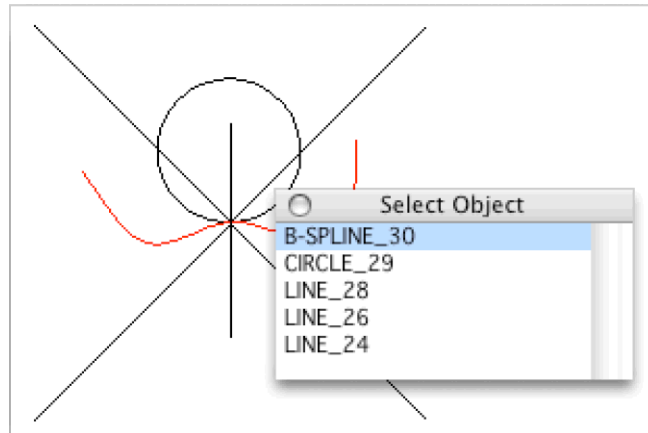
Press and Hold



Drag to and release

Select Ambiguity

The single pick select method assumes you want to select one object. If you ever select more than one object with the pick box select method, a pop-up window with a list of all the possible select objects displays to the screen. As you move the cursor over the object name in the pop-up, the color of the object changes to indicate which object will be selected. If you move the mouse beyond the extents of the pop-up window, the window will automatically move, enabling you to see what is underneath the window.



You can disable the ambiguity pop-up menu through an option located in the *Shark FX:Preferences:Select* dialog box. (*File:Preferences:Select* for PC).

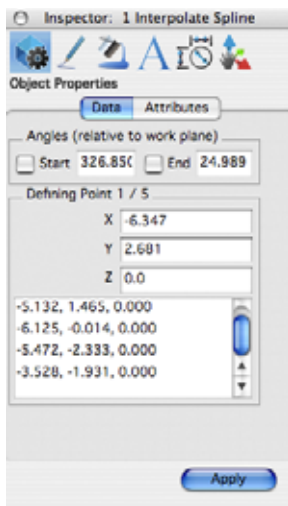
Show Point Editing

The arrow icon located in the upper right location of the main tool palette is used to dynamically modify the shape and position of an object. Select the object you wish to modify by pressing the mouse button on the object. Then hold the mouse button down while you move it to a new location. Selecting a show point will modify just that point. Selecting anywhere else on the curve will drag the entire object to a new location.

To display an object's show points, select the object and pick the *Edit:Show* command located in the Menu Bar.

Object Parameter Editing

Another useful technique for modifying an object is to double click on an object while in the Select tool (arrow icon in the main tool palette). Double clicking will bring up a dialog box of the object's data and attributes that are changeable.



The Inspector provides a means to change geometry parameters, as well as display settings, general attributes, and photo rendering material settings. For more information, see “Inspector”, which begins on page 35.

Editing and Intelligent Links

Many of the objects in Punch! Shark FX are associative. Associative objects retain a history of how they were created and a set of rules that define their geometric description. If you modify an object that defines another object (called a parent object) the dependent object (called a child) will automatically regenerate. For example, consider a skinned surface that is defined by two lines. If you modify one of the lines either via points or its parameters, the skinned surface will automatically regenerate.

Deep Select

The Deep Select tool provides a means to select objects and points that are nested deeper into the object definition. For example, use deep select if you need to modify an object that is part of a group. Deep select will also automatically turn on control points that can be used to alter the shape of an object. When you are done, these points are turned back off.

Partial Select

You can select an object by selecting only a portion of it from right to left. This functionality also includes support for surfaces and solids.

Tape Measure Tool

The Tape Measure tool identifies distance, distance components, and the angle relative to the workplane. The distance and angle values are displayed at the cursor location for quick reference.

Using the Tape Measure tool

- 1 From the Deep Select tool palette, click the Tape Measure tool.
- 2 Click to select the first point.
- 3 Drag and click to select the end point.

The distance and angle are displayed at the cursor location. The dX, dY, and dZ distance components are displayed on the Data Entry window.



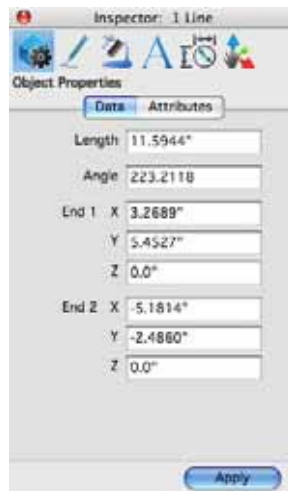
Inspector

The Inspector dialog displays a variety of general information regarding the current set of selected object(s), including Object Properties, Pen Properties, Fill Properties, Text Properties, Dimension Properties, and Gripper Properties.

Object Properties

Data Page

The Data Page displays values associated with an object's form and shape. Typical field values include positions, lengths, widths, heights, and diameters.



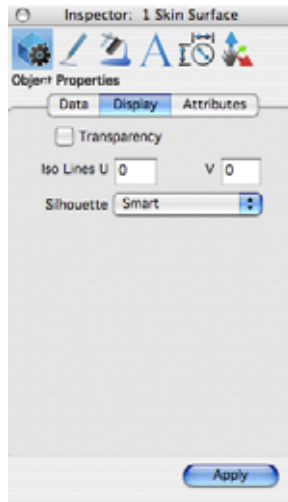
Attributes Page

The Attributes Page controls miscellaneous settings associated with the object. These include settings for resolutions (vector and triangle), line fonts & patterns, arrowheads, and user-defined names.



Display Page

The Display Page appears only for surfaces and solids. This page controls attributes with how the object is displayed to the screen. These include settings for transparency, iso lines, and silhouette edge display.



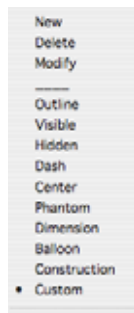
Pen Properties

The Pen Properties control the styles, colors, weights, and patterns of the text created using the Text tools.



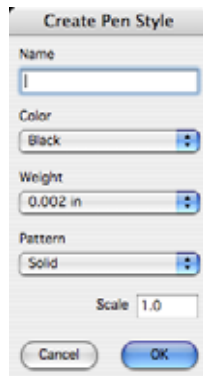
Style

A Pen Style is a collection of pen attributes such as color, weight, and pattern. A style is a means of preserving and recalling frequently used collections of these attributes. In addition, pen styles are associative. This means that after you create and assign pen styles to an object, you can go back and modify the pen style and have all objects using that pen style automatically update.



The top three menu options provide tools to create, delete, and modify user-defined pen styles. The proceeding pen styles are unmodifiable pen styles. Styles listed after those are user-defined pen styles.

The New... option creates a new user-defined pen style. Specify the name, color, weight, pattern, and pattern scale to define a new pen style.



Create Style Dialog

The Delete option allows you to remove a user-defined pen style.

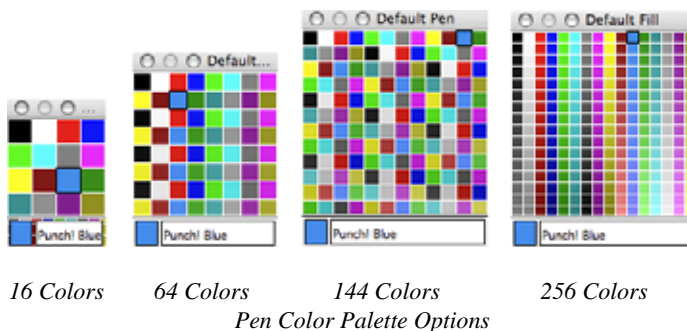
The Modify option provides a means to change a user-defined pen style attribute. Once again, the default, predefined pen styles are not modifiable; only user-defined pen styles can be modified.

Color

The Pen Color command specifies the object color of an entity. In the case of a surface or solid, the color defines the facet color for shaded display modes. The first eight entries provide selection to predefined colors.



The last option displays a tool palette with color swatches. Clicking in the upper right corner of the title bar will toggle the color palette into different versions. Clicking in the pen color palette will change all selected objects immediately to the selected color.



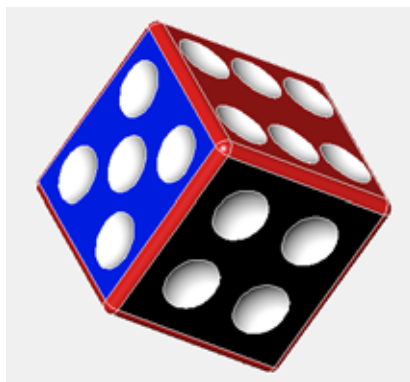
The RGB (**R**ed, **G**reen, and **B**lue) values and names of the colors used by the application are stored in the `rgb.txt` file located in the `Environ` folder.

Apply Color Per Face

You can apply different colors to each face of an object. Previously, a color would be applied to each side of an object.

Applying color per face

- 1 Click the Select Deep tool.
- 2 Select the face you want to color.
- 3 From the Pen Properties, choose the color you want from the Colors drop-down.
- 4 Continue to select faces and colors to apply different colors.



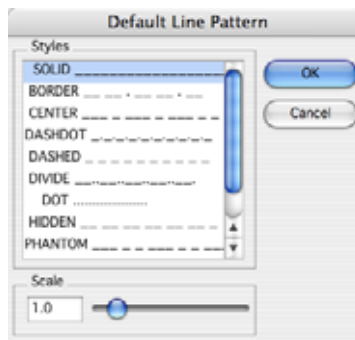
Pattern

The Pen Pattern menu allows specification of patterns to objects. The first eleven patterns are predefined.



Pen Pattern Menu

The last option displays a dialog box that has more patterns and the ability to set a scale associated with a pen pattern.



Line Pattern Dialog

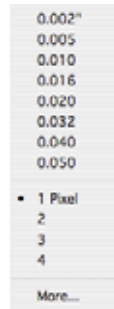
By default, Punch! Shark FX uses the descriptors located in the cadd.lin file located in the Environ folder. Users can manually add or modify line fonts with any text editor. Each style definition contains two lines in the cadd.lin file. The first line is the style name plus a descriptor string. The second line contains the length attributes for dashes, spaces, and dots. A zero value indicates a dot and a negative value indicates a blank space. The following are example definitions for the BORDER and CENTER line styles.

```
BORDER,_____ . _____ . _____ . _____ . _____ . _____  
A,.25,-.125,.25,-.125,0,-.125
```

```
CENTER,_____ . _____ . _____ . _____ . _____ . _____  
A,.75,-.125,.125,-.125
```

Weight

The Pen Weight menu allows specification of pen weights. The first eight pen weights are predefined thickness values. The next four are pixel-based thickness. The last option provides a means to specify a custom pen weight.



Arrow Start

The Arrow At Start command displays an arrow at the starting position of a curve. In the case of a line, it's the first point you clicked.



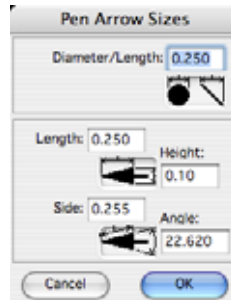
Arrow End

The Arrow At End command displays an arrow at the ending position of a curve. In the case of a line, it's the second point you clicked.



Arrow Size

This command controls the size of an arrowhead. The five fields are described below.



Diameter/Length Controls the diameter of circular arrowheads. Also controls the horizontal length of slash arrowheads.

Length Specifies the horizontal length of an arrowhead.

Height Sets the vertical height of an arrowhead.

Side Sets the length along the hypotenuse.

Angle Defines the angle of the arrowhead in degrees.

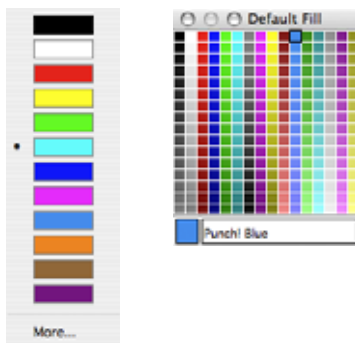
Fill Properties

The Fill Properties settings allow you to control the appearance of the fill that appears or does not appear, within an enclosed object.



Fill Color

This option specifies the fill color for smart polygons. Click More to display a tool palette with additional pattern fills. The pattern by which a color fills a space is controlled by the Fill Pattern option.



One Point Fill tool

You can fill a region with one mouse click using the One Point Fill tool. This tool supports auto-trimming and intelligent associativity to the neighboring untrimmed curves. Press the Shift key to select specific curves to be involved in the fill. The fill color and pattern are defined on the pen menu.

Using the One Point Fill tool

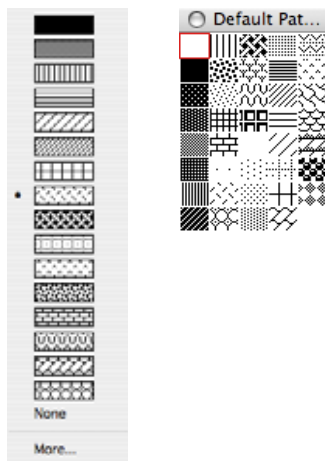
- Position your cursor in the center of the region you want to fill and click to place.



Note: Filling uses a bitmap to define the region. Use this tool if you have large regions to fill in a solid color in preference of the Hatch tool, which generates lines.

Fill Pattern

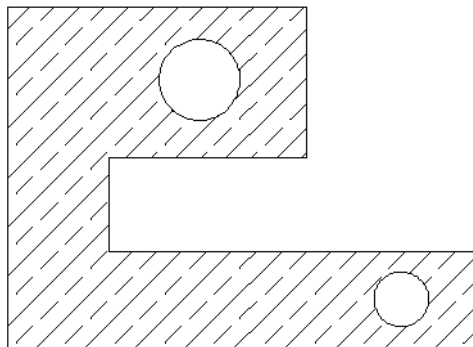
The Fill Pattern command specifies the pattern used for smart polygons. Unlike the Cross Hatch command, the fill pattern uses a bitmap fill. Click More to display a tool palette with additional pattern fills.



Fill Patterns

Hatch

The Hatch command applies hatch patterns to collections of closed 2D curves. The current hatch pattern is specified from the drop-down or by clicking the Options button when the Hatch tool is enabled.



To edit a hatch, select the hatch pattern, and then go to the Cross Hatch dialog. The Cross Hatch dialog box will fill the values with the selected hatch pattern.

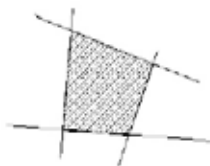


One Point Hatch tool

You can create a hatch with one mouse click using the One Point Hatch tool. This supports auto-trimming and intelligent associativity to the neighboring untrimmed curves. To change the default hatch pattern, click the Options button on the Data Entry Window and choose the hatch pattern you want on the Object Cross Hatch dialog box.

Using the One Point Hatch tool

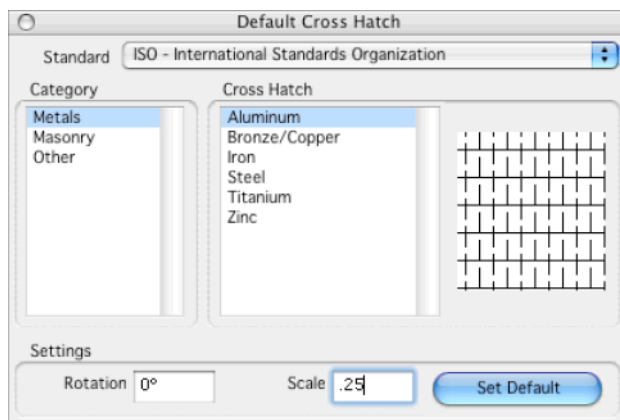
- Position your cursor in the center of the region where you want a hatch and click to place.



Note: Hatching uses a spacing and angle attribute to control scale and orientation. Hatches are comprised of individual line segments and recognizes interior holes.

Hatch Options

When the Hatch tool is active, the Options button appears on the Prompt toolbar. The default Cross Hatch dialog box allows you to set the current cross hatch pattern.



Cross Hatch Dialog

Standard	Specify ISO or DIN Standard patterns.
Category	The hatch categories for a given standard.
Cross Hatch	Patterns for the given category
Rotation	The rotation angle for the cross hatch pattern.
Scale	The scale value for the cross hatch pattern.
Set Default	Click this button to set the current standard, category, cross hatch pattern, and settings for newly created hatches.

Text Properties

The Text Properties provide a means for assigning text attributes to text entities and for setting the default attributes for newly created text. The text type, New or Selected, is displayed in parentheses.

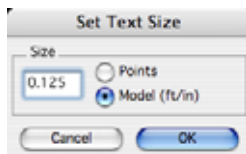


Font

The Font option displays a menu with a listing of all the font types available from the host system. Select the More option to display a floating tool palette that stays on the screen.

Size

The Size menu allows specification of text size in point and model values. Select the User option to display a dialog box for custom point and model values.



Style

The Style menu provides four options that impact text styles.

Align

Left - Changes the justification of box (paragraph) text to left aligned.

Center - Changes the justification of box (paragraph) text to center aligned.

Right - Changes the justification of box (paragraph) text to right aligned.

Case

Lower Case - Converts all selected text to lower case characters.

Upper Case - Converts all selected text to upper case characters.

Title Caps - Converts all selected text to upper case characters for the first letter of the word.

Show Only Used Fonts

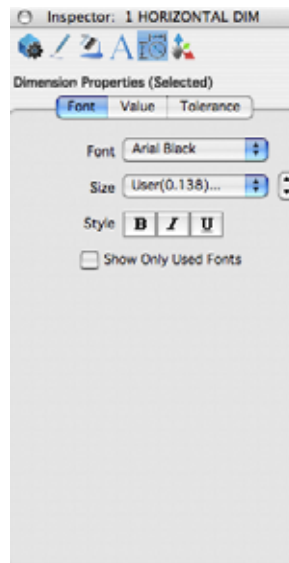
Select this option to limit the Font drop-down menu options to fonts that are actively used in your design.

Dimension Properties

Annotations are a critical part of creating any drawing. Dimensions, tolerances, and other annotations allow the user to clarify complicated parts, components or assemblies. Punch! Shark FX allows the user many powerful types of annotations, including linear and angular, dimensions and tolerances, balloons, witness lines, and center marks. This chapter presents descriptions of these powerful tools and how they integrate into your drawing.

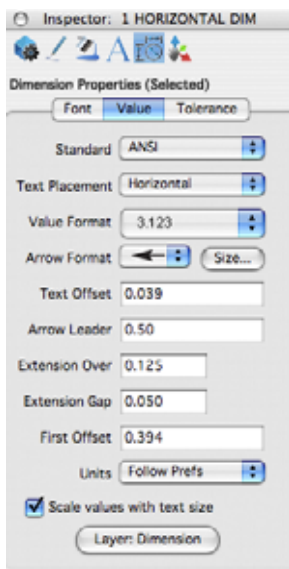
Dimension Text

The Dimension Text is controlled from the Font tab. You can change the text font, size, and style. You can limit the font selections to only those fonts used in your drawing; to do so, select the Show Only Used Fonts checkbox.



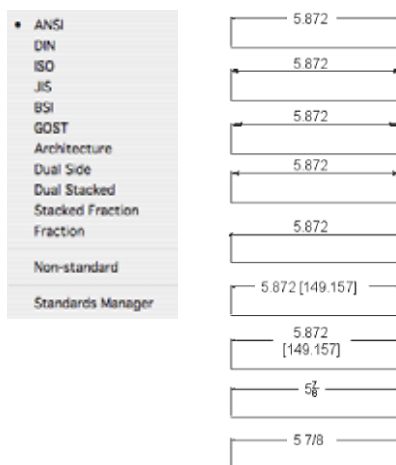
Dimension Values

The Values tab affects the actual value. Here you can change properties such as size, orientation, and position.



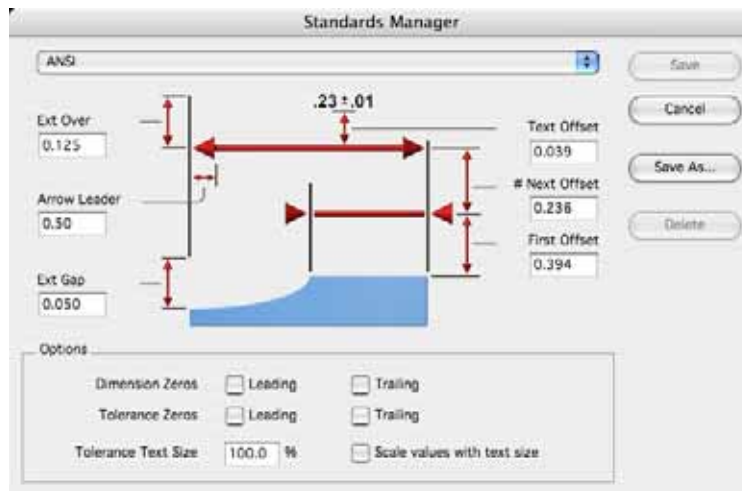
Standards Layouts

The Standards Layouts menu controls a collection of predefined attributes that define a particular standard. Example attributes include arrowheads, text placement, and additional spacing values.



Standards Manager

The Dimension Standards dialog box allows you to create custom standards for dimension settings.



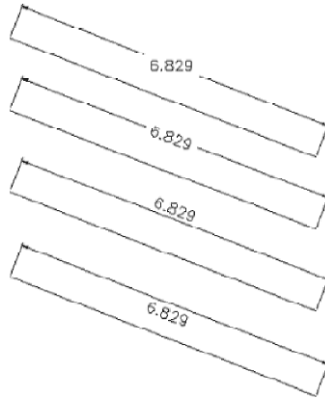
Standards	A list of predefined and user defined standards.
Text Offset	The text height above the leader line.
Next Offset	The distance between successive leader lines. This value is relevant for baseline dimensions.
First Offset	The distance between the object dimensioned and the leader line.
Arrow Leader	The distance for the arrow leader line. Used only when the arrows are placed outside.
Ext. Over	The distance above the leader line along the extension line.
Ext. Gap	The distance between the dimensioned point and the beginning of the extension line.
Dimension Zeros	Checkbox indicating whether to include leading and trailing zeros for the dimension value.
Tolerance Zeros	Checkbox indicating whether to include leading and trailing zeros for the tolerance values.
Tolerance Text Size	Tolerance size as a percent of the dimension text size.
Save	The save button will store all settings to the current user defined standard. Note: Save is available only for user defined standards. You cannot overwrite or change predefined standards.
Save As	Saves the existing settings to a new user defined standard.

Delete

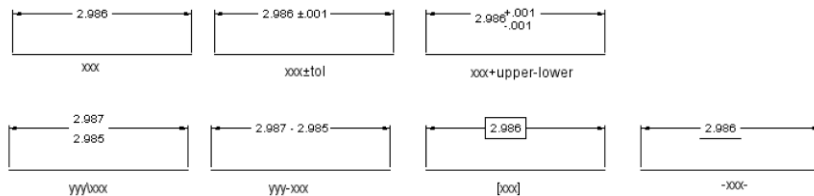
Deletes the current user standard for the standards drop-down list. Note: You can only delete user define standards. You cannot delete predefined standards.

Text Placement

The Text Placement options apply to both linear and angular dimensions. The four options, Horizontal, Break-In, Over, and Under placements control the placement of the dimension text relative to the leader line.

**Value Format**

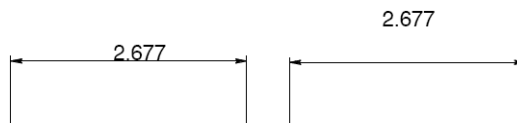
The seven options in the Format menu control the Value Format of the dimension text. These formats apply to both linear and angular dimensions. The value format is included within the text portion of the dimension.

**Arrow Format**

Choose the endpoint format and size for dimensions. For more information on changing the size, see “Arrow Size” on page 42.

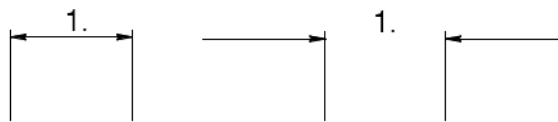
Text Offset

Used to control the distance between the text and the extension line for over or under text formatted dimensions.



Arrow Leader

Specifies the minimum length for a leader line in the dimension. If there is not enough room, the arrows are automatically flipped to the outside.



Extension Over

Specifies the distance that the extension goes about the leader line.



Extension Gap

The distance between the dimensioned point and the beginning of the extension line.

First Offset

Specifies the leader line distance above the measured points.

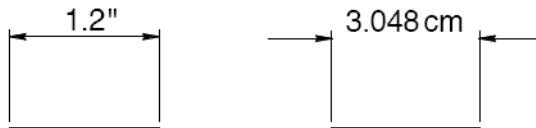


Units

The units pull down provides a means to mix units in a drawing.

- Follow Prefs
 - inches
 - feet
 - feet/inches
 - millimeters
 - centimeters
 - meters

Specifying units allows you to use various units of measure throughout a single design.



Scale Values with Text Size

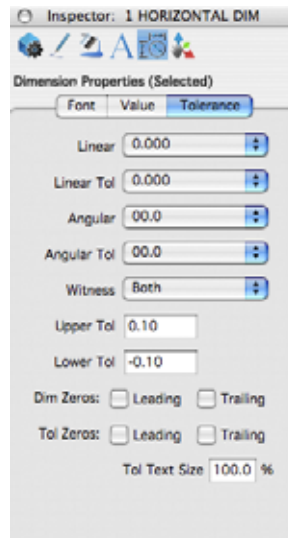
This checkbox will enable automatic scaling of all dimensions' values (arrow head sizes, offsets, leader lengths) when the text size is changed.

Layer Creation

The Create Layer menu item sets the layer that dimensions are initially assigned into. Use the *Edit:Change Layer* to assign a new layer to a dimension.

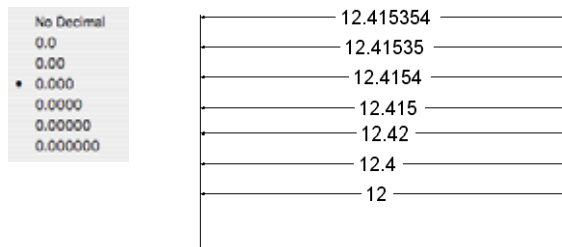
Dimension Tolerance

The Tolerance tab affects the limitations that are applied to the actual dimension value. Setting the Tolerance properties can help maintain realistic or consistent dimension values and formats throughout a design.



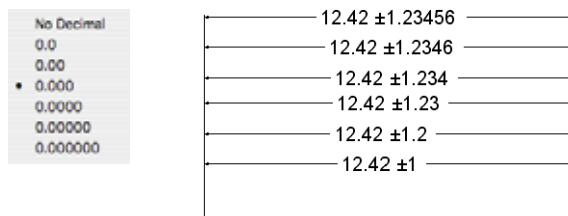
Linear

The Linear menu provides a means to specify the number of decimals for the linear value of the dimension.



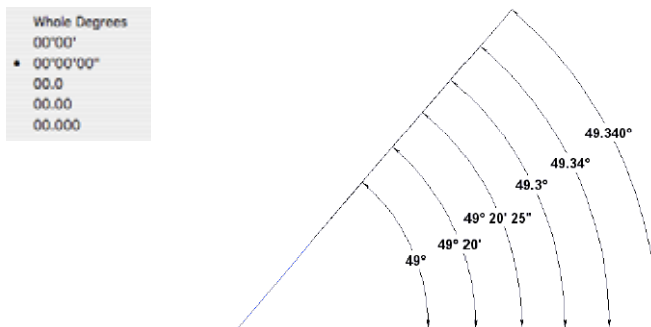
Linear Tolerance

The Linear Tolerance value controls the number of decimals used in the tolerance portion of a dimension.



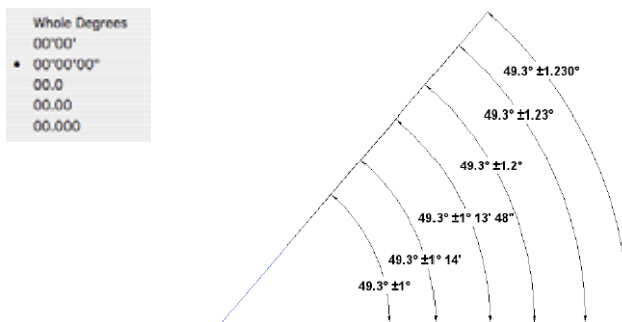
Angular

The Angular menu controls the display format for angular dimension value. Format options are available for degree/minute/second formats as well as decimal versions.



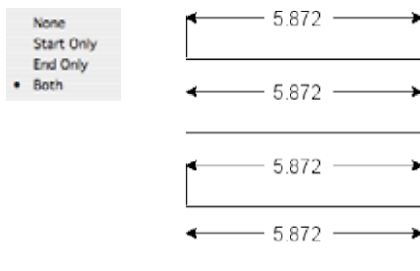
Angular Tolerance

The Angular Tolerance menu controls the display format for the tolerance portion of the angular dimension.



Witness

This option turns on or off the Witness Line for a dimension.



Upper Tol

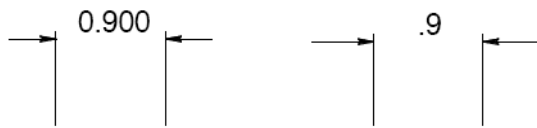
Defines the position of the upper-level tolerance value, if the applicable value format is selected.

Lower Tol

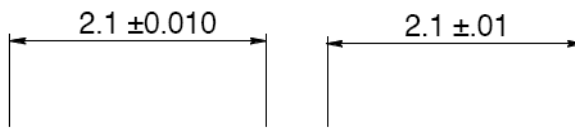
Defines the position of the lower-level tolerance value, if the applicable value format is selected.

Dim Zeros

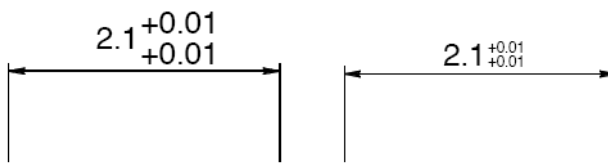
Specifies whether zeros will be removed if they are leading or trailing in the dimension text.

**Tol Zeros**

Specifies whether zeros will be removed if they are leading or trailing in the tolerance text.

**Tol Text Size**

Specifies the size of the tolerance text as a percent of the dimension text.

**Gripper Properties**

The 3D Gripper is an intuitive tool for transforming objects, including moving, rotating, and resizing. When the gripper is enabled, the Gripper Properties allow you to control its different functions and abilities.



Enable Gripper

This checkbox enables and disables the gripper.

Attach

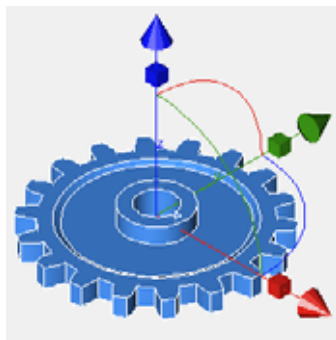
The drop-down allows you to choose whether to attach the gripper to a selected object or to a user-specified position.

Size

The drop-down offers four sizing options that control the appearance of the gripper.

Show Handles

There are four handles you can enable (or disable) to help transform an entity just the way you want.



Translation	The Translation handles move a selection along the specified axis. The Translation handles are displayed as cones at the end of each gripper axis.
Rotation	The Rotate handles rotate a selection. When enabled, an arc is visible at each axis in the color of the axis around which it rotates.
Scale	Controls the size of an entity along a particular axis. The Scale handles are displayed as cubes along each axis. Additionally, if the Uniform Scaling checkbox is not selected then the entity is controlled separately along each axis; if it is selected, each axis will follow the same scale.
Planar	Displays a transparent plane along each axis. When activated, the Planar handles restrict movement along the selected plane. This is useful for exact, flush positioning.



Show Planar Handles

Uniform Scaling

When the Scale handles are enabled, selecting the Uniform Scaling checkbox unites each axis so a change to the scale of one axis affects each axis the same, so the entity is resized together.

Gripper Positioning

Reset Axis	Repositions the gripper to 0,0.
Axis to Object	Allows you to choose the object with which to align the axis.
Set Origin	Allows you to select a point for the origin.
Object to Axis	Select an object to align with the axis.
Flip Axis	Rotates the axis 90-degrees around the Z-axis.

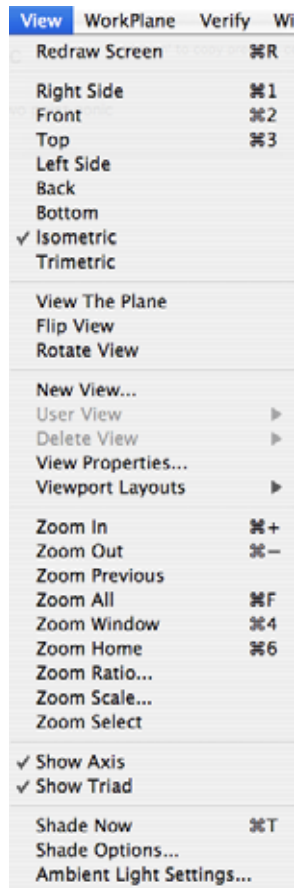
Controlling the Display

The display controls how three-dimensional objects are transformed to a two dimensional screen. Two important features for controlling the screen are the view and zoom tools. A view describes the orientation of your eye position and direction towards the model. Example view operations include the side, front, top, and isometric options. Zooms describe a window looking into the model defining the scale and clip extents. Example zoom operations are pan, extents, and scale. This chapter introduces you to the following display functions:

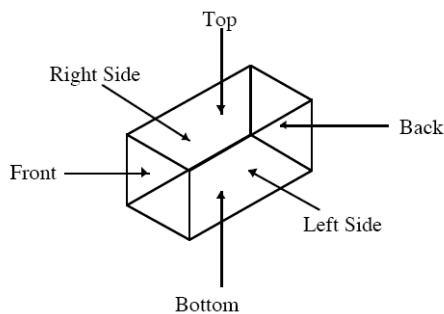
- Predefined Views
- Zoom Options
- View Tool Palette
- Predefined View Short Cuts

Predefined Views

Punch! Shark FX supports eight predefined views. You can change to a predefined view by accessing the *View* option from the Menu Bar on top of the screen.



Six of the predefined views locate a viewer such that they are looking parallel to the model axis. The other two views allow isometric viewing from the front and back of the model.

*Predefined Views*

Zoom Options

Zooming is a view operation used to increase or decrease the view scale of your drawing. Punch! Shark FX supports nine zoom view commands. Zoom commands include zoom in, out, previous, all, window, home, ratio, scale, select. For a full description of these commands see the documentation for Menu Bar commands.

Predefined View Short Cuts

The default configuration of Punch! Shark FX has the following short cuts defined for viewing operations. To change these settings, go to the Short Cuts dialog box located under the File option in the Menu Bar.

View Orientation

- a Right Side View
- s Front View
- d Top View
- f Isometric View
- g Trimetric View

Zooming

- e Zoom All
-] Zoom In
- [Zoom Out
- w Zoom To Window

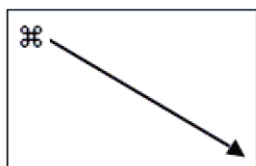
Predefined Mouse Operations

Punch! Shark FX has four predefined view operations associated with the mouse buttons.

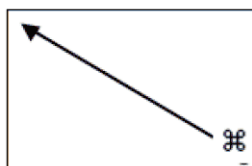
Mouse Button	View Operation
Space/Left Button	Pan Views
Wheel Button	Zoom In/Out at Cursor
Control/Apple Left Button	Rotate View via Cursor (Mac)
Alt/Right Button	Rotate View via Cursor (PC)

Some of the new mice will allow you to assign keys to the mouse buttons and the scroll wheel. If you have a mouse that does this, you can assign a key (i.e. “z”) to the mouse button, then within Punch! Shark FX assign the key (i.e. “z”) to one of the many viewing operations in the *File:Short Cuts* dialog box.

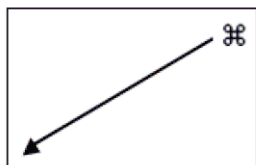
You can also perform in-stream zoom commands by pressing either the apple key (Mac) or control-shift (PC).



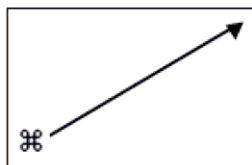
Zoom In



Zoom Previous



Zoom Out



Zoom Previous

In-stream Zoom Commands

Part 1 - Exercises

Using the Data Entry Window

Once you have drawn an object, you can adjust the geometry attributes such as length, angle, and location with the Data Entry Window. You must make these changes immediately, before you construct another object, select a different tool or choose a command.

Try the following exercises to create and change a single line with the Data Entry Window.

Altering geometry in progress with data entry boxes

- 1 Select the Single Line tool.
- 2 Click 2 locations in the drawing area. The length (L) box is automatically highlighted in the Data Entry Window.
- 3 Type 3. The 3 is entered directly in the L box.
- 4 Press the Tab key to select the next data entry box. The angle (A) box is now highlighted.
- 5 Enter 15.
- 6 Press the Return key. Pressing the Return key completes the data entry for this object. The line is redrawn 3 units long and at a 15-degree angle.

Creating additional geometry with data entry boxes

- 1 With the Single Line tool still selected from the previous example, click 2 more points. The application constructs the line.
- 2 Type 4.
- 3 Press the Tab key and type 25.
- 4 Press Return, another new line is drawn.

Creating new geometry with data entry boxes

- 1 Select the Selection tool so the Single Line data entry box clears.
- 2 Click the Single Line tool again. The X data entry box is active, showing that you can enter a value for the X coordinate of the beginning point of the line.
- 3 Type 0. Note: Do not press Return until you reach Step #8.
- 4 Press the Tab key. The Y box is highlighted.
- 5 Type 0.
- 6 Press the Tab key. The Length box is highlighted.
- 7 Type 45.
- 8 Press the Return key. The 10 units long line at 45 degrees is created.

Creating geometry offset from a point

If you want to create some geometry that is offset from an existing point, you can use the Data Entry Fields to specify the offset.

- 1 Select the tool you want to use.
- 2 Move the pointer over the control point from which you want the offset.
- 3 Click once to lock onto that point.
- 4 Click in the appropriate X or Y box in the Data Entry Window, placing the text cursor at the end of the entry.
- 5 Type the offset (such as + 3") and press Return.
- 6 Continue with your construction.

Part 2

Menu Commands

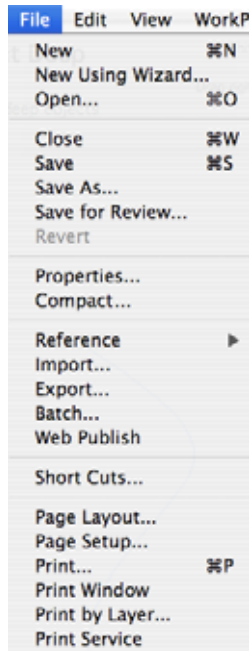
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File

When launching Punch! Shark FX from an icon, an empty drawing file called “untitled” is created. You can immediately start creating and modifying objects in your untitled drawing or you may load a preexisting drawing. Saving and retrieving files are performed with the File command located in the Menu Bar. This chapter covers the saving, retrieving, and other commands located in the File Menu.

File Menu

The File menu lists commands used to read/write files to disk, print, and the command to quit or exit Punch! Shark FX. A description of each option follows:



New

The new command will create and initialize a blank drawing. The default drawing title is labeled “untitled”. Previously created drawings will stay open until specifically closed or until the designer quits the program.

New Using Wizard

The New Using Wizard dialog box provides the ability to create a new drawing file from a collection of custom settings or selection of an existing file from a template. Click the default button to use the same settings used in the New command.

2D Template

Templates are existing files with predefined drawing and title blocks. A variety are provided to handle the major drawing formats. You can add additional templates to the drop-down menu by adding files to the Templates/2D folder, located in the Applications folder.



Custom Drawing

The Custom Drawing option allows you to specify a variety of settings used to set up the drawing. These settings include Unit Format, Printer Paper Size, Drawing Size, Drawing Scale, and Annotation Settings.



Unit Format

Specifies the unit type for the drawing from a drop-down menu consisting of metric and imperial unit formats.

Printer Paper Size

Select the page orientation or paper size from the associated drop-down menus. If the size is larger than the available printer size, tiled printing will automatically turn on. The width and height of the page are displayed as a reference.

Select the Show Page Breaks in Drawing Window option to distinguish the page boundaries on the drawing window.

Drawing Scale

Select a drawing scale from either engineering, architectural or metric. Select the Custom option to define unique scales relating printer page output with the actual world size. These fields support expression parsing, which allows you to put in unit identifiers.

Examples include:

1" = 1'

1cm = 1m

3/4" = 1'

Annotation Settings

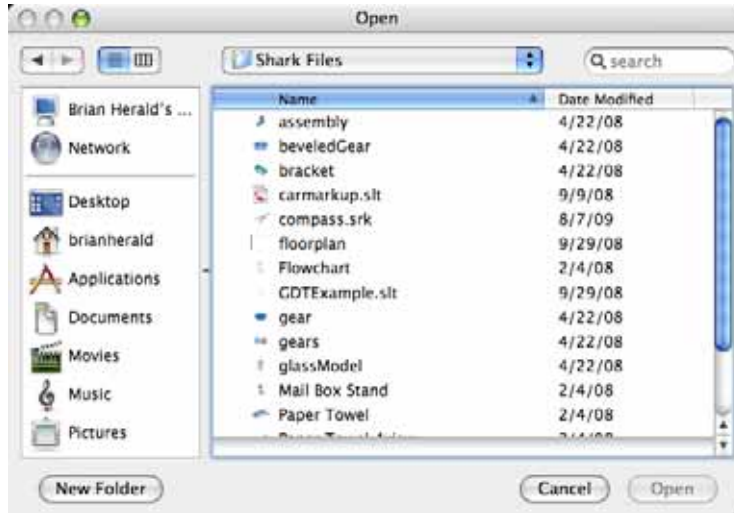
Specify text height as a function of world size or printer size. The relation between the two is defined by the drawing scale. If you enter a text height defined by the printer page, the actual world size will scale accordingly.

Specify the dimension format used in the drawing from a variety of standard settings.

The Scale Dimension Values with Text Height checkbox adjusts the dimension properties such as arrow head size, extensions, gaps, and leaders based on the change in the dimension text size.

Open

To read an existing file use the Open command. The Open command will display a dialog box prompting the user to select the file format and name.



Double clicking on a document will open that document and display it to the Punch! Shark FX screen. The default File Format is Punch! Shark FX, which reads in a compact binary version of your file that only Punch! Shark FX can read. To read in other formats, click the mouse over the file format option for a menu of additional options. Options available for reading are:

- Punch! Shark FX Modeling Files (*.vc3)
- All Files (*.*)

Close

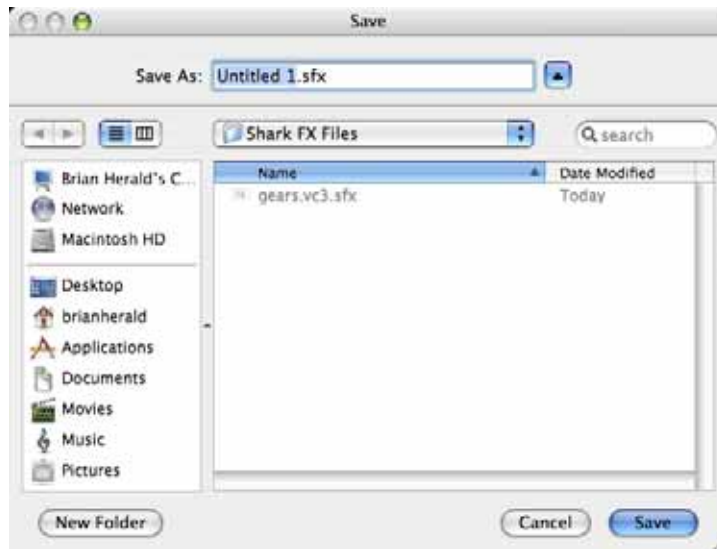
This command will close the current file. If other files are open, they are left open. If the file has changed since opening, the user will be asked if they would like to save the existing file.

Save

The Save command will write the file into the most recently saved file. No dialog will appear, as it uses the previous save information for file name and location. If you need to save the file as a new name use the Save As command.

Save As

The Save As command will write the objects into a file specified by the user. A dialog box will appear that allows the specification of file name and folder location.



Save for Review

The Save for Review command saves the current file to a new file name while locking the layer list. Locking the layer list prevents anyone from making changes to the file without first unlocking the layers. Use the Save for Review command in combination with the Markup tools.

Revert

The Revert command will restore the file to the last saved state. This command deletes all changes made in the drawing since the file open and is not undoable.

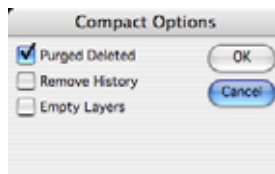
Properties

The File Properties dialog allows you to save information along with the file. By entering information in the text fields, you can identify the file with a title, subject, author name, keywords, and other important information about the file.



Compact

The *File:Compact* option allows you to remove temporary objects from the file, remove history, and empty the layers, compacting a file to its smallest size.



For more information on reducing file sizes, see “Filing” on page 91.

Reference Files

Reference Files are external files embedded into a drawing while maintaining a link to the original file enabling updates. Advantages of Reference Files are:

- Smaller file sizes
- Provides a means for many users to work on individual components of a large project
- Reference Files can be updated, added, edited at any time

Tools for creating and managing Reference Files are located under the File. The submenu has four options.

Attach

Specify a file to be referenced. This is the first step in linking another drawing to your current drawing. Any file type supported by the application can be attached, including DWG, SAT, STEP, PDF, ViaCAD or Shark files.

Reference Manager

A dialog box providing additional tools to manage references. The Reference Manager dialog provides tools for updating, detaching, replacing, and editing reference files.



Update - Updates the selected reference file. This will reload the file from disk into the file.

Detach - Removes the selected reference by removing the reference and the geometry from the file.

Replace - Replaces the selected reference file with another reference file.

Edit - Reads the selected reference file into a new drawing window for editing.

Bind - Removes the link to the reference file, but leaves the geometry in the file.

Update All

Update all references in the file.

Set Attach Point

Assigns an attachment point for a drawing. This attachment point is used if this file is later used as a reference file.

Import

The Import option located in the File menu provides options for reading data in, from other applications. Formats include those that read faceted data as well as precise curve, surface, and solid data.



The following import file formats are supported:

Punch! Shark FX

Imports a Punch! Shark FX file into the application. Import will merge the file with the existing file. Use Open to read the file into a new window.

3D Studio

The 3D Studio file import reads .3ds files. 3DS is a faceted file format developed in support of the Studio Max application. The following import options are supported:

Group faces

Creates a group entity that includes facets.

Smooth normals

Smooths the normals of facets that have shared vertices and edges.

ACIS® SAT

ACIS® SAT is a precise format for curves, surfaces, and solids. SAT is the format used by a variety of applications that support the ACIS® kernel developed by Spatial Technology. Typically SAT is the preferred choice for importing data into the application. Since ACIS® is the primary kernel, there is no translation required and data integrity is very high. There are no import options for this file format.

Tip: There are several applications that generate SAT data that need to be checked for integrity. This is because these systems don't actually use ACIS® and are manually recreating SAT data. In many cases these systems don't adhere to the precision tolerances associated with ACIS®. One such example is SAT files from Rhino. To avoid downstream problems with SAT files not generated from ACIS® we recommend you use the *Verify:Check Object* tool on this data. If there are problems found (usually tolerance related), use the Heal or Repair option to correct.

Adobe Illustrator®

Reads Adobe Illustrator® files created up to version 9. Version 9 and later use a PDF based format, whereas the earlier versions use a version of postscript. The following import options are supported:

Group curve segs

Creates a group entity enclosing segments from the same poly curve segment.

Polygon from fill

Creates a polygon entity from Illustrator fill type objects.

Join Curve Segments

Joins multiple curves into one continuous curve. Used only when adjoining curves are tangent and position continuous.

Adobe Photoshop®

This option imports a Photoshop file based on the .psd file extension.

AutoDesk Inventor 2010 parts and assemblies

Inventor assemblies are placed in unique layers defined by the original part name.

BMP

The BMP file format reads bit mapped image files. This type of file is useful for sketching data on top of scanned images. There are no import options for this file format.

CATIA® V4

The CATIA® V4 will import files from Dassault Systems CATIA® application. This functionality is available on both Mac and PC platforms. This will import version 4 CATIA® files. There are no import options for this file format.

CATIA® V5 (PC only)

CATIA V5, R6-R19, parts and models are now supported, for PC only. Extensions include: .CATPart, .CATProduct, .cgr. The CATIA V5 translator preserves layers and part colors.

DWG and DXF

Punch! Shark FX supports importing DWG or DXF data files through 2009. The DWG and DXF file format was created by AutoDesk as a means to share modeling and annotation data with their AutoCAD® application. DXF files contain the same information as a DWG file, the only difference is that DXF is ASCII, whereas DWG is binary. The following import options are supported:

DWG and DXF Layers

Provides three checkboxes for handling of imported layer names.

- Create Creates equivalent layers when checked. If not checked all geometry is placed into the current work layer.
- Create Empty Creates layer names even if they are empty of geometry.
- Display All Displays all layers after importing the file.

DWG and DXF File Units

Presents a pull down menu for specifying the file units. This is needed by DWG and DXF files, which do not support a unit setting in the format definition.

DWG Paper Space and Viewports

AutoCAD files support two drawing areas which include model space and paper space. All modeling is typically done in model space. Paper space is an area typically used to organize data for printing. The paper space sections of a DWG file may contain title blocks and annotations that are associated with the model. Paper space also contains Viewports, which define a viewing region and orientation of the geometry located in model space.

This application supports importing paper space and Viewports through layers and draw views. Paper space sections of the file are organized into layers and sublayers which preserves not only the separation of data from model data, but preserves the layer organization of the original file. Viewports, the ability to look at specific regions of model data, are also preserved precisely through the use of draw views which, like Viewports, view regions and orientations of model data.

The export tools such as Adobe Illustrator, DWG, and DXF also preserve the model and paper space data by maintaining layers on export. Since Viewports are not generally supported in many graphic applications, Viewports are exploded or flattened into 2D for increased interoperability support.

EPS

This format imports an encapsulated postscript file (EPS). There are no options for this file format.

Facet

This file format imports an ASCII file defined by Lockheed and supported by the U.S. Department of Defense. It was created to support sharing of faceted data with normals and material color codings for analysis programs. There are no import options for this file format.

FACT

This file format is used by the Electric Images Animation System.

GIF

This option imports files with the .gif extension.

Grid Surface

Imports an x m array of points to create a NURBS surface. There are no import options for this file format.

IGES

IGES stands for the Initial Graphics Exchange Specification. IGES is an industry standard format used to share precise CAD data that includes annotation, curves, surfaces, and limited solid modeling data. The following import options are supported:

Flavors

This drop-down provides options for Generic, AutoCAD®, and SolidWorks®. Typically you should leave this at Generic. If you're sharing data between AutoCAD® you should consider using DXF, DWG or SAT. Punch! Shark FX is one of the few applications that supports the precise surfaces and solids embedded in the DWG and DXF file formats. For SolidWorks® you should consider exporting SAT.

Trim Curve Options

This drop-down provides options for specifying the method used to handle trimmed surfaces. The three methods include Follow the File, Use 2D, and Use 3D. If the Follow the File method is set, the IGES translator will examine and use the flag specified by the exporting system in the IGES file. The Use 2D option will rebuild the 3D trim curves from the 2D parametric data. The Use 3D option will rebuild the 2D trim information from the 3D trim curves.

Auto Heal

The Auto Heal option will automatically detect groups of closed surfaces and create solids. If there are small gaps, this option will attempt to repair these problems.

Inventor (PC only)

AutoDesk Inventor 2009 parts and assemblies are supported. Inventor (.ipt) versions 6-11, and Inventor Assembly (.iam) version 11.

JPEG

This option will import a JPEG format into the current drawing. There is an option available on the Inspector for imported JPG files which allows you to set a transparent background for tracing.

Macro

The Macro File Import will read and process a list of textual commands. The commands provide options to create and manipulate geometry.

Command	Parameters	Comments
\$\$		Signifies a comment string
#Line	X1,Y1,Z1, X2,Y2,Z2	Start and end point
#Arc	X1,Y1,Z1, Radius	Center and radius
#Cylinder		
#Box		
#Cone		
#Sphere		
Move	BodyId, X1,Y1,Z1, X2,Y2,Z2	Moves a body a vector
RotateX	BodyId, X1,Y1,Z1, Angle	Rotates a body about the x axis at given angle
RotateY	BodyId, X1,Y1,Z1, Angle	Rotates a body about the y axis at given angle
RotateZ	BodyId, X1,Y1,Z1, Angle	Rotates a body about the z axis at given angle
#Subtract	BodyId 1, BodyId 2	Subtracts Body2 from Body1
#Intersect	BodyId 1, BodyId 2	Intersects Body2 with Body1
#Union	BodyId 1, BodyId 2	Adds Body2 with Body1
#Extrude	BodyId, Height, Angle	Creates an extruded body assigning it BodyId from the recently created curves. Extrusion distance is along z axis.
#Sweep		
#Resolve	BodyId, X1,Y1,Z1, X2,Y2,Z2, Angle	Creates a revolved body assigning it BodyId from the recently created curves. Angle is the amount the profile is revolved about the axis specified from the two points.

PDF

This option imports files of the Portable Document Format.

PICT (Macintosh® only)

PICT is the Apple format for bit images. This option is only available on Macintosh® computers. There are no import options for this file format.

PNG

This option imports files of the Portable Network Graphics format.

ProE®

This option provides support for PTC Pro/E parts (.prt) and assemblies (.asm). This includes support from Wildfire 4 and earlier. Pro/E import is supported on both the Mac and PC platforms. File extensions include: .prt, .prt*, .asm, .asm*, .xpr, .xas.

Punch! Objects

Imports a Punch! object file into the application. Import will merge the file with the existing file. Use Open to read the file into a new window.

Rhino 3DM

This option supports importing Rhino 2.0 and 3.0 files. Rhino is a NURBS surface modeler. Files are imported using the OpenNurbs library which preserves the NURBS features of 3dm files. The Rhino 3dm import option is supported on both Mac and PC. There are three options for importing 3dm data. These options are needed to deal with the potentially less accurate Rhino data. The following import options are supported:

Use 2D Trim Curves

Use the 2D trimming data to recreate the 3D curves for trimmed surfaces.

Use 3D Trim Curves

Use the 3D trim curves to recreate the 2D curves for trimmed surfaces.

Auto Heal Solids

Converts closed volumes of surfaces into solids. In addition, the resulting bodies are checked and repaired for possible geometry or topology errors.

SketchUp

This option reads data from the Google SketchUp modeling application. It will import SketchUp files (.skp) from version 7 or earlier. SketchUp data consists of planar data only. Data is converted into planar surface object files.

Spline

The spline import option will read an ASCII file containing x, y, z values as a spline. There are no import options for this file format.

STEP

STEP is considered the replacement file format to IGES. STEP is a newer industry standard format with respect to IGES and is used to share precise CAD data that includes annotation, curves, surfaces, and solid modeling data. The following import option is supported:

Check Object

There is one option for importing STEP files, which is Check Object. Turning this option on will closely examine each object for tolerances that adhere to the ACIS® kernel. If any issues are detected, this will attempt to repair the problems.

STL

This import option reads STL formatted files. STL is a triangular facet format used in the rapid prototyping industry. The following import options are supported:

STL FileUnits

Presents a pull down menu for specifying the file units.

Text

The text import file will read text strings from an ASCII formatted file and create text entities. There are no import options for this file format.

TIFF

This option imports a file with the TIFF format.

Truespace COB

This import format supports reading in of Truespace COB files. This file format supports faceted data and color attributes. There are no import options for this file format.

VDA

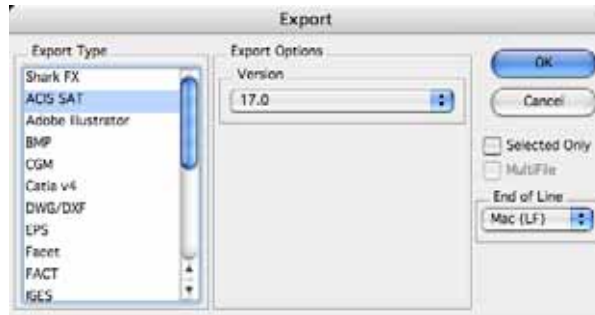
VDA-FS is a CAD data exchange format for the transfer of surface models from one CAD system to another. The name stands for “Verband der Automobilindustrie - Flächenschnittstelle”, which translates to the “organization of the automotive industry - surface translation format”. This file format standard was specified by the German organization VDA. VDA-FS has since been superseded by STEP, ISO 10303.

Wavefront OBJ

WaveFront OBJ is widely used when dealing with graphics technology. This file format allows you to import colors, names, texture coordinates, and normals. Importing texture coordinates is particularly useful for producing smooth renderings by preserving the original NURB.

Export

The export option provides a means to export geometry data to a wide variety of applications. Export options include precise and faceted formats.



In addition to settings unique to a file format, the export dialog has two checkboxes related to all exports. These two checkboxes are:

Selected Only

This option saves only objects which are selected with the Arrow tool.

Multi File

Saves each selected solid as an individual file. For example, if your file contains 10 solid objects, each object will be stored as a separate file. This option is only available if Selected Only is enabled.

If an ASCII based export format is selected, the following option is supported:

End of Line

This option sets the end of line terminator for Mac, PC or Unix.

The following export file formats are supported:

Punch! Shark FX

This option will export the file as a Punch! Shark FX file. This is useful for sharing data with older versions as you can specify the version upon export.

ACIS® SAT

Exports curves, surfaces, and solids using Spatial Technology's ACIS® kernel. The following export option is supported:

Version

Specifies the format version to use for export.

Adobe Illustrator®

This format exports the current display image to Adobe Illustrator®. There are no options for this file format.

BMP

The BMP file format reads bit mapped image files. This type of file is useful for sketching data on top of scanned images. There are no export options for this file format.

CGM

This format exports Computer Graphics Metafile (CGM) file. The resultant file is created from the 2D display of the data and is therefore dependent on the view in use at the time the export is created. There are no options for this file format.

CATIA® v4

Exports surfaces and solids into Dassault System's CATIA® v4 file format. There are no options for this file format.

CATIA® v5 (PC only)

Exports surfaces and solids into Dassault System's CATIA® v5 file format, for PC only. There are no options for this file format.

DWG and DXF

Exports the file as an AutoCAD® DWG or DXF file. The following export options are supported:

DWG and DXF Format

Specifies the format version to use for export. The following formats are supported:

R12

Pre ACIS® version of AutoCAD® that does not support surfaces or solids. Surfaces and solids export their facet display list using FACE3D entities. This version of AutoCAD® also did not support ellipses, conics, and splines. These types are converted automatically as polylines.

R13

ACIS® data is supported including surfaces, solids, ellipses, and splines.

R14

Same data conversions as R13 but uses a newer version of ACIS® for curves, surfaces, and solids.

R2000

Same data conversions as R13 and R14 but uses a newer version of ACIS® for curves, surfaces, and solids.

DWG and DXF File Units

Since both DXF and DWG are unitless formats, this option internally scales the data for the specified unit settings.

Tip: During DWG and DXF export the following conversions are applied: Hatch patterns are converted to lines. Groups are automatically exploded into individual geometry types. Draw views are automatically flattened into the sheet. ACIS® data only supported in DXF.

EPS

This format exports an encapsulated postscript file (EPS). There are no options for this file format.

Facet

This format exports the display mesh using a format defined by Lockheed. The facet file format includes normals per vertex and color information. There are no options for this file format.

FACT

This file format is used by the Electric Images Animation System.

IGES

The IGES option exports curves, surfaces, and solids out to the Initial Graphics Exchange Specification (IGES) format. The following export options are supported:

Flavor

The flavor pull down automatically assigns settings for particular applications that include AutoCAD®, SolidWorks®, Pro/E®, and Alias.

Write MSBO #186

This option will create Manifold Solid Boundary Objects when solids are exported into the IGES file.

Trimming Curve Preference

Specifies the trimming curve preference used to define trimmed surfaces. Check 2D parametric to have the 3D trim curves driven from the 2D parametric curves. Check 3D model space to have the 2D parametric curves driven from the 3D trim curves.

IGES File Units

Specifies the units for the exported file. The application will internally scale the data to the appropriate unit type.

JPEG

This option will capture and save the entire display screen to disk in the JPEG format.

PICT (Macintosh® only)

Exports objects displayed on the screen into the Apple PICT format.

RAW

This format is used primarily for exporting triangular facet geometry to the popular Persistence of Vision ray trace utility. To convert a RAW file into POV, use the POV utility program RAW2POV. There are no options for this file format.

STEP

Exports files that adhere to the STEP Standard AP203. There are no options for this file format.

Tip: This translator only supports modeling data such as curves, surfaces, and solids. Drafting and annotation data is not exported.

STL

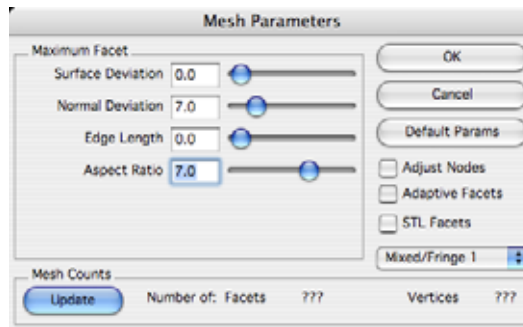
The STL option exports data in a format suitable for creating stereo lithography files. This format consists of a faceted representation of your surface or solid data. The following export options are supported:

STL Format

Specifies the storage format to use for export.

After initiating the export, you will be prompted with the Mesh Parameters dialog box. This dialog gives you control over the facet resolution to use for export.

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 of error that results from this conversion is controlled by the settings in the mesh parameters dialog box.



Convert to Mesh Dialog

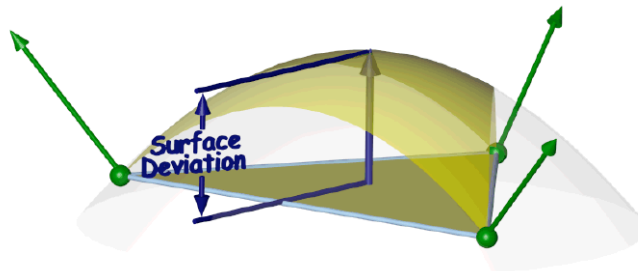
During the conversion, vertex points are distributed on the surface or solid. These vertices are then grouped into 3-sided and 4-sided facets. The conversion is deemed acceptable when the generated vertices and facets satisfy the settings. The five (5) available settings are Surface Deviation, Normal 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, then click the Update button to see the number of facets and vertices generated. Determining the combination of settings that will work for a given situation can be a little bit of an art. If one setting becomes too tight, the other settings will have no effect. If one setting becomes too loose, it will have no effect.

Keep in mind that under “real” circumstances the faceting algorithm only uses the settings if possible. It is often not possible to satisfy all settings simultaneously. In this situation, the algorithm decides which settings to “loosen” or ignore.

Surface Deviation

This setting controls the maximum allowed distance between any point on the actual surface or solid and the facet representing that point. The exaggerated figure below shows the largest distance between a patch on the actual surface (yellow) and the corresponding planar facet (brown).



Normal Deviation

This setting controls the maximum allowed angular difference between any normal on the actual surface or solid and the corresponding interpolated normal on the facet.

Edge Length

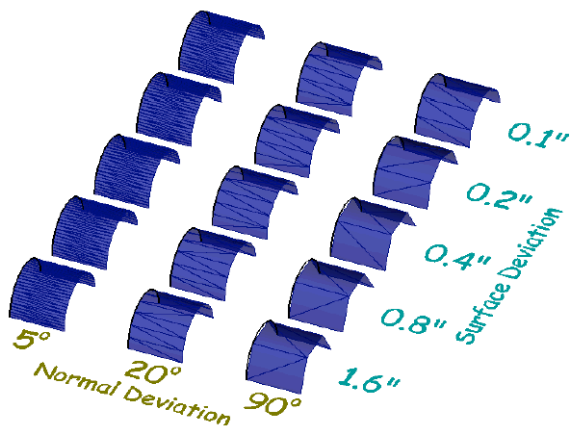
This setting controls the maximum allowed edge length of any given facet.

Aspect Ratio

This setting controls the maximum allowed aspect ratio of any given facet.

STL Facets

This setting will force the facets generated to be suitable for stereolithography usage. This setting is usually used when exporting STL files.



Text

Exports text entities into an ASCII file containing the text characters. There are no options for this file format.

VDA

VDA-FS is a CAD data exchange format for the transfer of surface models from one CAD system to another. The name stands for “Verband der Automobilindustrie - Flächenschnittstelle”, which translates to the “organization of the automotive industry - surface translation format”. This file format standard was specified by the German organization VDA. VDA-FS has since been superseded by STEP, ISO 10303.

Viewpoint Media

Exports meshes, surfaces, and solids into a folder that contains html formats necessary to view in a web browser using Viewpoint Media (VET) technology. The following export options are supported:

Geometry Quality

Controls the level of visual quality for the resulting VET data set.

Author Name

Name used for copyright information stored in the VET data set.

VRML

This format exports the facet data. VRML supports normals per vertex and color. VRML is an acronym for **V**irtual **R**eality **M**odeling **L**anguage. The following export options are supported:

VRML Format

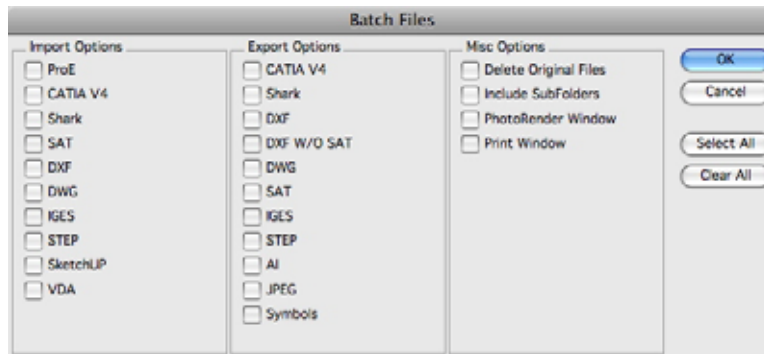
Specifies the format version to use for export.

Wavefront OBJ

Wavefront OBJ is widely used when dealing with graphics technology. This file format allows you to export colors, names, texture coordinates, and normals. Exporting texture coordinates is particularly useful for producing smooth renderings by preserving the original NURB.

Batch

The Batch File command located on the File menu will read in a variety of file formats and process into other file formats.



Misc Options

- Delete Original Files** Deletes the files after it is processed (use with caution).
- Include SubFolders** Process files located in nested subfolders.
- PhotoRender Window** Photorenders the contents of the window.
- Print Window** Prints the screen.

Web Publish

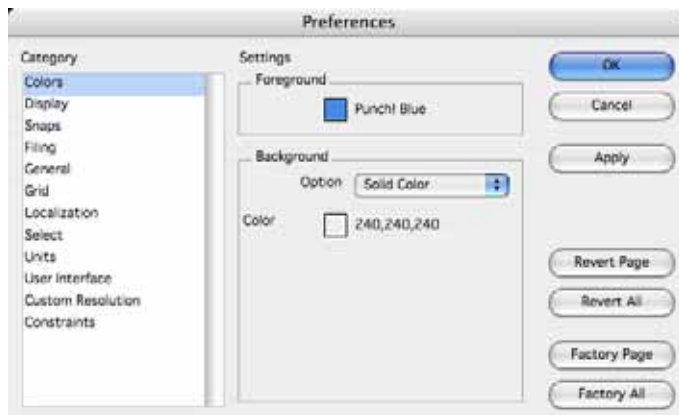
The Web Publish command is located under the File menu. The tool will capture the screen and embed the resulting JPEG within an HTML file suitable for display within an internet browser application.

Preferences...

The Preferences dialog, accessed from the Shark FX menu, allows you to customize various start-up and runtime settings for Punch! Shark FX. The Category section lists the available preference groups. The Settings section displays the interface for the selected category. The preference group settings are detailed below:

Colors

The colors category assigns the foreground and background color.



Background Color

Specify the background color of the drawing window.

Foreground Color

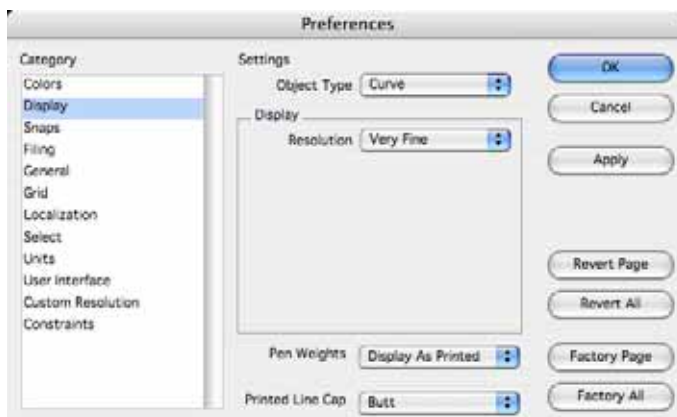
Specify the foreground color of the drawing window.

Preview Color

Displays the foreground and background into a preview area.

Display

The Display preferences page sets settings for default display resolutions of curves, surfaces, and solids.



Object Type

The Object Type pull down provides a means to set separate resolution settings for curves, surfaces, and solids.

Resolution

The Resolution pull down menu provides options for setting the resolution between coarse and super fine.

Pen Weights

The Pen Weights pull down menu provides options for the displayed pen weights that have been customized in the drawing, as they appear when printed.

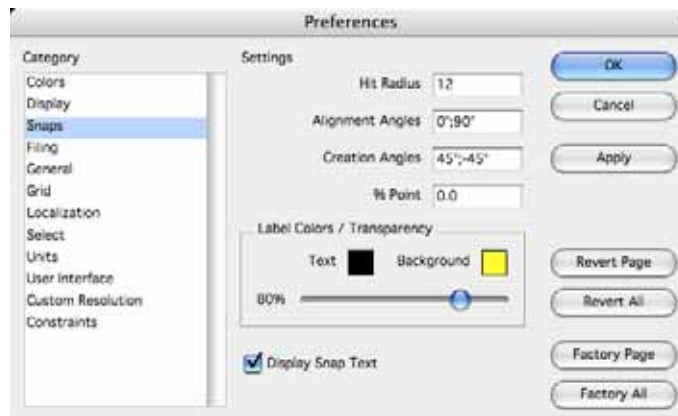
Printed Line Cap

The Printed Line Cap provides three options that control how line ends are printed. The three options are butt, round, and square. Note, these options are only activated upon printing.

*Butt**Round**Square*

Snaps

Sets the default settings for the Snap tool. For more detailed information on Snaps settings, see “Snap Settings” on page 26.



Hit Radius

The pixel radius which is examined for the Snap tool.

Alignment Angles

The angles that are highlighted when referenced from a wake up snap.

Creation Angles

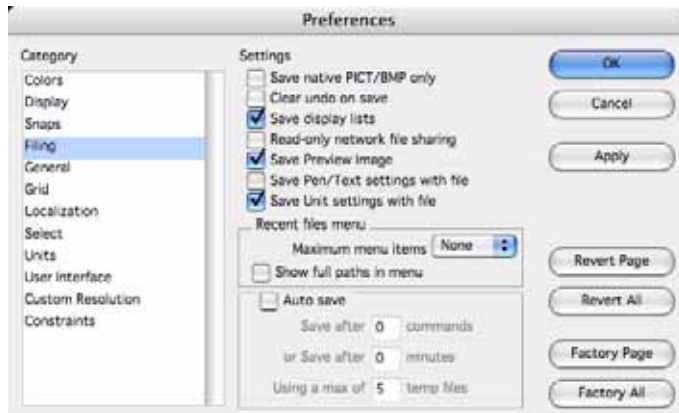
The angles that are highlighted relative to the first snap point.

% Point

Percentage along a curve that is examined for snap locations.

Filing

This category sets up the preferences for filing and back up file options.



Save native PICT/BMP only

When this option is selected, images are saved in the native format for either Mac or PC. The native format is more compact, but will not display on both Mac and PC computers.

Clear Undo On Save

Clears the contents of the undo buffer after a save file. Clearing undo buffers frees up considerable memory for additional commands.

Save Display Lists

Saves the display list for surfaces and solids. This includes the vector and facet display lists for each entity. Saving the display will make file loads faster, but file sizes larger.

Read-only network file sharing

With this option enabled, files are locked when read by the application. This prevents others from reading and writing a file you may be working on.

File Previews

Checking the Save File Previews option will save a preview image of the file. The preview is seen with the *File:Open* command. File previews are cross platform compatible.



Save Pen/Text Settings with file

Checking the Save Pen/Text settings with file option will save any annotations in the drawing along with the file.

Save Unit Settings with file

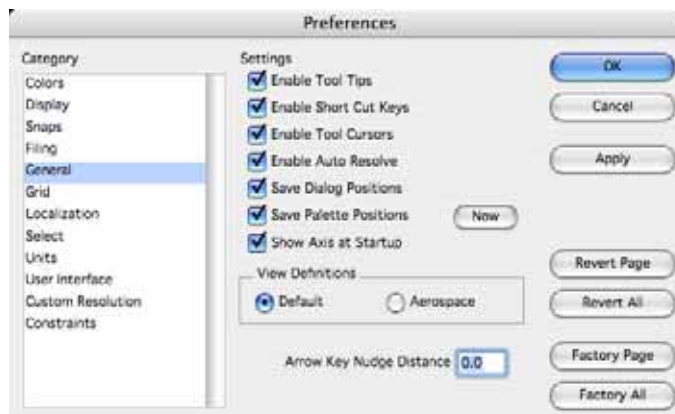
Checking the Save Unit settings with file option will save any localized measurement units you have set along with the file.

Auto save

Automatically saves the current drawing into a file located in the back up folder. Saves up to a user specified maximum. Back up files can be created every so many commands or every so many minutes.

General

This category assigns general settings including axis, tool tips, short cuts, and dialog positions.



Enable Tool Tips

Displays, in the prompt window, a brief description of an icon as you move your cursor over its tool image.

Enable Short Cut Keys

Turns on or off the ability to use short cuts for quick command access.

Enable Tool Cursors

Turns on or off the ability to use tool cursors. Two types of tool cursors exist which support snapping or selection operations.

Enable Auto Resolve

Immediately resolves dependent geometry when changes are made.

Save Dialog Positions

Saves all displayed dialog positions on the screen when you exit the program.

Save Palette Positions

Saves all displayed tool palette positions on the screen when you exit the program.

Show Axis at Start Up

Displays the XYZ axis located at the origin.

View Definitions

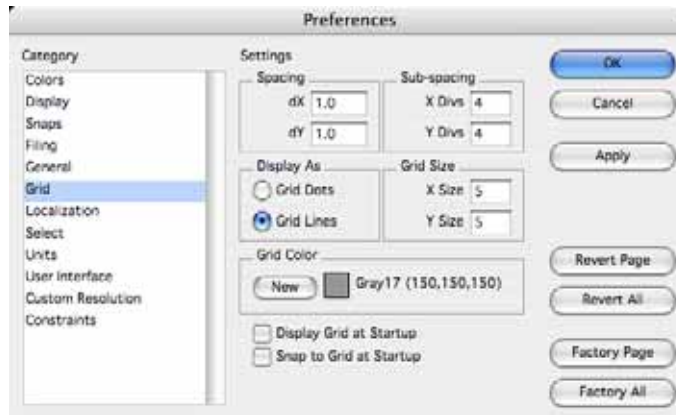
Changes the definitions of side and front views to reflect mechanical or aerospace design conventions.

Arrow Key Nudge Distance

The distance used to move objects while in the Select tool using the arrow keys.

Grid

This category assigns the default display grid settings.



Spacing

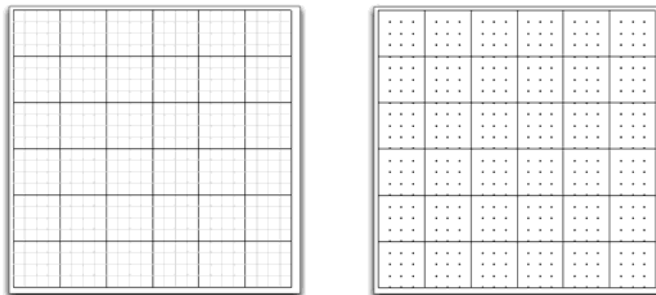
The dX and dY spacing between major grid lines.

Sub-spacing

The X Divs and Y Divs (minor grid lines) displayed between the major grid lines.

Display As

Displays the grid as dots or lines.



Grid Size

The number of cells in the grid.

Grid Color

The display color for the major grid lines. The minor grid lines are displayed as a different shade of this color.

Display Grid at Startup

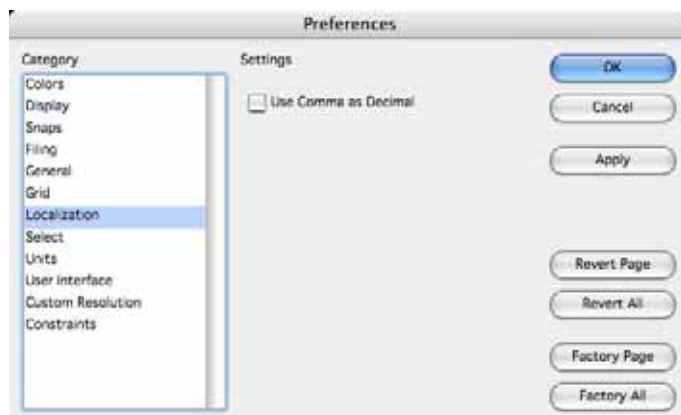
Enable this item to display the grid when the program is launched.

Snap to Grid at Startup

Enable this item to activate grid snapping when the program is launched.

Localization

The Localization category displays options specific to language localization.

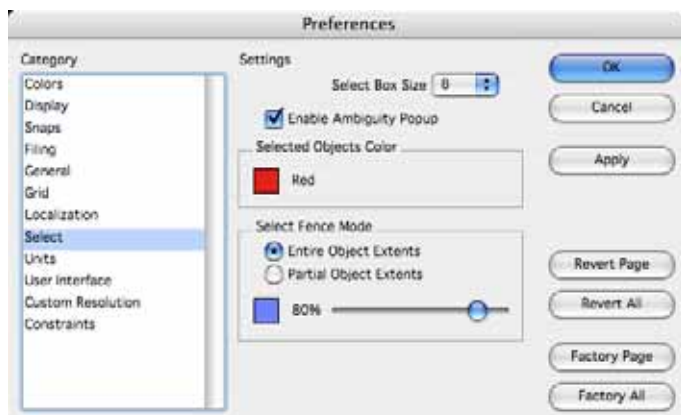


Use Comma as Decimal

Replaces the period used in floating point representations with a comma.

Select

The Select category displays options for picking objects.



Pick Box Size

The pick aperture size for the snap tool and selections.

Enable Ambiguity Popup

Displays a popup menu with all the objects in the pick region. The user then picks the object of interest. If off, the object closest to the view is selected. If the objects are at the same depth, the object that was first created will get selected.

Selected Entities Color

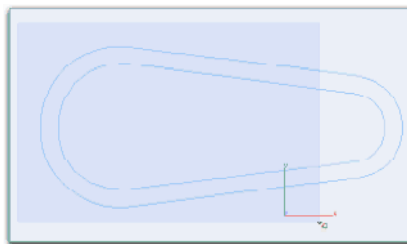
Sets color used to indicate a selected object.

Select Fence Mode

Sets the method used for picking with a bounding box. Entire object extents requires the user select the complete extents of the object, whereas partial extents requires only a portion be fenced to select.

Transparency

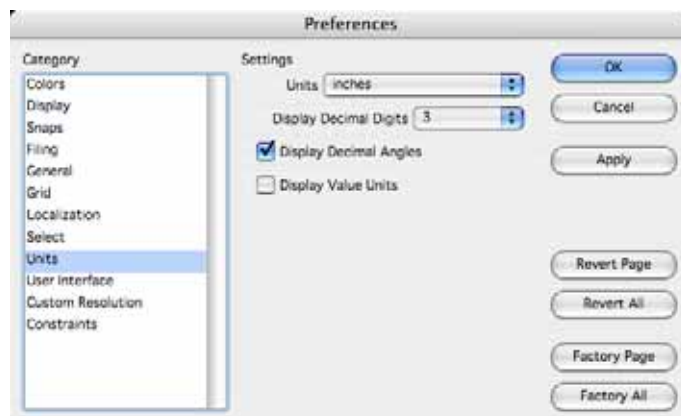
The selection box uses window overlays with transparencies. You can control the selection background color and transparency level with the transparency slider located at the bottom of the Select Fence Mode.



The user can modify the color and transparency level associated with the selection box. To change these attributes, display the Preferences dialog box and pick the Select category. The selection transparency and color are controlled under the Select Fence Mode control grouping.

Units

The Units category displays the options for units of measure used for display of numeric values.



Units

The Units pull down sets the unit format for numbers displayed in dialog boxes and Data Entry Windows. At this time, Punch! Shark FX supports six different units of measure. These units and their associated symbols are listed below:

<u>Unit</u>	<u>Symbol</u>	<u>Abbreviations</u>	<u>Example</u>
inches	"	in.	12.0"
feet	'	ft.	1.0'
feet/inches	' "	ft-in	1' 0"
millimeter	n/a	mm	1000 mm
centimeter	n/a	cm	1000 cm
meters	n/a	m	1 m

Display Decimal Digits

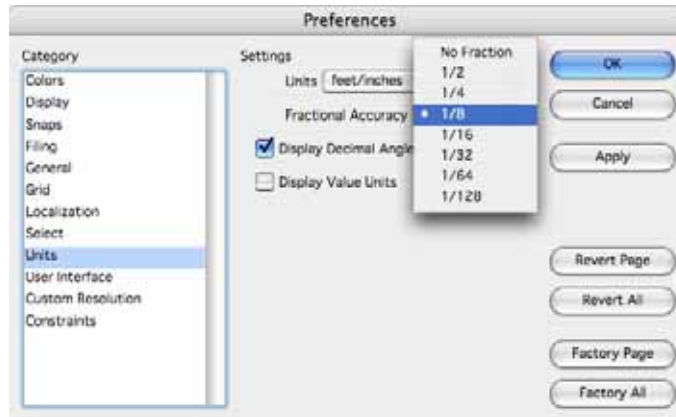
Sets the number of digits used to display floating point values in the Data Entry Fields and dialog boxes.

Display Decimal Angles

When checked, this option will display angles as decimal values instead of degrees, minutes, and seconds.

Architectural Units Note

Use the settings under the Units category to configure the application for working with feet-inches. The data entry window and unit display fields will reflect the current unit settings. When the unit settings is selected as feet-inches, the accuracy menu provides a drop-down list of fractional settings.



The Tool Bar data entry window reflects the preferences for unit settings. When using the feet/inches unit settings, data entry fields will display and process with feet/inches notation.



Feet/Inches Settings in Data Entry Window

All fields including the data entry window and dialog input fields support an intelligent unit parser. At any time you can type in a value supporting any unit expression. Examples include:

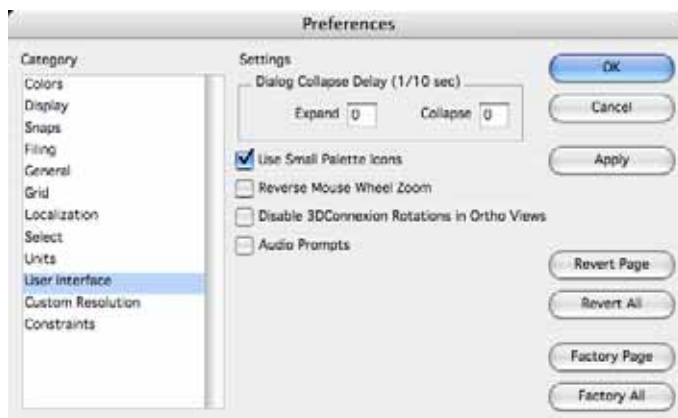
- 14' 11"
- 179"
- 3cm
- 4mm
- 14" + 63cm

Be careful about typing in a minus sign between the feet and inches. This will be interpreted as a subtraction expression.

14'-11" = 157inches

User Interface

The User Interface category displays options for controlling dialogs, icon display, and wheel scroll direction.



Dialog Collapse Timing

The Expand and Collapse time sets a delay for how long the cursor must be out or in a dialog to collapse.

Toolbar Layout

Click the Use Small Palette Icons check box to display icons 24 by 21 pixels. The default icons are 30 by 28 pixels.

Wheel Scroll Direction

For those users with a wheel button on their mouse, there is a new option to reverse the direction of the zoom.

Disable 3DConnexion Rotations in Ortho Views

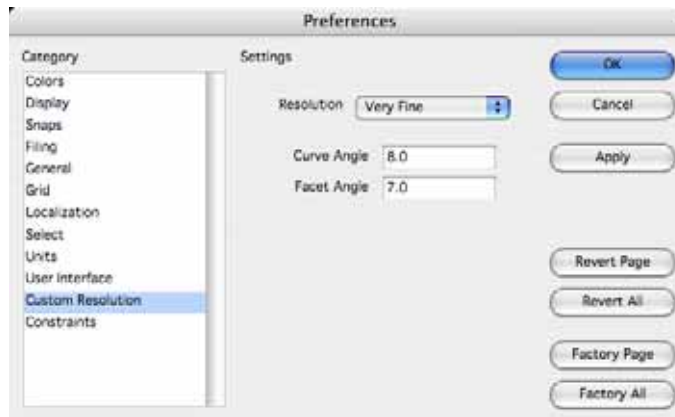
Select to disable 3DConnexion rotations while in Ortho views.

Audio Prompts

The Audio Prompts check button enables speaking of the message line prompts. This is particularly useful with tasks such as 3D digitizing where reading prompts is difficult.

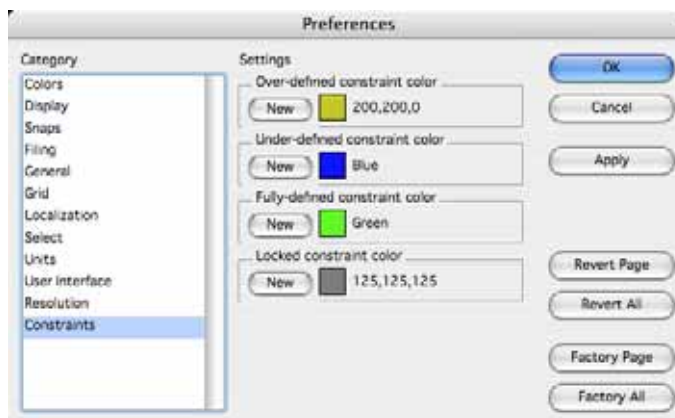
Custom Resolution

The Resolution category allows the user to define custom settings for display resolutions.



Constraint Preferences

The Constraints category displays options for dimension and constraint default colors.



Factory

Changes all the settings for all preference groups back to the original installation values.

Revert All

Resets all values to the values prior to opening the dialog box.

Cancel

Resets all values and exits dialog box.

OK

Exits and writes out all changes to the preferences file.

Short Cuts

Shortcut keys provide access to creating and modifying user defined command keys. Multiple shortcut key sets can be defined. These key sets can be used in support of multiple users sharing a common workstation or multiple design modes. The shortcut key dialog manager displays a list of commands and allows the designer to assign associated keys.



As you become more proficient with Punch! Shark FX, you may elect to use the shortcut keys to issue commands. Shortcut keys require less mouse movements to access frequently used commands, but require that the user memorize command keys or position a “cheat” sheet nearby. This chapter describes features for accessing capability through the typical Apple character prefix and special Punch! Shark FX keys.

Key Set

Used to activate the current key set.

New...

Click this button to input a name for a new key set.

Delete...

Click this button to delete an existing, user-defined new key set. The system-defined “Shortcuts” key set cannot be deleted.

Category

This section lists the available command groups.

Command

This section lists the available commands within the selected category. Any existing shortcut assignments are also indicated in this list.

Shortcut key

Displays the shortcut key for the selected command list item. Also used to change or assign a shortcut to the selected command list item.

Description

A brief description of the selected command list item.

Assign Key

Assigns the specified shortcut key to the selected command list item. Any existing shortcut for this command is removed.

Remove Key

Removes the specified shortcut key from the selected command list item.

Revert

Changes the keys back to the point where the dialog box was first displayed.

Factory

Changes the current shortcut key set assignments back to the original installation values.

Print

Prints a listing of all keys and their assigned keys.

Assigned

If checked, only the shortcut keys that are assigned to commands are printed. If not checked, all keys are printed.

Print Layout

The Print Layout command lets you set up the page size, orientation, and other page layout options for printing of a Punch! Shark FX file.

The Drawing Size command located in the Layout pull down menu allows you to see the size of the maximum plotting/printing area relative to the drawing and to specify the scaling of the drawing so it fits the paper size and orientation set by the Page Setup... command.

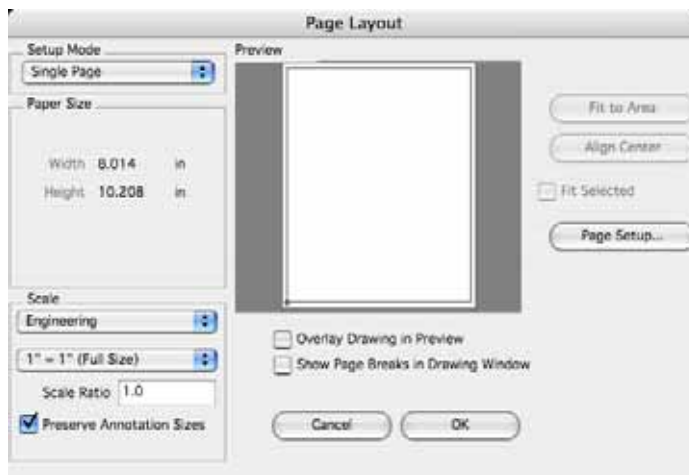
Setting up the Drawing Size

If the drawing is larger than a standard piece of paper or so small that it would be impossible to use it at full scale, you must set the Drawing Size. The Drawing Size command allows you to set the page width, height, show page breaks, tile prints, and set the drawing scale. With any of these methods, the geometry itself does not change scale; it only changes visually, not physically. Punch! Shark FX supports four (4)

modes for setting up a print layout: *Single Page*, *Height and Width*, *Rows and Columns*, and *Advanced*. The modes are detailed below.

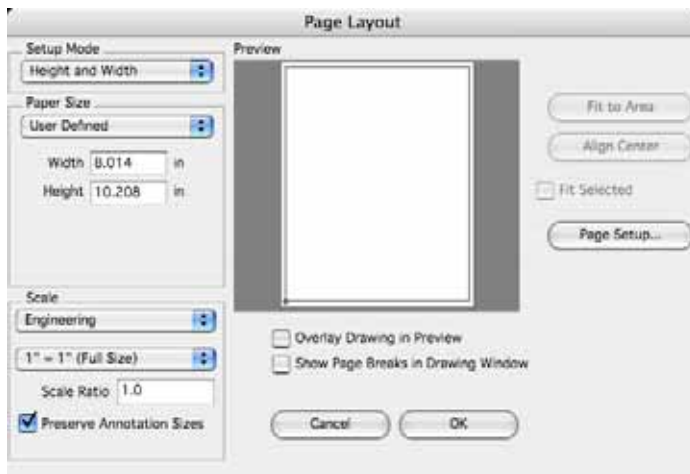
Single Page Mode

In this mode, all printing is forced to fit on a single sheet of paper as reported back from your system's printer interface.



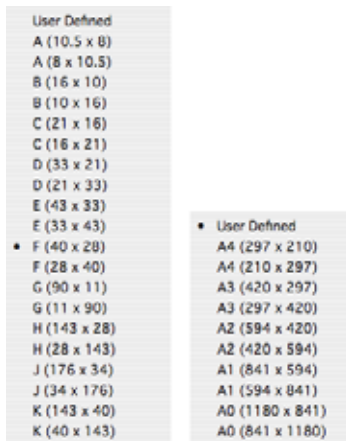
Height and Width Mode

In this mode, printing is targeted to a user-defined, arbitrary drawing size. The drawing size is specified using the current unit of measure. The drawing size is *not* limited to the physical paper size in your printer. As needed, the drawing will be tiled to fit the physical paper size supported by your printer. The tiles are indicated in the preview by the lighter matrix of lines within the drawing border. In this mode it is possible to have "excess" paper along the top and right edges of the drawing. The lighter tile matrix lines extending past the drawing border indicate this situation.



Paper Size

Selecting from this drop-down list sets the Width and Height fields according to standard drawing sizes. The list of standard drawing sizes (ANSI or ISO) corresponds to the current unit of measure. Set the unit of measure in the preferences.



Width

Sets the drawing width using the current unit of measure.

Height

Sets the drawing height using the current unit of measure.

Rows and Columns Mode

In this mode, printing is targeted to a user-defined, arbitrary drawing size. The number of tile rows and columns specify the drawing size. The drawing size is *not* limited to the physical paper size in your printer. The tiles are indicated in the preview by the lighter matrix of lines within the drawing border. In this mode it is *not* possible to have “excess” paper in the drawing. Setting the tile rows and columns to 1 x 1 with Overlap set to 0 is equivalent to single page mode.



Rows

Sets the drawing height as the number of tile rows.

Cols

Sets the drawing width as the number of tile columns.

Overlap

Sets the amount of overlap (using the current unit of measure) included along the top and right edges of each tile page. The overlap provides a duplicate printed region on each tile page that can be used for tile alignment, gluing or taping. This value may be set to 0.

Advanced Mode

In this mode, printing is targeted to a user-defined, arbitrary drawing size. The drawing size can be specified as a combination of *Height and Width* and/or *Rows and Columns*. The drawing size is *not* limited to the physical paper size in your printer. The tiles are indicated in the preview by the lighter matrix of lines within the drawing border. The lighter tile matrix lines extending past the drawing border indicate the presence of excess paper.

The dominant drawing size values are indicated in the dialog box by the bold labels. For example, the dialog below shows that the drawing width is 36 in and the drawing height is 3 Tile Rows. The dimmed

labels indicate the corresponding reference sizes. That is, the Width of 36 in corresponds to 5 Tile Columns and the height of 3 Rows corresponds to a Height of 30.875in.



In addition to the drawing size settings that vary according to the setup mode, there are additional settings that are common to all modes. These are detailed below:

Setting the Drawing Scale

You may set the print scale using either the drop-down list of common values or you can enter an arbitrary value in the Scale Ratio field.

Fit to Scale (Advanced mode only)

The drawing size Width and Height will be adjusted to fit the geometry at the current Scale value.

Fit to Area

The Scale value will be adjusted to fit the current drawing size Width and Height.

Align Center

Centers the drawing border to the geometry extents.

Fit Selected

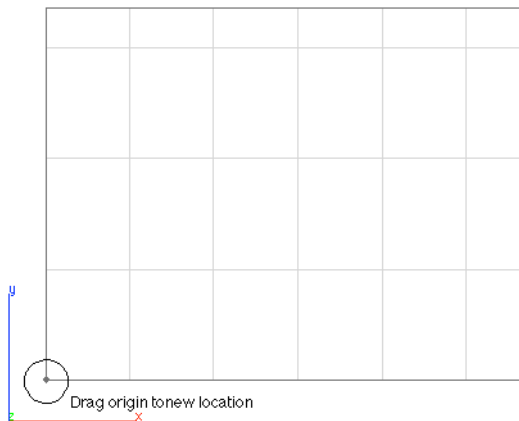
If checked, only the selected objects will be considered when making the Fit to Scale, Fit to Area, and Align Center adjustments.

Overlay Drawing in Preview

If checked, the actual drawing geometry will be displayed in the preview section. For very large drawings this may be too slow. If *not* checked, bounding rectangles are drawn in place of each drawing object.

Show Page Breaks in Drawing Window

If checked, the drawing border and any tile matrix lines are drawn to the drawing window. This is necessary if you need to move the drawing border. You may move the origin of the drawing border by positioning the pointer over the lower left corner of the border in the drawing window, pressing the mouse button, and dragging to new location.



Page Setup

The page setup command lets you set up the page size, orientation, and other page layout options for printing of a Punch! Shark FX file.

Print

Prints the current drawing using the page description defined in the page setup command. Prints to the active print device defined in the chooser.

Print Window

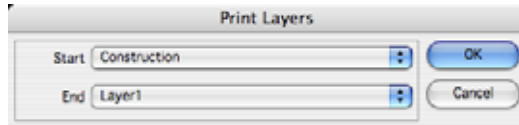
Prints the contents of the current drawing.

Print by Layer

This is extremely useful when you have multiple sets of data within one file that needs printing. One particular example is printing multiple sheet drawings.

Using the Print By Layer Command

- 1 Use Page Layout to define the print size, scale, and positions.
- 2 Select *File:Print by Layer*.
- 3 Select the Starting Print Layer.
- 4 Select the Ending Print Layer.



The Print by Layer tool is especially useful for creating PDF design books in association with drawings and parts. This is because the Print by Layer command was designed to create a single document with multiple pages. PDF is included with Mac OS X but requires a third party utility on the PC.

Print Service

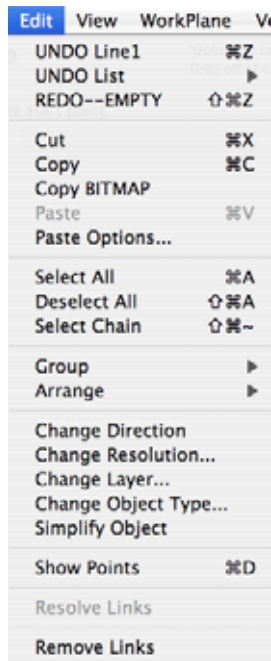
You can order a scaled, large format print of your design from Punch! Software. Clicking the Print Service option takes you to a Punch! website where you can upload your drawing and choose your print service options.

Quit/Exit

The Quit command will exit the Punch! Shark FX program. If the current file has not been saved you will be asked if you wish to do so. After quitting, you will return to your normal Macintosh® desktop.

Edit

The edit menu contains commands to select and modify objects. Example commands include cut, copy, paste, and select all. Typically, commands in the Edit menu require the entity be preselected.



Undo

This command will undo the last operation. Example commands that are placed in the undo stack include creation, modification, and deletion of entities. Example commands not placed in the undo stack include view operations such as zooming and change views.

Punch! Shark FX supports an unlimited number of undo commands. You can limit the number of undoable commands by checking the Clear Undo List Upon Saves in the *File:Preferences* dialog box.

Undo List

The Undo List displays a list of your recent actions, in descending order, in a submenu.

Redo

The Redo command will restore the last undo operation. The number of redo operations equals the number of undo operations.

Cut

The Cut command deletes the selected objects and places them in the Clipboard.

The contents placed into the system clipboard are 2D. The internal clipboard for Punch! Shark FX is 3D but does not save associative information.

Copy

The Copy command places a 2D copy of the selected data into the system clipboard, and an unassociative 3D copy into Punch! Shark FX internal clipboard. The Copy command does not alter the selected entities.

Copy BitMap

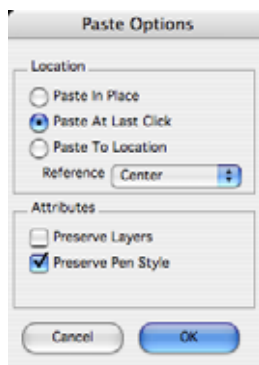
The Copy Bitmap command provides a means to copy a rectangular region as an image and store in the clipboard as either a bmp or PICT image. This image data can then be pasted into other applications.

Paste

The Paste command will copy the contents of the current clipboard into Punch! Shark FX. The contents are centered about the last mouse down position.

Paste Options

The Paste Options dialog provides settings regarding how items are pasted into the drawing using the *Edit:Paste* command. The dialog consists of location and attributes options for pasting content.



- Paste In Place** Preserves the original location of the objects in the paste buffer.
- Paste At Last Click** Inserts the paste buffer objects at the last click location.
- Paste To Location** The user specifies a location for the paste buffer objects.
- Reference** Displays a drop-down defining how the paste buffer objects are aligned to the user location for Paste At Last Click and Paste To Location. Options include:
- Center
 - Upper Left
 - Lower Left
 - Upper Right
 - Lower Right

These references are relative to the current workplane.

- Preserve Layers** If selected, the Preserve Layers checkbox maintains the original layers of the objects in the paste buffer. If this option is not selected, pasted objects will use the current construction layer.
- Preserve Pen Style** This option when enabled will keep the original pen styles consisting of color, weight, and pattern. If this is not checked, the pasted objects will use the current pen color, weight, and pattern.

Select All

Use the *Edit:Select All* command to select all objects in the displayed layers.

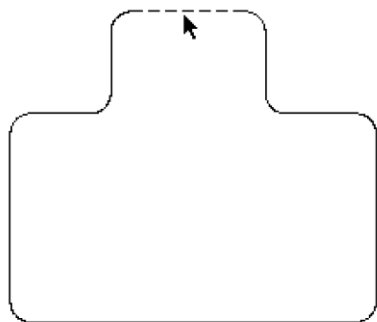
DeSelect All

Use the *Edit:DeSelect All* command to turn off the selection status for all objects.

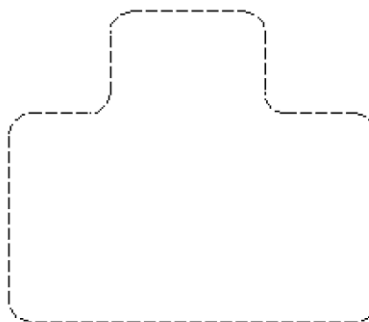
Select Chain

Selects curve objects that are connected to the currently selected object. To use the Select Chain command:

- 1 Select the start curve as in the figure below.
- 2 Select the *Edit:Select Chain* command.
- 3 All objects connected to the start or end are selected as in the figure below.



Select start curve

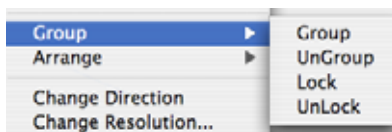


Connecting curves selected

Select Chain

Group

The Group command displays a menu with four options. These options are all related to grouping and locking commands.



A group is a collection of objects that are treated as one unit. Moving the group will automatically move all members of the group. To create a group, select the objects you want to group, then select the Group option from the *Edit:Group* menu. Punch! Shark FX supports nested groups.

UnGroup

The Ungroup command will remove the last group entity associated with the selected entities. If the object selected has nested groups, this command only removes the last group operation.

Lock

All Punch! Shark FX objects have a locking attribute flag. The lock attribute prevents all modifications of a locked object. Locked objects cannot be translated, rotated or any other operation that would change

the position or shape of an object. However, you can still reference locations from a locked object. To lock an object, use the *Edit:Group:Lock* command and select the objects to lock.

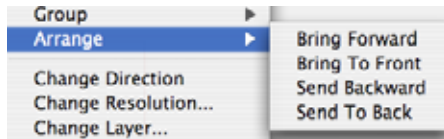
Unlock

The Unlock command will remove the lock attribute placed on an object. To unlock an object, select the Unlock command from the menu, then select the objects to unlock.

Tip: This is one of the few commands that requires the user select the object after they have selected the command from the Menu Bar.

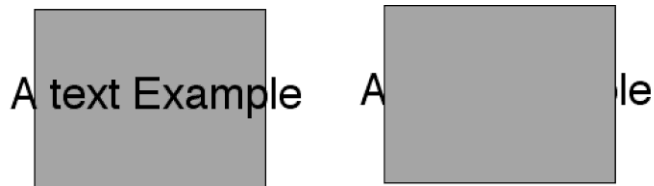
Arrange

The Arrange command changes the position of the selected objects within the entity list. The entity list is a list of all entities in the file. You can change the position using one of four commands:



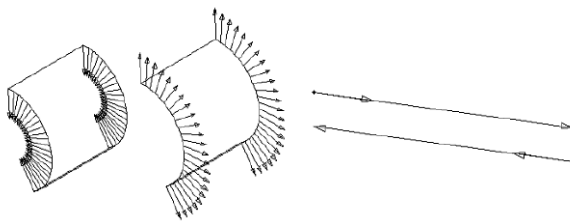
- Bring Forward Moves the object one position forward.
- Bring To Front Moves the object to the first position in the list.
- Send Backward Moves the object one position backward.
- Send To Back Moves the object to the last position in the list.

An object’s position in the entity list can impact how the object is displayed with respect to other objects. See figure below where a filled polygon was brought to the front of a text entity.



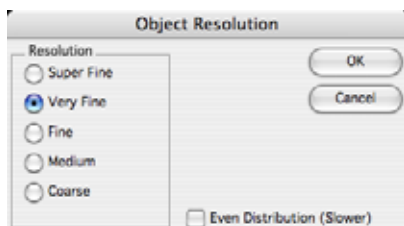
Change Direction

This command will change the orientation of a curve or surface. Changing direction of a curve is sometimes needed when creating certain surfaces such as net surfaces. For surfaces, changing the direction can affect the way the surface is rendered or shaded.

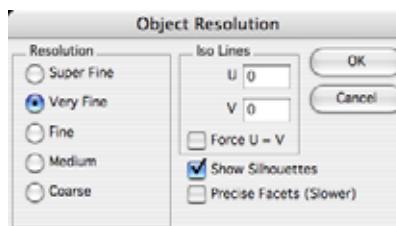


Change Resolution

The change resolution command modifies the display resolution of a curve, surface or solid.



For Curves



For Surfaces/Solids

Resolution

In the case of a curve, the Resolution controls the deviation of a line segment to the actual curve. For a surface or solid, the resolution impacts the normal deviation of a planar facet representing the precise definition.

Iso Lines

Iso Lines controls the number of isometric lines calculated across the surface. For example, if you specify 10 x 5 iso lines for a solid, each face on the solid will display a grid of 10x5 curves. Force UV will keep the u and v values the same.

Tip: The grid is applied to an untrimmed surface and then trimmed to the visible portions of the surface. Therefore it is possible you may not see all the Iso Lines you requested.

Show Silhouettes

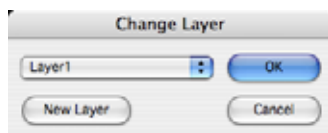
The show silhouettes checkbox tells the application whether or not to calculate view dependent edges. In the case of NURBS surfaces, this can be a performance drain.

Precise Facets

The Precise Facet checkbox employs an alternate faceting algorithm that takes longer and creates many more facets but can produce more favorable facets for NURBS surfaces with high curvature.

Change Layer

The Change Layer command displays a dialog box to change an entity's current layer to another layer or a new layer.

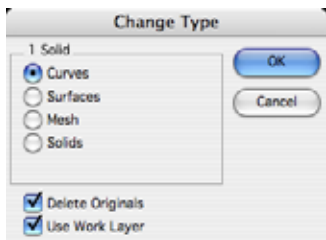


Change Object Type

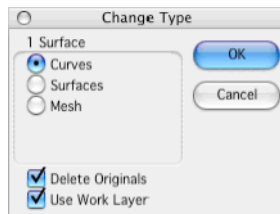
The Change Object Type command provides a means to change the geometry type of selected geometry. Some example operations you can perform with this command include:

<u>Change From</u>	<u>Change To</u>
Curve	Another Curve Type, PolyLines, Lines
Surface	Curves, Mesh
Solid	Curves, Surfaces, Mesh

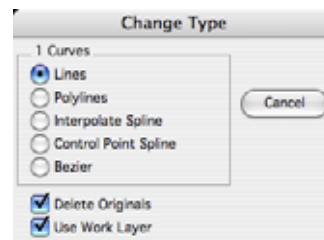
Generally when you convert an entity to another entity, any associativity is destroyed. The exception is when you change a curve to another curve type.



For Solids



For Surfaces



For Curves

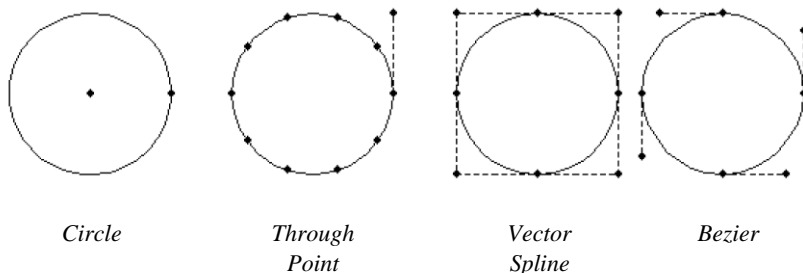
Change Curve

Selecting a curve (line, spline, arc, circle, ellipse or conic) with the Change Object Type will provide the below type conversion options:

Lines	Converts the curve's display list into individual line entities.
PolyLines	Converts the curve's display list into a single polyline with many straight segments.
Through Spline	Refits the selected curve using a fitting algorithm with a through point spline.
Vector Spline	Converts the curve into a NURBS, and exposes the control vertices through the show points command.

Bezier Spline Converts the curve into a NURBS and creates a Bezier spline from the resulting NURBS.

In the example below, a circle is converted into a through point spline, vector spline, and Bezier spline.



The Convert Curve command is also useful for minimizing over constrained spline curves (splines with too many control points). In the example below, an airfoil with 50 points was reduced to five with the Convert command, while maintaining a tolerance of 0.001 inches.



Minimize Curve

Change Surface

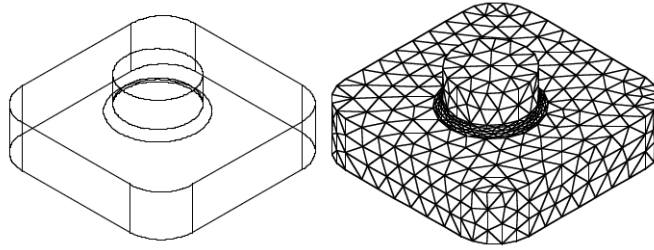
Selecting a surface with the Change Object type will provide the below type conversion options:

Curve	Converts the surface exterior edges into individual curve entities. If iso lines are shown, those are converted to curve entities.
Surface	Removes associativity with the surface.
Mesh	Converts the surface into a faceted mesh object.

Change Solid

The Change Solid can also be used to convert solids into other entity types. You can convert a solid into a collection of curves, surfaces or facet entity types. Changing a solid to a mesh will display a dialog box with options that control the mesh generation parameters. These options include:

Aspect Ratio	Ratio of triangle edges.
Surface Deviation	Maximum deviation of surface to facet.
Normal Deviation	Maximum deviation between normals.
Maximum Edge	Maximum edge length for a facet.



Explode Solid as Mesh

Simplify

The Simplify Object tool 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 NURBS (Non-Uniform Rational BSpline) surfaces or solids to analytics.

Potential NURBS shapes converted to analytic shapes include cylinders, cones, spheres, tori, and planes. Candidate shapes for simplification include data imported from IGES, STEP, Rhino, and Alias. In addition, some surfaces created in Punch! Shark FX, such as skins, are initially created as NURBS and may simplify to an analytic shape.

There are many advantages of analytic representation of shapes over NURBS. Analytic operations are typically considered exact, whereas NURBS are precise. Analytic operations execute much faster internally and take up less memory in RAM and less space when saved to a file. For example, an IGES file was reduced by half after the Simplify tool was executed.

Show Points

The Show Points command will show all editable points on an entity.

Resolve Links

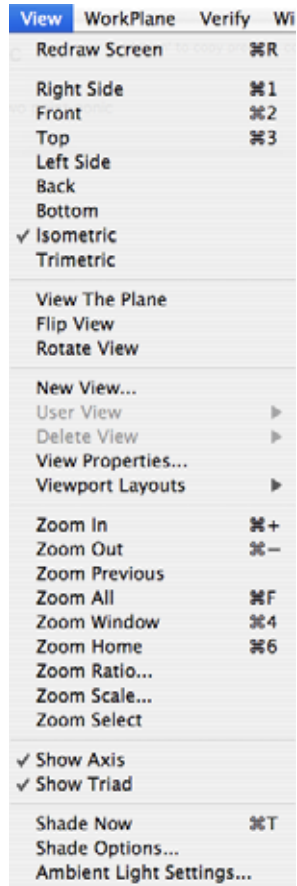
The Resolve Links option examines the current file for any unresolved links and resolves them.

Remove Links

This option takes the selected objects and removes all associativity related to the object.

View

The View menu contains commands for changing and creating view orientations, zooming, and assigning OpenGL settings.



Redraw Screen

Refreshes the drawing window.



Right Side

The Right Side option will change the view orientation to view the y, z plane looking down the x axis. The view normal for this orientation is $x = 1, y = 0, z = 0$.



Front

The Front option will change the view orientation to view the x, z plane looking down the y axis. The view normal for this orientation is $x = 0, y = -1, z = 0$.



Top

The Top option will change the view orientation to view the x, y plane looking down the z axis. The view normal for this orientation is $x = 1, y = 0, z = 0$.



Left Side

The Left Side option will change the view orientation to view the y, z plane looking down the x axis. The view normal for this orientation is $x = -1, y = 0, z = 0$.



Back

The Back option will change the view orientation to view the x, z plane looking down the y axis. The view normal for this orientation is $x = 0, y = 1, z = 0$.



Bottom

The Bottom option will change the view orientation to view the x, y plane looking down the z axis. The view normal for this orientation is $x = 0, y = 0, z = -1$.



Isometric

The Isometric option will change the view orientation to view the x, y plane looking down the z axis. The view normal for this orientation is $x = .577, y = -.577, z = .577$.



Trimetric

The Trimetric option will change the view orientation to view the x, y plane looking down the z axis. The view normal for this orientation is $x = .76$, $y = -.512$, $z = .39$.

Navigator

The Navigator is a viewing tool independent of your drawing. The Navigator has two viewing modes: Sphere Mode and Step Mode. To toggle between the two modes, click the Step Mode or Sphere Mode button in the top right corner of the dialog.



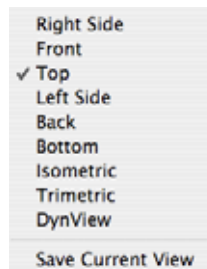
Sphere Mode

Step Mode

Navigator Modes

Views

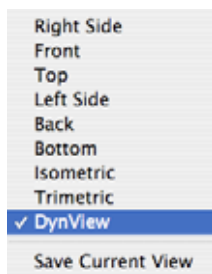
The Navigator dialog's Views drop-down is available while in Sphere Mode, from the bottom right corner. It allows you to quickly choose one of the 8 saved views or choose DynView to drag the Navigator and set your view.



Views drop-down

DynView

To activate DynView, use the Navigator to change your orientation. DynView also appears on the View drop-down. Each mode has rotation options which control the type of rotation and the origin of rotation.



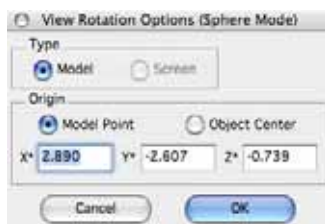
DynView - Drop-down

Using the DynView setting you can manually control your view, either by dragging the Navigator in Sphere Mode or by clicking stepper controls to change your orientation in Step Mode.

To auto-rotate, click the Steps in the center of the Navigator while in Step Mode, then choose a direction. Your view automatically rotates in that direction until you click the Steps again.

Sphere Mode Rotation Options

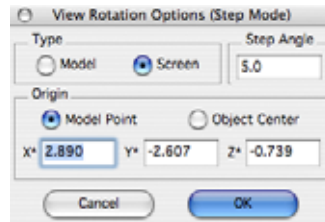
In the top left corner of the Navigator dialog, click the Rotation Options button.



In Sphere Mode, Model is the only rotation type available. You can select your rotation's point of origin by selecting either Model Point or Object Center. Using Model Point, you select a location on the design window or enter coordinate values in the text fields. Using Object Center, you select the object you want to use as your origin of rotation.

Step Mode Rotation Options

In the top left corner of the ViewBall dialog, click the Rotation Options button.



In Step Mode, you can select the rotation type, either Model or Screen. Model axis use the red/green/blue x, y, and z axis. Screen rotations are horizontal and vertical, relative to the monitor.

With Step Angle, you can define the increment that the object will rotate when you press the arrow keys.

Using Model Point, you select a location on the design window or enter coordinate values in the text fields. Using Object Center, you select the object you want to use as your origin of rotation.

Save View

You can save views using the Navigator. From the Views drop-down, choose Save Current View. For more information, see “New View”, which begins on page 123.

View The Plane

This option will change the view to that associated with the current work plane. The eye point is taken 500 units out along the work plane normal. The reference point is taken as the work plane origin.

Flip View

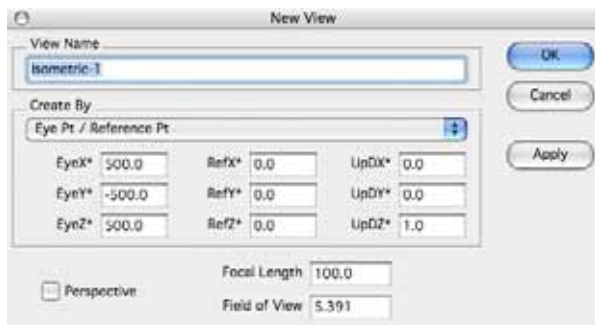
The Flip View command will change the current view by flipping the eyepoint to the other side of the view plane. Internally, the view normal changes the sign of its direction. For example, flipping the top view will yield the bottom view and flipping the front view will yield the back view.

Rotate View

The Rotate View command rotates the eye point 180 degrees about the up vector.

New View

The New View option provides a means to create custom user defined views. Options are described below:



ViewName Assigns a user specified name to the particular view. This name appears in the drop-downs that identify the view.

Create By A pull down menu that determines the method for specifying the eye point, reference point, and up vector. Methods exist for specifying these values as:

Eye Pt Reference Pt
Azimuth/Elevation
Rotation relative to current view

Perspective Sets a flag indicating whether this view has Perspective enabled.

Field of View The Field of View entry is a value that represents the angular value associated with the eye point, reference point, and viewable area. Small fields of view imply you are far away from an object.

Focal Length The Focal Length is the distance between the eye point and reference point. This value is only used in perspective mode.

User Views

The User Views Menu option provides a means to access user-defined views. When selected, a submenu will appear with all the user-defined views. Simply pick one and the current view is changed to the selected view.

Delete View

The Delete View command is used to delete user defined views. Select the view to delete from the submenu and the view will be permanently purged from the file.

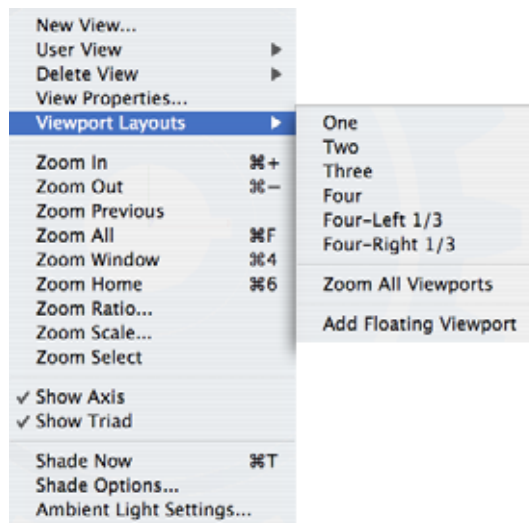
Tip: There is no undo for this operation.

View Properties

The View Properties dialog box will display all relevant information associated with the current view. In the case of user defined views you can use this dialog to modify values. The dialog displayed is the same used in creating new views.

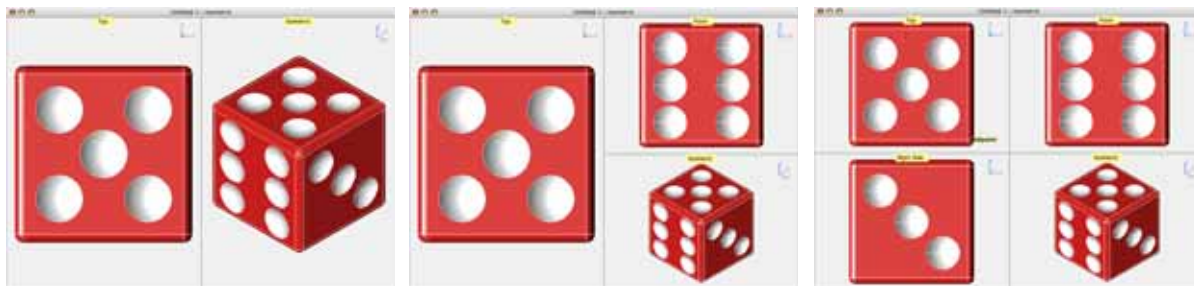
Multiple Viewports

Viewports allow you to display multiple views of your drawing at one time, each with a different orientation. For example, you can view your design from the top, while also viewing the left side and an isometric view.



Viewport Layouts Menu

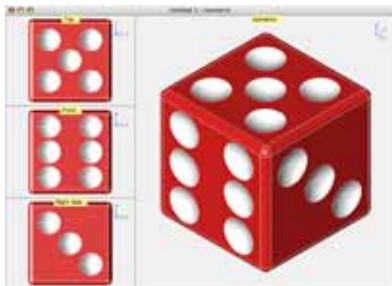
You can also see multiple views of your design simultaneously. In addition to the Viewport options available within the user interface, you can view a floating window, which you can move around your workspace.



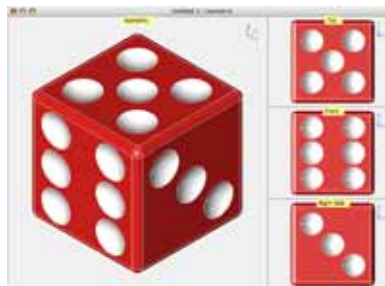
Two Viewports

Three Viewports

Four Viewports



Four-Left 1/3 Viewports



Four- Right 1/3 Viewports

Viewport Configurations

Viewing multiple Viewports

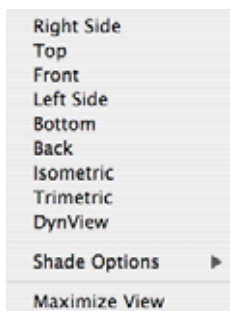
- 1 From the View menu, choose Viewport Layouts and, from the submenu, choose the Viewport configuration you want.
- 2 To change a Viewport's orientation, click the current view, at the top of the Viewport, and from the pop-up menu that appears, choose the view you want.

Viewport Sizing

As you move your cursor over a viewport's borders, the cursor will change indicating that it's possible to drag the border to reshape and resize the viewport.

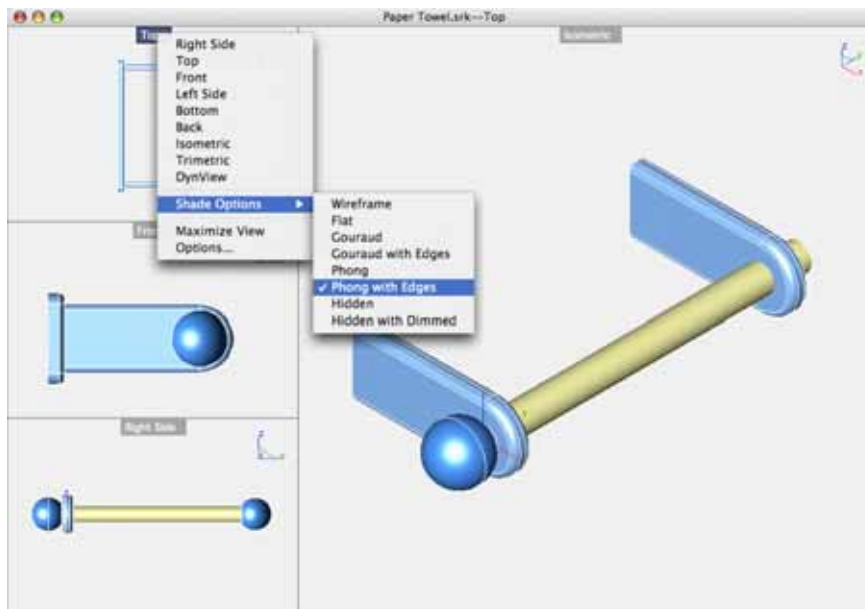
Shade Options

Clicking the label displays a drop-down with a list of options, including view orientations and shading.



View Orientations

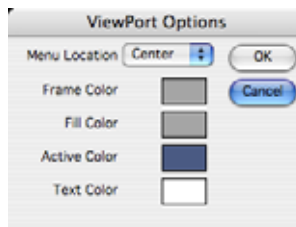
You can customize each view's shade options, so each orientation is easily recognizable.



Shade Options Menu

Viewport Options

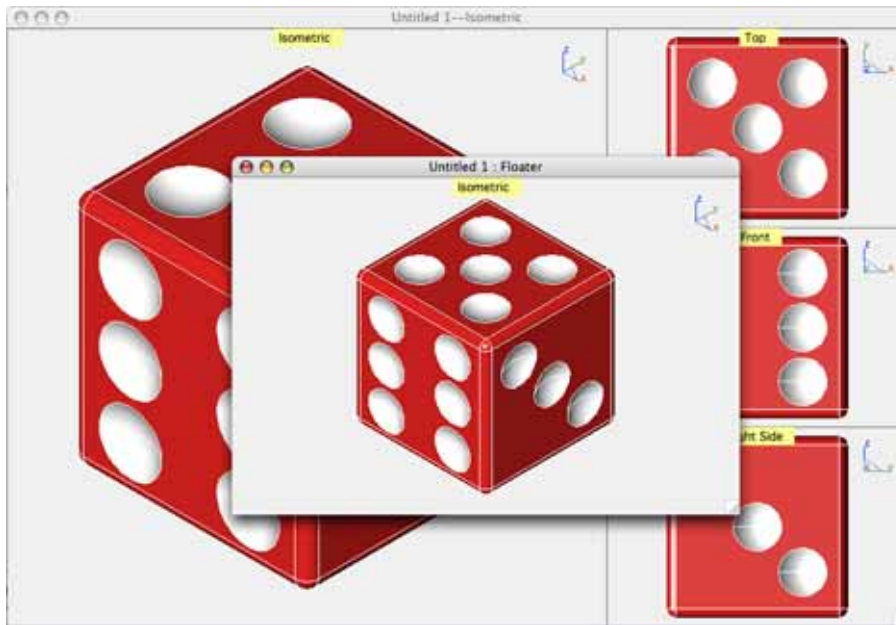
Each Viewport also has its own Viewport Options, which can be set to further customize each view. Click Options from the Viewport drop-down menu to display the Viewport Options dialog.



Here you can set the orientation of the Viewport menu, as it appears in the Viewport. You can also customize the colors associated with the Viewport.

Floating Viewport

A Floating Viewport is a viewport that is created in a new window. Floating Viewports can be used to support multiple monitors.



Floating Viewport Window

Opening a floating Viewport

- 1 From the View menu, choose Viewport Layouts and, from the submenu choose Add Floating Viewport.
- 2 To change a Viewport's orientation, click the current view, at the top of the Viewport, and from the pop-up menu that appears, choose the view you want.

Zoom In

The Zoom In command will enlarge the current view by 20 percent.

Zoom Out

The Zoom Out command will decrease the current view by 20 percent.

Zoom Previous

The Zoom Previous command will restore the zoom scale to the previous zoom setting.

Zoom All

This command calculates the extents of your drawing and zooms to a size that will fit the drawing in the current window. Extents are only checked for objects blanked on and in the current display layer set.

Zoom Window

Zoom Window prompts the user to define a rubber banding box which is used to describe the extents of a new window. This command will always enlarge the view, independent of how you start your banding box.

Zoom Home

Zooms to the default view used at start up. The default view scale is defined the *File:Preferences:General* controls.

Zoom Ratio

Scales the view up or down by a specified amount.

Zoom Scale

Zooms the view to a specific scale between window and model space.

Zoom Select

Zooms the view to the selected entities (includes edges and faces).

Show Axis

Turns on or off the model coordinate system axis located at 0,0,0.

Show Triad

Displays a small model axis and work plane in the upper right corner of the window. The triad is useful for orientation independent of view scale or position.

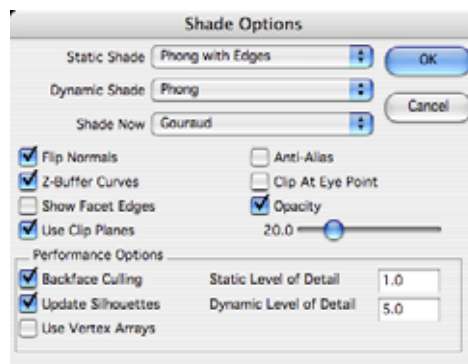


Shade Now

The Shade Now option renders the screen using the current OpenGL settings. This option is useful if you normally work in wireframe mode and need to occasionally see the data in rendered mode.

Shade Options

The Shade Options dialog provides controls for display settings related to OpenGL.

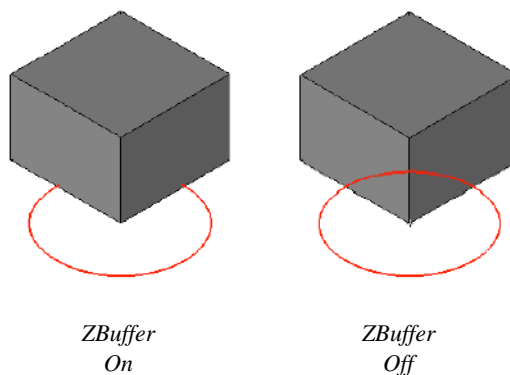


Flip Normals

When this option is checked, surface normals are automatically flipped such that surfaces pointing away from the viewer are not dark. Otherwise surfaces are displayed only with ambient light.

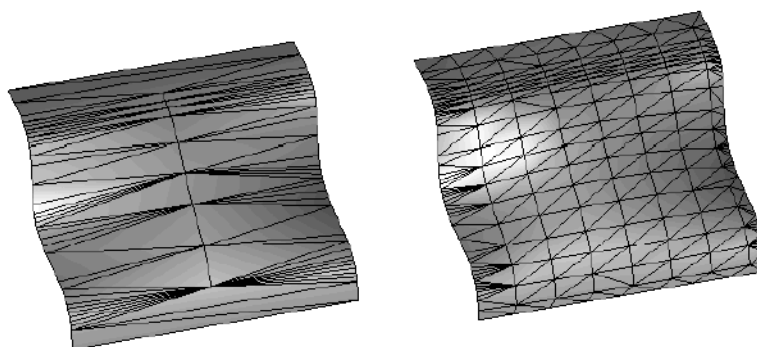
Z-Buffer Curves

This option turns on or off OpenGL depth buffering of curves as they are displayed in context with surfaces and solids. With this option on, curves behind surfaces or solids are hidden and visible when turned off.



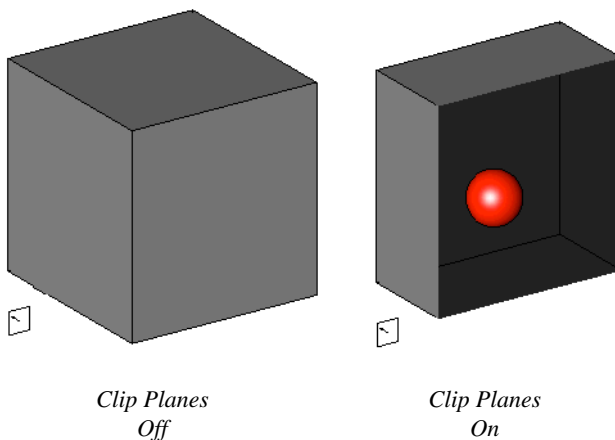
Show Facet Edges

The Show Facet Edges option turns on or off the edges of the display facets. The facet density is determined by the entity resolution.



Use Clip Planes

This option indicates whether to process clip planes for OpenGL rendering. When turned on, all planes entities that are marked as clip planes will clip the current view. To mark a plane as a clip plane, right click over the entity and select the appropriate option in the menu.



Anti-Alias

This option tells the OpenGL drivers to enable anti-aliasing of edges.

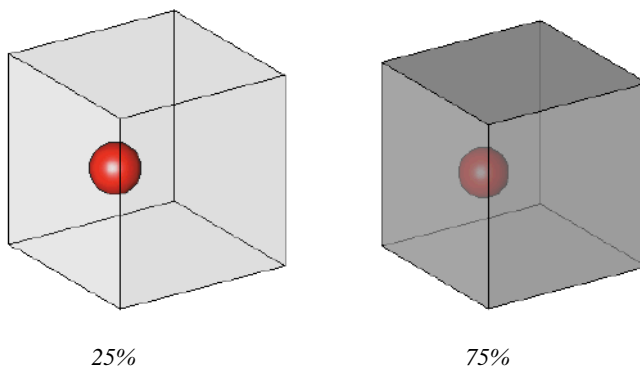
Clip At Eye Point

The Clip At Eye Point turns on or off clipping at the eye point. If off, clipping is automatically determined based on the extents of the given model.

Opacity

This option indicates whether to process objects marked with an opacity flag for OpenGL opacity. When turned on, all surface and solid entities that are marked as transparent will clip the current view. To mark an object as transparent, Control-click (right-click) an entity and select the transparent option from the menu.

The slider value indicates a global opacity setting. A setting of 100 implies no transparency and a value of 0 implies fully transparent.



Backface Culling

Backface Culling is an option that greatly 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 you have hybrid solids that mix open surfaces with solids, otherwise the surface may appear invisible in OpenGL display modes.

Update Silhouettes

Turns off the calculation of silhouette edges when the view orientation is changed. By default, this is on.

Use Vertex Arrays

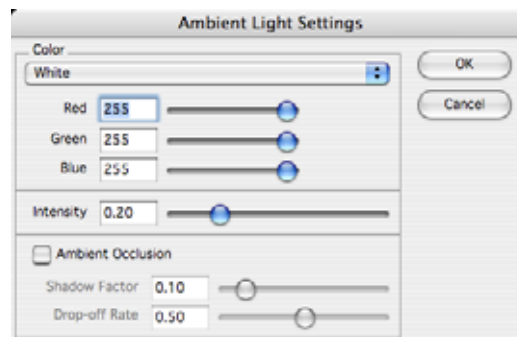
Uses the fastest path to OpenGL by sending vertex arrays to the graphics board. By default, this is on. This feature uses more RAM than when disabled.

Level of Detail

Objects that are under a certain size with respect to the viewing area are displayed as boxes. These values range between 0 and 100 percent of the screen viewing area. Specify 0 to turn this feature off. The Static Level of Detail applies to repaints, zooming, changing view type operations. The Dynamic Level of Detail applies to dynamic zooming, panning, and rotating.

Ambient Light Settings

This setting affects the color and intensity of ambient light to all objects in your displayed scene.



The Ambient Light dialog contains three sets of user controls described below:

Color

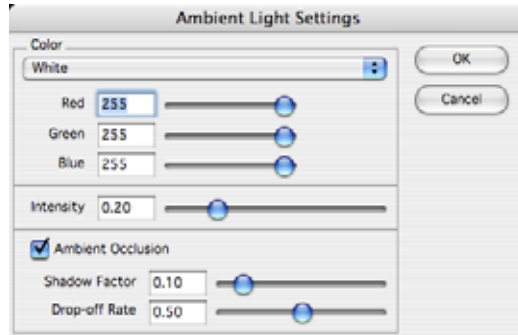
This control sets the color of the ambient light. You can adjust the color using Red, Green, and Blue sliders or select an existing color from the pull down menu.

Intensity

The Intensity Slider controls the strength of the ambient light. A zero turns off all ambient light. Values greater than 5 will wash out the model.

Ambient Occlusion

Ambient occlusion is a type of intelligent ambient lighting that takes into account the distance of the point being rendered to other objects in the scene. This distance is used to add shadows in areas such as beneath tables or in corners while leaving the rest of the room fully lit.

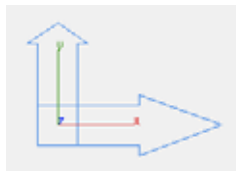


WorkPlanes

The WorkPlanes menu provides access to commands for creating, setting, and modifying the work plane. A work plane is an invisible flat surface used to define planar geometry. The work plane is also used to define the origination of the Z axis for the Snap tool.



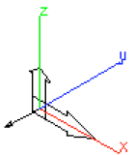
The current orientation of the work plane is displayed graphically with the *WorkPlane:Show Work Plane* command located in the Menu Bar. This shows a triangle representing the orientation with respect to the model axis.



Work Plane Symbol

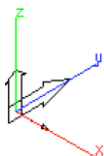
Global

The Global option sets the work plane to the global work plane. The global work plane is the fixed, uneditable coordinate system created by Punch! Shark FX that allows a uniform system between different files/models. This is equivalent to the world coordinate system.



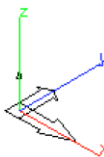
Front

The Front option changes the work plane to the front plane, which is parallel to the XZ axis.



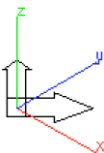
Side

The Side option changes the work plane to the side plane, which is parallel to the YZ axis.



Top

The Top option changes the work plane to the top plane, which is parallel to the XY axis.



Use View

The Use View option changes the work plane to use the view normal as the work plane z axis.

3 Pts

The 3 Pts command sets the work plane based on the input of three referenced xyz points. The three points are used to define the following information:

- Pt 1** Origin of the new work plane
- Pt 2** Direction of the x axis
- Pt 3** Direction of the y axis

The three point work plane command is very useful for creating oblique work planes (planes not ortho to the model coordinate system).

Pick Objects

The Pick Objects command sets the work plane based upon selecting curves, surfaces or faces. The example selections below will result in setting up a new work plane.

- 2 Lines The two non-parallel lines define a plane from which the new work plane is created. The origin is extracted from a point on the line.
- Circle The circle normal defines the work plane orientation. The user selected location on the circle defines the work plane origin.
- Face The face normal is used to calculate the work plane orientation. The origin is the user selected point on the circle.

Set Origin

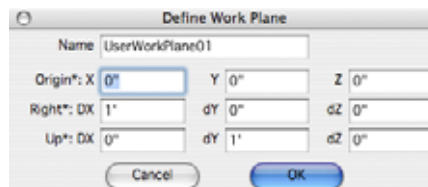
The Set Origin command moves the current work plane to the specified location.

Offset

The Offset command moves the work plane parallel to the current work plane xy axis.

New Work Plane

The New Work Plane command creates a work plane by dialog box field values.



Name Specifies the work plane name. This name is used in subsequent commands such as specifying a user work plane or deleting a work plane.

Origin Defines the work plane x, y, z origin.

Right Defines the x axis for the work plane.

Up Defines the y axis for the work plane.

The last three items have an asterisk, which means you can reference these values with the Snap tool in the drawing window.

User Work Plane

The User Work Plane option displays a menu listing all the user defined work planes. Pick the desired work plane and this will set the current work plane to the selected item.

Modify Work Plane

The Modify Work Plane command displays a menu listing all of the user defined work planes. Selecting a name will display the dialog box used to create a new work plane.

Delete Work Plane

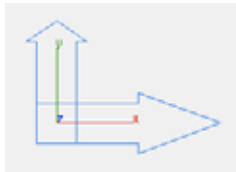
The Delete Work Plane command will display a menu list of all user defined work planes. Select the item you desire to delete. This operation is not undoable.

Show Grid

The Show Grid command turns on or off the grid. For a full description of the grid parameters see the discussion for the grid in the File:Preferences:Grid portion of this document.

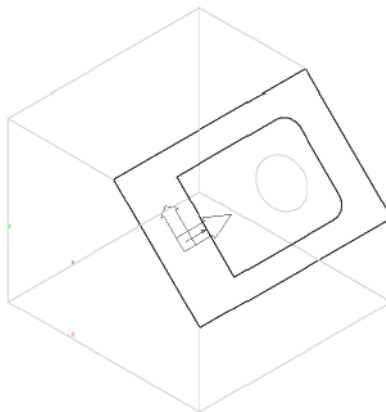
Show Work Plane

The Show Work Plane command toggles the work plane symbol on and off.



Show Dimming

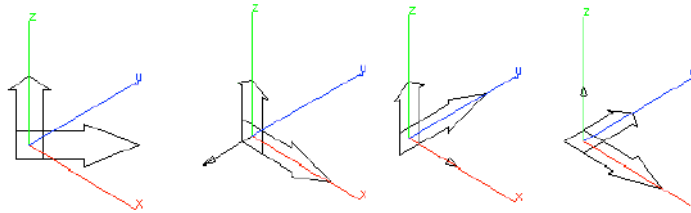
The Show Dimming command provides a means to dim objects that do not lie in the work plane. This tool aids in visually identifying objects in or out of the work plane. This command is a toggle command that turns on or off this feature.



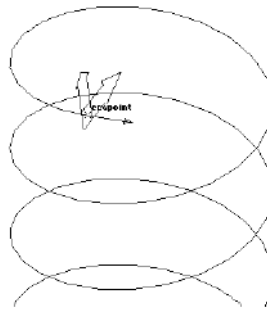
Dimming and Work Planes

Work Plane Snapping

A little known shortcut for changing the work plane is accomplished by pressing the ‘c’ key at an identified snap location. Repeatedly pressing the ‘c’ key will toggle the work plane to different orientations at the snap location.



Punch! Shark FX goes one step further with work plane snapping with the addition of aligning the work plane to be perpendicular to a curve. The work plane will snap perpendicular to a curve if you have a snap identified as either on, end, vertex or midpoint.



Work Plane Snapping to Curve

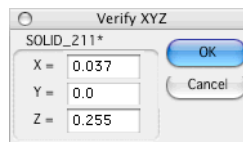
Verify

The Verify Menu option displays a collection of commands that allow you to verify geometry and data associated with the current file.



X,Y,Z...

The Verify XYZ option displays a dialog box with the selected point model space coordinate displayed within the fields.



Tip: The Verify XYZ dialog can also be used to modify the selected point.

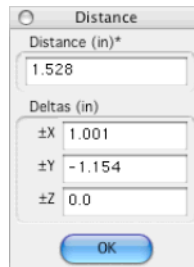
Angle 3 Pts

The Angle 3 Pts dialog is used to display the angle between three user supplied points.



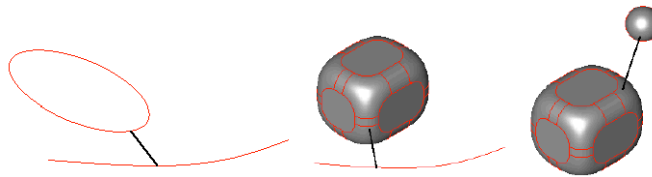
Dist Pt-Pt

The Dist Pt-Pt command is used to verify global coordinate distances. The dialog displays the distance in the current unit value and breaks down the distance into its dx, dy, and dz components.



Minimum Distance

The Minimum Distance command is used to calculate the minimum distance between two entities. Acceptable paired entities include curve/curve, curve/surface, surface/surface, curve/solid, surface/solid, solid/solid.



Length

The Verify Length command calculates the length of a curve.

Perimeter

Objects that are selected will be used to calculate the perimeter. Objects must be selected prior to selecting the Perimeter option from the menu.



Area

The Verify Area command calculates the area of a surface or solid. In the case of a solid, the area is the sum of the individual face areas.

Volume

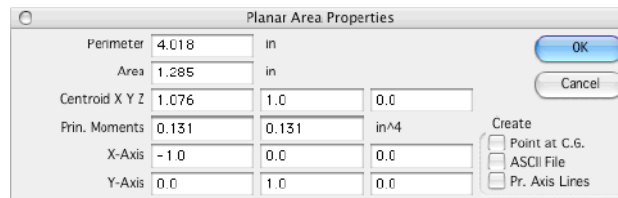
The Volume command prompts a dialog which displays the amount of three-dimensional space a selection occupies.

Properties

The Properties command in the Verify menu calculates area and mass properties.

Planar Properties

Planar properties are calculated when either curves or surfaces are selected. When selecting curves, the command requires they form a closed shape without overlapping elements.



Perimeter Sums the length of all the curves or surface edges.

Area Calculates the 2D planar area or 3D surface area.

Centroid Calculates the 2D area centroid. This value is not calculated for a non-planar surface.

Prin. Moments Calculates the principle moments of inertia for a 2D planar object. This value is not calculated for a non-planar object.

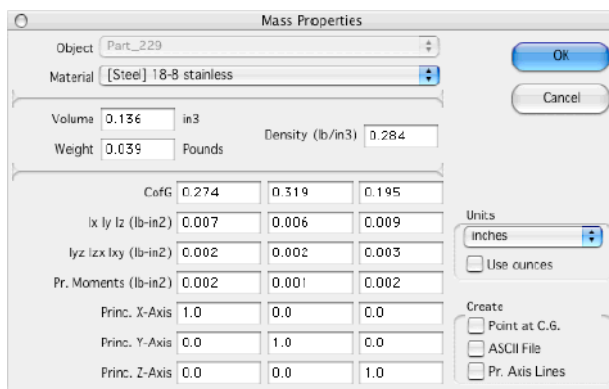
- X-Axis** Calculates the principle x axis of a 2D planar object. This value is not calculated for a non-planar object.
- Y-Axis** Calculates the principle y axis of a 2D planar object. This value is not calculated for a non-planar object.
- Point at CG** Creates a point entity at the centroid for 2D planar objects.
- ASCII** Dumps the results into a text ASCII file.
- Pr Axis Lines** Creates two lines corresponding to the principle axis lines. This object is only valid for 2D planar objects.

Mass Properties

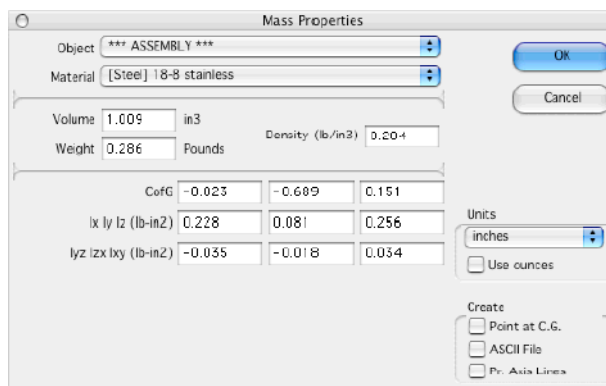
Selecting one or more solid objects will display the Mass Properties dialog. The mass properties information for a solid includes volume, weight, Density, CG (Center of Gravity), inertia, and Moments data. The mass properties command will accept more than one object selected. If more than one object is selected, the dialog will calculate the results as a combined assembly.

Using the Mass Properties Command

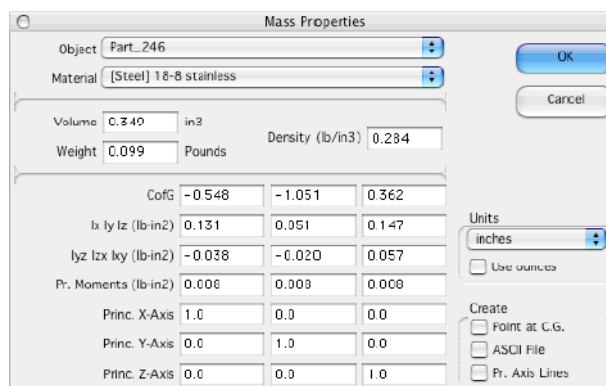
Select one or more solids. All selected objects will be used for the assembly calculations. Pick the *Verify:Properties* menu item to display the Mass Properties dialog box. If a single object is selected, the Object drop-down list is disabled and the object's values are displayed.



If 2 or more objects are selected, the Object drop-down list is enabled and the assembly's values are displayed.



Making a selection from the Object drop-down list changes the current object.



The current object is highlighted in the drawing. If the current object is the assembly, all the assembly's objects are highlighted.

An object's density value may be changed at any time using the Material drop-down list or the Density edit field. Changing the density of the assembly will change the density of *all* objects in the assembly. All computed values are updated as density changes are made.

The displayed units may be changed at any time using the Units drop-down list or the Use ounces/grams checkbox. All computed values are updated as unit changes are made.

Selecting the Create:Point at C.G. checkbox enables the creation of a point entity at the center of gravity of each object in the assembly and at the center of gravity of the assembly itself (if applicable).

Selecting the Create:ASCII File checkbox enables the creation of an ASCII dump file containing the mass property values for each object in the assembly and for the assembly itself (if applicable).

Selecting the Create:Pr. Axis Lines checkbox enables the creation of a principle axis triad (3 line entities) at the center of gravity of each object in the assembly and at the center of gravity of the assembly itself (if applicable).

Pressing the OK button will save the density values for each object and perform any selected Create options. Pressing the Cancel button will discard any density changes and ignore any selected Create options.

Example Mass Properties ASCII File

Mass Properties for *** ASSEMBLY ***

Volume: 134.18740 in3
Weight: 0.00000 Pounds
C.G.: 0.00000 0.00000 0.00000
Ix Iy Iz: 0.00000 0.00000 0.00000 (lb-in2)
Iyz Izx Ixy: 0.00000 0.00000 0.00000 (lb-in2)

Mass Properties for MyBlock

Volume: 50.96510 in3
Weight: 0.00000 Pounds
C.G.: -0.65655 6.20481 1.47284

Ix Iy Iz: 0.00000 0.00000 0.00000 (lb-in2)
Iyz Izx Ixy: 0.00000 0.00000 0.00000 (lb-in2)

Pr. Moments: 0.00000 0.00000 0.00000 (lb-in2)
Pr. X Axis: 1.00000 0.00000 0.00000
Pr. Y Axis: 0.00000 1.00000 0.00000
Pr. Z Axis: 0.00000 0.00000 1.00000

Mass Properties for My Other Block

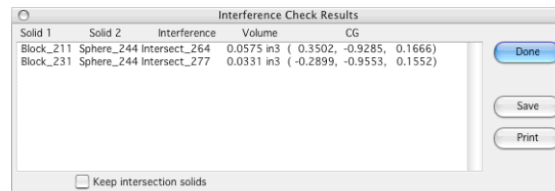
Volume: 83.22229 in3
Weight: 0.00000 Pounds
C.G.: 5.08555 9.38822 4.37342

Ix Iy Iz: 0.00000 0.00000 0.00000 (lb-in2)
Iyz Izx Ixy: 0.00000 0.00000 0.00000 (lb-in2)

Pr. Moments: 0.00000 0.00000 0.00000 (lb-in2)
Pr. X Axis: 1.00000 0.00000 0.00000
Pr. Y Axis: 0.00000 1.00000 0.00000
Pr. Z Axis: 0.00000 0.00000 1.00000

Interference

The Interference command checks to see if any two solids overlap. You can pick two or more solids to check interference between.



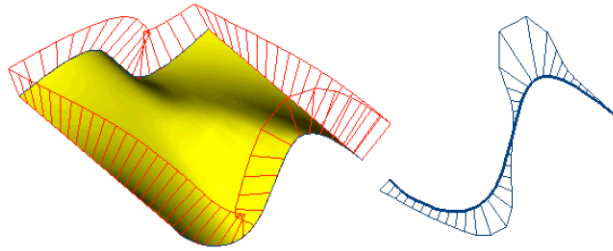
- Solid 1** The entity name of the first solid analyzed.
- Solid 2** The entity name of the next solid analyzed.
- Interference** The entity name corresponding to the solid that will get created if Keep intersection solid is checked.
- Volume** The volume of interference between Solid 1 and Solid 2.
- CG** The centroid of interference between Solid 1 and Solid 2.
- Keep Intersection** Keeps the solid of intersection between Solid 1 and Solid 2.
- Print** Sends the results to the printer.
- Save** Saves the results to a text file.

Show Direction

The Show Direction command displays the starting direction of a curve and the positive direction for a surface. When you show the direction on an object, it will stay shown until you turn it off with the Show Direction command.

Show Curvature

The Show Curvature displays the curvature of a curve or surface of the selected object. It will stay shown until you turn it off with the same command. By keeping on the curvature display, you can perform dynamic modifications of the curve and immediately see the effect of curvature.



Surface and Curve Curvature Plot

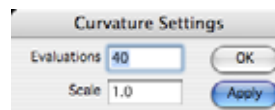
Curvature Circle

The Curvature Circle option creates a dynamic circle that is located next to the cursor and selected curve. The curvature circle is the circle that best approximates a circle at the location along the selected curve. As you move your cursor over the selected curve, the circle center and diameter update, including the Data Entry Fields. To end the command, press the mouse button. A circle is created at the cursor location. You can use the Undo function to remove the circle.



Curvature Settings

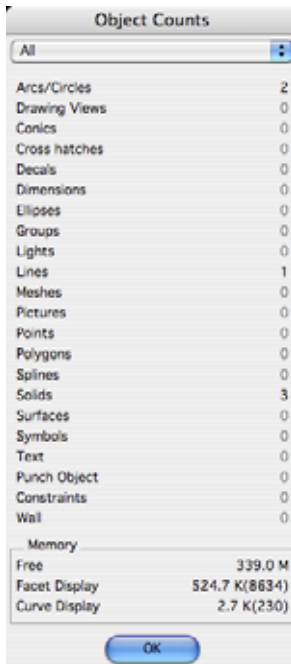
The Curvature Settings dialog box is located in the Verify menu. The dialog box controls the number of evaluations used to display curvature and a scale factor useful for increasing or decreasing the magnitude of the curvature.



Curvature Settings Dialog

Object Counts

The Verify Object Counts displays a dialog box that helps you examine object counts in your drawing. The pull down menu on the top allows for calculating counts for all, selected, visible, work layer only, and hidden objects. In addition, the Object Counts dialog box displays information related to memory and the display list.

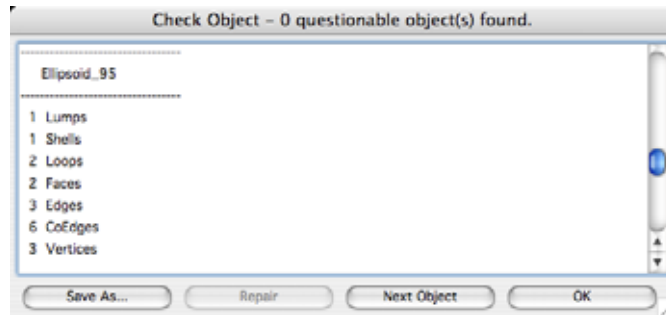


Check Object

The Check Object command examines 1 or more objects for potential problems. Potential problems include geometry and topological errors. Some example conditions examined include:

- Improper tolerances between vertices, edges, and faces (gaps).
- Improper orientations of edges and faces.
- Curve or surface discontinuities.
- Self-intersecting geometry.
- Bad representations of analytic or NURBS data.
- Incorrect trimming curve information.

Many of these conditions arise from data that is imported from other modeling systems. It is a good idea to use check object on imported data.



Save As Saves the check object dialog ASCII content to a text file.

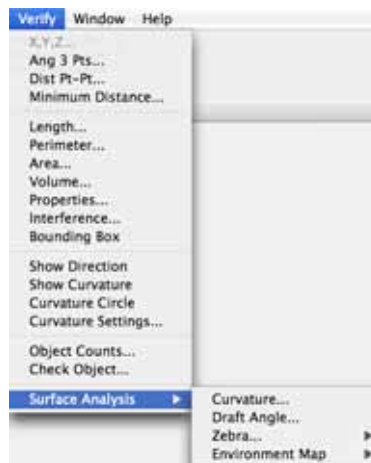
Repair Attempts to repair data that is determined bad.

Next Object Moves the scroll bar to the data associated with the next object.

Tip: Use the Stitch command as an alternative to the repair button. The stitch button will work on a solid and provides access to a variety of healing and repair options not available in the Check Object repair button.

Surface Analysis

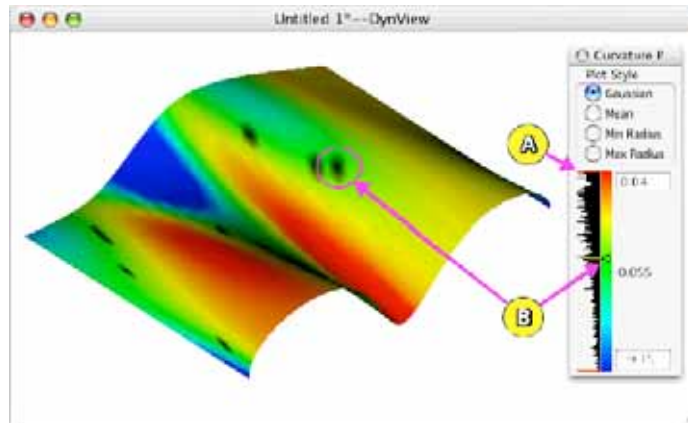
The Surface Analysis command displays a menu with four options that include curvature, draft angle, normals, and zebra plots.



Surface Analysis Menu

Curvature

The Curvature Analysis displays color analysis for Gaussian, Mean, Min Radius, and Max Radius of curvature.

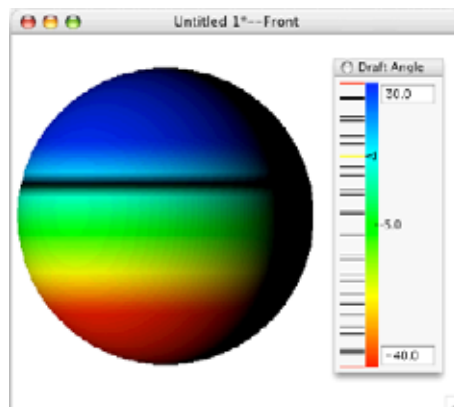


Gaussian Curvature

The “A” call-out points to a Red overloaded histogram bin. This occurs when the user edits the min/max values such that the histogram scale becomes too small. The “B” call-out points to the user selected “value of interest”. The user selects the color by clicking with the left mouse button on the color bar. The surface colors will change to black at the specified value. Also, the corresponding histogram bin will change to yellow at this location. If the user has edited the min/max values, selecting a radio button will reset the values back to the defaults.

Draft Angle

The Draft Angle Plot provides mold interference checking. The user can specify the minimum and maximum draft angle limits. This dialog also supports a user selected “value of interest”.



Draft Angle Plot

Zebra

The Surface Zebra Plot provides a quick check of a surface's smoothness. The view eye point will affect a zebra plot. A zebra plot maps stripes onto a surface or solid, and several stripe options exist including: horizontal, vertical, wide, and narrow.



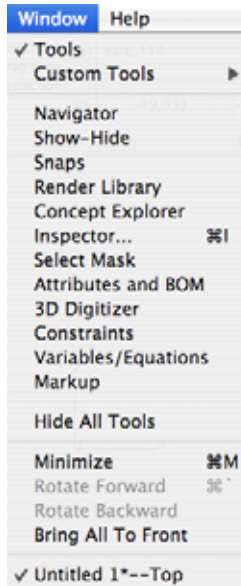
Environment Map

An Environment Map applies an image mapped to an object. Like zebra plots, environment maps are useful to find inconsistencies with surfaces, especially at the location where two surfaces join together.



Window

The Window Menu Bar provides access to commands that display tool palettes, non-modal dialog tools, and window management commands.



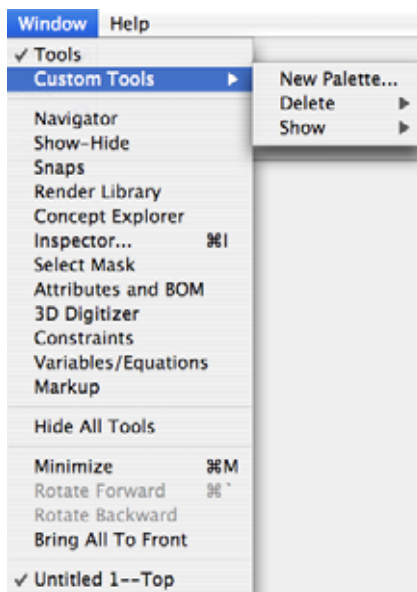
Tools

The Tools option will toggle on or off the tools palette. The Tools Menu is the main tool palette for the application as it contains selection, basic curve creation and modifications, annotations, and transformations.



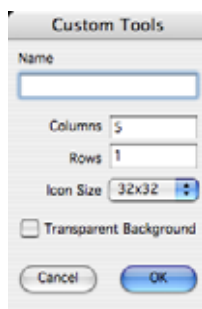
Custom Tools

The Custom Tool palette options provide a means to create user defined tool palettes.



New Palette

This option allows you to create a custom palette by setting the number of columns and rows, as well as the icon's size and even whether or not the palette is transparent.



Delete/Show

The Delete and Show options allow you to delete palettes you've created and turn on or off their visibility.

Navigator

The Navigator is a viewing tool independent of your drawing. The Navigator has two viewing modes: Sphere Mode and Step Mode. Sphere Mode allows you to drag the Navigator to control your orientation.

Step Mode allows you to click stepper controls to control your orientation. In Step Mode, you can also choose the auto-rotate option. For more information, see “Navigator”, which begins on page 121.

Show-Hide

The Show-Hide tools help manage the display of your drawing. Show-Hide will turn on or off the display attribute of an object. The result of a show-hide command is different from a delete command. The latter removes the object from the database, whereas a show-hide only affects an object’s display/select status. There are five options for show-hide:



Show

Turns on the display of hidden selected objects. (Requires that you know the area where they are hidden).

Show All

Turns on all hidden objects in the current displayed layers.

Show Only

Turns on selected objects for display and turns everything else off.

Hide

Turns off the display of selected objects.

Invert

Turns off all displayed objects and turns on all blanked off objects.

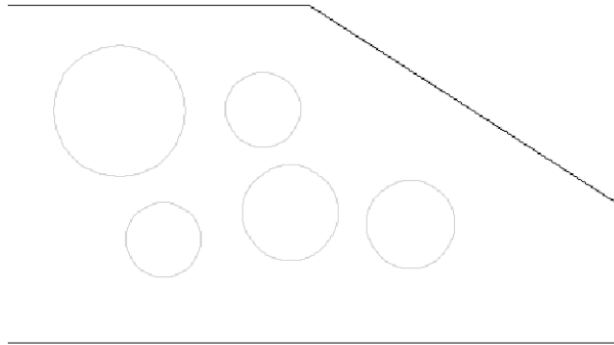
Show Parents

Turns on parents of the selected objects.

Hide Parents

Turns off parents of the selected objects.

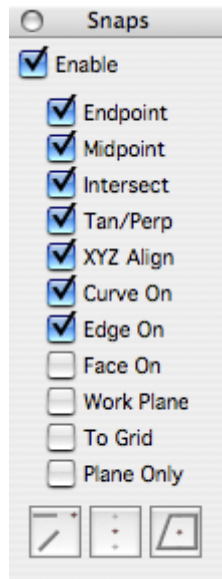
The Show/Hide Tool palette supports preselected objects and a one-time operation. This means after using one these tools, you return to the previous tool. To continue within the tool, press the Shift key. In addition, the Show option was modified to display hidden entities in light grey so that you can see the objects you want to convert from hidden to displayed.



Shows Hidden In Light Grey

Snaps

The Snap dialog displays a list of filters used for the Snap tool.



Enable	Turns off the Snap tool but preserves the settings of the individual snap masks.
Endpoint	Enable snapping to curve endpoints.
Midpoint	Enable snapping to curve endpoints and circle centers.
Intersect	Enable snapping to curve intersections.
Tan/Perp	Enable tangent and perpendicular alignments.
XYZ Align	Enable model axis alignments.

Curve On	Enable projections to nearest point on curve.
Edge On	Enable projections to nearest point on surface or solid edge.
Face On	Enable projections onto surfaces and solid faces.
Work Plane	Project final snap data to work plane.
To Grid	Snap to nearest grid point.
Plane Only	The Snap tool will only see a snap point IF that point is in the current work plane. Enabling this option, in essence, turns off the z-axis alignments in relation to the current work plane.
Intersection Snap	This option calculates the projected intersection of two curves.
Midpoint Snap	This option calculates the midpoint of two user specified locations.
Centroid Snap	This option calculates the centroid of a user selected object.

Render Library

The Render Library item will display a dialog box that is useful for dragging and dropping materials onto objects. For more information, see “Render Library”, which begins on page 479.

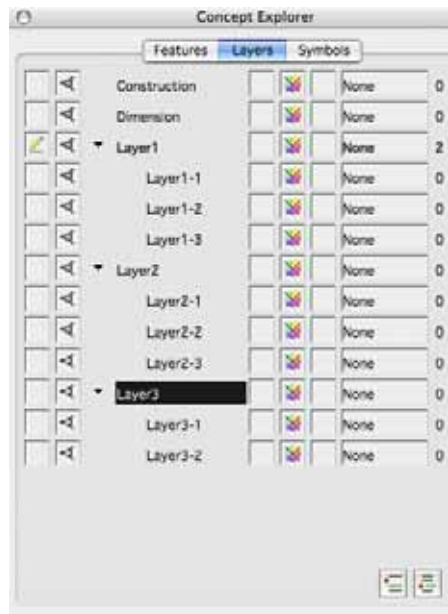
Concept Explorer

Concept Explorer provides a user interface to access commands for the layer manager, feature tree, and a symbol manager for conveniently dragging and dropping symbols and parts onto your drawing. Layers provide a means to organize entities into displayable levels and the feature tree provides a means to examine and modify associativity, history, and dependencies between objects.

Layer Manager Tab

Layers are a powerful tool for organizing related groups of drawing objects. For example, a user may want to separate the dimension information from the rest of the model. This can be accomplished by using layers. Layers are not new to designers, it is a practice that has been going on long before computer aided design systems. Designers took advantage of drawing on transparent plastic sheets to separate related groups of drawing objects. Designers could then mix and organize these sheets for different presentation purposes. When computers came along, this great idea was designed into most CAD systems.

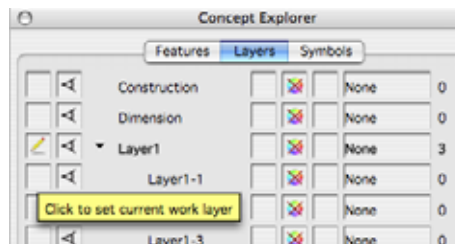
Punch! Shark FX exposes layers through the Layer Manager tab in Concept Explorer. For each new drawing, the Construction, Dimension, and Layer1 layers are created automatically. A maximum of 1200 layers can be created.



A model object can reside in only one layer at a time. Each layer has several attributes that can be used to control the behavior of newly created or assigned objects. These attributes are discussed below. Where appropriate, sub-layers inherit dominant attribute(s) from their parent layer(s). The dominant attributes are: Hidden, Locked, and Color.

Work Layer

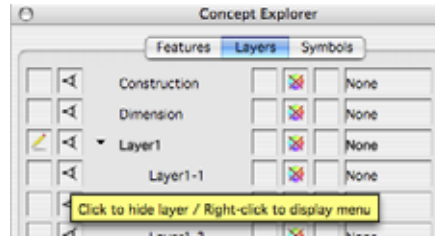
The first vertical column from the left indicates which layer is the current Work Layer. All newly created objects are automatically placed in the Work Layer. The pencil icon (✎) indicates the current Work Layer. Left-clicking on an empty cell or double-clicking on the layer name will change the Work Layer.



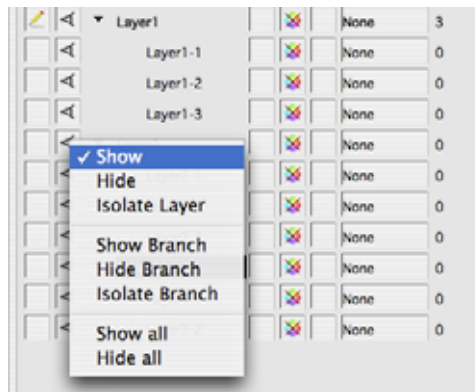
Visible/Hidden

The second vertical column from the left indicates which layers are visible or hidden. If a layer is visible, all objects in that layer are visible in the drawing. If a layer is hidden, all objects in that layer (and sub-layers) are *not* visible in the drawing. Visible layers display the eye icon (☞). Any sub-layers that are

hidden due to a parent override will display the dimmed eye icon (◑). If a layer is hidden, no icon is displayed. Left-clicking on a cell will toggle the visibility of the layer.



Control-click (right-click) cell to display the visibility pop-up menu. The layer's current visibility status is indicated by the check mark.



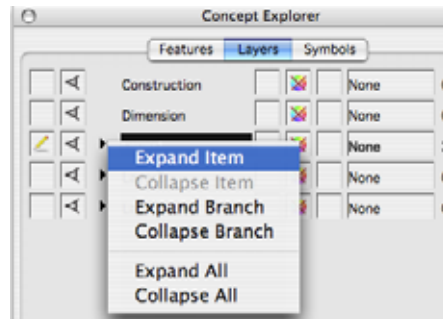
- | | |
|-----------------------|---|
| Show | Makes the selected layer visible. |
| Hide | Makes the selected layer hidden. |
| Isolate Layer | Makes the selected layer the Work Layer and hides all other layers. |
| Show Branch | Makes the selected layer and all its sub-layers visible. |
| Hide Branch | Makes the selected layer and all its sub-layers hidden. |
| Isolate Branch | Makes the selected layer the Work Layer and all of its sub-layers visible. All other layers are hidden. |
| Show all | Makes all layers and sub-layers visible. |
| Hide all | Makes all layers and sub-layers hidden. |

Name

The central area displays the layer tree hierarchy using a series of branches and layer names. If a layer has no sub-layers, the terminal node indicator (▶) is displayed.

If a layer has undisplayed sub-layers, the closed node indicator (▶) is displayed. Double-clicking on the closed node indicator will expand this layer and all sub-layers (see Expand Branch menu option below).

If a layer has displayed sub-layers, the open node indicator (◀) is displayed. Double-clicking on the open node indicator will close this layer and all sub-layers (see Collapse Branch menu option below). Left-clicking on a node indicator will toggle the display of sub-layers. Right-click on a node to display the tree menu.



Expand Item Displays the selected layer's immediate subtree. Pressing the + key is a shortcut for this option.

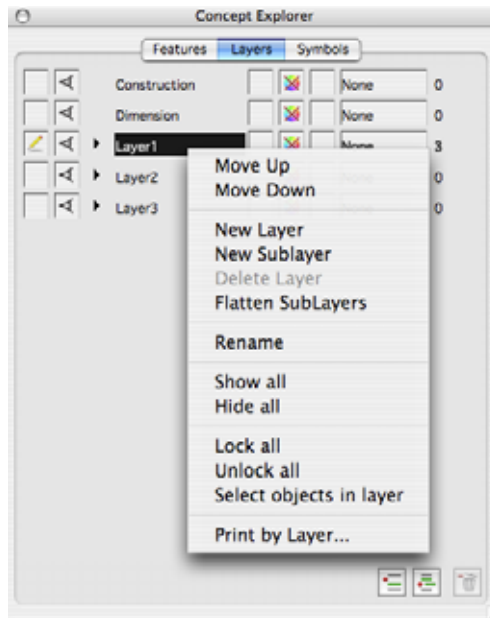
Collapse Item Closes the selected layer's immediate subtree. Pressing the - key is a shortcut for this option.

Expand Branch Displays the selected layer's complete subtree. Pressing the Ctrl+ key is a shortcut for this option.

Collapse Branch Closes the selected layer's complete subtree. Pressing the Ctrl- key is a shortcut for this option.

Expand All Displays branches for all layers and sub-layers.

Collapse All Closes branches for all layers and sub-layers. Right-click on a name to display the layer pop-up menu.



Drag to reorder To drag a layer to a new position, hold down the Shift key and drag the layer to the position you want.

Move Up Moves the selected layer up one position within its current hierarchy level.

Move Down Moves the selected layer down one position within its current hierarchy level.

New Layer Creates a new top-level layer.

New Sublayer Creates a new layer as a sub-layer under the selected layer.

Delete Layer Deletes the selected layer and its sub-layers. If any entities exist in this branch, the delete will be confirmed. This option is disabled if the Work Layer is set within this branch.

Flatten SubLayer Moves entities from existing sublayer(s) to the main layer, removing the sublayer.

Rename Activates the in-place edit of the layer's name. Clicking on the name of the selected layer also activates the in-place edit.

Show all Makes all layers and sub-layers visible.

Hide all Makes all layers and sub-layers hidden.

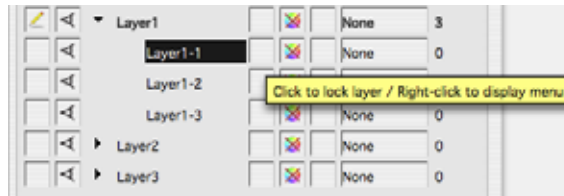
Lock all Makes all layers and sub-layers locked.

Unlock all Makes all layers and sub-layers unlocked.

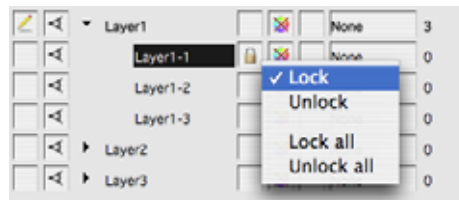
Select Objects in Layer Selects any object(s) that exist on that particular layer.

Locked/Unlocked

The vertical column just to the right of the names indicates which layers are locked or unlocked. If a layer is locked, all objects in that layer (and sub-layers) cannot be selected. If a layer is unlocked, all objects in that layer are eligible for selecting. Locked layers display the lock icon (🔒). Any sub-layers that are locked due to a parent override will display the dimmed lock icon (🔒). If a layer is unlocked, no icon is displayed. Left-clicking on a cell will toggle the lock on the layer.



Right-click on a cell to display the lock pop-up menu. The layer's current lock status is indicated by the check mark.



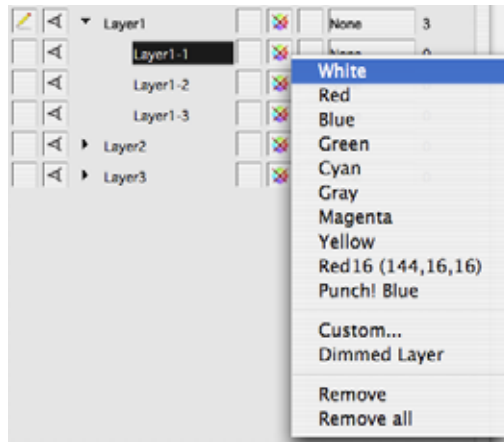
- Lock** Locks the selected layer.
- Unlock** Unlocks the selected layer.
- Lock all** Locks all layers and sub-layers.
- Unlock all** Unlocks all layers and sub-layers.

Layer Color

The second vertical column just to the right of the names indicates the layer color. If a layer does not have a color, the no-color icon (🎨) is displayed. If a layer has a color, a color sample (■) is displayed and all objects in that layer (and sub-layers) will be drawn with the layer color. The layer color takes precedence over the color already assigned to an object. Any sub-layers that are colored due to a parent override will display a multi-color sample (■). Left-clicking on a cell will cycle through the standard layer colors (see color menu below).



Right-click on a cell to display the color pop-up menu.



Black – Navy Blue Sets the selected layer's color a standard color.

Custom... Opens the color chooser dialog to assign a custom color.

Remove Removes any color from the selected layer.

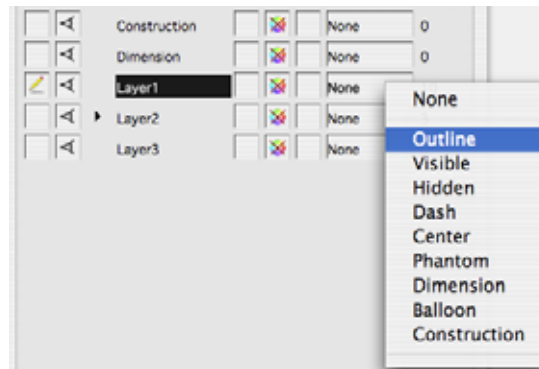
Remove all Removes any color from all layers.

Ignore when printed

The column to the right of the Layer Colors allows you to toggle the layer's visibility when printed. By default, each layer is printed, but clicking this cell will hide the layer when the design is printed.

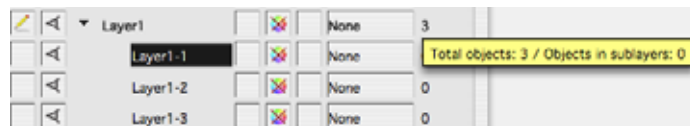
Pen Style Override

The second-to-last column allows you to override the pen style(s) on a particular layer with one uniform style.



Object Count

The number to the right of each layer name indicates object count. If there are no sub-layers or if the sub-layers are displayed, this count represents the number of objects in the layer. The total count and sub-layer count is available using the object count tool tip.



If the layer has undisplayed sub-layers, the count represents the number of objects in the layer and all sub-layers. This total count is differentiated by a + sign to the right of the number. The layer count and sub-layer count is available using the object count tool tip.

The Button Bar

The three buttons at the bottom of the Layer Manager provide easy access to the create layer, create sub-layer, and delete layer operations.



Create a new top-level layer.



Create a sublayer under the selected layer.



Delete the selected layer.

Feature Manager Tab

The Feature Manager Tab displays the associative information related to selected objects. Associative information that can be visually examined and in many cases modified through the feature manager include:

Feature Tree

Curve/Surface Associativity

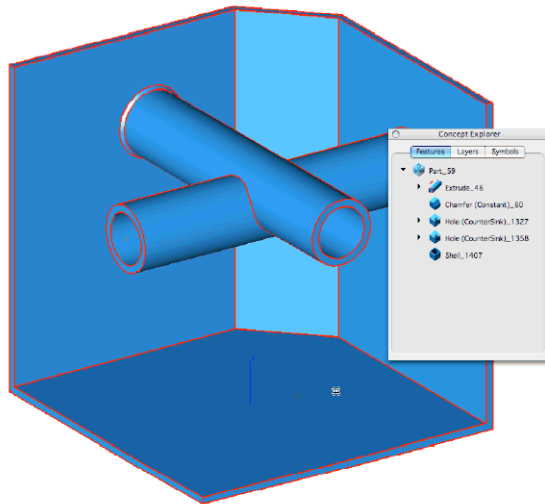
Feature Tree

The Feature Tree displays the history of the part with respect to the steps used to originally construct the part. Right-clicking on a history node will display a pop-up menu that lets you perform the following operations on a feature:

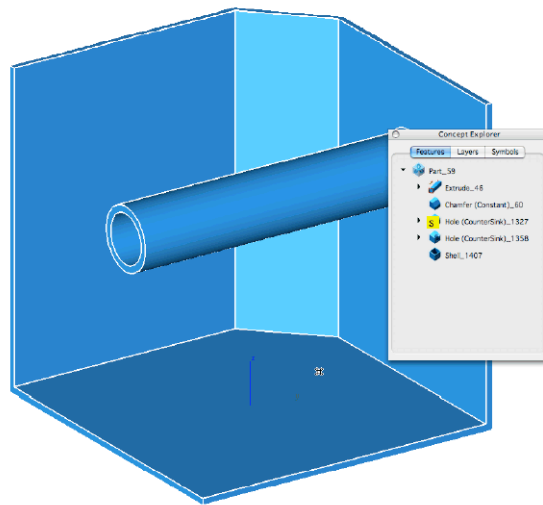
- Feature Suppress
- Feature Delete
- Feature Reorder

Feature Suppression

Feature Suppression allows you to turn off a particular feature operation in the history tree list of a solid. Suppressed features will rebuild the part as if the suppressed feature operation never occurred. You may unsuppress the feature and return it into the part at a later time.



Before Feature Suppression



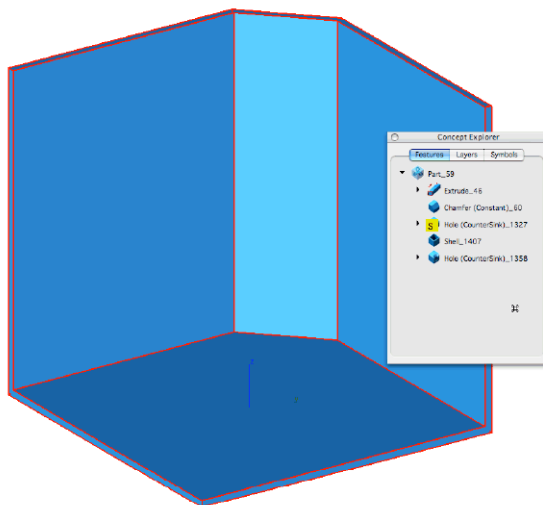
After Feature Suppression

Feature Delete

Picking this option will remove the selected feature operation.

Feature Reordering

Sometimes you will need to modify your part by rearranging its features. With the reorder command, you can move features up and down the feature tree. Some example uses of feature reordering include:

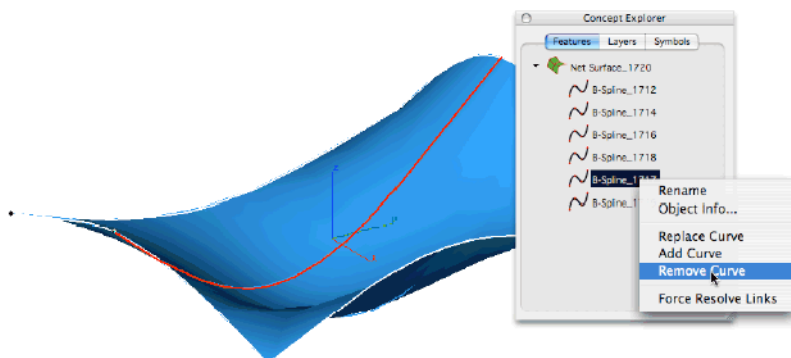


Curve/Surface Associativity

The feature manager tab does more than just show part history of solids. It also shows you any relationships that may exist with curves and surfaces as well. In addition there are commands to modify or change associative relationships using a pop-up window that appears with a right button click. These tools include commands for adding, removing, and replacing curves used with surfaces or solids.

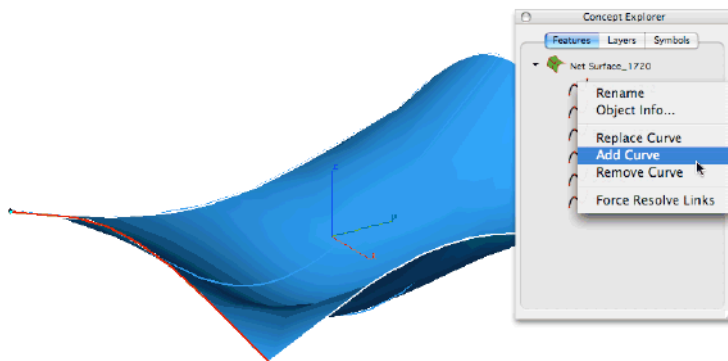
Remove Curve

Remove Curve will delete a curve reference used to define a skin or net surface. The actual curve is not deleted, just removed for the associative links of the surface. You can access the remove curve command using Concept Explorer and right clicking at the curve that you wish to remove.



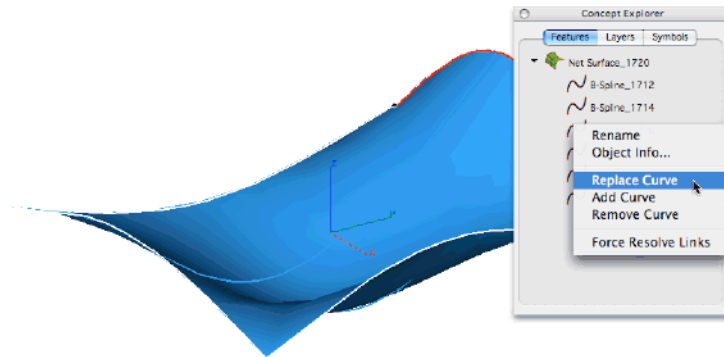
Add Curve

Add Curve will insert a new curve into a skin or net surface. You can access the add curve command using Concept Explorer and right clicking at the curve that you wish to add a new curve after.



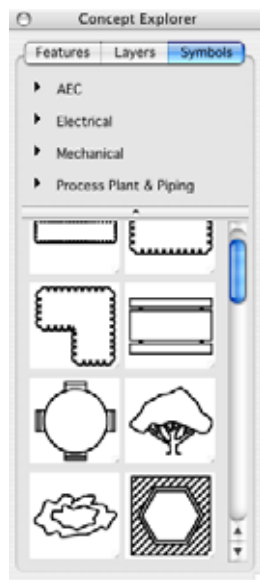
Replace Curve

Replace Curve will substitute a new curve for any curve you see in Concept Explorer. This includes curves for surfaces such as skins and nets as well as curves used to construct profiles for extruded, swept, and lathed solids.

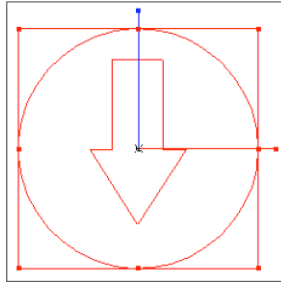


Symbol Manager

The Symbol Manager provides a user interface to drag and drop 2D and 3D symbols and parts into the drawing.



Symbols dragged in from the Symbol Manager display control points that are used to dynamically modify size, angle, and position of the object.



Additionally you can use the data entry fields to precisely modify the scale and rotation angle of the symbol.

Angle	X-Scale	Y-Scale
0.0	1.0	1.0

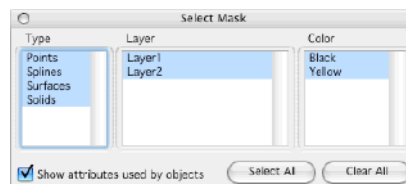
Inspector

The Inspector dialog displays a variety of general information regarding the current set of selected object(s), including Object Properties, Pen Properties, Fill Properties, Text Properties, Dimension Properties, and Gripper Properties.

For more information, see the chapter titled “Inspector”, which begins on page 35.

Select Mask

Select Masking provides a method for fine tuning your selections. Selection masks are available by color or object type. See figure below for the Select Mask dialog box.



Attributes and BOM

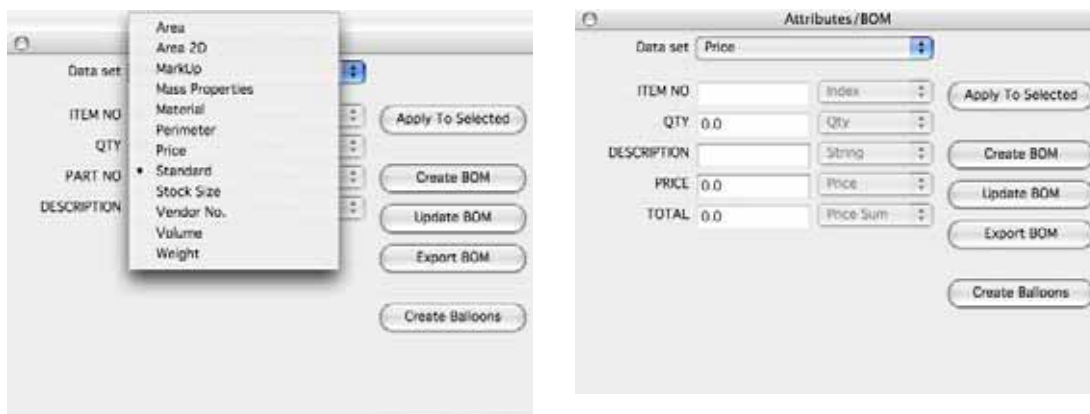
The Bill of Materials tools provide a means to apply custom attributes to entities and display that data in a table or export it to an external application such as Microsoft Excel.

There are twelve predefined attribute templates that come with Punch! Shark FX. These templates are:

Area Calculates the surface area of any polygon, surface or solid.

Area 2D	Calculates the 2D area and centroid properties for curves.
MarkUp	Assigns markups to an entity.
Mass Properties	Calculates the mass properties for solids. (Uses material assigned in <i>Verify:Mass Properties</i> command.)
Material	Assigns a material to an entity.
Perimeter	Calculates the 2D perimeter for curves.
Price	Assigns a price to an entity.
Standard	Assigns part number and description to an entity.
Stock Size	Assigns part number, stock size, and description to an entity.
Vender No.	Assigns part and vendor number, and description to entity.
Volume	Calculates the volume of a solid.
Weight	Calculates the weight of a solid. (Uses material assigned in <i>Verify:Mass Properties</i> command.)

Some of the templates automatically extract attribute information from the entity, others require the user to supply the information associated with the attribute. Any editable fields should be filled in manually.



The BOM dialog box has four button options for applying attributes, creating a Bill of Materials table, assigning balloon items, and exporting the attributes to a text delimited file.

Apply To Selected

This button option applies the current BOM attribute to the selected entities.

Create BOM

This option creates a Bill of Materials Table using the current set of attributes defined by the pull-down menu. The user is prompted to enter the 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

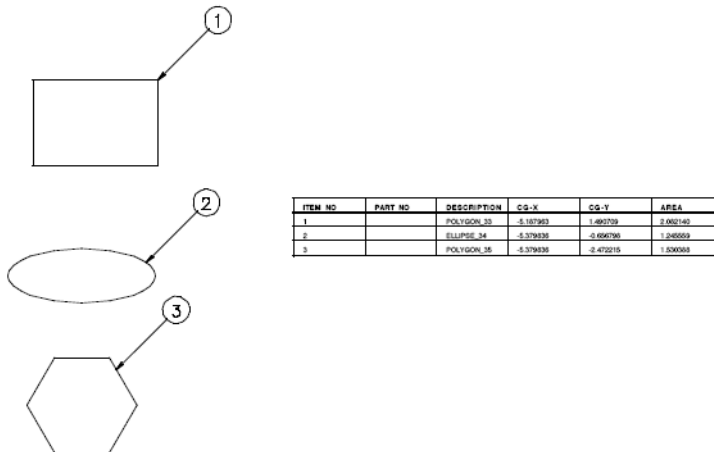
Items Numbers Down

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

Items Numbers Up

Create Balloons

Clicking the Create Balloons button will add balloon item dimensions to the geometry. This requires an item number type be used with the attribute. All of the supplied attribute files have item numbers.



Export BOM

The Export BOM option creates a comma delimited file that can be read into Microsoft Excel or other similar spreadsheets.

User Defined Templates

You can create user-defined templates by creating an attribute definition file. Attribute definition files are located in the BOM folder found in the standard 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 line contains two columns. The first column contains the attribute definition string, and the second, the attribute type. A comma separates the two columns. 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	Calculates the sum of the prevoius attributes.
Material	Extracts the material type from the entity.

Below is an example file for a user-defined attribute:

```

Material/Name
ITEM NO,      index
QTY,         qty
NAME,        name
MATERIAL,    string
DESCRIPTION, string

```

3D Digitizer

3D digitizing is a fast and flexible way to capture 3D measurements from a physical model. 3D scanning or 3D digitizing, involves creating a digital version of a physical object. Scanning allows a copy of a physical object to be represented electronically; then the electronic version can be manipulated in ways

the physical version cannot. An electronic model can be kept on a computer's hard drive, emailed, animated for use in simulation or used as a template for making physical reproductions.



MicroScribe

Punch! Shark FX processes raw 3D data from a MicroScribe device and Baces3D. MicroScribe uses a series of user-controlled snap inference filters. As you trace over your object, feedback is dynamically displayed to the user for endpoints, midpoints, centers, tangents, and alignments. The 3D snap inferencing ensures geometry created with the MicroScribe system is precisely aligned relative to neighboring geometry.

Baces3D

Baces3D is a portable measurement arm that acquires 3-dimensional measures, points, and surfaces within its work-space with a simple manual movement of the user. To use a Baces3D digitizer with Shark, simply plug the device into the USB port. This device is currently only supported on Windows Vista and XP.

3D Digitizer Tool Palette

3D digitizing offers a collection of specific tools.



3D Digitizer Connect

Use the 3D Digitizer Connect tool to initialize a new session with the digitizing device. Additionally, you will need to use this tool if the MicroScribe or tangent moves. This command creates a coordinate transformation between the target and Shark FX drawing coordinate system.

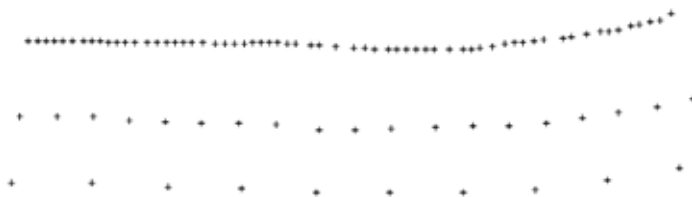
Using the 3D Digitizer Connect tool

- 1 Select the 3D Digitizer Connect tool from the tool palette.
- 2 Digitize a location representing the target origin.

- 3 Digitize a location representing a point along the target x axis.
- 4 Digitize a location representing a point along the target y axis.
- 5 Digitize two points representing the target extents. These two points should be the extent diagonals of the target.

Digitize Curves

The Digitize Curves tool provides a means to create points, lines, and splines from a continuous stream of digitized data. Additionally, this tool automatically turns off snap inferencing and alignments. If you need snap inferencing, use the Curve Creation tools on the Main Tool Palette. There is a drop-down on the Prompt Window that allows you to specify points, lines, splines, and closed splines. For each of these curve types, you can specify a point sampling value through the Data Entry window, labeled Point Spacing.



Point Spacing Examples

The spline option provides an additional option for fit tolerance. This is a tolerance used to fit a smooth spline curve to a collection of points. It is useful for removing noise from the sampled data. In the image below, the dark dots represent digitized data points with the curve passing through those points at a specific fit tolerance.



Spline Fit Tolerance

Using the Digitized Curve tool

- 1 Select the Digitized Curve tool from the tool palette.

- 2 Select the curve type, either point, line or spline.
- 3 Specify the desired point spacing.
- 4 If creating a spline, specify the spline fit tolerance.
- 5 Press and hold the digitizer button. Move the pen and release when done.

Section Surface

The Section Surface tool creates a surface as you digitize from curves spaced along sections. Options for this tool include creation of a mesh (3D facets) or NURBS (Non Uniform Bspline Surface). Additionally, you can specify the sections by continuously reading the digitizer or single click for each point on the curve.

Using the Section Surface tool

- 1 Select the Section Surface tool from the tool palette.
- 2 Select the surface type, either Mesh or NURBS.
- 3 Select the input, either Continuous or Single.
- 4 Specify the desired point spacing.
- 5 Digitize the first section of the surface. Release your mouse button to end.
- 6 Continue until all the sections for the target are entered.

Sections should not overlap and need to start from the same side of the target, throughout the Section Surface tool, to prevent twisting.

Contour Surface

The Contour Surface tool accepts scanned data in any order and reconstructs a surface from the data. Internally, it reconstructs sections from the scanned data by planar slicing and finally creating a surface from the sections. Slicing planes are controlled by the extent and orientation of the centerline and plane spacing. In the image below, the two spheres represent the start and end centerline. The red curve represents the outer contour to which everything is trimmed. The black curves are the raw MicroScribe scans used for section slicing. These scans can have any orientation, including overlapping. However, you must provide enough scans to fully cover the target area.



Using the Contour Surface tool

- 1 Select the Contour Surface tool from the tool palette.
- 2 Select the surface type, either Mesh or NURB.
- 3 Specify the desired point spacing.
- 4 Digitize two points for the center line. (These two points define the extent and orientation of the slicing planes).
- 5 Digitize the outer contour of the target. Press and hold the button around the entire outer contour.
- 6 Digitize inner contours.
- 7 Select new tool to terminate.

Align

The Align tool is used to reposition the digitized image. This is useful for digitizing models larger than the reach of the arm. Realigning is a two step process. First, you must mark the model in the current position with three marking points. Second, move the arm, reconnect, and then reference the three previous marking points.

Using the Align tool

- 1 Select the Align tool from the tool palette.
- 2 Select the Set Marking Points option.
- 3 Specify three points on the target as references.
- 4 Move the digitizer to a new position.
- 5 Use the first tool to reconnect to the MicroScribe.
- 6 Select the Align tool from the tool palette.
- 7 Select Align Coordinates from the drop-down.
- 8 Select the three marking points in the order that they are labeled/created.
- 9 Digitize three points on top of the marking points.

Constraints

Dimension and constraint tools are available to help manage the geometric relationships between 2D shapes. The Constraints tool palette contains a variety of constraint types, including distance, radius, diameter, coincident, tangent, horizontal, vertical, perpendicular, offset, concentric, parallel, equal, collinear, and symmetry. For more information, refer to the chapter titled “Constraints”, which begins on page 281.

Variables/Equations

The Variables/Equations dialog displays constraint variables that have been created and applied to geometry. For more information, see “Variables/Equations”, which begins on page 291.

Markup

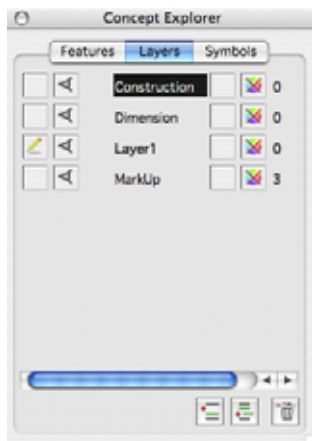
Markup tools are used to add comments or notes to a drawing. Markup tools are often used during the design review process. Shark provides a variety of markup tools including pens, markers, clouds, and callouts. In addition to tools for marking up a drawing, tools exist for making a table of changes and status. The Markup tool palette is accessed from the Window menu. From left to right the Markup tool palette provides the following functions.

- Pencil
- Marker
- Revision Cloud
- Balloons
- Measure
- Markup

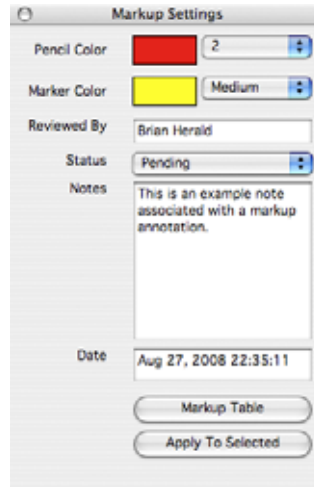


Markup Settings

Markup annotations are assigned to the Markup layer.



Markups also have pen attributes located in the Markup Settings dialog box. These attributes include pen and fill color and pen and marker thickness. The Markup Settings dialog box also includes the name of the reviewer, status, notes, and a date. Selecting an existing markup annotation will update the Markup Settings dialog to the selected entity. Use the Apply to Selected to change an existing markup attribute.



Pencil

The Pencil is used to draw a freehand line about a point of interest. The Pencil tool uses the Pencil Color and Pencil thickness attributes located in the Markup Settings dialog box for pen attributes. Reviewer, status, notes, and date attributes are attached to the resulting annotation.

Press the Option/Ctrl button to automatically close the pencil markup.

Using the Pencil tool

- 1 Click the Pencil tool.
- 2 On the workspace, click, and with your mouse button down, drag to define the pencil markup. The markup appears as dotted lines, as you are dragging, but will become a solid line when you finish.
- 3 Release to finish.

Marker

The Marker tool creates two types of marker attributes. The first type is a rectangular marker defined by opposite corner diagonals. The second type is a line marker defined by a start and endpoint. The Marker fill color and the Line Marker thickness is set in the Marker Settings dialog box.

Using the Rectangular Marker tool

- 1 Click the Marker tool.
- 2 On the Prompt Window, drag through the sub-toolset and choose the Rectangular Marker tool.
- 3 Click to set the start point.
- 4 Drag to extend the rectangle to the size you want.

- 5 Click to finish and place the rectangle.

Using the Line Marker tool

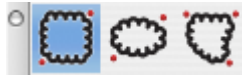
- 1 Click the Marker tool.
- 2 On the Prompt Window, drag through the sub-toolset and choose the Line Marker tool.
- 3 Click to set the start point.
- 4 Drag to extend the line to the length you want.
- 5 Click to finish and place the rectangle.



Revision Cloud

A Revision Cloud is a standard annotation in engineering, architecture, and construction for depicting a design or drafting change. Three types of revision clouds are supported which include:

- Cloud Rectangle
- Cloud Ellipse
- Cloud Freehand



Using the Cloud Rectangle tool

- 1 Click the Revision Cloud tool.
- 2 On the Prompt Window, drag through the sub-toolset and choose the Cloud Rectangle tool.
- 3 Click to set the start point.
- 4 Drag to extend the cloud to the size you want.
- 5 Click to finish and place the cloud.

Using the Cloud Ellipse tool

- 1 Click the Revision Cloud tool.
- 2 On the Prompt Window, drag through the sub-toolset and choose the Cloud Ellipse tool.
- 3 Click to set the start point.
- 4 Drag to extend the cloud to the size you want.
- 5 Click to finish and place the cloud.

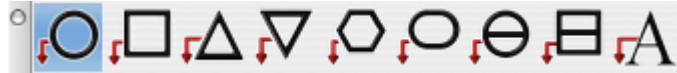
Using the Cloud Freeform tool

- 1 Click the Revision Cloud tool.
- 2 On the Prompt Window, drag through the sub-toolset and choose the Cloud Freeform tool.

- 3 Click and, with your mouse button down, drag to define the shape of the cloud. The markup appears as dotted lines as you are dragging, but will become a solid line when you finish.
- 4 Release to finish and place the cloud.

Balloon

The Balloon tool creates callout dimensions with labels and attaches to markup entities. The Balloon identifiers are useful for associating markup changes in a markup table.



Using the Balloon tools

- 1 Click the Balloon tool.
- 2 On the Prompt Window, drag through the sub-toolset and choose the Balloon tool you want.
- 3 On the workspace, click to where you want the balloon to point.
- 4 Drag to where you want the notation and click. You can edit the balloon's content on the Prompt Window.

Measure

The Measure tool creates a parallel dimension between two points in a drawing.

Using the Measure tool

- 1 Click the Measure tool.
- 2 On the workspace, click to set the dimension start point.
- 3 Drag, to extend to the length you want, and click to place the dimension.

Markup Table

The Markup Tables provide a means to identify the markups and status for a reviewed document in a table format. The Markup Table dialog displays four buttons providing the following features.

Create Table

Creates a table of markup annotations listed by Item number.

ITEM	REVIEWER	NOTES	DATE	STATUS
1	Tim Olson	Add larger line weight.	Aug 26, 2008 14:38:59	Cancelled
2	Tim Olson	Increase area coverage.	Aug 26, 2008 14:38:59	Accepted
3	Tim Olson	Move to right	Aug 26, 2008 14:38:59	Pending
4	Tim Olson	Remove from drawing.	Aug 26, 2008 14:39:10	Pending
5	Tim Olson	Customer requested two more be added.	Aug 26, 2008 14:39:10	Pending
6	Tim Olson	Feature not needed for customer.	Aug 26, 2008 14:39:18	Completed
7	Tim Olson	Remove this feature from drawing.	Aug 26, 2008 14:39:22	Pending

Update Table

Updates an existing table for markup changes since the last time the table was created.

Attach Balloons

Attaches balloon annotations to markup entities.



Export

Exports a table of markup annotations as a comma delimited text file suitable for reading into Excel or other spreadsheet programs.



Hide All Tools

A Hide All Tools option quickly hides all dialogs and tool palettes from the application. Hitting again will restore the dialogs and tools. It works especially well with photo rendering in the follow example:

- Select the Hide All Tools option.
- Render using your favorite Render Command.
- View entire screen.
- Select the Hide All Tools option to get back to work.

Minimize

Minimize the current design window.

Rotate Forward

When multiple design windows are active, Rotate Forward will bring forward the window created prior to the currently active window. So if the current window was the second window opened, Rotate Forward will make the first design window the active window. If the active window was the first window created, Rotate Forward will activate the most current window created.

Rotate Backward

When multiple design windows are active, Rotate Backward will bring forward the window created after the currently active window. So if the current window was the second window opened, Rotate Backward will make the third design window the active window. If the active window is the most current window, Rotate Backward will activate the first window created.

Bring All To Front

This option maximizes all the windows that have been minimized.

Tile Vertically

Rearranges and sizes all open drawing windows as columns. PC Only.

Tile Horizontally

Rearranges and sizes all open drawing windows as rows in a single column. PC Only.

Part 3

Drawing Tools

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Wireframe Modeling Overview

Introduction to Wireframe Modeling

A wireframe model of an object is the simplest mathematical representation of a part. The word “wireframe” is related to the fact that one may imagine a wire that is bent to follow the object edges to generate a model. Wireframes in Punch! Shark FX consist of points, lines, arcs, circles, ellipses, and conics. These particular wireframes are also collectively referred to as curves and may also be thought of as 2D entities. Although wireframes are very limited in the amount of model content they represent, they are very powerful building blocks for creating more complex, three-dimensional models composed of surfaces and solids.

Wireframe Tools

Tools associated with wireframe modeling methods are located in the upper portion of the main tool palette.



Snap Tool and Wireframes

The Snap tool recognizes several useful snap locations associated with wireframe objects. These dynamic snap locations include:

- Endpoints
- Midpoints
- Intersections
- Projections

Selection and Display

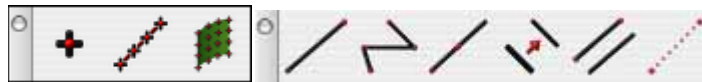
Wireframes are selected by picking a location along the length of the wire. Wires are selected when any portion of the wire passes through the rubberbanding select box or single pick box. Conics, ellipses, splines, and circles have resolution attributes that impact the screen display and printer output. Use the *Edit:Change Resolution* command from the Menu Bar to change a wireframe display resolution.



Points & Lines

Point and Line tools

The top of the main tool palette window contains two tear aways for creating points and lines.



Point tools

There are three different methods for creating point objects. From left to right along the Point tool palette they are:

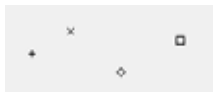
- Point
- Points Along Line
- Points Along Surface

Point

Create a single point at a specified x, y, z location. There are four point styles available from the drop-down menu on the Prompt window. To change the point style, choose a different style from the drop-down after the tool is activated.

- Cross-hair Plus
- Cross-hair X
- Empty Diamond
- Empty Square

To create a point, simply locate and press your mouse button at the desired location.

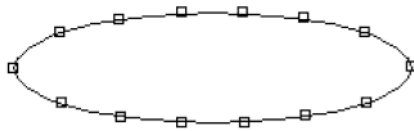


Different Point Styles At Various Locations

 ***Points Along Line***

Creates a user-specified number of points along a curve. There are three options for determining how points are distributed, from the drop-down on the Prompt window. Points can be distributed by:

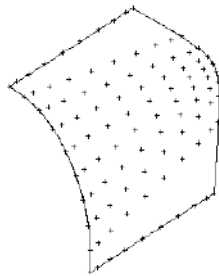
- Number of Points - points distributed based on a desired number of points.
- Even parametric locations along the curve - points distributed based on a desired distance between each point.
- Length - point positioned at a desired length.



Points Along Line

 ***Points Along Surface***

Creates a MxN grid of points along a surface.



Points Along Surface

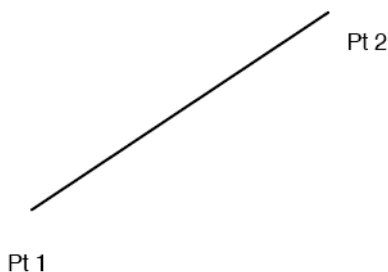
Line Tools

There are six different methods for creating point objects. From left to right along the Point tool palette they are:

- Single Line
- Multi Line
- Midpoint Line
- Parallel Line
- Double Line
- Construction Line

 ***Single Line***

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



Using the Single Line tool

- Click two endpoints of the line.
or
- Drag to indicate the endpoints of the line; press at the beginning and release at the end of the line. As you drag, you see a *rubberband* line that previews your construction.

Notice that after completing a line, the Message Line says “Option copy previous”. This indicates that if you hold the Option key down and click once in the drawing area, a line will appear that is identical to the one just drawn. The location of the click designates the location of the first endpoint.

X	Y	Z	Length	Angle
-6.805	4.934	0.0	1.459	218° 6' 59"

The Data Entry Fields allow you to specify the X,Y,Z coordinates of the beginning, the relative location of the end (X, Y, Z), the line length, and the angle from horizontal. Once a line is drawn, the line Length is the selected status box.

Drawing a line perpendicular to another object

- 1 Construct the object.
- 2 Move the pointer to the object until a snap on notation appears (endpoint, midpoint, quadrant, etc., will not work, only on).
- 3 Drag straight away from the object in a perpendicular direction. A perpendicular line will appear attached to the object. Keep holding the mouse down and move the cursor. Notice that the line stays perpendicular but slides along the object.
- 4 Drag the extent of the line.

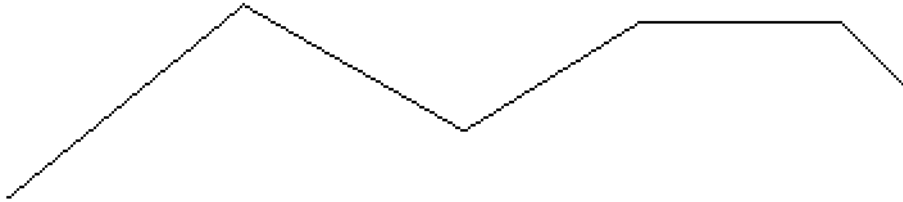
Drawing a line tangent to or perpendicular to a curve

- 1 Construct an arc, circle or ellipse.
- 2 Choose the Single Line tool.
- 3 Move the pointer to the arc until a snap **on** notation appears. The snap notation must be on rather than endpoint, quadrant or midpoint.

- 4 Drag in the appropriate direction (straight out for perpendicular or at an angle for tangent) until the snap perpendicular or tangent notation appears.
- 5 When the snap locks on to perpendicular or tangent, you can then drag the line around the arc to the location you want and extend the line to the length you want.

Multi Line

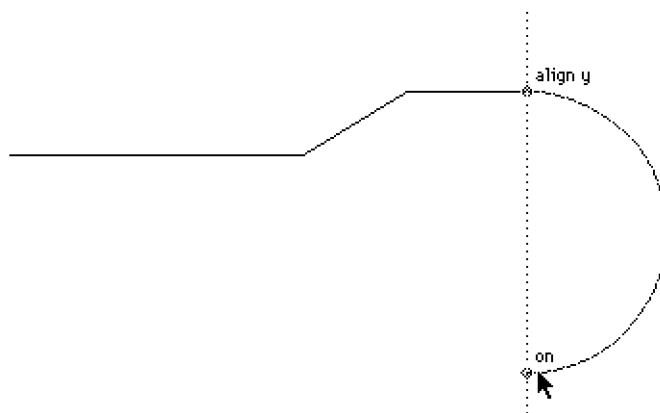
This tool draws lines in which the endpoint of one line segment is the beginning point of the next. Use the undo command to remove the previous segment of the connected line.



Using the Multi Line tool

- 1 Click or drag to indicate the endpoints of the line segments. If you click a point and then change your mind, press the Escape key or choose Undo to remove the last line. Pressing the Delete key removes all connected lines in the current construction.
- 2 Indicate the last point by double-clicking or by choosing another tool.

After completing at least one segment with the Multi Line tool, you can create a tangent arc off of the last line by holding down the Option key (notice that the pointer temporarily changes to an “arc” icon) and clicking or dragging the next point. Several tangent arcs can be strung together by continuing to hold down the Option key. (Note: the radius of these arcs cannot be edited with this method.)



Multi Line with Arc Option

The Data Entry Fields allow you to specify the XYZ coordinates of the beginning, the relative location of the end (X, Y, and Z), the line length, and the angle from horizontal. Once the line segment is created, line Length is the selected data entry box. By entering successive sets of data and hitting the Return key after each set, a continuous string of connected lines can be entered from the keyboard.

Midpoint Line

This tool draws a line whose first point is the midpoint. Use the undo command to remove the previous segment of the connected line. Use the Data Entry Fields to modify the length of the line relative to the line midpoint.

Using the MidPoint Line tool

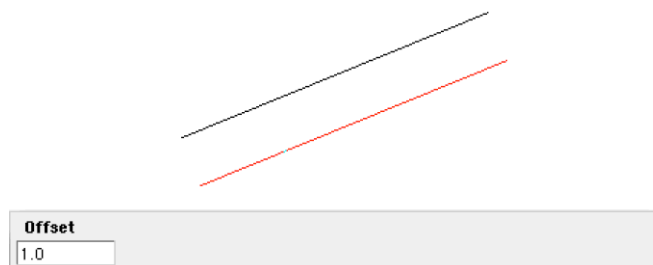
- 1 Click the midpoint of the line. Move the cursor to define the length and direction of the line.
- 2 Use the Data Entry Fields to specify the exact length and direction of the line if needed.

Parallel Line

This tool constructs lines parallel to existing lines. Use one of the following methods for creating parallel lines:

Using the Parallel Line tool

Select the Parallel Line tool, drag off the desired line, and release to indicate the position for the new parallel line. A rubberband line moves with the pointer. The Data Entry Fields show the distance you dragged. If you type in your own value and hit the Return key, the parallel copy will move to the offset defined by your value.



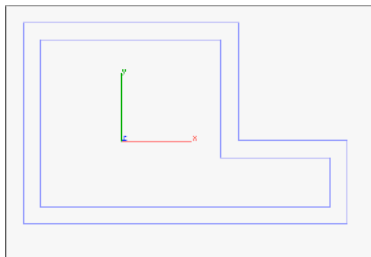
Double Line


The Double Line tool automatically offsets to create a thickness for the line.

Using the Double Line tool

- 1 Select the Double Line tool from the tool palette.
- 2 Click locations.

- 3 End with double click or hitting ESC key.



 **Construction Line**
Construction lines are useful for creating temporary lines used in the design process. Construction lines are created in the Construction layer and display with a dashed red font.



The Construction Line tool has two sub options. The first option creates an infinite rubber banding line between two points. The second option drags a construction off from an existing line.



The Construction Line tool automatically adds creation angles at 15-degrees. Plus ambiguity selection dialog box now ignores construction lines, making them easier to work with in general.

Using the Construction Line tool

- 1 Select the Construction Line tool from the tool palette.
- 2 Click to set the start point, and with your mouse button down, drag to set the line's angle.
- 3 Release to place.

Using the Construction Parallel Line tool

- 1 Select the Construction Line tool from the tool palette.
- 2 From the sub-tool palette, click the Construction Parallel Line tool.
- 3 Click an existing line or construction line, and drag to where you want the parallel line.
- 4 Release to place.

Arcs & Circles

Arc and Circle Tools

Punch! Shark FX supports ten circle and arc creation tools. From left to right along the Circle Tool palette these commands are:



Arc Tools

There are six tools for creating segmented arcs.

Two Point Arc

The Two Point Arc tool creates an arc from three points. The order of the points is the start point, the endpoint, then the centerpoint. The Two Point Arc tool is unique in that the two endpoints are static even as the radius of the arc changes.

Using the Two Point Arc tool

- 1 Click to set an endpoint.
- 2 Click to set the other arc endpoint.
- 3 Drag to set the arc's radius and click to specify its centerpoint.



 **Start, End, On Arc**

The Start, End, On Arc tool creates an arc using three points on the arc. The first point is the starting location, the second point is the ending location, and the third point is a point that lies on the arc. The diameter is calculated based on the supplied three points.

Using the Start, End, On Arc tool

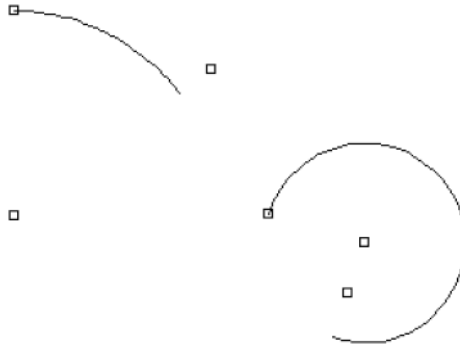
- 1 Click the arc starting point.
- 2 Click a point that is the arc endpoint.
- 3 Click a point that lies along the arc.

 **Center Point Arc**

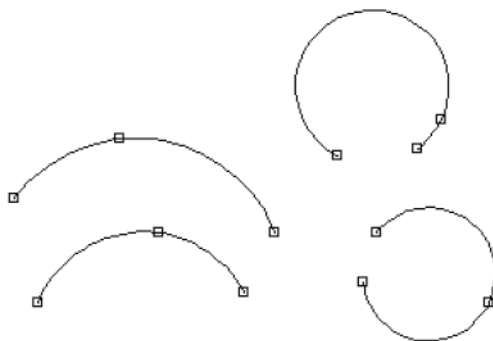
The Center Point Arc tool creates an arc from three points. The order of the points is center point, start point, and endpoint.

Using the Center Point Arc tool

- 1 Click the arc center point.
- 2 Click the arc start point.
- 3 Click a point that is scaled to the radius, terminating the arc.

 **Three Points Arc**

The Three Points Arc tool creates an arc using three points on the arc. This tool creates an arc using first point, point on, and endpoint. The diameter is calculated based on the supplied three points.



Using the Three Points Arc tool

- 1 Click the arc starting point.
- 2 Click a point that lies along the arc.
- 3 Click a point that is the arc endpoint.



Tangent Points Arc

The Tangent Points Arc tool creates an arc whose second point defines that tangent off of the first point.

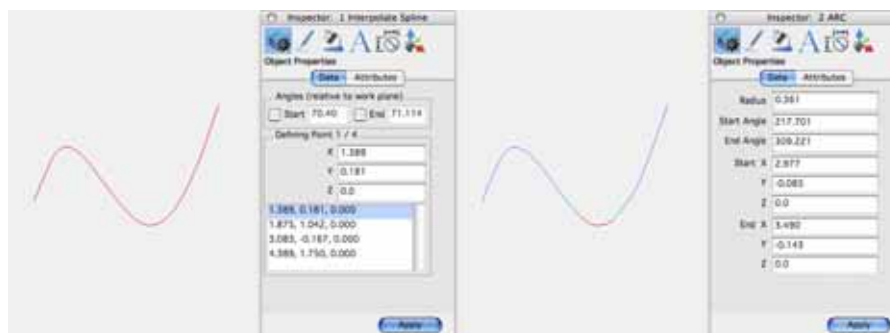
Using the Tangent Points Arc tool

- 1 Click the arc starting point.
- 2 Click a point that defines the starting tangent of the arc.
- 3 Click a point that is the arc endpoint.



Arcs From Curve

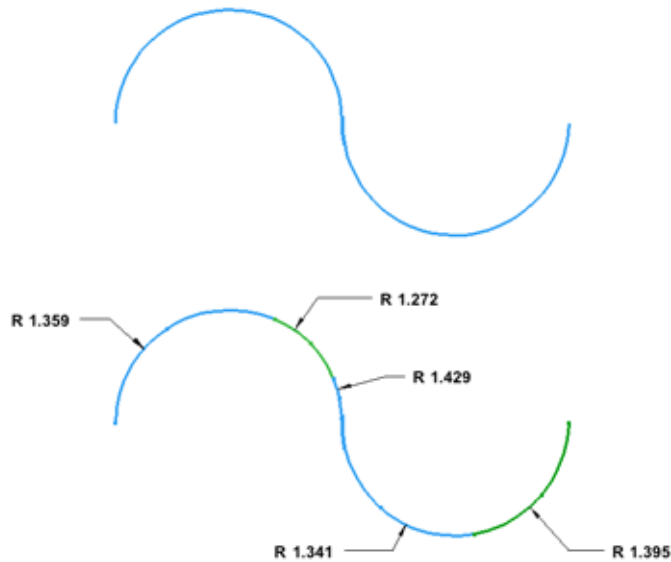
The Arcs From Curve tool creates a series of arcs that follow the shape of an existing spline, ellipse or conic. Once the tool is selected, you can choose the continuity you want, on the Data Entry window.



Position Continuity preserves the original curve. Tangent Continuity positions each arc tangent to its counterpart. You can also enter the tolerance you want. The lower the tolerance value, the closer the arcs appears to the original shape.

Using the Arcs From Curve tool

- 1 Click the arc starting point.
- 2 Click a point that lies along the arc.
- 3 Click a point that is the arc endpoint.



Circle tools

The Circle tool palette contains six tools for creating circles.

One Point Circle

This creates a circle from one point, the centerpoint. The diameter is defined on the Data Entry Window.

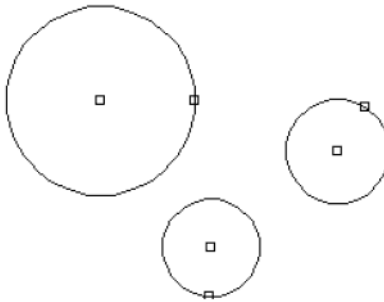
Using the One Point Circle tool

- 1 Enter the diameter you want on the Data Entry Window. Be sure to press Enter to accept new values.
- 2 Click to set the circle's centerpoint.



Center Point Circle

Creates a circle using the first input as the circle center and the second point as a point on circle circumference.



Using the Center Point Circle tool

- 1 Click the circle center point.
- 2 Click a point that defines the circle diameter.



Opposite Point Circle

The Opposite Point Circle command creates a circle from two input points. The points lie on opposite sides of the resulting circle.

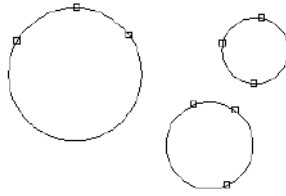
Using the Opposite Point Circle tool

- 1 Click a point that lies on the circle circumference.
- 2 Click a second point that lies on the opposite side of the circle.



Three Point Circle

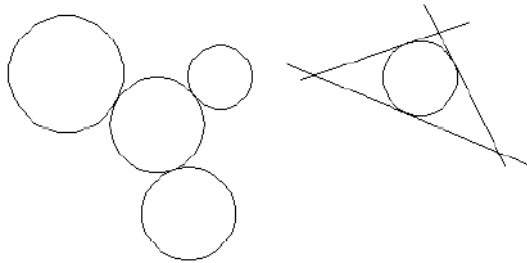
Creates a circle from three points on the circumference.



Using the Three Point Circle tool

- 1 Click the circle starting point.
- 2 Click a point that lies along the circle.
- 3 Click a third point that is the arc end circle.

Hold down the Option key to make tangent to referenced curves.

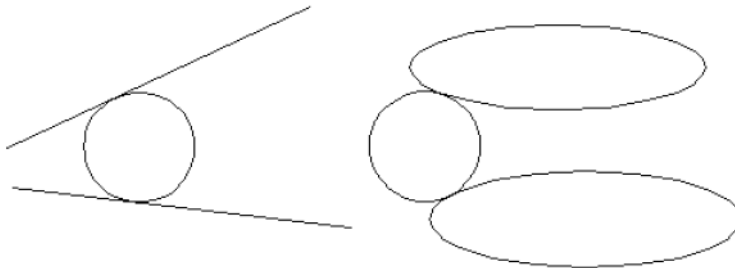


Tangent Point Circle

The Tangent Point Circle tool creates a circle tangent to two curves.

Using the Tangent Point Circle tool

- 1 Specify a diameter in the Data Entry Window.
- 2 Click the first curve.
- 3 Click the second curve.





Three Tangent Circle

The Three Tangent Circle tool creates a circle tangent to three curves.

Using the Three Tangent Circle tool

- 1 Click the first curve.
- 2 Click the second curve.
- 3 Click the third curve.

Conics & Ellipses

Conic and Ellipse tools

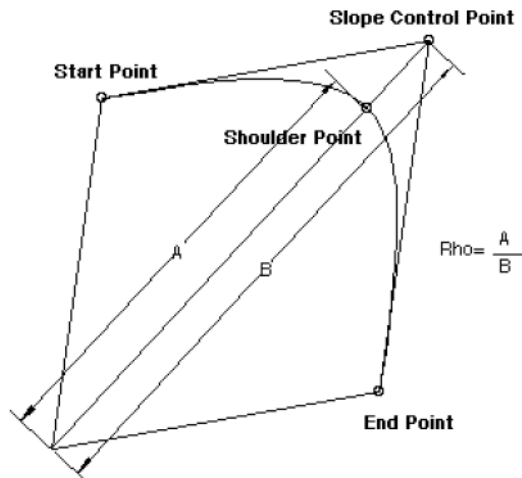
Conic curves or conic sections form the most general form of quadratic curves. There are eight methods for creating conics. From left to right along the Conic tool palette they are:



- Two Point Conic
- Three Point Conic
- Four Point Conic
- Center Point Ellipse
- Diagonal Ellipse
- Opposite Point Ellipse
- Three Point Ellipse
- Major Axis Ellipse

Conic

Punch! Shark FX uses the Conic form popular in the aerospace industry. The form consists of a start and end point plus a variety of means to specify the tangency. A graphical description of the relative points is shown below.



Conic Definition

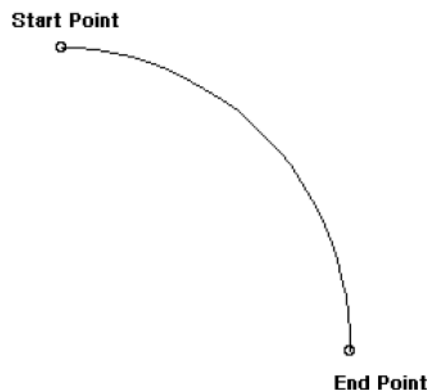
Punch! Shark FX provides three methods for creating conics that include 2, 3, and 4 point forms.

Two Point Conic

This tool creates a conic by specifying the start and endpoints plus specification of a rho value. A rho value is the ratio of the distance of the conic center point to shoulder point, over the distance of the center point to the slope control point. See example below. This form of the conic will imply horizontal and vertical slopes relative the current construction plane for the tangencies.

Using the Two Point Conic tool

- 1 Click a point defining the conic start position.
- 2 Click a point defining the conic end position.
- 3 Use the Data Entry Fields to change the Rho value, if needed.





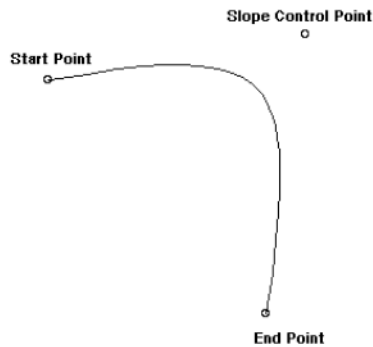
Three Point Conic

Creates a conic by specifying the start, end, slope control, and the Rho value.

Using the Three Point Conic tool

- 1 Click a point defining the conic start position.
- 2 Click a point defining the conic end position.
- 3 Click a point defining the conic slope control position.
- 4 Use the Data Entry Fields to change the rho value if needed.

Note: The slope control point is the intersection of the tangents extended from the start and end control points.



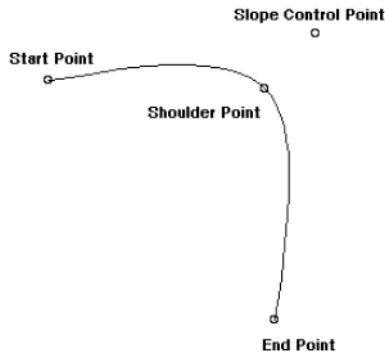
Four Point Conic

Creates a conic by specifying the start point, endpoint, slope control point, and shoulder point.

Using the Four Point Conic tool

- 1 Click a point defining the conic start position.
- 2 Click a point defining the conic end position.
- 3 Click a point defining the conic slope control position.
- 4 Click a point defining the conic shoulder control position.

Note: The shoulder point lies on the conic curve.



Ellipse

Ellipses are special forms of conic curves. Ellipses in Punch! Shark FX are defined by a center point, major axis, minor axis, and radius for each of the axes. The various user interfaces extract this information and create the appropriate ellipse. The default ellipse is created from 0 to 360 degrees, and is measured in a positive direction about the ellipse normal. The direction of the ellipse normal is defined as the cross product of the major and minor axis.

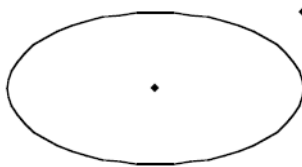


Center Point Ellipse

This command creates a 360-degree ellipse from two points that include the ellipse center and one diagonal.

Using the Center Point Ellipse tool

- 1 Click the ellipse center point.
- 2 Click the diagonal point that defines the major and minor axis lengths.

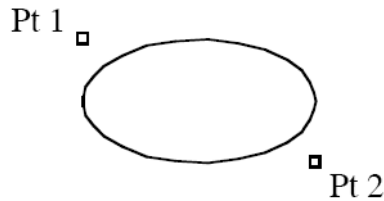


Diagonal Ellipse

Creates an ellipse by opposite diagonals by indicating the upper left corner of the ellipse and the lower right.

Using the Diagonal Ellipse tool

- 1 Click a point defining the ellipse upper left corner.
- 2 Click a point defining the ellipse lower right corner.



Opposite Point Ellipse

Creates an ellipse by opposite ends and a point defining the height of the minor axis by indicating three points.

Using the Opposite Point Ellipse tool

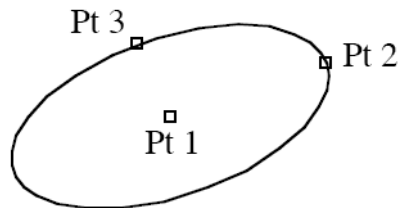
- 1 Click a point defining the ellipse left point for major axis.
- 2 Click a point defining the ellipse right point for major axis.
- 3 Click a point defining the ellipse height along minor axis.

Three Point Ellipse

This option creates an ellipse from three points. The first point is the ellipse center. The second point is the major axis endpoint. The third point is projected onto the minor axis to determine the correct distance.

Using the Three Point Ellipse tool

- 1 Click a point defining the ellipse center.
- 2 Click a point defining the ellipse major axis.
- 3 Click a point defining the ellipse minor axis.

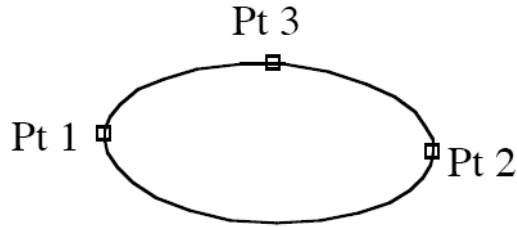


Major Axis Ellipse

This option creates an ellipse by specifying two points for the magnitude and orientation of the major axis and projects a third point along the minor axis for the correct distance.

Using the Major Axis Ellipse tool

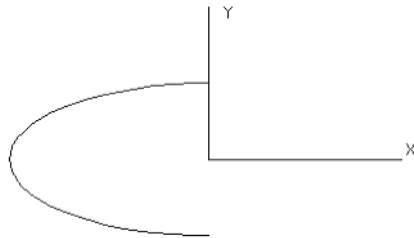
- 1 Click a point defining the ellipse major axis start.
- 2 Click a point defining the ellipse major axis end.
- 3 Click a point defining the ellipse minor axis end.



Ellipse by Major Axis Extents and Minor Axis

Ellipse Angles

Use the Inspector to modify an ellipse's start and end angles. For more information, see "Object Properties", which begins on page 35.



Splines

Spline Tools

Punch! Shark FX supports eleven spline tools. A spline is a smooth curve fit through a collection of user specified points.

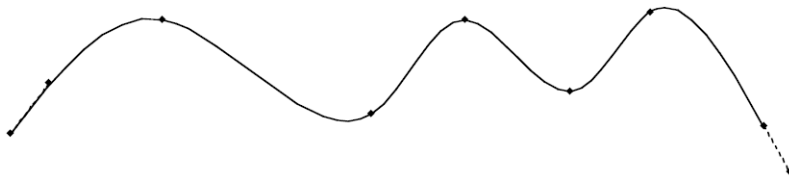


Interpolate Spline

Create a spline by supplying only the points on the curve. The slopes will be calculated automatically, and the spline will be updated as you move your cursor.

Using the Interpolate Spline tool

- 1 Click two or more points defining the spline.
- 2 Terminate with a right-click, ESC, double-click or activate new tool.



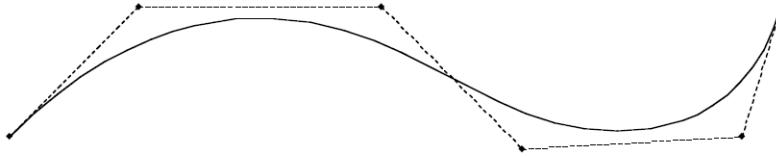
Note: If the start point equals the last point, the slope is adjusted as smooth through the shared point. The designer is allowed to modify only the start and end slope on this particular spline type. To see the slope controls, select the curve and select the Edit:Show points option from the Menu Bar.

Control Point Spline

Create a spline by supplying only the control polygon of a B-Spline. Only the start and endpoints will lie on the resultant curve, and the spline will be updated as you move your cursor.

Using the Control Point Spline tool

- 1 Click two or more points defining the spline.
- 2 Terminate with a right-click, ESC, double-click or activate new tool.

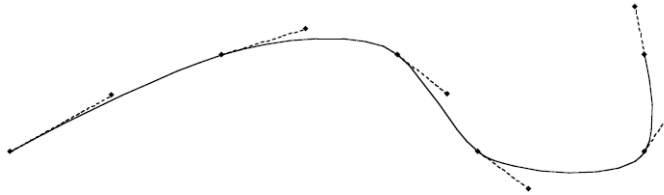


Bezier Spline

Create a Bezier Spline by supplying points that lie on the curve, and the spline will be updated as you move your cursor. The Bezier spline allows you to modify the slope at any control point. To modify a point, select the curve and pick the Edit>Show Points command.

Using the Bezier Spline tool

- 1 Click two or more points defining the spline.
- 2 Terminate with a right-click, ESC, double-click or activate new tool.



Sketch Spline

The Sketch Spline tool samples points as the cursor moves and creates a smooth spline through the sampled data.

Using the Sketch Spline tool

- 1 Press the mouse and drag the desired shape.
- 2 Release the mouse to terminate and create the final spline.

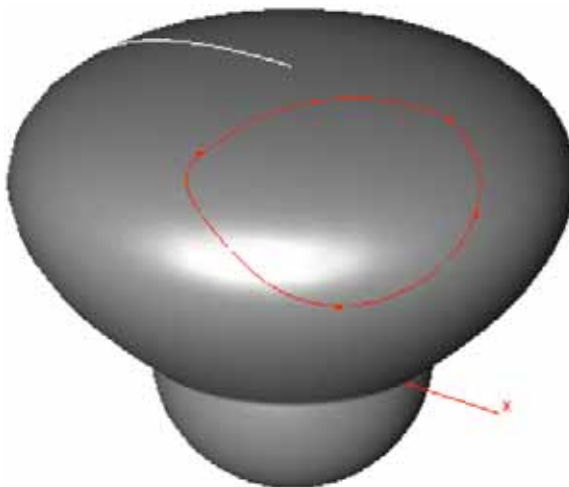


Spline On Surface

The Spline On Surface tool creates a spline that lies precisely on a surface or face of a solid.

Using the Spline On Surface tool

- 1 Select a surface or solid face on which to draw a spline.
- 2 Click two or more points defining the spline.
- 3 Double-click to finish.



Note: You can use this curve to trim a surface to or imprint onto, a solid. Use the Add and Remove Points tools to change the number of points on the curve. Use the Selection tool to relocate existing points.

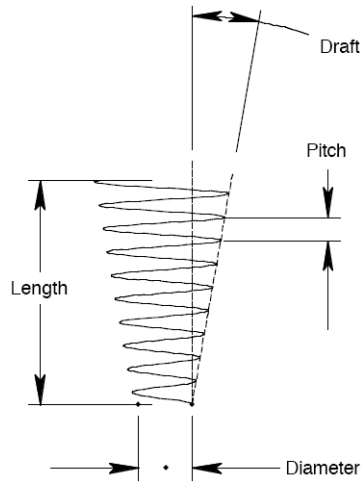
Helix and Spiral

This tool creates a helix spline, a spiral spline, and a helix along path spline. All three result in a vector spline based curve.



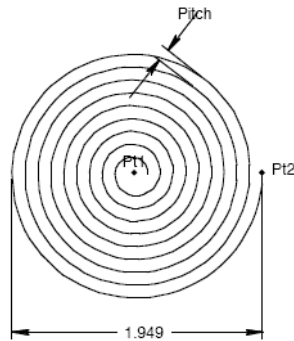
Using the Helix Spline tool

- 1 Click first point for the helix start.
- 2 Click last point for the helix end.
- 3 Use the Data Entry Fields to adjust the pitch, diameter, length, and draft angle.



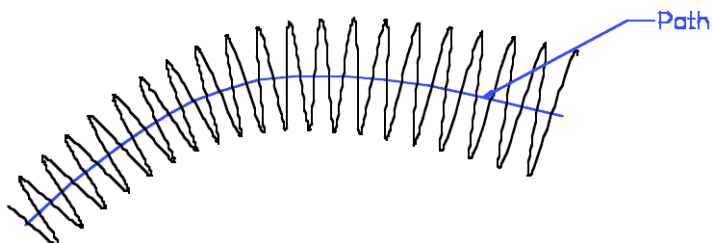
Using the Spiral Spline tool

- 1 Click first point for the spiral start.
- 2 Click last point for the spiral end.
- 3 Use the Data Entry Fields to adjust the pitch or diameter.



Using the Helix Path Spline Tool

- 1 Select the curve to use as the helix path.
- 2 Use the Data Entry Fields to adjust the pitch, diameter, and draft fields.



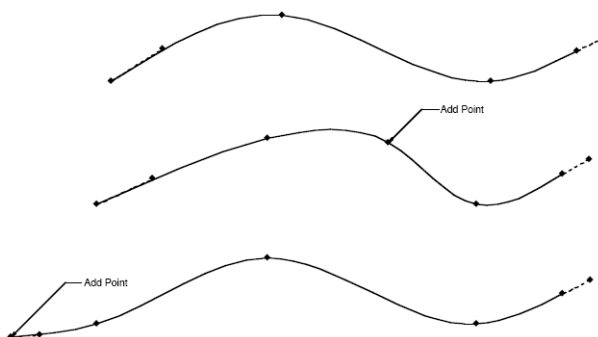
Add Spline Point

Adds a defining point to the spline curve. This is useful for fine tuning of the curve shape.

Using the Add Spline Point tool

- 1 Select the spline to modify.
- 2 Click location to add spline point.

Note: You can add points to extend a spline.



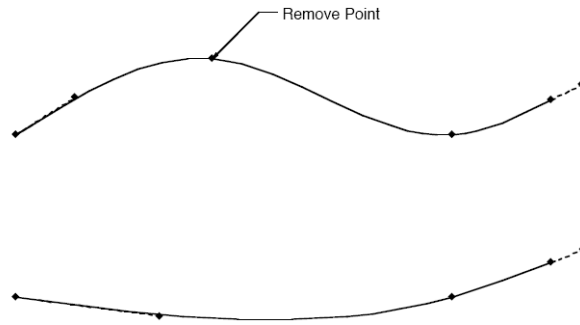
Remove Spline Point

Remove a control point from a spline. Some curves may have too many points resulting in an over constrained curve.

Using the Remove Control Point Spline tool

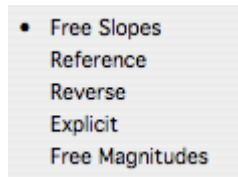
- Select the spline point to remove point.

Note: You can select any point but the slope points to remove.



Modify Slope

The Modify Slope tool displays a pop-up window that allows five options related to spline slopes. These five options are:



Free Slopes

Resets the slope back to the system default. This command works on the start or end slope of a through point spline, and any point on a Bezier spline.

Reference

This command allows you to set the slope of a curve to a referenced curve, surfaces or solid.

Reverse

Flips the slope 180 degrees. Sometimes needed in conjunction with the Reference Slope tool.

Explicit

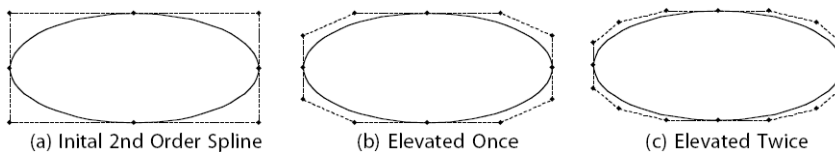
Set the slope to a specific angular value. The value is measured relative to the x axis of the current work plane.

Free Magnitudes

Resets the magnitude of a Bezier slope.

Elevate Curve

The Elevate Curve tool increases the internal polynomial order of the spline. You can select any curve (line, spline, conic, arc, circle, ellipse) to elevate.

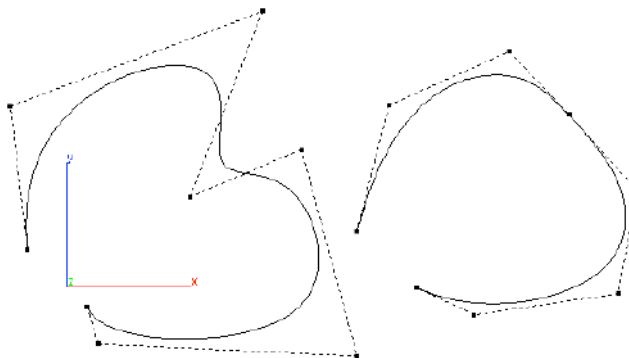


Smooth Spline

The Smooth Spline tool moves control vertices of vector splines into a “smooth” or “faired” position. The maximum amount any control point is moved is the tolerance value listed on the Data Entry window. This tool does not add or remove points in the fairing process. Existing points are moved to a location that is deemed “smoother”.

Using the Add Spline Point tool

- 1 Select the spline you want to smooth.
- 2 Continue to select until you have the desired spline or reset the tolerance value on the Data Entry window.



Polygons

There are seven tools for creating and manipulating polygons. These tools, from left to right, are:



- Rectangle
- Rounded Rectangle
- Slot Polygon
- Inscribed Polygon
- Circumscribed Polygon
- N-Sided Polygon
- From Curves Polygon

Polygon Fill Patterns and Colors are specified on the Inspector. For more information, see “Fill Properties” on page 42.

Rectangular Polygons

The Polygon Rectangle tool has four creation methods that provide different user interfaces for specifying a rectangle. To access each tool, click the Rectangle Polygon tool on the Polygon tool palette and drag through the sub-tools on the Prompt Window.




Rectangle Polygon sub-tools

Center Point Polygon

This tool creates a rectangle from two points by specifying the center and upper right diagonal.



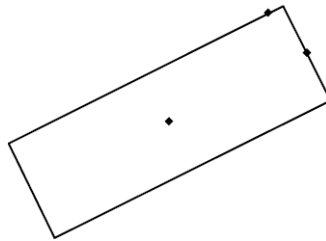
 **Diagonal Polygon**

This tool creates a rectangle from two points by specifying the lower left and upper right diagonal.



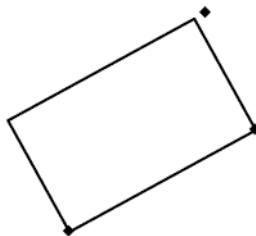
 **Three Point Polygon**

This tool creates a rectangle from three points specifying the center, width point, and height point.



 **Major Axis Polygon**

This tool creates a rectangle from three points where the points are width start, width end, and height.



Also in the message line is a pull down for Single Line or Smart Polygon rectangles. The Single Line option creates unassociative lines. The Smart Polygon creates a special entity that stores the resulting object as rectangle of width and height.

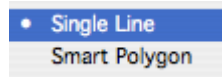
Rounded Polygon

The round polygon is defined by the length, width, and a radius. The radius is used to define a 90 degree arc at each corner of the polygon. There are four methods for drawing a rounded polygon. To access each tool, click the Rounded Polygon tool on the Polygon tool palette and drag through the sub-tools on the Prompt Window.



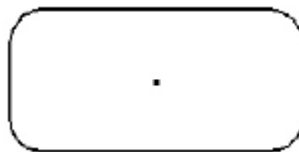
Rounded Polygon sub-tool palette

Also on the Data Entry Window is a drop-down menu for Single Line or Smart Polygon rectangles. The Single Line option creates unassociative lines. The Smart Polygon creates a special entity that stores the resulting object as rectangle of width and height.



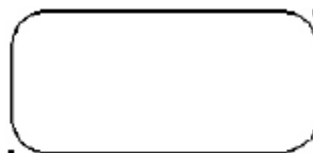
Center Point Round

This tool creates a round polygon from two points by specifying the center and upper right diagonal.



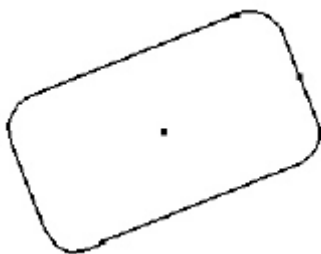
Diagonal Round

This tool creates a round polygon from two points by specifying the lower left and upper right diagonal.

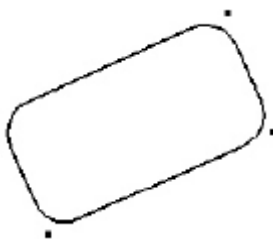


Three Point Round

This tool creates a round polygon from three points specifying the center, width point, and height point.

**Major Axis Round**

This tool creates a round polygon from three points where the points are width start, width end, and height.

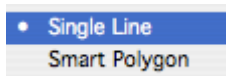
**Slot Polygon**

The slot polygon is defined by the length and width. The width controls the diameter of a 180-degree arc. There are four methods for drawing a slot polygon. To access each tool, click the Slot Polygon tool on the Polygon tool palette and drag through the sub-tools on the Prompt Window.



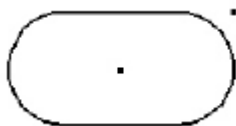
Slot Polygon sub-tool palette

Also on the Data Entry Window is a drop-down menu for Single Line or Smart Polygon rectangles. The Single Line option creates unassociative lines. The Smart Polygon creates a special entity that stores the resulting object as rectangle of width and height.



 **Center Point Slot**

This tool creates a slot polygon from two points by specifying the center and upper right diagonal.

 **Diagonal Slot**

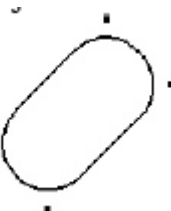
This tool creates a slot polygon from two points by specifying the lower left and upper right diagonal.

 **Three Point Slot**

This tool creates a slot polygon from three points specifying the center, width point, and height point.

 **Major Axis Slot**

This tool creates a slot polygon from three points where the points are width start, width end, and height.





Inscribed Polygon

The Inscribed Polygon tool creates an N-Sided polygon given a center point and diameter. The resultant polygon lies inside a circle.

Using the Inscribed Polygon tool

- 1 Click a point for the polygon center point.
- 2 Click a point for the polygon width.
- 3 Use the Data Entry Fields to adjust the center xyz, number of sides or diameter.



Circumscribed Polygon

The Circumscribed Polygon tool creates an N-Sided polygon given a center point and diameter. The resultant polygon lies outside a circle.

Using the Circumscribed Polygon tool

- 1 Click a point for the polygon center point.
- 2 Click a point for the polygon width.
- 3 Use the Data Entry Fields to adjust the center xyz, number of sides or diameter.



N-Sided Polygon

The N-Sided Polygon tool creates a polygon from an arbitrary number of points. The end is automatically closed.

Using the Arbitrary Points Polygon tool

- 1 Click three or more points defining the polygon shape.
- 2 Terminate with the right button, ESC, double click or select new tool.



Arbitrary Polygon

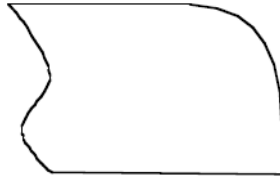


From Curves Polygon

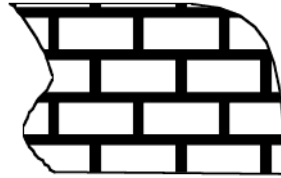
Creates a polygon from a collection of trimmed, closed curves. The resulting polygon uses the default fill color and pattern.

Using the Polygon From Curves tool

- Select one or more curves forming a closed shape.



Closed Curves

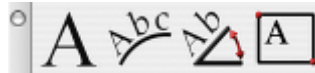


Resulting Polygon

From Curves Polygon

Text

There are four tools for defining text annotations to a drawing. From left to right in the Text tool palette they are:



- Horizontal Text
- Text Along Curve
- Text At Angle
- Bounding Box Text

Text properties, such as font, size, color, orientation, and more, are controlled on the Inspector. For more information, see “Text Properties”, which begins on page 46.

Horizontal Text

The Horizontal Text command creates horizontal (left to right) text at a user specified location.

Using the Text tool

- 1 Click location for text.
- 2 Type characters for text string.
- 3 Click mouse to end. (Hit Return to continue placing new text entities.)

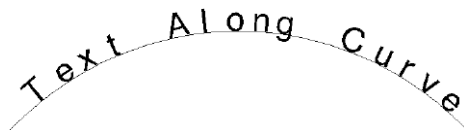
Along Curve Text

Text can be created with respect to a curve with the Along Curve command. The text is associative to the curve.

Using the Along Curve Text tool

- 1 Select a curve for the text path.
- 2 Click two points that specify the text up direction.

- 3 Type characters for text string.
- 4 Click mouse to end.

The text "Text Along Curve" is displayed in a white, monospace-style font, following the upper arc of a thin black semi-circle.

At Angle Text

Text can also be created at any angle. The angle uses the right hand rule about the construction plane normal.

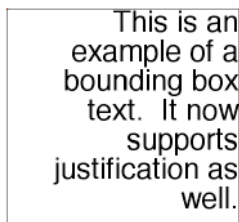
Using the At Angle Text tool

- 1 Use the Data Entry Fields to specify text angle.
- 2 Click location for text.
- 3 Type characters for text string.
- 4 Click mouse to end.

The text "TEXT At 45 Degrees" is displayed in a white, monospace-style font, rotated 45 degrees counter-clockwise.

Box Text

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. The new text box also supports left, right, and center justifications. In addition, you can double click on the text to edit directly on the screen. On-screen editing provides support for cutting, pasting, and copying into the text box. For example, you can copy a paragraph from another application and paste it into a text box.

A rectangular box with a thin black border containing the text "This is an example of a bounding box text. It now supports justification as well." The text is centered within the box and is justified to the width of the box.

Dimension

There are eleven tools for creating dimensions. From left to right, these tools are:

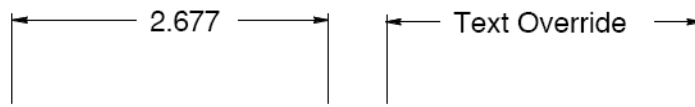


- Smart Dimension
- Horizontal
- Vertical
- Parallel
- Ordinate
- Radial and Diametrical
- Angular
- Center Mark
- Balloon & CallOut
- Length Along Curve
- Geometric Dimensioning and Tolerance

The Smart Dimension tool creates an associative link between the dimension and any referenced curve. This means that modifying the curve will automatically update the smart dimension. The other dimension tools do not preserve a relationship between geometry and the dimension.

Dimension Text

You can specify the dimension text or use the # symbol to automatically fill the field based on the measured value. The text is controlled on the Prompt window.



Dimension properties, such as format, orientation, and more, are controlled on the Inspector. For more information, see “Dimension Properties”, which begins on page 47.



Smart Dimension

The Smart Dimension tool creates horizontal, vertical, parallel, radial or diameter dimensions based on the object type selected and where you move your mouse. Dimensions created with this tool are associative to the curves referenced. Modifying the shape of the curve will automatically update the dimension.

Using the Smart Dimension tool (Example 1-HVP Dimension)

- 1 Position the cursor over a line such that the Snap tool says “on”. Click the mouse.
- 2 Move the cursor to place a horizontal, vertical or parallel dimension.
- 3 Click at desired location to accept the final dimension.

Using the Smart Dimension tool (Example 2-Diameter Dimension)

- 1 Position the cursor over a circle such that the Snap tool says “on”. Click the mouse.
- 2 Move the cursor to place a diameter dimension.
- 3 Click at desired location to accept the final dimension.

Using the Smart Dimension tool (Example 3-Point/Point Dimension)

- 1 Position the cursor over an object such that the Snap tool says “endpoint”, “midpoint”, “vertex” or “center”. Click the mouse to specify the dimension start point.
- 2 Position the cursor over an object such that the Snap tool says “endpoint”, “midpoint”, “vertex” or “center”. Click the mouse to specify the dimension endpoint.
- 3 Move the cursor to place a horizontal, vertical or parallel dimension.
- 4 Click at desired location to create dimension.

Using the Smart Dimension tool (Example 4-Angular Dimension)

- 1 Position the cursor over a line such that the Snap tool says “on”. Hold the Shift key and click the mouse.
- 2 While still holding the Shift key, position the cursor over a second line such that the Snap tool says “on”. Click the mouse.
- 3 Release the Shift key.
- 4 Move and click at desired location to accept the angular dimension.

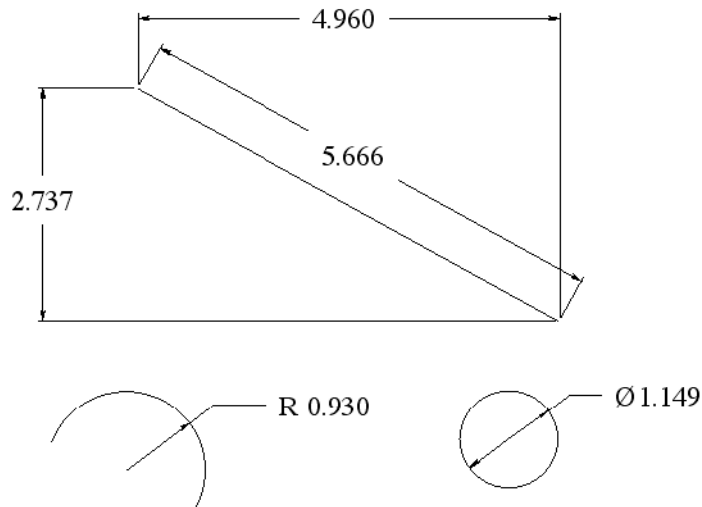
Using the Smart Dimension tool (Example 5-Minimum Distance Curve/Curve)

- 1 Hold the Shift key, and position the cursor over an object such that the Snap tool says “on”. Click the mouse.
- 2 Position the cursor over a second object such that the Snap tool says “on”. Click the mouse.

- 3 Move and click at desired location to accept the minimum distance dimension.
- 4 This creates a minimum distance dimension between two curves.

Using the Smart Dimension tool (Example 6-Minimum Distance Curve/Pt)

- 1 Hold the Shift key, and position the cursor over an object such that the Snap tool says “on”. Click the mouse.
- 2 Position the cursor over a second object such that the Snap tool says “endpoint”, “midpoint” or “center”. Click the mouse.
- 3 Move and click at desired location to accept the minimum distance dimension.
- 4 This creates a minimum distance dimension between a curve and a point on another curve.



Horizontal Dimensions

A Horizontal Dimension creates a dimension with horizontal leader lines and vertical extension lines. Horizontal and vertical for the drawing are defined by the x and y axis of the construction plane. To create a horizontal dimension select the object you desire to dimension and locate the position corresponding to the lower right text location. Modifying the object will automatically regenerate the dimension.

There are three types of horizontal dimensions available, from the sub-tool palette:

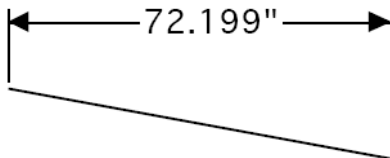


 **Horizontal Dimension tool**

This tool creates a single horizontal dimension, from a specified start point to a specified endpoint.

Using the Horizontal Dimension tool

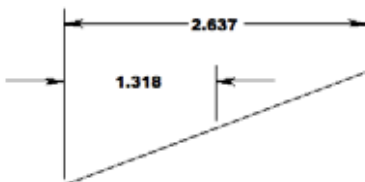
- 1 Click the mouse for the dimension start.
- 2 Click the mouse for the dimension end.

 **Horizontal Baseline tool**

This tool creates a horizontal series of dimensions with one baseline dimension of the entire distance.

Using the Horizontal Baseline tool

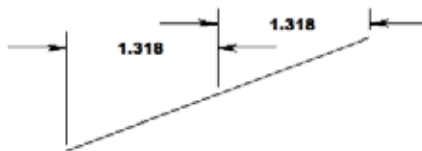
- 1 Click the mouse for the dimension start.
- 2 Click to set a series of horizontal dimensions.
- 3 Click the mouse for the dimension end.

 **Horizontal Chain tool**

This tool creates a horizontal series of individual dimensions over a specified distance.

Using the Horizontal Chain tool

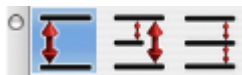
- 1 Click the mouse for the dimension start.
- 2 Click the mouse for the dimension end.
- 3 Continue to define individual dimensions along a surface.



Vertical Dimensions

A Vertical Dimension creates a dimension with vertical leader lines and horizontal extension lines. Horizontal and vertical for the drawing are defined by the x and y axis of the construction plane.

There are three types of vertical dimensions available, from the sub-tool palette:

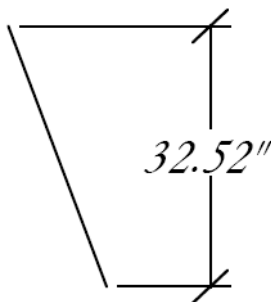


Vertical Dimension tool

This tool creates a single vertical dimension, from a specified start point to a specified endpoint.

Using the Vertical Dimension tool

- 1 Click the mouse for the dimension start.
- 2 Click the mouse for the dimension end.

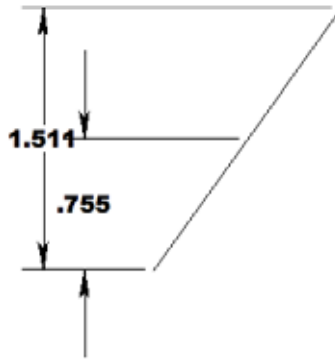


Vertical Baseline tool

This tool creates a vertical series of dimensions with one baseline dimension of the entire distance.

Using the Vertical Baseline tool

- 1 Click the mouse for the dimension start.
- 2 Click to set a series of vertical dimensions.
- 3 Click the mouse for the dimension end.

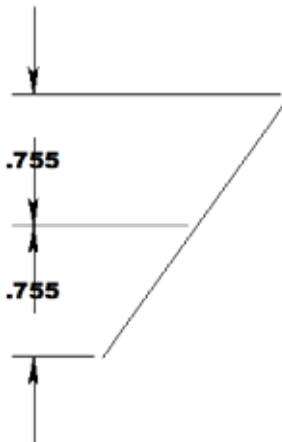


Vertical Chain tool

This tool creates a vertical series of individual dimensions over a specified distance.

Using the Vertical Chain tool

- 1 Click the mouse for the dimension start.
- 2 Click the mouse for the dimension end.
- 3 Continue to define individual dimensions along a surface.



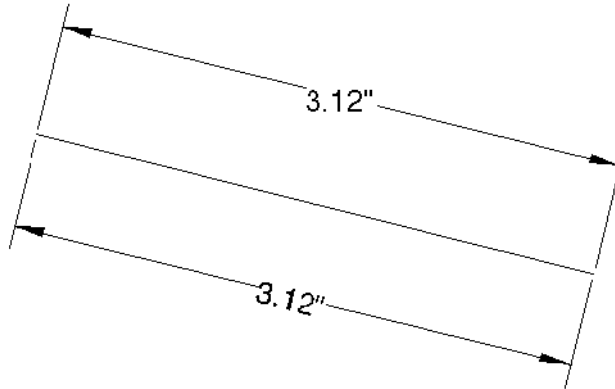
Parallel Dimension

A Parallel Dimension creates a dimension with parallel leader lines and perpendicular extension lines relative to the object selected.

Using the Parallel Dimension tool

- 1 Click the mouse for the dimension start.

- 2 Click the mouse for the dimension end.

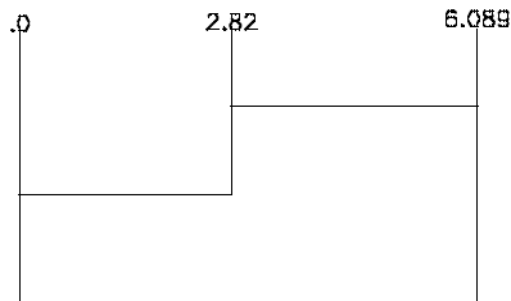


Ordinate Dimensions

The Ordinate Dimension tool dimensions distances relative to a base point. Two options are provided for creating horizontal and vertical ordinate dimensions.

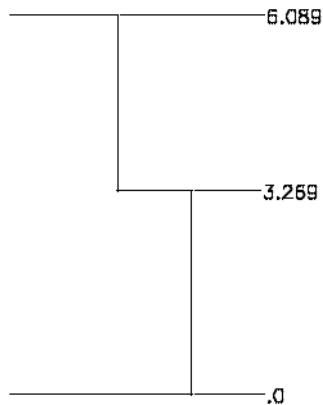
Using the Horizontal Ordinate Dimension tool

- 1 Click the mouse for the ordinate base point.
- 2 Click the mouse for additional ordinate dimensions.
- 3 Select a new tool to stop creating ordinate dimensions.



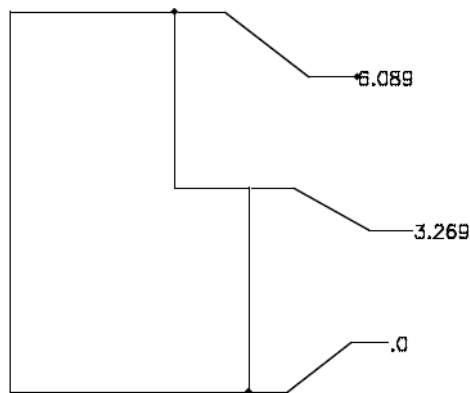
Using the Vertical Ordinate Dimension tool

- 1 Click the mouse for the ordinate base point.
- 2 Click the mouse for additional ordinate dimensions.
- 3 Select a new tool to stop creating ordinate dimensions.



Modifying Ordinate Dimensions

- 1 Select the Arrow tool to modify an ordinate dimension.
- 2 Box select the ordinate dimension near the text location.
- 3 Drag the text label (creating an elbow) to the new location.

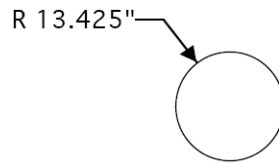
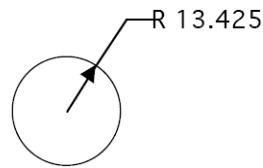


Radial Dimensions

A Radial Dimension measures the radius of a circle or arc and creates a dimension with either arrows inside or arrows outside. The resulting dimension is associative to the arc or circle.

Using the Radial Dimension tool

- 1 Select an arc or circle.
- 2 Click the mouse for the dimension end.

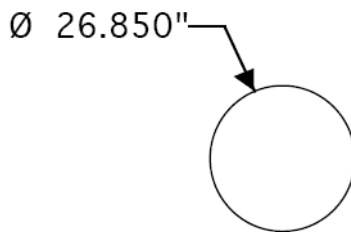
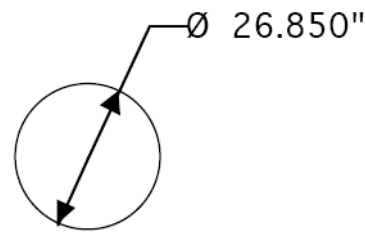
*Arrow Outside**Arrow Inside*

Diametrical Dimensions

A Diametrical Dimension measures the diameter of a circle. To create a diametrical dimension select a circle and indicate the location of the text. A diametrical dimension is linked to the original object such that any modifications to the object will automatically regenerate the dimension.

Using the Diameter Dimension tool

- 1 Select an arc or circle.
- 2 Click the mouse for the dimension end.

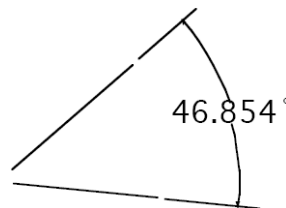
*Arrow Outside**Arrow Inside*

Angular Dimensions

Angular Dimensions measure the angle between any two lines.

Using the Angular Dimension tool

- 1 Select first line.
- 2 Select second line.



Center Mark Dimension

The Center Mark Dimension will locate and mark the center of a circle or entity. There are three tools available for finding a centerpoint.

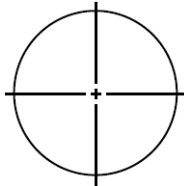


Center Mark Circle

To create a center mark simply select the desired circle. Note that if you change the circle by modifying the radius or moving its position the center mark will automatically regenerate because of the associativity that is created.

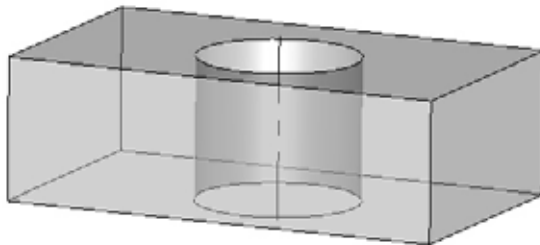
Using the Center Mark Dimension tool

- Select circle.



Centerline

The Centerline tool determines the center between two user-defined points. The Axis overlap text field, on the Prompt window, controls how far the line extends beyond each point.



Using the Centerline tool

- 1 Click to set the first point along which the centerline will follow.
- 2 Click the next point to draw the line.

Centerline Bisector

The Centerline Bisector tool references two lines. The bisector of the of the two lines defines the center line dimension.



Using the Centerline Bisector tool

- Click two lines (or circles) between which you want a centerline drawn.

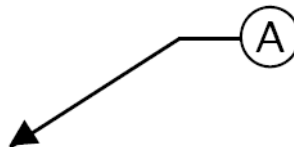
Balloon and Callout Dimensions

A Balloon Dimension creates a label with a surrounding graphic that may be a circle, square or triangle. A callout dimension contains no surrounding graphics. There are eight balloon dimensions and one callout dimension.



Using the Balloon Dimension tool

- 1 Select balloon type.
- 2 Click location for balloon arrow to point at.
- 3 Click location for text extension line to start at.
- 4 Use the Data Entry Fields to modify the text or width of the enclosing graphic.

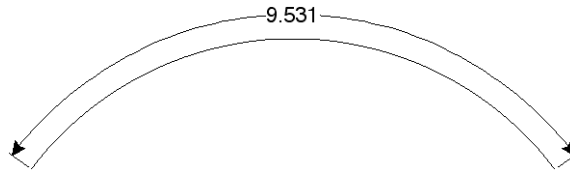


Length Dimension

The Length Dimension creates an associative dimension to the length of a curve.

Using the Length Dimension tool

- 1 Click dimension start location.
- 2 Click dimension end location.



Geometric Dimension and Tolerance

Geometric dimensioning and tolerancing (GD&T) is a language used on mechanical engineering drawings composed of symbols that are used to efficiently and accurately communicate geometry requirements for associated features on components and assemblies.



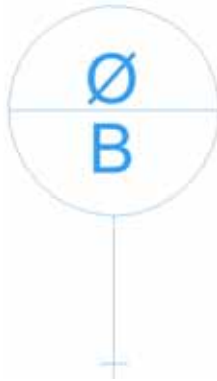
Datum Feature tool

A Datum Feature is the actual component feature used to establish a datum.



Datum Target tool

A Datum Target is a specified point, line or area on a part that is used to establish the datum reference plane for manufacturing and inspection operations.

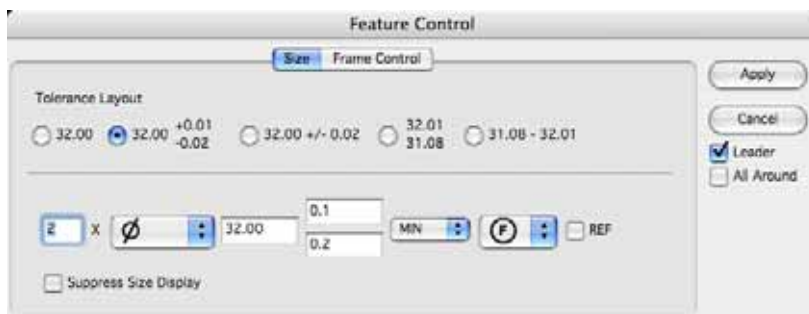


Feature Control Frame tool

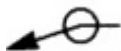
A Feature Control Frame is a rectangular box containing the geometric characteristics symbol, and the form, runout or location tolerance. If necessary, datum references and modifiers applicable to the feature or the datums are also contained in the box.

Size

The Size tab page is used to define the basic dimension and tolerance values along with other optional modifiers. This size specification is also known as the Limits of Size. Selecting one of the five Tolerance Layout radio buttons controls the tolerance style. The dialog controls are repositioned to match the current Tolerance Layout selection.

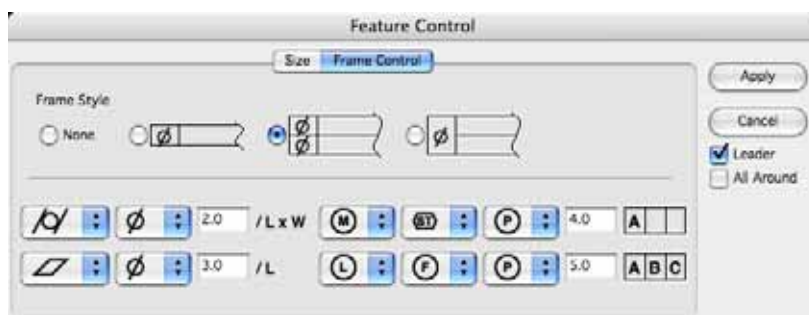


- Tolerance Layout – specifies the tolerance configuration.
- Number of Places – the X is used along with a value to indicate the number of times a dimension or feature is repeated on the drawing.
- Leader – adds an arrowhead and leader line.
- All Around – indicating that a tolerance applies to surfaces all around the part. Adds a circle about the leader line.



Frame Control Page

The Frame Control tab page is used to define the geometric relationships for the dimensioned feature. Selecting one of the four Frame Style radio buttons controls the layout for the feature control frames. The supported frame styles are None, Single, Dual, and Composite. The dialog controls are repositioned to match the current Tolerance Layout selection. The Feature Control tab also uses 2 additional support dialogs; Unit Basis and Datum Reference. The Unit Basis dialog is displayed by clicking on the region of the dialog box. The Datum Reference dialog is displayed by clicking on the (ABC) region of the dialog box.



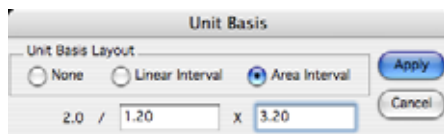
- Frame Style – indicates the type of frame for the feature control.
- *****/L – displays the Unit Basis dialog.



Displays the Datum Reference dialog.

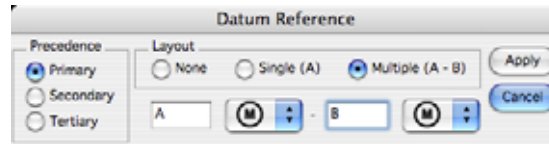
Unit Basis Dialog

When appropriate for a feature, the Unit Basis Dialog defines the unit basis interval (subregion) over which the tolerance value is applied. The unit basis interval can be one of “None”, “Linear”, and “Area”. A “None” interval (the default) applies the allowed tolerance over the entire feature. A “Linear” interval applies the allowed tolerance per any linear sub-length along the feature. An “Area” interval applies the allowed tolerance per any rectangular sub-area of the feature. If a linear interval is defined, the Feature Control tab page will display a /L indicator. If an area interval is defined, the Feature Control tab page will display a /LxW indicator.





















Datum Reference 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 and an optional modifier symbol. The letters I, O, and Q are not allowed. For a multiple datum, a second datum letter and optional modifier symbol follow a dash.



Symbol	Description	Geometry	Notes
	Straightness	Form	A condition where an element of a surface or an axis is a straight line.
	Flatness	Form	A condition of a surface having all elements in one plane.
	Roundness	Form	Describes the condition on a surface of revolution (cylinder, cone, sphere) where all points of the surface are intersected by any plane.
	Cylindricity	Form	Describes a condition of a surface of revolution in which all points of a surface are equidistant from a common axis.
	Profile of a Line	Profile	The condition permitting a uniform amount of profile variation, either unilaterally or bilaterally, along a line element of a feature.
	Profile of a Surface	Profile	The condition permitting a uniform amount of profile variation, ether unilaterally or bilaterally, on a surface.
	Angularity	Orientation	The condition of a surface, axis or centerplane, which is at a specified angle from a datum plane or axis.
	Perpendicularity	Orientation	The condition of a surface, axis or line, which is 90-degrees from a datum plane or a datum axis.

Symbol	Description	Geometry	Notes
	Parallelism	Orientation	The condition of a surface, line or axis, which is equidistant at all points from a datum plane or axis.
	Position Tolerance	Location	Defines a zone within which the axis or center plane of a feature is permitted to vary from true position.
	Concentricity	Location	Describes a condition in which two or more features, in any combination, have a common axis.
	Symmetry	Location	A condition in which a feature (or features) is symmetrically disposed about the center plane of a datum feature.
	Runout	Runout	The composite deviation from the desired form of a part surface of revolution through on full rotation (360 deg) of the part on a datum axis.
	Total Runout	Runout	The simultaneous composite control of all elements of a surface at all circular and profile measuring positions as the part is rotated through 360.
	Maximum Material Condition (MMC)	Modifier	A condition of a part feature wherein it contains the maximum amount of material within the stated limits of size.
	Least Material Condition (LMC)	Modifier	A condition of a part feature of size wherein it contains the least (minimum) amount of material; examples, largest hole size and smallest shaft size. It is opposite to maximum material condition.
	Projected Tolerance Zone	Modifier	Applies to a hole in which a pin, stud, screw, etc., is to be inserted. It controls the perpendicularity of the hole to the extent of the projection from the hole and as it relates to the mating part clearance.
	Tangent Plane	Modifier	Indicating a tangent plane is shown. The symbol is placed in the feature control frame following the stated tolerance.
	Free State Variations	Modifier	A term used to describe the distortion of a part after removal of forces applied during manufacture.

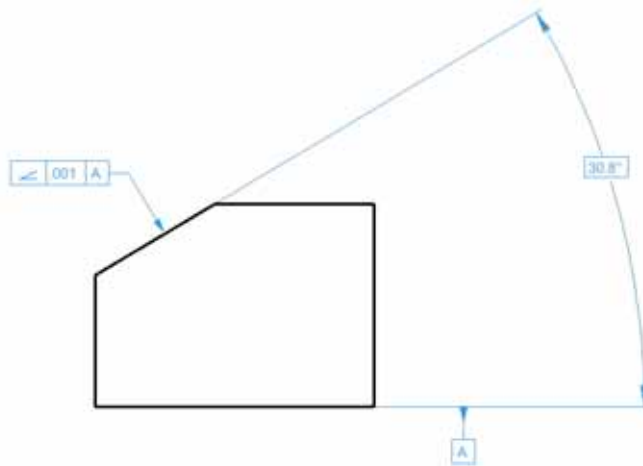
Symbol	Description	Geometry	Notes
	Diameter		Indicates a circular feature when used on the field of a drawing or indicates that the tolerance is diametrical when used in a feature control frame.
	Conical Taper		Used to indicate taper for conical tapers. This symbol is always shown with the vertical leg to the left.
	Slope		Used to indicate slope for flat tapers. This symbol is always shown with the vertical leg to the left.
	Counter/Spotface		Used to indicate a counterbore or a spotface. The symbol precedes the dimension of the counterbore or spotface, with no space.
	Countersink		Used to indicate a countersink. The symbol precedes the dimensions of the countersink with no space.
	Depth/Deep		Used to indicate that a dimension applies to the depth of a feature. This symbol precedes the depth value with no space in between.
	Square		Used to indicate that a single dimension applies to a square shape. The symbol precedes the dimension with no space between.
R	Radius		Creates a zone defined by two arcs (the minimum and maximum radii). The part surface must lie within this zone.
SR	Spherical Radius		Precedes the value of a dimension or tolerance.
S \emptyset	Spherical Diameter		Precedes the tolerance value where the specified tolerance value represents spherical zone. The symbol for spherical diameter precedes the size dimension of the feature and the positional tolerance value, to indicate a spherical tolerance zone.

Symbol	Description	Geometry	Notes
CR	Controlled Radius		Creates a tolerance zone defined by two arcs (the minimum and maximum radii) that are tangent to the adjacent surfaces.
ST	Statistical Tolerance		Assign tolerances to related components of an assembly on the basis of sound statistics.

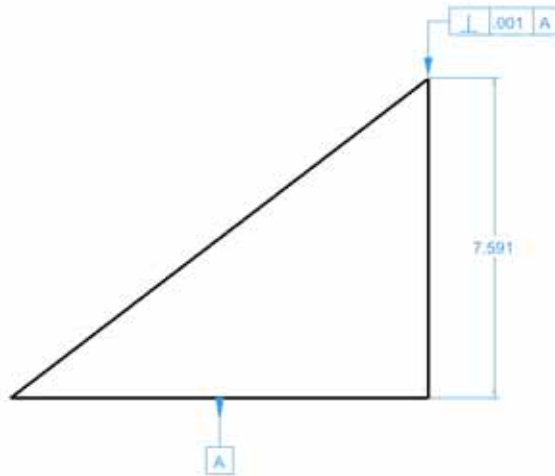
GD&T Examples

Below are several examples of uses of GD&T dimension tools relative to geometry.

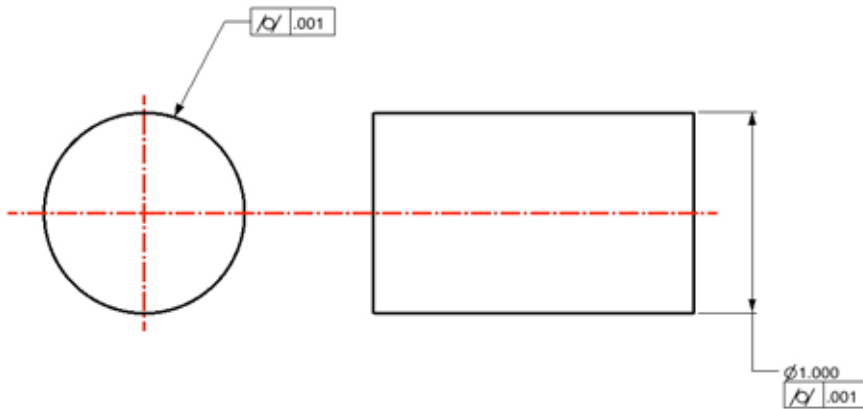
Angularity



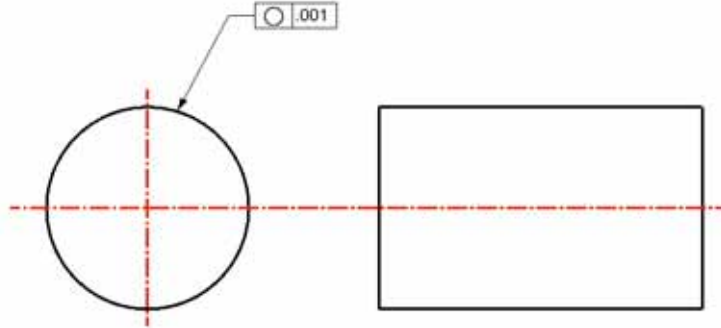
Perpendicularity



Cylindricity



Circularity



Fillet & Chamfer



These tools on the tool palette construct fillets and chamfers from corners formed by nonparallel lines or curves. The fillets and chamfers are automatically trimmed. If you do not want them trimmed, hold down the Option key (or Alt key) while you are selecting the objects to fillet or chamfer.

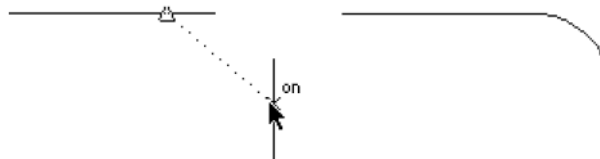


Fillet Two Lines

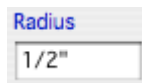
This tool constructs an arc tangent to the two objects you click.

Adding a fillet

- 1 Click the Fillet Two Lines tool.
- 2 Enter the arc radius in the Data Entry Window. The default radius is 0.25 inch.
- 3 Click the objects you want to fillet. You can also hold down the Shift key and click once inside the corner you want to fillet.



If you hold down the Option key (or Alt key) while you select the objects to fillet, the objects are not trimmed. The Data Entry Fields allow you to specify the Radius of the fillet either before or after you create the fillet.





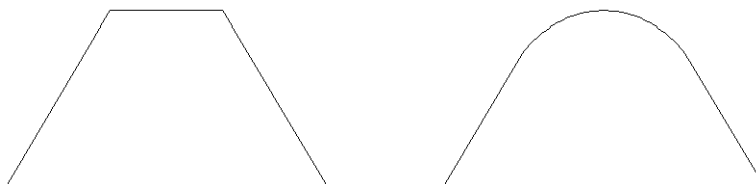
Fillet Three Lines

This tool constructs a fillet tangent to the three objects you choose.

Adding a fillet three line

- 1 Click the Fillet Three Lines tool.
- 2 Click the three lines you want to fillet.

If you hold down the Option key (or Alt key) while you select, the lines are not trimmed. There are no entries on the Data Entry Window.

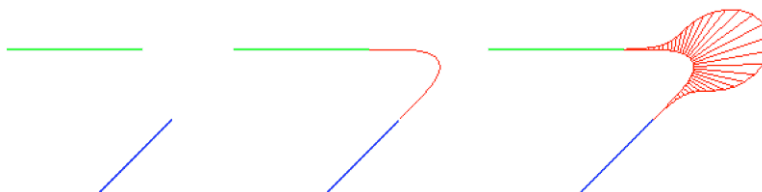


Smooth Fill Curve

The Curve Fill tool located in the Curve Fillet tool palette is used to create a smooth blended curve between any two curves. Unlike the Fillet tool, curves do NOT need to be planar.

Using the Smooth Fill Curve tool

- 1 Select the Smooth Fill Curve tool icon from the 2D fillet tool palette.
- 2 Select near the end of the first curve.
- 3 Select near the end of the second curve.



Chamfer Two Lines

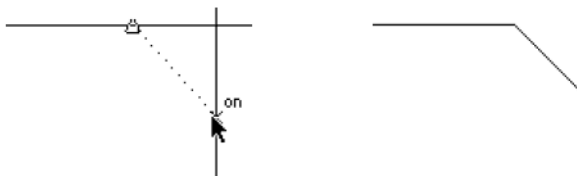
This tool creates a chamfer across a corner at the specified distance from the intersection of two lines.

Using the Chamfer Two Lines tool

- 1 Click the Chamfer Two Lines tool.
- 2 In the Data Entry Window, enter the distance you want the chamfer from the corner.

- 3 Click each line making up the corner you want chamfered.

You can also hold down the Shift key and click once inside the corner you want to chamfer.



The lines are automatically trimmed or extended. If you hold down the Option key (or Alt key) while you select the objects to chamfer, the objects are not trimmed.

The Data Entry Fields allow you to specify the distance (Length) from the chamfer to the intersection of the corner lines.

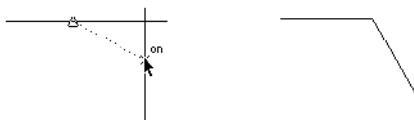


Chamfer Angle

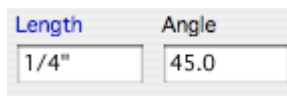
This tool creates a chamfer at the specified angle and distance from the corner. The specified angle is the angle between the chamfer and the second line of the corner. The specified length is the distance between the corner and the intersection of the chamfer and the second line of the corner.

Using the Chamfer Angle tool

- 1 Click the Chamfer Angle tool.
- 2 In the Data Entry Window, enter the length you want the intersection of the chamfer and the second line of the corner, from the corner.
- 3 In the Data Entry Window, enter the angle you want between the chamfer and the second side.
- 4 Click each line making up the corner you want chamfered.



The Data Entry Fields allow you to set the distance (Length) from the intersection as well as the Angle.



Trim Tools



The Trim tool palette contains commands for trimming, extending, relimiting, connecting, and offsetting curves.

Trim

The Trim tool shortens a line to the specified boundary. If you hold down the Option key (or Alt key) while using this tool, it becomes the Relimit tool.

Using the Trim tool

- 1 Select the object that limits the trim.
- 2 Select the Trim tool. If necessary, use Shift-Click to select more boundary objects.
- 3 Click the section of the object to be discarded.



There are no Data Entry Fields entries for the Simple Trim command.

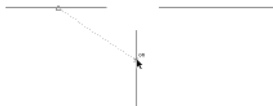
Corner Trim

This tool creates a corner from the two objects you specify. Lines are extended or shortened to create the corner.

Using the Corner Trim tool

- 1 Click the Corner Trim tool.

- Click each object. You can also press and hold the Shift key and click inside the about-to-be-created corner.



There are no Data Entry Fields entries for the Corner Trim tool.



Extend

This tool extends the boundaries of an existing curve a designated distance. There are three methods by which a curve can be extended and three styles to choose from. These settings are available on the Prompt window.

Extension Method

- Extend to Curve
- Extend by Distance
- Extend dynamically

- Extend to Curve - select a boundary to which you want to extend and then select the point you want extended.
- Extend by Distance - enter the distance by which you want an end extended, then select the end you want to extend.
- Extend Dynamically - select the end you want to extend, then drag to the desired length.

Extension Style

There are three extension styles available, depending on the resulting geometry you want.

- Linear
- Quadratic
- Cubic

Extending to an existing curve

- From the Method drop-down, choose Extend to Curve.
- (optional) Choose the extension style you want.
- Select the existing curve to which you want to extend.
- Click the end you want to extend.

Extending a specified distance

- From the Method drop-down, choose Extend by Distance.

- 2 (optional) Choose the extension style you want.
- 3 Enter the distance you want to extend. Press the Enter key to accept the value.
- 4 Click the end you want to extend.

Extending dynamically

- 1 From the Method drop-down, choose Extend Dynamically.
- 2 (optional) Choose the extension style you want.
- 3 Click the end you want to extend.
- 4 Drag to the point where you want and click.



Segment To Curve

This tool divides a line or curve at intersections with other lines or curves.

Using the Segment To Curve tool

- 1 Select the objects that limit the segmentation.
- 2 Select the Segment To Curve tool. If necessary, use Shift-Click to select more boundary objects.
- 3 Click the object to be segmented.

The selected object is segmented at the boundary objects. Even though you cannot see the segmentation on the screen, you can select parts of the segmented line by clicking. If you hold down the Option key while selecting the line to be segmented, the new segment appears in the characteristics of the current pen style.



There are no Data Entry Field entries for the Segment To Curve tool.



Segment at Interval

The Segment at Interval Tool subdivides a curve into a specified number of equal length segments.

Using the Segment At Interval tool

- 1 Set the #Segments value in the Data Entry Window.
- 2 Select the object to be divided.



The Data Entry Field sets the number of segments the curve is to be divided.

#Segments

Connect Curve

The Connect Curve tool is used to merge endpoints between two curves. The tool will move the endpoint of the second selected curve to the end of the first selected curve.

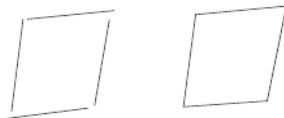
Using the Connect Curve tool (Example 1)

- 1 Click the Connect Curve tool.
- 2 Click each object near the ends that are to be connected.



Using the Connect Curve tool (Example 2)

- 1 Click the Connect Curve tool.
- 2 Set the search tolerance in the Data Entry Window.
- 3 Box select a collection of curves to merge endpoints.



Join Curve

The Join Curve tool is used to smoothly join two curves. Joining of the curves will create a single curvature continuous curve from the original two curves. If the curves are not tangent, the shape at the merge location will not precisely match.

Using the Join Curve tool

- 1 Click the Join Curve tool.

- 2 Set the join tolerance in the Data Entry Window. The smaller the tolerance, the closer the joined curve will be to the originals.
- 3 Click each object near the ends that are to be joined.



The two curves are joined as one continuous G2 curve. If the original curves were not tangent, there may be some deviation from the resulting curve to the original near the join point.

Offset to Value

There are three tools available for placing offset curves. To access the sub-tools, click the Offset Curve tool on the Trim tool palette and drag the sub-toolset from the Prompt window.



Offset To Value

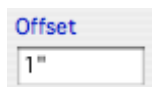
This tool creates a curve that is offset from an existing curve by specifying a distance and direction.

Using the Offset to Value tool

- 1 Click the Offset To Value tool.
- 2 Select the curve to offset.
- 3 Click to indicate the offset direction for the curve.



The offset distance can be changed by entering a new value in the Data Entry Window.

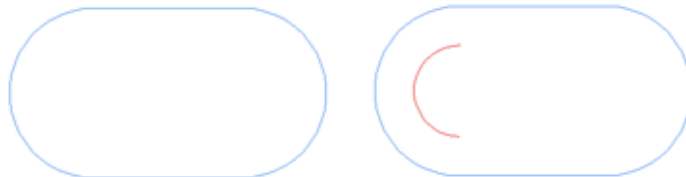


Offset by Point

This tool creates a curve that is offset from an existing curve by specifying the point where you want the curve to appear.

Using the Offset by Point tool

- 1 From the Offset Curve tool sub-toolset, click the Offset by Point tool.
- 2 Select the curve to offset.
- 3 Position your cursor where you want the curve to appear and click.

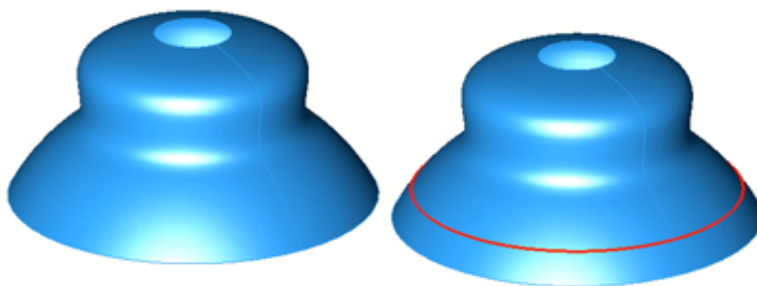


Offset Face Edges

This tool offsets an edge along a face or surface, a specified distance. The resulting offset curve lies precisely on the surface.

Using the Offset Face Edges tool

- 1 From the Offset Curve sub-toolset, click the Offset Face Edges tool.
- 2 Select the surface or solid face to offset edge.
- 3 Click the edge to offset.



Region Trim

The Region Trim tool supports rapid trimming of closed regions. Click inside or outside to control which side is trimmed. Hold down the Option key (Control key on PC) to create a copy of the trim results. Pressing the Shift key will allow the user to select which curves are to be used in the trimming operation.

Using the Region Trim tool

- 1 Click the Region Trim tool.
- 2 Click inside or outside the region to trim.

The three images below show the original trim region and the impact on clicking outside and inside the region.



Trim Region



Click Outside



Click Inside

Transformations



Object transformations are a powerful method for modifying the shape and position of an object. There are eleven tools for transforming objects available through this tool palette. From left to right, across the transformation tool palette they are:

Basic Transforms

- Translate
- Rotate
- Scale/Stretch
- Mirror
- Linear Duplicate
- Polar Duplicate

Advanced Transforms

- Path Duplicate
- Align
- Mate and Connect
- Stretch
- Twist

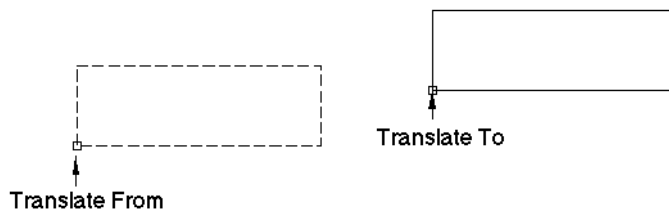


Translate

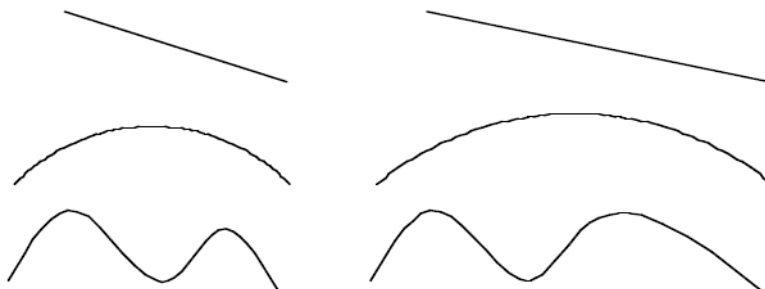
The Translate tool is used to move an object from one location to another. Hold the option key to perform a translate copy.

Using the Translate tool

- 1 Click the Translate tool.
- 2 Select the objects to move if not already selected.
- 3 Specify two locations that indicate the start and end movement vector.
- 4 Use the Data Entry Fields to precisely adjust dx, dy, and dz move components.



The move tool will move entire objects or just selected points. In the next example we move only the selected endpoints of a collection of curves.



Translate Selected Points

Rotate

The Rotate tool changes the angle of an object about an axis. The Rotate command has the following options:

- Rotate about one point
- Rotate between three points
- Hold down the Option key to copy

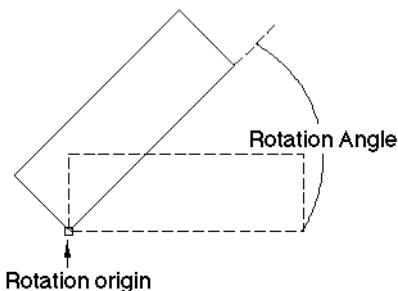


Rotate One Point

The Rotate One Point option rotates selected objects about a user supplied origin and angle. The default axis of rotation is the construction plane normal. Select the pull down menu in the prompt line to choose an alternate rotation axis.

Using the Rotate One Point tool

- 1 Select objects to rotate if not already selected.
- 2 Pick the axis to rotate about if not the work plane.
- 3 Type in the rotation angle in the Data Entry Window.
- 4 Click a point which is the rotation origin.



Press and hold the Option key (Alt key) if you would like to perform a rotate copy.

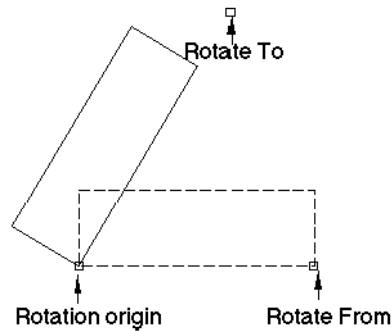


Rotate Three Points

The Rotate Three Points command rotates an object by three points specified by the user. The first point defines the center of rotation, the second is the from point, and third point is the rotate to point. Internally, a transformation matrix is created that rotates the object an angle defined by the three points.

Using the Rotate Three Points tool

- 1 Select objects to rotate if not already selected.
- 2 Click a point that is the rotation center.
- 3 Click a point that is the rotation from point.
- 4 Click a point that is the rotation to point.
- 5 The Data Entry Fields fills the angular value for the three points. Use the Data Entry Fields to change to a different value if needed.



Press and hold the Option key (Alt key) if you would like to perform a rotate copy.



Scale

The Scale command provides options to enlarge or shrink an object. The user interface provides two types of scaling: uniform and differential. Uniform scaling provides the same scale factor to the x, y, and z components. Differential (or non-uniform) scaling can scale each axis with a different value. Use the pull down menu to select uniform or differential scaling.

Uniform Scale

The Uniform Scale tool will scale using input from mouse clicks or a scale value entered in the Data Entry Window. The following two examples show how to accomplish uniform scaling.

Using the Uniform Scale with Data Entry Window

- 1 Select objects to rotate if not already selected.
- 2 Click an anchor point for the scale operation. This represents the scale origin and stays fixed throughout the scaling operation.
- 3 Click a reference point for the scale magnitude.
- 4 Click an ending point for the scale magnitude.

From the above mouse clicks, the scale origin is the first click and the scale value is the distance as $\text{Scale} = (\text{endpoint} - \text{Anchor Point}) / (\text{Reference Point} - \text{Anchor Point})$. The mouse click method supports both shrinking and stretch of curves, surfaces, and solids. Press the option key to perform a scale copy.

Using the Uniform Scale with Data Entry Window

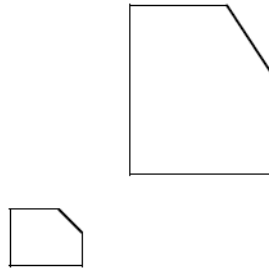
- 1 Select objects to rotate if not already selected.
- 2 Click an anchor point for the scale operation. This represents the scale origin and stays fixed throughout the scaling operation.
- 3 Type in a scale value using the Data Entry Fields and hit Enter on the keyboard.

Differential Scale

The Differential Scale tool will expand or shrink a curve, surface or solid, in a non-uniform manner with respect to the x, y or z axis. The following example shows how to accomplish differential scaling.

Using the Differential Scale tool

- 1 Select objects to rotate if not already selected.
- 2 Use the Data Entry Fields to specify the x, y, and z scale components.
- 3 Click an anchor point for the scale operation. This represents the scale origin.
- 4 Hold the option key to perform a copy.



Differential Scale Copy (x=2, y=3)

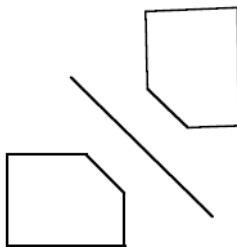


Mirror

The Mirror command produces a mirror image of objects through a mirror plane. A mirror plane is defined by one of the primary axes (x, y, z) or a user-defined vector. Use the pull down in the message line to specify the plane or axis.

Using the Mirror 2 Pts tool

- 1 Select objects to rotate if not already selected.
- 2 Select the 2 Pts Option from the Prompt pull down.
- 3 Click beginning point for mirror reference line.
- 4 Click ending point for mirror reference line.
- 5 Hold down the Option key to perform a mirror copy.

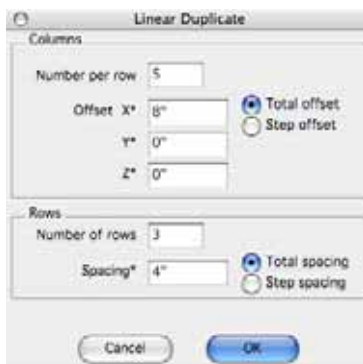


Linked Mirror

The Linked Mirror tool creates an associative, mirrored copy of one or more objects so when the original (parent) object is edited or moved, the associative (child) object will as well.

Linear Duplicate

The Linear Duplicate tool creates copies of an object based on specifying rows, columns, and spacings. Columns are the vertical series in an array whereas rows are the horizontal series. The command displays the dialog box below to gather information related to columns, rows, and spacing.



Column Information

Number per Row Specifies the total number of objects per row. This includes the selected object.

Offset X, Y, Z The horizontal x, y, and z distances associated with the combined columns or individual column offsets. The asterisk indicates that these fields can be filled in from the Snap tool by referencing data in the drawing.

Total Offset A radio button indicating the Offset XYZ values correspond to the combined columns.

Step Offset A radio button indicating the Offset XYZ values correspond to the individual column offsets.

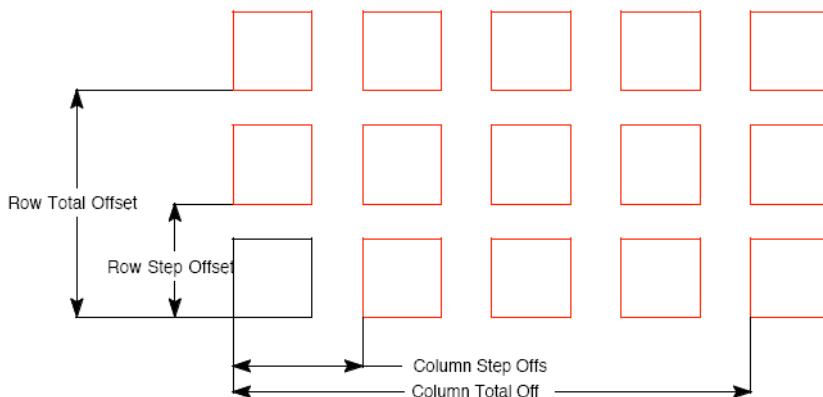
Row Information

Number of rows Specifies the total number of rows.

Spacing The vertical spacing between rows.

Total spacing A radio button indicating the vertical spacing corresponds to the combined row height.

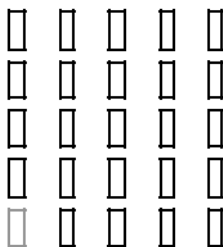
Step spacing A radio button indicating the vertical spacing corresponds to the individual row spacing.



Linear Duplicate Settings

Using the Linear Duplicate tool

- 1 Select objects to rotate if not already selected.
- 2 Specify the row, column, and spacing information in the linear duplicate dialog box.

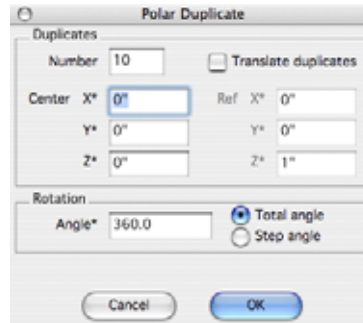


5x5 Linear Duplicate



Polar Duplicate

The Polar Duplicate tool creates multiple copies of objects arranged in a uniform circular pattern.



Duplicates Information

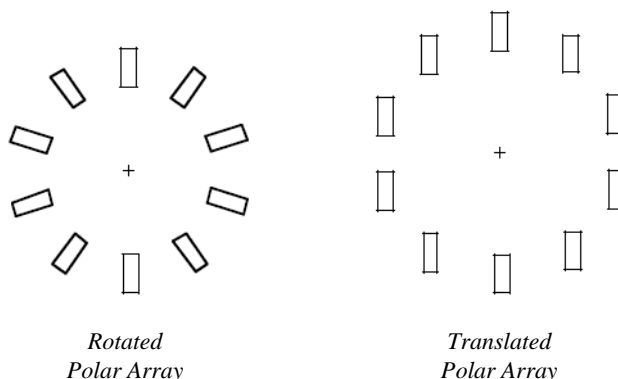
- Number** Specifies the number of rotational copies. This includes the selected object.
- Center X, Y, Z** The center of the polar array. The asterisk indicates that these fields can be filled in from the Snap tool.
- Ref X, Y, Z** The reference point when translate duplicates is checked. This point serves as the origin for the object rotated.
- Translate duplicates** A checkbox indicating that the objects are only translated into the polar position. If not checked, the objects are rotated at each duplicated position.

Rotation Information

- Angle** Specifies the polar angle.
- Total Angle** A radio box indicating the angle is the total polar angle.
- Step Angle** A radio box indicating the angle is the individual angle between successful polar increments.

Using the Polar Duplicate tool

- 1 Select objects to rotate if not already selected.
- 2 Specify the number, center, and angle information in the polar duplicate dialog box.

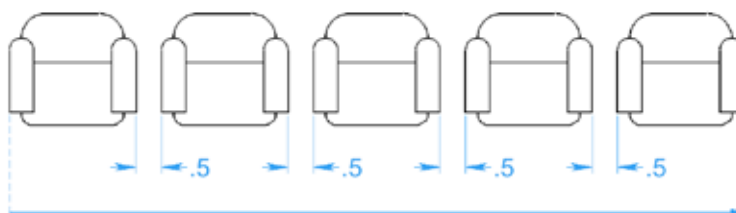


Linear Array

This tool creates and distributes copies based on the starting reference point and cursor location. As you move your cursor, the angle and number of copies increases or decreases. The spacing value describes the absolute distance between each consecutive object.

Using the Linear Array tool

- 1 Select the object you want create an array with.
- 2 Click the Linear Array tool.
- 3 Click to select the starting point for the array.
- 4 (optional) Enter a value in the Spacing text field, on the Data Entry window. Press the Enter key.
- 5 Drag your cursor in the direction you want the linear array. As you drag your cursor the array increases or decreases in size, depending on which way you're dragging.
- 6 Click to set the end of the array.



Linear Array

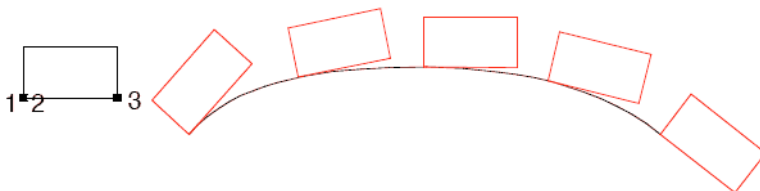


Path Duplicate

This tool creates copies of an object along a curve (line, spline, arc, circle or ellipse). The tool also has the ability to reference surface normals, aligning the selected object with the normals of a surface.

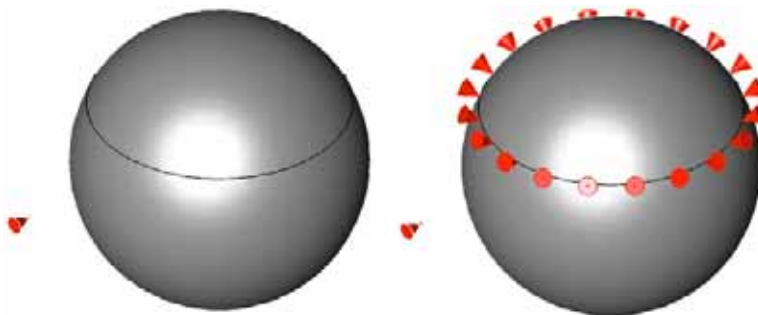
Using the Path Duplicate tool

- 1 Select objects to rotate if not already selected.
- 2 Click origin point for the objects you want to translate along path. This point will move to the path start (1 in the figure below).
- 3 Click two points defining an alignment vector. This vector defines the alignment to the slope path (2 and 3 in the figure below).



Referencing surface normals

- 1 Press and hold the Ctrl key (Option for Mac) to enable the path translation.
- 2 Click the translation path for the object.
- 3 Click the copy origin. The location from the selected body will be translated to a location along the path.
- 4 Select two points for the alignment axis. The axis will be aligned with the surface normal calculated from the location on the path.
- 5 Select the body for normals alignment. The surface normals will be calculated from this referenced body, which can be a surface or solid.



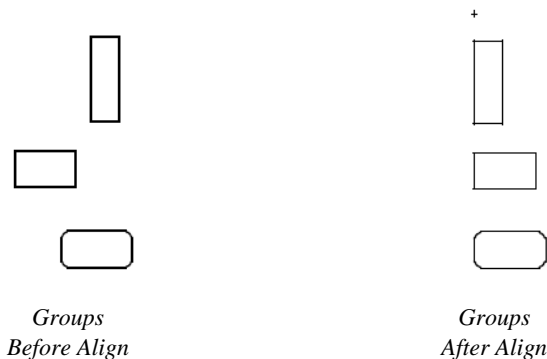
Align

The Align tools provide options for positioning objects relative to other objects by:

- Left & Right sides
- Centers
- Tops & Bottoms

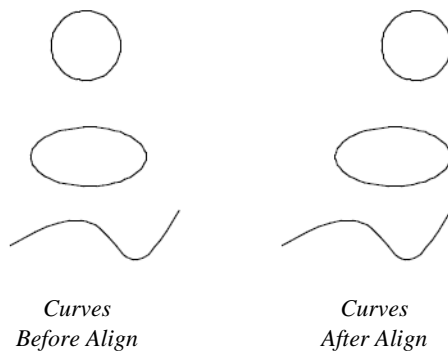
Align Left Sides

The Align Left Sides option will move selected objects such that their left extents are aligned.



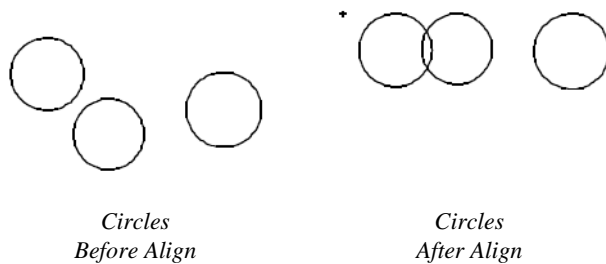
Align Right Sides

The Align Right Sides option will move selected objects such that their right extents are aligned.



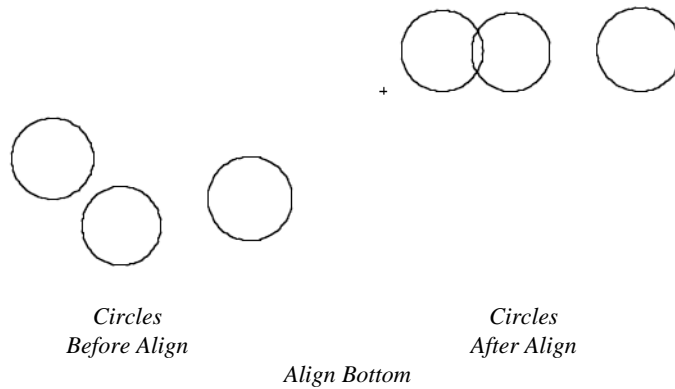
Align Tops

The Align Tops option will move selected objects such that their top extents are aligned.



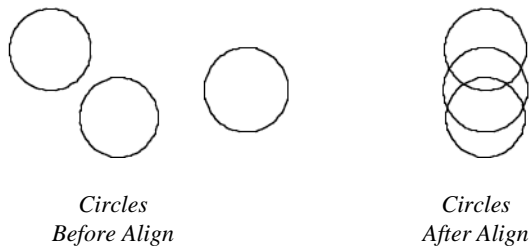
Align Bottoms

The Align Bottoms option will move selected objects such that their bottom extents are aligned.



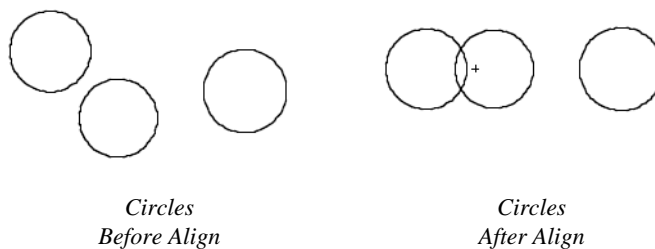
Align Centers Horizontal

The Align Center Horizontal option will move selected objects such that the center of their bounding extents are aligned.



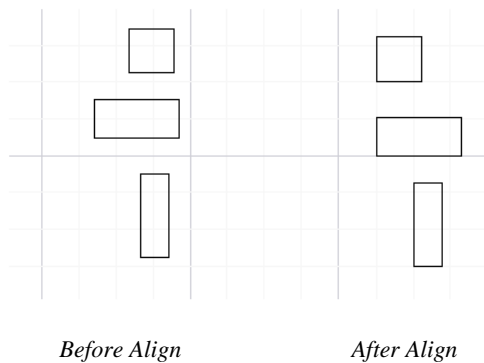
Align Centers Vertical

The Align Center Vertical option will move selected objects such that the center of their bounding extents are aligned.



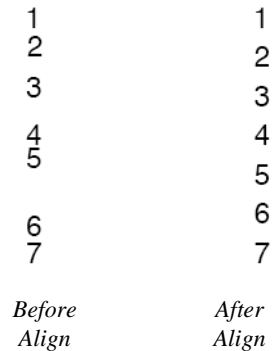
Align Grid

The Align Grid option will move selected objects such that the lower left of their bounding extents is at a grid node.



Align Spaced Vertical

The Align Spaced Vertical option will move selected objects such that the vertical bounding box spacing is even between the objects.



Align Spaced Horizontal

The Align Spaced Horizontal option will move selected objects such that the horizontal bounding box spacing is even between the objects.



Mate and Connect

The Mate and Connect tools provide options for positioning and aligning faces of parts relative to other faces by:

- Connect Point
- Mate Faces
- Align Faces
- Insert Faces



Connection

The Connection tool will move a part to another part, connecting two points between the parts. The Connection is associative; moving the parent part will move the other to maintain the connection.

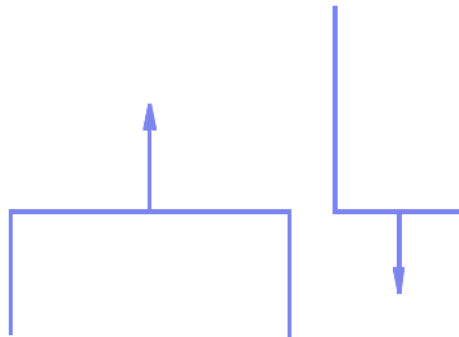
Using the Connection tool

- 1 Select the Connect tool from the Advanced Transforms tool palette.
- 2 Select the face to move.
- 3 Select the connection point on the face to move from.
- 4 Select the face to move to (parent part).
- 5 Select the connection point on the face to move to.



Mate Faces

The Mate Face tool will rotate a selected planar face such that the face normal is exactly the opposite as the referenced face. The Mate Faces tool is associative; moving one part will move the other to maintain the mated faces.

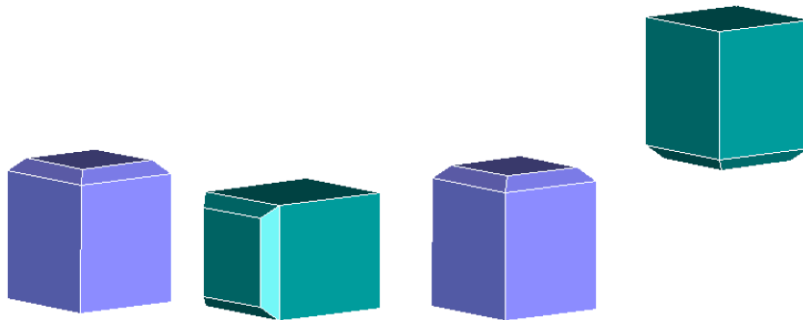


Mate Face Resulting Orientation

Using the Mate Face tool

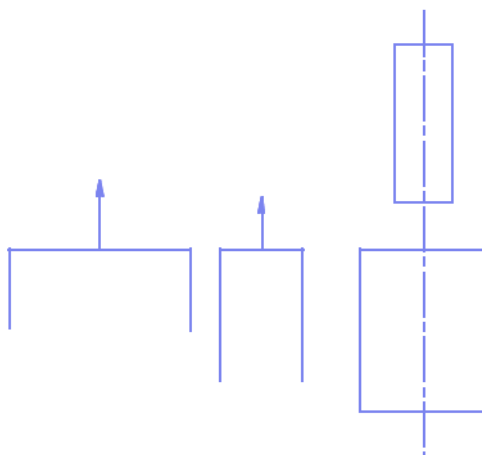
- 1 Select the Mate Face tool from the Advanced Transforms tool palette.

- 2 Select the planar face to align.
- 3 Select the planar face to align to.

*Align Tool*

Align Faces

The Align Faces tool will rotate a selected planar face such that the face normal is exactly the same as the referenced face. Align will also allow selection of cylinder faces in which case the axis will be aligned to each other. The Align Faces tool is associative; moving one part will move the other to maintain the aligned faces.

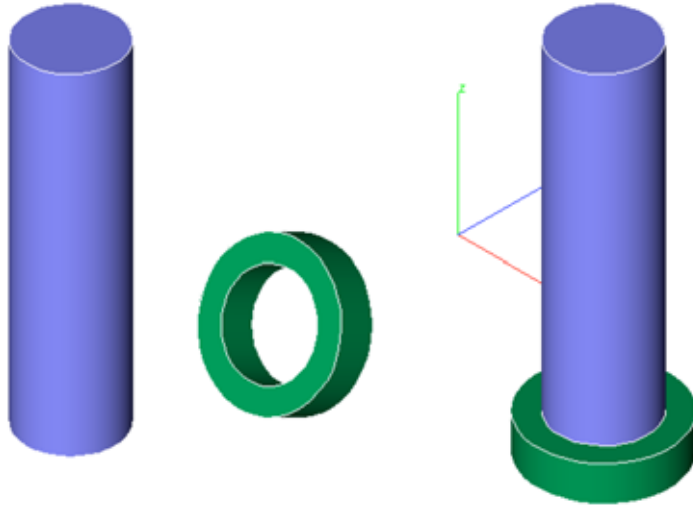


Planar *Cylinder*
Align Face Resulting Orientation

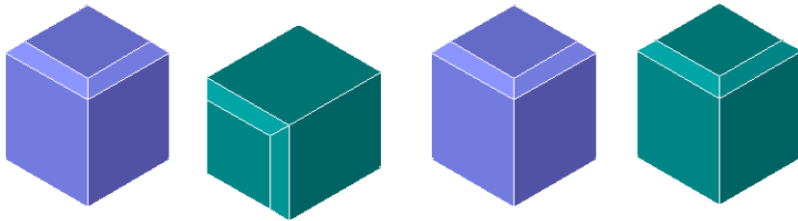
Using the Align Face tool

- 1 Select the Align Face tool from the Advanced Transforms tool palette.
- 2 Select the face to align (planar or cylindrical).

- 3 Select the face to align to (same face type as above).



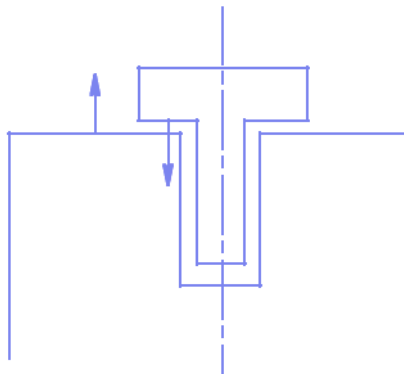
Cylinder Axis Align



Planar Face Align

Insert Face

The Insert Face tool will move and align cylindrical faces. Typically this is used to insert a bolt type shape into a hole type shape. The Insert Face tool is associative, moving the hole shape will automatically update and move the inserted face.

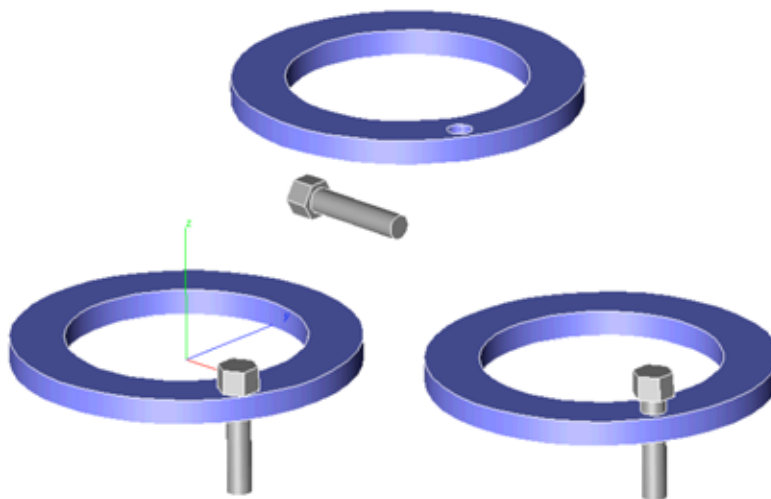


Insert Face Resulting Orientation

Using the Insert Face tool

- 1 Select the Insert Face tool from the Advanced Transforms tool palette.
- 2 Select the face to align (planar or cylindrical).
- 3 Select the face to align with (same face type as above).
- 4 Select the face to mate.
- 5 Select the face to mate with.

If the first face you select is cylindrical, the second alignment face must also be cylindrical. In the case of cylindrical faces, the axes of the cylinders are aligned.



Insert Face Examples



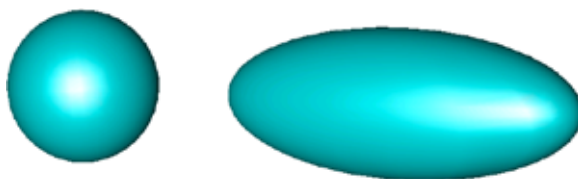
Stretch

The Stretch tool is used to stretch curves, surfaces, and solids. Stretching is a non-uniform scaling operation along a stretch axis.

The stretch axis is defined by two points specified by the user. The material to be stretched is bounded by planes perpendicular to the stretch axis at the two supplied points. The third point defines how much the body is stretched. The distance from the third point to the second point defines the distance stretched.

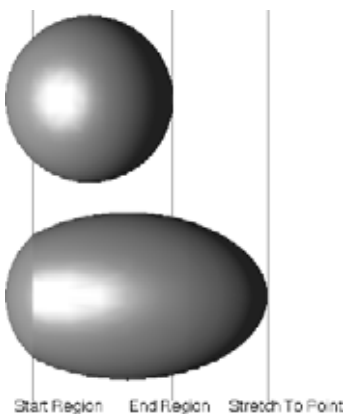
Using the Stretch tool

- 1 Select the Stretch tool from the advanced transformations tool palette.
- 2 Specify the start and end regions for stretching.
- 3 Specify the stretch to point. Material is stretched from the second point to this stretch point.
- 4 To stretch the entire body, specify two points that lie outside the body.



Global Stretch of a Sphere

The Stretch tool can also be used to stretch regions within a body. To perform a local stretch, specify one or both points within the body.



Local Stretch Example

Note: Local stretching introduces new faces and edges at the stretch start or end region. Continuity at the new edges is positional only.

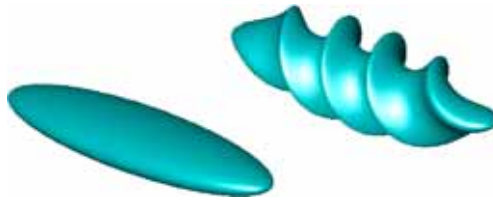


Twist

The Twist command is a warp class tool that will apply an angular twist to a curve, surface or solid. Use the Twist tool to warp a simple body into a shape not achievable with normal parametric modeling methods. Note you can twist a curve, surface or solid.

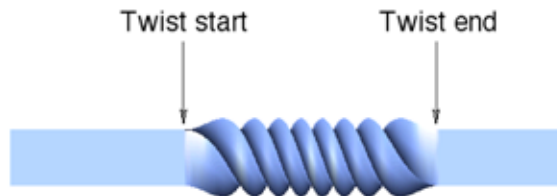
Using the Twist tool

- 1 Select the Twist tool from the advanced transform tool palette.
- 2 Specify the twist angle in the data entry window.
- 3 Specify the start point for the twist region.
- 4 Specify the endpoint for the twist region.



Global Twist Example

The start and endpoints specify the axis for the twist. Perpendicular to the axis start and endpoints are two planes defining the volume of material impacted by the twist operation. If the two points lie within the body, a localized twist operation occurs.



Local Twist Example

Constraints

Dimension and constraint tools are available to help manage the geometric relationships between 2D shapes. These relationships are used to remember design intent of 2D sketches, which typically drive 3D solids. Modifying any object within a system of constraints will update all other dependent objects accordingly.

The Constraints tool palette contains a variety of constraint types, including distance, radius, diameter, coincident, tangent, horizontal, vertical, perpendicular, offset, concentric, parallel, equal, collinear, and symmetry.



Sketches may also be defined on a face of a solid. In this situation, the sketch is dependent on the geometry of the face and updates as the face changes. This includes the ability for the sketch to change its orientation to match any changes of the face orientation.



Smart Dimension

This tool creates a user-defined distance constraint.

Using the Dimension tool

- 1 Click the Dimension tool on the Constraints tool palette.
- 2 Click on a 2D shape to start the dimension.
- 3 Drag your cursor in the direction you want the dimension placed.
- 4 Click again to place the dimension.



Auto Constraints

This tool automatically applies constraints to user-defined curves and dimensions. This is particularly useful when working with shapes and objects created using a different drawing tool. This tool adds the following constraints to selected objects:

- Horizontal
- Vertical
- Tangent
- Perpendicular
- Concentric
- Coincident
- Equal
- Smart Dimensions

Using the Auto Constraints tool

- 1 Click the Auto Constraint tool on the Constraints tool palette.
- 2 Click the objects you want to constrain.

Horizontal Constraint

This tool adds a horizontal constraint to a line. You can select one or more line segments to apply the constraint. Horizontal is defined by the x-axis.

Using the Horizontal tool (Method 1)

- 1 Click the Horizontal tool on the Constraints tool palette.
- 2 Click one or more lines to constrain horizontally.



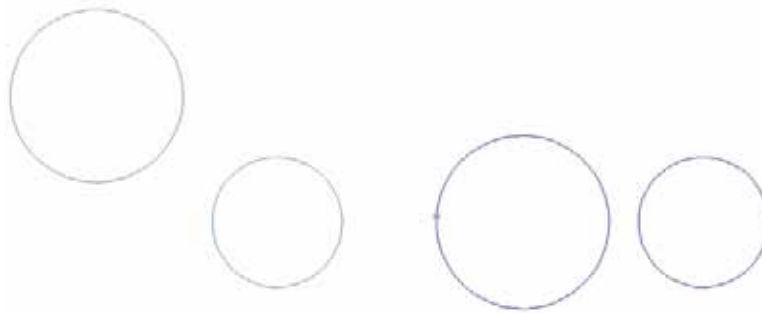
Horizontal Constraint

The tool tip line has an option to make selected points horizontal. Use this option to constrain selected points on an object instead of the entire object. The below example constrains the center point of a circle.

Using the Horizontal Constraint tool (Method 2)

- 1 Click on the Horizontal Constraints tool.
- 2 Hold down the Option key (CTRL for PC).
- 3 Click the first object.
- 4 Click point on object to constrain.

- 5 Click second object.
- 6 Click point on object to constrain.



Horizontal Point Constraint

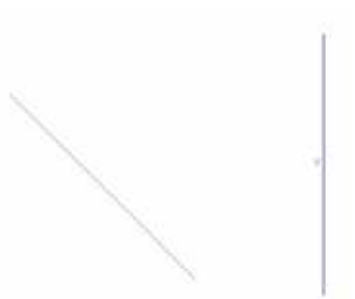


Vertical Constraint

This tool adds a vertical constraint to a line. You can select one or more line segments to apply the constraint. Vertical is defined by the y-axis.

Using the Vertical tool (Method 1)

- 1 Click the Vertical tool on the Constraints tool palette.
- 2 Click one or more lines to constrain vertically.



Vertical Constraint

The tool tip line has an option to make selected points vertical. Use this option to constrain selected points on an object instead of the entire object. The below example constrains the center point of a circle.

Using the Vertical tool (Method 2)

- 1 Click the Vertical tool.
- 2 Hold down the Option key (CTRL for PC).
- 3 Click the first object.

- 4 Click point on object to constrain.
- 5 Click the second object.
- 6 Click point on object to constrain.



Vertical Point Constraint

Coincident Constraint

This tool adds a coincident constraint between two or more objects.

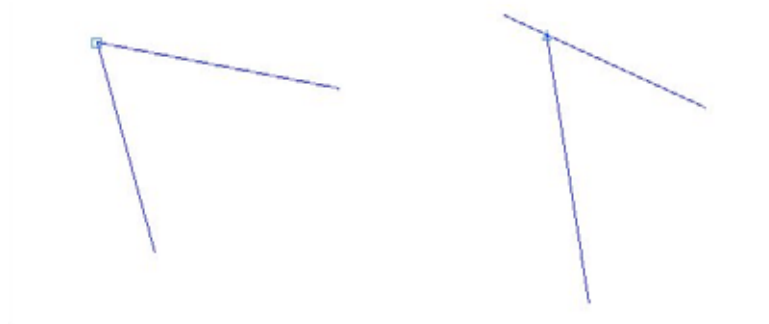
Using the Coincident tool (Method 1)

- 1 Click the Coincident tool on the Constraints tool palette.
- 2 Click the first curve.
- 3 Specify a location on the curve to constrain.
- 4 Click the second curve to constrain.
- 5 Specify a location on the curve to constrain.

Selecting a defining point will tie the constraint to this location. Selecting along a curve will preserve the parametric location (relative position along the curve).

Using the Coincident tool (Method 2)

- 1 Click the Coincident tool on the Constraints tool palette.
- 2 Click two or more objects to constrain. All shared endpoints with a tolerance are automatically constrained.



(a) Endpoint

(b) Percent Along

Coincident Constraints

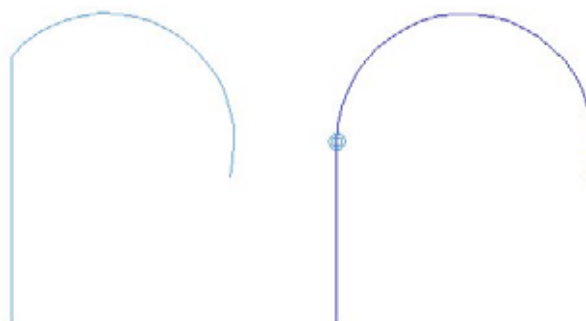


Tangent Constraint

This tool adds a tangent constraint to two or more curves.

Using the Tangent tool (Method 1)

- 1 Click the Tangent tool on the Constraints tool palette.
- 2 Click the first curve near the shared tangent location.
- 3 Click the second curve near the shared tangent location.



Tangent Constraint

When using a tangent constraint, don't forget to add a coincident constraint if you want the tangent constraint at a particular location.

Using the Tangent tool (Method 2)

- 1 Click the Tangent tool on the Constraints tool palette.
- 2 Click two or more objects to tangent constrain.

The latter method will automatically add tangent constraints to curves that are within a distance and angular tolerance.

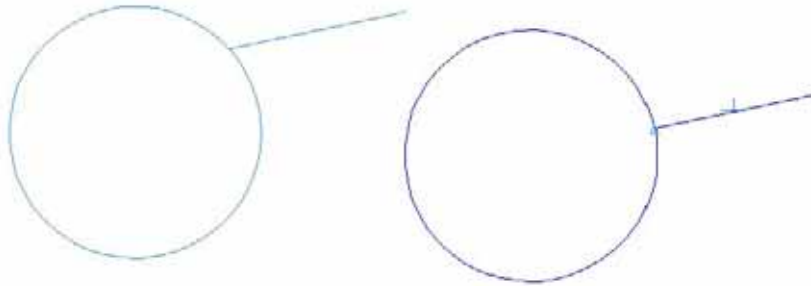


Perpendicular Constraint

This tool adds a normal constraint between a line and a curve.

Using the Perpendicular tool

- 1 Click the Perpendicular tool on the Constraints tool palette.
- 2 Click one or more objects to constrain.



Perpendicular Constraint



Parallel Constraint

This tool adds a parallel constraint between two lines.

Using the Parallel tool

- 1 Click the Parallel tool on the Constraints tool palette.
- 2 Click one or more objects to constrain.



Parallel Constraint



Offset Constraint

This tool adds an offset constraint between two objects. The offset distance can be set on the Data Entry window.

Using the Offset tool

- 1 Click the Offset tool on the Constraints tool palette.
- 2 Click one or more objects to constrain.
- 3 (optional) Enter a new value in the Offset text field, on the Data Entry window.



Offset Constraint



Equal Constraint

This tool constrains two lines to be the same length or two circles the same radius.

Using the Equal tool

- 1 Click the Equal tool on the Constraints tool palette.
- 2 Click one or more objects to constrain.



Equal Constraint



Fixed Constraint

This tool locks a curve so it will not move. The constraint solver will not modify a locked curve in any manner.

Using the Fixed tool

- 1 Click the Fixed tool on the Constraints tool palette.
- 2 Click one or more objects to constrain.



Fixed Constraint

Locked curves are displayed using the Locked Constraint Color defined in the File:Preferences:DCM dialog.

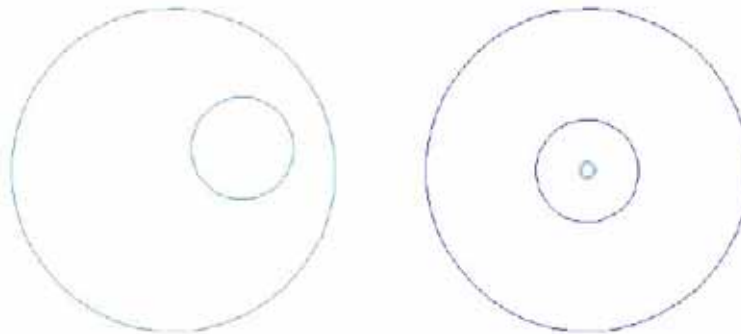


Concentric Constraint

This tool makes two circles share the same centerpoint.

Using the Concentric tool

- 1 Click the Concentric tool on the Constraints tool palette.
- 2 Select two or more circles you want to share one centerpoint.



Concentric Constraint

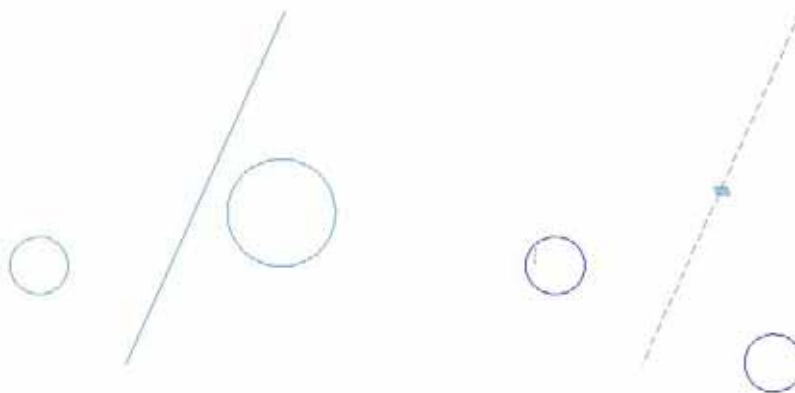


Symmetric Constraint

This tool makes two curves symmetrical about a line.

Using the Symmetric tool

- 1 Click the Symmetric tool on the Constraints tool palette.
- 2 Select the symmetry reference line.
- 3 Select the first curve.
- 4 Select the second curve to complete the symmetrical constraint.



Symmetry Constraint



Midpoint Constraint

This tool creates a midpoint constraint between a curve (point, line, circle) and the midpoint of a reference line.

Using the Midpoint tool

- 1 Click the Midpoint tool on the Constraints tool palette.
- 2 Click one or more objects to constrain.



Midpoint Constraint

Collinear Constraint

This tool makes two lines collinear or lie in a straight line.

Using the Collinear tool

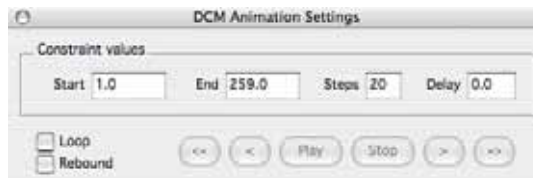
- 1 Click the Collinear tool on the Constraints tool palette.
- 2 Click one or more objects to constrain.



Collinear Constraint

Animating Dimensions

This tool animates a sketch by modifying a dimension value through a range of values. If you animate a sketch while in sketch mode, only the sketch is updated. If you animate a sketch outside of the sketch mode, dependent surfaces and solids update accordingly.



Constraints Animation Dialog

Using the Animate tool

- 1 Select a dimension.
- 2 Click the Animate tool on the Constraints tool palette. The DCM Animation Settings dialog appears.
- 3 Enter the start and ending values.
- 4 Enter the number of steps to use between the start and end values.
- 5 Click Play.

Under and Over Constrained Geometry

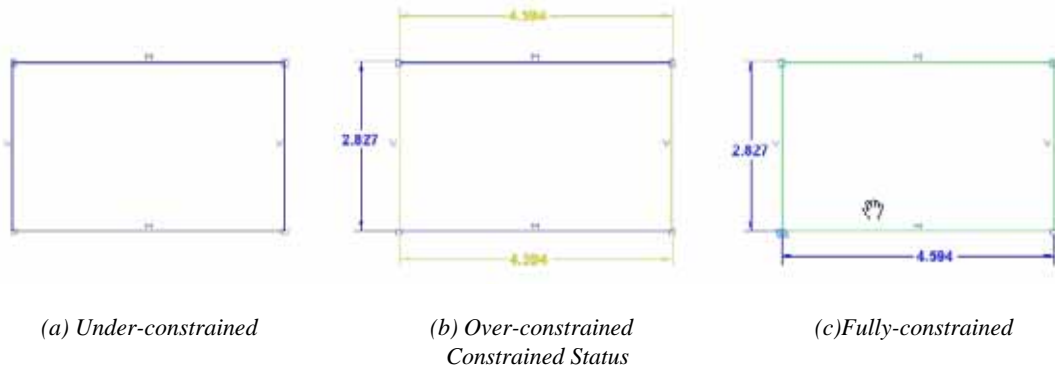
A major feature of the constraint management system is that it is designed to handle under and over constrained data in such a way that the user is helped to build a properly defined object interactively.

Under-constrained data is defined as data having insufficient dimensions and logical constraints to define the geometry uniquely. The default under-constrained object color is blue.

Over-constrained data is defined as data having too many or conflicting dimensions and legal constraints. The default over-constrained object color is dark yellow.

The fully constrained object color is green.

Under, over, and fully-defined object default colors can be changed in the File:Preferences:DCM category.



Dimension Driven Geometry

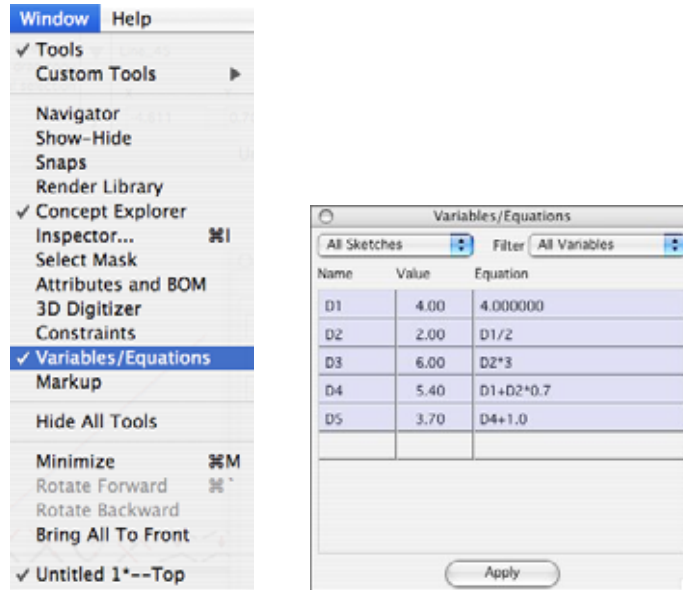
Dimensions created in the sketch mode are, by default, driving dimensions. This means that changing the dimension value will force curves associated with the dimension to update to the new value. A dimension that is driven by the curve is called a reference dimension. Outside the sketch mode, dimensions created are reference dimensions. To change a dimension from driving to driven, right-click on a dimension.

Sketch Axis

To turn on a sketch axis, open the Concept Explorer and Ctrl-click on a sketch and toggle between turning the sketch axis on and off. Moving the sketch axis will move the entire sketch, and rotating the axis will rotate the sketch.

Variables/Equations

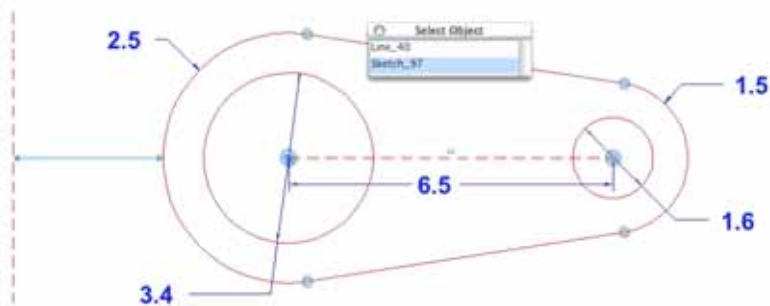
The Variables and Equations dialog allows you to create variables which may be constant or mathematical expressions. To access the dialog, to go *Window:Variables/Equations*.



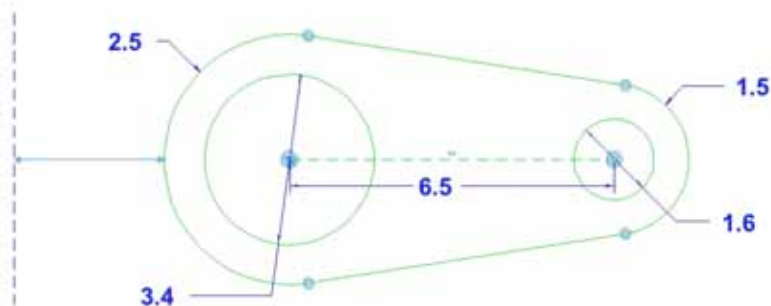
As you create and assign dimension constraints to geometry, the equation dialog automatically updates with new entries. The Name field contains a unique identifier from the object. The Value field represents its current evaluation expression. The Equation field contains a string expression defining a mathematical relationship. The following types of variables within equations are supported:

- A variable associated with a dimension.
- A simple variable not associated with a dimension.
- Linear equations of the form $a1*v1+a2*v2+a3*v3+...+c = 0$.
- Non-linear equations of the form $f(v1,v2,v3) = 0$.

Sketch to 3D



Use line patterns for objects not to be extruded; sweep.

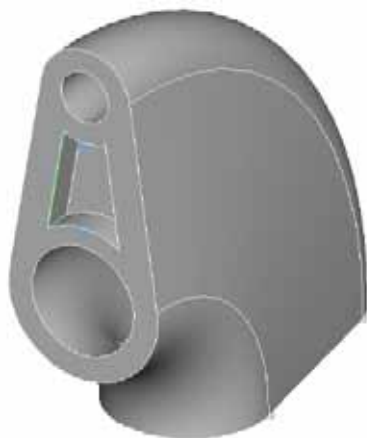


Sketching on Faces

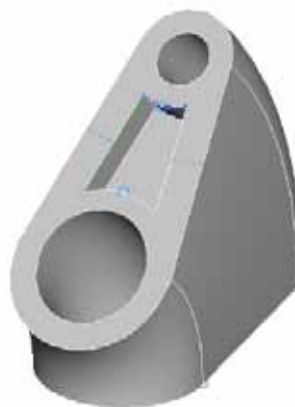
Punch! Shark FX supports the creation of sketches on the face of a solid. In this case, the sketch is dependent on the geometry of the face and updates as the face changes. This includes the ability for the sketch to change its orientation to match any changes of the face orientation.

Establishing a sketch on the face of a solid is accomplished by first setting the work plane to the face using the WorkPlane:Set Objects command located in the menu bar. The orientation and origin for the workplane define the sketch axis when constraints on the face are applied. The sketch on face remains active until the workplane is changed by the user.

Face edges can be used as references for sketches. Since these edges are defined by the solid and cannot be changed, they are marked as “Locked”. Horizontal and vertical references are relative to the sketch x and y axis, respectively.



Lathe at 90-degrees



Lathe Modified to 45-degrees

Sketch On Face

Use the “Distance” option for protrusions and cutouts to maintain a distance that is relative to face normal.

Adding and Removing Items from a Sketch

To remove a curve or dimension from a sketch, simply select the item and choose Edit:Cut. Curves or dimensions are automatically added to a sketch when they are used as part of a constraint involving the sketch.

Part 4

Surface Modeling

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Intro to Surface Modeling

A surface model of an object 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 that spans boundary curves with smooth equations. Consequently, surface modeling provides a means for the designer to model extremely complex shapes such as cars, ships, and aircraft.

Surfaces in Punch! Shark FX are based upon 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 Punch! Shark FX are suitable for precise geometry analysis and can be passed to computer aided design or manufacturing applications that support precise NURBS formats such as IGES, STEP or SAT.

Surface Menus

The surface modeling tools are accessible from the main tool palette. The surface tool palettes provide tools for creating surfaces from curves, surface features, surface utilities, local surface operations, and advance surface tools.

Surface From Curves

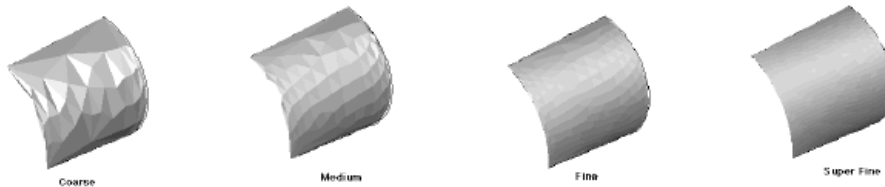
Examples of surfaces that are driven from curves include net, ruled, skin, cover, revolved, extrude, 1-rail sweep, 2-rail sweep, and tube. Curves are automatically added or removed to preserve the history for covers and profiles.

Surface Utilities

Once a surface is defined, you can generate plane/surface, surface/surface and curve/surface intersections. In addition, Punch! Shark FX supports curve projections onto surfaces and exploding surfaces into meshes. The utility palette also includes a Trim Surface tool for removing unwanted portions of a surface.

Surface Select and Display

Surfaces are selected by pressing the mouse anywhere within the surface boundary. However, you will notice faster response by selecting a surface edge (The selection algorithm first checks for an edge and if no edge is selected, fires a ray into the surface area). A surface is displayed by its boundary edges along with any possible interior holes. You can increase the surface edge resolution with the *Edit:Resolution* command.



Surface Resolutions

You can change the default resolution that surfaces are created with by adjusting the resolution setting in the Files:Preferences:Display (or Shark FX:Preferences:Display) dialog box.

Surface Associativity

Surfaces are associative to the original curves and parameters used to create the surface. Changing the curve will automatically change the dependent surface. For example, if the designer creates a skin surface from three arcs and changes the radius of one of the arcs, the skin surface will update for the change in the arc radius. Use the Edit:Remove Links command to remove all associativity with the surface if you do not want curve changes tied to the surface. Another important issue to remember with the curve/surface associativity is that if the curve is deleted, the surface will lose all associativity.

Surface From Curves

Punch! Shark FX supports nine surface types that are created by referencing curves. From left to right across the surface tool palette they are:



- Infinite Plane
- Net
- Ruled and Skin
- Cover
- Revolution
- Extrude
- Swept
- Bi-rail
- Tube

Infinite Plane

The Infinite Plane Surface is the simplest surface supported by Punch! Shark FX. An infinite plane is defined by a location and normal. The plane surface is useful for generating cross-sections through meshes, surfaces or solids.

The infinite plane has five creation methods:

X-Station	Create a plane with normal $X=1, Y=0, Z=0$.
Y-Station	Create a plane with normal $X=0, Y=1, Z=0$.
Z-Station	Create a plane with normal $X=0, Y=0, Z=1$.
Normal & Location	Create an arbitrary plane by specifying the location and normal.
Three Points On Plane	Specify a plane by three points. Normal is calculated from the three points.

Using the Infinite Plane Surface tool

- 1 Select the infinite plane sub type (x-station, y-station, etc.).
- 2 Click at the location of the plane.



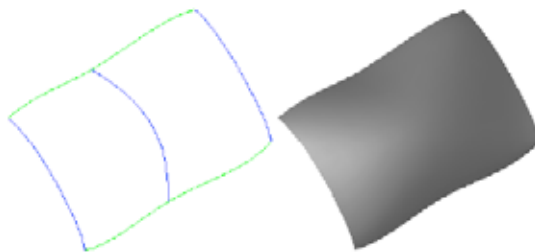
Infinite Plane

Net Surface

The Net Surface is created from a collection of $m \times n$ curves. The curves may be combinations of lines, splines, arcs, ellipses, conics or circles. Each curve must be a single curvature continuous curve. No groups (composited curves) are allowed. A point is allowed at either the start or end of the surface. The supplied curves do not need to be planar and do not need to touch.

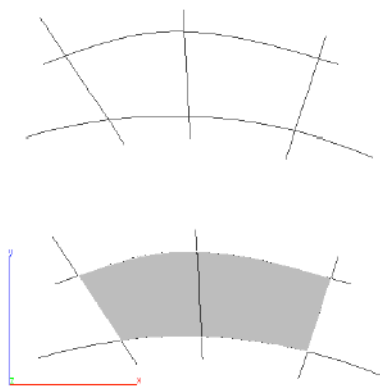
Using the Net Surface tool

- 1 Hold the Shift key and pick the curves defining the m direction. Release the Shift key when done selecting.
- 2 Hold the Shift key and pick the curves defining the n direction. Release the Shift key when done, to create the net surface.



Net Surface

The net surface does not require that the curves intersect. This means that nets such as the one shown below are possible.



Overlapping Curves Net Surface

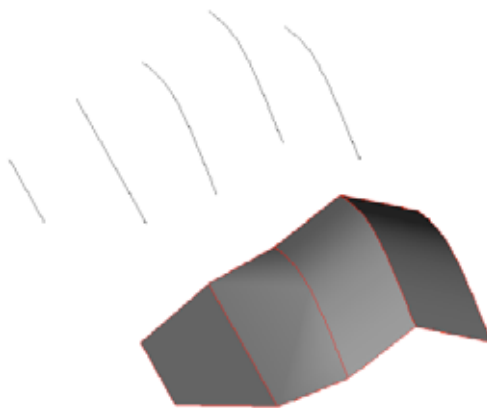


Ruled Surface

A Ruled Surface has straight spans between each section. Sections can be of any curve type. This tool is available from the sub-tool palette.

Using the Ruled Surface tool

- 1 Hold the Shift key and pick the curves.
- 2 Release the Shift key when done selecting.



Ruled Surface



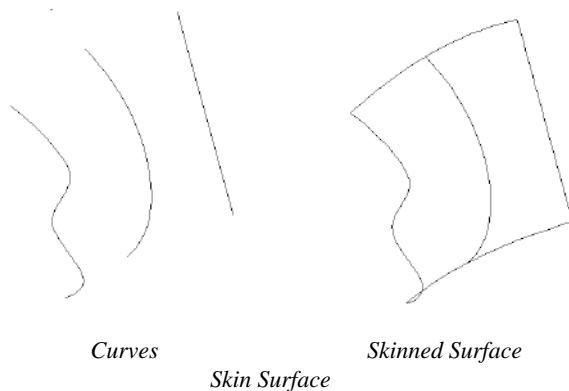
Skin Surface

A Skin Surface fits a network of NURBS patches to a collection of curve cross sections. The curves can either be all opened or all closed. Curves suitable for skinning include lines, arcs, circles, ellipses, conics, and splines. Each curve must have continuous tangents (no polylines) and be oriented in the same

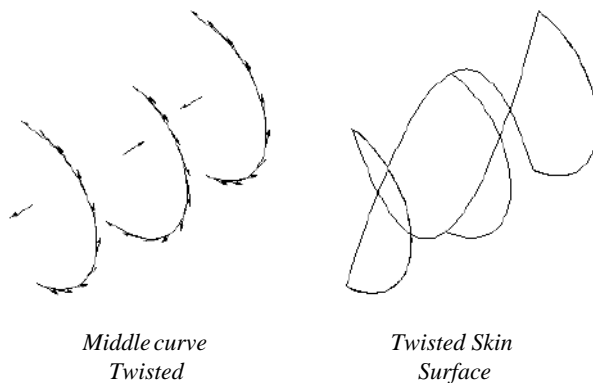
direction. If the curves have different directions, the resulting surface will twist from section to section. Also, sections cannot share an endpoint (each section start and end must be a unique position in coordinate space). This tool is available from the sub-tool palette.

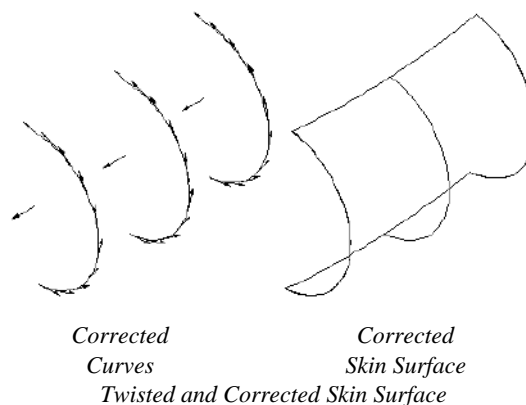
Using the Skin Surface tool

- 1 Hold the Shift key and pick the curves (line, spline, arc, circle, ellipse).
- 2 Release the Shift key when done selecting.



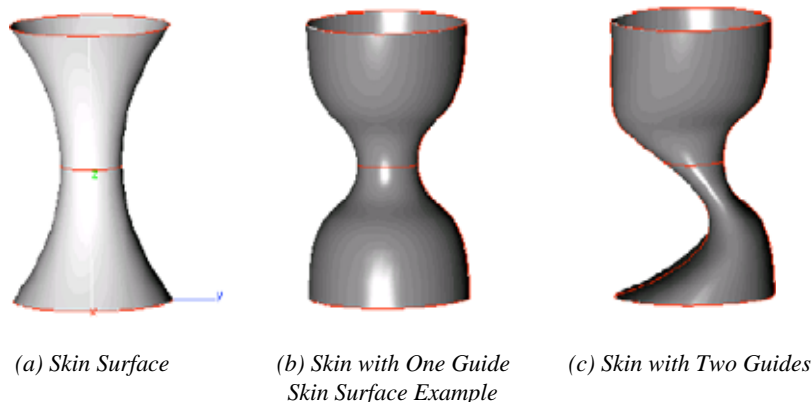
By default, the skin tools will Auto Align curves such that their direction and orientation are irrelevant. If you come across a condition where the auto align is turned off or does not apply you will need to manually align each curve section. To do so, use the Verify:Direction command located in the Menu Bar to examine curve direction. You can use the Edit:Change Direction command to reverse the orientation of a curve.





Guide Skin Surface

A Guide Skin Surface is identical to a skin surface with the addition of an unlimited number of edge guide curves. The guide curves control the surface's edge shape between the skin sections. Either one or both surface edges can be guided. Each guide curve must go through a point on the skin profile curves. Curves suitable for guides include lines, arcs, circles, ellipses, conics, and splines. This tool is available from the sub-tool palette.



(a) Skin Surface

(b) Skin with One Guide
Skin Surface Example

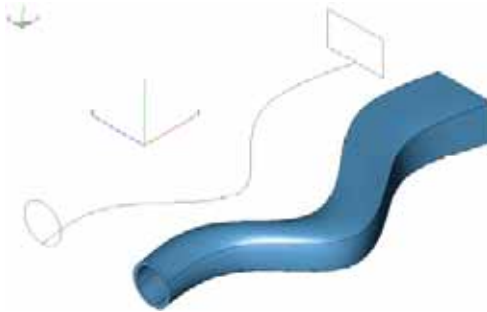
(c) Skin with Two Guides

In the figure above are three skin surfaces created from the same three arcs. Figure (a) is the skin surface without guide curves. Figure (b) shows the same three arcs with a single spline guide curve. The spline goes through the starting point of each circle and the slopes are modified. Figure (c) shows a skin with two guide curves. One is at the beginning of each curve, another is at the midpoint. The slopes on each spline have been modified to produce an unusual surface. The figure illustrates how the intermediate shape of a skin is drastically controlled using guide curves.

Using the Guide Skin Surface tool

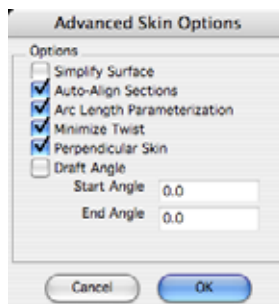
- 1 Hold the Shift key and pick 2 or more skin profile curves (line, spline, arc, circle, ellipse).

- 2 Release the Shift key when done selecting.
- 3 Select the edge guide curve(s). Hold the Shift key to select multiple curves.



Skin Options

Both the Skin Surface and Guide Skin Surface tools provide many options that can be used at the time of creating a skin surface. After a tool is active, click the Options button on the Prompt window.



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 iso-parametric parameterization of the skinning surfaces. In iso-parametric parameterization, the surface parameter in the v direction follows the cross

section curves. In Arc Length Parameterization, the surface parameter follows lines of constant length. The default is arc length parameterization.

Minimize Twist

The Twist option may be used to minimize the twist of the surface produced. Twist minimization aligns closed curves such that the start of the second curve is aligned to the start of the first curve. Even if a body's shape is unaffected by twisting, a surface with a twist could produce unexpected results when faceting and rendering. By default, twist minimization is on. Twist minimization is also an involved calculation that some users may not want to have carried out.

Perpendicular Skin

The take-off vector is a tangent vector going out of the starting edge or surface and into the skinned surface. The Perpendicular option is used to specify the direction of the take-off vector, perpendicular to the edge. The default is in the loft direction, because a perpendicular take-off vector can cause self-intersections to the surface.



Default Skinning

Perpendicular Skinning

Perpendicular Skin Option

Draft Angle

Skinning with draft angles provides the ability to control the take-off vectors of the two outer skinning profiles. The “draft” angle is defined as an angle off the plane of the wire at every point along the skinning profile. In addition to the user supplying the angle itself, one may also supply a magnitude for the take-off vector. The draft angle and magnitude is constant for the entire profile. However, one may apply different draft angles to the two outer profiles. In addition, skinning with draft angles supports open and closed profiles and skinning to a point. When skinning to a point, the algorithm constructs its own normal vector. The outer profiles must be planar when not degenerate. Use the Edit Object dialog box to access the skin magnitude.



*Standard Skin
Between Two Circles*

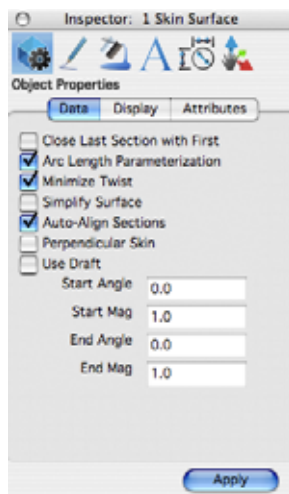


*Various combinations of
Draft Angles and Draft Magnitudes
Skinning Two Circles*

In the figure above, note the different number of shapes that can be obtained from the same two circles for a skinned surface. To access the skin parameters, select the surface and edit the parameters in the Inspector.

Editing Skinned Surfaces

The Inspector dialog contains the various options used for creating the skin surface. You can change any of these options and have the surface regenerate to the new option.



Skin Properties



Cover Surface

The Cover Surface is an extremely versatile and powerful utility for creating a surface from curves connected end to end. Curves suitable for cover surfaces include arcs, circles, lines, splines, ellipses, and conics. Curves are not required to be in a plane. Each individual curve must have continuous curvature (no polylines or groups). If the curves all lie in the same plane, a simple planar surface with boundaries is created. If the curves lie out of plane, a Gregory surface is fit to the boundary curves.

The Cover Surface tool also has the ability to cover planar bodies that include interior cutouts. The cover automatically updates according to fillets and chamfers, and if a hole is removed or resized.

Covers must be created from curves connected end to end. If one or more of the curves are not connected, Punch! Shark FX displays the message “can only cover closed wires”. You can force wire ends to precisely close with one of two commands. The first command is the Move Selected Points tool, located in the main tool palette under transforms. The second command is connect curve ends located in the tool palette under Curve Utilities.



Invalid Curves for Cover

Using the Cover Surface tool

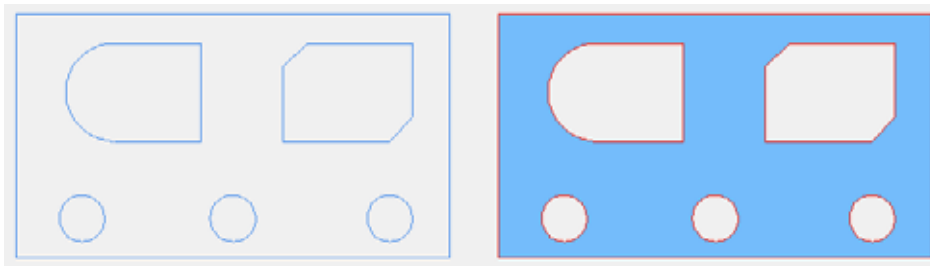
- 1 Hold the Shift key and pick the curves (line, spline, arc, circle, ellipse).
- 2 Release the Shift key when done selecting.



Cover Surfaces

Covering a surface with holes

- Drag a selection window around the entire surface you want to cover.



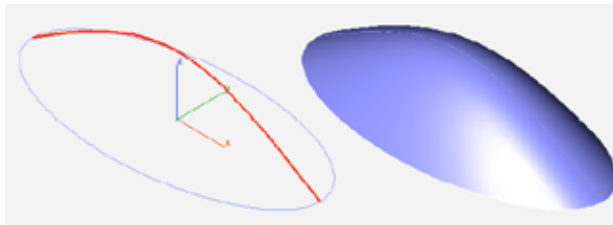
Surface with holes

Covered surface with holes



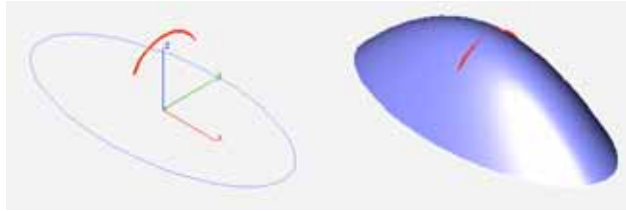
Cover Surface with Guides

The Cover with Guide Curves tool provides an option to not only cover the boundary curves but also include interior guide curves (or points).



Cover with Guides

Guide curves can be of any type and orientation. Guide curves can also include points. Guide curves can cross over each other and do not need to touch the boundary curves as shown in the image below.



Guide Curve Example

Using the Cover with Guides tool

- 1 Select the Cover with Guides tool from the Cover Surface tool palette.
- 2 Pick the curves defining the outer boundary of the surface.
- 3 Pick the objects defining the inner guide curves.

Surface of Revolution

The Surface of Revolution is used to model axis symmetric objects. The designer can choose from one of two methods:

- Revolve About 2 Points
- Revolve About Curve

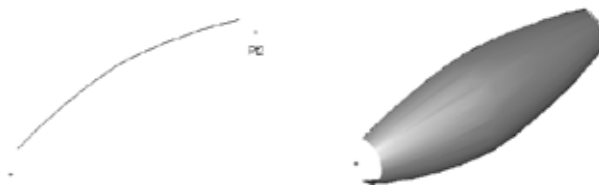


Two Point Revolved Surface

The surface of Revolution About 2 Points option will revolve the curve a user specified angle about an axis created from two user specified points.

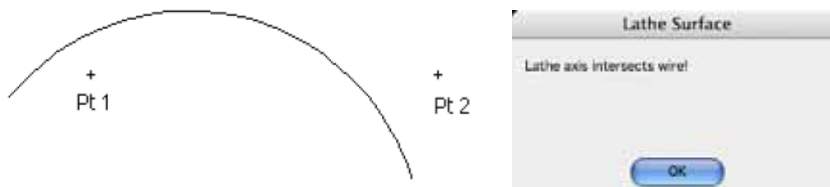
Using the Two Point Revolved Surface tool

- 1 Hold the Shift key and pick the curves to revolve (line, spline, arc, circle, ellipse).
- 2 Release the Shift key when done selecting.
- 3 Specify the angle of revolution in the Data Entry Window.
- 4 Click two points defining the axis of revolution.



Surface of Revolution

The direction of the angle is defined by the order of the points defined for the axis. Use the right hand rule to determine the direction that the surface will revolve. If the axis intersects the lathe curve, Punch! Shark FX displays the error message below. Your only course of action is to pick a new non-intersecting axis.



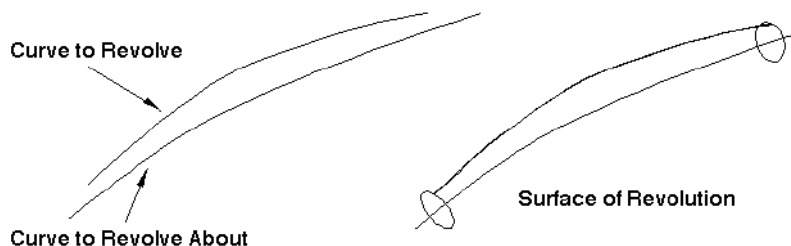
Intersecting Axis Error

Curve Revolved Surface

The second method revolves a curve about another non-linear axis such as a curve. The curved axis must be as long or longer than the curve you are revolving.

Using the Curve Revolved Surface tool

- 1 Pick one curve to revolve (line, spline, arc, circle, ellipse).
- 2 Type in the angle of revolution in the Data Entry Window.
- 3 Pick one curve to revolve about (line, spline, arc, circle, ellipse).



Revolve About Curve

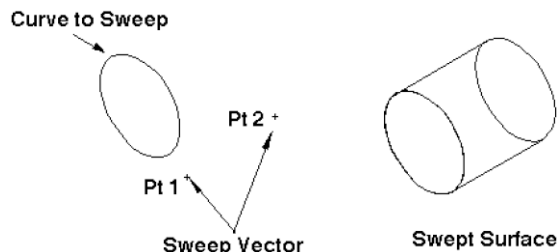


Extrude Surface

The sweep 2 points surface sweeps a curve, collection of curves or group along two user supplied points that define the direction and length of the sweep.

Using the Extrude Surface tool

- 1 Pick one or more curves to sweep (line, spline, arc, circle, ellipse, groups).
- 2 Click two points that define the direction and length of the sweep from the curves.



Sweep Surface 2 Points

The resulting surface is associative to the original sweep curve. Changing the curve will automatically update the surface. You can also edit the sweep length and direction with the Object Info command; this opens the Inspector.



One Rail Sweep Surface

The One Rail Sweep Surface sweeps a curve, collection of curves or group along a user supplied rail curve. The resulting surface is associative to both the path and sweep curve. Modifying either will automatically update the associative sweep surface. The Sweep One Rail command has two pull downs for options in the prompt line. The first pull down controls how the profile orientation sweeps as it is moves along the rail. It contains the following three options.

- Sweep In Place (profile is not translated or aligned to path).
- Sweep Perp (profile is translated and aligned perpendicular to path).
- Sweep Rigid (profile orientation is maintained irregardless of path tangent).

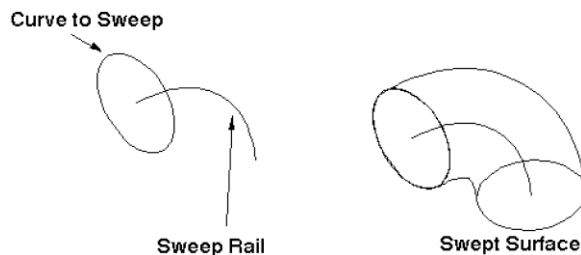
The second pull down controls how the profile is terminated. It contains the following three options.

- Curve extents (profile sweeps across entire curve extents).
- To Body (profile is terminated at a surface or solid).
- Between Points (user specifies two points on the path).

Sweep Perpendicular rotates the sweep profile to be perpendicular to the take off path. Sweep in Place leaves the sweep as is; no aligning of the profile occurs with this option. Sweep To Entity sweeps the profile along the given path and terminates to a reference surface or solid.

Using the One Rail Sweep Surface tool

- 1 Pick one or more curves to sweep (line, spline, arc, circle, ellipse, groups).
- 2 Pick one curve for the rail path.



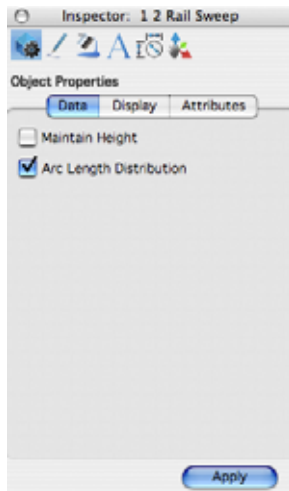
Sweep Surface One Rail



Two Rail Sweep Surface

The Two Rail Sweep Surface tool creates a swept surface by sweeping an open or closed curve between two rail curves. The rail curves define the orientation and scale of the surface as it moves between the two curves.

The Inspector offers two options after the surface has been created. Select the surface, and the options are available on the Data tab.



Using the Two Rail Sweep Surface tool

- 1 Pick one or more curves to sweep (line, spline, circle, ellipse, groups).
- 2 Pick the first rail path.
- 3 Pick the second rail path.

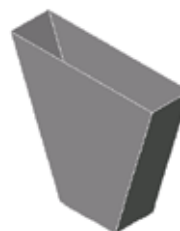
- 4 Hold the Option key at step one if you want to maintain the profile height as it is swept.



Original Curves



*Sweep Normal
Sweep Surface Two Rails*



Sweep Maintain Height

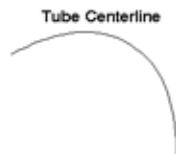


Tube Surface

Creates a tube or pipe surface along a curve with a given radius.

Using the Tube Surface tool

- 1 Type in a tube radius in the Data Entry Window.
- 2 Pick the tube path which is the tube centerline.



Tube Surface

Surface Features

The Surface Feature tool palette contains surfaces that are created by referencing other surfaces in addition to other optional parameters. Example surface features include:



- Offset
- Loft
- Draft
- Fillet
- Tangent Cover

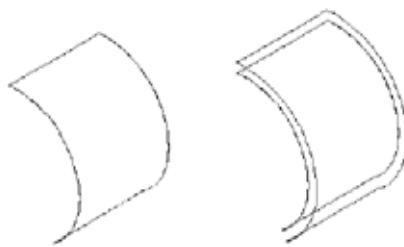


Offset Surface

Offset surfaces maintain a constant distance normal to the parent surface. An offset surface is created by first selecting the parent surface and specifying the offset surface. The direction of the offset is in the direction of the surface normal. You can specify negative offsets. Typically you cannot offset a surface a distance greater than the smallest radius of curvature.

Using the Offset Surface tool

- 1 Type in an offset value in the Data Entry Window.
- 2 Pick the surface to offset from.
- 3 Type a negative into the Data Entry Fields if the offset is desired on the opposite side.



Offset Surface

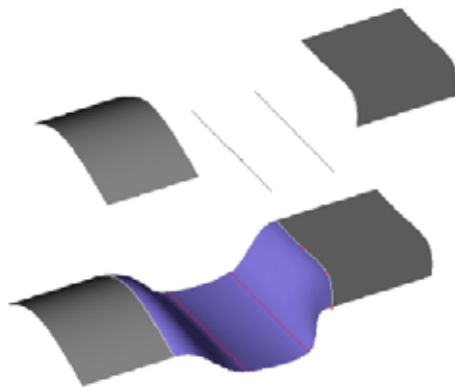


Loft Surface

The Loft Surface creates a tangent continuous surface between one or two surfaces. The Loft Surface also accepts a bulge factor which influences the strength of the slope coming off of the referenced surfaces.

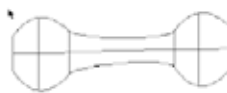
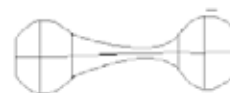
Using the Loft Surface tool

- 1 Hold the Shift key and pick the starting surface edge to reference.
- 2 Hold the Shift key and pick intermediate curves
- 3 Pick the ending surface edge for reference.
- 4 Type in bulge values to adjust tangent magnitude.



Lofted Surface

Once created, the blend surface can be modified with the Inspector. The start bulge factor controls the magnitude of the slope from the starting edge, whereas the end controls the slope influence at the end curve. The reverse curve options are provided to fix twisting of the blend surface resulting from different edge directions.

*Bulge = 0.25**Bulge = 1.0**Start Bulge=3,
End Bulge=0.5**Bulge Factor Examples***Loft Surface Guides**

The Loft Surface with Guides option allows you to specify guide curves. The guide curves can help control the shape of the surface away from the tangent.

**Draft Surface**

The Draft Surface tool creates a surface at an angle relative to a referenced surface. For example, zero degrees will create a surface tangent to the referenced surface.

Using the Draft Surface tool

- 1 Use the Data Entry Fields to specify the draft angle and surface length.
- 2 Pick the starting surface edge.

*Draft Surface*

The draft surface also has an option that controls how the surface slope is calculated with respect to the referenced surface. In the example below, the first draft is aligned with the edge and the second is aligned with the surface.

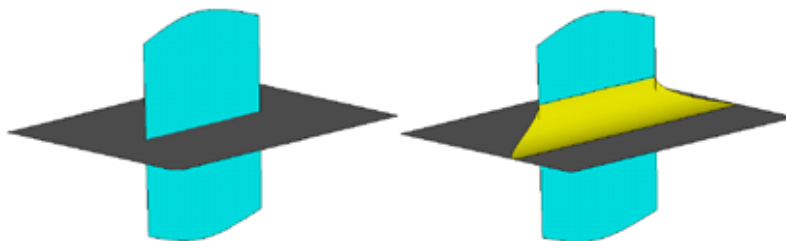
*Draft Surface Align Option*

Fillet Surface

The Fillet Surface tool will create a smooth radial surface between two surfaces. Use the option to trim the surfaces back to the fillet. The default fillet style is Tangent Continuous, there's also a Curvature Continuous option for curvature continuous blends between two surfaces.

Using the Fillet Surface tool

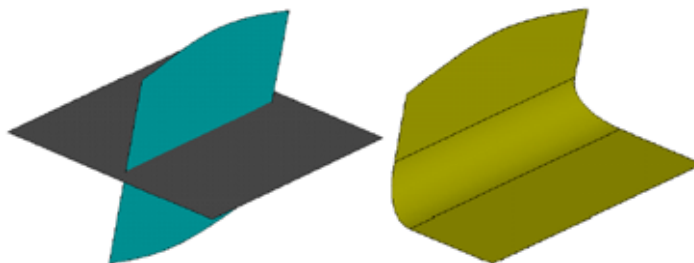
- 1 Use the Data Entry Fields to specify the fillet radius.
- 2 Pick the first surface.
- 3 Pick the second surface.



Fillet Surface

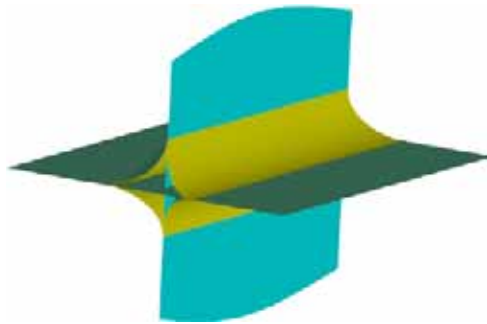
Using the Fillet Surface tool (Trim Option)

- 1 Use the Data Entry Fields to specify the fillet radius.
- 2 Press the option key.
- 3 Pick the first surface.
- 4 Pick the second surface.



Fillet Surface-Trim Option

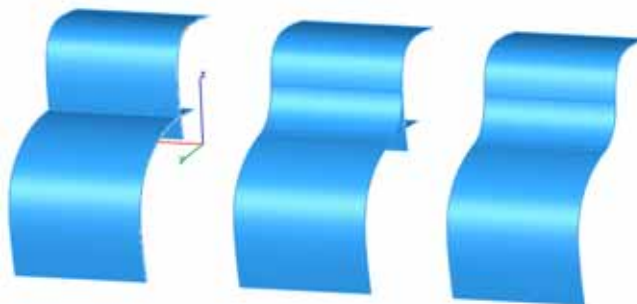
The Fillet tool automatically determines the quadrant for the fillet, based on where the user selects.



Fillet Surface-Auto Quadrant Selection

Using the Fillet Surface tool (Curvature Continuous)

- 1 Choose Curvature Continuous from the drop-down on the Data Entry Window.
- 2 Specify a radius in the Data Entry Window.
- 3 Select two surfaces to blend.



Original Surfaces

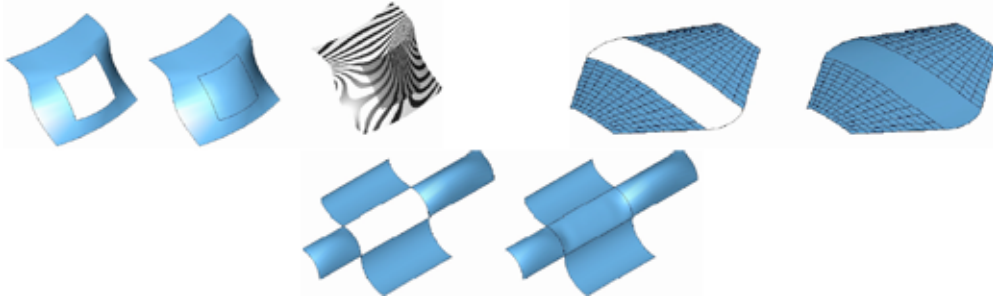
Fillet No Trim

Fillet Trim



Tangent Cover Surface

The Tangent Cover Surface creates tangent boundary surfaces. The resulting surface is tangent to selected surfaces. This tool will accept any combination of four curves and surfaces. The tool also recognizes selected surface edges without exploding first to edges. The resulting surface maintains its associativity to the parent curves and surfaces.



Example Tangent Cover Surfaces

Using the Tangent Cover Surface tool

- 1 Pick surface edges or curves that define a closed loop.
- 2 Use the Data Entry Field to influence the effect of the tangent vector.



Tangent Cover Guide Surface

The Tangent Cover Guide Surface tool is just like the Tangent Cover Surface tool, but with a Guides option that allows you to specify guide curves. The guide curves can help control the shape of the surface away from the tangent.

Surface Utilities



There are two palettes of surface utilities. These two palettes contain tools useful for manipulating surfaces. These tools, from left to right, are:

Surface Utilities

- Cut Section
- Project Curve Surface
- Surface/Surface Intersect
- Curve/Surface Intersect
- Silhouette Curve
- Explode Curve
- Project Curve To Plane

Advanced Surface Utilities

- Surface Booleans
 - Add Surface
 - Subtract Surface
 - Intersect Surface
- Join Surface
- Split Surface



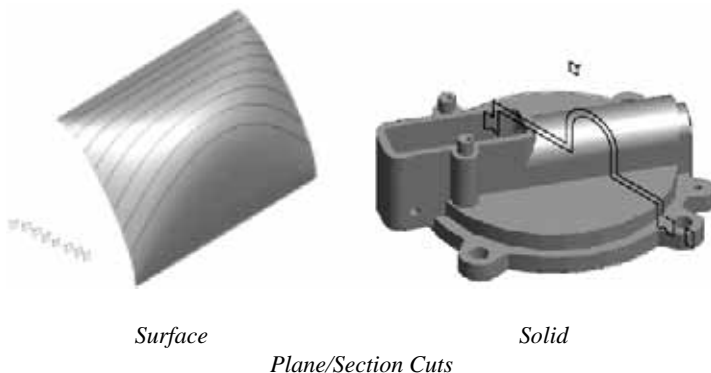
Cut Section

Calculates the intersection of an infinite plane and a curve, surface or solid. The resulting curve is automatically associated with the plane and surface such that if either is modified, the cut curve will regenerate.

Using the Cut Section tool

- 1 Select the planes to intersect with objects.

- 2 Press option to use auto layers, if desired.
- 3 Select the objects to intersect the planes with.

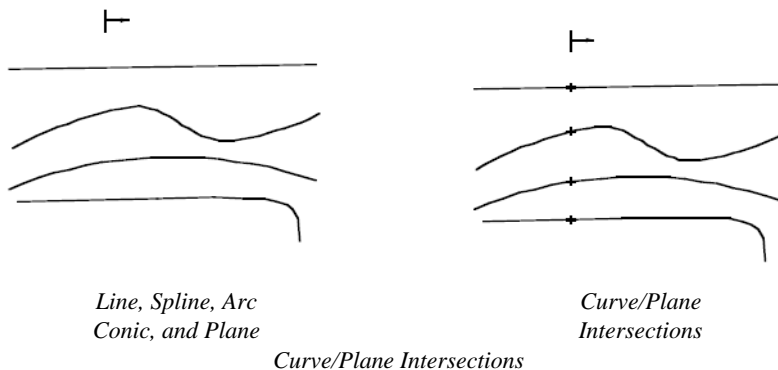


Auto Layers Option

The Auto Layers Options allows you to specify what layers each individual section cut will go in. Used in conjunction with the 9 and 0 quick keys (increment up/down layer) it is an effective tool to examine two dimensional cuts of a 3D model.

Cut Curve

Creates a point at the intersection of a plane and curve. Select the plane and the curves. At the intersection of the infinite plane and curve, a point entity is created. The point entity is associated to the plane and curve. Because the point is associative, modifying either the plane or the curve will automatically update the location of the point.





Project Curve Surface

The Curve Projection tool projects a curve along a direction vector onto a surface or solid. The direction is supplied as two points specified by the user. The resulting curves are associative to the surface. Modifying the original curve or surface will automatically regenerate the curve projection.

Using the Project Curve Surface tool

- 1 Select the curve to project.
- 2 Select the surface or solid to project onto.



Curve/Surface Projections

This tool has an option to imprint the results directly onto the surface or face. Imprinting may introduce additional faces due to splitting.

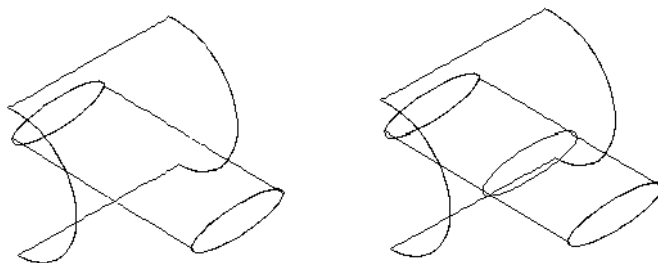


Surface/Surface Intersect

Calculates the intersection of two surfaces or solids. The designer selects the two surfaces and one or more curves are created. The resulting curves are associative to both surfaces. A change in either surface will regenerate the intersection curve. To remove the associative link, use the Edit:RemoveLink command located in the Menu Bar.

Using the Surface/Surface Intersect tool

- 1 Select the first surface or solid.
- 2 Select the second surface or solid.

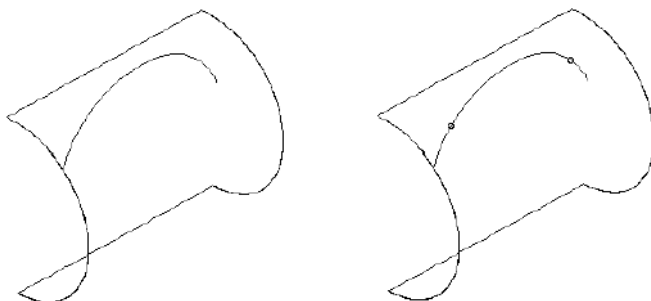
*Surface/Surface Intersection*

Curve/Surface Intersect

Calculates the intersection of a curve and surface or solid. The result of a curve/surface intersection is a collection of point objects. The resulting points are, by default, associative between the curve and surface.

Using the Curve/Surface Intersection tool

- 1 Select the curve.
- 2 Select the surface or solid.

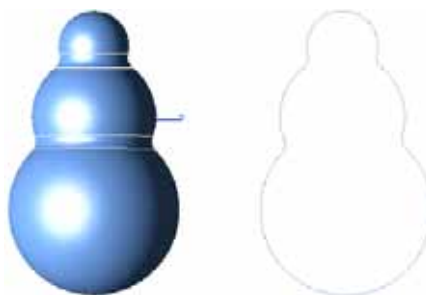
*Curve/Surface Intersection*

Silhouette Curve

This tool provides a means to create precise silhouette curves from surfaces or solids. This tool does not generate a silhouette for planar surfaces or faces.

Using the Silhouette Curve tool

- 1 Select the first surface.
- 2 Select silhouette generation based upon view or two points.
- 3 (Two Point Option) Click two points defining the view direction.



Silhouette Curves



Explode Curve

The Explode Curve tool takes an edge of a surface or solid and converts it into an independent but associative curve.

Using the Explode Curve tool

- 1 Click the Explode Curve tool.
- 2 Select the surface or face edge to explode.
- 3 Hold shift to select more than one curve.



Project Curve To Plane

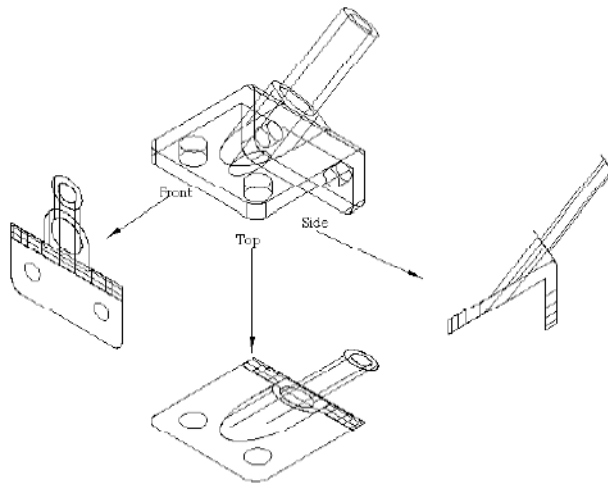
The Project Curve To Plane tool projects 3D curves into a 2D plane. Default planes include XY, XZ, YZ, Work Plane, and Plane object. Use the construction plane to define user defined projections and depths. Arcs, circles, and ellipses maintain precise shape when projected normal to their definition, otherwise are spline fit to the default curve tolerance (0.001").

Using the Project Curve To Plane tool

- 1 Click the Project Curve To Plane tool.
- 2 Select the plane to project the curves into:



- 3 Select the curves to project.



Project Curve


Add Surface

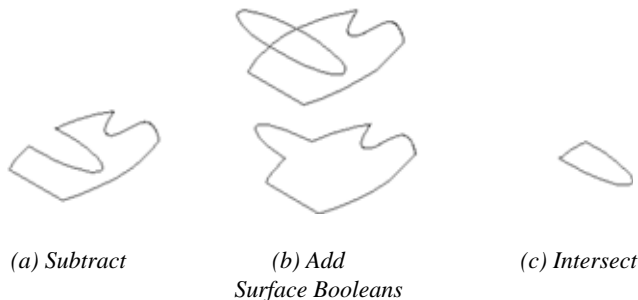
Subtract Surface

Intersect Surface

Punch! Shark FX supports 2D Boolean operations on planar surfaces. Planar surfaces are created with the cover surface command. With the Boolean tool, you can add, subtract, and intersect surfaces. The results of Boolean operations are useful for 2D area property analysis or as profiles for solid extrusions, sweeps, and lathe operations.

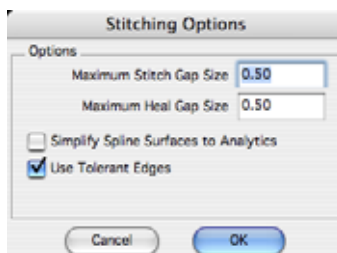
Using the Add/Subtract/Intersect Surface tool

- 1 Select the first surface.
- 2 Select the second surface.



Join Surface

The Join Surface provides a means to join two or more surfaces. The Connect Surface command will attempt to heal any gaps along shared edges and combine the final surfaces into one surface entity. If the surfaces form a closed volume, an alert dialog box is displayed.



Connect Surface Options

Maximum Stitch Gap Size The maximum distance between two shared edges.

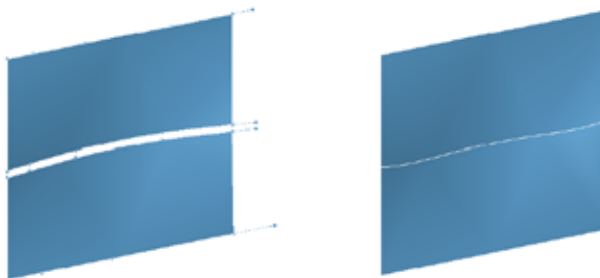
Maximum Heal Gap Size The maximum distance the software will attempt to heal two shared edges.

Simplify Spline to Analytics Examines the surfaces for possible conversion from NURBS to analytics.

Use Tolerant Edges Instead of healing the two surfaces, just assigns a tolerant edge.

Using the Join Surface tool

- 1 Press the Option key to specify advanced connect settings.
- 2 Select two or more surfaces.



Connect with Heal Option

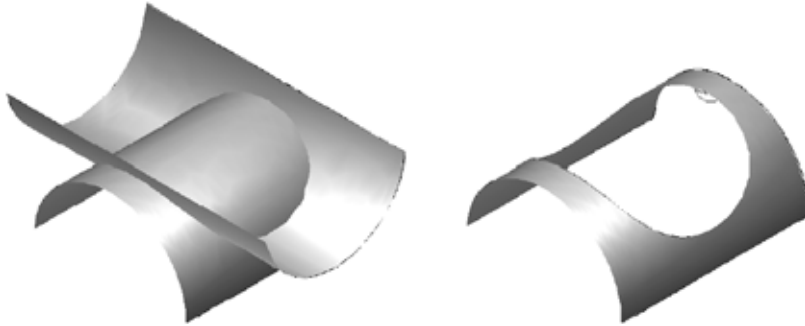
Split Surface

The Split Surface tool allows the designer to cut away portions of a surface. You can trim a surface to another surface or a solid. To create a split surface, select the surface you desire to trim and then the

split surface/solid. Punch! Shark FX will split the surface at the intersection of the two surfaces. Use the Delete tool to remove portions that are not desired.

Using the Split Surface tool

- 1 Select the first surface.
- 2 Select the body to trim the surface.



Split Surface

The split tool will split to curves that lie on the surface, other surfaces, and solids.

Local Surface Utilities

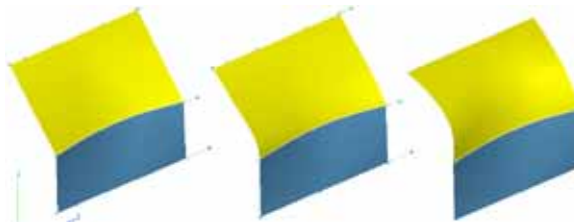


Match Surface

The Match Surface tool modifies the surface continuity at a given edge to a referenced edge. The surface that is to be modified must not be a trimmed surface.

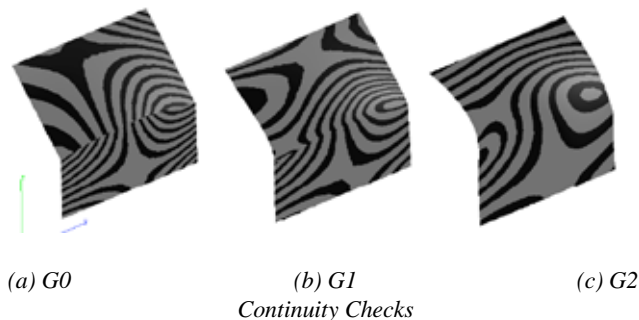
Using the Match Surface tool

- 1 Select the surface edge that will be modified in the match operation.
- 2 Select the surface edge that will be matched to.



Match Face Along Edge

The Verify:Surface Analysis:Zebra plot is very useful in combination with the Match Surface tool. The Zebra Plot helps visualize continuity between two surfaces. G0 continuity will show positional discontinuity in the strips. G1 shows positional continuity but strips do not transition smoothly. G2 show positional and smooth transitions between strips.



Rebuild Surface

The Rebuild Surface tool reconstructs an approximating surface to the referenced surface. 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 original condition.

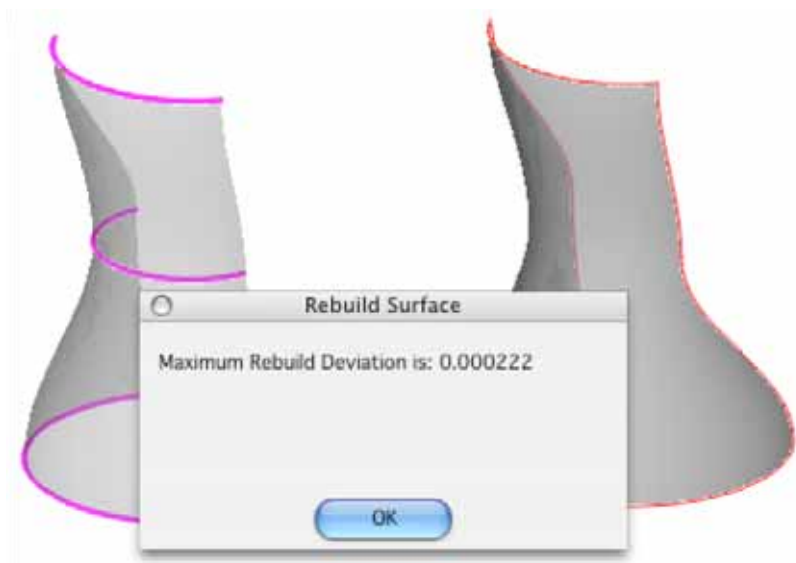
The Rebuild Surface tool is useful for converting analytics to NURBS, trimmed surfaces to untrimmed surfaces, and repairing surfaces. The rebuild tool is limited to surfaces with 3 or 4 sides.

Rebuild uses refitting algorithms to recalculate a new NURBS surface from the original. The rebuild will perform the following:

- Turns the resultant surface into a cubic surface.
- May reduce the number of control points.
- Fits an untrimmed surface to a trimmed surface.
- Repairs bad or questionable underlying geometry.
- Removes any discontinuities with the surface.

Using the Rebuild Surface tool

- 1 Select the surface to rebuild.
- 2 There are no options or Data Entry Fields for this operation.



Surface Rebuild

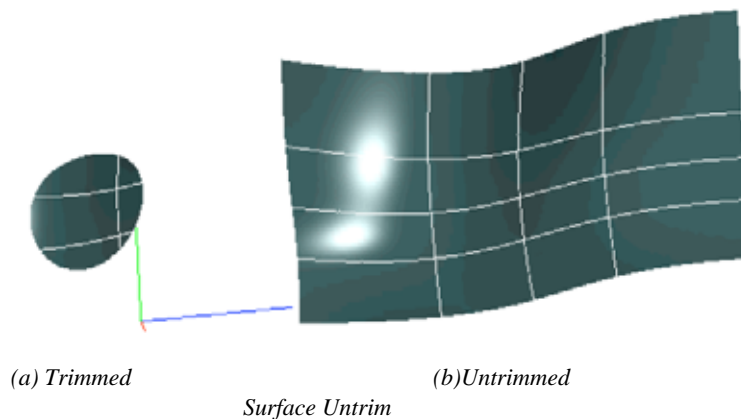


Untrim Surface

The Untrim Surface tool will remove all bounding curves on a surface.

Using the Untrim Surface tool

- 1 Select the surface to untrim.
- 2 There are no options or Data Entry Fields for this operation.

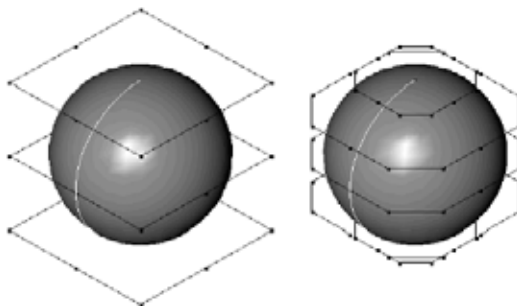


Elevate Surface

The Elevate Surface tool manipulates the underlying mathematical formula of a NURBS surface. NURBS are piecewise polynomial equations of different degrees. Typically surfaces created from skins, covers, and nets are third degree. Some shapes created from cylinders and spheres are second degree. By increasing the degree of a surface, more control points are introduced to aid local modifications. Normally, elevating to degree 5 or 7 provides enough control points to manipulate a surface, although Punch! Shark FX supports up to degree 22.

Using the Elevate Surface tool

- 1 Select the surface to elevate.
- 2 There are no options or Data Entry Fields fields for this operation.



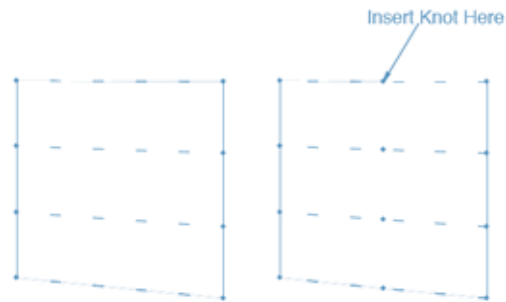
Elevate Surface

Insert Knot

The Insertion Knot tool introduces a new row or column of control points to a surface. By inserting a knot into a surface you gain more control points to manipulate for finer control over a shape. To use the Insertion Knot tool, pick a boundary edge on which to insert the knot.

Using the Insert Knot tool

- 1 Select the surface to insert a knot.
- 2 There are no options or Data Entry Fields for this operation.



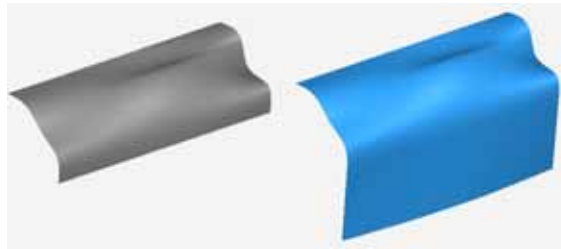
Surface Knot Insertion

Surface Extend

The Surface Extend tool will lengthen a side of the surface along its curvature. Selecting a face extends all of the edges associated with that surface. All of the underlying associativity is preserved and the offset distance by which the surface is extended can be defined on the Prompt window.

Using the Surface Extend tool

- Click a surface or edge to extend it the amount specified by the Offset field.

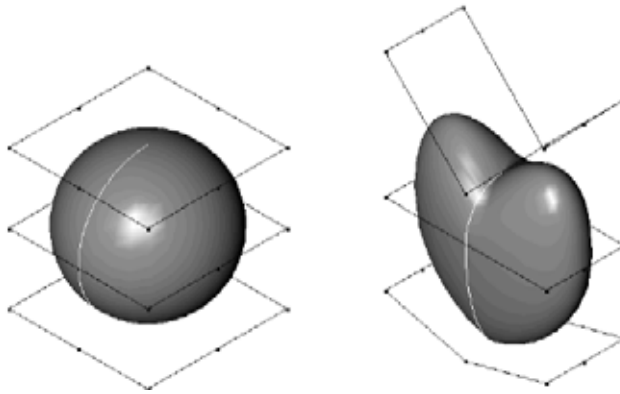


The resulting surface is offset in a manner that the 3D distance between the two is the specified offset distance; the distance is not measured along the actual surface (distance along the arc).



Surface Control Vertex Modification

Punch! Shark FX supports the manipulation of control points for a NURBS surface. This function is limited to “Surface Face” surfaces. For example, if you have a net, skin or cover surface, first change the object type by removing the associative links. Then use the Show Points tool to display the surface control points. In the example below, a solid sphere was converted to a Surface Face. Next, Show Points was used to expose the surface control points. The control points were then translated and moved to new positions to reshape the sphere into a heart shape. It is now possible to create a crease in the surface with this tool. Be sure to use surface analysis tools in conjunction with control vertex modification if surface smoothness and continuity are an issue.



Surface Control Vertex Modification

Part 5

Solid Modeling

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Intro to Solid Modeling

A solid model is an unambiguous 3D representation of an object. Solids are unambiguous because their mathematical descriptions completely define the inside and outside of a 3D object. Unlike wireframes and surfaces which define lengths and areas, solids accurately provide volume, mass, moments of inertia, centroids, and interference information. Solids can also be used to generate stereolithography models useful for demonstrating prototype concepts or for creating forms for molds or castings.

Punch! Shark FX uses a sophisticated solid modeling kernel that provides a precise boundary representation of solids suitable for design projects sensitive to accuracy. In Punch! Shark FX, linear and quadric geometry are exactly represented analytically while free-form geometries are represented as Non Uniform Rational B-Splines (NURBS). Unlike faceted solid modelers, Punch! Shark FX maintains a high level of accuracy as you perform Boolean or feature operations. Because of its strict accuracy, parts are suitable for precision-demanding numerical control applications.

Solid Modeling Menus

Access to the solid modeling tools is available from the Main Tool Palette. The Primary Solid Tool Palette contains five options with associated tear out menus.

The five tool palettes are described as follows:

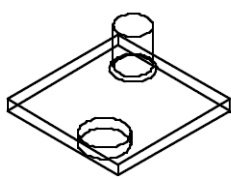
Primitives	Primitives are simple shapes defined by linear or quadric geometry. Punch! Shark FX supports spheres, blocks, cones, pyramids, prisms, tori, and slabs.
Profiles	A profile is a closed curve or collection of curves that can be swept, lathed, extruded or sectioned into a solid.
Features	A feature is a set of operations that simplify the building or modification of a solid by a common design task. Features include fillets, chamfers, holes, bosses, shelling, and bending.

Local Face Operations Local face operations include tools for drafting, removing, replacing, deforming, matching, moving, and offsetting individual faces within a solid.

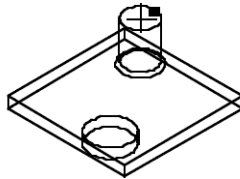
Utilities Punch! Shark FX provides a variety of solid modeling utility operations that include Boolean operations (union, subtract, and intersect), splitting, trimming, and thickening.

Snap tool and Solids

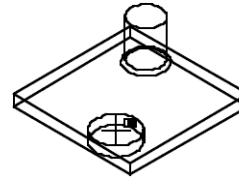
The Snap tool recognizes a variety of intelligent snap locations for solids. The Snap tool recognizes all hard vertices, user supplied points (grips), hole centers, cylinder centers, and fillet centers.



Vertex



*Cylinder Center
Solid Snaps*



Hole Center

Solid Selections and Display

Solids display two types of edges. The first is called a hard edge. Hard edges are permanent edges with respect to the solid and are present in all views. The second edge type is called a silhouette edge. A silhouette edge is a temporary edge that is dependent upon the view. A silhouette edge is where the surface normal makes a perpendicular angle with the view normal.

A solid is selected by pressing the mouse on an edge or within a face. Selecting the solid by its edge is considerably faster than casting a ray to see if it pierces the solid.



Coarse



Medium

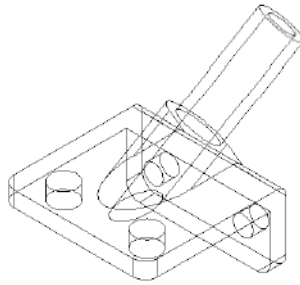


Fine

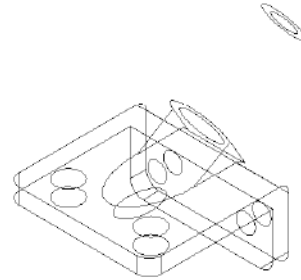


Super Fine

Resolution Options



Silhouette On



Silhouette Off

Silhouette Edges

Solid Primitives



Primitives are simple shapes internally represented as linear or quadric geometry. The designer creates primitives from one of several choices, one point, two point, and by diagonals. Once created, each primitive is modified by a set of parameters unique for each type. Primitives supported by Punch! Shark FX include spheres, blocks, cones, pyramids, prisms, tori, and slabs.

Sphere

A sphere is defined by a radius and origin. Three methods exist for creating spheres that include Sphere 1 Pt, Sphere 2 Pt, and Sphere Diagonals. The Snap tool registers the sphere center and points used by the designer to define the sphere.

Sphere One Point

The Sphere One Point method prompts the designer for the center location of the solid sphere. The designer defines the radius in a dialog box. One point spheres are independent of the orientation of the construction plane.

Using the Sphere One Point tool

- 1 Click a point defining the sphere center point.
- 2 Use the Data Entry Fields to adjust the center or diameter.

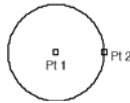


Sphere Two Point

Sphere Two Point creates a solid sphere from a center point and a point on the sphere extents. The distance from point 1 to point 2 defines the sphere radius.

Using the Sphere Two Point tool

- 1 Click a point defining the sphere center point.
- 2 Click a point defining the sphere radius point.
- 3 Use the Data Entry Fields to adjust the center or dx, dy, dz distances to point 2.



Sphere Diagonals

The Sphere Diagonals tool prompts the designer to specify a box by diagonals. The sphere center is positioned at the middle of the two points. The radius is calculated from the smallest length of the enclosing box.

Using the Sphere Diagonals tool

- 1 Click a point defining the sphere upper left extents.
- 2 Click a point defining the sphere lower right extents.
- 3 Use the Data Entry Fields to adjust the center or dx, dy, dz distances to point 2.



Blocks

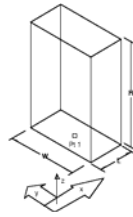
The Block primitive is a rectangle whose mathematical definition is provided by width, height, and depth. Three methods are available for defining blocks that include 1 Pt, 2 Pt, and by diagonals. For all three methods, the construction plane is used to orientate the block.

Block One Point

The Block One Point method creates a block from one point supplied by the designer. The point provided is the base center. The designer supplies the length, width, and height in a dialog box. The orientation of the block is defined by the length, width, and height values along the x, y, and z axis of the construction plane.

Using the Block One Point tool

- 1 Click one point for the block base center.
- 2 Use the Data Entry Fields to adjust center, length, height, and width fields.

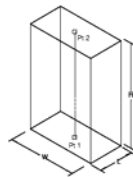


Block Two Points

The Block Two Points method creates a block whose height and z axis are defined by the two points supplied by the designer. The length and width are defined along vectors closest to the construction plane x and y axis.

Using the Block Two Points tool

- 1 Click a center point.
- 2 Click a point signifying the block height and z axis.
- 3 Use the Data Entry Fields to adjust center, length, and width fields.

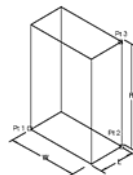


Block Diagonals

The Block by Diagonals method creates a block where point one is the block start, point 2 defines the length and width, and point 3 defines the block height.

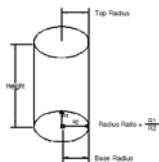
Using the Block Diagonals tool

- 1 Click base upper left point.
- 2 Click base lower right point.
- 3 Click a point along block height.
- 4 Use the Data Entry Fields to adjust center, length, height, and width fields.



Cylinder

The Cylinder primitive is defined by a top and base radius, base radius ratio, and height. The base radius ratio defines the ratio between the major and minor radius. A base radius ratio of 1 is a circle. Values not equal to one yield elliptical cylinders.

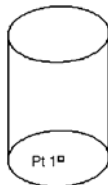


Cylinder One Point

The Cylinder One Point method creates a cylinder from a point that is the center of the cylinder base. The orientation of the cylinder is determined by the z axis of the construction plane.

Using the Cylinder One Point tool

- 1 Click cylinder base point.
- 2 Use the Data Entry Fields to adjust the base center, height, and diameter.

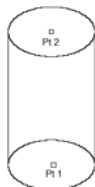


Cylinder Two Point

The Cylinder Two Point method prompts the designer for the base center and top center. The orientation of the cylinder is determined by a vector from pt2 and pt1.

Using the Cylinder Two Points tool

- 1 Click cylinder base point.
- 2 Click cylinder height point.
- 3 Use the Data Entry Fields to adjust the base center, height, and diameter.



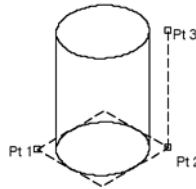


Cylinder Diagonals

The Cylinder Diagonals method creates a cylinder from three points. Point 1 and point 2 determine the base center extents. Point 3 determines the height. The cylinder may be elliptical.

Using the Cylinder Diagonals tool

- 1 Click cylinder base lower left point.
- 2 Click cylinder base lower right point.
- 3 Click cylinder height point.
- 4 Use the Data Entry Fields to adjust the center and height.



Cone

The Cone primitive defines a solid with an elliptical base degenerating into a point. Three methods exist for creating cones:

- One Point** Defines a cone from its base center.
- Two Points** Defines cone axis and height.
- Diagonals** Define cone by box diagonals.



Cone One Point

The Cone One Point prompts the designer to specify the cone base center.

Using the Cone One Point tool

- 1 Click cone base center.
- 2 Use the Data Entry Fields to adjust the center, diameter, and height attributes.





Cone Two Points

Cone Two Points creates a cone whose height axis is aligned with the given points. The height is extracted as the distance between the provided points.

Using the Cone Two Points tool

- 1 Click cone base center.
- 2 Click cone height.
- 3 Use the Data Entry Fields to adjust the center, height, and diameter attributes.

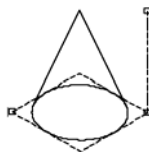


Cone Diagonals

The Cone Diagonals method creates a cone from three points. Points 1 and 2 define a rectangle for an elliptical cone base. Point 3 is used to determine the height of the cone. If point 3 is in the plane of point 1 and 2, a default height for the cone is used. By holding the Shift key after locating point 1, you can force the rectangle to be square providing a radius ratio of 1 (a circular base).

Using the Cone Diagonals tool

- 1 Click cone base lower left.
- 2 Click cone base upper right.
- 3 Click cone height point.
- 4 Use the Data Entry Fields to adjust the center and height attributes.



Torus

The Torus primitive is generated by revolving a circle about an axis. The torus is defined by an outer radius, inner radius, and center.

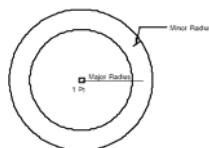


Torus One Point

The Torus One Point method prompts the user for the center point. The torus is aligned with the construction plane z axis.

Using the Torus One Point tool

- 1 Click torus center point.
- 2 Use the Data Entry Fields to adjust the major and minor radius values.

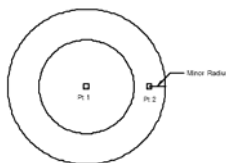


Torus Two points

The torus two point method prompts the torus center and tube center. The designer specifies the radius value for the tube in the torus parameter dialog box.

Using the Torus Two Points tool

- 1 Click torus center point.
- 2 Click torus radius point.
- 3 Use the Data Entry Fields to adjust the major and minor radius values.

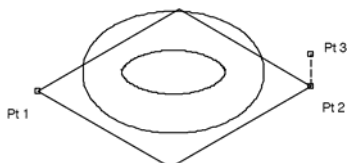


Torus Diagonals

The Torus Diagonals tool prompts the designer for three points. The first two points define a rectangle that locates the torus center and major radius length. The third point defines the radius of the tube (torus height).

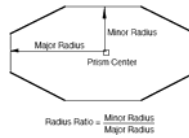
Using the Torus Diagonals tool

- 1 Click lower left point for torus base.
- 2 Click upper right point for torus base.
- 3 Click height point for torus minor radius.



Prism

The Prism primitive creates n-sided faced solids. The faces all form a 90 degree angle with the base. Prisms are defined with a major and minor radii, centers, and heights.



Prism Ratio Definition

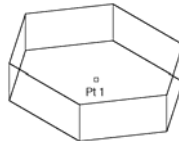


Prism One Point

The Prism One Point method prompts the designer for the prism base center. The height direction is taken from the z-axis of the current construction plane. The designer is prompted for additional radius values from the Prism dialog box.

Using the Prism One Point tool

- 1 Click the prism base center.
- 2 Use the Data Entry Fields to adjust the center, diameter, height, and number of sides attributes.

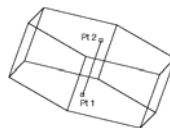


Prism Two Points

The Prism Two Point method creates a prism from two points defining the prism vertical axis. The designer is prompted for radius values with the Prism dialog box.

Using the Prism Two Points tool

- 1 Click the prism base center.
- 2 Click the prism height point.
- 3 Use the Data Entry Fields to adjust the center, diameter, height, and number of sides attributes.



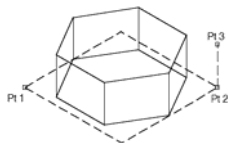


Prism Diagonals

The Prism Diagonals command creates a prism from three points. The first two points define a rectangle that specifies the prism's radius and radius ratio. The third point defines the prism height.

Using the Prism Diagonals tool

- 1 Click the first prism base corner.
- 2 Click the second prism base corner.
- 3 Click the prism height point.



Pyramid

A Pyramid contains n-sided flat faces sharing a common vertex tip. Punch! Shark FX supports three methods for creating pyramids.



Pyramid One Point

A One Point Pyramid method prompts the designer for the pyramid center. The height and radius parameters are defined in a dialog box. The one point pyramid is orientated along the construction plane z axis.

Using the Pyramid One Point tool

- 1 Click the pyramid base center.
- 2 Use the Data Entry Fields to adjust the pyramid height, diameter, and number of sides.



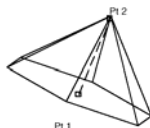
Pyramid Two Points

The Two Point Pyramid method prompts the user for two points that define the pyramid height and orientation.

Using the Pyramid Two Points tool

- 1 Click the pyramid base center.

- 2 Click the pyramid height center.
- 3 Use the Data Entry Fields to adjust the pyramid height, diameter, and number of sides.

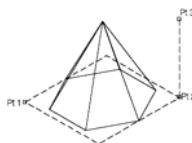


Pyramid Diagonals

The Pyramid Diagonals method prompts the designer for three points. The first two points define the length and width of the pyramid. The third point defines the pyramid height.

Using the Pyramid By Diagonals tool

- 1 Click the pyramid lower left point.
- 2 Click the pyramid lower right point.
- 3 Click the pyramid height center.
- 4 Use the Data Entry Fields to adjust the pyramid height, diameter, and number of sides.



Ellipsoid

The last primitive in the tool palette is an Ellipsoid. Similar to a sphere, ellipsoids are extremely smooth and are useful as a foundation for some very interesting shapes. The Inspector supports modifying the three radius values for an ellipsoid.



Ellipsoid One Point

A One Point Ellipsoid method prompts the designer for the ellipsoid center and radius values for the x, y, and z axis.

Using the Ellipsoid One Point tool

- 1 Click the Ellipsoid center.

- 2 Use the Data Entry Fields to adjust the center and diameters along the x, y, and z axis.



Ellipsoid Two Points

The Two Point Ellipsoid method prompts the user for two points that define the ellipsoid major axis.

Using the Ellipsoid Two Points tool

- 1 Click the ellipsoid center.
- 2 Click the ellipsoid major radius point.
- 3 Use the Data Entry Fields to adjust the center, minor radius 1, and minor radius 2.



Ellipsoid Diagonals

The Ellipsoid Diagonals method prompts the designer for three points. The first two points define the length and width of the ellipsoid. The third point defines the ellipsoid height.

Using the Ellipsoid Diagonals tool

- 1 Click the ellipsoid lower left point.
- 2 Click the ellipsoid lower right point.
- 3 Click the ellipsoid height.
- 4 Use the Data Entry Fields to adjust the ellipsoid center, and three radius values.

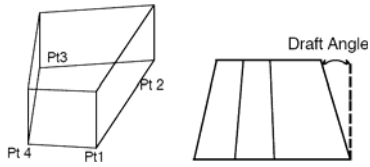


Slab Primitive

A Slab is a collection of closed planar points that are extruded into a solid. If the points do not close, Punch! Shark FX will automatically close them. The extrusion direction is determined by the order in which the points were entered. Clockwise points will produce a slab in the direction of the right hand rule. Likewise, counterclockwise points will produce a slab in the opposite direction of the right hand rule. Although points must be coplanar, they can be independent of the current construction plane.

Using the Slab Primitive tool

- 1 Click on 3 or more planar points.
- 2 Press the Esc key, double-click, Enter key or right mouse button to terminate slab creation.
- 3 Type in the height and optional draft using the Data Entry Fields.



Slab Primitive Points and Draft Angle

Edit a Primitive Slab

Once a slab is created, you can later modify the slab height and draft angle. You can change the direction of the slab by specifying a negative value in the height field.

Solids From Profiles



A profile is a closed 2D section created from curves, lines or polygons. The curves must lie in the same plane. A 2D profile can be lathed, swept, extruded or sectioned to create a solid. When a profile is used to create a solid, Punch! Shark FX creates a parent/child relationship between the profile (parent) and the solid (child). Any changes made to the parent will regenerate the child. For example, if you lathe a rounded rectangle polygon, you can edit the polygon radius and the solid will regenerate for the new shape.

Lathe Solid

The Lathe Solid command rotates any closed 2D polyline, circle, ellipse, closed spline or polygon. You can also lathe groups of closed curves. You cannot lathe a profile that intersects upon itself (possible for polylines, splines, and polygons).

Using the Lathe Solid tool

- 1 Select a closed profile (curves, groups).
- 2 Select a line for the revolution axis.
- 3 Use the Data Entry Fields to adjust angle and draft.

Note: You can specify positive or negative angles.

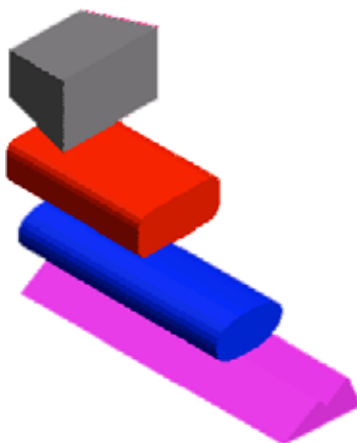


Extrude Solid

The Extrude Solid command takes a profile and extends the shape a user specified distance. You can extrude 2D polylines, circles, ellipses, closed splines, and polygons. There are five extrude methods.

- | | |
|---------------------|---|
| Distance | Extrudes a user defined distance. Uses profile normal for extrusion direction. |
| Vector | Extrudes a distance and direction defined by two points. |
| To Entity | Extrudes a direction defined by two points and terminates at a used defined surface or solid. |
| Mid Plane | Extrudes on both sides of the profile. |
| Thin Extrude | Extrudes open or closed curves with a thickness and distance. |

Many of the extrusion options include a draft angle. The draft angle default is zero, which results in the sides of the solid being parallel to the profile normal. If the draft angle is positive, the sides will be forced inward. Likewise, negative draft angles will force the sides outward. Some draft angles and extrusion heights will cause the solid to self intersect. If this happens you must reduce the draft angle or increase the extrusion height.



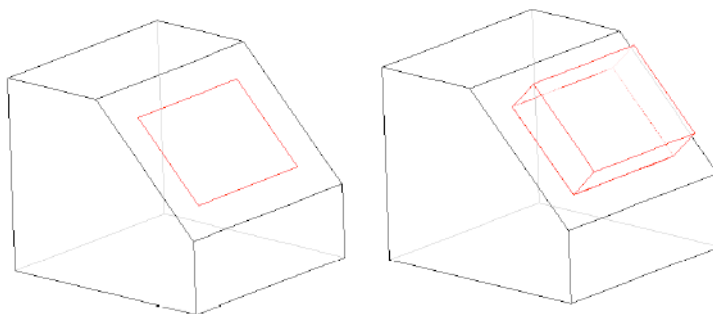
Solid Extrude

Distance

Distance extrudes normal to the plane of the profile a specified distance. The distance can be positive or negative.

Using the Extrude Distance tool

- 1 Select a closed profile (curves, groups).
- 2 Use the Data Entry Fields to adjust the extrusion distance.



Extrude Distance

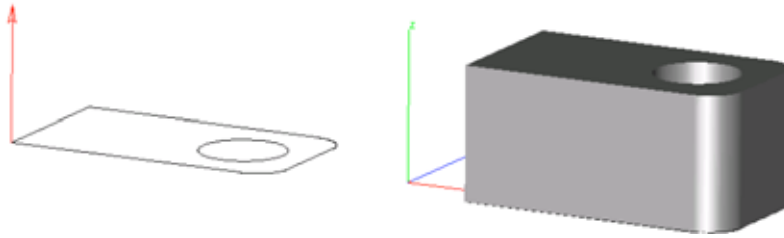
Vector

Vector extrudes a profile along a vector whose distance is defined by two user supplied points.

Using the Extrude Vector tool

- 1 Select a closed profile (curves, groups).

- 2 Click two points defining the direction and magnitude of the extrusion.
- 3 Use the Data Entry Fields to adjust the extrusion distance.



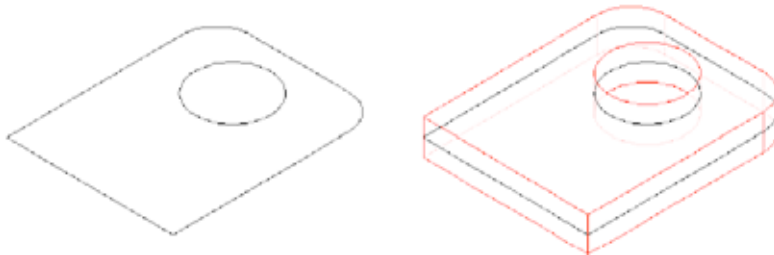
Extrude Vector

Mid Plane

Mid Plane extrudes a profile in both directions using the plane of the profile.

Using the Extrude Mid Plane tool

- 1 Select a closed profile (curves, groups).
- 2 Use the Data Entry Fields to adjust the extrusion distance.



Mid Plane Extrude

To Entity

To Entity extrudes along a vector terminating at a surface or solid. The terminating body is not unioned with the extruded body.

Using the Extrude To Entity tool

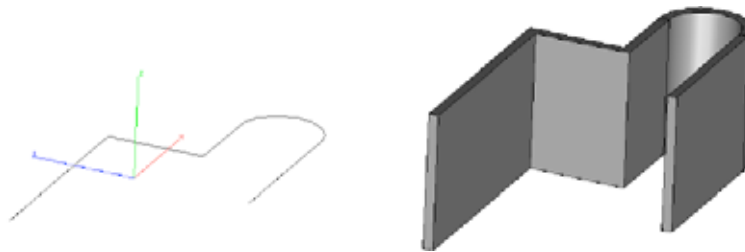
- 1 Select a closed profile (curves, groups).
- 2 Select a terminating surface or solid
- 3 Use the Data Entry Fields to adjust the draft.

*Extrude To Entity****Thin Extrude***

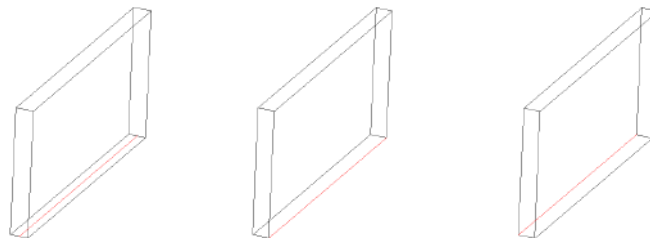
Thin Extrude extrudes an open or closed profile by a vector and thickness.

Using the Thin Extrude tool

- 1 Select one or more curves.
- 2 Click two points defining the direction and magnitude of the extrude.
- 3 Use the Data Entry Fields to adjust the height and draft.

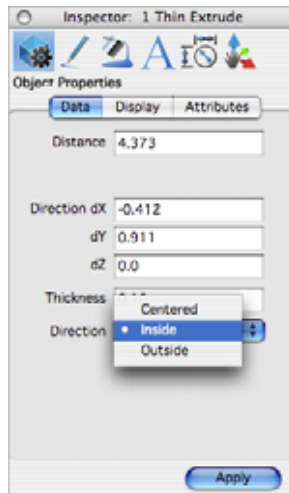
*Thin Extrude*

Use the Option key to toggle between both sides, right side and left side, for offset direction.

*Middle**Right
Thin Offset Sides**Left*

The Inspector includes edit fields for distance, direction, thickness, and offset options for centered, inside, and outside.

There are certain situations where the Thin Extrude tool will fail and produce the error message “Gap cannot be filled”. This error message results from curves that cannot be extended (this will often be noticed with splines). One means to work around this is to thicken the extrusion to the opposite offset side. You can accomplish this by specifying a negative offset value.

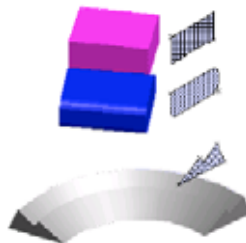


Thin Extrude - Inspector



One Rail Sweep Solid

A sweep is similar to an extruded solid except instead of an extrusion vector you specify a sweep path. A sweep path can be a line, circle, arc, ellipse, conic or spline.



The One Rail Sweep has two pull downs for options on the Prompt window. The first drop-down controls how the profile orientation sweeps along the rail. It contains the following three options:

- Sweep In Place (profile is not translated or aligned to path).
- Sweep Perp (profile is translated and aligned perpendicular to path).
- Sweep Rigid (profile orientation is maintained irregardless of path tangent).

The next drop-down controls how the profile is terminated. It contains the following three options:

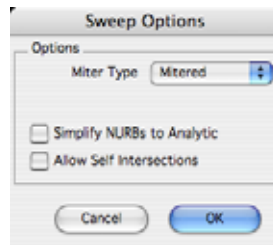
- Curve extents (profile is swept across entire curve extents).

- To Body (profile is terminated at a surface or solid).
- Between Points (user specifies two points on the path).

Sweep Perpendicular rotates the sweep profile to be perpendicular to the take off path. Sweep in Place leaves the sweep as is; no aligning of the profile occurs with this option. Sweep To Entity sweeps the profile along the given path and terminates to a reference surface or solid.

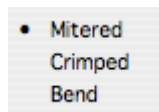
Sweep Options

The Options button displays the Sweep Options dialog. Here you can further control the behavior of the tool.



Miter Type

Controls how material is added along paths that are not tangential.



Mitered Corners are finished with two 45-degree angled edges.

Crimped Corners are finished with a curved section connecting the two edges.

Bend Corners are finished with a combination of curved sections and angled edges.

Bend Radius

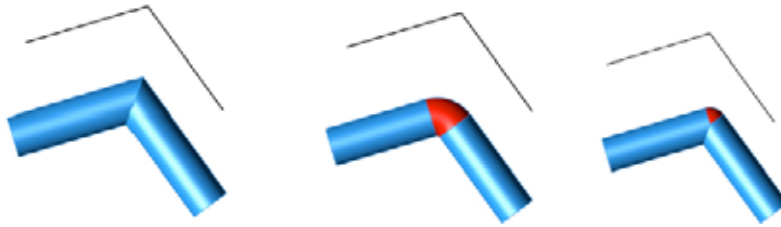
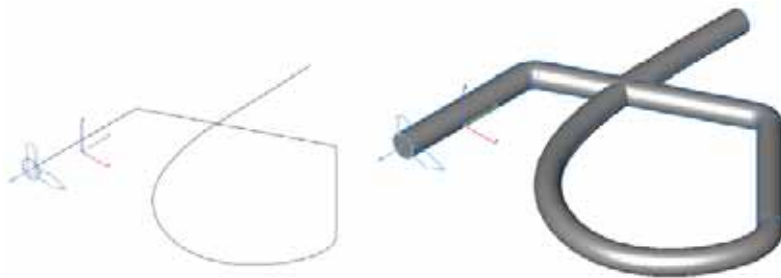
Displayed when using the Bend Miter type to set the radius added material.

Simplify NURBS to Analytics

Converts NURBS surfaces into analytics, where possible. Analytics typically take up less memory and provide faster calculations.

Allow Self Intersections

Enables sweeps to self intersect along the path.

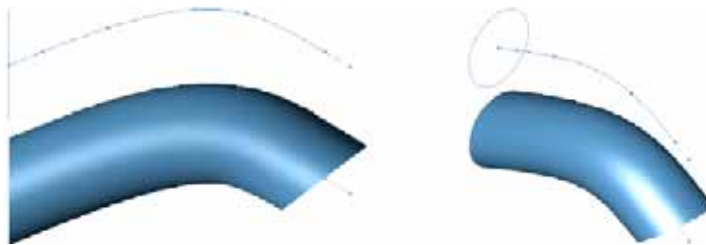
*Mitered**Crimped
Miter Options**Bend**Crimped Sweep with Self Intersection*

Sweep In Place

Sweep In Place preserves the starting orientation of the profile with respect to the path.

Using the Sweep In Place tool

- 1 Select profile curves.
- 2 Select rail curve for sweep path.
- 3 Use Data Entry Fields to adjust optional twist and draft.

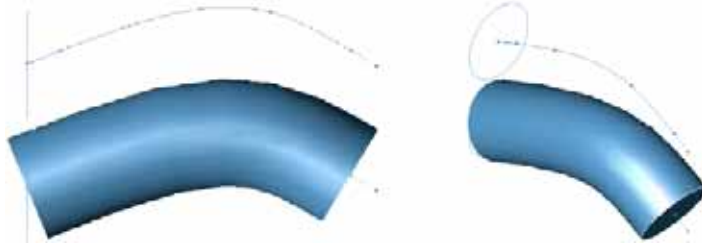
*Sweep In Place*

Sweep Perpendicular

The Sweep Perpendicular method realigns the starting orientation of the profile with respect to the path.

Using the Sweep Perpendicular tool

- 1 Select profile curves.
- 2 Click one point defining profile origin. The origin of the profile is transformed to the start of the path.
- 3 Select rail curve for sweep path.
- 4 Use Data Entry Fields to adjust optional twist and draft.



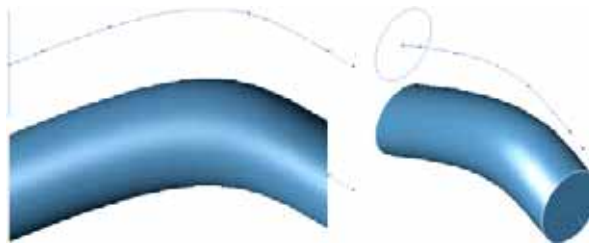
Sweep Perpendicular

Sweep Rigid

The Sweep Rigid method preserves the starting orientation of the profile throughout the entire sweep. The profile is never rotated or aligned to the path.

Using the Sweep Rigid tool

- 1 Select one or more curves.
- 2 Click two points defining the direction and magnitude of the extrude.



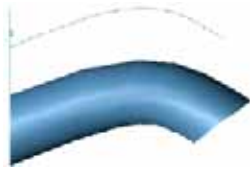
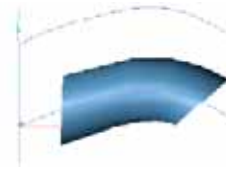
Sweep Rigid

Sweep Termination

Sweeps can be terminated by curve extents, to body or between points. The following image demonstrates the three termination methods.

- Curve extents (profile sweeps across entire curve extents).
- To Body (profile is terminated at a surface or solid).

- Between Points (user specifies two points on the path).

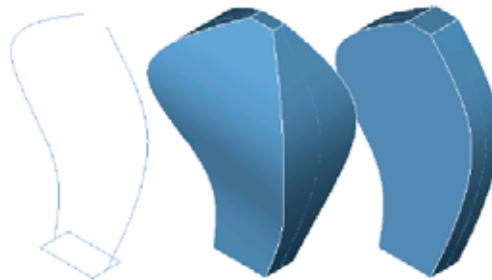
*Curve Extents**To Entity
Sweep Termination**Between Points*

Two Rail Sweep Solid

The Two Rail Sweep Solid tool creates a swept solid by sweeping a closed curve between two rail curves. The rail curves define the orientation and scale of the profile as it moves between the two curves.

Using the Two Rail Sweep Solid tool

- Pick a profile to sweep.
- Pick the first rail path.
- Pick the second rail path.
- Hold the Option key at step one if you want to maintain the profile height, as it is swept.

*Curves**Default
Sweep**Maintain
Height**Sweep Two Rail Solid*

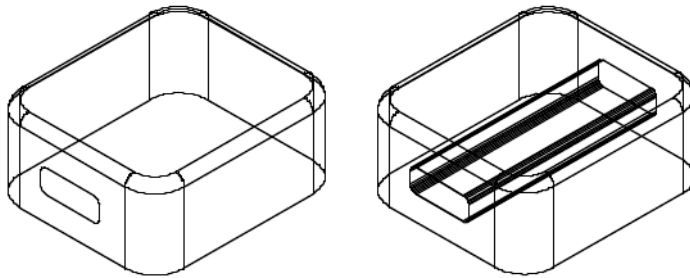
CutOut Solid

CutOut Solid is a method for quickly subtracting material from a solid.

Using the CutOut Solid tool

- Select a profile to cutout.
- Select a solid.

- 3 Click two points defining the direction and distance for the profile to be swept and removed from the solid.
- 4 Use the Data Entry Fields to adjust the distance and draft angle.

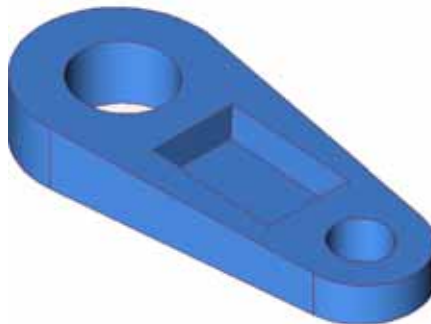
*Cutout Solid*

CutOut By Distance

The CutOut tool supports material cutouts by distance. The direction is relative to the selected face upon which the 2D profile lies.

Using the CutOut By Distance tool

- 1 Click the CutOut tool.
- 2 Click the drop-down on the Data Entry window and choose Distance.
- 3 Select the face upon which the profile lies.
- 4 Select the curves to cutout.

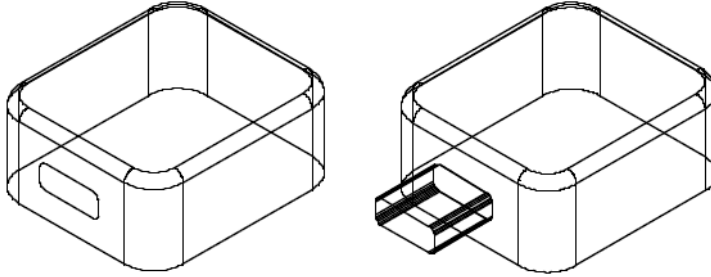
*Cutout By Distance*

Protrusion Solid

A protrusion feature is a quick method for adding material to an existing solid.

Using the Protrusion Solid tool

- 1 Select a profile to protrude.
- 2 Select a solid.
- 3 Click two points defining the direction and distance for the profile to sweep and be added to the selected solid.
- 4 Use the Data Entry Fields to adjust the distance and draft angle.



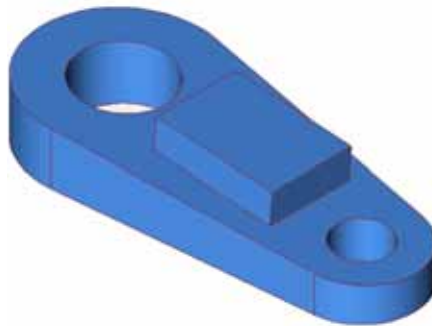
Protrusion Solid

Protrude By Distance

The Protrude tool supports protrusions by distance. The distance is relative to the selected face upon which the 2D profile lies.

Using the Protrude By Distance tool

- 1 Click the Protrude tool.
- 2 Click the drop-down on the Data Entry window and choose Distance.
- 3 Select the face upon which the profile lies.
- 4 Select the curves to protrude.



Protrude By Distance



Skin Solid

A Skinned Solid is created from a collection of closed 2D profiles. There are two methods for skinned solids. The first creates a solid from the given profiles. The second allows the user to select guide curves that control the shape between the sections. The profiles must be selected in the order they create a solid (use Shift select). If a profile has more than one curve, be sure to group the curves.

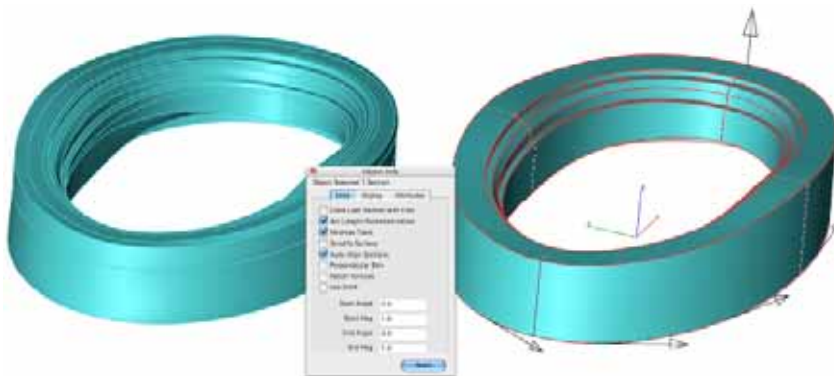
The Skin Solid data properties attributes, located in the Inspector, now includes an option for turning on/off Match Vertices. This option suppresses the vertex-matching algorithm which ensures that all profiles consist of the same number of co-edges. A heuristic approach is used to determine which vertex pairs are good matches. Profile co-edges are then split where additional vertices are needed. This option is forced to TRUE if the co-edge numbers of the profiles are not equal. Its default is true.

Using the Skin Solid tool

- 1 Hold the Shift key and select profiles; release when done.
- 2 There are no Data Entry Fields options for this tool.



Skinned Solid



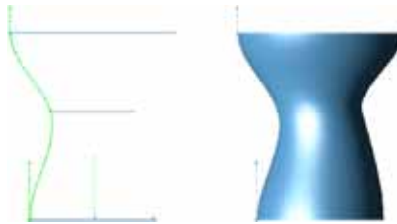
Match Vertices

 **Guide Skin Solid**

This tool guides curves. The resulting body will pass through the provided guide curves. Be sure your guide curve passes precisely through the profile section or it will be ignored. Guide curves must be single (no groups), curvature continuous curves. You may have any number of guide curves.

Using the Guide Skin Solid tool

- 1 Select the second icon in the message line for skin with guides.
- 2 Hold the Shift key and select profiles; release when done.
- 3 Hold the Shift key and select guide curves; release when done.
- 4 There are no Data Entry Fields options for this tool.



Skinned Solid with Guide Curves

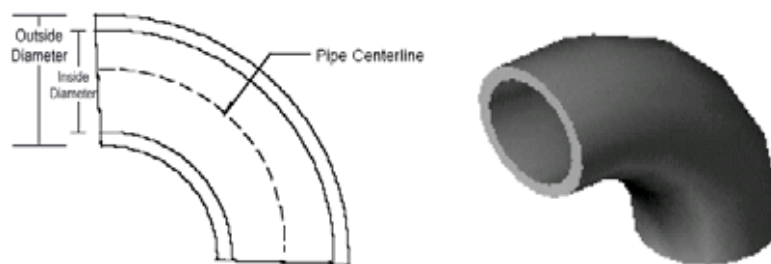
 **Pipe**

The Pipe Solid command provides a rapid feature for creating pipes and tubes. To create a pipe you select a curve which is the pipe centerline. Then specify the pipe inner and outer diameter values.

When a pipe is created Punch! Shark FX creates an intelligent link between the solid and the pipe centerline. Whenever the pipe centerline is moved or modified, the pipe solid will regenerate. In addition to changing the pipe path, you can perform a *Window:Inspector* to verify or change the pipe diameter values. Specifying an inside diameter of 0 will provide a pipe without a hollowed inside.

Using the Pipe tool

- 1 Select the curve for the pipe center line.
- 2 Use the Data Entry Fields to adjust the inner and outer diameter values (or the Inspector).



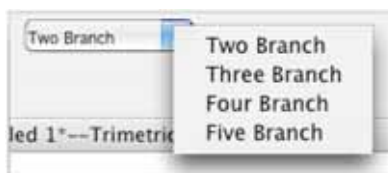
Pipe Solid



Branched Solid

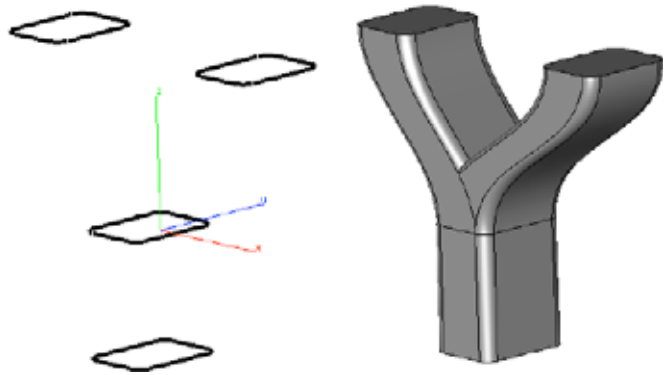
The Branched Solid tool is used to create shapes with profiles that resolve two or more branched conditions. The Branch tool is located as the last item within the Solid From Profiles tool palette.

The Branch tool requires the user first select a collection of profiles for the trunk. The trunk profiles represent the core set of surfaces from which all others join. The second set of profiles are called branches. These branches are smoothly integrated into the trunk profiles. Punch! Shark FX supports up to five branches from the trunk.



Using the Branched Solid tool

- 1 Select the Branched Solid tool from the Solid From Profiles palette.
- 2 Select the Number of Branches from the drop-down menu.
- 3 Hold the Shift key and select one or more profiles for the trunk body.
- 4 Hold the Shift key and select one or more profiles for the first branch.
- 5 Repeat for the number of branches desired.



Branched Solid

Feature Based Solids



A feature is a set of operations that simplify the building or modification of a solid by a common design task. Features include Booleans (add, subtract, intersect), fillets, chamfers, countersink, counterbore, boss, cutouts, protrusions, and splitting.

Fillet/Blend

The Blend tool creates a constant radius fillet or rounds the edge of a solid. A fillet is a blend operation that adds material, whereas a round subtracts material. Punch! Shark FX divides its blending functions through two icons that represent tools for constant and variable cross section blending.



Constant Cross Section

- Radial
- Elliptical
- Vertex Blend

Variable Cross Section

- Linear
- By Position
- Radius Curve
- Fixed Width
- Hold Line
- Three Face Blend
- Curvature Continuous

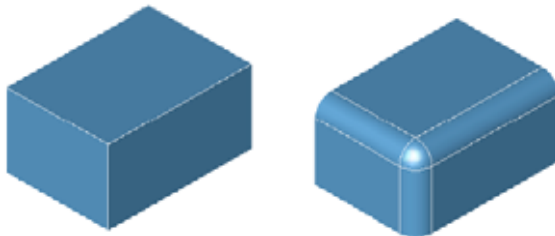
Constant Blend

The Constant Blend tool creates a constant radius fillet across one or more edges. There are three types of constant blends available on Data Entry window.

- Radial
- Elliptical
- Vertex Blend

Using the Constant Radial Fillet tool

- 1 Select the solid edge or face you want to fillet. Hold down the Shift key if you need to fillet more than one edge.
- 2 Use the Data Entry Fields to adjust the radius value.



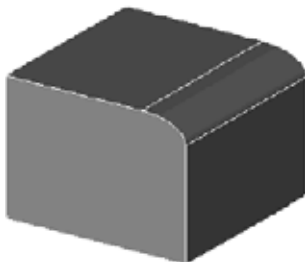
Constant Radial Fillet

Elliptical Fillet

The Elliptical Blend option defines an elliptical cross section for the blend. The ellipse ratio is the relationship between the major and minor radii of an ellipse. The major radius is defined in the Data Entry Fields and edit box. The minor radius is the product of the major radius and its ratio.

Using the Constant Elliptical Fillet tool

- 1 Select the Constant Blend tool from the Blend Edge sub-tools palette.
- 2 On the Data Entry window, click the drop-down and choose Elliptical.
- 3 Select the edge you want to fillet. Hold the Shift key down if you need to fillet more than one edge.
- 4 Use the Data Entry Fields to adjust the radius value.



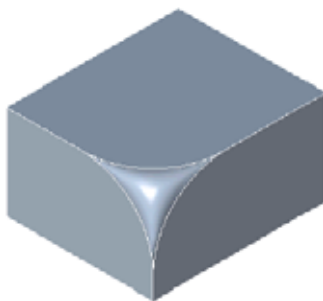
Elliptical Blend

Vertex Blend

A Vertex Blend places a blend at the corner, where three or more edges meet. The vertex blend is limited to corners that share the same convexity.

Using the Vertex Blend tool

- 1 Select the Constant Blend tool from the Blend Edge sub-tools palette.
- 2 On the Data Entry window, click the drop-down and choose Vertex Blend.
- 3 Select the vertex you want to fillet.
- 4 Use the Data Entry fields to adjust the radius value.



Vertex Blend

Variable Linear Blend

The Variable Blend tool creates a fillet that has a starting radius value that is different from the ending radius value. There are seven types of variable blends available on the Data Entry window.



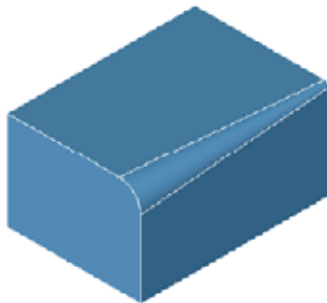
Linear

The Linear option fillets an edge along a selected line. The R1 and R2 values specify the radius.

Using the Linear Fillet tool

- 1 Select the Variable Blend tool from the Blend Edge sub-tools palette.
- 2 Select the edge you want to fillet. Hold down the Shift key to select multiple edges.

- 3 Use the Data Entry Fields to adjust the start and end radius value.



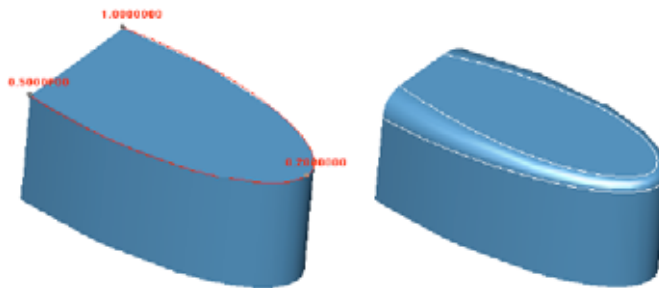
Linear Fillet

By Position

The By Position Fillet tool allows you to specify radius values at specific locations along the edge to blend.

Using the By Position Fillet tool

- 1 Select the Variable Blend tool from the Blend Edge sub-tools palette.
- 2 On the Data Entry window, click the drop-down and choose By Position.
- 3 Select the edge you want to fillet. Hold the Shift key down to select multiple edges.
- 4 Click at a location on the edge and specify the associated radius value.
- 5 Terminate with a right click, double click or hit Enter from the keyboard.



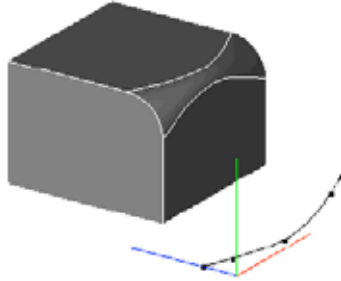
By Position Fillet

Radius Curve

The Radius Curve option allows you to select an edge to blend and a curve to define the radius distribution. The curve must be in the positive xy quadrant with the y value representing the radius. The percent along the curve is mapped to a percent along the edge being blended. One of the advantages of controlling the radius by a curve is that you can add or remove control points and set the slope.

Using the Radius Curve tool

- 1 Select the Variable Blend tool from the Blend Edge sub-tools palette.
- 2 On the Data Entry window, click the drop-down and choose Radius Curve.
- 3 Select the edge you want to fillet.
- 4 Select radius curve for the fillet.



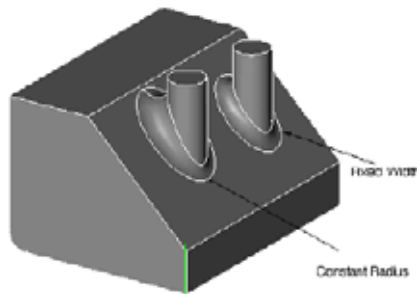
Variable Blend by Radius Curve

Fixed Width

The new Fixed Width function, located in the variable cross section pull-down, sets the explicit width between the tangent lines. In this case the radius value fluctuates for a constant width blend.

Using the Fixed Width tool

- 1 Select the Variable Blend tool from the Blend Edge sub-tools palette.
- 2 On the Data Entry window, click the drop-down and choose Fixed Width.
- 3 Select the edge you want to fillet.
- 4 Use the Data Entry Fields to specify a fixed width radius value.



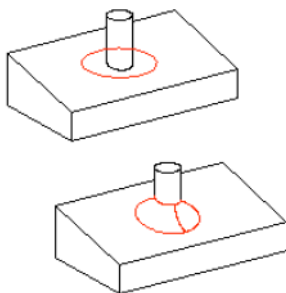
Fixed Width Blend

Hold Line

The Hold Line blend specifies one of the tangent lines of the resulting blend. A variable radius blend is created to match the hold line.

Using the Hold Line tool

- 1 Select the Variable Blend tool from the Blend Edge sub-tools palette.
- 2 On the Data Entry window, click the drop-down and choose Hold Line.
- 3 Select the edge you want to fillet.
- 4 Select curve for the fillet hold line.



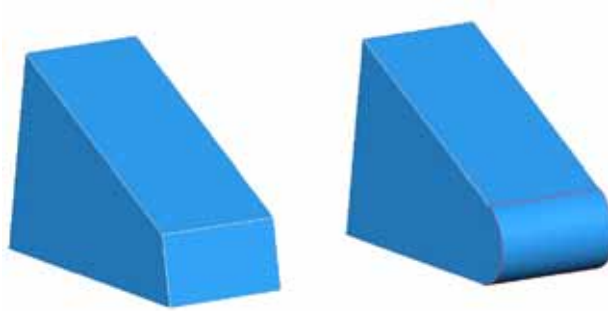
Hold Line Blend

Three Face Blends

The Three Face Blend tool creates a variable radius blend between three faces. The center or shared face is replaced with a smoothly blended surface. The radius is controlled completely by the geometry of the three faces.

Using the Three Face Blend tool

- 1 Select the Variable Blend tool from the Blend Edge sub-tools palette.
- 2 On the Data Entry window, click the drop-down and choose Three Face Blend.
- 3 Click the first side of the blend.
- 4 Click the second side of the blend.
- 5 Click the center of the blend.



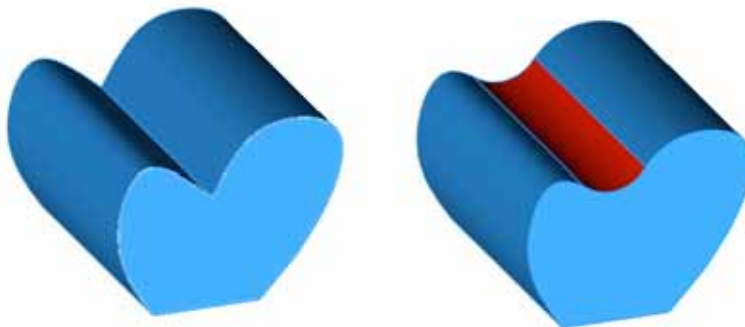
Three Face Blend

Curvature Continuous

The Curvature Continuous blend tool creates a G2 fillet surface along selected edges. A G2 blend takes into consideration the second derivative of the adjacent faces, creating a blended surface where the radius of curvature is continuous across the shared boundary.

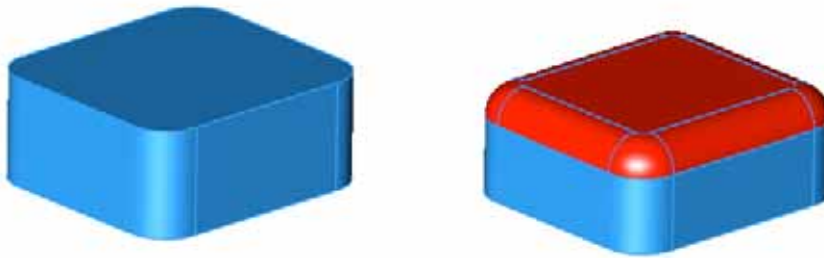
Using the Curvature Continuous blend tool

- 1 Select the Variable Blend tool from the Blend Edge sub-tools palette.
- 2 On the Data Entry window, click the drop-down and choose Curvature Continuous.
- 3 Specify the radius in the Data Entry Window
- 4 Select the edges to blend.



G2 Edge Blending

This tool also allows you to pick a face for blending. When a face is selected, all of its edges are assigned a variable radius.



G2 Face Blending

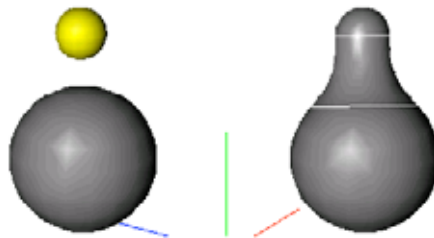
Important: There is no automatic blend reordering, as with constant radius blending. It is recommended that larger radius edges are blended first. Also, chained edges must use the Tangent Continuous method; non-tangent edges will be ignored.

Face/Face Blends

The Constant Radius Fillet tool allows the selection of face/face blends instead of the normal edge based blending. Face/face blending allows you to perform disjoint blends where the faces do not share a common edge.

Creating Face/Face Blends

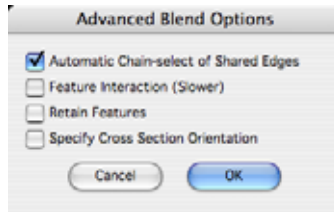
- 1 Pick the Constant Radius Fillet tool.
- 2 Use the Data Entry Fields to specify a suitable radius value.
- 3 Hold the Shift key, and select the first face.
- 4 While still holding the Shift key, select the second face.
- 5 Click a point between the two faces for a helper point.



Disjoint Body Blend

Blend Options

The Blend tools have advanced options that are available. Click the Options button to display the Advanced Blend Options dialog.



Fillet Options

Automatic Chain-Select of Shared Edges

When you select an edge, all edges that are tangent continuous are automatically selected as well.

Feature Interaction

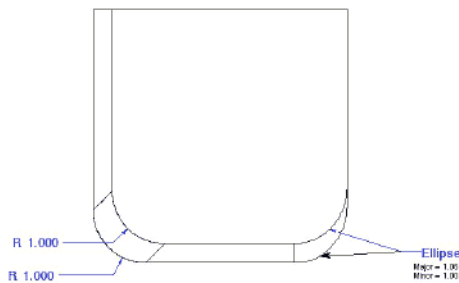
The Feature Interaction option will recognize and preserve additional intersections such as protrusions and cutouts within the blend region. This requires a longer blending time.

Retain Features

The blending has an option to preserve cutouts or protrusions.

Specify Cross Section Orientation

Normally the blend cross section is perpendicular to the edge being blended. Using this option, select a curve representing the spine that defines the cross section orientation. For example, in the figure below, the blend on the right was created by the defaults. The one on the left side represents the blend created by picking a vertical edge for the cross section orientation.



Blend Orientation Option



Chamfer

A Chamfer feature adds or removes a flat section of material from a solid along a collection of one or more selected edges. The flat section creates a beveled surface between the surfaces common to the selected edges.



Chamfer

Punch! Shark FX supports nine user interface options for creating chamfers. These nine options are classified into constant cross section and variable cross section chamfers. Constant chamfers maintain the cross section properties from start to finish, whereas variable cross sections change lengths by a variety of options.



Constant Cross Section

Length
Two Lengths
Length/Angle
Vertex Corner

Variable Cross Section

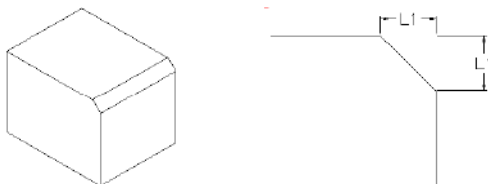
Lengths
Four Lengths
Lengths/Angles
By Position
Fixed Width

Length

This tool creates a constant length chamfer across one or more selected edges. The length from the common vertex is the same.

Creating Length Chamfers

- 1 Use the Data Entry Fields to specify a suitable chamfer value.
- 2 Select the edges for chamfering.



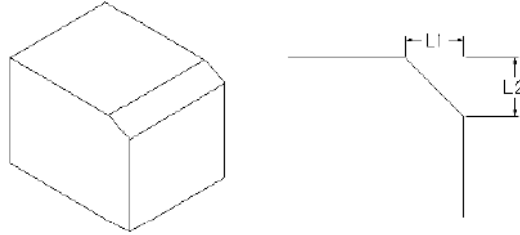
Constant Length Chamfer

Two Length

This tool creates a constant two length chamfer across one or more selected edges. Two lengths are used to specify the distance from the common vertex.

Creating Two Length Chamfers

- 1 Use the Data Entry Fields to specify chamfer length 1 and length 2 values.
- 2 Select the edges for chamfering.



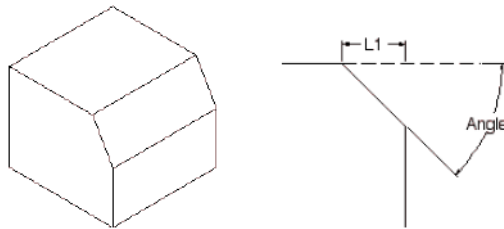
Constant Two Length Chamfer

Length/Angle

This tool creates a constant length/angle chamfer across one or more selected edges. A length offset is used to measure the distance for side one and angle for the other.

Creating Length/Angle Chamfers

- 1 Use the Data Entry Fields to specify a suitable chamfer length and angle values.
- 2 Select the edges for chamfering.



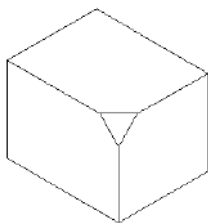
Constant Length/Angle Chamfer

Vertex Corner

The Vertex Corner Chamfer creates a notch at the corner of a solid, where three or more edges meet. The setback is the distance from the vertex back along the shared edges to the vertex. Vertex Corner requires the surfaces be planar.

Creating Vertex Corner Chamfers

- 1 Use the Data Entry Fields to specify a suitable chamfer distance.
- 2 Select the vertex for chamfering.



Vertex Corner Chamfer

Variable Chamfer

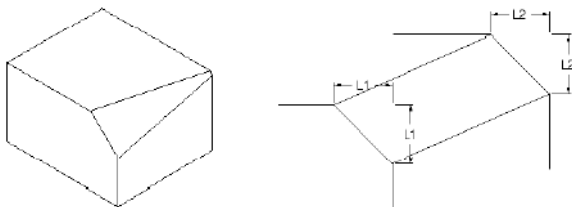
This tool creates a variable lengths chamfer across one or more selected edges.

Lengths

The Lengths option chamfers an edge along a selected line. This uses two length offsets where the lengths are applied to the start and end of the chamfer.

Creating Variable Lengths Chamfers

- 1 Use the Data Entry Fields to specify the two lengths.
- 2 Select the edges for chamfering.



Variable Lengths Chamfer

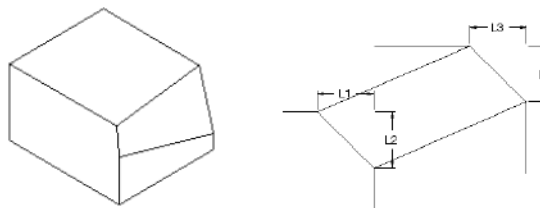
Four Lengths Chamfer

This tool creates a variable lengths chamfer across one or more selected edges. This uses four length offsets where two lengths are applied to the start and two at the end of the chamfer. Length 1 and 2 are applied to the start of the edge.

Creating Four Length Chamfers

- 1 Use the Data Entry Fields to specify the four lengths.

- 2 Select the edges for chamfering.



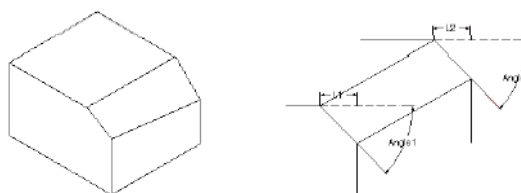
Variable Four Lengths Chamfer

Lengths/Angles Chamfer

This tool creates a variable lengths chamfer across one or more selected edges. This uses a length/angle setting at the start and another at the end.

Creating Lengths/Angles Chamfers

- 1 Use the Data Entry Fields to specify the length/angle pairs.
- 2 Select the edges for chamfering.



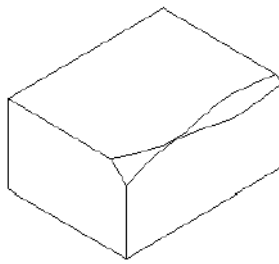
Variable Lengths/Angles Chamfer

By Position Chamfer

The Chamfer by Position tool specifies set-back values at locations along an edge. This tool's user interface is similar to the Variable Radius by Position tool.

Creating By Position Chamfers

- 1 Select the edges for chamfering.
- 2 Click a position along the edge and type in a chamfer length in the Data Entry Window.
- 3 Continue defining positions, end with ESC, ENTER, right click or double click.



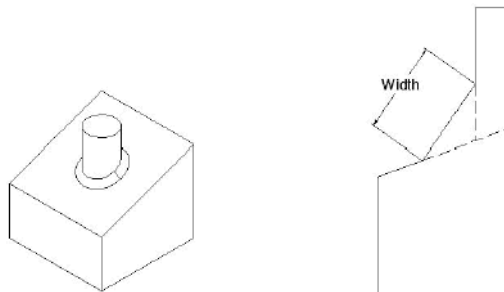
Chamfer by Position

Fixed Width Chamfer

The Fixed Width Chamfer creates a chamfer that controls the distance between the chamfer edge lines, similar to controlling the hypotenuse of a triangle. The distances vary to maintain the fixed width value.

Creating Fixed Width Chamfers

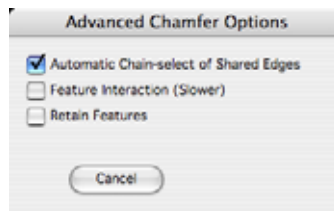
- 1 Use the Data Entry Fields to specify a fixed width value.
- 2 Select the edges for chamfering.



Fixed Width Chamfer

Chamfer Options

The chamfer options have attributes for chaining neighboring tangent edges, facilitating feature interaction, and retaining features.



Chamfer Options

Automatic Chain-Select of Shared Edges

When you select an edge, all edges that are tangent continuous are automatically selected as well.

Feature Interaction

The Feature Interaction option will recognize and preserve additional intersections such as protrusions and cutouts within the chamfer region. This requires a longer chamfering time.

Retain Features

The blending has an option to preserve cutouts or protrusions.

Hole

Holes are very useful for modeling bolt holes found frequently in mechanical design parts. In Punch! Shark FX holes are a predefined feature that removes cylinder shapes from an object. Holes, like other features in Punch! Shark FX, are associative and can be modified at any point in the design process. Punch! Shark FX supports three hole types:

- Simple
- Counterbore
- Countersink



Example Holes

All three hole creation tools will prompt the user to project the snap point if the snap point does not lie on the face. The snap is projected along the hole orientation to the selected face.

Hole Depth

The Prompt window contains a drop-down menu with options that control the depth of the hole. The options include:

- To Depth** The hole depth is defined by a user defined distance.
- Through** The hole goes completely through the body.
- First Blind** The hole stops at the first outward face it intersects.

To Face The hole terminates at a user defined face.

Hole Orientation

The Prompt window contains a drop-down menu with options that control the orientation of the hole. The options include:

- Normal** The hole is perpendicular to the face normal.
- X-Axis** The hole axis is aligned with the global x axis.
- Y-Axis** The hole axis is aligned with the global y axis.
- Z-Axis** The hole axis is aligned with the global z axis.
- 2-Pts** The hole axis is aligned with user supplied two points.

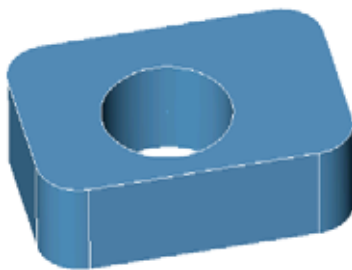


Simple Hole

A Simple Hole is defined by a hole center point, radius, and depth.

Creating a Simple Hole

- 1 Select the hole depth of “Through”.
- 2 Select the hole orientation of “Normal”.
- 3 Pick the face for the hole.
- 4 Specify location for hole center.
- 5 Adjust the hole center, diameter, and draft angles using the Data Entry Window.



Simple Through Hole

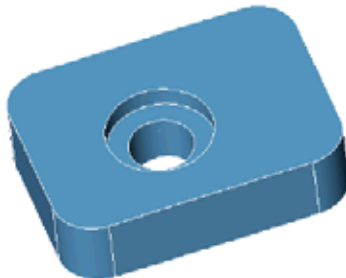


Counterbore Hole

A Counterbore feature allows the designer to specify a bore depth and bore radius in addition to the main radius and depth. The bore radius should be greater than the hole radius and the bore depth must be less than the hole depth. Unlike the countersink which has angled sides, the counterbore has straight sides.

Creating a Counterbore Hole

- 1 Select the hole depth of “Through”.
- 2 Select the hole orientation of “Normal”.
- 3 Pick the face for the hole.
- 4 Specify location for hole center.
- 5 Adjust the hole center, diameter, bore depth, bore diameter, and draft angles using the Data Entry Window.



Counterbore Through Hole

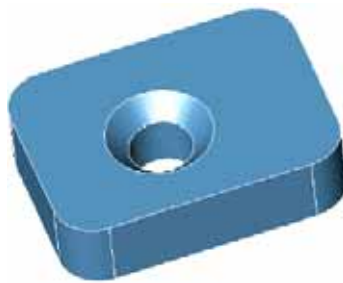


Countersink Hole

A Countersink feature allows the designer to specify a sink angle and sink diameter in addition to the main radius and depth. Unlike the counterbore which has straight sides, the countersink has angled sides.

Creating a Countersink Hole

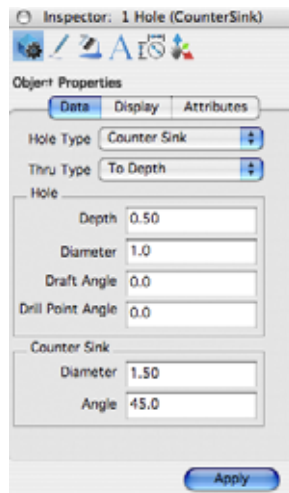
- 1 Select the hole depth of “Through”.
- 2 Select the hole orientation of “Normal”.
- 3 Pick the face for the hole.
- 4 Specify location for hole center.
- 5 Adjust the hole center, diameter, sink angle, sink diameter, and draft angles using the Data Entry Window.



Countersink Through Hole

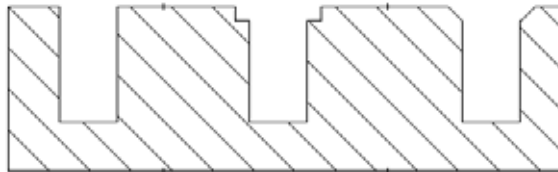
Drill Point Angle

The drill point is an attribute of holes; you can specify a drill point angle. When a hole is selected, the Drill Point Angle field is available on the Inspector. You can enter a value to create a drill point angle.

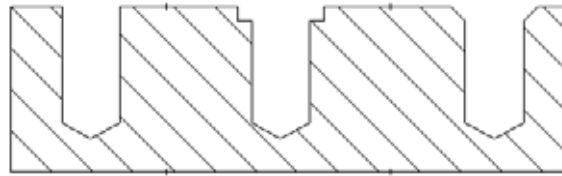


Drill Point Angle on Inspector

Below is an example of a simple hole, counterbore, countersink, (left to right):



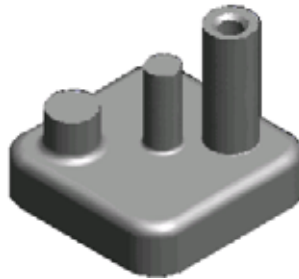
Drill Point Angle at 0



Drill Point Angle at 118

Boss

A Boss is a cylinder added to another solid with a radius applied at the intersection of the cylinder and base. You can also add draft to the cylinder.



Example Boss Features

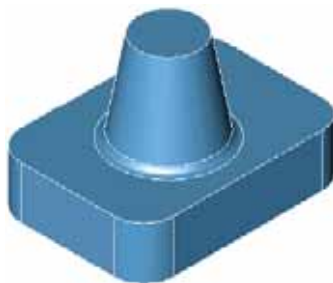
Boss Orientation

The Boss message line contains a pull down menu with options that control the orientation of the boss. The options include:

- Normal** The boss is perpendicular to the face normal.
- X-Axis** The boss axis is aligned with the global x axis.
- Y-Axis** The boss axis is aligned with the global y axis.
- Z-Axis** The boss axis is aligned with the global z axis.
- 2-Pts** The boss axis is aligned with user supplied two points.

Creating a boss feature

- 1 Select the boss orientation of “Normal”.
- 2 Pick the face for the boss.
- 3 Specify location for boss center.
- 4 Adjust the boss height, diameter, radius, and draft angles using the Data Entry Window.



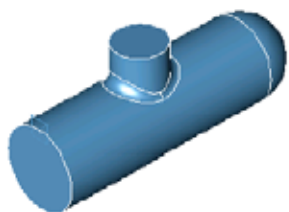
Boss with Draft and Fillet Radius

Shell

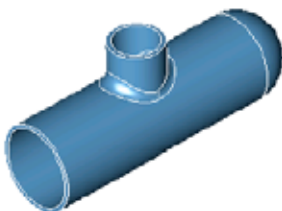
The Shell tool creates a parametric feature that is frequently used for cast and molded parts. Material from the part interior is removed leaving a hollow cavity. The Shell tool allows you to select faces to leave open; if no faces are selected, the part is hollowed inside. Use the Option key to keep the material removed (Keep Core).

Using the Shell tool

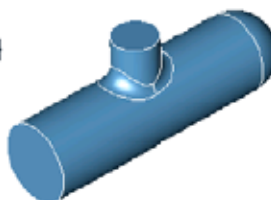
- 1 Select the part to shell.
- 2 Hold the Shift key and select open faces (click outside the part to hollow entire part).
- 3 Adjust the shell thickness using the Data Entry Window.



Before



*Shelled
Shell*



Core

Use the Inspector to set thickness values to individual faces.

Bend

The Bend tool will bend a solid about an axis. There are two Bend tool options in the Message Line:

Bend Axis

Bends a solid using the axis as the initial bend plane.



Center Bend

Bends a solid using a center bend.



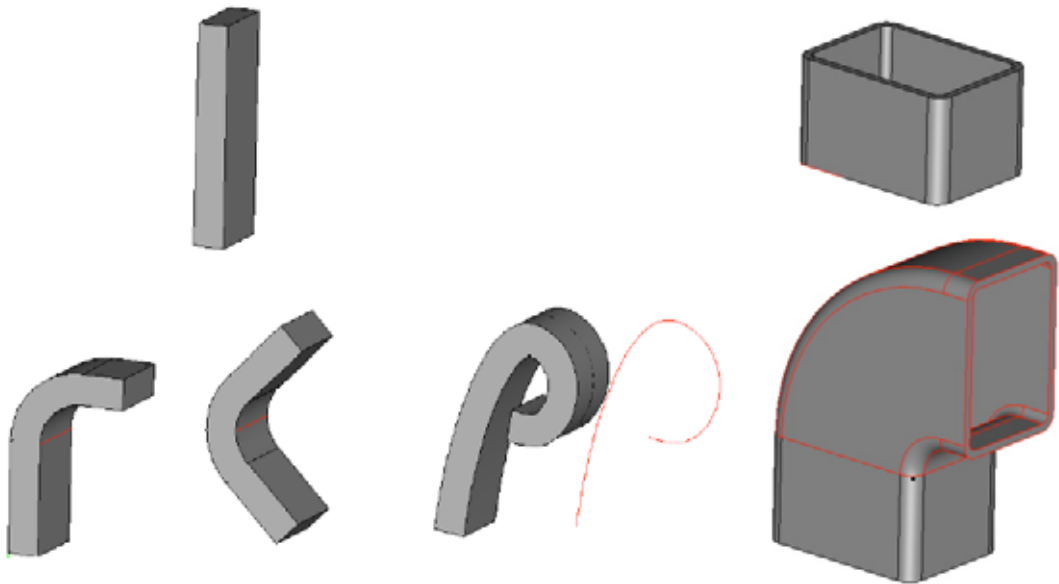
Bend Along Curve

Bends a solid along a curve.



Extend Bend

Lengthens a solid by the specified amount and introduces a bend at given angle and radius.



Bend Axis

Center Bend

Bend Along Curve

Extend Bend

Bend Tool Methods

Using the Bend Tool

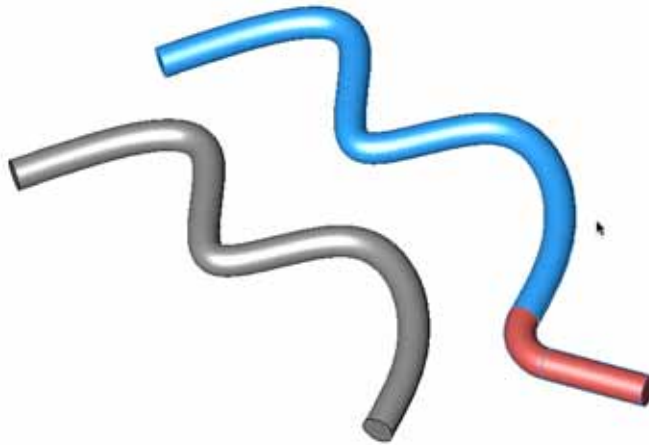
- 1 Pick the bend option you want.
- 2 Select the part to bend.
- 3 Specify two points for the bend axis.
- 4 Adjust the bend radius and angle using the Data Entry Window.



Bend Axis

Using the Bend Tool

- 1 Pick the extend bend solid icon (last icon in the sub-toolset).
- 2 Select the face to extend.
- 3 Click an edge to specify to direction in which you want the extension.
- 4 Adjust the bend radius and angle using the Data Entry Window.



Extend Bend

Edge Treatments

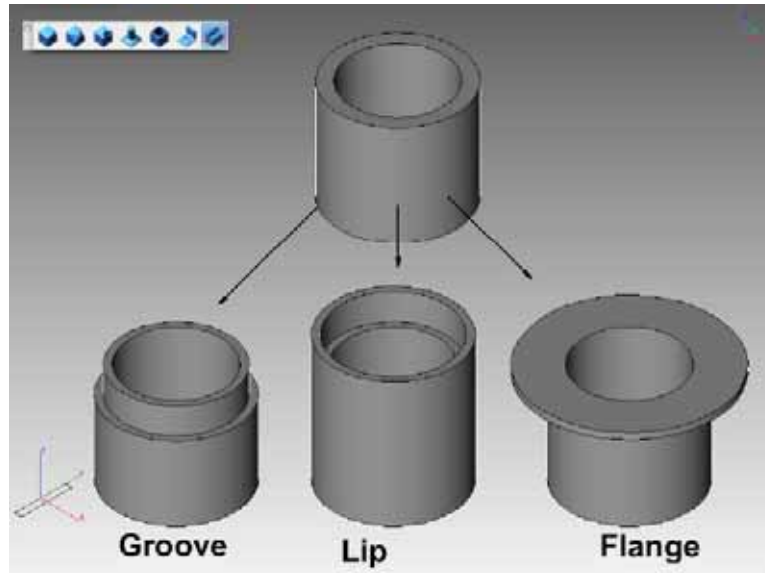
Edge treatments add or remove material based on user-supplied parameters and are associated with the edge of a part. There are three edge treatments supported:

- Lip
- Groove

■ Flange



The parameters or properties, of the resulting geometry are controlled on the Prompt window.



Groove

The Groove tool removes material below the selected face on the selected edge using the supplied height, width, and angle.

Using the Groove tool

- 1 Select the face for the groove.
- 2 Click an edge for the removed groove.



 **Lip**

The Lip tool adds material above the selected face, on the selected edge, using the height, width, and angle values.

Using the Lip tool

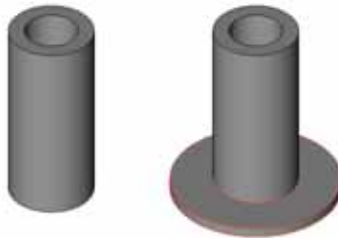
- 1 Select the face where you want the lip to appear.
- 2 Click the edge you want to remain. The other edge is repositioned to create the lip.

 **Flange**

The flange tool creates a support structure on a given part. Parameters for a flange are offset and thickness.

Using the Flange tool

- 1 Select the face for the flange.
- 2 Click an edge in the direction for the flange.

**Feature Editing**

One of the more powerful features of Punch! Shark FX is the transparent parent/child associativity for solid objects. As you create and add features to your part, Punch! Shark FX automatically establishes relationships between the parts. When you modify or change a parent the children all get regenerated. For example, consider the following history of a part:

- 1 Create a block primitive.

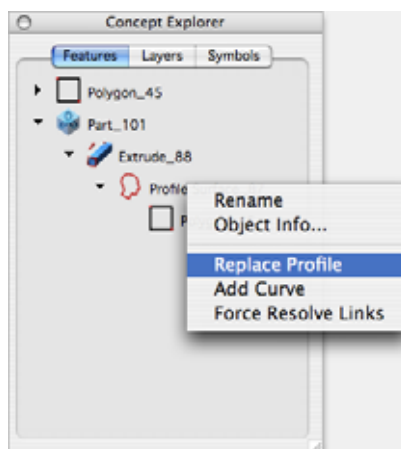
- 2 Fillet the edges of the block.
- 3 Create a boss at the block center.
- 4 Create a countersink hole at the boss top center.

Now try these modifications on the part:

- 5 Select any face or edge associated with the original block from step 1. Object Info the block parameters and change the length, width, and height. Note all steps 2-4 automatically regenerate based on the new l, w, h.
- 6 Select any face or edge introduced by the fillet command from step 2. Change the radius value, and note that steps 3 and 4 automatically regenerate.

Profile Updates

Feature based solids are created from collections of 2D curves called profiles. The Concept Explorer displays a Profile Surface entity that contains a list of curves defining a closed profile. Right-clicking on the Profile Surface entity displays a menu that provides options to replace the entire profile or add a curve to the existing profile.



Replace Profile

This option prompts you to select a new set of curves that will redefine the profile.

Add Curve

This option prompts you to select a curve in the plan of the profile to add as part of the profile.

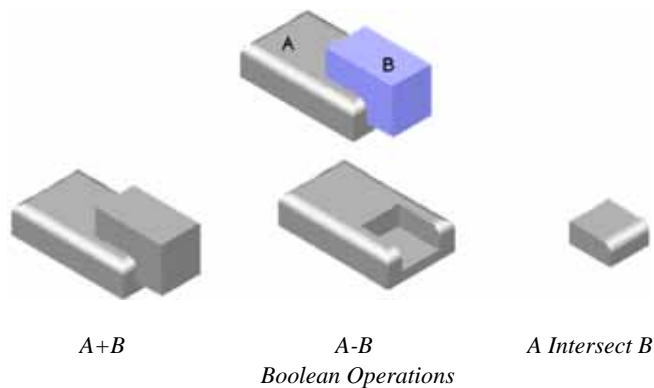
Remove/Replace Curve

Selecting a curve that is listed under a profile will display a menu that allows you to remove or replace the curve.

Solid Utilities



Boolean operations provide the designer with tools for adding, removing, and intersecting solids with other solids. The figure below illustrates the three Boolean operations on two solids.

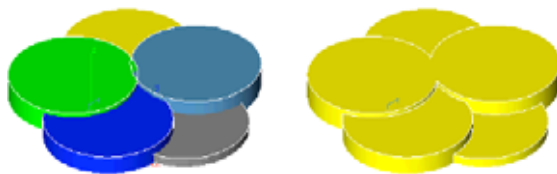


Union

The Union Boolean (also called add) operation is used to combine two shapes into one. You can add more than one solid to the base solid (first solid selected).

Using the Union tool

- 1 Select the part to add other parts to.
- 2 Select one or more parts to add to the previously selected part.



Boolean Union

In the above example, the first solid selected was yellow. The other solids were selected in step 2. The color and material attributes from the first solid are preserved on the resulting part.

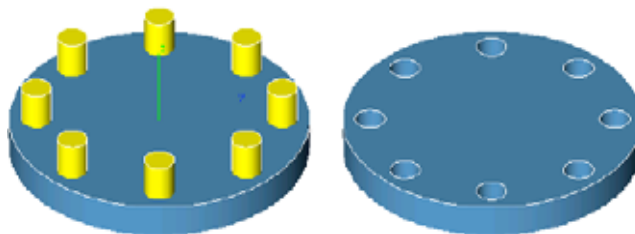


Subtract

The Subtract (also called difference) operation is used to subtract material from a solid.

Using the Subtract tool

- 1 Select the part to subtract other parts from.
- 2 Select one or more parts to subtract from the previously selected part.



Boolean Subtract



Intersect

The Intersection operation creates a shape equal to their common volume.

Using the Intersect tool

- 1 Create a lens using two spheres and the Intersect tool.
- 2 Select the first part for the Intersect tool. (New part has the color/materials of the first part.)
- 3 Select the second part for the Intersect tool.

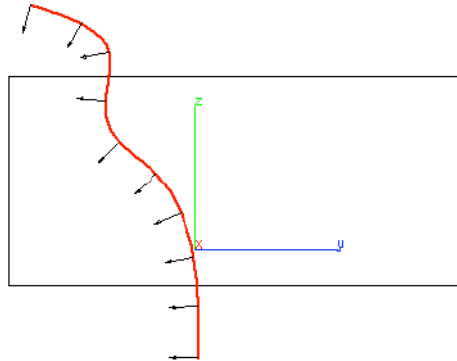
*Boolean Intersect*

Trim

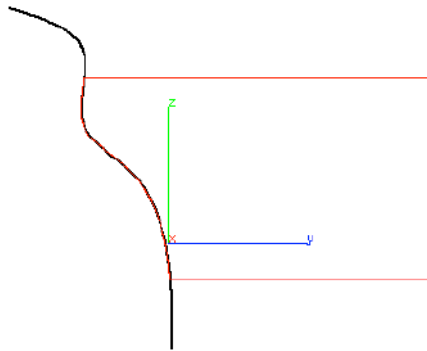
The Trim Solid tool is used to remove unwanted portions of a solid using a curve or surface. When you pick a curve to remove away portions of a solid, the curve is extended through the screen. Arrows are drawn that indicate the side of material that is to be removed. Use the Option key to flip the direction. You may also pick a surface to trim a solid.

Using the Trim tool

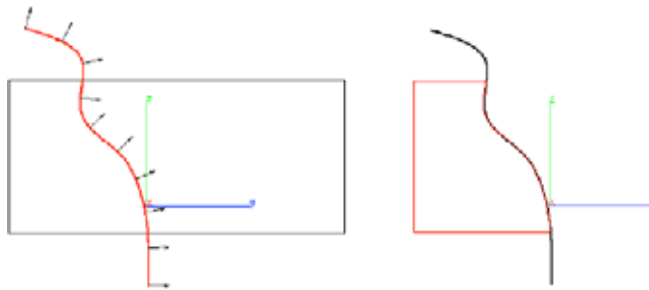
- 1 Pick a curve or surface to trim solid with.



- 2 Pick the solid that you want to trim. The trim operation removes the material in the direction of the arrows.



- 3 Hit the Option key to toggle the trim to the opposite side.

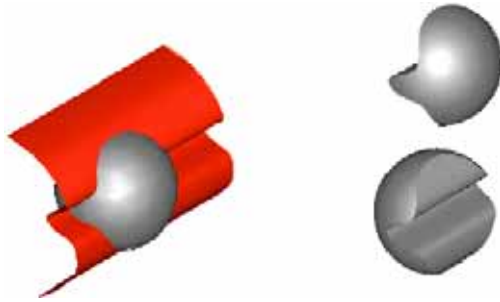


Split

The Split Solid command divides a solid into two components. You split a solid by first creating an infinite plane surface from the Surface command. Select the plane and the solid to split into two components. The Split Solid command is useful for displaying solid sections in drawings and for creating large stereolithography models.

Using the Split tool

- 1 Pick the solid to split.
- 2 Pick the surface or solid to split the first part.



Split Solid with a Surface (and moved)



Stitch

The Stitch tool provides a way to stitch, or sew, separate surfaces together to make a solid body that is topologically complete. Internally, stitching essentially involves replacing coincident groups of edges or vertices in the data with a single edge or vertex.

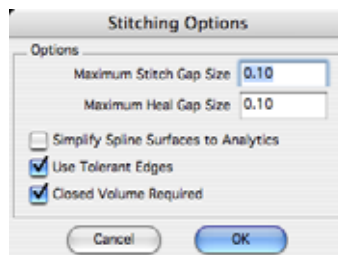
Using the Stitch tool with Surfaces

- 1 Pick one or more surfaces to stitch into a solid.
- 2 Press the Option key for advanced stitching options.

The Stitch tool also allows the selection of a solid. In this case the repair features of the Stitch tool will examine the part and repair any potential problems. In this case, the repair operation is added as a feature operation and repeats the repair operation on regeneration.

Stitch Options

The Stitch command displays a dialog box with advanced settings associated with stitch and healing surfaces.



Stitch Dialog Options

Maximum Stitch Gap Size

The maximum distance between two shared edges.

Maximum Heal Gap Size

The maximum distance the software will attempt to heal two edges. Healing geometry typically involves changing the geometry slightly at the shared vertex or edge.

Simplify Spline Surfaces to Analytics

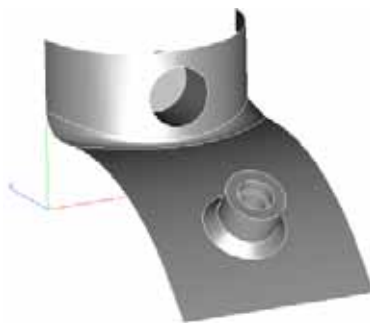
Examines the surfaces for possible conversion from NURBS to analytic.

Use Tolerant Edges

Instead of healing the two surfaces, just assign a tolerant edge.

Closed Volume Required

By unchecking the “Closed Volume Required” option, the stitcher will not require that the body be watertight. This command allows you to treat collections of surfaces as if they were closed. This then provides a means to do what were previously solid modeling operations on surfaces (sometimes called hybrid modeling). For example variable radius blends, chamfers, holes, and cutouts can be applied to surfaces as in the illustration below.



Feature Operations On Surfaces



Thicken

The Thicken tool will thicken a surface into a solid. You can also use the Thicken tool to thicken a specific face or the entire solid. Negative values are acceptable for thickness values. In the case of a surface, a negative value thickens in the opposite surface direction of the surface normals. For solids a negative value will remove volume.

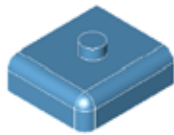
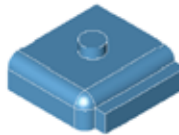
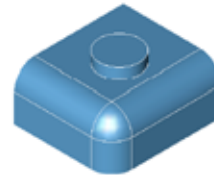
Using the Thicken tool on Surface

- 1 Select a surface to thicken. (Polysurfaces are acceptable as in below example.)
- 2 Use the Data Entry Fields to adjust the thickness value.

*Thicken Surfaces*

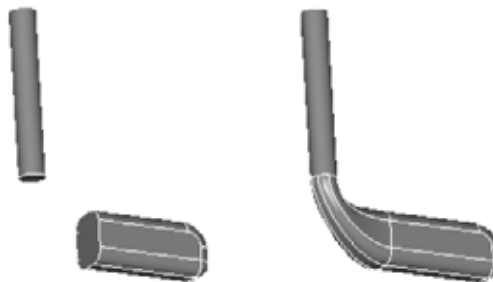
Using the Thicken tool on Solids

- 1 Select a face or body to thicken.
- 2 Use the Data Entry Fields to adjust the thickness value.

*Original Solid**Single Face
Thicken**Entire Body
Thicken**Thicken Face or Body*

Lofted Faces

The Lofted Faces tool creates a solid that is tangent between two neighboring solids. The resulting solid automatically joins the two bodies involved into one new body.

*Loft Solid*

Using the Lofted Faces tool

- 1 Select the two faces to loft between.



Loft Solid Face Selection

- 2 Use the Data Entry fields to change the bulge factor.

Once you've created a lofted solid, you can go back and modify the bulge factors using the Inspector. In the image below, object (b) uses two bulge factors, 5 and 5. Object (c) uses 20 for the first and 5 for the second. Object (a) uses 11 and 11 for each face. Be careful not to apply a bulge too high that will result in a self-intersecting body. Also beware of using too small of bulge factors which will prevent later features operations, such as shelling and blending.



(a) 11,11

(b) 5,5

(c) 20, 5

Bulge Factors for Face1 & Face2



Lofted Faces Guides

This tool allows you to specify guide curves for the Lofted Solid tool.

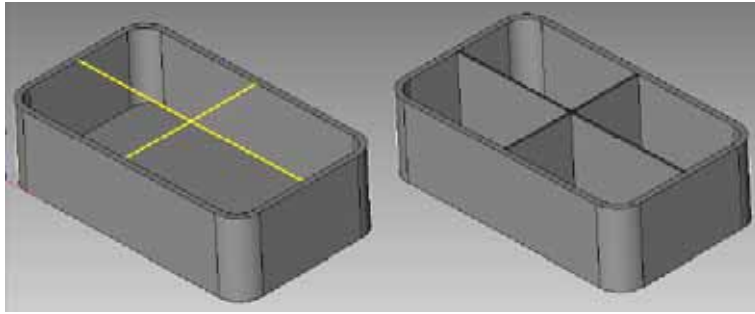


Rib

The Rib tool is used for the construction of a support feature. It includes parameters for thickness and draft angle, which are set on the Prompt window.

Using the Rib tool

- 1 Select the solid where you want a rib to appear.
- 2 Click the 2D profile that you want the rib to follow.
- 3 Click two points to define the direction of the rib.



Local Face Operations



Draft

The Draft Face operation will rotate a collection of faces about an edge. The Draft Face tool will extend or relimit adjacent faces. The following terms are used when creating a drafted face.

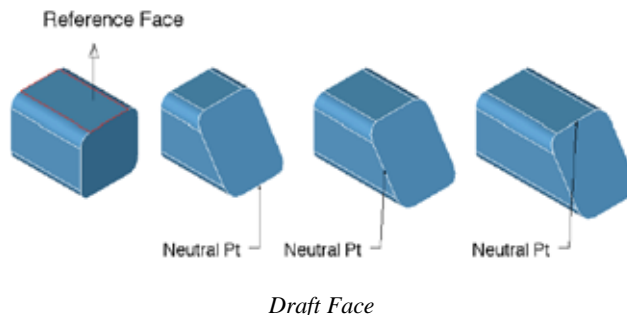
Reference Face A face whose normal defines the draft angle. Sometimes this is referred to as the pull direction for molds.

Reference Edge A linear edge whose direction defines the draft angle.

Neutral Position A face whose normal defines the pull direction. Since this position doesn't change, it is called the neutral position.

Using the Draft tool

- 1 Select the reference face for the draft angle.
- 2 Select the faces to draft. Use Shift key for multiple faces.
- 3 Specify the neutral point.
- 4 Use the Data Entry Fields to change the draft angle.



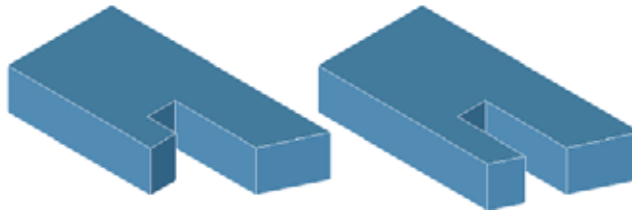


Match Face

The Match Face command will modify the orientation and position of a selected face to that of a referenced face.

Using the Match Face tool (To Face)

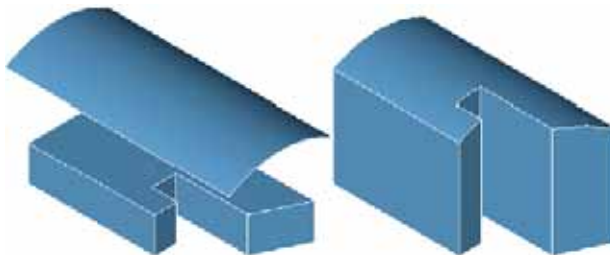
- 1 Select face to modify.
- 2 Select the face to match.
- 3 There are no Data Entry Fields entries for this command.



Match Face To Face

Using the Match Face tool (To Surface)

- 1 Select face to modify.
- 2 Select the surface to match.
- 3 There are no Data Entry Fields entries for this command.



Match Face To Surface



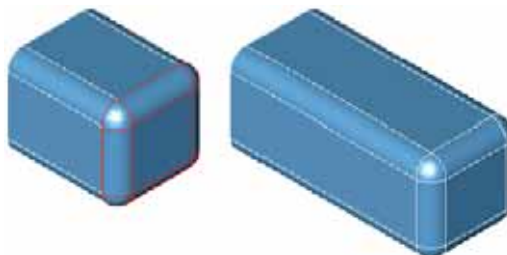
Move Face

The Move Face tool will translate one or more faces by a vector amount. Neighboring faces are extended or relimited to account for the new position.

Using the Move Face tool

- 1 Select one or more faces to move.

- 2 Specify two points for the move direction and magnitude.
- 3 Use the Data Entry Fields to adjust the dx, dy, and dz components.

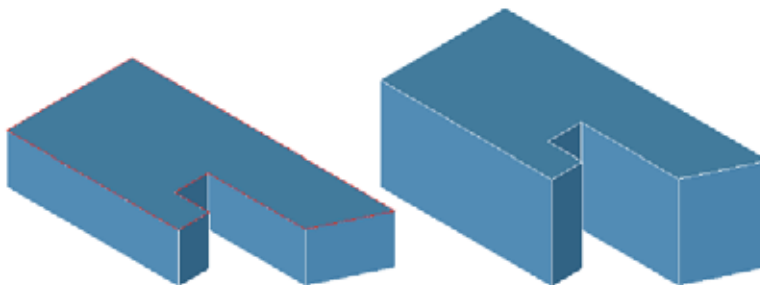
*Move Faces*

Offset Face

The Offset Face tool will offset one or more faces a positive or negative distance. Neighboring faces are relimited to the new offset face.

Using the Offset tool

- 1 Use the Data Entry Fields to specify an offset value.
- 2 Select one or more faces to offset. In the figure below, top face in red is offset upwards.

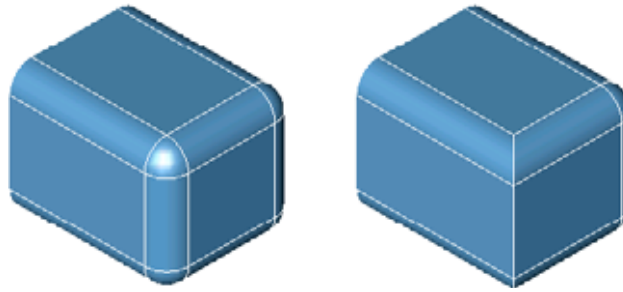
*Offset Face*

Remove Face

The Remove Face tool will delete one or more faces from a part. More than one face can be removed in one operation. Internally, neighboring faces are extended and relimited to account for the removed faces.

Using the Remove Face tool

- 1 Select faces to remove. Hold the Shift key for multiple faces as in example below.
- 2 There are no Data Entry Fields entries for this tool.



Remove Face



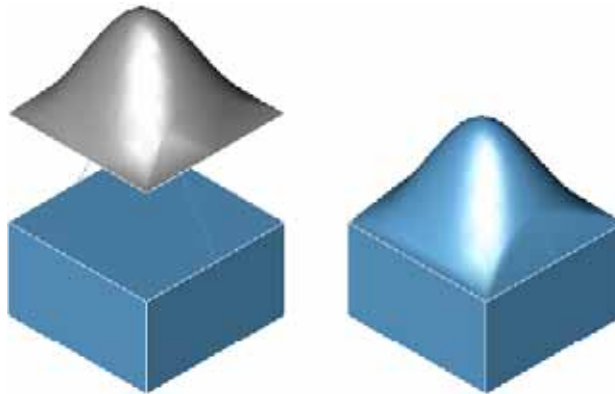
Replace Face

The Replace Face tool will substitute a new surface for the selected face. The surface must lie precisely along the edges of the face to replace.

Using the Replace tool

- 1 Select the face to replace.
- 2 Select the surface to replace with.

The example below shows the surface translated up for clarity. The surface must lie on the face edges that are being replaced.



Replace Face

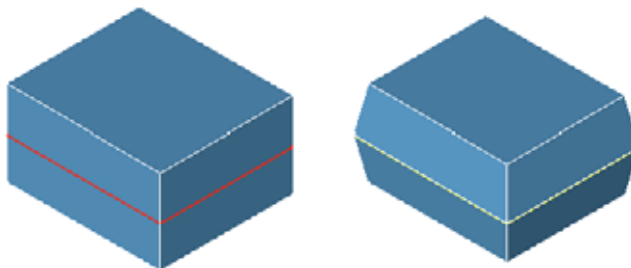


Parting Line

The Parting Line tool will take a collection of curves that lie on a solid, imprint them onto the solid (creates split faces), and add draft to the upper and lower surfaces relative to the parting line.

Using the Parting Line tool

- 1 Select the part for the parting line operation.
- 2 Use the Data Entry fields to specify upper and lower draft.
- 3 Select the curves that lie on the solid for the parting line.



Parting Line

Deform Face

The Deform Face tool is used to inflate or deflate a face to an optional point or curve constraint. The deform tool has three creation methods.



Pressure Gain Inflates or deflates based on a pressure gain. Useful for doming faces.

To Point Inflates or deflates a face to a point.

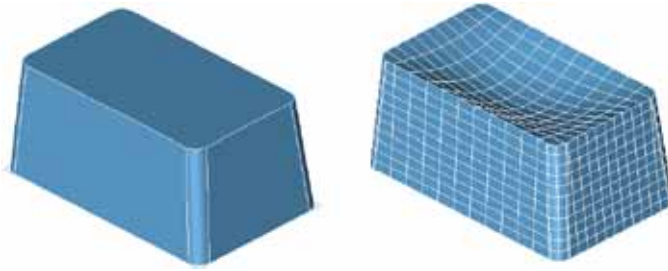
To Curve Inflates or deflates to a curve.



Using the Deform tool (Gain)

- 1 Select the first icon which deforms to a pressure gain.
- 2 Select the face to deform.
- 3 Use the Data Entry Fields to adjust the gain. Negative gains (as in example below) will move the surface in a direction opposite of its surface normals.

Iso lines are displayed in the figure to the right to better visualize the deform.

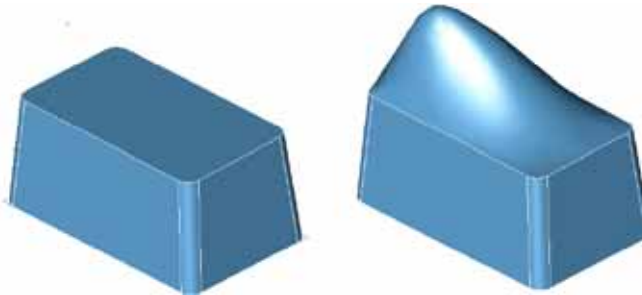


Deform Face to Gain



Using the Deform tool (To Point)

- 1 Select the second icon which deforms to a point.
- 2 Select the face to deform.
- 3 Specify two points that define a from to vector. (A point entity is created that you can later modify.)
- 4 Use the Data Entry Fields to adjust the surface gain.



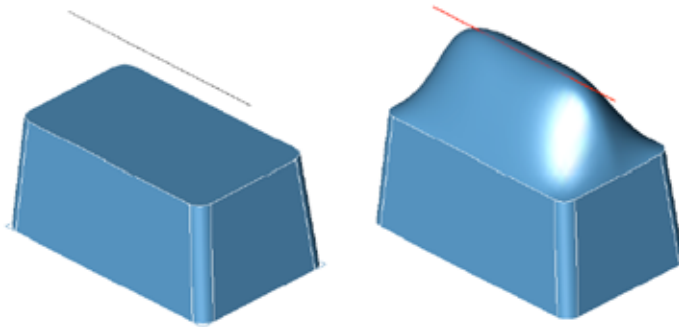
Deform Face to Point



Using the Deform tool (To Curve)

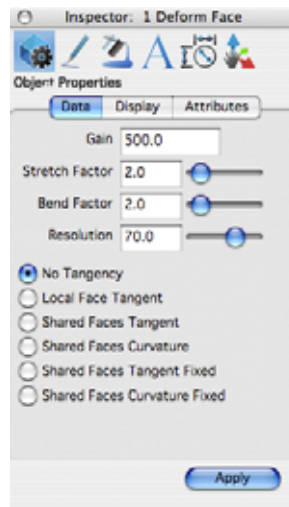
- 1 Select the third icon which deforms to a curve.
- 2 Select the face to deform.
- 3 Select the curve to deform to.

The curve must project completely onto the face to deform.

*Deform Face to Curve*

Deform Options

The Deform feature has a variety of options for controlling how the face is deformed. These are accessed through the Inspector.

*Deform Face Properties*

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 tool icon. It does not apply to deformations to a point or curve. Be sure to use higher resolutions when using the below options to insure tangency and curvature precision with the results.

No Tangency

The No Tangency option deforms the face and allows the shape at the edges to deform.

Local Face Tangent

The Local Face Tangent option deforms the shape but preserves the existing tangencies of the face.

Shared Faces Tangent

Shared Faces Tangent will modify all faces that share edges with the face being modified. The modification will impose tangencies at all shared edges.

Shared Faces Curvature

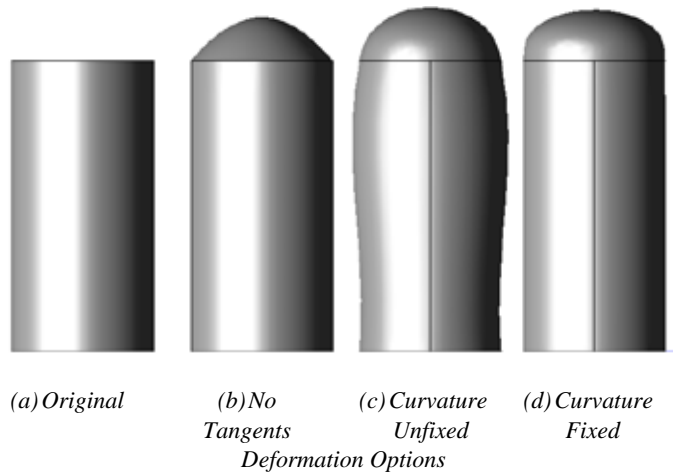
The Shared Faces Curvature option will modify all faces that share edges with the face being modified. The modification will impose curvature continuity at all shared edges.

Shared Faces Tangent Fixed

The Shared Faces Tangent Fixed option will only modify the selected face to be tangent to all faces that share an edge with the selected face.

Shared Faces Curvature Fixed

The Shared Faces Curvature Fixed option will only modify the selected face to be curvature continuous to all faces that share an edge with the selected face.



Pattern

Solid modeling often involves the repetition of features or objects arranged in a regular or irregular manner (copied or transformed), which may be referred to as patterns. Examples of patterns include the radial arrangement of holes in a shower head, the linear grating of ventilation holes on a computer monitor or the treads on a tire. Creating such patterns can become unnecessarily burdensome, especially when the number of repetitive elements grows large.

The Pattern tool reduces this burden by offering pattern types to facilitate pattern creation. The following eight patterns are available on the Data Entry window, once the tool is activated:

- Linear
- Polar
- Polar Grid
- Radial Grid
- Cylindrical
- Spherical
- Along Path
- Copy to Locations

Linear

Creates a linear pattern specified by quantities along the x, y, and z axis. Spacing is controlled by the dX, dY, and dZ filed on the Data Entry window. Use the Hex option to stagger the rows to create a hexagon pattern.

Polar

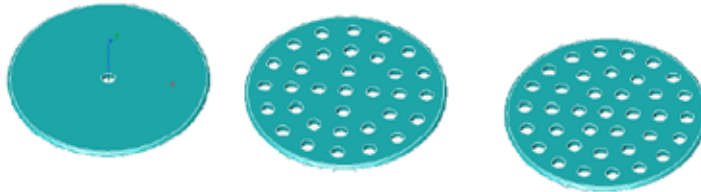
Creates a polar array pattern specified by the number of angles.



(a) Original (b) Polar Array
Polar Array Feature

Polar Grid

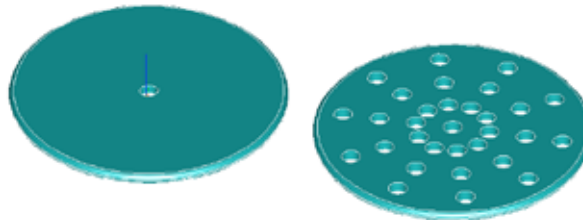
Creates a polar grid pattern specified by the number of angles.



(a) Original (b) Polar Grid (c) Polar Grid Hexagon
Polar Grid Feature

Radial

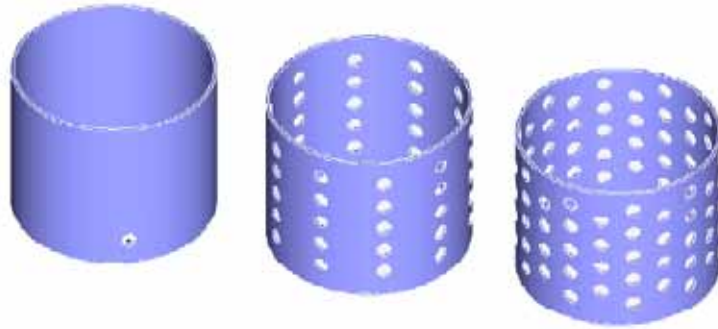
Creates a radial pattern specified by the number of angles.



(a) Original (b) Radial
Radial Feature

Cylindrical

Creates a cylindrical pattern specified by the number of angles.



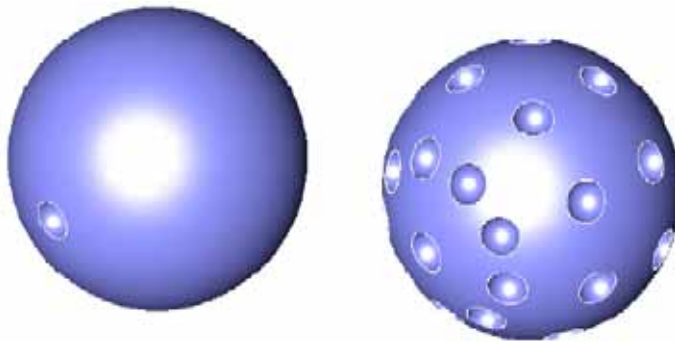
(a) Original

(b) Cylindrical
Cylindrical Feature

(c) Cylindrical Hexagon

Spherical

Creates a spherical pattern specified by the number of angles.

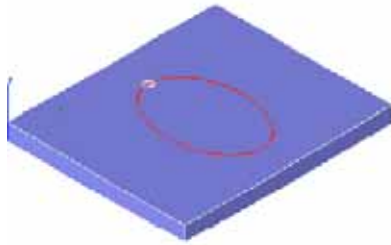


(a) Original

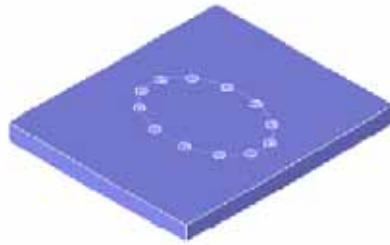
(b) Spherical
Spherical Feature

Along Path

Creates a pattern along a curve path.



(a) Original

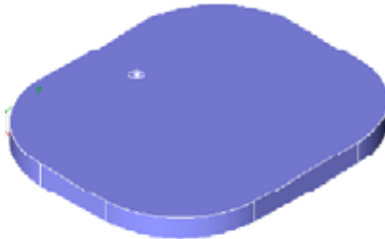


(b) Along Path

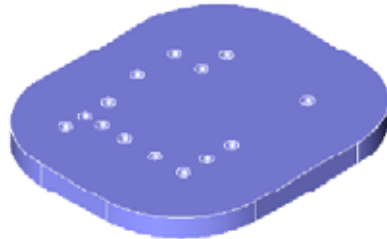
Along Path Feature

Copy to Locations

Creates a pattern by specifying locations.



(a) Original



(b) To Locations

To Locations Feature

Part 6

Drawing Composition

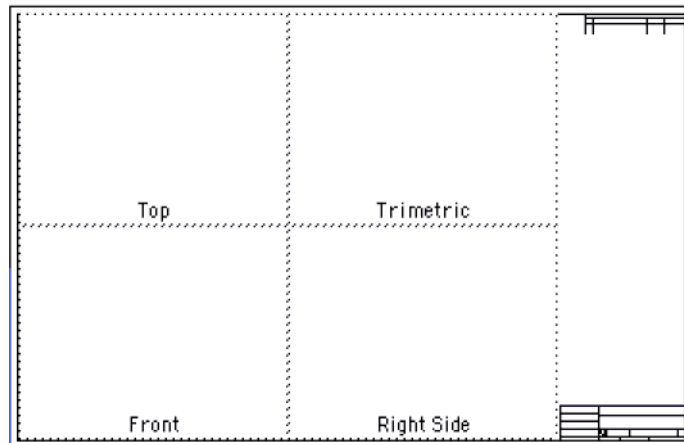
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Model To Sheet

Punch! Shark FX allows you to quickly create 2D drawings from one or more curves, surfaces, and solids. The 2D drawing views are bi-directionally associative. Changes made to objects used in the Model To Sheet tool automatically update the 2D drawings. Likewise, you can use the Inspector to change parameters of the objects in the 2D drawing, and have the part automatically update. Punch! Shark FX allows automatic creation of 2D drawings with the Model To Sheet tools located in the Main Tool Palette Menu.

Model To Sheet Overview

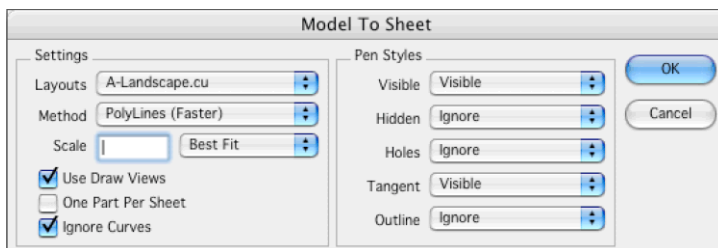
The Model To Sheet tool automates the process of creating drawings by the use of predefined templates. Templates are empty drawings with prearranged drawing views that are embedded within a variety of drawing borders and page formats. For example, in the Layouts folder you can find over 40 templates ranging from A-E and A0-A4 drawing sizes.



Example Template

Model To Sheet Dialog

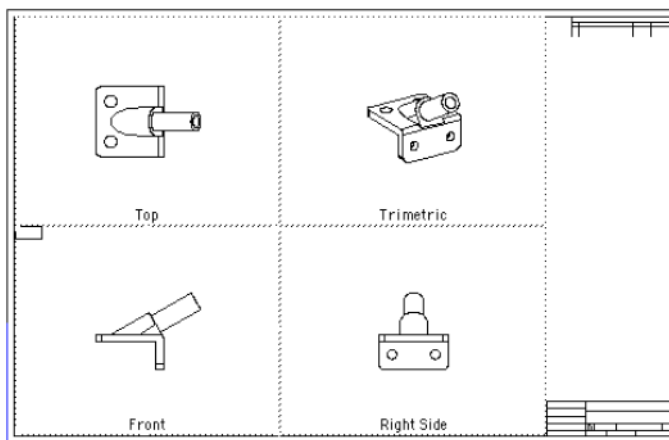
The first step in creating a drawing is to select the objects you wish to make a drawing out of using the Arrow tool. Once the objects are selected, you can choose the Model To Sheet tool located in the Menu Bar under the Layouts option. The following dialog box will appear.



Model To Sheet Dialog

Layouts

Pressing the drop-down next to the Layouts option will provide a list of all predefined Model To Sheet templates located in the Layouts folder. The Layouts folder is found in the Install folder. All Punch! Shark FX files in this folder are listed in the pull down menu. You may create, modify or customize any of the templates, based upon your drawing file format preferences.



Model To Sheet Example

Method

The Model To Sheet methods determine the type of data created in each of the draw views associated with the template. The five types of methods are:

Precise

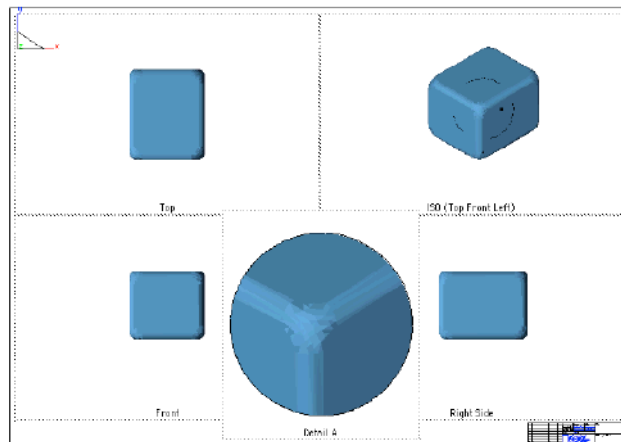
Precisely calculates the visibility of edges in a draw view. This method calculates visibility between edges and either the NURBS or analytic definition of the body. Resultant edges are lines, arcs, circles, ellipses, and splines. Precise also accurately calculates the visibility of silhouette edges.

Polyline

The Polyline method quickly calculates the visible and hidden edges by simplifying the edges into polylines and comparing its visibility with a facet representation of the body. These edges are less precise and difficult to dimension. Polylines calculate the visibility of silhouette edges as polyline segments.

Flat

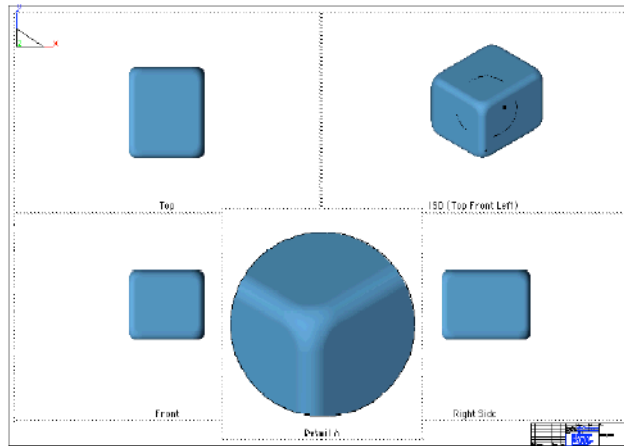
The image that appears in the draw view is generated as an OpenGL shaded image using a Flat lighting model. Because there is no vertex smoothing, you will see facet boundaries.



Flat Draw Views

Gouraud

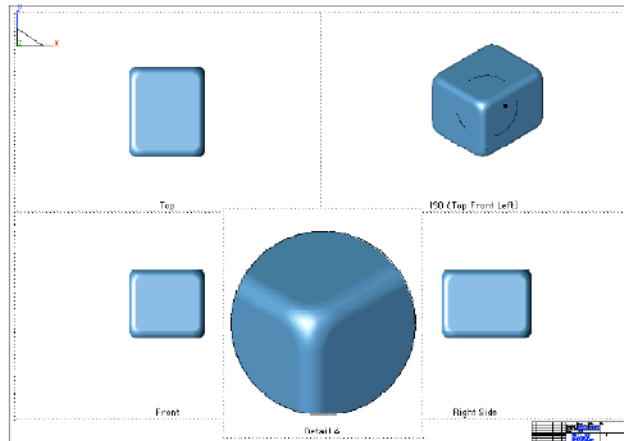
The image that appears in the draw view is generated as an OpenGL shaded image using a Gouraud lighting model. The Gouraud lighting model interpolates color across each facet and produces a smoother image than Flat.



Gouraud Draw Views

Phong

The image that appears in the draw view is generated as an OpenGL shaded image using a Phong lighting model. The Phong lighting model interpolates the vertex normal across each facet and produces a smoother image than Gouraud.



Phong Draw Views

Draw views, section views, and detail views can use the OpenGL methods.

Scale

This field sets the size relationship between the draw view and the sheet view. The Best Fit option will calculate a scale that fits the drawing view size.

Use Draw Views

When checked, Draw Views are created from the Model To Sheet command. Otherwise geometry is placed directly on the sheet. Draw views are required for changing the view orientation or scale as well as generating section, detail, and auxiliary views.

One Part Per Sheet

Each solid selected for Model To Sheet is placed in a separate sheet. For example, four parts will create four separate sheets. Curves and surfaces are combined into a dedicated sheet.

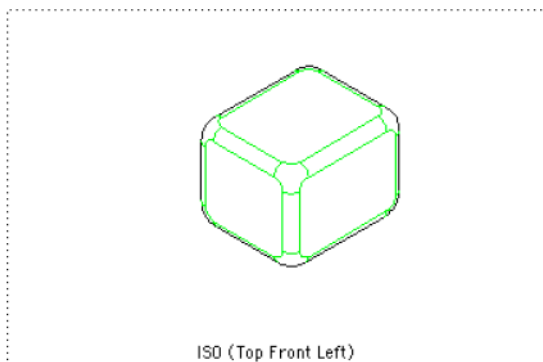
Ignore Curves

Curves such as lines, circles, arcs, and splines are ignored for the Model To Sheet operation.

Pen Styles

The Pen Styles option defines the line font format used by hidden and visible edges. The five options are:

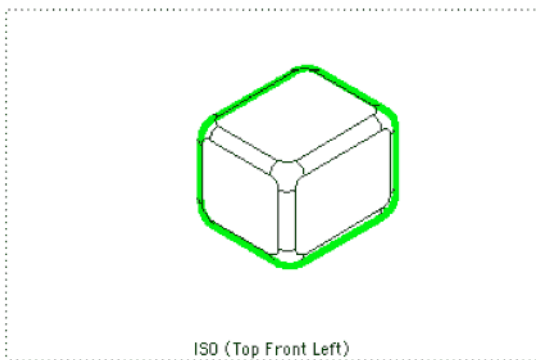
- Visible** Attaches a predefined or user-defined pen style to visible edges. By default, the Visible pen style is used with the following attributes:
- | | |
|---------|-------|
| Color | Black |
| Weight | 0.02" |
| Pattern | Solid |
- Hidden** Attaches a predefined or user-defined pen style to hidden edges. By default, the Hidden pen style is set to Ignore. Ignore tells the draw view NOT to display hidden line edges.
- Holes** Attaches a predefined or user-defined pen style to hole edges. A hole is defined as a hidden cylindrical surface. By default, the Hole pen style is set to Ignore. Ignore tells the draw view NOT to display hole edges.
- Tangent** Attaches a predefined or user-defined pen style to tangent edges. A tangent edge is defined by two surfaces that share an edge with G1/G2 continuity. By default, the Tangent pen style is set to Visible.



Tangent Edges In Pen Style

Outline

Attaches a predefined or user-defined pen style to outline edges. An outline edge is object profile. By default, the Outline pen style is set to Ignore.



Outline Edges In Pen Style

Activating Drawing Views

Once you've created a drawing, you can activate individual views by clicking a rectangular region surrounding the view. Once activated, the view boundaries will highlight in a red dashed border. Clicking outside will deactivate the drawing view.

When activated, all geometry in the view becomes activated. The Snap tool will now recognize all snap points in the activated view, facilitating the creation of new entities, such as dimensions. All entities created in an active view are viewed only in the view in which they were created.

Since the geometry created in the drawing views are true 2D wireframe, you may assign additional line fonts or layers to each entity.

Drawing View Menu

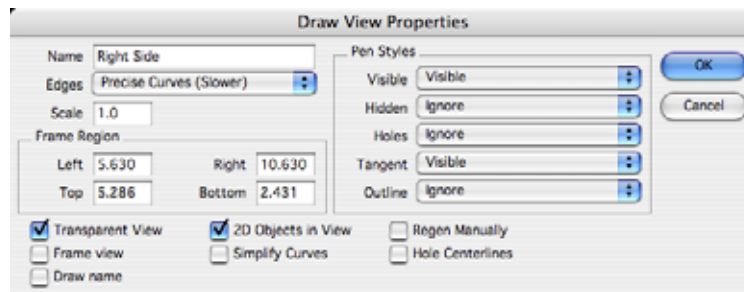
Drawing views have a variety of special commands accessible via a pull down menu in the drawing view. You can access these properties by activating the view and clicking the mouse in the upper left region of the window. You may also get this dialog box by clicking the right button of your mouse. On the Mac you may click the button in conjunction with the Control key.



Draw View Options

Properties

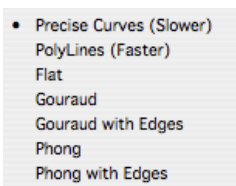
Each drawing view has a set of properties and attributes that impact the draw view. These properties adjust the hidden line method, name, scale, pen styles, and other attributes described below.



Properties Dialog Box

Name The Name option allows you to specify the name that appears when the draw name checkbox is activated.

Edges The Edges option defines the method in which hidden line edges are calculated and displayed in the view.



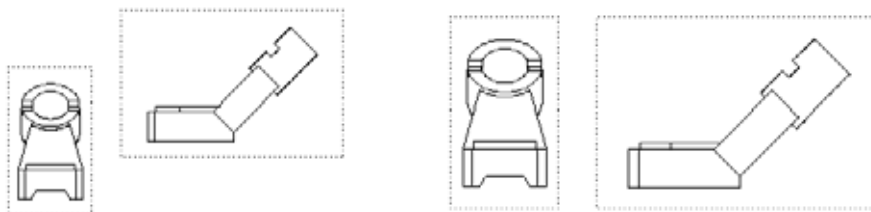
Scale	The Scale defines the size relationship between the drawing view and sheet view. A sheet view is the window that contains draw views.
Frame Region	The Frame Region defines the rectangular region on the sheet view where the drawing view is located.
Transparent View	Erases the background of the sheet view before drawing the draw view.
Frame View	Draws a border around the draw view.
Draw Name	Displays the draw view name centered at the lower border of the frame.
2D Objects In View	Projects the hidden line data into 2D. Circles may get projected into ellipses, depending on the view.
Simplify Curves	Attempts to simplify curve edges from splines to lines, arcs or circles.
Regen Manually	When checked, the draw view is updated manually by the user. Use the Regen View tool icon in the Sheet tools to manually update a view.
Pen Styles	Changes the pen style setting for hidden and visible edges.
Hole Centerline	When checked, hole centerline dimensions are automatically added.

Delete

Removes the drawing view and its contents from the drawing file.

Align

The Align tool will align the active view with another selected view. You can use the Align tool on section views, general drawing views, and auxiliary views.



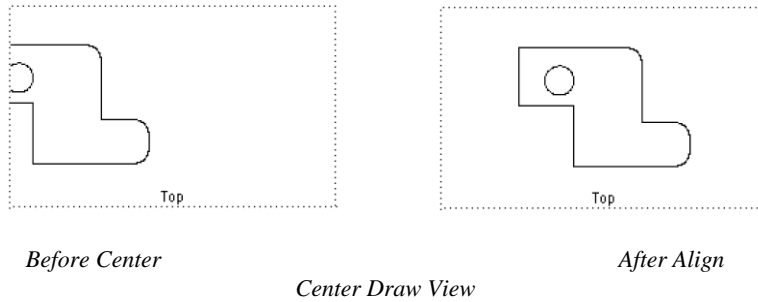
Before Align

Align Draw Views

After Align

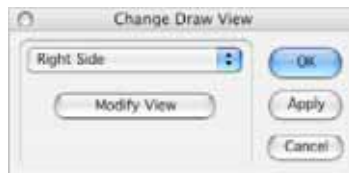
Center

Centers the objects in a draw view within the frame boundaries. The scale is preserved.



Change View

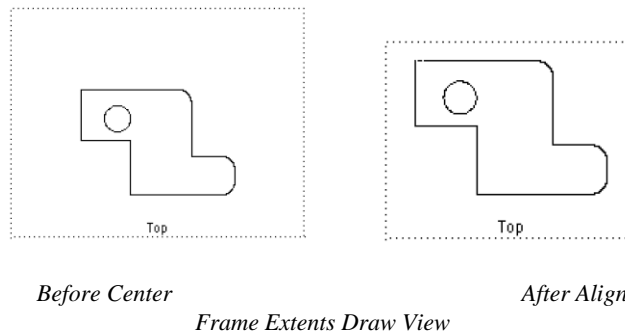
The Change View tool changes the current drawing view orientation to one of 20 predefined views or modifies the existing view by a variety of methods that include eye/ref point and rotation.



Change Draw View

Frame to Extents

Shrinks the frame to the extents of the geometry in the view. The scale is preserved.



Flatten

The Flatten option will delete the drawing view and place the objects within the view into the view containing the drawing view. The resulting objects will be scaled by the drawing view scale. This command destroys any associativity between the objects and the part that created them.

Zoom Extents

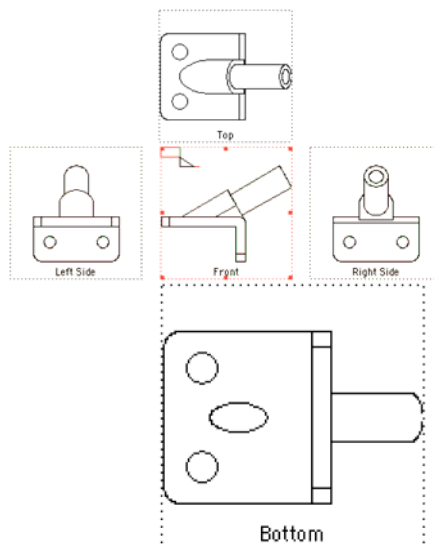
Zooms in on the geometry so that it fits to the extents of the frame. The scale is preserved.

Insert Reference File

Import a reference file into the view. For more information, see “Reference Files”, which begins on page 72..

Unfolding New Views

You can easily create new views from existing drawing views. First select the Arrow tool and activate the view you will unfold a new view from. Next, hold the Control key to indicate a copy action, then drag the view to the left, right, top or bottom. A view will be created that is the view rotated 90-degrees from the direction of the copy.



Pasting Into Views

Examines the contents of the paste buffer and inserts into the active view. The inserted object is associative to the original copied object. Use this in conjunction with the New Drawing view to make drawings manually.

Sheet Tools



Model To Sheet

This tool creates a Model To Sheet from the selected objects. For a full description of the Model To Sheet tool see the chapter titled “Model To Sheet”, which begins on page 419

Using The Model To Sheet Tool

- 1 Select curves, surfaces or solids for placement into a drawing sheet.
- 2 Set the appropriate options in the Model to Sheet dialog box.

The Model to Sheet tool automatically creates a new layer called Sheet #. Nested layers for the title block, dimensions, and drawing data are also created.

New Drawing View

Creates a new and empty drawing view. To add geometry to a new drawing view, select the geometry and copy it with the Edit:Copy tool. Then activate the view such that its border is red. Finally, use the Paste tool to add geometry to the view. The geometry in the view is associative to the original copy. Geometry created in an activated view is displayed only in that view.

Creating New Draw Views

- 1 Click the upper left corner position of the draw view.
- 2 Click the lower right corner position of the draw view.



New Draw View

To place objects (curves, surfaces, solids) into a draw view follow the directions below:

Adding Objects to a Draw View

- 1 Use the Arrow tool to select the objects to add.
- 2 Select the Edit:Copy command to copy into the paste buffer.
- 3 Click on the draw view to activate.
- 4 Select the Edit:Paste command to put the objects into the draw view.

The objects placed in the draw view are associative to the original parent objects.



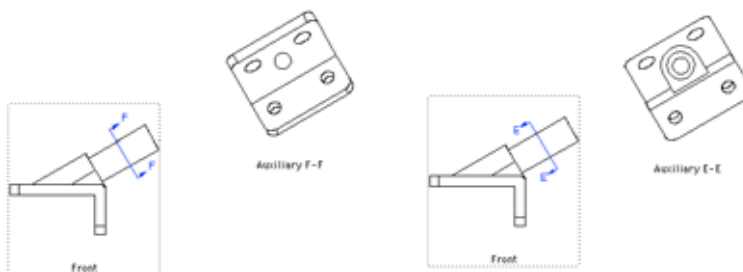
Auxiliary View

Creates an Auxiliary View from a referenced drawing view. Define the fold line with two points referenced from the parent view. Move the view and it will align automatically.

Creating an Auxiliary View

- 1 Click in the draw view to create an Auxiliary View from.
- 2 Click two points representing the fold line.
- 3 Move your cursor to locate the new Auxiliary View.

The direction of the fold line defines the line of sight. Changing the order of the two points will flip the direction. You can also edit the flip direction of the auxiliary view dimension in the info dialog box.



Auxiliary Views

Section Views

The Section View tools allow you to create horizontal, vertical, and a variety of other views through parts. The section dimension is associative to the section view. Moving the dimension will automatically update the section view. Section views are associative to the geometry that they are created from and the location of the section dimension. By moving the section dimension in the parent view, the section will automatically update.

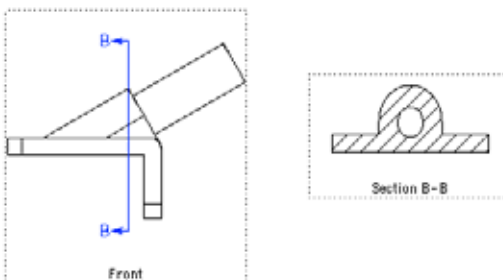


Vertical Section View

This creates a view, vertically, through a part.

Using the Vertical Section View tool

- 1 Click the Vertical Section View tool.
- 2 Click a start point to define the section position.
- 3 Drag your cursor to position the view.

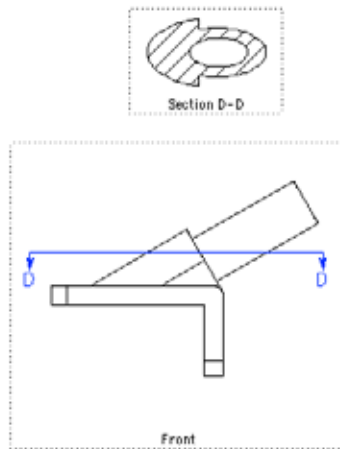


Horizontal Section View

This creates a view, horizontally, through a part.

Using the Horizontal Section View tool

- 1 Click the Horizontal Section View tool.
- 2 Click a start point to define the section position.
- 3 Drag your cursor to position the view.



Arbitrary Section View

This allows you to define a view through a part.

Using the Arbitrary Section View tool

- 1 Click the Arbitrary Section View tool.
- 2 Click a start point to define the section position.
- 3 Drag your cursor to position the view.

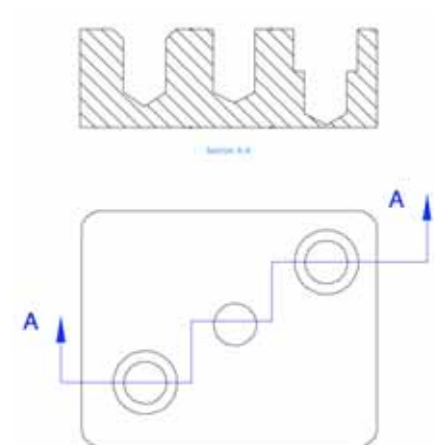


Offset

The Offset Section tool creates a sectional drawing view through a cutting plane, bent at right angles, to features as though they were in the same plane. A frequent use for an offset section is to provide a section view through a variety of hole centers.

Using the Offset Section View tool.

- 1 Select the Offset Section tool.
- 2 If a draw view is not active, click in the draw view that will contain the section annotation.
- 3 Enter locations for Offset corners. Right-click when completed.
- 4 Move the offset draw window to the desired location.

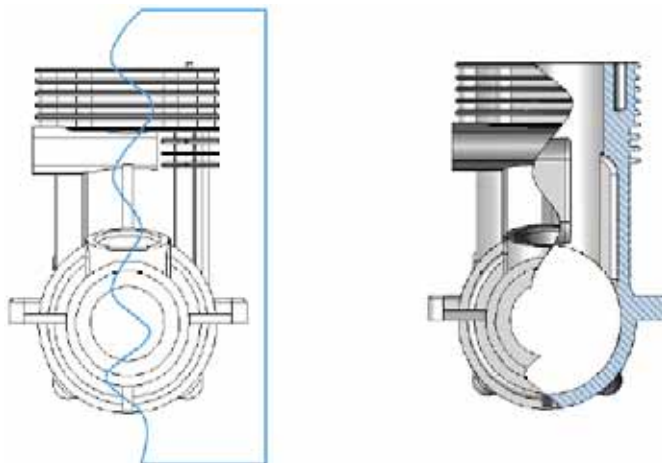


Break-away

The Break-away Section tool lets you define regions of a part you want to break away to display interior features of a part.

Using the Break-away Section View tool

- 1 Select the Break-away Section tool.
- 2 Select the closed curves to define the break away region.
- 3 Pick a draw view from which to define the depth to the break.
- 4 Pick a point to define the depth.
- 5 Drag the new section view to a location on the drawing.

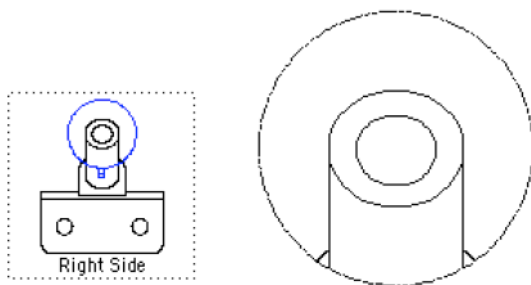


Detail View

Creates a detail view from a draw view. The detail dimension is associative to the detail view. Moving the detail dimension will automatically update the detail view. Detail dimensions are associative to the geometry in the parent view as well as the location and diameter of the detail dimension. Moving the circular dimension will automatically update the detail dimension.

Creating a Detail View

- 1 Click a point defining the detail center.
- 2 Click a point defining the detail radius.
- 3 Move your cursor to locate the new detail view.
- 4 Use the Data Entry Fields to change the detail scale.



Regen View

The Regen View tool is used to update all Draw Views that are out of date or in need of regeneration. A draw view becomes out of date if the geometry within the draw view changes. Normally draw views are updated automatically. However there is an option in the Draw View Properties dialog box for manually controlling regeneration of draw views. If you turn on Regen Manually, draw views will ONLY regenerate if you select the Regen View tool icon.

Part 7

Rendering Overview

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Rendering Overview

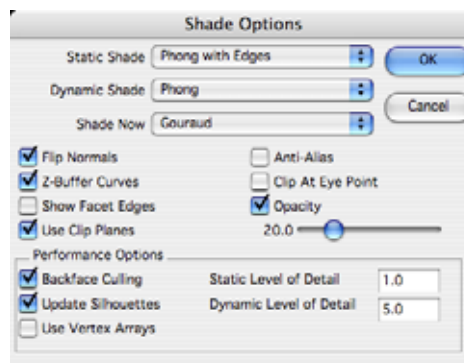
Rendering is the process of creating shaded images of 3D scenes. Punch! Shark FX supports two distinct types of rendering. The first method uses OpenGL for fast interactive manipulation. The second uses Rendering which utilizes slower raytracing methods but creates photorealistic images. This chapter provides a general overview of both OpenGL and Rendering.

OpenGL Shading

OpenGL shading uses software and hardware technology that is already preinstalled on most computers. OpenGL is designed for fast interactive manipulation of 3D geometry. OpenGL is an industry 3D standard that is a very popular rendering option for CAD software as well as the gaming communities.

For more information about OpenGL, visit the OpenGL consortium at <http://www.opengl.org>.

Punch! Shark FX uses OpenGL shading for static and dynamic repainting of the screen. Commands such as repaint, change view, zoom view, etc. all use OpenGL. There are various different options for shading in OpenGL that include wireframe, flat, Gouraud, and Phong plus combinations of showing or not showing associated surface or solid edges. To set the rendering method and a variety of other options, see the Shade Options dialog box under the View menu.



OpenGL Shade Option

The Static Shade mode applies to commands such as:

Repaint, Change View

Zoom All, Extents, Home

The Dynamic Shade mode applies to commands such as:

Wheel Mouse

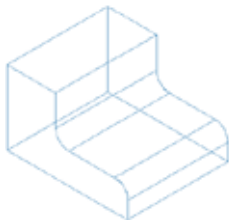
Dynamic Zoom, Pan, Rotate (in View Tool Palette)

The Shade Now option applies to a repaint operation associated with hitting the Shade Now command or a short cut key assigned to the Shade Now command.

The pull down menu for Static, Dynamic, and Shade Now contains a list of rendering and edge methods.



OpenGL Shade Methods



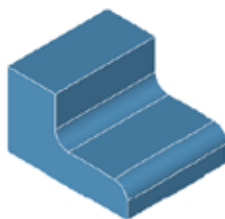
(a) Wireframe



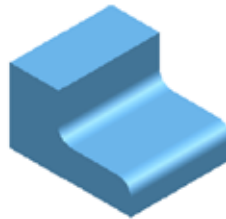
(b) Flat



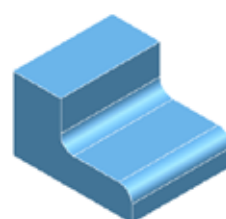
(c) Gouraud



(d) Gouraud w/Edges



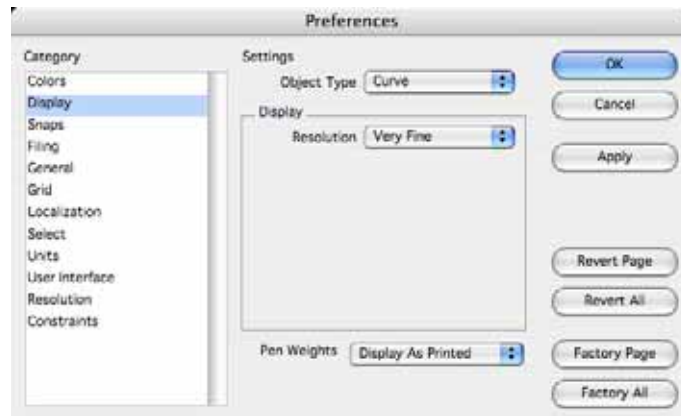
(e) Phong



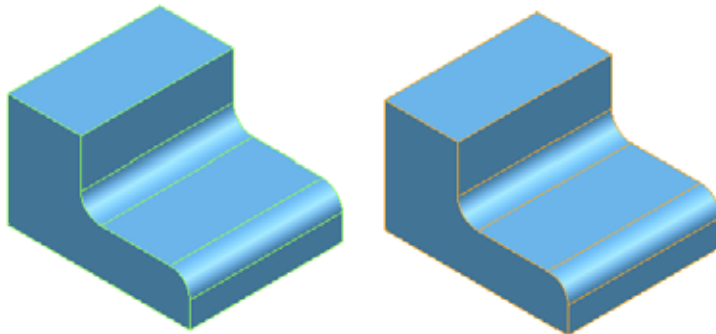
(f) Phong W Edges

OpenGL Render Methods

The render methods that include edges have options for the user to change the edge color. To change the edge color, access the Preferences:Display dialog box located in the File menu. Please note this is a global setting for all surfaces or solids.



Edge Color Methods



Different Edge Color Options

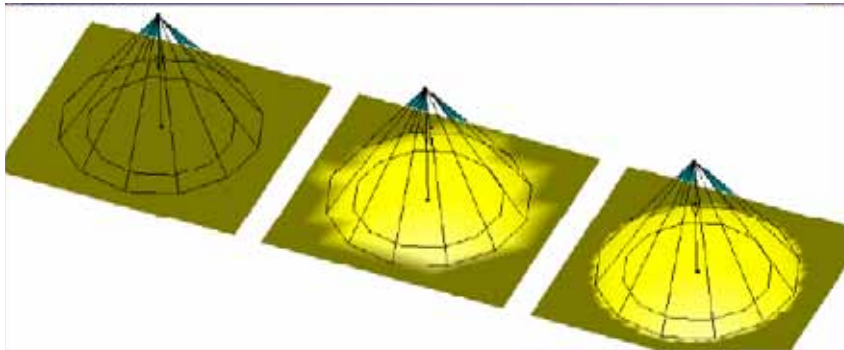
Drivers

The quality and performance of OpenGL rendering can depend on the choice of graphics adapter and driver version. If you are experiencing problems, be sure to check that you have the latest software driver for your given adapter board. In several cases, using a different graphics adapter and driver version combination results in serious display problems and in some cases causes instability problems.

Lights

OpenGL and Render use the same light tools for either render method. Use the light tool palette to create distant, point, and source lights. Use the Inspector to change parameters associated with a light. And turn on the light defining points with *Edit:Show Points* to modify position directly within the drawing. However there are some notable differences between the two. For example, in OpenGL shadows are not supported. In addition, facets are illuminated according to how much light falls on their vertices. If a light is positioned over a large flat face (such as a floor), it will not seem to be casting light. Setting the

object's resolution to "Very fine" or "Super fine" and setting the Precise Facets settings will increase the facet vertex density.



(a) *Fine*

(b) *Very Fine*
Vertex Density and OpenGL Lighting

(c) *Super Fine*

OpenGL Settings

The View:Shade Options dialog box contains many useful settings associated with the OpenGL shaded environment. For more information, see the chapter titled "View", which begins on page 119

Render

Render is a set of tools that controls the definition and generation of photo-realistic images. Unlike OpenGL shading, Rendering is not used during geometry construction and editing. Rendering is a post process tool that interacts with the geometry already created.

To effectively use Rendering, the user needs to create a scene. A scene may consist of one or more positional lights, ambient light characteristics, materials, decals, and background and foreground properties.

Lights

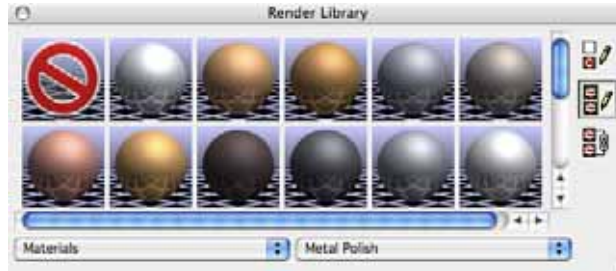
The positional light tools are used to define the lighting characteristics of a model. There are six light tools available. Use the Ambient Light Settings dialog box to define the ambient light characteristics. For more information on lights, see the chapter titled "Lights", which begins on page 491.



Render Library

The Render Library window is used to apply materials and decals to objects, and to select the background and foreground properties. The Render Library supports a drag-and-drop approach to

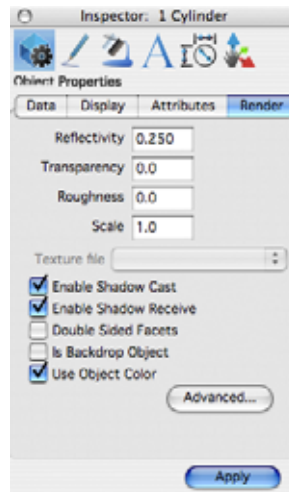
applying materials, and decals onto objects. To use a material, just drag it onto the object. For more information, see the chapter titled “Render Library”, which begins on page 479.



Render Library

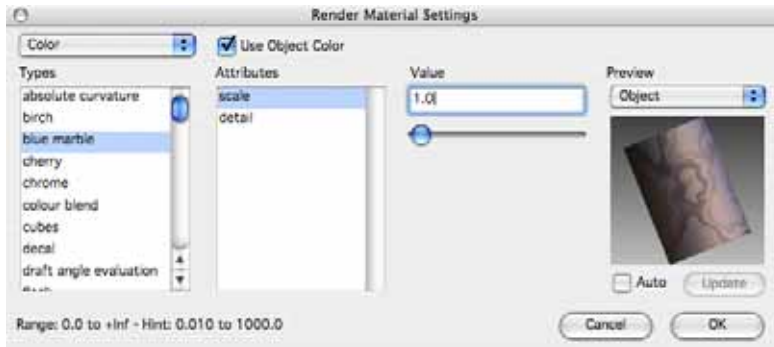
Advanced Material Editing

The Render Library provides an extremely easy means to add material characteristics to your geometry. The Advanced Material Editor provides you with a means to further tweak a material. To access the advanced material editor, select the object and display the Inspector. Next hit the material tab such that you see the Advanced button in the lower right.



Inspector Advanced Material Button

Clicking the Advanced button displays a dialog box for advanced material editing.



Render Material Settings

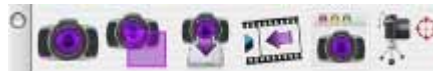
Advanced material editing provides a means to modify the following material attributes:

- Shader Class
- Shader Types
- Attributes
- Attribute Values

For more information, see the chapter titled “Advanced Material Editing”, which begins on page 497.

PhotoRender

The Render tool options are located in the main tool palette and provide access to generation of photo realistic images to the entire screen, a selected area of the screen, to file, display previous render, and adjust advanced settings.



Render Tool Palette

Render Window

The Render Window is the first icon in the Render Palette. The tool renders the entire window. The Render Window command has a sub tool palette which includes rendering the window with the following settings:

- User Specified
- Metal
- Glass
- Plastic
- Mirror
- Sketch



Render Options

The Prompt window has a pull down option for defining certain options for rendering regarding raytracing quality and drop shadows.



These options are further defined below:

Preview Render [shadows off]

This option utilizes a scan line algorithm for quick preview renders. Shadows are turned off.

Preview Render [shadows on]

This option utilizes a scan line algorithm for quick preview renders. Shadows are turned on.

Raytrace Render [shadows off]

This option utilizes a ray tracing algorithm for high quality renders. Shadows are turned off.

Raytrace Render [shadows on]

This option utilizes a ray tracing algorithm for high quality renders. Shadows are turned on.

Raytrace Render [shadows off, Anti-Alias]

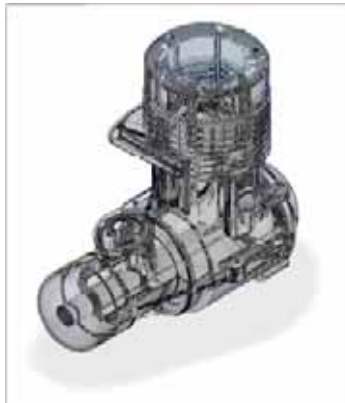
This option utilizes a ray tracing algorithm for high quality renders. Shadows are turned off, but anti aliasing is enabled.

Raytrace Render [shadows on, Anti-Alias]

This option utilizes a ray tracing algorithm for high quality renders. Shadows are turned on and anti aliasing is enabled. This option takes the longest, but produces the most realistic images.

Drop Shadow

The Render tool has a drop-down menu that provides an option to display a shadow.



Drop Reflector

The Render tool has a drop-down menu that provides an option to display a reflection on the floor.



The “shadow off” modes achieve fast image rendering speeds by skipping the scene's shadow calculations. The shadow “on modes” do produce shadows in the image at the expense of processing time. The Raytrace Render (shadows on, Anti-Alias) mode is intended for final image generation. It uses all the accuracy of ray tracing in combination with anti-alias over-sampling. The computation time is significantly longer, but the end result is very impressive.

RapidRender Environments

The Render Window tool also supports five RapidRender Environments. Use the RapidRender Environments to easily and immediately create stunning images from your models. These are preset rendering environments that automatically set up lighting, environment maps, and materials. The five RapidRender Environments are Metal, Glass, Plastic, Mirror, and Sketch.

For more information on Sketch Rendering, the chapter titled “Sketch Rendering”, which begins on page 467.



(a) Metal



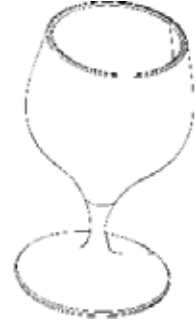
(b) Glass



(c) Plastic



(d) Mirror



(e) Sketch

RapidRender Environments



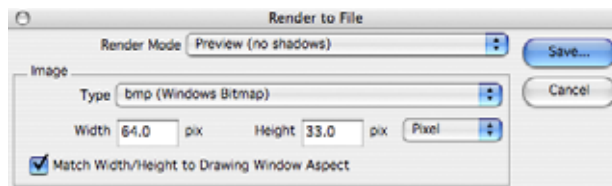
Render Area

The Render Area tool allows you to select an area of the window to render. The Render Area tool uses the user specified materials to generate an image.



Render To File

While rendering to a window is sufficient for proof of image and informal viewing, publication quality images require a resolution (>300 dpi) that far exceeds the resolution of a typical computer monitor (~90 dpi). The generation of high resolution images is controlled by the Render to File dialog box. This dialog is displayed using the Render:Render to File... menu option.



Render To File

The Render Mode drop-down list specifies which of the 6 modes is used for image generation. These match the modes described in the Rendering to a Window section above.



Render Modes

The Image section sets the output limits for the image file. The Type drop-down list specifies the format of the generated image file. The Width and Height values specify the image size in pixels. If the Match

Width/Height to Drawing Aspect is checked, the width and height will be automatically synchronized such that image's width-to-height ratio will be equal to the drawing window's width-to-height ratio. If the Match Width/Height to Drawing Aspect is unchecked, any appropriate values may be entered.



Image Output Options

There are seven image format types supported: Windows bitmap (bmp), Targa (tga), TIFF (tif), JPEG (jpg), Lightworks image (lwi), Encapsulated postscript (eps), QTVR Panoramic Movie (mov).



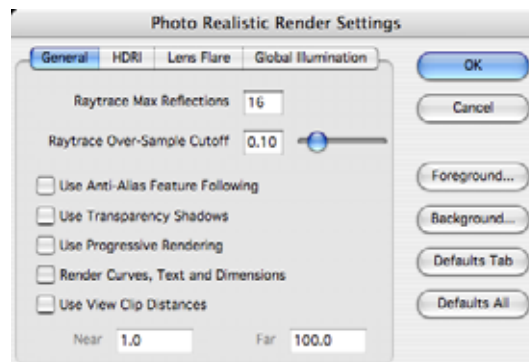
Show Last Image

Displays to the drawing window the last rendered image.



Render Settings

The Render Settings tool icon displays a dialog box with useful parameters associated with rendering.



Render Settings Dialog Box

For detailed information on all render settings, see the chapter titled “PhotoRender Settings”, which begins on page 451.

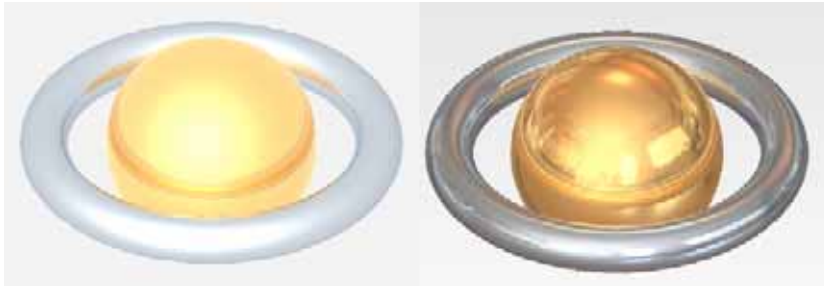
Multiple Processor Rendering

Punch! Shark FX supports multiple processors when using the photorendering command on Mac OSX. Additional processors are automatically enabled if you have a dual-processor Mac.

Environment Maps

A key to generating eye-popping images is placing an object in an environment or scene that will reflect realistically back onto the object. A very fast way to obtain this effect is to use a specially constructed image that projects onto the objects in a scene. These are called environment maps, which are supported in Punch! Shark FX through an easy to use drag and drop interface.

Environment maps are especially effective with reflective surfaces such as metals. In the example below the same scene is rendered with and without an environment map.



(a) Without Environment Map

(b) With Environment Map

Chrome and Gold Materials

Environment maps are implemented in Punch! Shark FX as Background Shaders. To access a background shader, display the Render Materials Dialog and select the Backgrounds option in the pull down menu. Next select the Environment shaders. Drag and drop any of these shaders to the background to add an environment map to your rendered image. For more information, see “Render Library”, which begins on page 479.

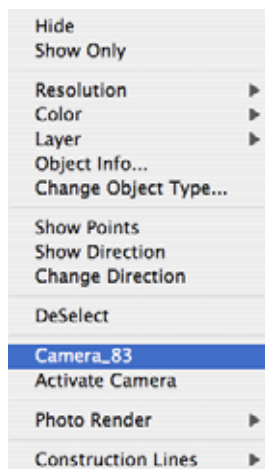


Background Environment Shaders

Camera Object

This tool creates a camera object saved as geometry, within the model. Use a camera object to store a location, direction, and field of view. The camera object is located as the last item in the photo-rendering tool palette.

A camera object is created by specifying the camera eye and reference point. Use the Data Entry window to specify a field of view. To activate a camera, right click on the camera symbol to display the menu.



Select the Activate Camera to change the view orientation to that specified by the camera.

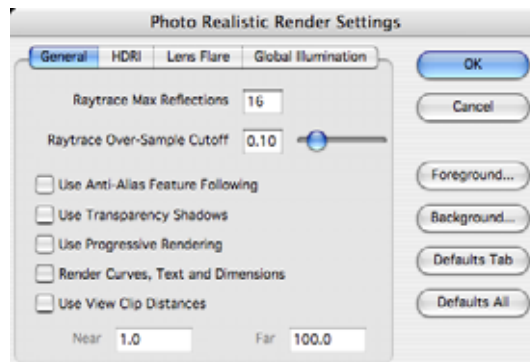
The Render To File tool also recognizes cameras. To render multiple view orientations, create the cameras and have them selected prior to picking the Render To File tool.

PhotoRender Settings

The Photo Realistic Render Settings dialog box controls general render, HDRI, Lens Flare, and Global Illumination settings. To access the Photo Realistic Render Settings dialog box, click the Render Settings tool from the PhotoRender toolset.

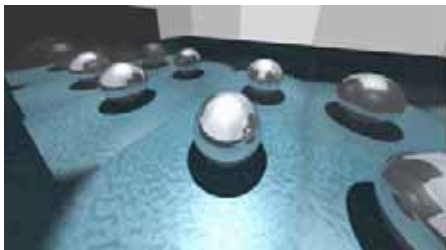


General Settings

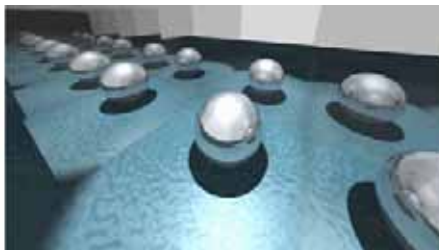


Raytrace Max Reflections

The Raytrace Max Reflections value controls the maximum number of “bounces” a ray will be allowed to make. Once a ray has reached this limit, no further color calculations will occur for that ray. The images of a mirror-walled room below demonstrate the effect of changes in the Raytrace Max Reflections value.



(a) Reflections = 4

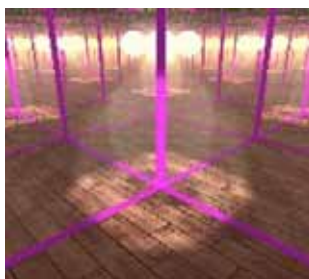


(b) Reflections = 8

Max Reflections

Raytrace Over-Sample Cutoff

The Raytrace Over-Sample Cutoff value controls the threshold for adaptive image over-sampling. The image will be sampled until adjacent color samples differ in the largest of their red, green, and blue components by an amount not exceeding the Raytrace Over-Sample Cutoff value. The valid range is from 0.0 to 1.0. The default value is 0.1.



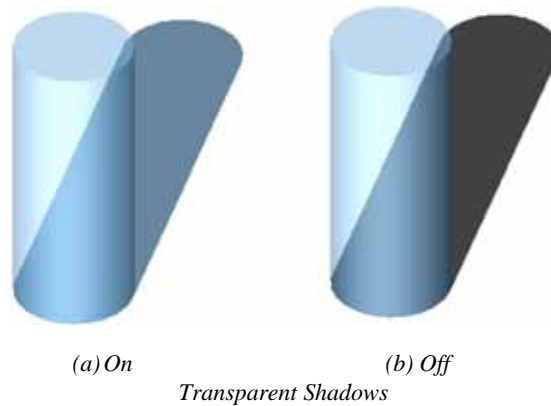
Over Sample Cutoff

Use Anti-Alias Feature Following

The Use Anti-Alias Feature Following (AAFF) checkbox controls the application of AAFF. If checked, a second pass will be made over an image that brings out small geometric feature details that may have been lost due to ray sample aliasing.

Use Transparency Shadows

The Use Transparency Shadows checkbox controls the behavior of shadow generation for transparent objects. If checked, the shadow cast through a transparent object uses the object color. If not checked, the transparent object will cast an opaque shadow.



Use Progressive Rendering

Progressive Rendering provides immediate feedback of a final image with a fast preview of the lighting and materials in a scene; it provides users with a way to get a desired image with less iteration.

Render Curves, Text, and Dimensions

This option enables photorealistic rendering of curves, text, and dimension entities. Use this feature to add annotations to a final rendered image.

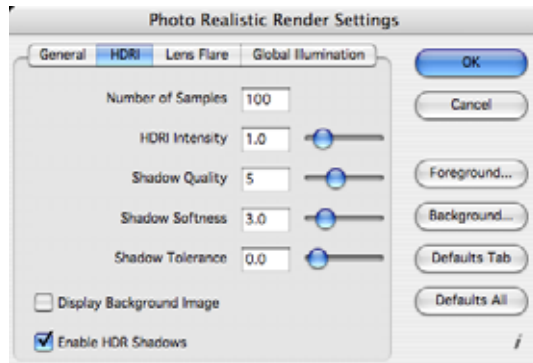
Use View Clip Distances

The Use View Clip Distances checkbox and the Near and Far values control the near and far clipping plane behavior. The near and far clipping planes are normal to the view vector and are at the specified distance from the eye point. Any objects (or portions of objects) that lie aft of the far clipping plane and forward of the near clipping plane are ignored for the image. As implied, an object can be “sliced” if it is intersected by a clipping plane.

If Use View Clip Distances is checked, the Near and Far distance values are used for the clipping plane distances. If Use View Clip Distances is unchecked, the near and far clipping planes will be automatically set to the near and far view extents of the model (all objects are rendered for the image). Unless you are trying to achieve a particular result, Use View Clip Distances should normally be left unchecked.

HDR Render Settings

For information on applying HDR Images, see “HDR Backgrounds Library” on page 481.



Number of Samples

The quality of lighting decides how good the lighting seen on objects will correspond to the environment. It depends mostly on the "number of samples", i.e., the number of lights used to simulate environment lighting. The more samples used, the more accurate the lighting. On the other hand, increasing the number of samples makes rendering longer.

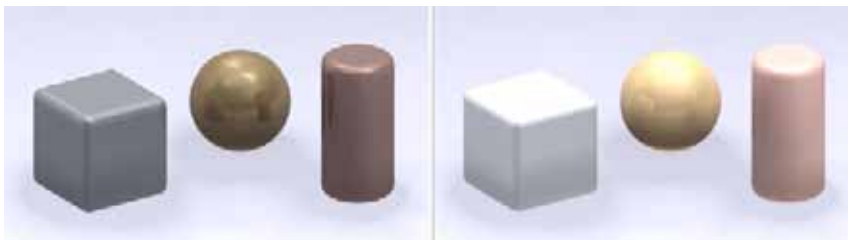


50 Samples

500 Samples

HDRI Intensity

The HDRI intensity parameter decides the strength of the lights which are lighting a scene and the overall brightness of the rendered image. Typically you will want to match brightness of lit objects with the brightness of the environment (for example, if the environment is bright, objects lit by it should be bright also). Choosing a value which will make lit objects match the background requires experimentation.

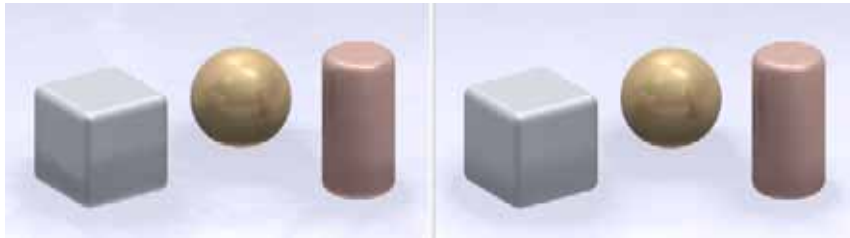


0.5 Intensity

2.0 Intensity

Shadow Quality

Increases the quality of shadows at the cost of rendering time.

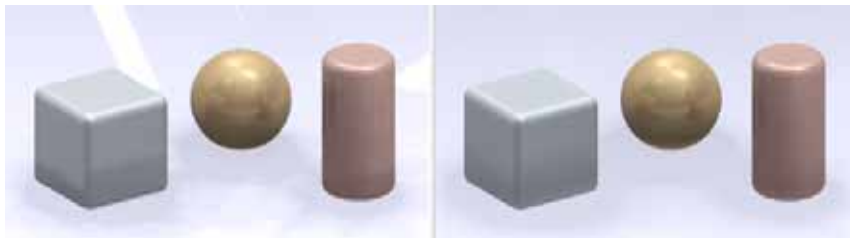


1 Shadow Quality

8 Shadow Quality

Shadow Softness

Determines how soft the boundaries of shadows appear.



1 Shadow Softness

4 Shadow Softness

Shadow Tolerance

Modifies the shadow map sampling distance. It is used to deal with self-shadowing artifacts that can be apparent with low resolution, high softness shadows. They usually appear as either grid-like patterns of smudges or moire patterns. A value of 0.0 specifies no effect, a value of 1.0 causes no shadows. Values around 0.1 to 0.2 are usually sufficient to deal with most artifacts. The value of this parameter should be as small as possible, because large values can cause shadows to disappear from objects close to the object casting the shadow. A value of 1.0 will cause all shadows to vanish.

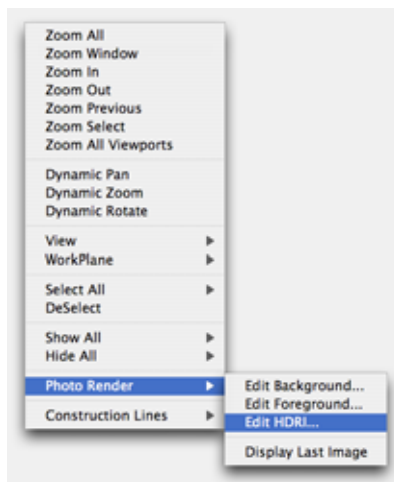


0 Tolerance

0/4 Tolerance

HDRI Map Type

The HDRI Setting dialog controls the HDRI Map Type. To access the dialog, right-click the design window and choose Edit HDRI from the PhotoRender menu.



There are seven options available on the drop-down menu.



Spherical Map

Usually created by photographing a reflective sphere with a telephoto lens.

Latitude/Longitude Map

Sometimes known as a panorama, where the entire environment is flattened into a single, flat panoramic image, rather like a map of the globe.

Angular Map

Sometimes known as a “light probe” image. It is similar to a spherical map from a reflective sphere, except the radial dimension is mapped linearly with the angle, instead of being pushed towards the edge of the circle. This gives a more accurate sampling around the edges of the image.

Vertical Cross Cube Map

A cubic map created by reading a single image, which consists of six sub-images laid out in the form of an unfolded cube.

Auto Detect

This examines the image data for solid black areas and decides on the type, depending on the pattern made by the areas of black. It is, therefore, capable of correctly distinguishing the following types of images only: angular, horizontal cross cube, vertical cross cube, and latitude/longitude.

Horizontal Cross Cube Map

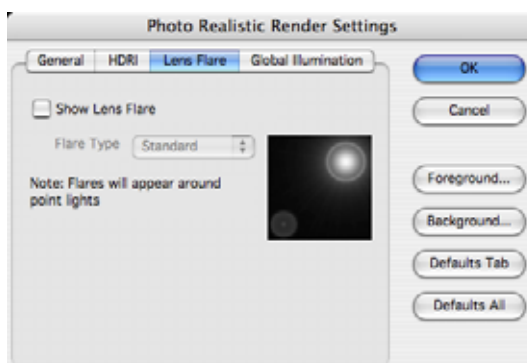
A cubic map created by reading a single image, which consists of six sub images laid out in the form of an unfolded cube.

Vertical Strip Cube Map

A cubic map created by reading a single image, which consists of six sub images laid out flat into a single image in 1:6 aspect ratio, consisting of 1 by 6 square sub-images representing the cube.

Lens Flare Settings

Access to Lens Flare settings is available under the PhotoRendering Settings dialog box, Lens Flare Tab. Options available include turning on or off the lens flare and lens flare types.



Note: A lens flare requires the presence of a Point Light type in the model located near the desired location of the flare. The intensity of the flare is driven by the intensity of the light. To change the light intensity, select the light and change the intensity value on the Data tab of the Inspector. For more information, see “Point Lights”, which begins on page 494.

When real cameras are pointed toward bright light sources, some sort of lens flare effect is produced as a by-product of the direct light interacting with the system of lenses within the camera. Effects include:

- central area of glow
- random and regular streaks of light
- halos around light sources
- ghost images of the lens iris

The main components of such effects may be circular, polygonal or polygonal with rounded edges, depending on the shape of the camera’s iris. Although such effects might be thought of as a side-effect of

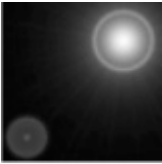
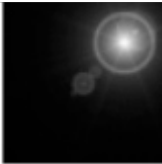
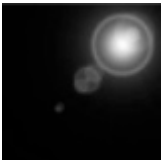
the design of real life cameras, they are often used by photographers and cinematographers to produce pleasing visual effects.


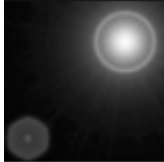


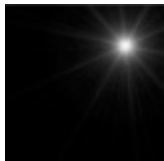
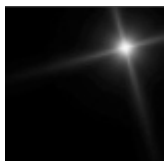
Punch! Shark FX simulates lens flare effects as an image post-processing effect. To enable Lens Flare effects, display the Photo Realistic Render Settings dialog box and turn on the “Show Lens Flare” checkbox. The scene will render normally, then render once more adding in the lens effect.



Show Lens Flare

This checkbox enables post processing of Point Light types as Lens Flare effects.

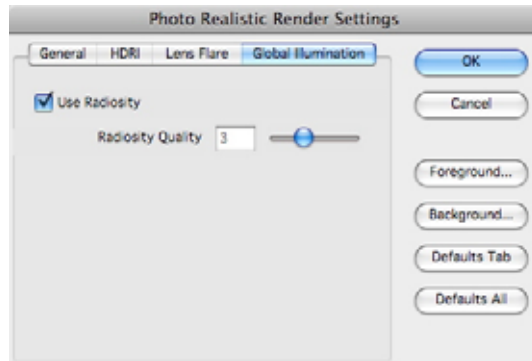
Lens Type	Description	Example
Standard	Standard camera. One circular ghost and random light streaks.	
Lens 35	Mixed light streaks (random and regular), five ghost images.	
Lens 50	Random light streaks, accentuated bloom, eight ghost images.	

Lens Type	Description	Example
Lens 105	Mixed random and regular streaks, fuzzy halo, and twenty rounded polygonal ghosts.	
Polygon	Mixed random and regular streaks, fuzzy halo, and twenty rounded polygonal ghosts.	
Polygon 35	Same as LENS F35 but with sharp polygonal ghosts.	
Polygon 50	Same as LENS F50 but with sharp polygonal ghosts.	
Spark	Mixed random and regular streaks, no halo or ghosts.	
Star 4	Slightly accentuated bloom, regular streaks. Simulates “star” filter lens.	

Global Illumination Settings

Global Illumination is a lighting method used to evaluate the distribution of all the light energy in an environment. This feature is enabled through the Photo Realistic Rendering Settings dialog box.

The final rendering solution that is generated encapsulates the light distribution throughout the scene, accounting for all diffuse interreflections, and is independent of any particular viewpoint. Simpler rendering techniques (non-global illumination) attempt to approximate the effect of diffusely reflected light in a scene by applying a constant “ambient” value of illumination throughout the entire scene.

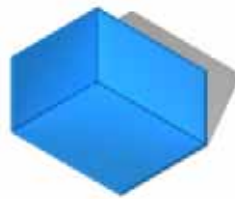


Radiosity is essentially a technique which correctly computes the distribution of this “ambient” light, accounting for light reaching a surface after being reflected from one or more diffuse surfaces. To enable this feature, click on the Global Illumination tab and Use Radiosity checkbox.

The quality settings is a natural number, lying in the range 0-9, which is used to set groups of internal control variables such as shadow quality and sampling. The default value is 2, with 0 indicating low quality and 9 indicating a very high quality.



Radiosity Off

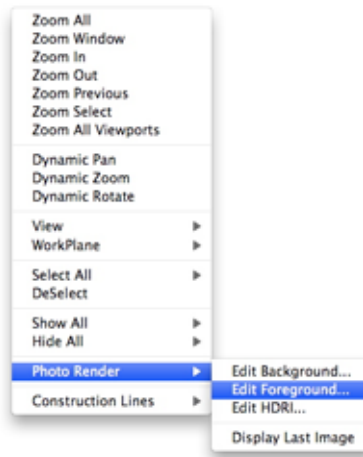


Radiosity On

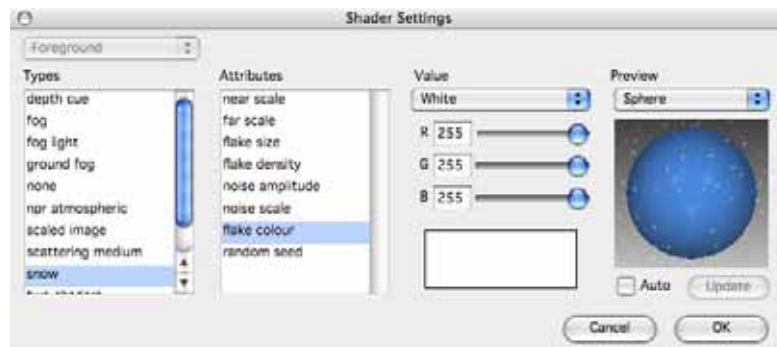
Note: Enabling radiosity can significantly increase rendering times.

Edit Foreground

The Edit Foreground menu option displays a dialog box that allows editing of the foreground used in the Render commands. You can access the Foreground Shader Settings from the Render Setting dialog or from the right-click menu.



The dialog box has the same familiar interface of the Advanced Material editor which consists of Shader Types, Attributes, and Attribute values.



Foreground Edit Dialog

Below is a list of the Foreground shader types and associated descriptions.

depthcue A foreground shader that adds a background color specified by parameter “background color” according to the distance of the surface from the viewer. At distances less than that given as “near”, no background color is added. At distances greater than “far”, the full background color is returned. Between the two distances, the output is linearly interpolated between the surface and background colors. The effect is that depth information in the geometry is enhanced in the final image.

fog A foreground shader that implements atmospheric absorption to create a ‘fog’ effect. The contribution made by the fog whose color is given by “fog color”, is attenuated asymptotically using the formulation: $1 - e^{-d}$ where d is the distance of the shading

point divided by the reference distance supplied as “distance”. Increasing this value advances the fog further from the viewer.

The density of the fog is clamped to a maximum value specified by parameter “max density”. By default this value is 1.0. Values between 0.0 and 1.0 are sensible – values less than zero will be interpreted as zero and values greater than one will be interpreted as one.

The fog effect can be enabled or disabled for background pixels using “ignore background”. By default this is set to TRUE, i.e., there is no fog effect applied to background pixels.

foglight

Atmospheric scattering from point and spot light sources. A constant distribution of particles in the atmosphere is assumed.

The light fall-off used in the scattering computations is $1/d^2$, so light sources tend to look saturated near their center. Light sources should therefore also use a $1/d^2$ type fall-off for consistency between atmospheric and surface shading.

Also note that spot lights should have an almost constant angular intensity distribution (i.e., a hard-edged illumination cone). Volumetric shadows are not implemented by this shader.

grndfog

This shader is used to create more subtle fog like effects than the standard “fog” shader. With this shader the fog density decreases exponentially along a user specified axis. The parameter “fog height” sets the rate of decrease (if this is set to zero or a negative value, the effect is the same as with the standard “fog” shader).

The parameters “ground point” and “ground normal” define the place where the fog lies. It is assumed to be the plane $z=0$ by default.

Note that fog does not exist at all below the ground plane, so in the default case there is only fog in the $z>0$ halfspace.

The density of the fog is clamped to a maximum value specified by parameter “max density”. By default this value is 1.0. Values between 0.0 and 1.0 are sensible – values less than zero will be interpreted as zero and values greater than one will be interpreted as one.

The fog effect can be enabled or disabled for background pixels using “ignore background”. By default this is set to TRUE, i.e., there is no fog effect applied to background pixels.

noneforeground No foreground. Pixel values are output unmodified.

scatteringmedium This shader enables simulation of a full range of effects occurring in a participating medium, including:

- attenuation within medium

- light filtration through colored medium
- first order light scattering inside medium with volumetric shadows

The light scattering effects are modeled for all light shaders except “ambient”, “eye” and “sky” (though are available in “area sky” shader). The “first order scattering” mentioned above means that the shader visualizes direct effect of scattering in medium – scattered light coming to observer but does not consider secondary effects – multiple bounces of light within the medium or illumination of surfaces by scattered light.

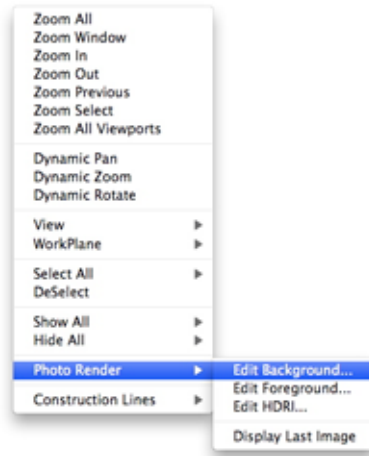
The shader performs attenuation of original surface color according to given medium attenuation coefficient and medium color. As one could expect, the longer distance from shaded point the dimmer and more colored by medium it becomes. In addition to attenuation it is also possible to specify medium ambient light scattering which gives overall veiling effect. Finally the shader models scattered light from all the supported light sources if “scattering” parameter of the source is set to TRUE.

snow

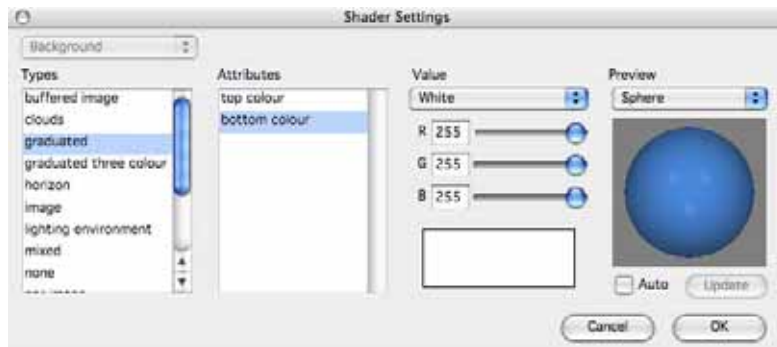
A foreground shader that gives the effect of snowflakes falling in front of the camera. The effect is produced by two overlaid planes of randomly dispersed “snowflakes”. The scale of each of the two planes is specified by the parameters “near scale” and “far scale”. The size and density of the flakes is given by the parameters “flake size” and “flake density”, and the irregular appearance of the edges of the flakes is controlled by the “noise amplitude” and “noise scale” parameters (these last two values should lie in the range 0.0 to 1.0). The color of the snowflakes is specified by the parameter “flake color”. Finally, it is possible to change the distribution of the snowflakes from the default by changing the “random seed” parameter. This provides an initial seed value for the random distribution.

Edit Background

The Edit Background menu option displays a dialog box that allows editing of the background used in the Render commands. You can access the Foreground Shader Settings from the Render Setting dialog or from the right-click menu.



The dialog box has the same familiar interface of the Advanced Material editor which consists of Shader Types, Attributes, and Attribute values



Background Edit Dialog

Below is a list of the Background shader types and associated descriptions.

- cloudsbackground** A background giving a cloudy appearance. The color of the clouds and the background (sky) may be specified in arguments “clouds color” and “background color”, respectively. The detail or complexity of the texture may be controlled by means of the “detail” argument. A value of 1 will give a simple outline of the clouds, while a larger value such as 5 or 6 will give fine detail on the cloud outlines. An overall scaling factor is provided in argument “scale”; increasing this value will make the clouds appear larger.
- envmapbackground** When this background shader is used, the current global environment map is sampled at background pixels.

The “intensity” argument allows the ‘brilliance’ of the reflection to be altered – the color value calculated for each pixel of background is multiplied by the intensity value.

The “angle” specifies the angle, in radians, over which the environment map is sampled for each pixel of background. This allows the environment map to be “blurred” slightly. By default the value is 0.0, meaning that the color of each pixel is established from a single point sample taken through the center of the pixel.

graduated A background providing a graduation from the top of the image to the bottom of the two colors specified as arguments “top color” and “bottom color”, respectively.

horizon The horizon is shader provides a simple horizon with a graduated sky and ground colors.

image A background shader that maps an image.

The name of the file containing the image data may be provided as a string to parameter “file name”, which can be in any format for which the appropriate image driver is installed.

mixed A background shader which mixes the results of two other background shaders (“base shader” and “mixed shader”) according to the ratio given as parameter “mixing ratio”.

nonebackground No background. Regions of the image not covered by any surface will appear black.

plainbackground A plain background with a single, uniform color. The color is supplied in argument “color”.

raycube A problem with ordinary background shaders is that they don’t handle reflections or refractions. This is because all pixels at which the background is deemed to be visible are shaded in the same way, using a background which is located only in image space, rather than in real 3D space.

Thus, for example, a mirror, which should reflect the background behind the viewer, in fact shows the background exactly as it would be if the mirror were not there (i.e. all the mirrors become transparent). This would even be the case if the mirror has geometry behind it, so the effect becomes one of a hole in the geometry.

This shader allows two other background shaders to be used, one for the areas of background directly visible to the viewer, and another for reflections.

Thus the “primary shader” is used for primary rays which do not intersect any geometry, and the “secondary shader” for reflection rays which do not intersect any further geometry (i.e., reach the background).

“Reflected shader” or “refracted shader” allow background shaders to be specified for reflected and refracted rays. If either is defined then it will be used in preference to any “secondary shader” that is specified. The “refracted shader” also applies to any rays that have passed through a transparent medium. If either of the shaders is not defined, then black is used as the default background color.

A situation where this shader is very useful for rendering images where the directly visible background is supposed to be replaced in post-production. The environment background would be used for reflections, and the plain background for the directly visible background, which can then be replaced by an image. This may be useful when producing animations.

scaledimage

A background of an image read from a file, and scaled to fit into the viewport. The image may also be rotated.

twoplanes

This shader allows two background images to be located in world space (one in front of the observer and one behind). These images can then be ‘reflected’ in any mirrors, thus producing the correct effect (refraction is also correctly handled).

Sketch Rendering

Sketch Renderings are designed to support rendering at the conceptual design stage or for stylized presentations. The supported styles include:

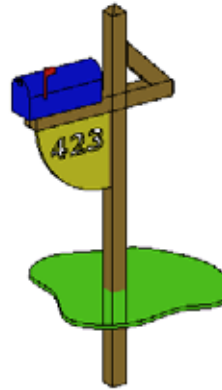
- cartoon
- colour wash
- contour
- hand drawn
- hatch
- ink print
- lines and shadow
- mosaic
- oil painting
- rough pencil
- soft pencil
- stipple

The Sketch Rendering tool activates the different tools on the Prompt window.



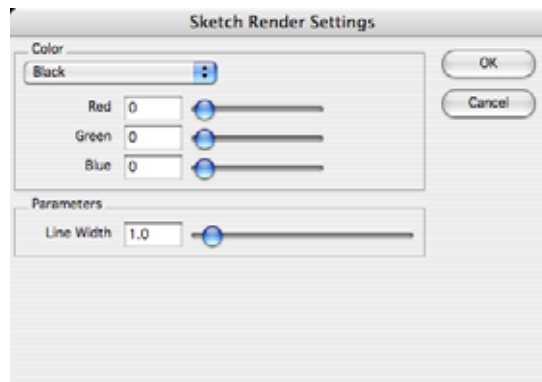
Cartoon

A classic cartoon style effect, where silhouette and boundary edges are picked out with bold lines and colors are somewhat simplified and stylized. It is possible to control the color and width of the lines in the Options dialog. The colors painted between the lines will depend on the materials, but are quite flat (i.e., very little tonal change as you look across a surface) to reflect the simplified palette employed by cartoon artists. This style uses hidden-line.

*No Render**Cartoon Render*

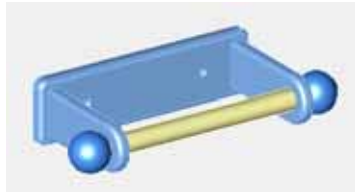
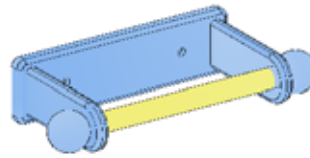
Options

Click the Options button to access the Sketch Render Settings dialog box. Here, you can control the color and line width.



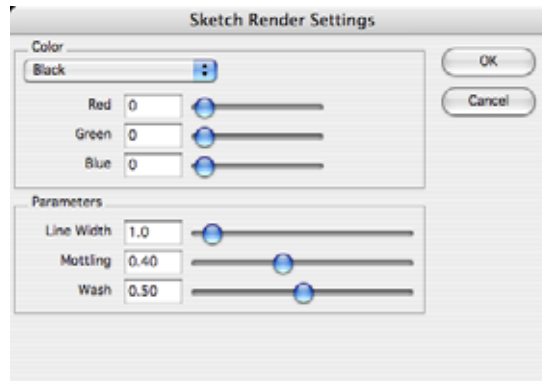
Colour Wash

As in the cartoon style, edges are picked out with lines, and the spaces in between are filled with simplified color. In the case of colour wash, however, the colors are less cartoon-like, and can be washed-out to look much more pastel and less intense. The width and color of the lines is still specified via the options dialog, but it is also possible to control the degree of color variation across objects, and the degree to which the colors are washed. This style uses hidden-line.

*No Render**Colour Wash Render*

Options

Click the Options button to access the Sketch Render Settings dialog box. Here, you can control the color, line width, and other parameters associated with the style.



Contour

Creates a very “painterly” effect, with strokes and swirls of color which reflect and represent the orientation of the underlying geometry. The effect may also be interpreted as being akin to using colored pencils to shade the image (with the direction of the pencil strokes matching the underlying geometry at all times). This style does not use hidden-line.

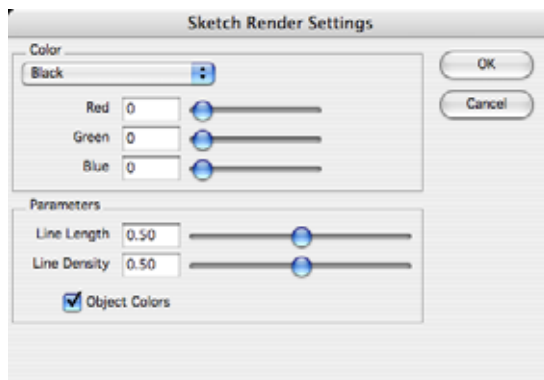
*No Render**Contour Render*

Options

The length and density of each stroke can be edited.

A low value for the line or stroke length will lend a finer, more detailed appearance to the final image. A low value for the density of the strokes or lines will result in more of the background color or paper color, showing through. Therefore, a background of either paper or canvas, certainly a light color, is likely to be more appropriate in many cases.

Normally, the colors of the strokes would be taken from the color of the objects being represented however, this can be changed so that a particular line color is used; simply uncheck the Object Colors checkbox.



Hand Drawn

This is a style in which objects are rendered using lines, but where the lines appear to be made up of individual strokes. Note that the current background is interpreted as being the “paper” or “canvas” onto which the lines are drawn, so it is important to choose a color for the lines which will be visible against your particular background. This style uses hidden-line.



No Render

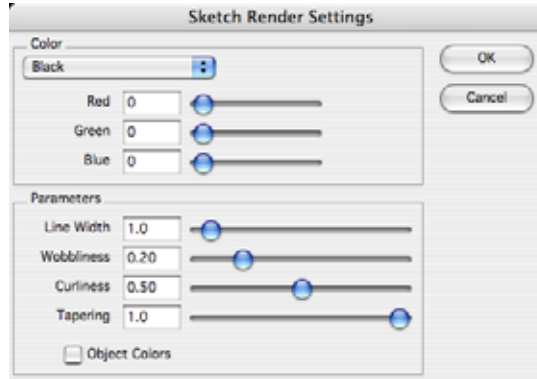


Hand Drawn Render

Options

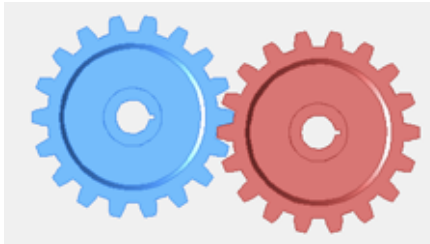
The color and width of the lines can be controlled in the Sketch Render Settings dialog, as can the degree from which each stroke varies from true, the smoothness of the strokes, and also the degree of taper from the start to the end of the stroke. All of these factors can be controlled via shader arguments, allowing the

effect to be suited to different models at different resolutions, with different drawing styles or pencils or pens.



Hatch

This style produces an effect similar to a shaded pencil drawing, in either monochrome or color. The image is made up of a series of pencil “strokes” all in a similar orientation (short strokes, from bottom left to top right in direction, as though a right handed artist were shading roughly with a pencil). All of the strokes are to provide shading - no lines are drawn. This style does not use hidden-line.



No Render

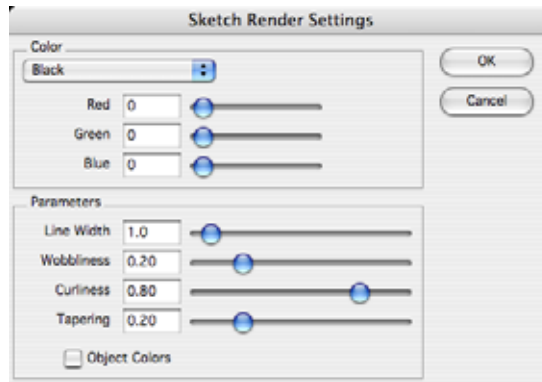


Hatch Render

Options

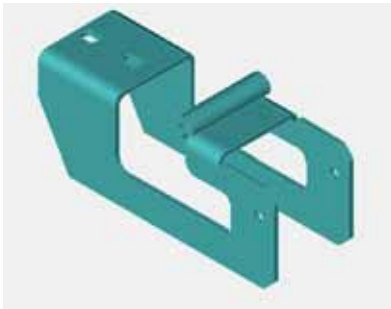
The Sketch Render Settings dialog offers controls for the color, width, length, and density of the pencil strokes, as well as how straight or curved the strokes appear.

A low value for the density of strokes will result in a very sparse looking image, with plenty of “paper” showing through, while a higher density will result in dark regions being almost completely shaded (note that lighter regions will still show up as the paper color or background).

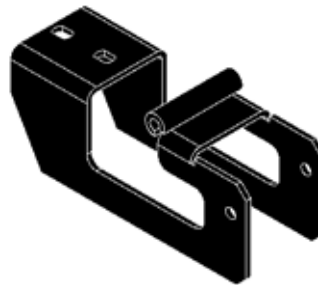


Ink Print

This style can be thought of as almost a photographic negative of other line-based styles. The entire image is rendered in a given “ink color”, save for the lines, which are simply left as “gaps” in the ink color. This style uses hidden-line.



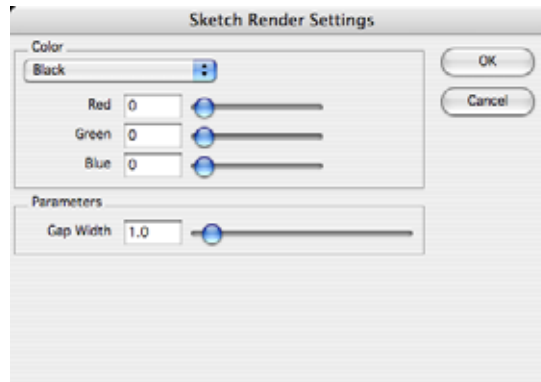
No Render



Ink Print Render

Options

The Sketch Render Settings dialog allows you to specify the ink color and the size of the gaps.

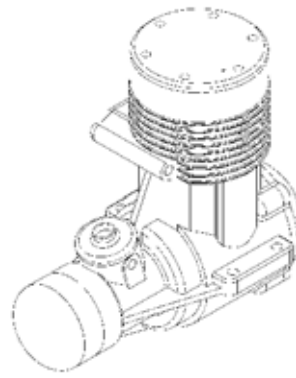


Lines and Shadow

This style combines a simple line representation of geometric objects with a (monochrome) shaded effect for areas in shadow. The result is very much like a pencil drawing where edges and shape are captured first using lines and then some shading work is done with the edge of the pencil. This style uses hidden-line.



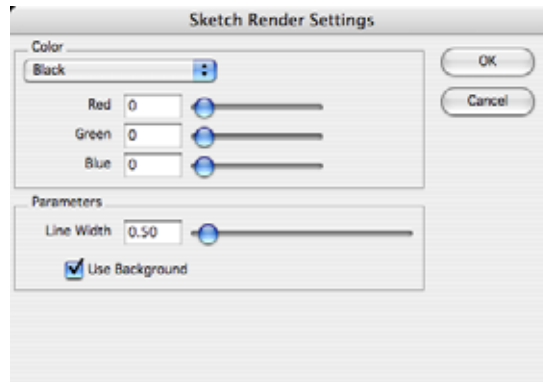
No Render



Line & Shadow Render

Options

The color and line width can be controlled on the Sketch Render Settings dialog.



Mosaic

This has the effect of rendering the image as though it were constructed as a mosaic of small colored tiles. If desired, it is possible to leave areas of exposed background in the scene untitled, in other words only to produce a mosaic pattern for the geometry, then apply some other pattern or image for the background. This style does not use hidden-line.



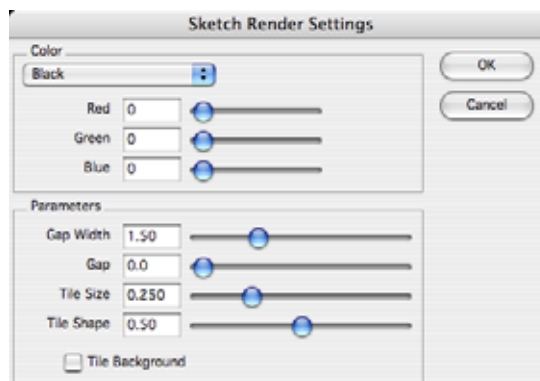
No Render



Mosaic Render

Options

The size and shape of the tiles can be controlled on the Sketch Render Settings dialog, as can the size and color of the gaps between the tiles. It is also possible to use the background color as the color for the gaps between the tiles or to blend a background and a given gap color.

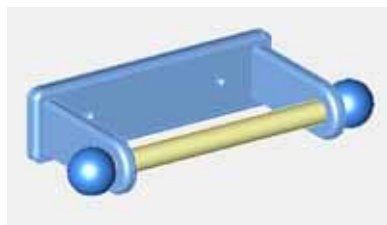


Oil Painting

This style creates an effect reminiscent of some impressionist painters, with individual blobs of paint (which could be brush or palette-knife strokes) of a single color applied across the canvas, so that the overall image only resolves itself if you view from far enough away.

The brush strokes are always slightly circular, roughly C-shaped marks. The shape of the strokes is not affected by the orientation, proximity or color of the underlying geometry that the stroke represents.

The effect is impressive with image scenes that have a strong shape and do not rely too heavily on detail to convey their meaning. Scale, and how the detail relates to scale, is quite important; if the size of the brush strokes means that important detail is lost then the results may be unimpressive. This style does not use hidden-line.



No Render

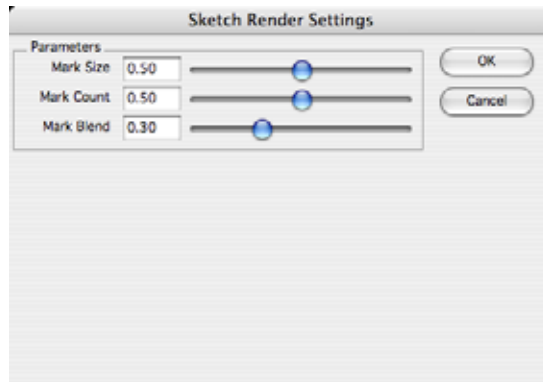


Oil Painting Render

Options

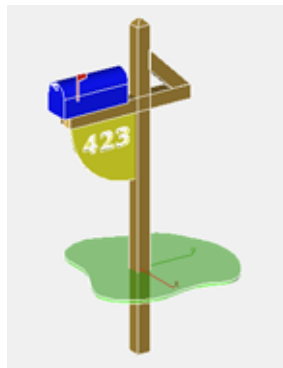
Using the Sketch Render Settings dialog, it is possible to alter the size of the individual marks, and the number that appear in the whole image (it is important to choose values for these parameters such that any detail required in the final image will be resolved, without losing the 'painterly' effect).

It is also possible to control the degree to which the marks obscure the background color. The entire canvas can be filled with marks that overlap or there can be gaps between the marks, creating a lighter feel.



Rough Pencil

Rough pencil produces an effect as though an artist had gone over each line several times, each with a small margin of error. The style is well suited to situations where a design is still quite tentative, as the effect is of a very rough, quick pencil sketch. This style uses hidden-line.



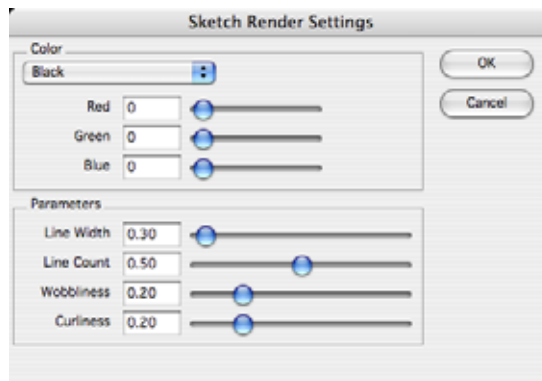
No Render



Rough Pencil Render

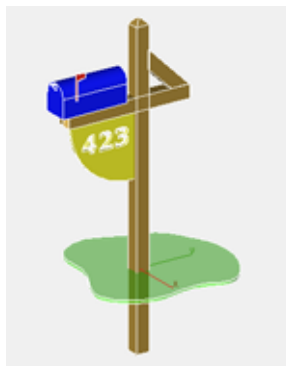
Options

The color, width, and number of lines can be controlled on the Sketch Render Settings dialog, as can the ‘wobbliness’ of lines (how far each line can deviate from true) and the ‘curliness’ of lines (how far from straight or jagged each line is).



Soft Pencil

The soft pencil style mimics the effect of a pencil drawing done with a soft pencil, so that in some places the pencil hardly leaves a mark. This style uses hidden-line.



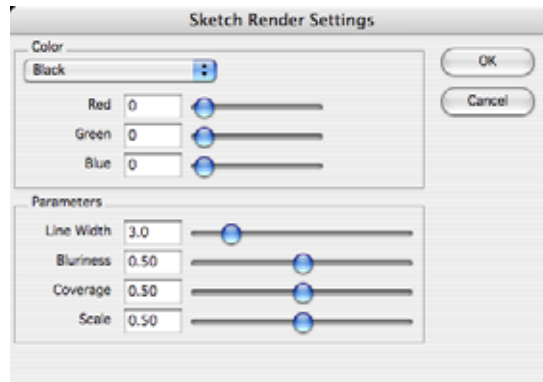
No Render



Soft Pencil Render

Options

The color and width of the pencil line, as well as the line sharpness, can be adjusted on the Sketch Render Settings dialog. It is also possible to control the extent to which the pencil line appears to fade in and out as the pencil traveled over the paper.

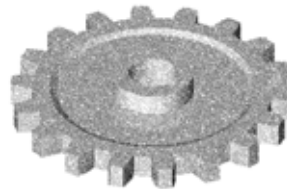


Stipple

This effect causes the image to be rendered as a series of irregular dots or stipples. The effect is similar to how photographs are reproduced in newsprint, only more pronounced. By default the stipples are all rendered in one color (black by default, but the 'ink' color can be altered). This style does not use hidden-line.



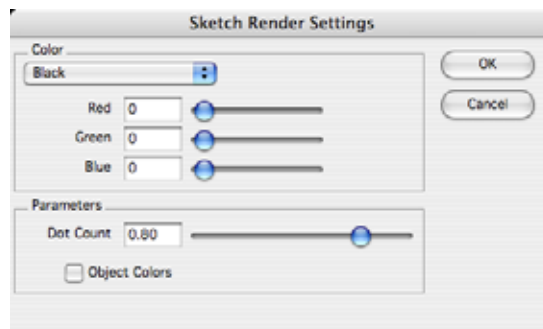
No Render



Stipple Render

Options

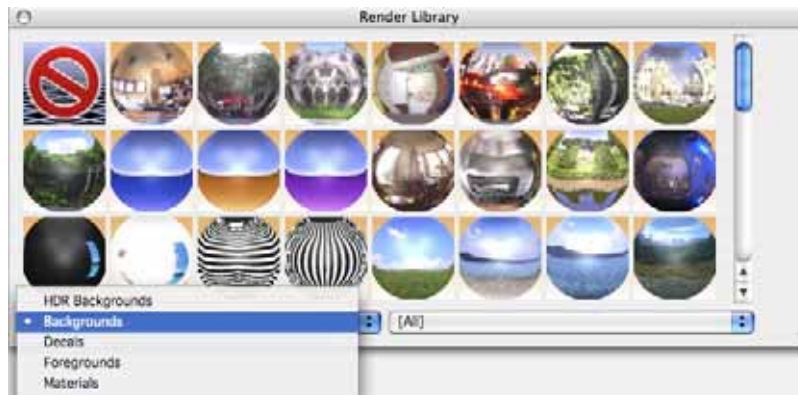
Using Stipple, a color can be specified or you can choose to use the color of the underlying geometry for each stipple. If this option is chosen, the effect becomes reminiscent of the 'pointillism' school of impressionistic painting, where scenes were rendered by points of color or light. You can also adjust the amount of dots that appear.



Render Library

The Render Library window is used to apply materials and decals to objects, and to select the scene's background and foreground properties. The Render Library window is accessed through the Window:Render Library menu option. The window displays preview images that indicate the render properties of a particular item. The library type and category drop-down controls the items displayed in the window.

The Library Type drop-down list controls which library is displayed on the palette. Depending on the layout, this is the left-most or top-most drop-down (Decals in the figure to the left). The library types supported include HDR Backgrounds, Backgrounds, Foregrounds, Decals, and Materials.



Each of the material and color libraries is divided into categories, which appear on the pop-up menu on the right. Choosing a category controls what appears on the Preview Bar. In addition to viewing categories, you can view based on where the material or color came from. [All] displays all materials and colors. [System] displays materials and colors that are included with Shark FX. [User] displays colors and materials imported or created by the user.



One more general note regarding the Render Library window. By clicking in the lower right corner you can resize the window. Resizing the window provides a means to customize the size for your particular user environment.

Accessing the Render Library

- On the Window menu, choose Render Library. The Render Library window is displayed.

Note: You can reshape and resize the Render Library, as well as move it around the workspace, as you design.

There are three methods for applying materials.



Apply Normal

The Apply Normal option applies materials to a single face.

Applying colors or materials to an individual face

- 1 Select Apply Normal, at the top of the Render Library.
- 2 From the Library Type drop-down, choose the library you want. All of the library's contents appear on the Preview Bar. Notice, as you move your pointer over each thumbnail, its name appears at the top of the Render Library.
- 3 (optional) From the Categories drop-down, choose a category to specify what appears on the Preview Bar.
- 4 Drag a selection onto a surface.



Apply Body

The Apply Body option applies materials to an entire object.

Applying colors or materials to an entire object

- 1 Select Apply body, at the top of the Render Library.
- 2 From the Library Type drop-down, choose the library you want. All of the library's contents appear on the Preview Bar. Notice, as you move your pointer over each thumbnail, its name appears at the top of the Render Library.
- 3 (optional) From the Categories drop-down, choose a category to specify what appears on the Preview Bar.
- 4 Drag a selection onto an object.



Apply Similar

The Apply Similar option applies materials to objects with the same color or material as the object to which you are applying.

Applying colors or materials to all similar surfaces

- 1 Select Apply Similar, at the top of the Render Library.
- 2 From the Library drop-down, choose the library you want. All of the library's contents appear on the Preview Bar. Notice, as you move your pointer over each thumbnail, its name appears at the top of the Render Library.
- 3 (optional) From the Categories drop-down, choose a category to specify what appears on the Preview Bar.
- 4 Drag a selection onto an object.

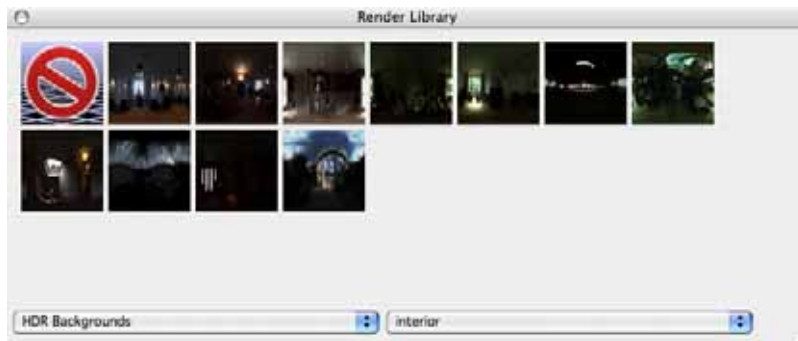
HDR Backgrounds Library

High Dynamic Range (HDR) images are images that contain complex lighting data in addition to the visible image. This allows for a straightforward way to introduce complex lighting environments into your rendered scene. These images can be captured with specialized equipment or obtained from outside sources. A number of HDR images are included and can be accessed in the Render Library.

User-supplied HDR images can be added to a custom category folder:

- PC- <User Documents>\Punch!\Shark FX\PhotoRender\Materials\<<category>
- Mac-Users/<User Name>/Documents/Punch!\Shark FX/PhotoRender/Materials/<category>

Custom images are displayed in the Render Library under HDR Backgrounds, with a category of the same name as the folder you created. All HDR images must have an .hdr extension. Supported formats include spherical, latitude/longitude, angular, vertical cross cube, horizontal cross cube, and vertical strip cube maps.



Applying an HDR Background

- 1 From the Library Type drop-down, choose HDR Backgrounds. HDR Backgrounds are displayed on the Preview Bar.
- 2 (optional) From the Categories drop-down, choose a category to specify what appears on the Preview Bar.
- 3 Drag the background onto the design.
- 4 To see the effects, activate a rendering tool.

Settings for HDR images are controlled on the Photo Realistic Render Settings dialog. For more information, see “HDR Render Settings”, which begins on page 453.

Applying a custom HDR background

- 1 Save an image to the path specified above, depending on the type of computer you are using.
- 1 From the Library Type drop-down, choose HDR Backgrounds. HDR Backgrounds are displayed on the Preview Bar.
- 2 From the Categories drop-down, choose the category that corresponds to the image you want.
- 3 Drag the background onto the design.
- 4 To see the effects, activate a rendering tool.

Settings for HDR images are controlled on the Photo Realistic Render Settings dialog. For more information, see “HDR Render Settings”, which begins on page 453.

Materials Library

The materials library is a collection of predefined material property settings that can be applied to renderable objects within a model. A material defines the surface color, transparency, reflectivity, and displacement properties of an object. A wide variety of visual appearances can be produced with the proper combination of these attributes.

Each material has a potentially large set of attributes that is determined by the internal material definition. These attributes have been mapped to a smaller, basic set of common attributes. The common attributes are edited using the Materials page in the Inspector. A Material tab will be displayed in the Inspector when the object is selected. The simultaneous editing of common material attributes for multiple selected objects is supported in the Material edit page. The ability to edit the full set of material attributes is supported through the advanced button located in the lower right corner of the Inspector.

Note: All objects that do not have a material, will have a default material applied to them the first time the scene is rendered. The DefaultCSMFile entry in the [RenderOptions] section of the Render.ini file controls the default material. If the DefaultCSMFile entry is not found, the material specified by the MiscPlain.csm material file is used.

Applying a material

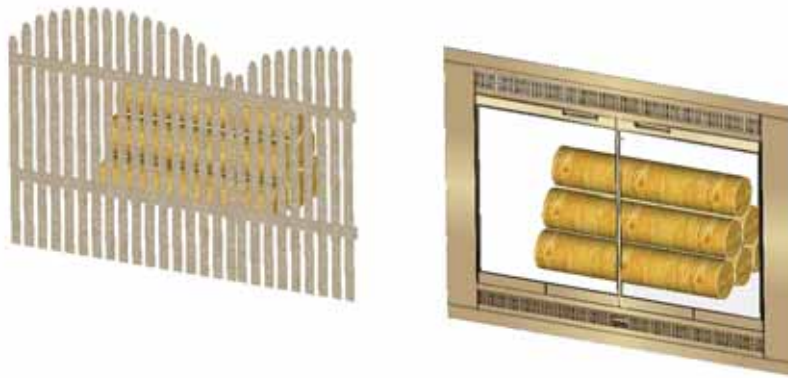
- 1 From the Library Type drop-down, choose Materials. Materials are displayed on the Preview Bar.
- 2 (optional) From the Categories drop-down, choose a category to specify what appears on the Preview Bar.
- 3 Drag the material onto the surface where you want it to appear.

Applying a materials for multiple objects

- 1 Press and hold the Shift key, then select the objects.
- 2 Position the cursor over a preview image and Ctrl-click.
- 3 From the shortcut menu, choose Apply Material to Select Object(s). The material is applied to the selections.

Transparent Textures

The Render Library includes textures with transparent settings. Textures shown with a black background indicate a transparent texture.



Transparent Materials

Texture Folder Locations

Textures are loaded into the Render Library with the program's initial launch. These textures are loaded from two directories: Textures and Custom Textures.

Mac Location

Textures /Library/Application Support/Punch!

Custom Textures /Users/Shared/Punch!

PC Location

Textures Install folder

Custom Textures Install Folder

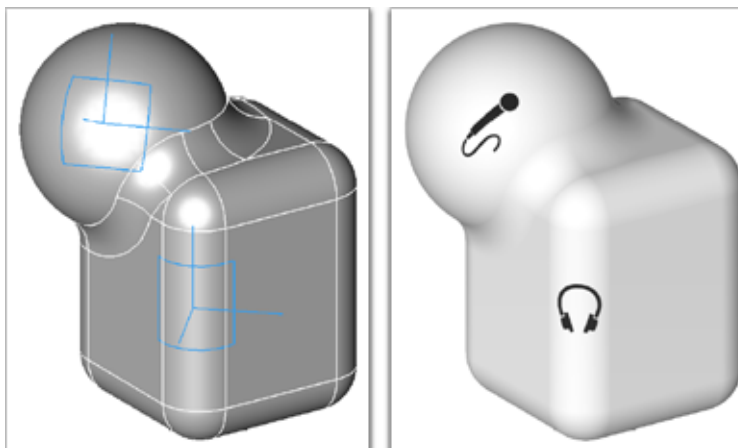
You can add user-defined textures in BMP or JPEG format to these folders. Textures must be divisible by 8 for OpenGL.

Decals Library

The Decals library is a collection of predefined decals that can be applied to renderable objects within a model. A decal overrides the local surface color properties of an object to give the appearance of an appliqué.

Applying decals

- 1 Position the cursor over a decal image.
- 2 Press and hold down the left button, then drag the decal over to the object where you want it to appear.
- 3 Release the mouse button. Any number of decals may be applied to an object.



Example Decals

Decal Wrapping

As a decal is dragged over an object, the decal symbol will snap to the surface of the object of interest, changing shape (planar rectangle, cylindrical patch, and spherical patch) to indicate the decal's wrapping mode. The wrapping mode of the decal is automatically determined by the curvature of the surface at the current drop point.

Planar Wrap

Planar wrap mode projects the artwork along the decal's normal vector. The artwork will appear on the object surface wherever it lands. Planar decals will appear on the opposite side of an object, and drag along any faces parallel to the normal vector.

Cylindrical Wrap

Cylindrical wrap mode projects the artwork along the surface normals of an imaginary cylinder that passes through the decal's location point. Any portion of the object that lies within the cylindrical wedge radiating out from the cylinder's axis through the decal's cylindrical patch will display the artwork. If the decal extends past the "end" of the object, the decal will drag towards the center of the wrap cylinder.

Spherical Wrap

Spherical wrap mode projects the artwork along the surface normals of an imaginary sphere that passes through the decal's location point. Any portion of the object that lies within the spherical wedge radiating out from the sphere's center through the decal's cylindrical patch will display the artwork.

Editing Decals

Once applied, decals are selectable geometric objects. They are edited like all other geometry, by using the Data page of the Inspector. The edit page has controls for settings the rotation, width, height, wrap mode, auto wrap, lock normal, masking, and color.



Decal Info Parameters

Rotation Angle Rotation angle, in degrees, turns the decal about its normal axis.

Width/Height Width and height change the decal's coverage area. The values reflect linear or arc lengths depending on the wrap mode.

Auto Wrap Local Normal

The Wrap Mode drop-down list, the Auto Wrap, and Lock Normal checkboxes control the decal's wrap mode behavior. The wrap mode determines how the decal artwork is applied to the object surface. The wrap mode may be Planar, Cylindrical or Spherical.

If Auto Wrap is checked, the wrap mode is matched to the underlying object's surface curvature at the decal's center point. The computed wrap mode is displayed in the disabled Wrap Mode drop-down list. If the decal is moved to another region of the object, the wrap mode will be recalculated.

If Auto Wrap is unchecked, the wrap mode is set using the Wrap Mode drop-down list. If the decal is moved to another region of the object, the wrap mode will not change.

Masking The Masking drop-down list controls how the decal artwork is applied to an object's surface. Masking can be set to Stencil or Factor.

Stencil Masking Stencil masking uses an image file to define the regions of the decal artwork that will show on an object's surface. The stencil image file is selected from the Stencil drop-down list. The red component (per RGB) of each pixel in the stencil image file is used as a color mix factor. A red value of 0.0 indicates that 0% of the artwork color is used at that pixel location. A red value of 1.0 indicates that 100% of the artwork color is used at that pixel location. Any intermediate value will result in a mixture of the artwork color and the underlying material color.

Typically, a stencil file is composed of fully red and fully black pixels. This provides a clean cropping of decal artwork. However, interesting "dirty" or "aged" effects can be created using a stencil image file that consists of a mottled red patch on a black background.



Example Stencil

Factor Masking Stencil Factor masking uses a single mix factor to compute the artwork material color combination. This is similar to the Stencil masking except there is only 1 value used for all pixels. The mix factor is set using the Factor slider.



Typically, Factor masking is used to put a full image on an object surface. This can be used for rectangular logo artwork or placing artwork on walls.

Logo The Logo drop-down list controls the source of the decal's artwork. Logo can be set to Color or Image.

Color logos use a solid color for the decal artwork. The color of the decal entity is used for the artwork color. The color can be changed using the Pen menu or the Data page in the Inspector.

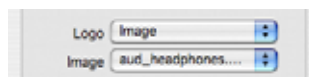
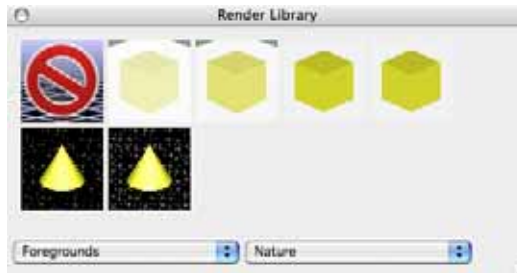


Image logos use an image file for the decal artwork. The image artwork file is selected from the Image drop-down list. Image logos are used for complex artwork decals. This is often used to apply company logos to objects being rendered for proof of concept engineering images.

Foregrounds Library

The Foregrounds library is a collection of predefined foreground effects that can be applied to a model. Foregrounds effect the way the space between the eye point and the scene objects alter the rendering results. Common foreground effects include fog and snow. See the section on Render:Foreground Edit for more detail on the available foreground shader types.



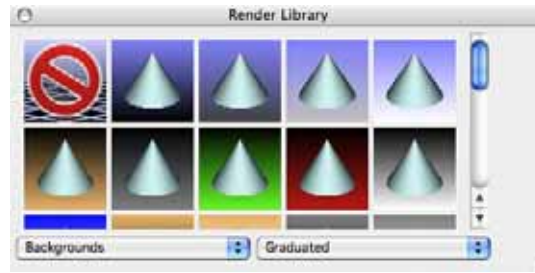
Foreground Library

Applying a foreground

- 1 Position the cursor over a preview image.
- 2 Press and hold down the left mouse button then drag the foreground over to the model.
- 3 Release the mouse button to place.

Backgrounds Library

The Backgrounds library is a collection of predefined background effects that can be applied to a model. Backgrounds control the appearance of those regions of a scene that do not contain objects. Common background effects include 2 color gradients, solid colors, and backdrop images. See the section on Render:Background Edit for more detail on the available background shader types.



Background Library

Applying a background

- 1 Position the cursor over a preview image.
- 2 Press and hold down the left mouse button, then drag the background over to the model.
- 3 Release the mouse button to place.

Lights

Light tool palette

The Light tool palette provides a means to define positional lights. Positional lights contribute light color and intensity to all objects in a scene that lie within their influence. If enabled, positional lights will cast shadows in the Render modes. Lights do not cast shadows in the OpenGL shading modes. Positional lights are created using the light palette toolbar. The Light Palette is displayed by selecting the Window:Lights menu option. The Light Palette contains 3 positional light tools: Distant, Spot, and Point.



Distant Light

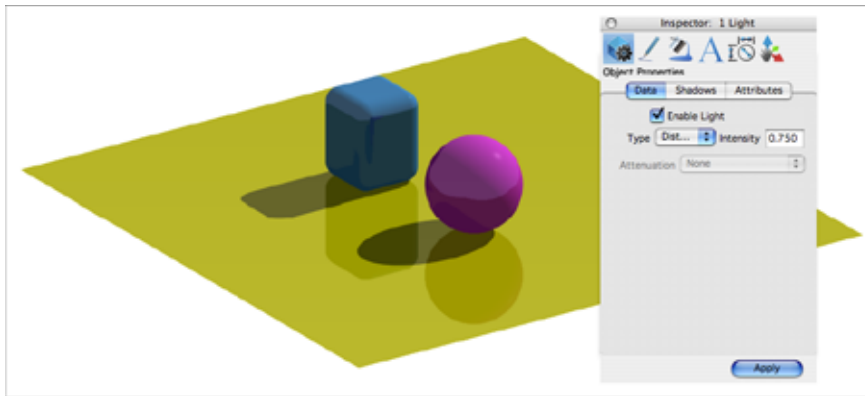
A distant light illuminates a scene with parallel rays of light as if they emanated from a very distant source. The sun is a distant light source. A distant light is defined by two user-specified points that define the light's location and direction. The exact distance is not important. Only the light's intensity and direction is used to determine its contribution to a scene.

A distant light appears as a wireframe cylinder with an arrow pointing along the centerline. If Edit:Show points is ON, the center line is extended from the location point to the direction point.

The distant light Object Properties, in the Inspector, control the setting the lights intensity, activation, and shadow casting behavior.

Using the Distant Light tool

- 1 Select the Distant Light option from the tool palette.
- 2 Click the first point for light location.
- 3 Click the second point for light direction relative to the first point.

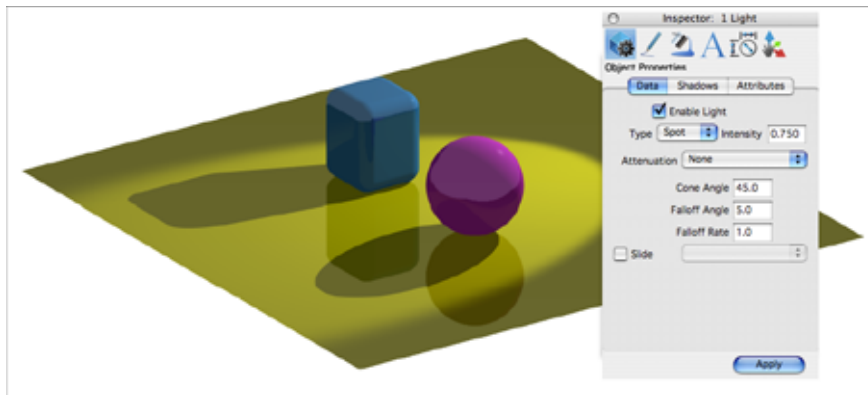
*Distant Lights*

Spot Lights

A spot light illuminates a scene with a cone of light emanating from a local source. A flashlight is a type of spot light. A spot light is defined by two points that specify the light's location and direction. The distance of the light may be important, depending on the light's attenuation setting.

A spot light appears as a wireframe outer and inner cone with an arrow pointing along the centerline. If Edit:Show Points is ON, the centerline is extended from the location point to the direction point and the sides of the cone are extended to a plane normal to the direction at the direction point.

The spot light Object Properties page has controls for setting the light's intensity, attenuation, activation, shadow casting behavior, cone angle, falloff angle, falloff rate, and slide image.

*Spot Light*

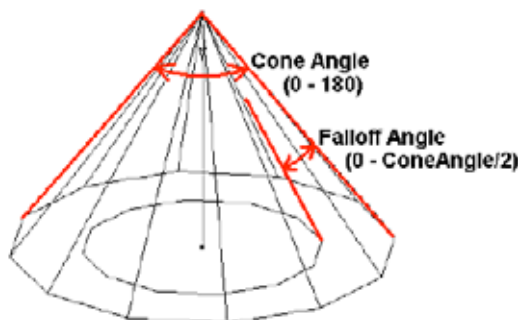
The attenuation controls how quickly the intensity of a light diminishes with respect to distance from the light. Attenuation can be set to one of None, Clamped Linear, Clamped Quadratic, Unclamped Linear, Unclamped Quadratic.



Light Attenuation

None	Light intensity does not diminish with distance.
Clamped Linear	Light intensity diminishes as $I / (d + 1)$.
Clamped Quadratic	Light intensity diminishes as $I / (d^2 + 1)$.
Unclamped Linear	Light intensity diminishes as I / d .
Unclamped Quadratic	Light intensity diminishes as I / d^2 .

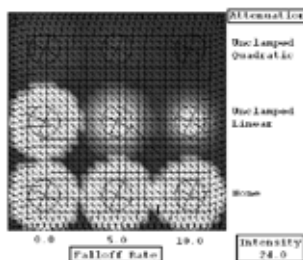
The clamped attenuations are provided for situations where the distance from a light source to an object is < 1 . Clamped attenuations will not increase intensity when $d < 1$. The unclamped attenuations are more realistic and can be used in most situations.



Cone and Falloff Angle

The cone angle controls the maximum spread of the spot light. Any objects outside this cone will not receive any light. Objects within the cone will receive light according to the attenuation, falloff angle, and falloff rate. Cone angle can be any value from 0 to 90 degrees.

The falloff angle controls the sharpness of the spot light's edge. The smaller the angle, the sharper the edge. The light intensity diminishes from full intensity at the inner cone to zero at the outer cone. The falloff angle can be any value from 0 to $\text{ConeAngle}/2$ degrees.



Falloff Rate

The falloff rate controls how light is distributed within the spot light's cone. The intensity diminishes from the centerline of the cone outward, with respect to the cosine of the angle raised to the power of the falloff rate. A falloff rate of 0 results in uniform light within the cone. Higher values produce a more pronounced falloff. The falloff rate can be any value from 0 to 10.

The slide image, if specified, turns a spot light into a slide projector. The image specified is projected into the scene as a slide projector would.

Using the Spot Light tool

- 1 Select the Spot Light option from the tool palette.
- 2 Click the first point for light location.
- 3 Click the second point for light direction relative to the first point.

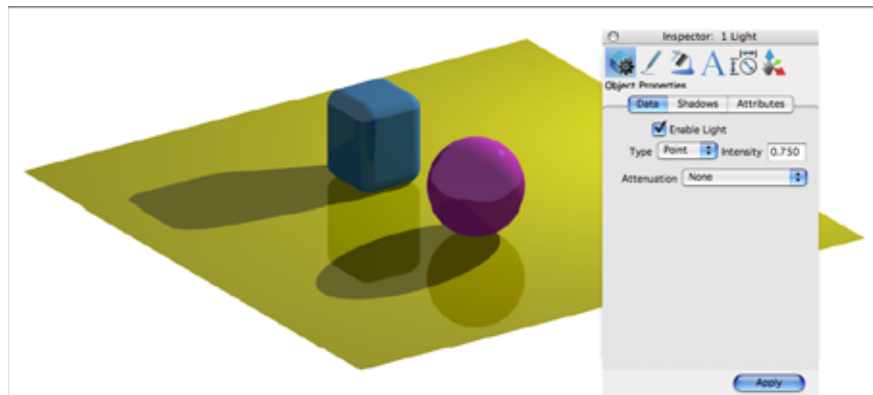


Point Lights

A Point Light illuminates a scene with light emanating in all directions from the light's location. A candle or a table lamp is a type of point light.

A point light is defined by one point which specifies the light's location. The distance of the light to geometry may be important, depending on its attenuation setting. A point light appears as a wireframe sphere with double headed arrows pointing outward. If Edit:Show points is ON, the light's location point is highlighted.

The Point Light Object Properties page has controls for setting the light's intensity, attenuation, activation, and shadow casting behavior.



Point Light and Info Page

Using the Point Light tool

- 1 Select the Point Light option from the tool palette.
- 2 Click the point for light location.



Strip Light

The Strip Light models neon lights, strip lights, and other sources which emit light from a line-like object.

Using the Strip Light tool

- 1 Select the Strip Light option from the tool palette.
- 2 Click the first point for light location.
- 3 Click the second point for light direction relative to the first point.



Area Light

The Area Light allows light emission from an area in space rather than a single point. It can emit light from a surface object or face of a solid.

Using the Area Light tool

- 1 Select the Area Light option from the tool palette.
- 2 Click the face or object from where you want to omit light.



Dome Light

The Dome Light provides a fast approximation of a true sky light. It models the sky as a simple, uniform dome.

Using the Dome Light tool

- 1 Select the Dome Light option from the tool palette.
- 2 Click a point for light location.

Light Color

You can assign pen color attributes to the point lights. The pen color light specifies the color of the light emitted and the intensity. Color values range from 0 to 255 for the red/green components.

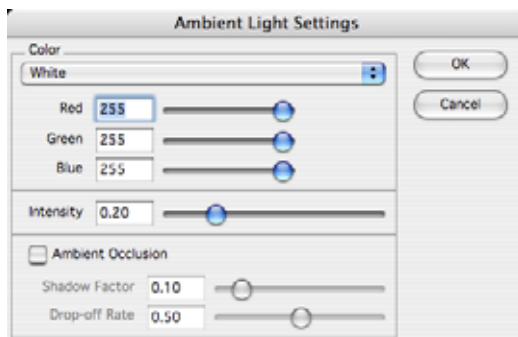
Light Intensity

Each of the positional lights have an attribute for intensity. The intensity setting is located on the Inspector. Intensity values range from 0 (off) to 1 (full intensity).

Ambient Light

The ambient light contributes light color and intensity to all objects in a scene. Ambient light does not cast shadows. Ambient light penetrates into all holes, nooks, and crannies of an object, illuminating all

surfaces equally. The ambient light settings are changed using the Ambient Light Settings dialog box. The Ambient Light Settings dialog is accessed through the View:Ambient Light Settings... menu option.



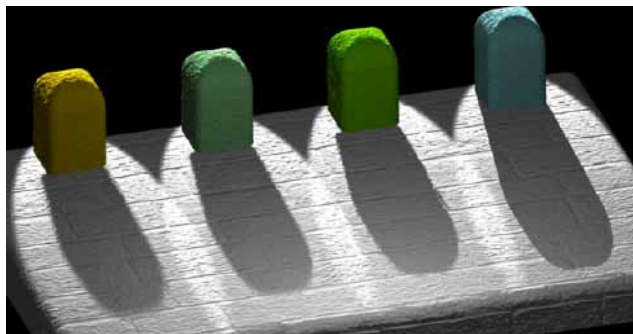
Ambient Light Settings

The ambient color value may be set by selecting a standard color from the drop-down list or by changing the RGB values using the edit fields or sliders. A preview of the color is displayed to the right of the sliders.

The ambient intensity may be set using the edit field or slider. An intensity value of 0 will turn off all ambient light. Ambient light settings are not saved between sessions.

Soft Shadows

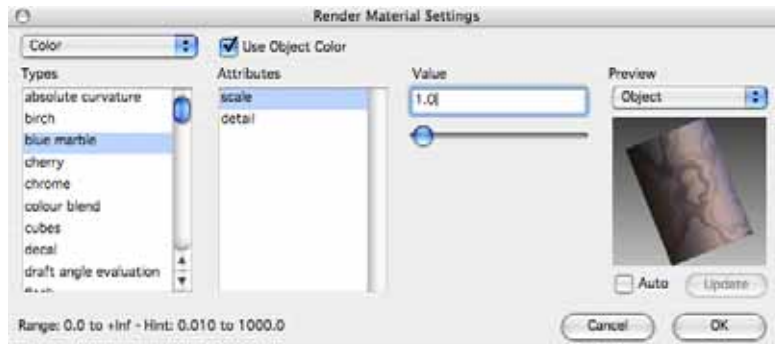
Shadow softness is controlled on a per-light basis in the Inspector. The Shadows page contains a drop-down list with four levels of shadow softness, and a Custom option. The image below shows (left to right) Blurry, Soft, Medium, and Hard shadow softness.



Blurry, Soft, Medium, and Hard shadow softness

Advanced Material Editing

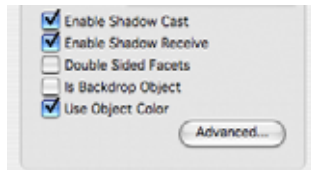
The Advanced Material Editor provides a means to edit existing materials or to create new custom materials. The dialog box provides a pull down menu to specify one of five Shader Classes. Each shader class supports a variety of shader types. Each shader type has a collection of attribute values that define the characteristic of the shader. This implementation presents over 300 rendering attributes for materials. The preview pane shows the immediate impact of an attribute on the material. Select the auto checkbox for the preview pane to update automatically or use the Update button to do so manually.



Advanced Material Editor

There are two ways to access the Render Material Settings dialog for material editing.

- Right-click (Ctrl-click) an object with a material applied and from the Photo Render submenu, choose Edit Material.
- Select an object with a material applied and, on the Inspector, click the Render tab and choose Advanced.

*Advanced Material Button*

Shader Class

The Shader Class displays a pull down menu that contains five classes for defining material Color, Displacement, Reflectance, Transparency, and Texture Space.

Shader Types

The Shader Types list contains all the different shader types for a given shader class.

Attributes

The Attributes list contains all the different attribute types for a given shader type.

Attribute Values

The Attributes value section of the dialog box is the location where you adjust floats, integers, and strings for an associated attribute type.

Preview Sample

The Preview Sample area of the dialog box displays a sample thumbnail image of a shape with the given material settings. Use the object type pull down to change the shape of the geometry used in the preview image.

Color Shader Class

The Color Shader class is used to define the method used to define an object's base color. The color may be calculated from a variety of methods that include curvature, texture, evaluation, wrapped images, wrapped textures, and decals. The example color shader types are listed below

abscurve	False color shader which shows the 'absolute curvature' of a surface.
birch	A version of the general "wood" advanced wood color shader with predefined values appropriate to a birch-like appearance.
bluemarble	A blue marble pattern.
cherry	A version of the general "wood" advanced wood color shader with predefined values appropriate to a cherrywood-like appearance.

- chrome** A color source providing simple chrome-like reflections. The source has a base color, specified by argument “base color”, which is mixed with bands of colors based on the orientation of the surface relative to a direction specified by argument “vector”. A “mix” argument is used to specify the ratio of mixing of base color with the reflection colors. This value should lie in the range 0 to 1, where a value of 1 indicates all base color and no reflection colors.
- colour blend** A color shader that blends two wrapped images or user-specified colors together. Blending an image with a colour is also supported. The input textures or colors are mixed using the drop off factor, which changes with the angle between surface normal and the chosen drop off direction.
- cubes** A three-dimensional lattice of cubes with alternating colors.
- decal** A shader which provides the entry point to support color shade trees. This shader switches between two color input channels. The current ‘layer’ is presented by the “decal color” shader, and the layer beneath by the “base color” shader. The switching between the two is accomplished by the alpha shader global set by the “decal transparency” shader. Both the “decal color” and “decal transparency” shaders are executed using the texture space supplied as “decal texture space”.
- draftangle** Surface evaluation for mold design. The “pull direction” parameter is the direction in which the cast object is supposed to be pulled out of the mold. It will not be possible to pull it out if part of mold “overhangs” the object, i.e. angle between pull direction and surface normal is smaller than 90 deg. If there is a part of surface that has such property then it will be marked with “overhang color”. Next it will be impossible to pull the object out if tangent to the surface is parallel or almost parallel to the pull direction. They must differ by at least “draft angle” (usually 1 deg). So the parts of surface with normals that form angles between 90 deg and $90 \text{ deg} + \text{“draft angle”}$ will be marked with “fail color”. The parts of surface with normals that fall into range between $(90 \text{ deg} + \text{“draft angle”})$ and $(90 \text{ deg} + \text{“draft angle”} + \text{“tolerance angle”})$ can be pulled out, although with some difficulty. So they are marked with “warning color”. Finally, if the surface normal forms any angle between $(90 \text{ deg} + \text{“draft angle”} + \text{“tolerance angle”})$ and 180 deg with “pull direction” everything is nice and those areas are marked with “pass color”. Each of the angles must fall into the range (0, 90) deg and also their sum must fall into this range. Otherwise the default angle values are used. The length of the pull direction vector must be greater than 0.0001. Otherwise the default pull direction is used. Setting 0.0 for “draft angle” results in “fail color” zone disappearing.

Setting 0.0 for “tolerance angle” results in “warning color” zone disappearing.

Setting both angles to 0.0 results in having displayed “pass color” for normal directions opposite to “pull direction” and “overhang color” for normal directions consistent with “pull direction”.

fleck	Color shader which creates a pattern with coloured flecks as seen in some plastics, composite materials or tiles.
gausscurve	False color shader which shows the “gaussian curvature” of a surface.
geomcurve	False color shader which shows the selected type of surface curvature (absolute, mean or gaussian) by false color encoding.
granite	A solid texture giving a granite like pattern.
layered	A shader that combines color from layers defined by color shaders. Any color shader can be used for the base layer, the shaders for the layer above should be ones that allow the background color to show through, such as “wrapped paint splat”.
maple	A version of the general “wood” advanced wood color shader with predefined values appropriate to a maple-like appearance.
marble	A marble pattern.
meancurve	False color shader which shows the ‘mean curvature’ of a surface.
oak	A version of the general “wood” advanced wood color shader with predefined values appropriate to a oak-like appearance.
paving	A color shader creating patterns of pavings made of irregular stones.
pine	A version of the general “wood” advanced wood color shader with predefined values appropriate to a pine-like appearance.
plaincolor	A plain, uniform color.
simplewood	A simple wood pattern with concentric rings of light and dark wood.
solidclouds	A cloudy appearance.
solidpolka	A solid polka-dot pattern.
surfeval	Color shader which simulates reflection of a cylinder with longitudinal bands surrounding the surface being considered. Gives good visual clues to the surface’s curvature.
turbulent	The shader simulates turbulent medium variations (smoke, patchy fog). Although designed for use as medium density defining shader for

“scattering medium” foreground it may be used also on its own for interesting surface effects.

wood

A solid texture corresponding to a wood pattern. The wood is based around the idea of having a tree trunk centered on a given axis with concentric rings of light and dark wood colors. This shader provides more subtle control over the pattern than the more basic “simple wood” shader. Using all the features of this shader it is possible to recreate the appearance of virtually any type of wood. For common types of wood such as oak, maple, birch, cherry and pine, however, specific shaders are provided which simply have the relevant values for the arguments preset (color shaders “oak”, “maple”, and so on). It is recommended that, for most applications, you begin with one of these shaders and modify argument values gradually to achieve the effect you want. The arguments/features of this general shader are presented here so that you can see how to create your own specific types of wood if you desire. The orientation of the tree from which the wood is taken is specified by a point which lies on the axis in argument “trunk center” and a direction vector pointing along the axis of the tree in argument “trunk direction”.

An overall scaling factor is supplied in argument “pattern scale”, which determines the radius difference of two adjacent rings. 0.01 means that there are 100 rings per unit length along the trunk radius. Choose a value appropriate to the units in which your geometry is modeled. The default value is appropriate for modeling in meters. The color of the rings and the wood between the rings is specified in arguments “ring color” (the darker rings) and “wood color” (the lighter colored wood representing spring growth in the tree), respectively. The width of the rings relative to the tree’s trunk is specified by argument “ring width”. 0.0 means no ring at all; 1.0 means that all the trunk area is covered by the ring color. The default value, 0.5, gives a reasonable look for many simple situations. Only values between 0.0 and 1.0 are acceptable. In addition, the “fuzziness” of the boundary between the areas of ring color and standard wood color can be specified using the arguments “ring fuzz in” and “ring fuzz out”. These specify the sharpness of the edges of the inside and outside of the rings respectively (the values actually represent the fraction of the ring width over which the interpolation from one color to the other is done). Values between 0.0 and 1.0 are acceptable. Again, sensible default values have been chosen for these values, which essentially represent common growing cycles in temperate climates (a reasonably quick start of new growth in the spring followed by a slower decline in growth during the late summer). The “gnarl” argument allows random perturbation of the regular rings inside the

trunk. This allows localized “knot” patterns to form, for example. The higher value of the parameter the more “gnarled” the rings pattern becomes.

This shader also accounts for the appearance of random flecks or grains within the wood pattern, and for random perturbations of the ring pattern, which helps give a much more life-like appearance to the wood. The “grain” parameter determines the intensity of the random grain effect. 0.0 means no grain at all, with higher values making the grains more distinct and visible (values in the order of 0.5, 1.0 or 2.0 create reasonable effects). The color of these grains is specified by the argument “grain color”. The amplitude of the high-frequency perturbations of the ring edges is controlled by the argument ring fuzz grain. (the value represents the fraction of the ring width over which the largest perturbations extend – values of less than 0.5 are recommended). Finally, “grain scale” determines the size of the grain in relation to ring size (both the random grain flecks and the ring fuzz grain). Values between 0.1 and 2.0 are reasonable – 1.0 means that the grain size is comparable to the distance between two adjacent rings.

wrappedbirchfloor	A version of the general “wrapped wood floor” color shader with predefined values appropriate to a birch-like appearance.
wrappedbrick	A wrapped brick pattern.
wrappedbrickbonds	An extended wrapped brick pattern.
wrapped brick formation	A wrapper brick pattern.
wrappedchecker	A wrapped checker board pattern.
wrappedcherryflr	A version of the general “wrapped wood floor” color shader with predefined values appropriate to a cherrywood-like appearance.
wrappeddiagonal	A pattern giving a line across the leading diagonal of the texture space.
wrappedfiltereding	A wrapped texture to provide image color mapping with additional color filtering.
wrappedgridcolor	A grid color pattern.
wrappedimagecolor	A wrapped texture to provide image color mapping.
wrappedmaplefloor	A version of the general “wrapped wood floor” color shader with predefined values appropriate to a maple-like appearance.
wrappedoakfloor	A version of the general “wrapped wood floor” color shader with predefined values appropriate to a oak-like appearance.
wrappedpaintslab	Paint splat effect shader that emulates paint dots on a plastic surface.

wrappedpinefloor	A version of the general “wrapped wood floor” color shader with predefined values appropriate to a pine-like appearance.
wrappedpolka	A wrapped polka-dot pattern.
wrapped random image	A shader that returns colour of one of the specified colour shaders (which would normally be images) based on a selector argument. Each of the supplied images can be rotated giving four variants for each image shader to choose from.
wrappedrooftiles	Wrapped color shader simulating various styles of roof tile.
wrappedsstripe	A pattern giving a line along the s-axis of the texture space.
wrappedtstripe	A pattern giving a line along the t-axis of the texture space.
wrappedtexturedbrk	A wrapped texture to textured brick pattern.
wrappedwoodfloor	Reproduces wooden flooring using a number of different plank replication patterns and wood types.

Displacement Shader Class

The Displacement Shader Class is used to define the roughness of an object. Displacement types include 3D space displacements (casting, flat, leather, rough), wrapped displacements (dimple, knurl, leather), and wrapped images. Example displacement types are listed below.

birch	A version of the general “wood” advanced wood displacement shader with predefined values appropriate to a birch-like appearance.
blue marble	A blue marble displacement pattern.
casting	A displacement providing an irregular casting pattern. This pattern is derived from two superimposed noise functions, providing displacements and indentations, with the indentations being used to scale the displacements. The magnitude of the surface displacements, expressed as a factor, typically in the range 0.0 to 1.0, is specified by argument “casting amplitude”, and the amplitude of the indentations that superimpose the displacements is specified by “dented amplitude”. An integer value which controls the detail or complexity of the texture is specified by argument “detail”, such that a value of 1 will give a simple perturbation, while a larger value such as 5 or 6 will produce fine detail in the pattern. An overall scaling factor may be supplied in argument “scale”; increasing this value will make the surface perturbations appear larger. Scaling of the indentations can be controlled separately by the argument “dented scale”. The relative contributions made by the displacements and indentations are controlled by argument “dented threshold”. This value should vary between

0.0 and 1.0, with a value of 0.0 causing maximum interaction. This displacement is suitable for any surface where an irregular roughness is required.

- cherry** A version of the general “wood” advanced wood displacement shader with predefined values appropriate to a cherrywood-like appearance.
- colourdisplacement** A displacement shader which calculates displacement to match a specified colour shader.
- cubes** A three-dimensional lattice of cubes with alternating heights.
- granite** A solid texture giving a granite like pattern.
- link to colour** A displacement shader which creates a displacement to match the material's colour shader.
- local leather** Models surface texture of leather using a solid displacement texture (the basic pattern is formed by a series of more or less rectangular cells). This shader is identical to the “leather” displacement shader, except that it respects transformations.
- maple** A version of the general “wood” advanced wood displacement shader with predefined values appropriate to a maple-like appearance.
- marble** A marble pattern.
- nonedisplacement** No displacement.
- oak** A version of the general “wood” advanced wood displacement shader with predefined values appropriate to a oak-like appearance.
- paving** A displacement shader creating patterns of pavings made of irregular stones.
- pine** A version of the general “wood” advanced wood displacement shader with predefined values appropriate to a pine-like appearance.
- rough** A displacement providing a rough, cast metal-like finish. The appearance of the roughness can be controlled precisely. The magnitude of the surface perturbations, expressed as a factor, typically in the range 0.0 to 1.0, is specified by argument “amplitude”. An integer value which controls the detail or complexity of the texture is specified by argument “detail”, such that a value of 1 will give a simple perturbation, while a larger value such as 5 or 6 will give fine detail of the roughness. Argument “sharpness” is an integer value which controls the sharpness of the perturbations. A value of 1 will give abrupt, sharp changes between the peaks and troughs of the displacements, while a larger value such as 3 or 4 will give smoother transitions. An overall scaling factor may be supplied in argument “scale”;

increasing this value will make the surface perturbations appear larger. This displacement is suitable for any surface where an uneven appearance is required.

simple wood

A simple wood pattern with concentric rings.

simple clouds

A cloudy appearance.

simple polka

A solid polka-dot pattern.

solidleather

Displacement shader which mimics the surface detail of leather using a solid texture. The basic pattern is a series of “cells” of unit size which vary from square to irregular convex shape using the parameter “irregularity” which varies from 0.0 to 1.0. The height of each cell is controlled using “cell amplitude” and the smoothness of the edges is controlled by a smooth step function with the minimum and maximum values set using “smooth min” and “smooth max” parameters.

The shape of the grooves between cells can also be controlled. The parameter “curve amplitude” determines how curved the edges of the cells are, the parameter “curve frequency” determines how wiggly the edges are, with a value of 1.0 indicating one cycle per cell. The detail in the grooves between cells can be controlled using “curve detail”. This varies from 1 to 10 and gives the grooves an increasingly creased appearance.

The bumpiness of the surface can be controlled using two sets of “rough” and “fold” parameters. These add bumps to the surface, the amplitude controlled by “rough amplitude” and “fold amplitude”, the detail controlled by “rough detail” and “fold detail”, and the frequency by “rough frequency” and “fold frequency”. The “rough” parameters refer to high frequency bumps which occur within cells, and the “fold” parameters refer to widespread undulating pattern used to simulate wrinkles in the material.

The number of cells per unit square is determined using the “scale” parameter. When set to one there is on average one cell per unit square. The amplitude of cells scales inversely with the scale factor, hence the more cells per unit volume the lower the amplitude of the cell.

turbulent

Turbulent displacement pattern.

wood

An advanced wood pattern with concentric rings wood with different height. Provides more complex and subtle features than “simple wood”.

wrapped birch floor

A version of the general “wrapped wood floor” displacement shader with predefined values appropriate to a birch-like appearance.

wrapped brick

A wrapped brick pattern.

- wrapped brick bonds** An extended wrapped brick pattern.
- wrappedbumpmap** A bump map displacement derived from an image file.
- wrapped checker** A wrapped checker board displacement pattern.
- wrapped cherry floor** A version of the general “wrapped wood floor” displacement shader with predefined values appropriate to a cherrywood-like appearance.
- wrapped diagonal** A pattern giving a line across the leading diagonal of the texture space.
- wrappeddimple** A regular dimpled appearance.
- wrapped grid** A grid displacement pattern.
- wrapped height map** A bump map displacement derived from an image file with enhanced tiling capabilities.
- wrappedknurl** A knurled appearance.
- wrappedleather** Displacement shader which mimics the surface detail of leather. The basic pattern is a series of “cells” of unit size which vary from square to irregular convex shape using the parameter “irregularity” which varies from 0.0 to 1.0. The height of each cell is controlled using “cell amplitude” and the smoothness of the edges is controlled by a smooth step function with the minimum and maximum values set using “smooth min” and “smooth max” parameters.
- The shape of the grooves between cells can also be controlled. The parameter “curve amplitude” determines how curved the edges of the cells are, the parameter “curve frequency” determines how wiggly the edges are, with a value of 1.0 indicating one cycle per cell. The detail in the grooves between cells can be controlled using “curve detail”. This varies from 1 to 10 and gives the grooves an increasingly creased appearance.
- The bumpiness of the surface can be controlled using two sets of “rough” and “fold” parameters. These add bumps to the surface, the amplitude controlled by “rough amplitude” and “fold amplitude”, the detail controlled by “rough detail” and “fold detail”, and the frequency by “rough frequency” and “fold frequency”. The “rough” parameters refer to high frequency bumps which occur within cells, and the “fold” parameters refer to widespread undulating pattern used to simulate wrinkles in the material.
- The number of cells per unit square is determined using the “scale” parameter. When set to one there is on average one cell per unit square. The amplitude of cells scales inversely with the scale factor, hence the more cells per unit volume the lower the amplitude of the cell.

wrapped maple floor	A version of the general “wrapped wood floor” displacement shader with predefined values appropriate to a maple-like appearance.
wrapped oak floor	A version of the general “wrapped wood floor” displacement shader with predefined values appropriate to a oak-like appearance.
wrapped pine floor	A version of the general “wrapped wood floor” displacement shader with predefined values appropriate to a pine-like appearance.
wrapped polka	A wrapped polka-dot pattern.
wrapped roof shingles	A displacement shader reproducing roof shingles pattern.
wrapped roof tiles	Wrapped displacement shader simulating various styles of roof tile.
wrappedrough	A wrapped displacement providing a rough, cast metal-like finish. The appearance of the roughness can be controlled precisely. The magnitude of the surface perturbations, expressed as a factor, typically in the range 0.0 to 1.0, is specified by argument “amplitude”. An integer value which controls the detail or complexity of the texture is specified by argument “detail”, such that a value of 1 will give a simple perturbation, while a larger value such as 5 or 6 will give fine detail of the roughness. Argument “sharpness” is an integer value which controls the sharpness of the perturbations. A value of 1 will give abrupt, sharp changes between the peaks and troughs of the displacements, while a larger value such as 3 or 4 will give smoother transitions. An overall scaling factor may be supplied in argument “scale”; increasing this value will make the surface perturbations appear larger. This displacement is suitable for any surface where an uneven appearance is required.
wrapped s stripe	A pattern giving a line along the s-axis of the texture space.
wrapped t stripe	A pattern giving a line along the t-axis of the texture space.
wrapped textured brick	A wrapped texture to textured brick pattern.
wrappedtreadplate	A wrapped displacement shader providing a regular tread plate pattern to a material. The indentations may be thought of as cylinders with rounded (spherical) ends that protrude above the surface of the material. The fractional radius of the cylinders and spheres is given by “radius”, which is a fractional value in the range 0.0 to 1.0. Small radii produce longer thinner indentations. A blend may be applied to the edges of the indentations where they meet the flat surrounding surface. The size of this blend is specified by “blend”, which corresponds to a fractional value in the range 0.0 to 1.0. A value of 0.0 produces no blend, and a value of 1.0 results in the whole of the indentation being a blend region.

Two scaling factors may be applied. The “amplitude” parameter applies a scale to the height of the displacements. A value between 0.0 and 1.0 will flatten the indentations, and values greater than 1.0 will accentuate the relative differences in height over the indentation. Values less than 0.0 will reverse the direction of the indentations. An overall scaling factor can be specified by argument “scale”; increasing this value will make the indentation pattern appear larger.

wrapped wood floor Reproduces wooden flooring using a number of different plank replication patterns and wood types.

Reflectance Shader Class

The Reflectance Shader class defines how objects simulate reflected light. This shader models the interactions of illumination with surface reflection properties to calculate the appropriate color of a surface using a reflection model and a shading model. Example reflectance types are listed below.

- blurred conductor** Physically accurate metallic simulation using ray tracing, supporting reflection. Also supports blurring of reflected images.
- blurred dielectric** Physically accurate glass-like simulation using ray tracing, supporting reflection and refraction. Also supports blurring of reflected and refracted images.
- blurred glass** Approximation of glass-like materials using ray-tracing, supporting reflection and refraction and blurring effects.
- blurred mirror** Approximation of mirror reflecting materials using ray tracing, supporting reflection and blurring effects.
- chrome2d** A reflectance model providing a chrome-like effect which generates a reflection pattern from a two dimensional array of colors. The map is projected onto surfaces using a spherical mapping. Reflectance coefficients are provided which allow specification of the amount of ambient, diffuse, specular and ‘chrome’ light reflected, corresponding to arguments “ambient factor”, “diffuse factor”, “specular factor”, and “chrome factor” respectively. The sharpness of the specular reflection highlights can be controlled by means of argument “roughness”. The reflection is made sharper with small values for the roughness, such as 0.1. Larger values, such as 1.0, decrease the specular fall-off. This model is suitable for shiny or highly polished chrome-like materials.
- conductor** A reflectance model that supports secondary mirrored views through ray tracing. Fresnel filtering is incorporated for accurate modeling of light interaction on the surfaces of objects. Reflectance coefficients are provided which allow specification of the amount of ambient, diffuse and specular

light components, and the contribution made by light reflected in the mirror direction, corresponding to arguments “ambient factor”, “diffuse factor”, “specular factor”, and “mirror factor”, respectively. The sharpness of specularly reflected highlights can be controlled by means of argument “roughness”. The reflection is made sharper with small values for the roughness, such as 0.1. Larger values, such as 1.0, decrease the specular fall-off. The conductor’s indices of refraction for red, green, and blue light are given by the arguments “refraction red”, “refraction green”, and “refraction blue” respectively. Similarly, the absorption coefficients for red, green, and blue light are given by “absorption red”, “absorption green”, and “absorption blue”, respectively. This model is appropriate for simulating metallic surface finishes accurately.

- constantreflectance** A reflectance model providing a constant color. This model ignores the effect of all light sources and would give the same result as an environment containing a single, white, ambient source with intensity 1.
- decalreflectance** Reflectance shader that provides support for surface property trees.
- dielectric** A reflectance model that supports an accurate simulation of dielectric (glass-like) materials that have both reflective and transmissive properties. Secondary mirrored and transmitted views are incorporated by the use of ray tracing. Fresnel filtering is incorporated for the specular reflection, the mirrored view and the transmitted view, for accurate modeling of light interaction on the surfaces of objects. Reflectance coefficients are provided which allow specification of the amount of ambient, diffuse and specular light components, and the contribution made by light from the mirror and transmission directions, corresponding to arguments “ambient factor”, “diffuse factor”, “specular factor”, “mirror factor”, and “transmission factor”, respectively. The sharpness of the specular reflection highlights can be controlled by means of argument “roughness”. The reflection is made sharper with small values for the roughness, such as 0.1. Larger values, such as 1.0, decrease the specular fall-off. The dielectric’s index of refraction for all wavelengths of light is given by the argument “refraction”. The default value for the refractive index corresponds to that for glass. The appropriate value for other dielectrics can be obtained from published data in handbooks of optical constants. This model is appropriate for simulating glass surface finishes.
- eye light plastic** Glossy plastic-like reflectance with a fixed eye-light.
- glass** A reflectance model that supports an approximation of glass-like materials that have both reflective and transmissive properties. Secondary mirrored and transmitted views are incorporated by the use of ray tracing. Reflectance coefficients are provided as arguments which allow

specification of the amount of the specular light component (“specular factor”), and the contribution made by light from the mirror (“mirror factor”), and transmission (“transmission factor”) directions. The sharpness of the specular reflection highlights can be controlled by means of argument “roughness”. The reflection is made sharper with small values for the roughness, such as 0.1. Larger values, such as 1.0, decrease the specular fall-off. The index of refraction for all wavelengths of light is given by the argument “refraction”. The default value for the refractive index corresponds to that for glass. The appropriate value for other materials can be obtained from published data in handbooks of optical constants. This model is appropriate for approximating glass surface finishes.

- glossy dielectric** Simulation of a glass-like dielectric surface with colour of the reflected and transmitted light affected by the drop off effect. Also supports blurring of reflected and transmitted light.
- glossy glass** Approximation of glass-like materials using ray-tracing, supporting reflection, refraction, and blurring effects. The colour of reflected and transmitted light is affected by the drop off effect.
- glossy metal** Simulation of a metallic surface using ray tracing, supporting reflection affected by the drop off effect. Also supports blurring of reflected light.
- glossy mirror** Approximation of mirror reflecting materials using ray tracing, supporting reflection. Reflections can be modified by the drop off effect and blurring.
- lit appearance** Physically accurate dielectric-like scattering, combined with a user-defined glow. Gives the effect of a surface which is internally as well as externally lit (such as a lampshade).
- matte** A reflectance model providing a dull matte appearance. Reflectance coefficients are provided to allow specification of the amount of ambient and diffuse light reflected in arguments “ambient factor” and “diffuse factor”, respectively.
- This model is suitable for non-glossy materials such as brick or fabric.
- metal** A reflectance model providing a specular metallic appearance. Reflectance coefficients are provided to allow specification of the amount of ambient and specular light reflected in arguments “ambient factor” and “specular factor”, respectively.
- The sharpness of specular reflections can be controlled by means of the “roughness” argument. The reflection is made sharper with small roughness values (less than 1), such as 0.1. Larger values, such as 1.0, decrease the specular fall-off. This model is suitable for most metallic materials such as steel or brass.

mirror

A reflectance model that supports secondary mirrored views through ray tracing. Reflectance coefficients are provided as arguments which allow specification of the amount of ambient light (“ambient factor”), diffuse light (“diffuse factor”), and specular light (“specular factor”) components, and the contribution made by light reflected in the mirror direction (“mirror factor”). The sharpness of the specular reflection highlights can be controlled by means of argument “roughness”. The reflection is made sharper with small values for the roughness, such as 0.1 or less. Larger values, such as 1.0, decrease the specular fall-off. This model is appropriate for representing mirror-like surface finishes.

multilayerpaint

A reflectance that simulates the effects of multi-layer paint used in the automobile industry. The reflectance effect is achieved by a combination of several different reflectance models into a single shader.

The shader is built using a combination of other reflectance shaders. This means that there will be parameter duplications. This is necessary to allow for individual control of each layer’s properties such as the reflection from the lacquer layer and the metallic layer.

The lacquer layer is based on the “dielectric” reflectance shader. The main modification is the removal of the transmission ray tracing. Instead the transmitted color is calculated by the reflectivity of the underlying layers.

The metallic layer is based on the “conductor” shader rather than the “metal” shader since the former is more physically accurate. This layer is to simulate the metallic flakes present in real metallic paints. A rough displacement shader is used rather than micro-facets to simulate the distribution of the flakes surfaces in the paints. The incident ray is the transmitted ray from the lacquer layer rather than the original incident direction. Likewise, the reflect ray from the flakes will also be refracted by the lacquer layer. This layer is optional and its contribution is determined by the “metallic factor”.

The base layer is the actual color of pigment in the paint surface. As a result, it is assumed to be a Lambertian surface and is based on the “matte” surface. Like the metallic layer, the incident ray is the transmitted ray from the lacquer layer and thus is subject to possible total internal reflection at certain viewing angles. This layer’s contribution is combined with the metallic layer before being filtered by the lacquer layer for the overall color.

phong

A reflectance model conforming with the popular Phong model in which reflections are greatest in the “mirror” direction of a surface opposite the viewing direction with respect to the surface normal.

plastic

A reflectance model providing a specular effect which is similar to the Phong model.

Reflectance coefficients are provided which allow specification of the amount of ambient, diffuse, and specular light reflected, corresponding to arguments “ambient factor”, “diffuse factor”, and “specular factor”, respectively.

The sharpness of the specular reflection highlights can be controlled by means of argument “roughness”. The reflection is made sharper with small values for the roughness, such as 0.1. Larger values, such as 1.0, decrease the specular fall-off. The color of the specular highlights is filtered by the color argument “specular color”.

This model is suitable for shiny or highly polished materials such as plastic or varnished surfaces.

shadowcatcher

Allows shadows to fall upon hidden geometry that is placed in a scene. Usually the supplied geometry will follow some detail depicted by a background image. This then allows objects that have been placed in the scene to cast shadows that appear to interact with the background in a natural way.

translucency

A reflectance model providing a translucent, backlit appearance. The diffuse lighting component is obtained from lights on opposite side of the surface to the viewer, i.e., surface normal is effectively reversed.

Reflectance co-efficients which allow specification of this amount of ambient light and degree of translucency are supplied by arguments “ambient factor” and “translucency factor”, respectively.

This model is suitable for non-glossy, translucent materials.

translucent plastic.

A reflectance model providing a specular effect which is similar to the Phong model with translucency.

Reflectance is as for the “plastic” shader but an additional diffuse term is added which represents the diffuse contribution from lights on the reverse side of the surface. The degree of translucency is controlled by the argument “translucency factor”.

Reflectance coefficients are provided which allow specification of the amount of ambient, diffuse, and specular light reflected, corresponding to arguments “ambient factor”, “diffuse factor”, and “specular factor”, respectively.

The sharpness of the specular reflection highlights can be controlled by means of argument “roughness”. The reflection is made sharper with small

values for the roughness, such as 0.1. Larger values, such as 1.0, decrease the specular fall-off. The color of the specular highlights is filtered by the color argument “specular color”.

This model is suitable for shiny or highly polished translucent materials such as translucent plastic.

- wrappedanisotropic** Approximation of surface with parallel scratches or ridges, such as brushed metal.
- wrappedcircularanisotropic** Approximation of surface with many small areas of concentric rings of scratches or ridges repeated over the surface.
- wrappedmirror** Approximation of mirror reflecting materials using ray tracing, supporting reflection with degree of reflectance determined by red component of an image map.
- wrapper specular map** Approximation of mirror reflecting materials using ray tracing, supporting reflection with the specular and mirror factors determined by image maps.
- wrappedwovenanisotropic** Approximation of woven materials such as satin cloth.

Transparency Shader Class

The Transparency Shader class defines how light passes through an object. Example transparent types are listed below.

- basetransparency** A transparency source shader that copies the input transparency to the output transparency. The input transparency is available if transparencies are assigned to the vertices of polygonal geometry. If transparencies are not available at vertices then the input transparency will be black, implying that the surface is opaque.
- eroded** A transparency shader operating as a stencil that creates the illusion of erosion on a surface. The scale of the erosions is given by parameter “scale”; increasing this value will increase the size of the erosions. The degree of erosion is controlled by the “coverage” parameter, this being a value in the range zero to one where one denotes full coverage (no erosions) and zero denotes no coverage (complete erosion). The edges of the erosions are given a fuzziness whose value is based upon the “fuzz” parameter, which should be in the range zero to one, with larger values producing softer edges to the erosions.
- glow** Transparency shader which varies coverage (alpha) value depending on angle between surface normal and view direction. Useful for creating glow effects when applied to spheroid surfaces (such as spheres or ellipsoids).

Such spheres are placed around a light source, this transparency shader is applied, and the effect is of a spherical glow around the light source.

The coverage can be set for the center (“center coverage”) and the edge (“edge coverage”).

nonetransparency	No transparency: this results in an opaque surface.
plaincoverage	Applies a constant alpha transparency value to a material. Set the parameter “coverage” with a value and the material will be rendered with an alpha transparency equal to that value, e.g., set coverage to 0.5 and the object will appear 50 transparent. Set to 1.0 for opaque and 0.0 for invisible. The transparency is constant across the object.
plaintransparency	A plain, uniform transparency. The transparency is specified as a color filter value by argument “color”. This enables colored transparency and translucency, such as colored glass, to be simulated. Each of the red, green and blue components of the “color” argument should lie in the range 0.0 to 1.0, where a value of 0.0 corresponds to completely transparent, and 1.0 to completely opaque.
wrappedchecktransparency	A wrapped checker board pattern transparency.
wrappedgridtransparency	A grid transparency pattern.
Wrappedimagetransparency	A wrapped texture to provide image transparency mapping.
wrappedmasktransparency	Wrapped image in which the red channel is interpreted as the transparency alpha value.
Wrappedsquaretransparency	A square region of transparency.
x ray	X-Ray like transparency effects.

Texture Space

The Texture Shader class defines how textures are projected or mapped onto an object. Example texture space types are listed below.

arbplane	A texture space in which all points are mapped onto an arbitrary plane. The plane is specified in terms of a point which lies in the plane as argument “origin”, a normal vector perpendicular to the plane as argument “normal vector”, and an orientation expressed as an “up” vector passed in argument “up”. The point passed as “origin” is used for the origin of textures such that the texture origins appear at this point. An overall scaling factor can be passed as argument “scale”. Increasing this value will have the effect of increasing the scale of all wrapped textures which use this texture space. The aspect ratio of the texture space, defined as one unit of its height
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divided by one unit of its width, may be set by the value passed to argument “aspect ratio”. This is interpreted such that the horizontal scale is equal to the value passed as argument “scale”, and the vertical scale is equal to the product of this value and the value passed for argument “aspect ratio”.

- autoaxis** A texture space which selects one of the three coordinate axes (x, y and z-axes) whose plane is most closely aligned with the surface at each point. The points in space are then mapped onto this plane. An overall scaling factor is provided by argument “scale”. Increasing this value will have the effect of increasing the scale of all wrapped textures which use this mapping.
- autoplane** Automatically assign coordinates so that wrapped shaders map along normal vectors of surfaces.
- cylindrical** A texture space in which all points in space are mapped onto a cylinder. The cylinder is specified in terms of a point on its axis in argument “center point”, and a direction vector along the axis in argument “axis direction”. The origin of wrapped textures is specified by means of argument “origin”. Scaling factors are provided in the direction around the axis in argument “scale around axis”, and in the direction along the axis in argument “scale along axis”.
- localautoaxis** A texture space which selects one of the three local coordinate axes (x, y and z-axes) whose plane is most closely aligned with the surface at each point. The points in space are then mapped onto this plane. Operates in the same way as “auto axis” but with axis selection and, thus, mapping performed in the primitive’s local coordinate system rather than in world space. An overall scaling factor is provided by argument “scale”. Increasing this value will have the effect of increasing the scale of all wrapped textures which use this mapping.
- spherical** A texture space in which all points in space are mapped onto a sphere. The sphere is specified in terms of its centerpoint, passed as argument “center point”, and a point which corresponds to the origin of wrapped textures, passed as argument “origin”. The orientation is specified by an “up” vector passed as argument “axis direction”, which corresponds to the upward pointing axis of the sphere. Scaling factors are provided, one in the latitudinal direction (“latitude scale”) and one in the longitudinal direction (“longitude scale”).
- xplane** A texture space in which all points in space are mapped onto a plane of constant x. The positive z-axis is taken as an “up” direction. An overall scaling factor is specified by argument “scale”. Increasing the value of this

factor will have the effect of increasing the scale of all wrapped textures which use this texture space.

yplane

A texture space in which all points in space are mapped onto a plane of constant y . The positive z -axis is taken as an “up” direction. A overall scaling factor is specified by argument “scale”. Increasing the value of this factor will have the effect of increasing the scale of all wrapped textures which use this texture space.

zplane

A texture space in which all points in space are mapped onto a plane of constant z . The positive y -axis is taken as an “up” direction. A overall scaling factor is specified by argument “scale”. Increasing the value of this factor will have the effect of increasing the scale of all wrapped textures which use this texture space.

Animation

The animation tools located in the main tool palette provide commands for generating QuickTime movies by moving relevant camera locations. There are seven options for creating movies that include walk through, fly by, paths, object VR, panoramic VR, and object-based animations. In addition to the camera animations, you can record QuickTime movies of the screen.



Animation Tool Palette

Walk Through

The Walk Through command creates a movie where the camera position follows a curve or collection of curves. The camera eye point is a point on the curve. The camera reference point (look at) lies out on a tangent to the curve at the camera point.

Using the Walk Through tool

- 1 Select the Walk Through option in the Animation menu.
- 2 Pick the curves used for the camera travel path.
- 3 Set the appropriate values in the Animation Dialog.
- 4 Set the appropriate values in the Compression Dialog.
- 5 A progress bar displays as each frame in the movie is calculated, compressed, and saved to a QuickTime file.

Fly By

The Fly By command creates a movie where the camera eye point follows a curve and the reference point is assigned a fixed location.

Using the Fly By tool

- 1 Select the Fly By option in the Animation menu.

- 2 Pick the curve used for the camera travel path.
- 3 Click a location in the drawing that is the fixed camera reference point.
- 4 Set the appropriate values in the Animation Dialog.
- 5 Set the appropriate values in the Compression Dialog.
- 6 A progress bar displays as each frame in the movie is calculated, compressed, and saved to a QuickTime file.



Paths

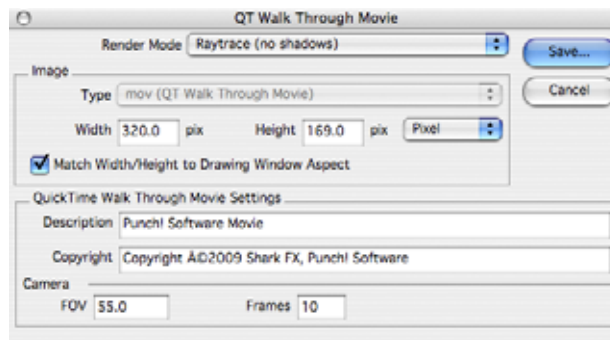
The Paths command creates a movie where the camera eye point follows a curve and the reference point also follows a curve. Positions along the two curve sets are evaluated at equally spaced positions to calculate the camera eye and associated reference point.

Using the Paths tool

- 1 Select the Paths option in the Animation menu.
- 2 Pick the curve used for the camera eye travel path.
- 3 Pick the curve used for the camera reference path.
- 4 Set the appropriate values in the Animation Dialog.
- 5 Set the appropriate values in the Compression Dialog.
- 6 A progress bar displays as each frame in the movie is calculated, compressed, and saved to a QuickTime file.

Animation Dialog

The WalkThrough, FlyBy, and Path animation tools each display a dialog box for details associated with the animation.



Animation Dialog

Render Mode

This pull down menu provides options for the type of rendering associated with the movie. The pull down includes the various preview and raytrace modes.

Image Width/Height

These two fields specify the width and height of the movie. The larger these values, the longer the animation will take to generate. There are four options for determining the width and height of the image: Pixel, 72 dpi, 150 dpi, and 300 dpi. You can choose the option you want from the drop-down menu.

Match Width/Height

This checkbox will force the width/height ratio of the animation window to that of the current drawing window.

Description/CopyRight

User defined strings for description and copyright notices.

Camera FOV

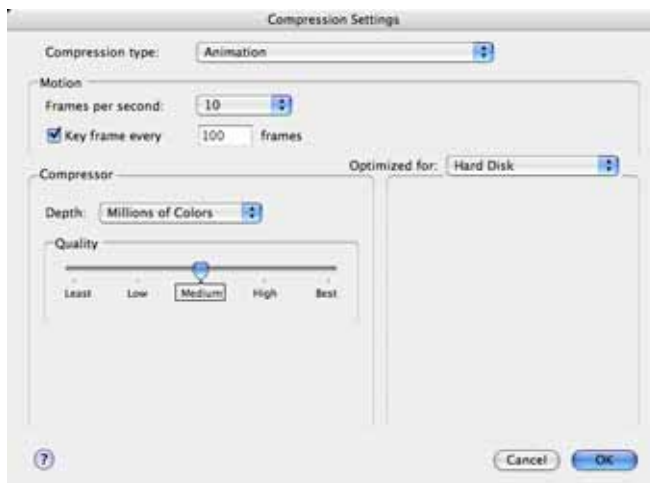
The Field of View relative to the camera eye point.

Camera Frames

The number of frames to generate for the movie.

Compression Dialog

The Compression Settings dialog box is displayed by QuickTime as a means to control the compression options for the animation. This dialog box has options for the user to choose a compression algorithm, and color, quality, and motion settings.



Compression Settings Dialog

QuickTime supplies a variety of image-compression algorithms with the Image Compression Manager. This section discusses some of these compressors and identifies their strengths and weaknesses. You can use this discussion as a guideline for choosing a compression algorithm for your specific situation. All the compressors support both temporal and spatial compression, except for the Photo and Raw Compressors, which support only spatial compression.

For Punch! Shark FX, we typically recommend the Animation Compressor.

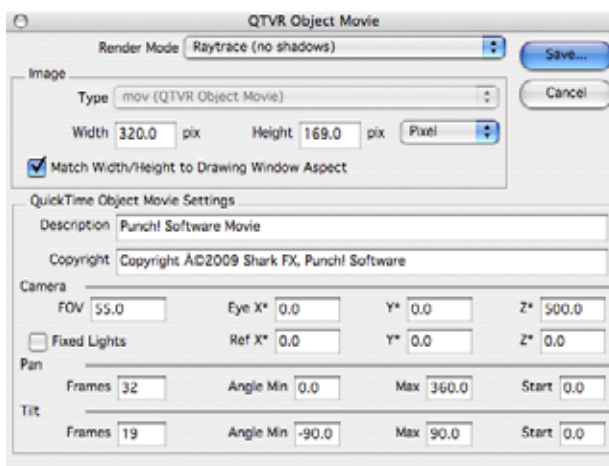
Object VR

This option creates an object-based virtual reality QuickTime movie. Object VR keeps the reference point fixed, but the eye is free to move about. Typically, the reference point is the center of the object, hence the term Object VR. In order to calculate an Object VR movie, many numerous images at different angles are calculated and composed into an interactive movie.

Creating an Object VR Movie

- 1 Select the Animation:Object VR menu item.
- 2 Assign settings in the Object VR dialog box.

The object VR dialog box provides a variety of settings to customize your movie. These settings are described below.



Object VR Dialog

Render Mode

This pull down menu provides options for the type of rendering associated with the movie. The pull down includes the various preview and raytrace modes.

Image Width/Height

These two fields specify the width and height of the movie. The Larger these values, the longer the animation will take to generate. There are four options for determining the width and height of the image: Pixel, 72 dpi, 150 dpi, and 300 dpi. You can choose the option you want from the drop-down menu.

Match Width/Height

This checkbox will force the width/height ratio of the animation window to that of the current drawing window.

Description/CopyRight

User defined strings for description and copyright notices.

Camera FOV

The Field of View relative to the camera eye point.

Camera Eye, Ref

The camera eye point and reference point. Click on the label to activate the field, then use the Snap tool to reference a location in the drawing.

Fixed Lights

The Fixed Light checkbox will keep light sources fixed with respect to the geometry, otherwise the lights move with the camera.

Pan

The Pan Frames value sets the number of images to be generated around the sphere equator (latitudes) for the movie. The extents of the eye point movement (in degrees) and the initial pan angle are set using the Angle Min, Max and Start angle values.

Frames	The number of frames in longitude.
Angle Min	The minimum angle for pan frames. Default 0.
Angle Max	The maximum angle for pan frames. Default 360.
Angle Start	The starting angle for pan frames. Default 0.

Tilt

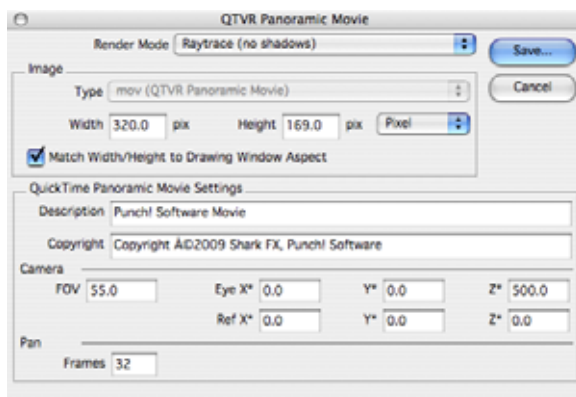
The Tilt Frames value sets the number of images to be generated from pole to pole for the movie. The extents of the eye point movement (in degrees) and the initial tilt angle are set using the Angle Min, Max and Start angle values.

Frames	The number of frames in latitude.
Angle Min	The minimum angle for pan frames. Default 0.
Angle Max	The maximum angle for pan frames. Default 360.
Angle Start	The starting angle for pan frames. Default 0.



Panoramic VR

A panoramic movie keeps the eye point fixed as the observation point is rotated 360 degrees. This gives the visual effect of turning 360 degrees in place. When the QuickTime VR (QTVR) Panoramic Movie type is selected, the QTVR Panoramic Movie Settings dialog box below is displayed.



Panoramic VR Dialog

Render Mode

This pull down menu provides options for the type of rendering associated with the movie. The pull down includes the various preview and raytrace modes.

Image Width/Height

These two fields specify the width and height of the movie. The Larger these values, the longer the animation will take to generate. There are four options for determining the width and height of the image: Pixel, 72 dpi, 150 dpi, and 300 dpi. You can choose the option you want from the drop-down menu.

Match Width/Height

This checkbox will force the width/height ratio of the animation window to that of the current drawing window.

Description/CopyRight

User defined strings for description and copyright notices.

Camera FOV

The Field of View relative to the camera eye point.

Camera Eye,Ref

The camera eye point and reference point. Click on the label to activate the field, than use the Snap tool to reference a location in the drawing.

Pan Frames

The number of frames to generate for the movie.

Object-Based Animation

The Object Animation palette provides tools for animating objects including curves, surfaces, and solids, animating cameras, and recording scenes.

***Set Object Axis***

This tool sets a coordinate system and origin to a part. This is required for object animations that are about a coordinate system other than the global coordinate system.



Set Object Action

The Set Object Action tool creates an animation effect for an object consisting of either a translation or rotation during a specified time.



Set Camera

The Set Camera Action tool defines a camera moving along a path during a specified time.



Animate Scene

The Animate Scene tool will calculate and display a set of frames for a set of object and camera actions. The user can select outputting the results to the screen or to a QuickTime movie.

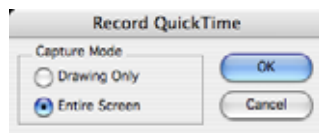


Record QuickTime

The Record Quicktime option is used to time capture screen shots from the drawing, frame or entire screen. The resulting images are compressed and stored for playback as a QuickTime movie. One use of the Record QuickTime tool is to generate user training movies associated with Punch! Shark FX.

Using the Record QuickTime tool

- 1 Select the Record QuickTime menu item.
- 2 Select from the menu, whether you want to capture just the drawing window, the application frame or the entire display/monitor screen.



Screen Capture Options

- 3 Specify the Compression settings and frame rates. The higher the frame rate, the more images captured.
- 4 Specify the file name for the resultant QuickTime movie.
- 5 Select the QuickTime tool once again when you have completed the recording.

Glossary

Accelerators

An accelerator is an application-defined keystroke that provides the user with a quick way to perform a task.

ACIS®

3D modeling kernel developed by Spatial Inc.
<http://www.spatial.com/>

Anti-aliasing

Each pixel in the rendered image is calculated by averaging several neighboring pixels. The resulting image appears smoother.

Arc

A section of a circle limited by a start and end angle.

ASCII

Abbreviation for American Standard Code for Information Interchange. This format is used to store text in most computer systems.

Associativity

An intelligent link between two objects. The link also contains a set of rules that tell how the dependent object should update itself if something changes.

CAD

Computer Aided Design. The process of creating drawings and models on a computer system.

CAM

Computer Aided Manufacturing. A type of computer application that helps automate a manufacturing process.

Conic

A class of curves that contains hyperbolas, parabolas, and ellipses. Conics are popular in the aerospace industry because of their smoothness and flexibility.

Coons' Patch

A NURBS surface with three or four sides.

Coplanar

Geometric objects lying in a common plane are said to be coplanar.

Cover Surface

A surface defined by the outlines of a curve boundary. The resultant surface may be created from 3 or more sides.

Curvature Analysis

Displays a curvature plot on the surface or solid.

Curve

A one dimensional mathematical object that describes length. Example curves include lines, circles, conics, and splines. A curve is a path in 3D space.

DWG and DXF

File format used by CAD system to translate data back and forth to dissimilar systems. This interface will allow Punch! Shark FX the capability to share data with AutoCAD® and other systems which adhere to the DWG and DXF specification.

EPS

Encapsulated PostScript (EPS) is a standard file format for importing and exporting PostScript files.

Flat Shading

A single, constant color across the extents of a facet, rather than smoothly interpolated colors. See Gouraud.

Helix

A curve that spirals in 3D.

IGES

Abbreviation for Initial Graphics Exchange Specification. Used to transfer CAD data to other CAD systems supporting IGES.

Iso Lines

A display option for showing the iso parameter lines for a surface or solid.

Keyboard Shortcuts

Assign a key to a function for accelerating access to a particular command.

Layer

A named collection of model objects. Layers are created and edited using the layer manage tab in the Concept Explorer dialog box.

Line

A straight path in 3D space defined by a start and endpoint.

Links

This is the associative relationship that exists between parent/child objects where a modification to the parent will also modify the child.

Material

A collection of shaders used during rendering of an object's surface that determines an image's pixel color value. The shader types in a material include color, reflectance, displacement, transparency, and texture space.

Moments

Moments or Moments of Inertia, is the name given to rotational inertia, the rotational analog of mass for linear motion. It appears in the relationships for the dynamics of rotational motion.

Net Surface

A surface created from an MxN collection of curves.

Normal

A three component plane equation that defines the angular orientation, but not position, of a plane or surface.

NURBS

Non-Uniform Rational B-Spline. One of several forms available to modeling a curve or surface. The mathematical form describes the shaping and blending information.

Object

A CAD entity that describes a logical grouping for an individual or collection of entities. Example objects include point, line, spline, conic, circle, polyconic, ruled, cone, and sphere CAD shapes.

Phong

A method of rendering that calculates pixel color based on interpolating normals.

Perpendicular

Two lines, vectors, planes, etc., are said to be perpendicular if they meet at a right angle.

Perspective

The appearance of things relative to one another as determined by their distance from the viewer.

Point

A location in model space specified by the 3 coordinates x, y, and z.

Plane

An infinite two-dimensional surface.

RHO

Used in defining a conic, it is the ratio of the center point – shoulder point distance and the center point – slope control point.

Smart Silhouette

A silhouette that displays only if it does not degrade the performance.

Spline

A piecewise parametric curve that consists of a collection of polynomial functions which describe a smooth shape.

Solid

A three dimensional object that describes length, width, and height. Solids describe regions that define volumes. Example solids include sphere, square, cone, prism, and cylinders.

Stitched Solid

A solid created from joining surfaces together.

Surface

A two dimensional object that describes length and width. Whereas a curve spans a length, a surface spans areas. Example surfaces include ruled, polyconic, fillet, and sculptured.

Uniformed Scaling

Scales an object equally in all directions.

Viewport

A viewing region looking into the 3D space of a model.

VRML

Virtual Reality Modeling Language (VRML) “a universal description language for multi-participant simulations.” (from VRML Mission Statement)

Wireframe

A CAD entity that represents length and perimeter. Curves such as lines, splines, conics, arcs, and circles are a class of wireframe entities.

Z-Buffer

An accumulation buffer used in rendering to calculate what object is closest to the viewer.

Zoom

The tool or command that allows you to magnify or reduce an image.

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