

# MODELER



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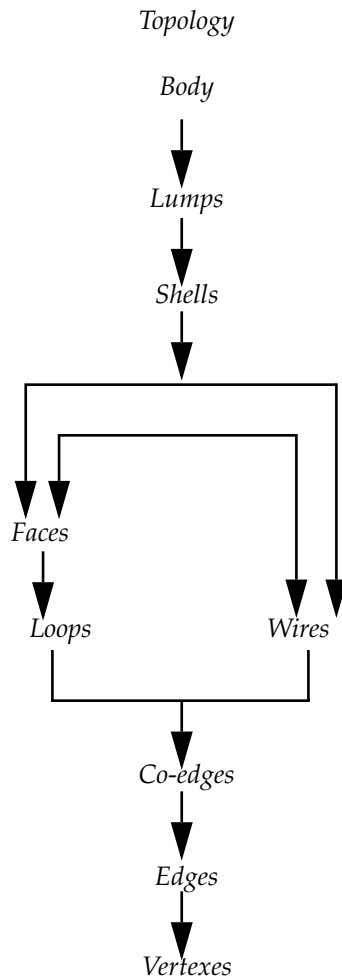
# Topology

## 1.0 Introduction

A key part of being able to build complex shapes in the Universe Modeler is a general knowledge of how the program works with the various elements of an object. The Universe Modeler is built around ACIS technology, and it approaches modeling in a different manner than polygonal modelers. While reading about the various structural elements of shapes may not sound like a lot of fun, especially to someone who wants to dive in and start building, understanding the structure of the model will help you to understand how to use the tools properly, and give you a better grasp of the amazing possibilities the program has to offer.

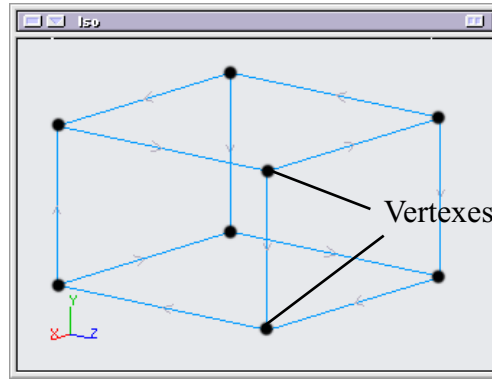
Understanding how shapes are created in the program is to understand the model's topology. Topology is a word you're going to see pop up throughout this manual. Topology is the term used in the program to describe the spatial relations between the various elements used to create a shape. The Universe Modeler stores geometry as boundary representations, and it is the framework of the boundary representation that make up a model's topology.

A vital part of understanding the concept of topology is learning the fundamental building blocks of an object, what their names are and what they represent. Topology and its boundary representations follow a pretty clear hierarchy, from base elements going up to the complete object. Following is the definition of these objects. Digest them in with gusto, you need to know them if you want to tap into the power of the program.



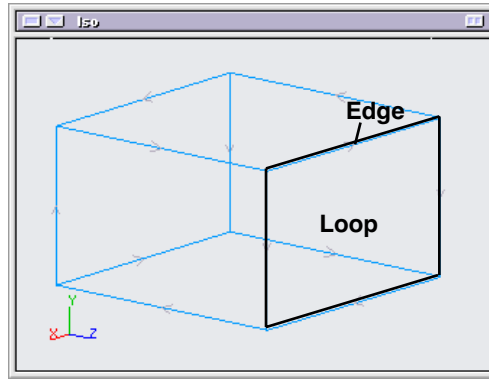
*Figure 1.0 — Graphical Topology Representation*

**Vertexes** A vertex is a corner. It has a position in object space and is owned by the edges that it bounds.

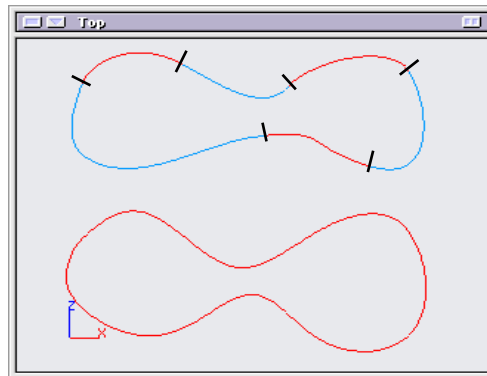


*Figure 1.1 — A cube with 12 coedges has eight vertexes*

**Edges** Edges are curves between vertices. A circle, for instance, is created with a single edge that loops all the way around and comes back to the beginning. A rectangle, on the other hand, is created from a series of four separate edges. Drawing a shape with the Bezier or NURBS curves can create an edge which are geometrically similar, but topologically different when it comes to edges.



*Figure 1.2 — A loop is made from a series of edges*

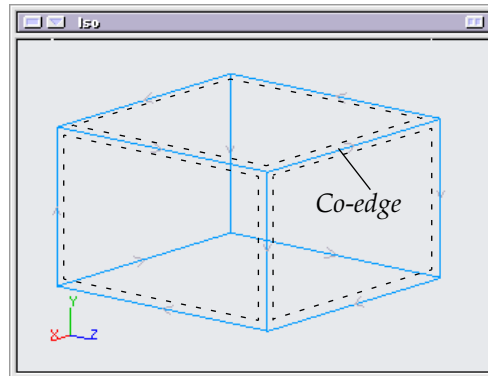


*Figure 1.3 — The top wire was created with the Bezier tool, and contains 6 edges. The bottom wire was created with the NURBS curve and contains a single edge.*

**Coedges** Two loops can share the same edge, creating a co-edge. A good example is a cube, where each edge is shared with two loops and all the loops are cre-



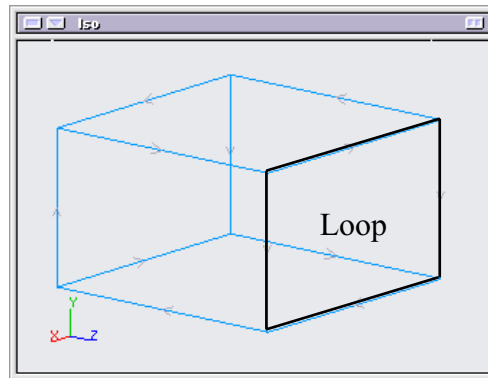
ated using coedges. Coedges are unifying elements that hold together solid and surface objects.



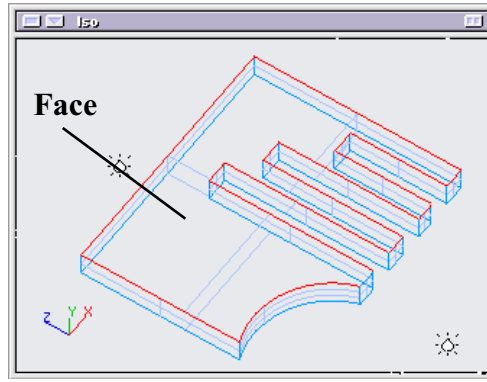
**Figure 1.4 — The dotted lines above show the loops of edges that form coedges where a single edge is used by two loops.**

- Wires** A wire is a connected collection of edges that are not attached to faces and do not enclose any volume. Wires may represent abstract items like profiles, construction lines and center lines. They are also commonly used to form wire frames to be surfaced to form solid-bounding shells.
- Loops** A loop represents the connected portion of a boundary of a face. It consists of a set of coedges linked in a chain which may be circular or open-ended. To be complete, a loop must use either a single edge, or a series of edges that form a closed circuit. If the circuit is open, the loop is incomplete and cannot be filled for create a face.
- Face** The boundary of a face is made up by loops of edges. Each loop determines a portion of the surface that is inside the face and a portion that is outside the face. A faces sidedness refers to whether a face is single sided or double sided, and if single sided, which direction the face is going. Viewing the

sidedness of an object can be done by going to the Edit menu and pulling down to the System Preferences and then going to the Display tab and toggling on “Show face directions.” Certain shapes require certain types of sidedness, and while you build your shapes you’ll want to keep track of the structure of your object.



**Figure 1.5 — One face of this cube is defined by the loop, as shown above. This cube has six loops that define the boundaries for six faces.**



**Figure 1.6 — The top face of the object above is bounded by a continuous loop.**

The “direction” of a face refers to the direction that the normals of the face are pointing.

It is important to note the difference in thinking between polygonal modeling and ACIS modeling when it comes to defining a face. In the polygonal world, a single polygon is represented by a face. In ACIS modeling, a face isn’t a polygon, but a filled area bordered by loops.

A face isn’t necessarily flat, but can have undulations, therefore a single face can have the ability to contain a good deal of information about the shape of an object.

### **Single-sided, forward Sense**

The face bounds a solid region. The inside of the face is solid and the outside face is void. Any solid primitive would be an example of this type of sidedness.

### Single-sided, reverse sense

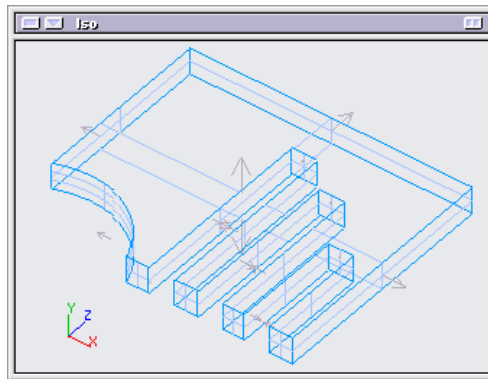
The face bounds a solid region (either completely or partially). The inside of the face is void and the outside of the face is void.

### Double-sided, both inside

The face is embedded in a solid. Both sides of the face are solid. Normals point to the inside.

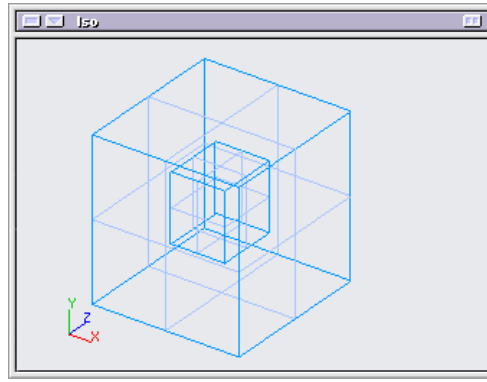
### Double-sided, both outside

The face is a 2D region. Both sides of the face are void. Normals point outside.



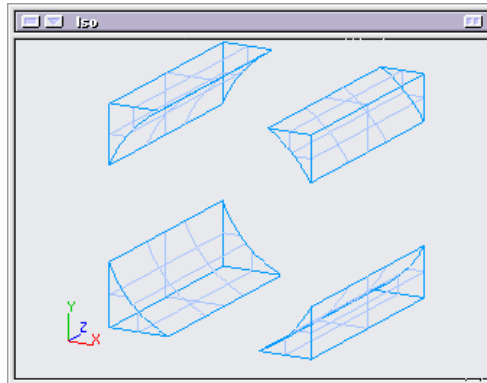
**Figure 1.7 — The directions of a face can be displayed on the screen by going to the System Preferences and toggling on “Show Face directions under the Display tab”.**

**Shells** A shell is an entire connected set of faces and/or wires. A solid block has one shell, while a block with a cavity would have two shells. The most common type of shell is made up only of complete, finite single-sided faces, two of which meet at every edge, with compatible insides and outsides.



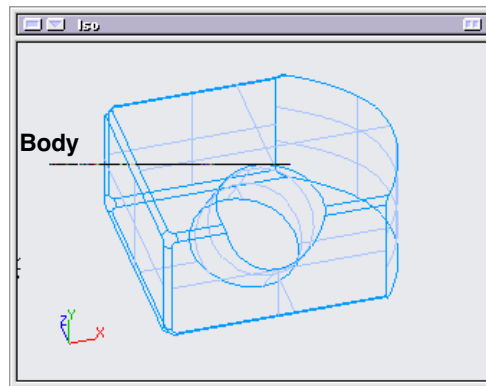
*Figure 1.8* — This block has had a second block removed from the inside forming a cavity. This shape has two shells, one on the inside and one on the outside.

**Lumps** A lump represents a bounded, connected region in space. A lump is an entire connected set of points.



*Figure 1.9* — The shape above was formed by removing a cylinder primitive from a block primitive. The result is a single body made up of four lumps.

**Body** This is the top level of an object's topology. A body is a single shape that is made up of all the other topological elements. A body can contain zero or more lumps, each of which represents a set of points that are disjoint from those represented by all of the lumps of the body. A single scene may have a single body, or hundreds of bodies. If you're building a model of an automobile, for instance, single shapes such as the tires, hub caps and seats would all be individual bodies. Each of those bodies would be made up of lumps, shells faces and so forth.



*Figure 1.10* — A body is the complete collection of vertices, edges, shells and other topological elements that form an object.

### 1.1 Topology Creation

By using the tools in the Universe Modeler's toolbox you create these different types of topology indirectly. By drawing a curve with the Bezier tool, you create a wire body. By using the Fill tool, you create a face, making a surface body, and by using the Extrude tool you create a series of co-edges, loops and faces that form a solid body. Knowing that you created these, and understanding how they function, will allow you to go further into the object and manipulate it to create complex forms.

## Quickstart

Nothing is quite as frustrating as having a great new toy, but not knowing how to use it. It is highly recommended that you sit down with this manual and, as mundane as this may sound, read it. Some new conventions are presented by the program and it is important that you familiarize yourself with them.

Being realists, we know most of you will use the manual only once you've hit a wall and need to figure out how to perform a certain function. To that end, we have created this Quickstart section. By going through this section, you will learn to:

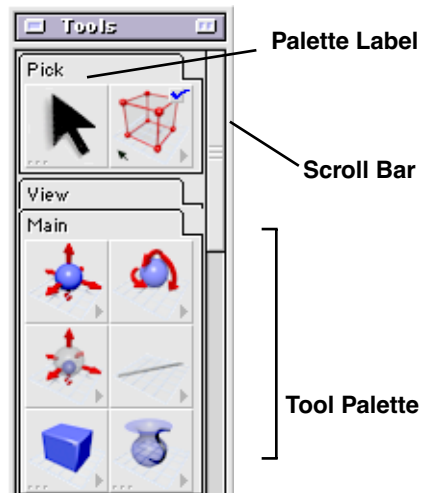
- Work with the toolbox
- Manipulate windows
- Create primitive objects.
- Move the objects.
- Move the camera around the objects.
- Create a 2D primitive shape with the NURBS tool.
- Extrude that 2D primitive shape.
- Create a 2D primitive shape with the polyline.
- Revolve that 2D shape around an axis.
- Remove a portion of the extruded shape from the revolved shape.
- Blend the edges of the new form.

By the time you finish this section, you will have a good feel for how the program works. The Quickstart guide is designed to give just a fleeting glance at the program. Step-by-step tutorials are provided for each tool later in the manual.

### 2.0 The Working Environment

The window set will look familiar to ElectricImage Animation System users. The default window settings give you four orthographic windows from the top, side and front; the fourth window on the top right of the screen is the iso view. On the left side of the window is the toolbox. All the tools you will need are nested here. On the right side of the toolbox is a scroll bar that can be used to slide the toolbox up and down to gain access to tools outside of the current view. Users also can click in the areas above and below the sliding bar to quickly expose hidden tool areas.

### 2.1 The Tool Box



*Figure 2.0 — Tool Box*



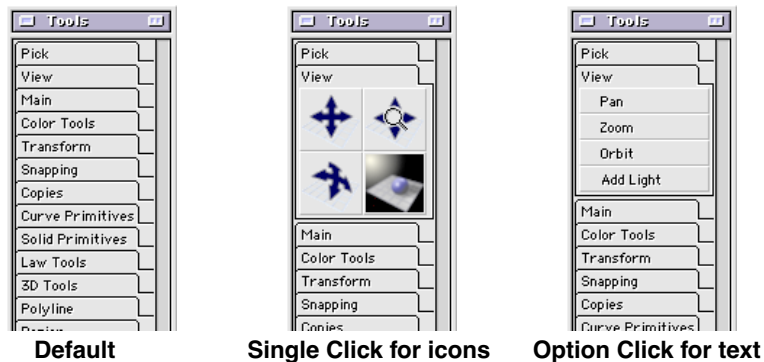
All the tools are available on this single palette and are represented by icons. Tool groupings are separated by folders in the toolbar. To make navigating easier, more experienced users may customize the toolbar in two ways. They are:

- Change the icons to text.

*To do this hold down on the option key (PC- Alt key) and click on the label of the tool palette, for example, View. The icons will be replaced by a more condense text version while all the features of those tools will still be available.*

- Hide the tool palettes altogether.

*To do this, click on an opened tool palette label. This will leave just the name. If you weren't interested in Snapping or Modeling aids, you could click on these to give you more room for the tools you want quick access to.*



**Figure 2.1 — Tool box options**

All the tool palettes can tear-off from the main tool bar. This allows you to custom-design your own tool preferences. With a two-screen setup, for instance, you can tear off favorite tools and place them so they would all be visible on the 2nd screen, leaving the main screen open for the window views. Further palette editing can be done by control-clicking on the tool palette.



Figure 2.2 — Torn off palette

By using numeric keys, you can move through the tools in the most recently used palette without having to click on them. By pressing the + or - key you can also scroll through a palette. This illustration shows the flow of tools selected by the key strokes.

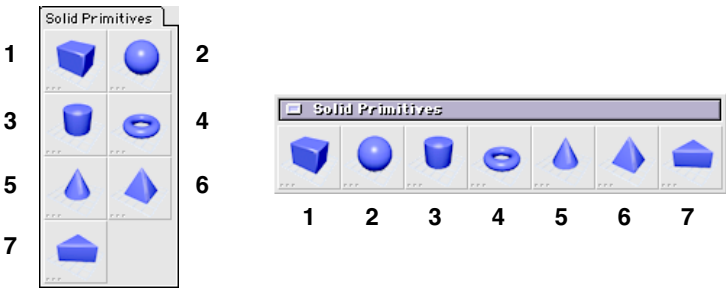


Figure 2.3 — Numeric tool shortcuts

Other existing shortcuts, (*PC- Ctrl + M*) Cmd + M, Free Move, (*PC- Ctrl + P*) Cmd + P, Pick tool, and (*PC- Ctrl + R*) Cmd + R, Free Rotate, won't change the selected palette.

These shortcuts will only work when the focus is in one of the 3D views. when you have placed the focus in the status box, the numeric keys will be used to enter values. Click the mouse in the title bar of one of the 3D views to return the focus there.

## 2.2 Getting Around in the Tool Box

Because of the great number of tools available in the Universe Modeler's toolbox, it would be impossible to display them all, even in small size or in text views. To get around through the toolbox you can scroll by either clicking and dragging on the vertical scroll bar on the right side of the tool box, or you can hold down the space bar and "push" the toolbox up or down. To help relieve the tedium of getting through the toolbox, the most-used tools in the toolbox have been placed in a separate palette called Main. In this window you will find quick access to tools that will allow you to move and rotate objects, access the line tools, create primitives, extrude shapes, perform booleans, create and edit UberNURBS and to delete objects. Each object in the Main election contains a sub-menu of additional features for each tool which can be accessed by simply holding the mouse down on each icon. Like all the other palettes, these can be torn off. Another way to quickly access often used tools is through the menu items.

## 2.3 Shrinking the Tool Box

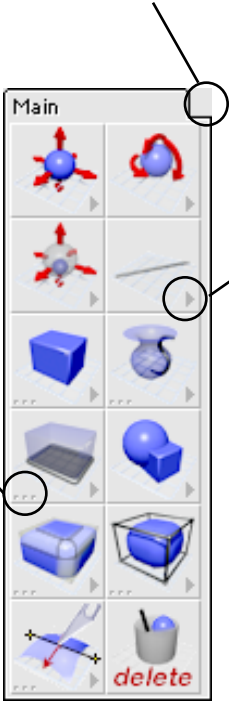
You may find that the toolbox takes up more real-estate than you would like, or that scrolling through the toolbox takes too much effort. One way to simplify the toolbox is by shrinking the size of the entire box.

To do this, click on the top right “Expand/Contract” button. This will make the icons in the tool bar almost half their normal size.

Icon views for the entire toolbox can also be quickly collapsed to a text list by double-clicking to the right of the text tab.

**Double-clicking on the null space to the right of a tab changes the entire toolbox from icons to text, showing only the palette names. Repeating the double-click reverts it back to icons. Holding down the option (PC -Alt) key and clicking here will change all of the tools to text. Option (PC -Alt) clicking again reverses the effect.**

The three dots “...” on the lower left hand corner of a tool button indicate that options are available in the tool settings. Double-clicking on the tool will bring up an options box.



The triangle in the lower right hand corner indicates that there are more tools of same nature available. Clicking the mouse and holding will show the additional tools. These tools can also be torn off to create their own palette.

Figure 2.4 — Getting around the toolbox

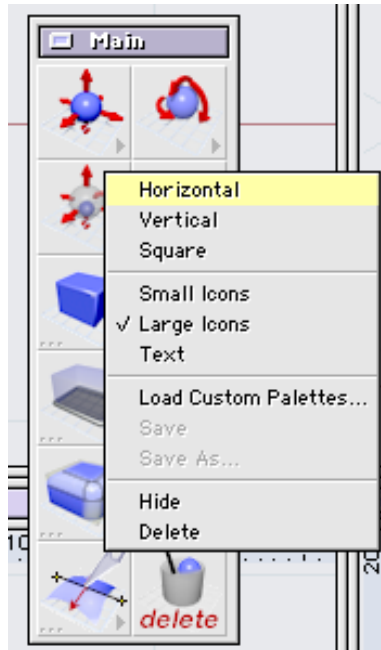
**Tool Options** Many tools have additional option settings. Double clicking on a tool with additional preferences will bring up a dialog box. These tools are indicated by “...” in the lower left corner. A discussion of the settings of each can be found in the following chapters associated with each tool.

### Multi-option tools

Tools that have a little grey arrow in the lower right hand corner indicate that a series of tools that work in the same family can be found by either holding the mouse down on the tool, or by tearing it off.

## 2.4 Contextual Toolbox Menus

Many windows in the Universe Modeler have contextual menus. This means that by holding down the control key (*PC- right-clicking*), and clicking in any window, you will bring up a pop-up menu selection. This includes the palettes where the menus reside. The options for the palettes tools vary, depending on whether the tool is resides in the main toolbox, or has been torn off menu. The toolbox contextual menu will let you choose small or large icons, as well as save and load user tool palettes. A torn off menu will give you the same options, but it will also let you make the palette vertical or horizontal and will let you hide or delete the palette. You can tell if a window has a contextual menu by holding down the control key and passing the cursor over the window (*PC- right clicking*). If the cursor arrow changes to the inverted triangle, that window has a contextual menu associated with it.



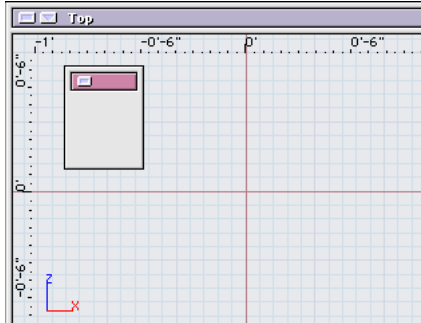
*Figure 2.5 — Contextual menu options*

## 2.5 Creating User Palettes

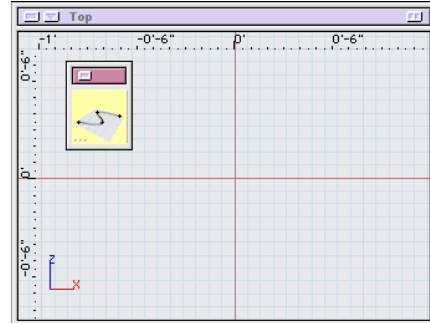
Every person using a modeler will have particular ways of working, and choices in tools that they prefer. The Universe Modeler program will allow you to design your own custom tool palette, or several palettes, and will remember how you placed them on the screen, so that you can call up your own set of options simply by loading your own user palette.

**In short, you can create your own user palette in a few easy steps.**

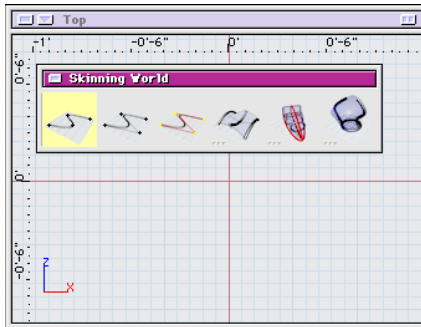
1. Go to the Windows menu, pull down to User Tool Palette.
2. Give your tool palette a name
3. Hold down the shift-key and drag the tool of your choice on top of your new tool palette. Let go of the mouse button.



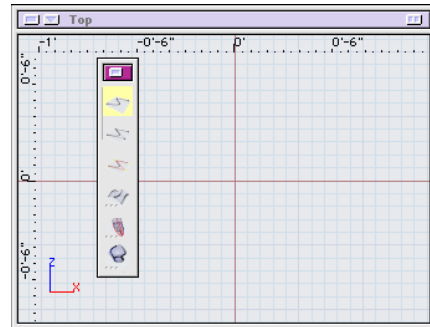
**Choose the User Tool Palette option under the Windows menu. After naming it, an empty palette will appear.**



**Hold down the shift-key, and drag tools from the main tool box into the new palette.**



**Add as many tools as you like; drag them over the position where you want them and release the mouse.**



**Use the contextual menus to further customize the palette.**

*Figure 2.6 — Creating a custom tool palette*

4. Continue to add as many tools as you like. You'll see a purple line that will indicate the placement of the currently chosen tool to place into the palette. The tool you're adding will show a blue line around it, if a blue line does not appear, that means the tool you are attempting to add is already in the palette.

If you wish to remove a tool, option-click (*PC- alt-click*) on the tool.

In addition to this, you can also create a unnamed customized palette by shift-dragging any individual tool off the tool palette and adding more tools to it by shift-dragging.

You can then use contextual menus to shrink your icons, or change them from horizontal to vertical if you wish. You can also place the palettes anywhere on your screen.

### Saving and Loading User Tool Palettes

After creating your new palette, or series of palettes, you can save the settings so you can use them every time you run the program. This allows several people to have their own preferences, and not have to re-design their favorite tool layout every time they start the program.

There are two ways to save the current palettes displayed on your screen. You can go to the **File>Tool Palettes** menu option and the **Save User Palette** or **Save User Palette As...** submenu. You can also save it using the **contextual menu**, hold down the Control key (*PC- right-click*) and choose the **Save Palette** option.

The resulting file will be saved wherever you indicate on your hard drive. Like any other file it can be moved, copied and even uploaded wherever you want.

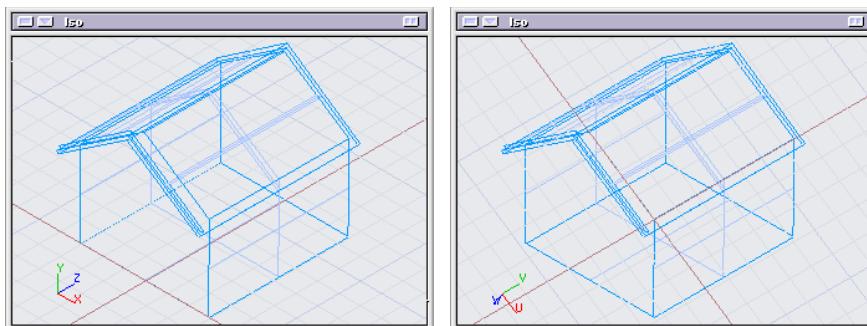


To load the files you also have two choices, you can do it through the **File>Tool Palette** menu option.

### 2.6 Workplanes

The program has a default workplane already set up for you. It lies on the XZ plane. The workplane is a user-definable grid that can serve as an important device for the construction of objects. While the default plane is useful, there will be times when you need to create an object that lays flat in the default grid. By altering the workplane you'll be able to constrain and snap to any number of directions. If you were working on a model of a slanted roof, for instance, and you wanted to use a workplane for exact placement of the vents on the roof, you could tailor the workplane to your needs and use the resulting grid for the measurement.

The tool is simple to use and requires the simple selection of new horizontal and vertical elements, like the edges of the roof below, to determine the placement and orientation of the new workplane.



The default workplane exists on the XZ plane. While this might be useful in many cases, such as determining how wide and long to make the house above, it does little in providing accurate information on the roof because of the slope.

By choosing the Workplane tool, click on the side and front edge of the roof, and click the Null space. A new workplane is quickly established. Workplanes can be saved in a scene and opened up at any time.

*Figure 2.7 — Creating a custom workplane*

## 2.7 Working with the Windows

The Universe Modeler window setup is based on the unique premise that a window is just a window, what kind of window it is going to be is completely user definable. When you start up the program you'll see four windows that are common in most 3D modeling programs. These are set up as the original default windows, and many users may choose to never alter them. Any window can be used for any view, and any window can also be used to edit the layer system. Users can create as many or as few windows as they want for any number of views. Each window can be positioned anywhere the user desires, simply by dragging the window to a new location.

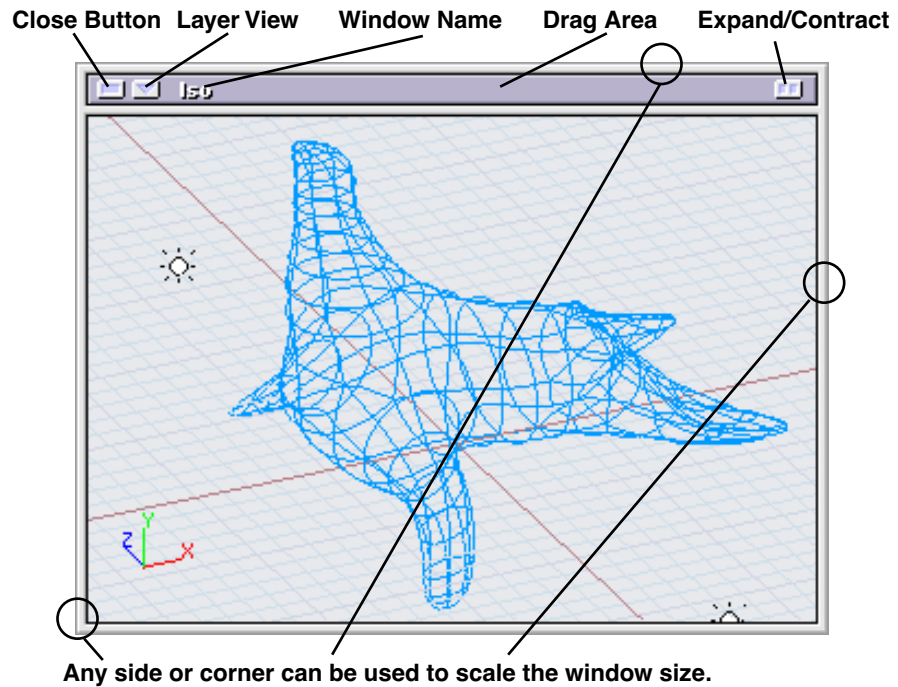


Figure 2.8 — Parts of a window

### Parts of the Window

**Close Box.** Clicking on this closes the box

**Layer View.** Clicking on this takes you to either the World View, Layer View, UCS view or the View Manager.

**Window Name.** The current view name is placed in this window.

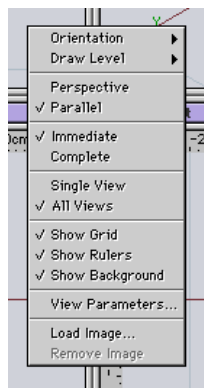
**Drag Area.** Click and drag here to reposition the window.

**Expand/Contract.** On world views this will make the window the size of the current screen, clicking again will bring it back to it's normal size.

**Status Window.** On the lower left of the screen you'll find the Status Window. While using the program's tools, the status bar will walk you through steps and will indicate any errors in the process. On right side of the bar is a pop-up window for Relative or Absolute. This will be reflected in an objects change of position, whether it is relative to its former position, or in relation to the to the world coordinates, or absolute.

### Window Options

Each window can be custom designed by a host of user options. A contextual pop-up menu will come up when the control key (*PC-Right Click*) is held down and the mouse button is held on any window. The options control how elements in the window are viewed, with variables for shading, grids, backgrounds and orientation.



**Control + clicking (*PC- Right Click*) in any window brings up the window's options.**

*Figure 2.9 — Window options*

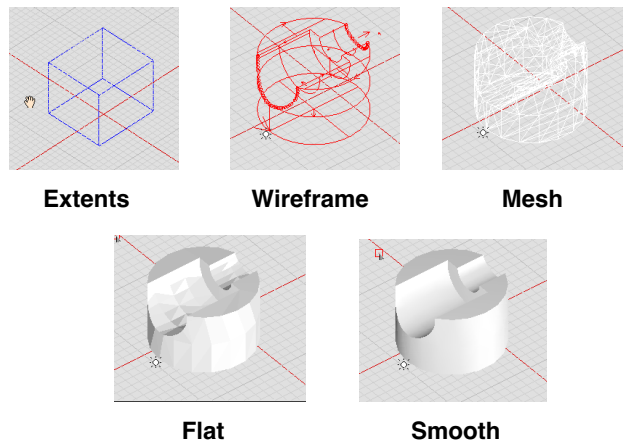
### **Orientation**

Each window can be viewed by the Top, Front, Left, Right, Back, Bottom or Iso. The Iso view is a flexible user-definable view that allows the user to freely orbit around an object. To orbit you can use either the Orbit tool, or hold down the “O” key and click and drag on any window. Even windows currently labeled as Top, Front and Left by default can be made to orbit.

### **Draw Level**

The user can set up the shade view for any window, whether as a working view or to quickly check the model. This allows the user to hop from a wireframe view while building a model to a smooth shade to review the work, without having to move the mouse up to the Menu bar, or to remove your hands from your mouse to type in a key command to preview.

## **2.8 Draw Quality**



*Figure 2.10 — Draw Quality Types*

- Extents Only

Only the boundary box information will be included. While this is the fastest way to navigate around complex models, it also offers the least amount of information.

- Simple Wireframe

In this mode the basic geometry of the object as shown by its “isoparms” is displayed. This is a good working mode for complex models.

- Mesh

This mode will give you a good feel for the surface space of an object. If you’re editing objects and you want to know where surfaces start and stop, but still need the “see through” freedom of a wireframe, this is the setting for you.

- Flat Shade

This shader will give you instant feedback on overall lighting and surface interaction.

- Smooth Shade

Since most of your final images will be smooth, this quickly drawn Phong shader gives you more precise feedback on your final image without the distracting facets of flat shading.

## **2.9 Window View**

### **Perspective**

Any window can be viewed with perspective. For most applications users will use the perspective setting in the iso view, or in a new window, to check the appearance of a scene.

### **Parallel**

This setting will give no perspective to a scene, but will offer a straight orthographic view. This setting works well for constructing objects while viewing from the top, front or side views.

## **2.10 Interactive Drawing**

Below the screen shade modes are Immediate and Complete. This effects the screen feedback during translations, panning, zooming and orbiting.

### **Immediate**

If Immediate is selected, when the user navigates or moves an object or moves the view, the objects in the view are shown in the extents mode. This allows for smoother motions with more complex objects.

### **Complete**

When Complete is chosen the shade view for the window remains. For instance, if you are viewing a window in Smooth shade mode and you zoom in, during the zoom the object would remain a smooth object. Users can control the feedback mode without going into the windows options pop-up box by holding down the control key (*PC-Right click*). If you are in Immediate working mode, holding down the control key (*PC-Right click*) will force the screen into Complete mode. If you are in Complete working

mode, holding down the control key (*PC-Right click*) will force the window into Immediate mode.

### Single view

With this option chosen, any transformation that is done to an object such as moving it, will be apparent only in the working view until the function is complete. Since only one window needs to be updated, this is a faster working mode.

### All Views

With this option chosen, any transformation will be automatically updated in all the views. This feature is important when aligning objects, since you get feedback from other views.

**Note** *“Immediate/Complete” and “Single/All views” are settings for the viewing engine and not of each view separately. Therefore, if you change the setting from Immediate to Complete, it affects ALL views, not just the one in which you set the options.*

### Show Grid

Users can toggle the grid on and off using this pop-up selection. Settings for the grid can be found under the **Preferences** menu.

### Show Rulers

Users can toggle the rulers on and off for each window view using this selection.

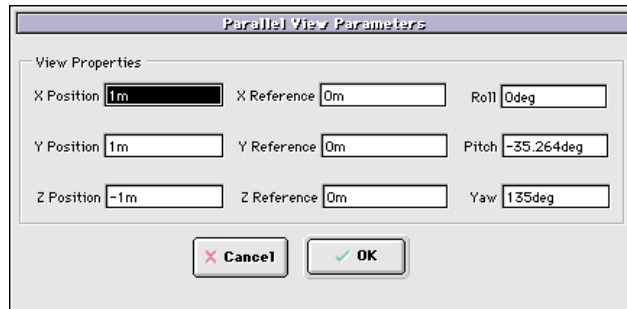
### Show Background

Users can toggle the background on and off using this selection. Backgrounds are loaded using the “Load Image” in the contextual menu.



### 2.11 View Parameters / Camera Parameters

If a window is in an orthographic mode, like front, right or iso, you can adjust the settings for the view. If it is in a perspective mode, the menu changes to Camera Parameter, and camera options can be set.



Both the “View Parameters” and the “Camera Parameters” will give you numeric input to adjust the view positions. These include X,Y and Z positions, which define the placement of the view in the window, the X,Y and Z References, which determine what the view is looking at, and Roll, Pitch and Yaw input, which determine the orientation of the view.

*Figure 2.11 — Parallel View Parameters*

The expanded camera parameters gives you the same camera position fields as well as settings for perspective properties. On the top left of the window are size settings for the camera view. These can be set manually, or can be altered by picking preset sizes in the available pop-up window.

Camera Parameters offer the same position, reference and orientation options as the Non-Perspective view parameters, but also give options important for setting up a scene. These options are available only in windows that are set for “Perspective” viewing.

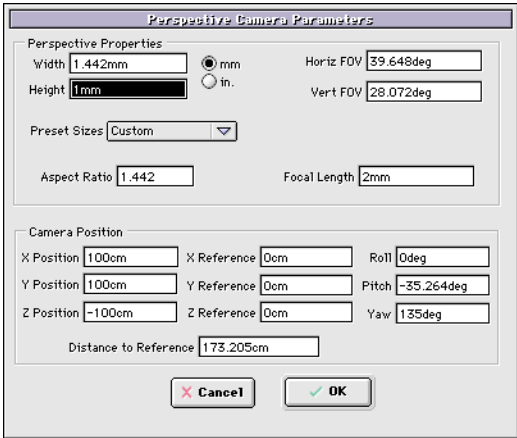


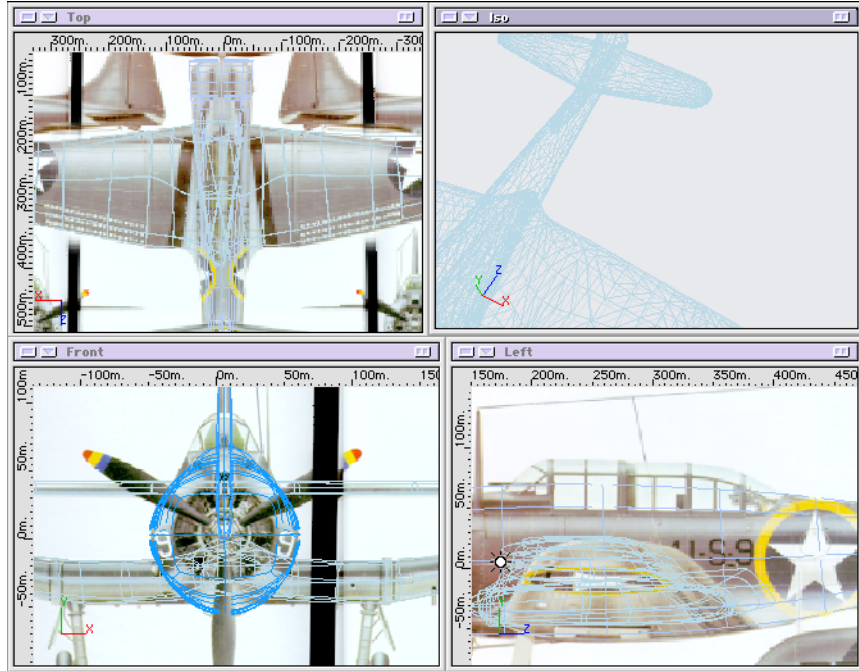
Figure 2.12 — Parallel View Parameters

To adjust the Field of View, alter the fields located in the upper right hand of the window. The larger the angle, the more wide-angle the perspective will be. The lower the angle, the more telescopic the view will be. Changing either the horizontal field of view or the vertical field of view will automatically alter the other.

**Load  
Image**

Image files saved in the Electric Image format can be brought into the program as background images and can serve as templates for creating models. The images enter view in the same window in which the pop up menu was invoked. A different image can be imported in each window.

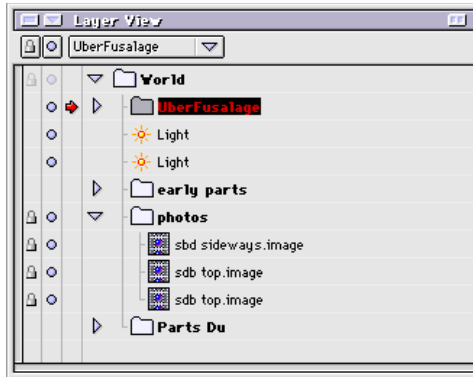
Image files will act much like other elements in the window, they can be selected, moved, rotated, locked, scaled and hidden.



Each window can display its own image. Each image can be positioned and scaled as needed, creating a perfect environment for introducing a template to build a model.

*Figure 2.13 — Loaded Images*

Images can also be brought into the iso view. When in iso view, they come in on a flat plane to the camera and any changes in perspective or position will be reflected on the image, just as if it was a flat, planar object.



*Figure 2.14 — Images as seen in the view window*

*Note: An image brought into a window will change in perspective with the rest of the scene.*

## 2.12 Navigation

There are two ways to navigate through views, using the tools available in the toolbar, and by using modifier keys.



*Figure 2.15 — View tools*

**Pan** To move any screens view up or down, left or right, hold down the spacebar and click and drag with the mouse. When you have held the spacebar down you will see the hand cursor on the screen.

To use the Pan tool, simply click on the tool, and then click in the window you want to pan the view with, and click and drag the mouse.

**Zoom** To move closer or farther from an object in a scene, hold down your spacebar and the command key (*PC- control key*) and click and drag. You will see the magnifying glass cursor. Moving to the right zooms in, moving to the left zooms out.

To use the Zoom tool, simply click on the tool, and then click in the window you want to zoom the view with, and click and drag the mouse. Dragging to the right will zoom in, the left zooms out.

**Orbit** To change the direction of the camera, you can orbit by either holding down the spacebar, command key (*PC- control-key*) and shift keys and click / drag on the screen, or you can hold only the “O” key down and click / drag. While you can orbit in any view, orbiting is best done in the camera, or iso view. Orbiting from one of your working windows (top, side, front) will just break that window out of its axis-aligned view and turn it into an axonometric view.

To use the Orbit tool, simply click on the tool, and then click in the window you want to orbit the view with, and click and drag the mouse.

### **Fit objects in window**

Holding down the F key and clicking in any 3D window will fit all the objects in the scene into the window. Holding down the Shift and F keys will fit only the selected object or objects into the window.

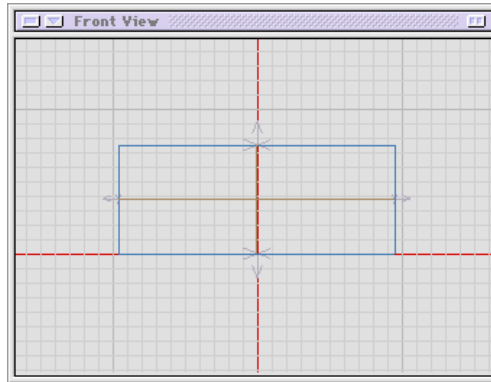
### 2.13 Creating Primitive Solids

Seven different types of solid primitives are included in the toolbox: cube, sphere, cylinder, torus, cone, pyramid, and prism.



*Figure 2.16 — Solid Primitives with block chosen*

The actual object's proportions are determined by the user when placing them in the world. For instance, click on the cube tool. Then, click and drag in the top view window. As you drag, you see the width and depth of the cube form. When you release the mouse, your dimensions will be set. Without clicking, move your mouse to the front or side view and you'll notice the height of the cube will follow the cursor. When you've positioned the cursor for the height, click on the mouse. The grid on the object, called an isoparm, is a reference device so the viewer can get a feel for an object's surface.



*Figure 2.17* — **Block created in the front view**

### 2.14 Selecting Objects

The selection tools give you the option of selecting a vertex, an edge, a loop, a face, the whole body, or all the bodies in a group. To select the cube above, click on the body selection tool, then select the cube by clicking on it. The highlighted object turns red (or whatever the highlight preference color is set to).



*Figure 2.18* — **Filter Toolbox with body filter chosen**

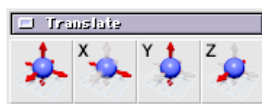
*The Pick tool works hand-in-hand with the pick filter selection, allowing the user to select between a single vertex or an entire group.*

### 2.15 Deselecting Objects

With the Pick tool chosen, hold down the shift key and click on an object that was previously selected. The object will be de-selected. If you have several objects selected and you want to de-select them all, hold down the shift key and click in a blank area.

### 2.16 Moving Objects

Translation tools provided let you move the object in any direction. All the selected objects are affected by the move. If the objects are already selected, you don't have to click directly on them to move them—click anywhere in the workspace and move the cursor, the object(s) will follow along. Holding down the X, Y or Z keys will constrain the movement to that axis.



*Figure 2.19 — Translation Tools*

### 2.17 Deleting Objects

Delete the object by selecting the Delete tool and clicking on the shape or using the Selection tool and hitting the delete (PC- *backspace*) key.

### 2.18 Creating Wires

The foundation of most shapes are the wire tools. With simple lines, curves, Beziers, and NURBS, any 2D and 3D shape can be constructed. Once a wire form is created it can be extruded, lathed, used to create a path or a source of a shape, or used as a rib of a shape and then skinned.





The Curve Primitives palette contains nine tools for creating basic wires. For more information on how each tool works, See “Curve Primitive Palette” on page 131.

**Figure 2.20 — Curve Primitives Palette**

When a wire shape is initially formed, it has no bulk to it. Several tools are available that can transform your wire into a solid or surface object. The Extrude tool used on an open wire will create a double sided surface shape. A closed loop wire can also be used to create a surface object, just make sure the “Caps” option is toggled off in the Extrude tool first. If the “Caps” option is on, a solid, single-sided object will be formed.

Wires can be created in many ways. Tools for drawing them directly are supplied, including Bezier, NURBS and wire primitives that cover a range of lines, circles, polygons, arcs and ellipses. In addition to this, wires can be created from existing shapes by using tools like Intersection Wires or Project Wire Onto Body. Even solid surfaces can have their loops turned into wires using the Convert Loop to Wire tool.

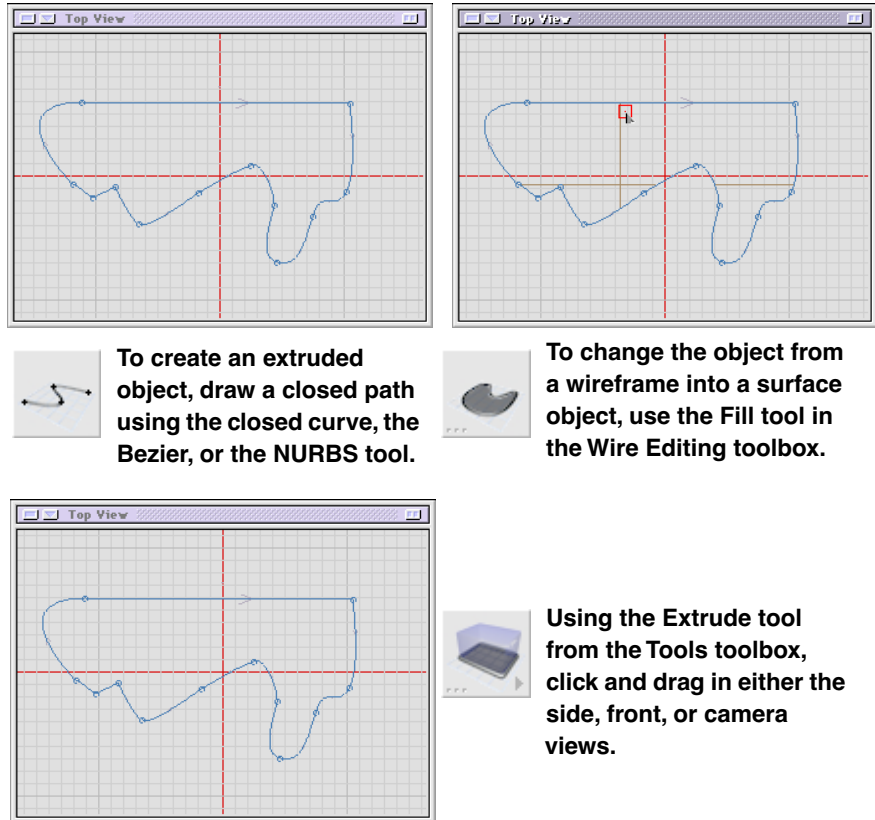


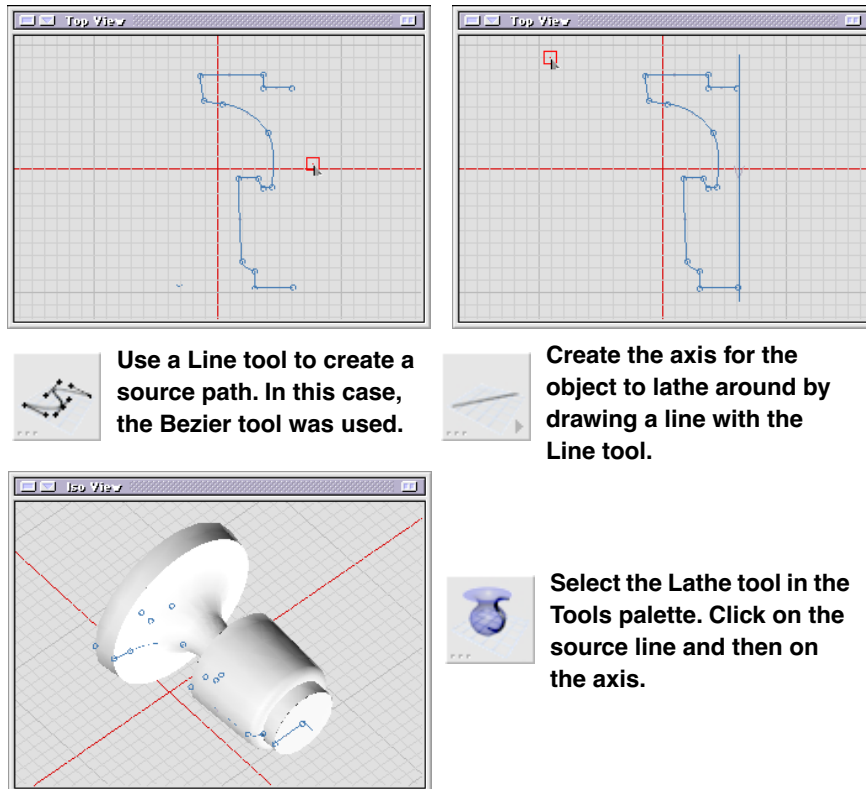
Figure 2.21 — Creating an extrusion

Some tools, like Extrude and Revolve, require the construction wires be planar. This means that they were drawn on a single plane, and don't go through all three planes at the same time, in short, that they are flat. If you get a message box stating that the tool cannot be used on a non-planar wire, you may have to construct a wire that is planar.

Usually this means working in a single window when you create the initial wire, or use the planar snap option when creating the wire. Some tools require several wires be used for the creation of a single shape. Revolve tools require source shapes and axis lines, for instance. Once the new shape is created, the source shapes remain in the picture. You can delete them, or hide them for later use if you want. Wires can be exported provided the option for export is turned on. Filled wires used as source objects will render as surface objects, so be sure to hide them.

### 2.19 Creating Revolved Shapes

Revolved shapes begin their lives as 2D wire primitives or faces. Using any of the 2D wire tools, create a wire object in any planar view (For instance, top, side, or front). After you finish creating this "source" line, create the axis that you want to revolve the shape around using the line tool. It is important that no part of the source line cross over the axis line.



*Figure 2.22 — Revolving an object*

Before you use the Revolve tool, you may want to adjust the settings for the revolve. Like many other tools in the toolbox, additional settings can be accessed by double clicking on the tool icon in the toolbox.

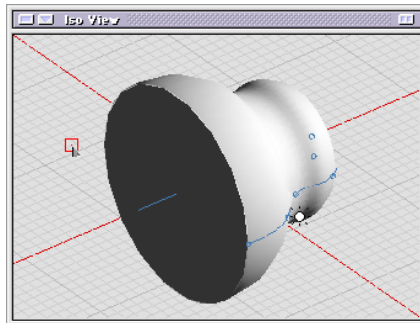
The revolve dialog box is accessed by double-clicking on the Revolve tool icon.



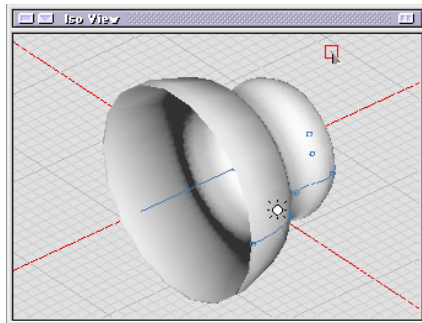
*Figure 2.23 — Revolve tool settings*

While the settings for revolved objects are covered in-depth in the Toolbox section of the manual, there are a couple options you may want to alter before creating even a simple lathe.

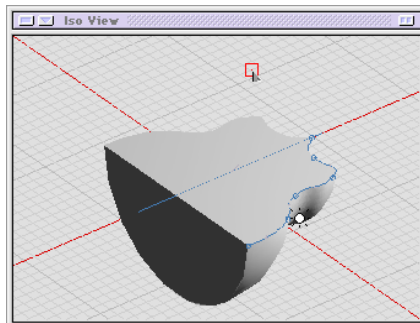
On the bottom left of the dialog box is a selection for Solid or Surface. If Solid is selected, an open face of a revolved object will be filled. If Surface is selected, an object with an open face will not be filled and the revolved shape that will be created will be a surface object.



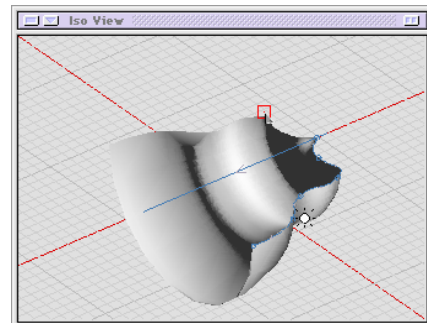
**Revolved as a solid.**



**Revolved as a surface.**



**180 degree revolved as a solid.**



**180 degree revolved as a surface.**

**Figure 2.24 — Revolved shapes with different settings**

You also can create a partial revolve by changing the angle of revolution. In the example, Figure 2.24, the source line was swept only 180 degrees as both a surface and a solid object.

## 2.20 Creating Swept Objects

For shape creation, many of the tools in the program, such as sweep, birail, and profile tools, require making a cross-section and a path. In some cases, several paths or several profiles are required. The most elementary use of this tool type is the sweep. In this example, we'll create a path, make a cross-section and perform a sweep.

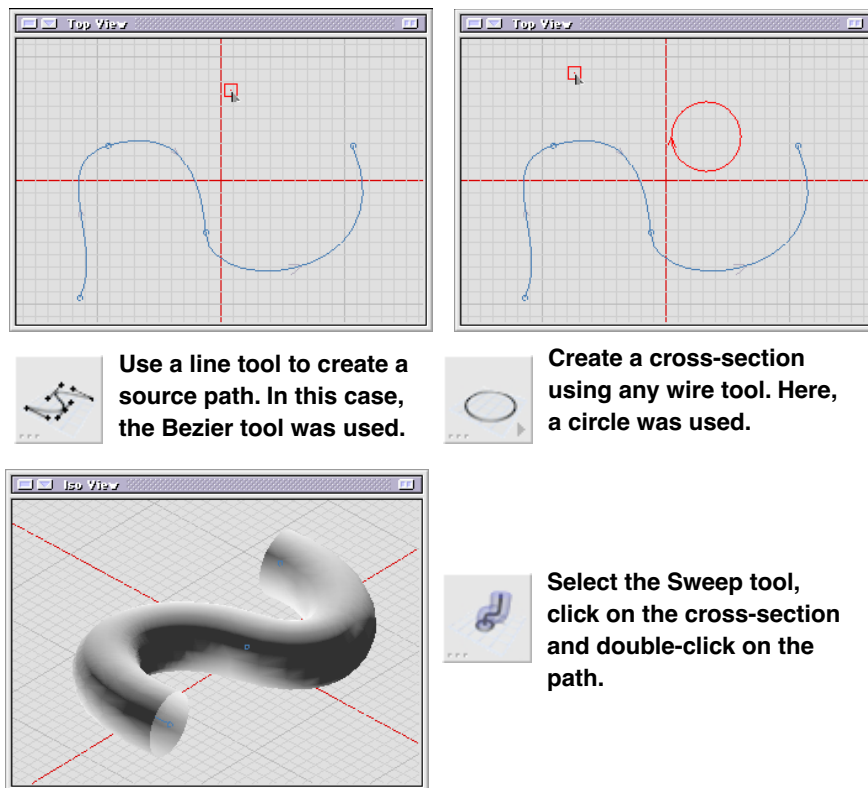


Figure 2.25 — Sweeping an object

Begin by creating a new document or deleting objects in your current scene. To create our path, use either the NURBS tool or the Bezier tool. If you really want to have fun, the wire tools are non-planar, meaning you can click and drag in any of the windows, front, side, and top and create a path that is a 3D wire.

Next you'll create a cross-section to follow this path. Choose your primitive line tools again and pick the circle. In the top view draw a small circle—be careful not to make it too big. If the diameter is too big for the object to sweep, the operation will be rejected.

If a face was applied to the circle before it was swept, the object would be solid; if not, it would be hollow. In our example, the circle was left unfilled, and in the final sweep you can see the tube is open-ended.

To create the sweep, select the source (the circle in this case) and then double-click on the path. Many tools in the Universe Modeler require multiple selections to perform a function, and they all require a double-click on the last object, or a double-click in null space to tell the program to perform the function.

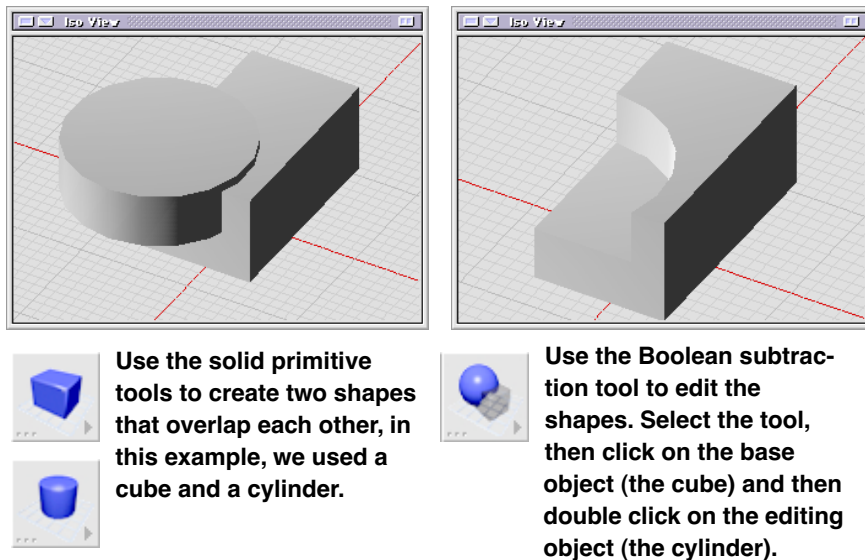
### 2.21 Editing Objects

Once you've created an object, the program shows its strength with its ability to edit the forms. First we'll create a Boolean shape and use a body editing tool to round the edges. Begin by creating two primitive solid shapes that overlap each other.



Use the Boolean Subtraction tool to edit the cylinder out of the cube. Select the Boolean Subtraction tool. Click on the cube and double-click on the cylinder. Your new edited form is now ready for your next operation.

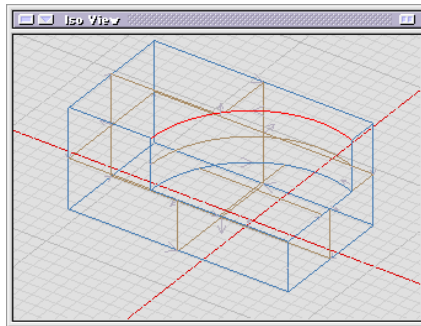
The Universe Modeler allows you to either pre-pick or post-pick items for different functions. As an example, we'll round off some edges of our shape. Change the Pick Filter in the Pick Palette to Edge Filter.



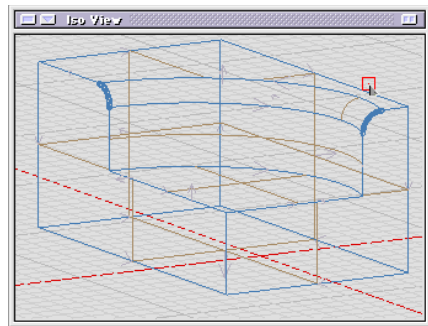
*Figure 2.26 — Boolean subtracting an object*

### Pre-picking Objects

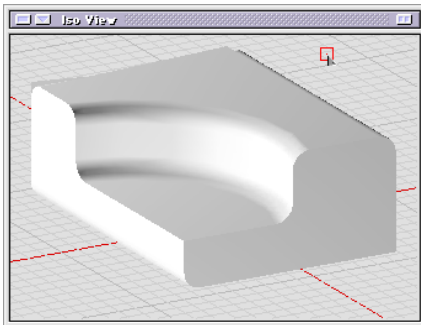
Using the Pick tool, select a few of the edges in the shape. After your edges are highlighted, go to the Tools palette and select the Rounding tool.



In the Pick palette, select the Edge Pick filter and select the Pick tool. Use the Pick tool to select an edge.



Under the tools palette, select the Rounding tool. With the Rounding tool highlighted, click on the edge you want to round.



You can continue to round any edges of the model you want. This example shows a few more edges rounded.

*Figure 2.27 — Rounding an edge*

Clicking in null space will cause all the selected edges to be rounded.

### Post-picking Objects

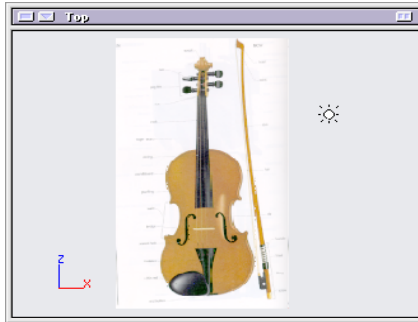
Select the Edge Filter in the Pick Palette, then select the Rounding tool. Any edge you double-click on will be rounded.

After finishing these exercises, you should have a fundamental understanding of the program. But there's a lot more power behind the tools and the next chapter walks you through them in more detail.

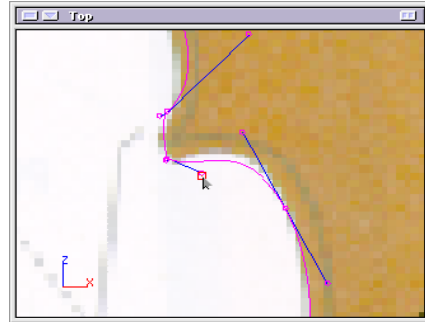
**Saving your work** At any point in the process, save your work by doing a menu File/Save, or Command + S (PC- *Ctrl* + S). When you're ready to export a file and bring it into another program, use the menu File/Export and pick the file format of choice.

### 2.22 Building a Violin

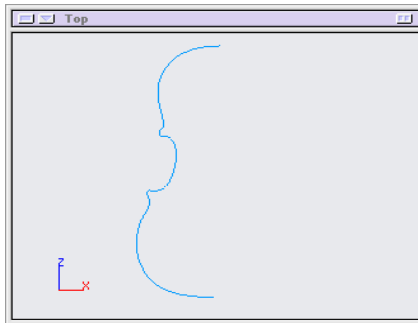
In the following pages the steps used to create a violin will be described. These steps are not intended to be used as a tutorial on how you should create a violin, but rather to help enlighten you on the steps involved in shape creation, so you can better understand the inter-play of tools with each other. Of course, if you want to try to follow along, dive on in. You'll want to refer to each tool as you go along for a more complete understanding of its function.



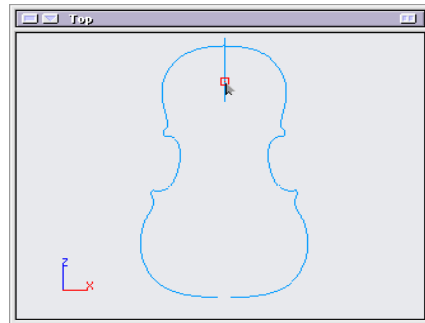
To get a perfect violin shape, an image of a violin was scanned and saved as an EI Image format. By bringing up the contextual menu command-click (*PC- right click*) on any window) the image was loaded.



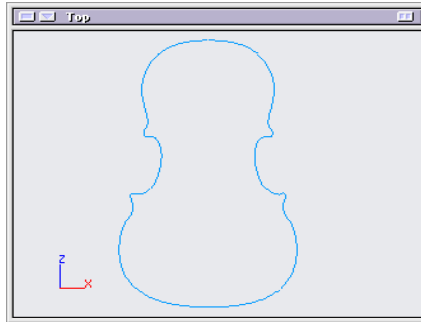
To get into the detail, the image was zoomed into holding down the space bar and command (*PC- Control*) key and dragging the mouse. The scan and the Bezier Curve is enlarged to create the outline.



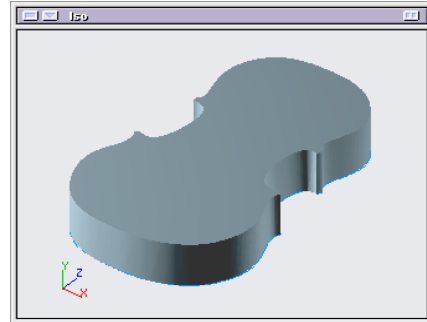
Half the violin was drawn using the Bezier Curve tool.



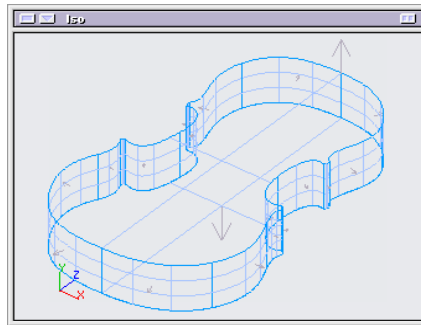
Using the Line tool, an axis was drawn and then the Reflect tool was used to mirror the wire.



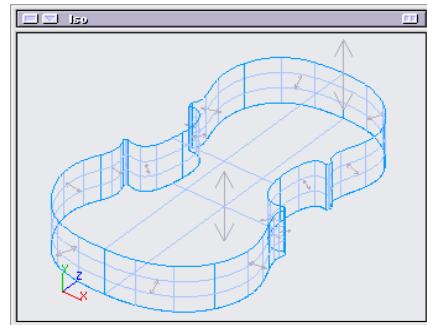
The Join Bezier tool was then used to bring the two separate wires together.



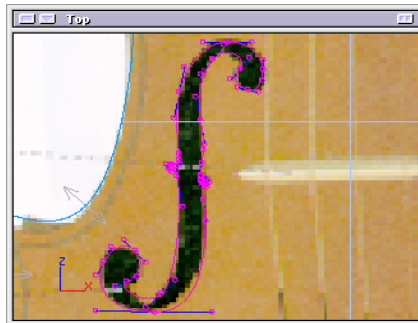
Then the Extrude tool was used with the “Caps” option turned on, creating a solid object.



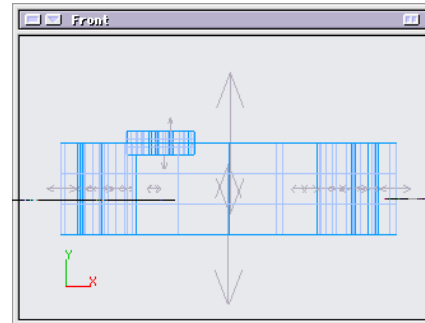
While this extruded shape is currently a single-sided solid object, in the near future it's going to have to become hollow, and that means making it two-sided.



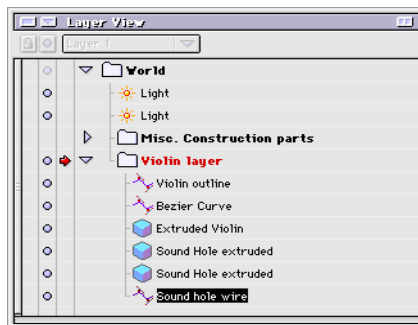
With the System Preferences Display tab set to show face directions, the Make Two Sided tool from the Body Editing Palette is used, note the change in the face directions.



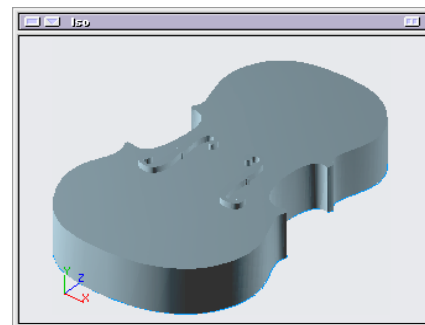
To create the shapes that are going to “poke” the holes in our violin, the scan is brought back and the shape of one of the sound holes is created using the Bezier tool, and it is then extruded.



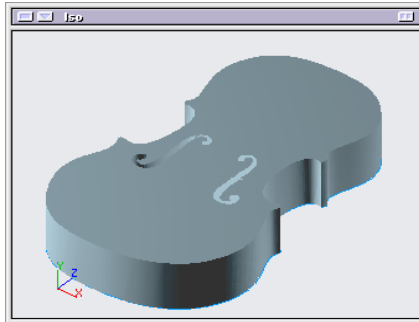
The extruded solid shape for the sound hole is positioned to intersect with the top of the violin.



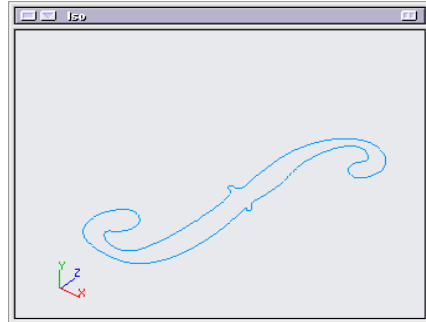
Keep track of things with your layers window, and name your objects as you go.



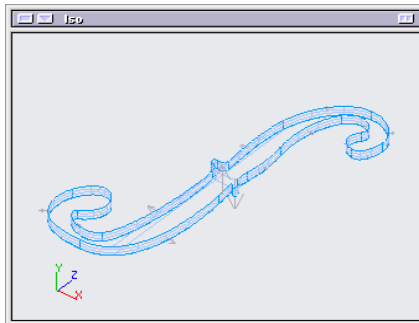
Using the reflect tool, the sound hole shape is repeated for the other half of the violin.



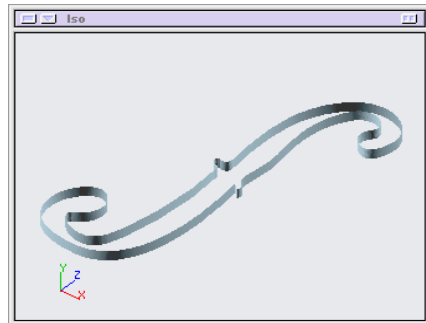
Using the Boolean Subtract tool the sound hole shapes are removed from the violin shape.



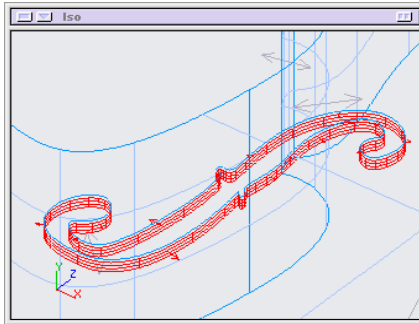
To keep the violin from seeming paper thin, the original sound hole wire is made active with the rest of the scene hidden.



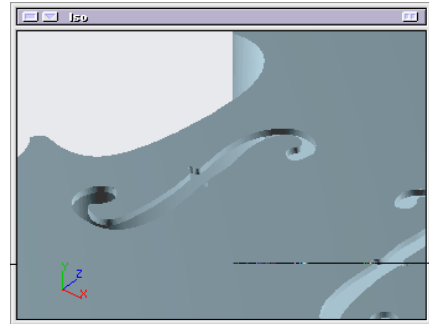
The wire is then extruded. By accident the “caps” option was left on.



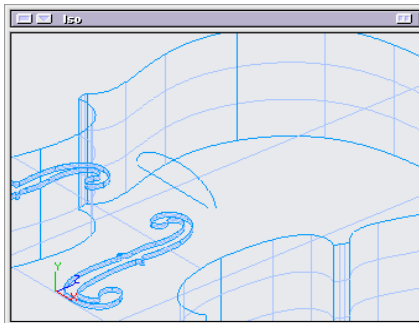
By using the Remove Face tool in the Body Editing palette, both the top and bottom faces are quickly removed.



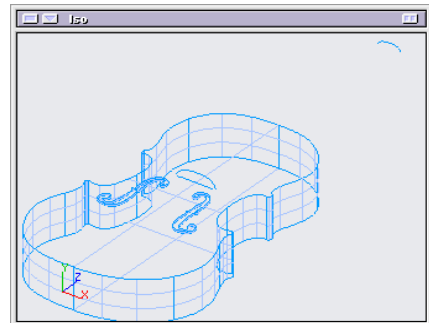
The sound hole object is now positioned right below the hole in the violin.



Using the Blend Edges tool the edges of the violin hole, and the edges of the sound hole surface object are selected and the blend surface is performed, melding the two together into a single body.

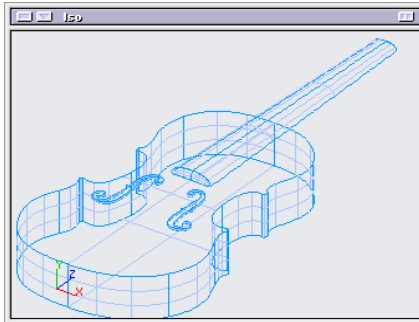


Using the Bezier Curve tool, two wires are made to form the base of the neck.

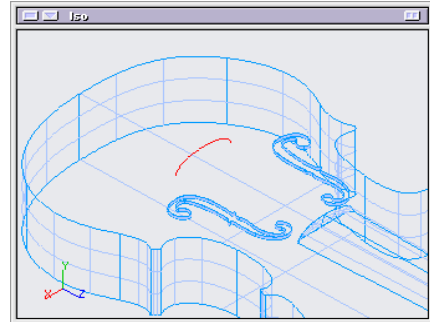


The larger curved shape is copied, moved and reduced in size to create the top of the neck.

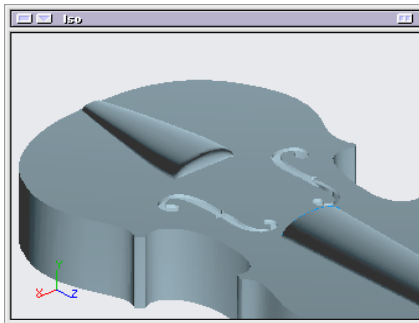




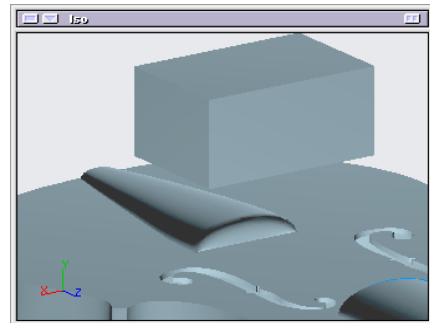
Using the Skin tool, and clicking on the three wires, the neck is formed.



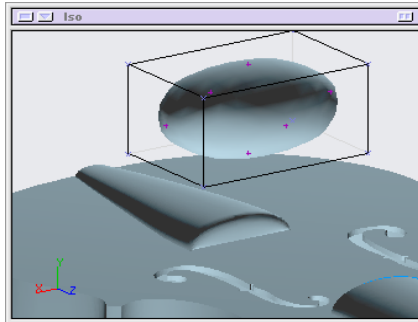
The same procedure is used to form the tailpiece, the curve is copied and reduced, forming ribs.



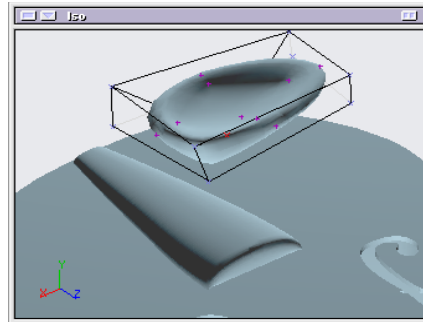
Then the Skin tool is used to create the tailpiece.



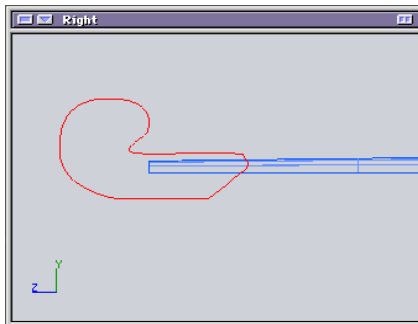
To create the chin rest, a Block Primitive is introduced to the scene.



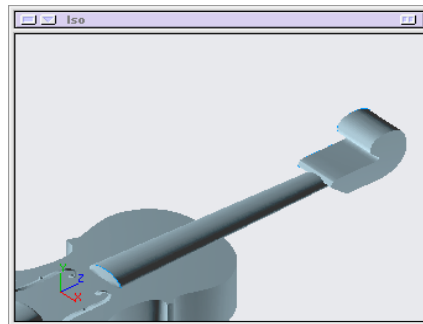
The UberNURBS tool is selected and the block is clicked on, an editing cage forms around the UberNURBS shape.



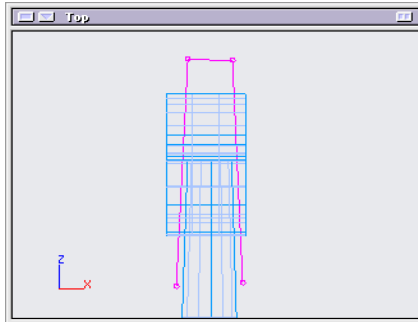
By grabbing the top vertices and pulling them down, the shape is “squashed” and the top face is subdivided radially. The new vertex in the center is moved down slightly to form the depression.



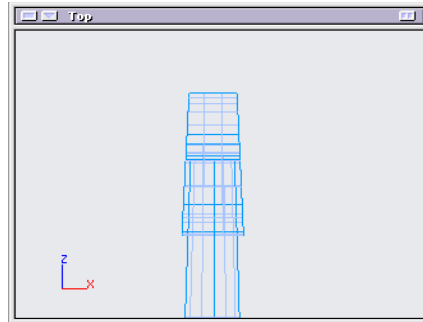
To create the peg box, a wire form was created to approximate the side outline of the shape.



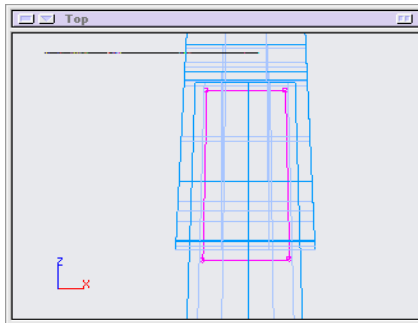
Using the Extrude tool with the “caps” option on, the wire is extruded.



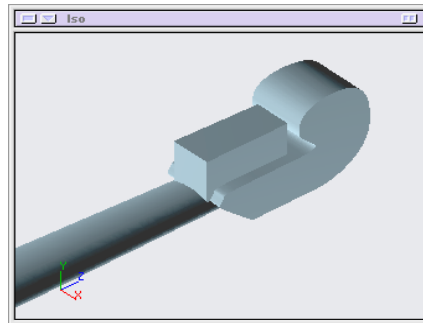
In the top view, the polyline tool is used to create a wire that will be used to edit the pegbox.



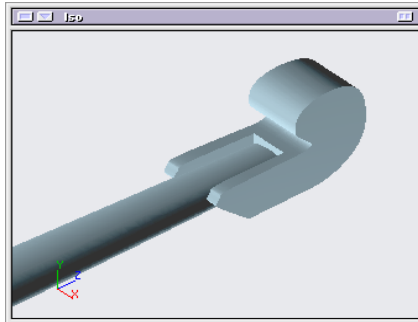
The Wire Knife tool is then used to edit the form, the excess parts are deleted.



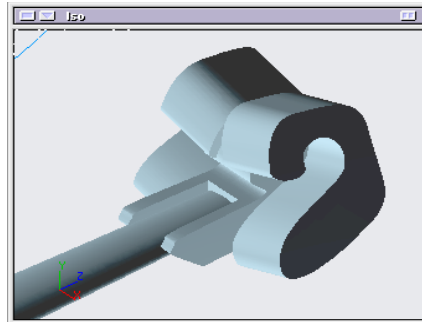
Still in the top view, the Polyline tool is used to create the negative area in the peg box.



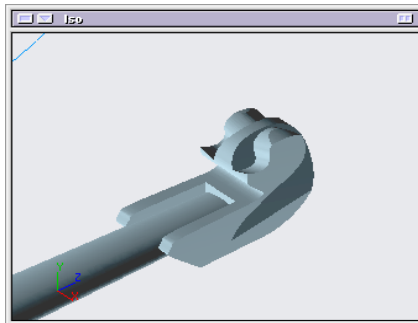
The shape is then extruded with the “caps” option turned on, creating a solid shape.



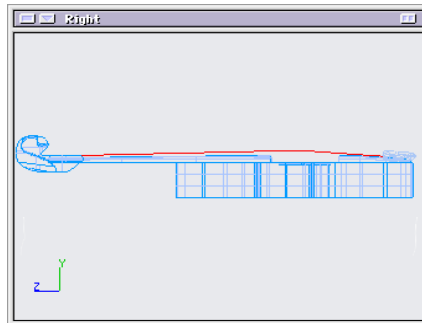
The Boolean Subtract tool is used to remove this shape from the peg box.



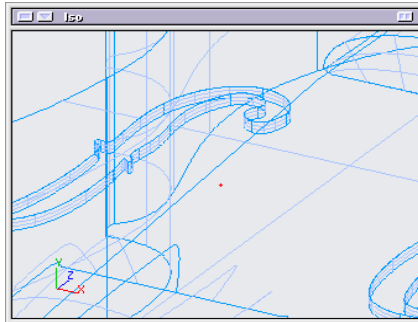
To sculpt the top of the peg box, a shape is created and mirrored, the shapes are carefully positioned to overlap the peg box.



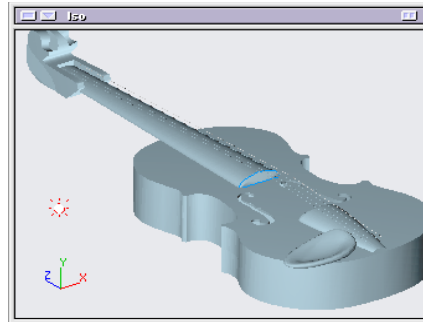
Using the Boolean Subtract tool the two solid shapes are removed from the solid peg box, sculpting the object into its final shape.



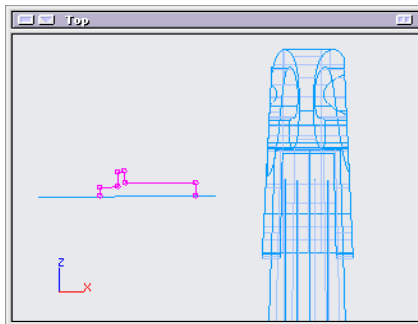
To create the strings, use the Bezier, Polyline or NURBS Curve tool to create the path.



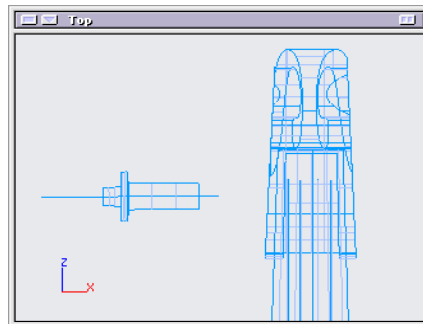
To make the cross section, the Circle tool is used. It'll be small, you might want to zoom in.



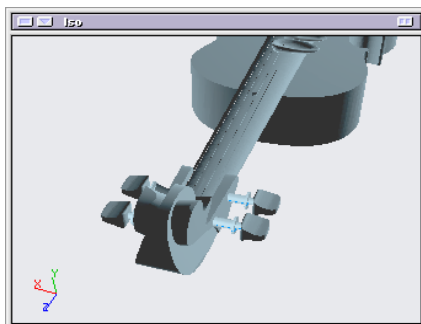
Using the Sweep tool, clicking on the circle, then the path, creates the string, which is then copied.



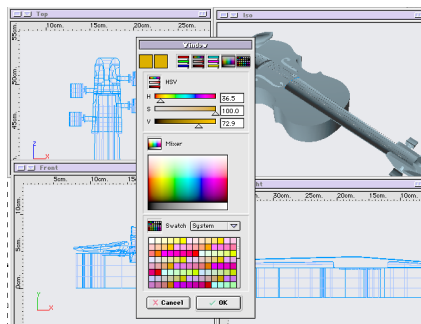
Time to create the string pegs. Create an axis and a cross section using any curve tool.



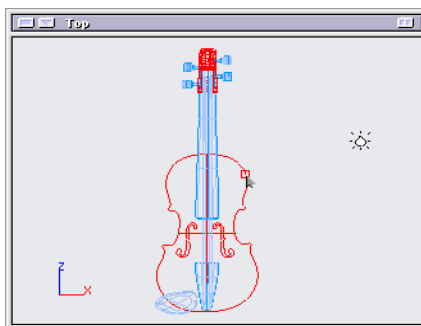
Choose the Revolve tool, click on the cross section, then the axis, and the peg will be created.



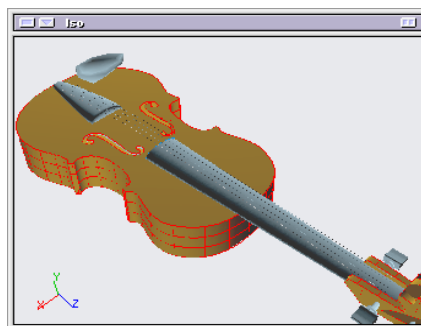
Use the Free Move tool to position the pegs, and then use the Free Move tool while holding down the “D” key to duplicate the peg, and position the rest of the keys.



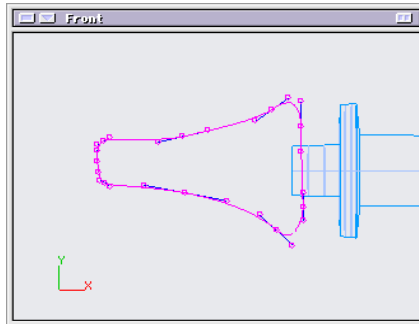
Click on the Pick Color tool in the Color Tools Palette, and create a new color.



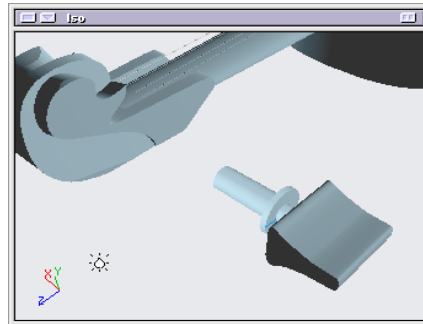
Select objects in the scene that you want to make that color.



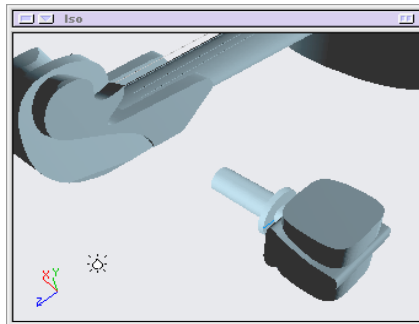
Click on the Apply Color tool in the Color Tools palette, all selected objects will be colored.



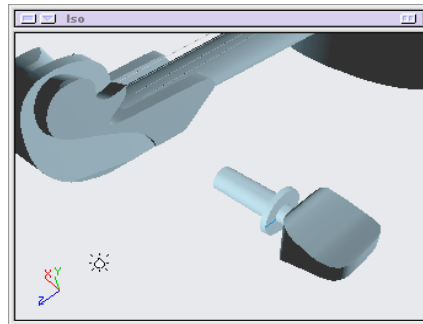
The Join Bezier tool was then used to bring the two separate wires together.



Then the Extrude tool was used with the “Caps” option turned on, creating a solid object.



Create from the top view the shape of the peg. Place it so that it fits entirely inside the side extrusion on the peg.



Using the Boolean Intersection tool, click on the side and top extrusion and the key will be formed. The Knife tool could have been used here, but hey, we’ve done that already.





# Menus & Preferences

By design the menu selections in the program are spartan. All of the options for the tools can be found by double-clicking on the tools, and controls for the windows can be found under the contextual menus by holding down the control key and clicking (*PC- right clicking*) in the window. This leaves the menu list to deal with the basics.

## 3.0 The File Menu

This menu controls the system-wide concerns such as opening, closing and saving the scene files.

**New.** Choosing this option will close the current project and open a new one that will be based on the default preferences. If you have not saved the current project, you will be given the option to do so.

**Open.** Previously created projects are opened by using this menu. On the PC only (.eim) files will be seen in the dialog box used to open files.

**Close.** This menu command closes the currently open project.

**Close Window.** This command closes the currently active window view. The active window is indicated by a darkened Drag Bar.

**Save.** On a previously unsaved project, this command will bring up the standard “**Save File**” dialog box. If the file has been saved before, this command will update the disk file, saving changes made to the file.

**Save as...** To save the current scene under a different name, use the **Save As...** command. The current version of the project will have the new name, and the previous version will remain unaffected. PC versions will automat-

ically add “.eim” to filenames. If you will be moving files between Mac and PC versions, we recommend use of “.eim” extension for both platforms.

**Note:** *When transferring a file from PC to MAC, you will have to set the file type (TYPE: AMS Creator: EIM) for Modeler to be able to recognize it.*

**Save a Copy.** To save a copy of the current project, and change its extension, without changing the name of the current version, use this option

**Tool Palettes.** Custom tool palettes can be created and loaded using these settings.

**Import.** The program imports SAT Text, SAB Binary, FACT files, IGES, and Illustrator EPS Files (up to Illustrator 8 format). EPS objects may come in at very large sizes. You may have to zoom out to see them.

**Export.** The program exports SAT Text (versions 4.x, 5.x, and 6.x), SAB Binary, FACT, STL, IGES and OBJ files. Most users will be interested in the FACT file export for importing the scene into Electric Image. When FACT file is chosen, the scene will be tessellated using the options set in the Document Preferences.

**Quit.** This closes out the program. If the current project has not been saved, a Save dialog box will appear.

### 3.1 The Edit Menu

This menu affects the objects inside a window, including undoing and redoing actions, object selection, duplication, locking and hiding. At the bottom of the menu are the preferences.

**Undo.** At any point in the modeling process you can take a step back and undo what you just did. You can, in fact, undo all the way back to your last save. Command-Z is your friend.

**Redo.** After you've used the **Undo**, you can use the **Redo** to go back to where you were before the undo.

**Cut.** Only text in edit fields can be cut. Removing them from the scene and placing them on the clipboard. The clipboard is a virtual holding spot for text and will remain there until another block of text is copied or cut, or until the program is exited.

**Copy.** Much like the **Cut** command, the **Copy** places text fields on the clipboard, without deleting it from the current text field.

**Paste.** Any text field previously cut or copied can be taken from the clipboard and placed in other text fields by pasting it.

**Clear.** Any selected text field will be deleted using this command. The text will not be placed on the clipboard, it will simply be removed from the field.

**Select All.** This command will select all the items in a scene.

**Deselect All.** Any and all objects selected in a scene will be deselected using this command.

**Preferences.** These are settings that determine many key factors in the program, from the color of the grid lines to the resolution of the models. The **System** and **Document preferences** are covered extensively in the pages that follow.

**Toggle Background.** This option makes the background images visible or invisible at the click of the mouse.

### 3.2 Create, Object, Tools, and Tessellation Menus

These four menus give you direct access to particular tools, instead of having palettes up on your desktop or scrolling all the palettes of tools. More information on these tools are available in the Toolbox section of the manual.

### 3.3 The Windows Menu

The user's door to the program is through the windows on the screen. These windows are managed directly through options in the **Windows menu**. These options allow for the creation and management of the windows, which are used to present the scene, info, status, layers and more.

### 3.4 Preferences

Under the Edit menu can be found two items, **System Preferences** and **Document Preferences**. Each of these windows contains a tabbed subset of preference selections. The **Document Preferences** is used in setting up the tessellation of an object and the grid unit, ruler and snap settings. Tessellation determines how objects will be faceted when objects are exported, and also controls the overall resolution of the images in the scene. **System Preferences** determine how the screens look, with adjustments for showing directions and colors, as well as fundamental building preferences, for multicopy, creation, export grouping and color.

#### Document Preferences

In the Document Preference window you will find two tabs, one for Tessellation, and one for Grid/Units/Ruler/Snap.

### Tessellation

Tessellation settings are faceting options to convert ACIS bodies into objects that can be viewed and exported. For those who want to dive into the program and start building models, the defaults used should serve as a good foundation. The settings provide a good smooth, average sized mesh resolution. Altering the settings for higher or lower resolution will be covered both in this chapter, and in the section for exporting.

Global changes made to these fields will effect the creation of all objects with the “Use Project Settings” box checked. Selecting Object Display or Export Tessellation Settings make it possible to create an object with one Normal Tolerance setting different from all other objects. In other words, altering the **Object Tessellation settings** will not alter the whole scene, just the objects selected, and when the Regenerate tool is applied

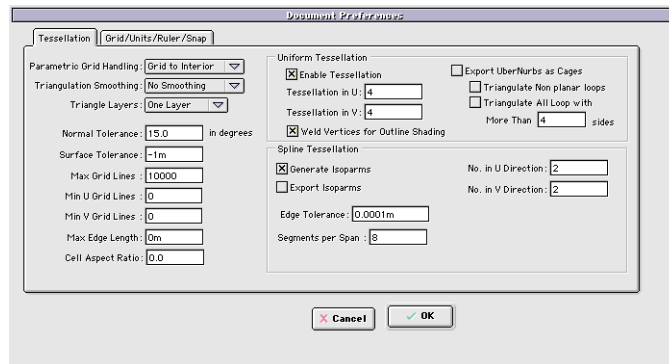


Figure 3.0 — Tessellation preferences

Faceting is the operation that generates polygonal representations for the faces of entities. The ACIS faceter supports only view-independent faceting. Faceted representations are used in rendering, in analysis and in oper-

ations where an approximation is acceptable in order to simplify calculations.

Face faceting is performed by subdividing the face in parameter space with a grid whose increments are determined through settings in the tessellation fields. These settings determine the accuracy of the faceted representation. Tessellation settings also control how triangulation is performed and whether smoothing is used to improve the aspect ratios of triangles.

The faceted representation of a face is also called a mesh. Altering the tessellation settings will have a significant effect on the rendering process. The tighter the facets, the smoother the rendered surface will be. However, the tighter the facets, the slower the rendering process becomes. On the other hand, the looser the facets, the faster the rendering process becomes, but the rendered surface will be rougher.

Because ACIS supports various types of geometric representation (e.g., planar, conical, toroidal, and spline surfaces), faceting relies on the curvature measurements of each faceted face to capture the underlying geometry. All types of surface geometry, except spline, can be well approximated with a regular grid whose lines are equidistant in parameter space. However, due to their intrinsic free-form nature, spline surfaces necessitate a different approach. Adaptive faceting is employed to lay a grid of non-equidistant lines, rather than a regular grid, for spline surfaces.

The non-regular grid is based on the curvature computation of the spline surface at some parameter space sampling locations. Sampled curvature distribution controls the density of grid lines in the parameter space. Higher curvature areas get a greater number of grid lines, while lower curvature areas get fewer lines. In all cases, faceting tries to satisfy the refinement criteria set by the application. The adaptive faceting algorithm now

subdivides internal triangles according to the specified refinements in the case of faceting a body with triangles when no UV grid was selected.

**Parametric Grid Handling.** This determines whether a grid is used and whether the points where the grid cuts the edges should be inserted into the edge discretization. The grid mode uses the UV parameter grid of the surface for faceting. The allowable values are:

- No Grids: Does not subdivide face with a grid.
- Grid To Edges: Uses a grid and creates intersection points of the grid with the edges.
- Grid Interior: Uses grid only in the interior. Triangles are generated from the edge to the grid. This is the default value.

**Triangulation Smoothing.** Determines whether triangles will be smoothed or not.

**Triangle Layers:** This specifies how much triangulation to perform. If Grid Interior is specified for Parametric Grid Handling, triangulation will be performed at least at the fringe cells.

**The allowable values are:**

- No Triangles: effective when used with Grid To Edges for Parametric Grid Handling
- One Layer: Triangulate at the fringe layer.
- Two Layers: Triangulate two (2) fringe layers.
- Three Layers: Triangulate three (3) fringe layers.
- Four Layers: Triangulate four (4) fringe layers.
- All Triangles: Triangulate all facets.

The Grid Interior mode generates triangles at least near the first fringe, even if the triangulation mode is No Triangles. In this case, triangulation mode No Triangles gives the same result as One Layer. When triangulation

mode is set to Two Layers, two outermost peripheral grid facets are triangulated. Up to four fringe levels can be specified by the triangulation mode, or All Triangles can be specified for all triangular facets.

If Grid To Edges is used, and the triangulation is set to anything other than All Triangles, the faceter could produce  $n$ -sided facets ( $n > 4$ ). It however does not create T-junctions. It matches nodes on either side of the Edge. However one must Import such exported FACT models into Universe Animator. Adding the model will not do the necessary triangulation of the  $n$ -sided facets, but importing will.

**Triangulation Smoothing.** This mode works when a grid is used along with triangulation of facets. It tries to adjust the facet node positions to smooth the triangles. It specifies whether triangles should be smoothed. The No Grid preserves the plane of cells by avoiding points that are corners of a cell.

The allowable values for Triangulation Smoothing are:

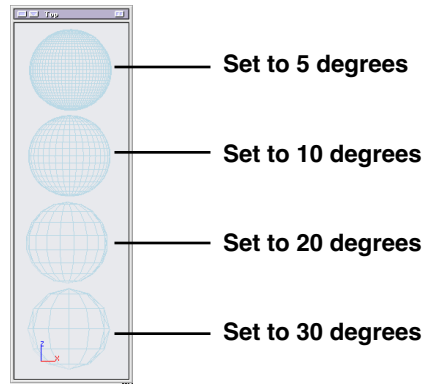
- No Smoothing: which doesn't do any smoothing.

Smooth Non Grid: Applies to points that are not part of a cell. This is the default value.

- Smooth Everywhere: Adjusts all points connected to triangles.

**Normal Tolerance.** The normal tolerance is the angle between the surface normals at the two adjacent nodes of a facet. By setting this normal deviation refinement, the user specifies how accurately the facets are representing the surface and the quality of rendering desired.





**Figure 3.1 — Normal tolerance examples**

It is desirable to use this refinement control because it is independent of the model size. The proper value is usually independent of the model size. The default is 15 degrees.

**Note:** *Lowering the normal tolerance can significantly increase RAM requirements. Many times another Tessellation Setting will give the required results instead.*

**Surface Tolerance.** The surface tolerance is the distance between the facet and the part of the surface it is representing. By setting this refinement, the user specifies how accurately the facets represent the surface. It is desirable to provide surface tolerance when facets obtained with normal tolerance do not represent the surface well. The proper value is dependent on the model size. The default is -1m.

**Max Grid Lines.** Smaller values for normal and surface tolerance can quickly increase the polygon count. To control this the Max Grid Lines can be reduced to a smaller number and the faceter will not exceed the maximum grid lines by the number specified.

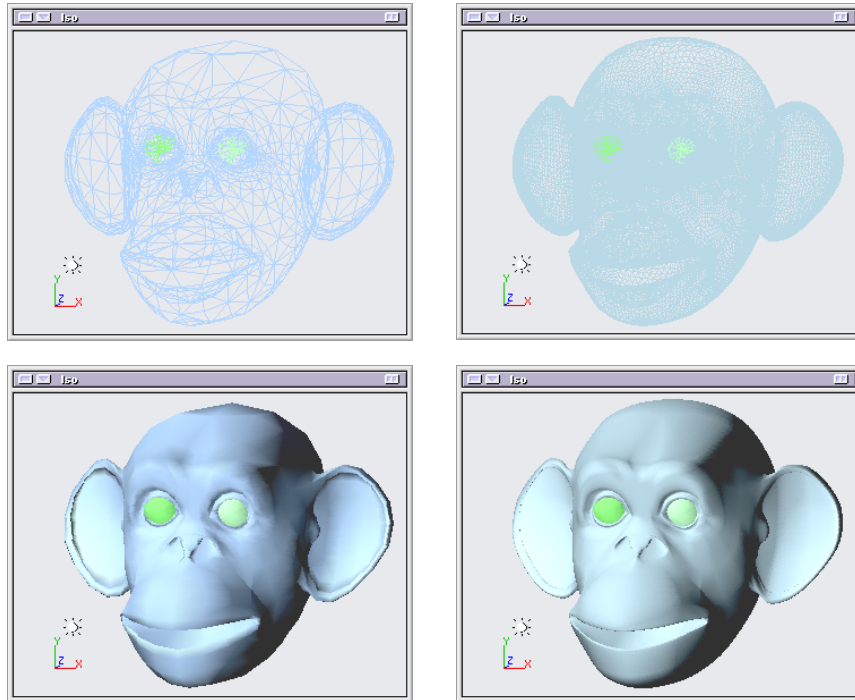
**Min. U/V Grid Lines.** This setting allows you to control how many grid lines there are for both the U and V direction of a surface. Sometimes having more grid lines in just one direction is preferable.

**Max Edge Length.** Edge length can be used to create a denser mesh by using a smaller length. The faceter will add more polygons to cover a face if there are edges in the current mesh that will have lengths greater than the Max Edge Length input by the user.

**Cell Aspect Ratio.** The grid aspect ratio is the ratio of the U and V lengths of a UV grid cell. Thus, the aspect ratio is used only when the Parametric Grid Handling is something other than No Grids. The u and v lengths are in object space. The algorithm tries to make the aspect ratio equal to the one specified, but this is NOT GUARANTEED. The grid aspect ratio is defined as the ratio of the geometric length of a grid in the u parameter direction to the geometric length of the grid in the v parameter direction. The faceter attempts to make grids having a ratio specified by the Grid Aspect Ratio parameter. For instance, setting the aspect ratio to 1.0 indicates that the grid should be as square as possible.

**Uniform Tessellation.** The settings for UberNURBS are under the Uniform Tessellation group. This setting is used to tessellate all the UberNURBS models which have not been converted fully to ACIS bodies. The option to convert UberNURBS to ACIS bodies or to keep them as UberNURBS attributes can be found in the dialog box to the UberNURBS tool.

By default the setting is 4 by 4, providing a very high density mesh. 1 by 1, on the other hand, would provide a very low density mesh. If you're planning to make morph targets, you would want to keep your UberNURBS as UberNURBS attributes and use the Uniform Tessellation settings for adjusting your UberNURBS resolution.



**Adjusting the Uniform Tessellation will effect how an UberNURBS object exports. The monkey head above is viewed in mesh mode and shaded mode with a Uniform setting of 4.**

**Adjusting the Uniform Tessellation will effect how an UberNURBS object exports. The monkey head above is viewed in mesh mode and shaded mode with a Uniform setting of 8.**

*Figure 3.2 — Adjusting uniform tessellation*

UberNURBS, can be exported with a tessellation that enables morphing. For fully converted UberNURBS (which are true ACIS objects) one has two options: to use the ACIS faceter (which does adaptive tessellation i.e. it will add more polygons where necessary in order to not exceed the user speci-

fied tolerance values), or, one can use uniform tessellation. As you know all UberNURBS objects are made up of smaller four-sided NURBS surfaces. These surfaces are parametric i.e. have U and V, and uniform tessellation can be done by sampling in equal intervals of U and V. This tessellation always produce triangles. Also, if you don't change U and V values. you will get the same number of triangles each time. Thus you can see how it is useful in morphing. The semi-UberNURBS can only be tessellated uniformly. So, one must check uniform tessellation if they want to output polygons for semi-UberNURBS.

### Uniform Tessellation Options

- **Enable Tessellation:** Allows for text input in U and V.
- **Weld Vertices for Outline Shading:** This option will combine the vertices where they overlap. This is used when using Outline shading in your program. Without this checked, extra outlines can be viewed. Polygon counts and vertex order does change when selecting this option, making morphing not possible.
- **Export UberNURBS as cages:** When this option is selected, instead of exporting the smooth UberNURBS mesh, the Universe Modeler will export the cage used to create the mesh as polygons. This is very good for using a proxy for character animation. Using a proxy will speed up your redraw speed.
- **Triangulate Non Planer Loops:** This option will help reduce the amount of pinching that can occur in non planer loops.
- **Triangulate All Loops With More Than (x) sides:** This gives more explicit control with the triangulation of loops. This way the polygon counts can be lower.

**Spline Tessellation.** These settings effect all the objects created with standard, non-UberNURBS tools. The isoparms are generated so you can view each object controlled in this area. The higher the number of isoparms, the

more visual feedback you get on your surface, but in the bargain you also get a more cluttered screen and slower screen refresh rates.

### Spline Tessellation Options

**Generate Isoparms:** Toggles Isoparms on or off. For a complex scene, you can toggle off the isoparms.

- **Export Isoparms:** Export Isoparms check box will export isoparms as polylines with the mesh data. Universe Animator can display isoparms (simple wireframe) instead of wireframes for object representation.
- **No. in U, V Directions:** Number of Isoparms generated in the U and V directions.
- **Edge Tolerance:** This setting is used for sampling the edges before displaying them in the program. Edges are sampled such that the polyline sample is no farther than the true edge by the user specified tolerance. True edges are mathematical functions. To draw these one must sample them. If the sampling is too coarse, you will see segmented edges. They can be made smoother by taking higher number of samples, but then the trade off is drawing speed. For example, if you sample a circle 12 times you will have twelve lines to draw, but it would not look much like a circle. If you sample it 120 times, it will look more circular but it will take 10 times longer. Currently, this setting control only straight and elliptical edges. Also when the user zooms in, it will appear faceted. All edges are sampled when the display structures are being generated.
- **Segments per Span:** For spline edges, one must use segments per span. A span is part of a spline between two knots.

### 3.5 System Preferences

The **System Preferences** window allows for adjustments to the way you work. Five tabs are available to alter settings with the system, multi-copy, creation, export grouping and color.

**System** Two groupings are found here, one for display, the second of diagnostics.

**Tab** The Display section deals with how objects on the screen will be shown. While some might consider it vital to have all the elements displayed, others might find it distracting, or unneeded, and just toggle on the options when needed. Show Isoparms and Show Silhouettes are toggled on by default, since they will be needed to show your work on the screen at a minimum.

- **Show Knots on Curve:** Displays the knots on a curve, useful for showing the structure of a curve when not in editing mode.
- **Show Vertexes:** Shows ACIS vertexes. These are not CV's or Uber-NURBS vertexes.
- **Show Edge Directions:** There will be times, such as skinning, when knowing your edge direction will be critical in creating a proper form.
- **Show Face Directions:** In the construction of objects you may want your face in specific directions, such as using a boolean tool. You may also want to check to make sure your face is going in the right direction after creating a shape.
- **Show Isoparms:** Toggles Isoparm display on and off, useful for reducing clutter in complex scenes.
- **Show Silhouettes:** Determines if the silhouettes of an object will be shown.
- **Show Numeric Values:** This option will show numeric values for knots in NURBS curves and NURBS surface tools
- **Overlay Wireframe in shaded:** Shows isoparms on top of shaded views.

- **Control Hull on NURBS:** Displays the CV's on a NURBS surface.
- **Back Face Culling:** Culls the back faces to speed up screen redraws.
- **User Relative Left/Right Views:** The Universe Modeler's default Left and Right views are relative to the objects, the left view shows the view of the object from it's left side. If you want the view to be relative to the user, not the objects, toggle the option on.
- **Check Faces and Edges:** Toggled on this will check the faces and edges when the display for each new body is created. This can take longer times with more complex bodies.
- **Remove Bad Faces:** Any faces that ACIS finds problematic may be deleted from the body with this option. This could cause solid bodies to be opened up and should be considered more of a diagnostic option.
- **Save Separate Bodies While Loading:** Each geometry that is loaded into the project is automatically saved to the current directory. This serves as a backup if the program closes before the project is saved. You can retrieve all objects for a new project. These backup files are recognized by name as "individualsat0" and continue by number. Lights and images are not saved.

**Multicopy** The default setting here is 1. When you use a copy tool, or a command-D  
**Tab** when dragging, this is the number of copies that will be made. With a setting of 1, a single copy will be made. With a setting of 5, five copies will be made, and they will be placed at the same distance, rotation or scale made with the original translation.

- **Divide Total Transformation:**

The transformation is divided into a smaller transformation. For example, if you translated 10 units with 5 copies, each copy will be 2 units apart.

- Creation Tab** By default any curve primitive shape created will be automatically filled, becoming a two-sided surface object. If you want to create a wire object using a Curve Primitive tool, toggle off this setting. If you wish to make a single sided surface object, keep the Fill Curve Primitive toggle on, and toggle off the Make Surfaces 2 Sided option.
- Export Grouping Tab** Objects can be exported in several ways, with layers as groups, the selection as a group, bodies as groups or the entire scene as a group. By default Bodies as groups is selected as this generally is the way most people will work. With this selection each body in the scene will remain it's own entity when imported into another program.
- Color Tab** All the working elements in a scene can be assigned their own color. While defaults are provided for elements such as the background, rulers and grids, the user can customize these by clicking on the color box to the right of each element. You might find, for instance, that if you have several color objects in your scene red, that the red default highlight color is ineffective for showing you picked objects, so you might want to change the highlight color to something more suitable.



# The Toolbox

## 4.0 Working with the Toolbox

The toolbox contains 25 palettes, or categories, with each responsible for a different aspect of model creation. If a tool has additional options that can be accessed by double-clicking, that tool's icon will have a “...” in the lower left portion of the box. If a tool has sub-palettes with additional tools, it is marked with a small triangle on the lower right. These palettes also can be torn off and placed anywhere on the screen. Some tools can be toggled on or off, like Grid Snapping. When a tool is toggled on it will have a small check mark on it.

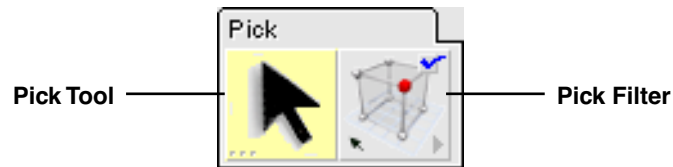


*Figure 4.0 — Special tool markers*



## The Pick Palette

To select objects in your scene, use the Pick tool. To determine what exactly it is that you're picking, set the Pick Filter. Many of the Universe Modeler's tools work in either pick or pre-pick mode. This means you either choose your tool and click directly on the object, or you pre-pick an object using the Pick tool combined with the Pick Filter to select the object and then choose the tool and double-click in null space to perform the function.



*Figure 5.0 — Pick Palette*

Two main reasons to pre-pick are:

1. You can pick exactly what you need without any surprises. For instance, if you want to round a shape's edge, you would use the Edge Pick Filter and click on the edge, which turns red. Once you are satisfied with your selection you can then use the rounding tool.
2. You can pre-select several objects, whether they are vertices, edges, faces, or whole bodies and then use a single tool to affect them all. For example, if you have a complex shape and you want to round five of the edges on the object, pick the edges using the Edge Pick Filter and the Pick tool. By double-clicking in null space, all the edges become rounded at one time.

### Pick Modifier Keys

Without using any modifier keys (like the shift key), you can continue to click and select as many objects as you wish. Clicking in a null space, unlike other programs, will not deselect your object. This is a built-in safety to prevent accidental deselection. If you want to deselect a single object, hold down the shift key and click on that object. If you want to deselect everything, hold down the shift key and click in null space.

### Pick Options

Double-clicking on the Pick tool brings up the Pick Settings box. Here you can adjust the Pick Aperture and toggle on and off the small square box that indicates the aperture's size. The aperture setting determines the width and height in pixels of a small box surrounding the point of the Pick tool. Anything falling within this area will be selected.

If you don't like having the small box on your screen indicating the pick area, you can toggle it off by clicking the Show Aperture box.



**Aperture 5**



**Aperture 11**



**Aperture 15**

*Figure 5.1 — Special tool markers*

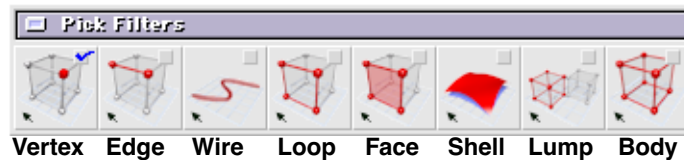
The aperture size also controls the snapping area when using the snapping tools.

## Pick Filters

The Pick Filter tool allows you to select the kind of topological element you want to pick, from a small vertex to the entire body.

Choose the topological element you want to pick before using the Pick tool. Your options outline the topology that the program uses to build objects.

Most filters require that you click on the element you want to pick. With the Face, Lump, and Body filters, you can click on an object's isoparm.



*Figure 5.2 — Pick Filters*

Although picking an object seems fairly simple, you may want to get used to the concept that you can make multiple selections without holding any key down, as well as how to deselect one or all the objects.

When you're done, hold down the shift key and click in null space, deselecting all the objects.



## The View Palette

There are two ways to navigate through your window views, using the tools available in the tool bar, and by using modifier keys.



Figure 6.0 — View Palette

**Panning** To move any screen's view up or down, left or right, hold down the spacebar and click and drag with the mouse. When you hold down the spacebar you will see the hand icon on the screen.

To use the Pan tool, simply click on the tool, and then click in the window you want to Pan the view with, and click and drag the mouse.

**Zooming** To move closer, or farther from an object in a scene, hold down your spacebar and your command key (*PC-Control Key*) and click and drag. You will see the magnifying glass cursor, moving to the right zooms in, moving to the left zooms out.

To use the Zoom tool, simply click on the tool, and then click in the window you want to Zoom the view with, and click and drag the mouse. Dragging to the right will zoom in, the left zooms out.

**Orbiting** To change the direction of the camera, you can orbit by either holding down the spacebar, command (*PC-Control*) and shift keys and click/drag on the screen, or you can hold only the “O” key down and click/drag. While you can orbit in any view, orbiting is best done in the camera, or iso view. Orbiting from one of your working windows (top, side, front) will just break that window out of its axis-aligned view and turn it into an axonometric view.

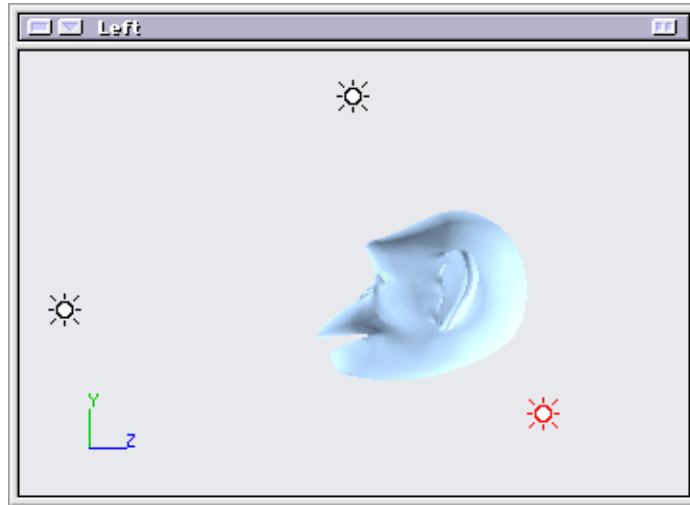
To use the Orbit tool, simply click on the tool, and then click in the window you want to Orbit the view with, and click and drag the mouse.

**Add light** Additional lights can be added to the scene using this tool. Lights can be manipulated like any other object. Select them with the pick tool, move them with the translation, delete them by selecting the light, and pressing the delete key (*PC- Backspace key*).

To add a new light, click on the tool icon, then click in any world view, to place the light.

The intensity and color of lights is adjusted using the Element Information Window, either under the menu **Windows/Element Info Window** or by pressing command + I (*PC- Ctrl + I*)





**Any number of lights can be added to a scene using the Add Light tool. The lights can be repositioned like any object in the scene, and the light settings can be altered under the Element Info window.**

*Figure 6.1 — Face with added lights*

Lights in the Universe Modeler are available only in “Radial” or “Point” form, meaning that the light will radiate out from the point to infinity.



# The Main Palette

The Main Palette is a composite of some of the most used tools in the tool-box, conveniently grouped to make locating frequently used tools easier. Most of the tools in the Main Palette are completely detailed later in the manual in their own groups. The Delete tool is only found in the Main Palette and is discussed here.

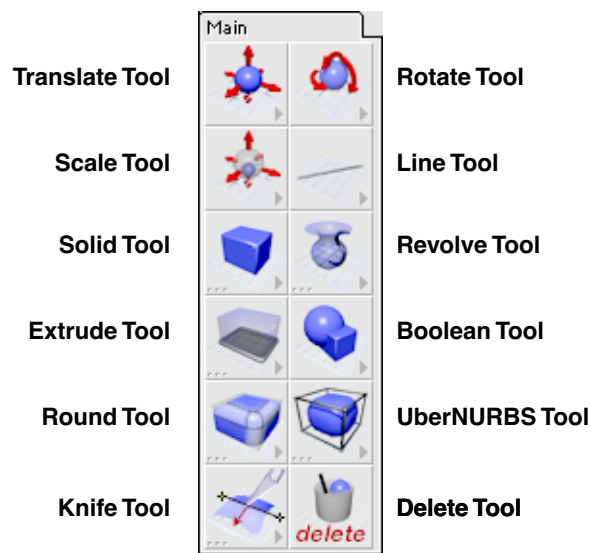


Figure 7.0 — Main Palette

**The Delete Tool** To use the Delete tool, either pre-pick or post-pick. With the pre-pick option, use your Pick tool to select all the objects to be deleted. Then select the Delete tool to delete the objects. With the post-pick option, choose the Delete tool and click on each item to delete. Another way to delete objects is to select them with the Pick tool and hit your delete (*PC- Backspace*) key. You can also delete objects by selecting them in the Layers Window and hit-

ting the delete (*PC-Backspace*) key. More information on working with layers can be found on page 537.

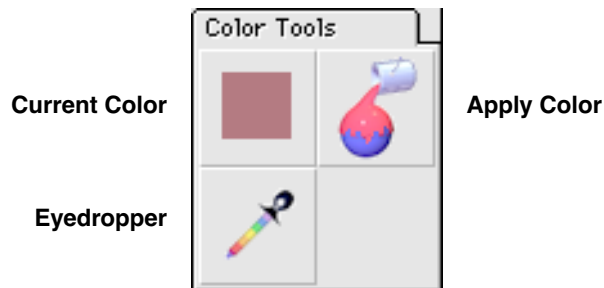
# Color Tools Palette

## 8.0 Introduction

Objects in the project can be set to any color for shading purposes. Objects can be colored at the body level, where every face of the object is colored the same, or at the face level, where you can have different colors for specific faces that you pick.

To color objects or their faces requires the use of the Color Tools palette. The Color Tools palette includes three tools:

- Color Picker, for choosing which color will be applied. The color picker uses several color models which you choose to arrive at the desired color. There is also a mixing area and a swatch area to create and store specific colors.
- Apply Color, which is the tool that you will use to apply the selected color to either the object or object faces.
- Eyedropper, which you will use to retrieve colors from objects or project elements such as background images.



*Figure 8.0 — Color Tools Palette*

### 8.1 Color Picker

Clicking on the Pick Color tool will activate the color picker. Five color palettes are available. Three palettes are active in the color picker by default, the HSV color sliders, the mixer and the swatch. Two other available palettes are ARGB and CMYK. Any of the palettes can be dragged in and out of the work area at any time. The color picker will retain the most recent configuration that you have set.

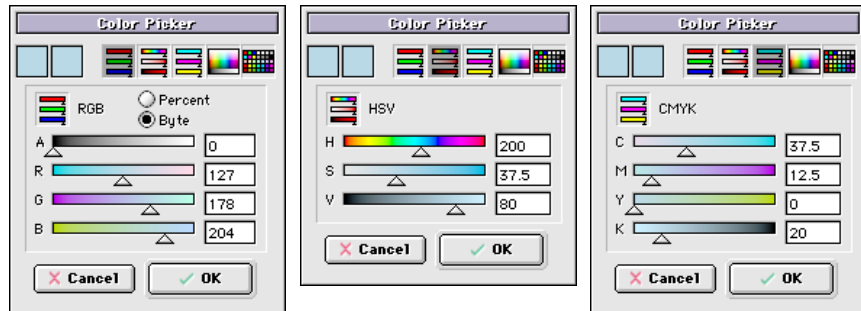
#### Creating Colors

To create the colors, a variety of color palettes are available, including a color mixer as well as RGB, HSV and CMYK palettes. Any color palette can be used by itself or in combination with other palettes.

The currently selected color can be applied to existing objects, and it will become the default color for newly created objects.

**Note** If **System Preference > Color** “Inherit Body Colors” is turned on, the wireframe will be drawn in the object’s color. If “Use Edge/Iso Colors” is turned on, the colors chosen in the entity scheme will be used to display the wireframe.

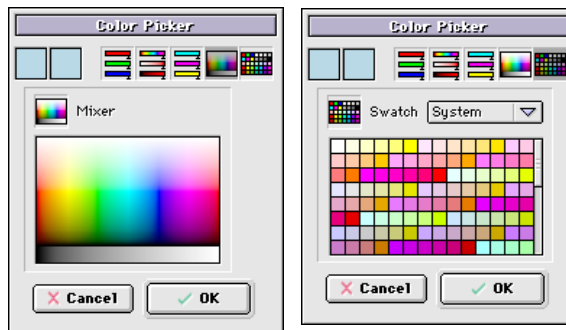
The illustrations in Figure 8.1 present the different color palettes and their specific functions. Each of these palettes are interconnected, and the ARGB, HSV and CMYK palettes will all update in real time to reflect the current color that you set in any of those palettes, as well as colors chosen in the mixer or swatch palettes.



**RGB:** Adjusts colors using the Red, Green and Blue settings, with an additional Alpha setting.

**HSV:** Adjust colors using Hue, Saturation and Value.

**CMYK:** Adjust colors by allowing alterations to the Cyan, Magenta, Yellow and Black.

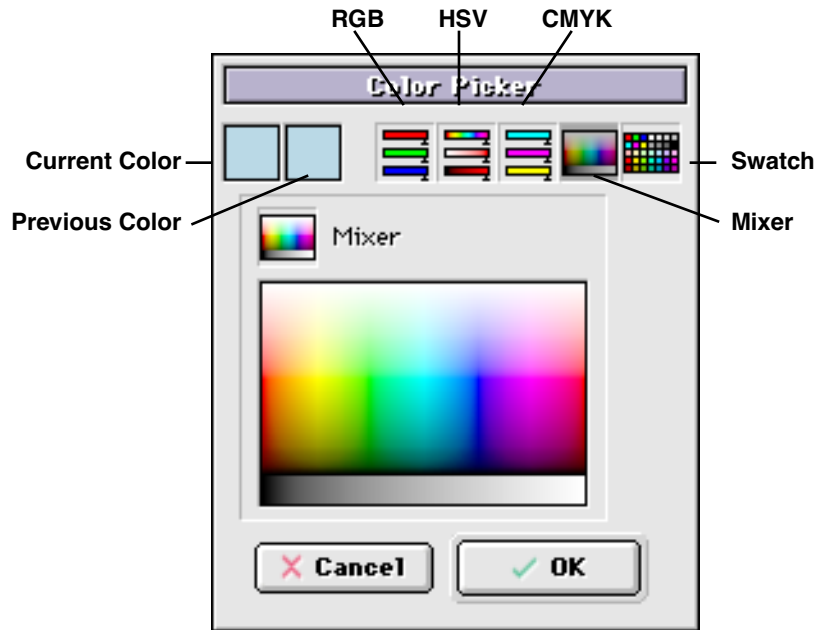


**Mixer:** This allows the user to use a less structured palette to create colors.

**Swatch:** Specific palettes can be used to allow a specific range of available colors. Palettes can be loaded and saved.

*Figure 8.1 — Available color schemes*

The illustration in Figure 8.2 shows the parts of the color picker that you need to be the most concerned with.



*Figure 8.2 — Color Pickers*

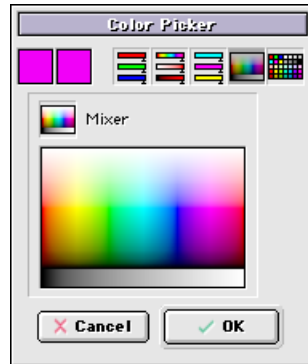
The color swatch at the top left of the window is the current color. This color is the active current color in the project. The color to the immediate right is the previous color, and will be the color stored as the active color if you press the Cancel button instead of the OK button.

### **Saving and Recalling Colors**

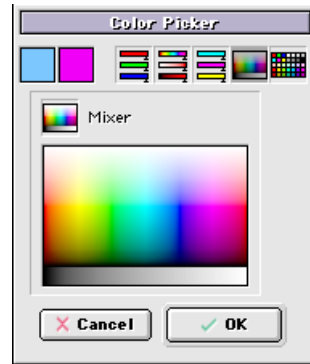
If you need to save a color that you would like to use later, place the color into the swatch palette. To do so, move the cursor into the swatch palette work space. As you pass the cursor over colors in the swatch palette, the cursor will change into an eyedropper icon. This indicates that you can




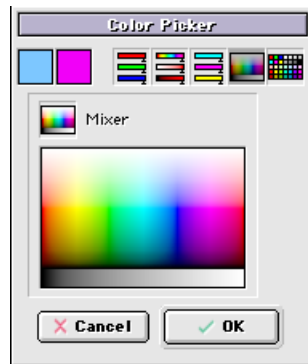
choose one of the colors underneath. As you move the cursor into an empty area of the swatch palette, the cursor will change into a bucket icon. This indicates that you can click the cursor to place the current color into the swatch palette for storage and later retrieval.



 Click on the Pick Color tool, the color palette box will come up.



 Use the color palette to change the color, the new color will be in the upper left box.



 Click on OK to make the color the currently selected color.

*Figure 8.3 — Creating a color using the color picker*

### Saving and Loading Color Swatches

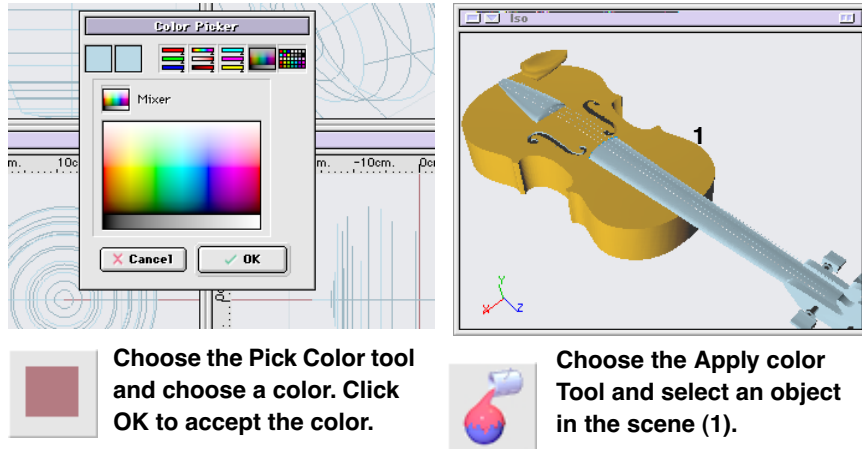
You can save and load whole swatches of color from within the color picker. To do this, use the **Swatch** popup menu in the Swatch palette. The menu offers a variety of functions as well as preset color palettes:

- New Swatch...
- Load Swatch...
- Save Swatch...
- Rename Swatch...
- Delete Swatch...
- Import Color Table (from Photoshop)
- System
- Black Body
- Gray Scale
- Spectrum
- Metals

## 8.2 Applying Colors to Objects

The Apply Color tool is used to take the current color and to apply it to an object or objects in the scene. Once a color has been chosen using the Pick Color tool, you can apply the color either by directly selecting the object with the Apply Color tool, or pre-selecting the object, and then using the Apply Color tool.

You can change the color of multiple objects to the same color at one time. To do this, use the pick tool, and then choose Apply Color. Using this method any number of objects can be selected before the Apply Color tool is used, and the color will be applied to all the selected objects.



*Figure 8.4 — Applying a color to an object.*

Another way to apply color is to use the Layer Window. With the Layer Window open, select all the objects that you want to color. Unlike in the world view, to make multiple selections in the Layer Window you have to hold down the shift key. Once all your selections have been made, go to the toolbox and click on the Apply Color tool. The currently selected color will be applied to all the highlighted objects.

You can also change the color of an object from the Element Info Window. To do this, choose the object and activate its Element Info Window (command or control “i”). When the window comes forward, activate the Properties tab. There you will see the color swatch for the object. Click on the swatch to activate the color picker and choose a new color for the object.

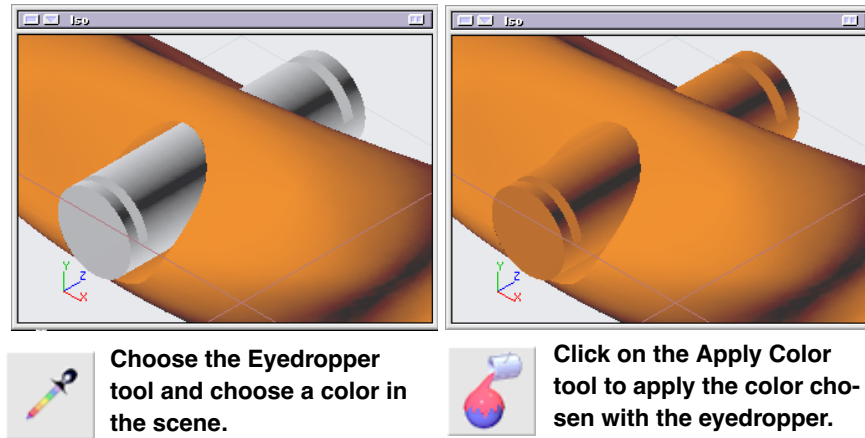
### 8.3 Eyedropper

The Eyedropper is used to select the current color of an object and place it in the color swatch so that the color may be applied to other objects. It can also be used to extract color from a background image or any color applied as object shading in any window.

You can use the eyedropper tool to insure that those objects that should be the same color are, in fact, so. The eyedropper tool is especially useful if you neglected to previously store the color into the swatch section of the color picker for later application.

To use the eyedropper tool to retrieve a color from another object, choose the tool while passing over an object, click on it. The current color will change to the new object's color in the Pick Color tool swatch.

To use the eyedropper tool to retrieve any color from the scene, such as a background image color, choose the tool and hold down the shift-key and the mouse button. As you drag, any color that passes under the cursor will change the value in the Pick Color tool swatch. When the mouse is released the color underneath the cursor will be set.



*Figure 8.5 — Applying a color from one object to another*

### 8.4 Some Notes on Color

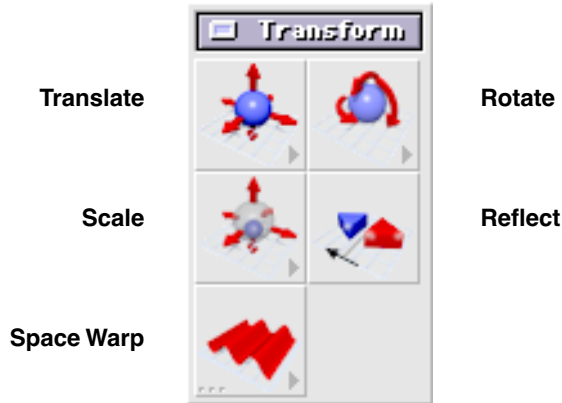
Working with colors in Universe Modeler is very easy, as long as you know the rules:

- New objects will always be created with the current color
- Any object that is created as the result of a boolean operation will be colored with the color of the first object selected. Face colors will not be retained. Booleans use object color only
- If you want to keep a color, store it into the Swatch palette
- You can use the Eyedropper to retrieve the color of any object in the project
- Hold down the shift key to retrieve any shaded color value, or the color of a pixel from a background image. This will also grab background colors, grid colors, and so on.
- To color a face, you must pre pick, and then use the Apply Color tool



## The Transform Palette

Objects can be moved, rotated, scaled, mirrored, and selected with tools from the Transform Palette.



*Figure 9.0 — Transform Palette*

### Translate Tool

This is the tool used to move an object. You can move any selected object on the screen using the pre-pick method, or you can simply choose the tool and click and drag on a body to be moved. The tool moves objects (horizontally and vertically) in each window. So if you want to move an object on the Z-plane, use either the top or side views to move the object back or forward.

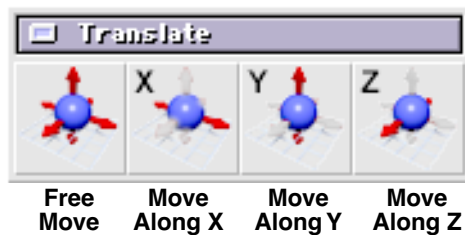


Figure 9.1 — Translation Tools

### Translate Modifier Keys

Movement can be constrained by holding down the X-, Y-, or Z-keys in relationship to the world, not the window, while moving the object. If you want to move an object straight to the left or right, hold down the X-key and click and drag on the object. You'll see that this constrains to move only on the X-plane. The same holds true for the Y- and Z-planes. Notice that you can't force items back on the window's Z-plane (back and forth) but you can achieve the same effect by going to another view and moving the object.

To duplicate an object when moving it, hold down the D-key and drag the object. A copy is left behind and the original object is moved. The copy is created in the object's original layer. The D-Key also works when rotating or scaling an object.

*Holding down the "c" key while dragging objects will enable the collision detection system. This prevents you from dragging one object through another. On the PC, collision detection may also be toggled on and off using the F3 key.*



### Rotation Tools

Objects can be either pre-selected, or selected after activating the Rotation tool. If the Rotation tool is used, it works much the same way as the Pick tool, you can click on several objects without using any modifier keys. To deselect an object, hold down the shift key and click on it. Once all the objects to be rotated are chosen, click and drag in the null space to rotate.

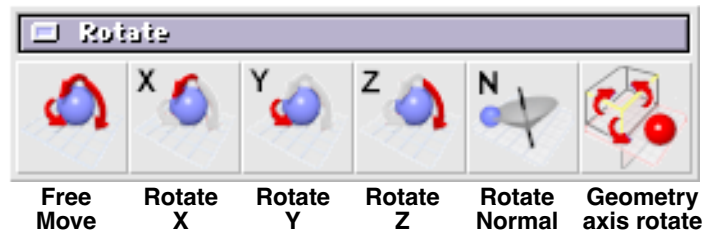


Figure 9.2 — Rotate Tools

Objects are pivoted, either on their own center axis, or on a user-defined axis, using the rotation tools. The Free Rotation tool will rotate any object on it's axis clockwise or counter clockwise in any window it is used, without a modifier key.

### Rotate About Normal

The Rotate About Normal tool will revolve objects around its normal axis. Multiple objects selected will rotate about the first object selected normal. Objects such as spheres and cylinders whose normal may not be determined will rotate in relationship to each individual window.

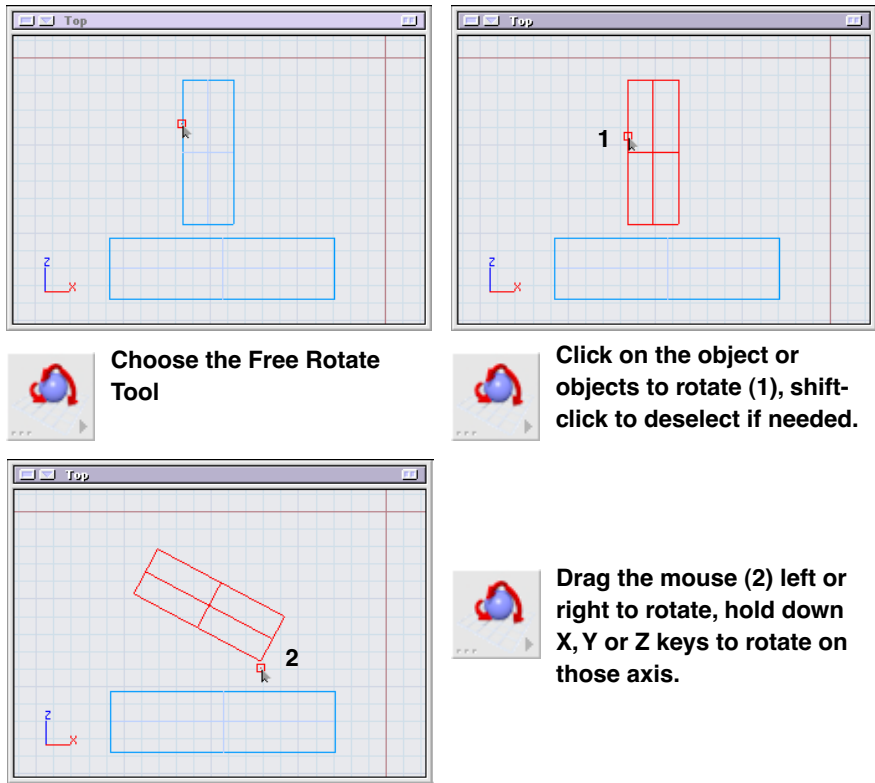


Figure 9.3 — Standard rotation

If several objects are selected at once, and then rotated, the objects will pivot off the average center point of all the objects.

### Rotate Modifier Keys

When using the Free Rotate tool, hold down the X-, Y-, or Z-keys to constrain the rotation so the spin occurs only on the X-, Y-, or Z-axis. You can perform several constraints on a single move by holding down any sequence of keys, one at a time. Holding down the D key with any of the rotation tools duplicates the object.

### Rotate About an Axis

Objects can be orbited around a selected object, wire, loop, face or lump. To use the tool, click once on all the objects to be rotated. You can use the orbit tool, or you can pre-select them. Double-click in null space once all the objects are selected, then click on the object they are to orbit.

The object can be any type of shape, and you can set the pick filter to choose what kind of element you want it to orbit. If you orbit a cube, for instance, and the pick filter was set at “Edge” then the selected objects would orbit the edge. If the “Body” filter was chosen, the objects would orbit the center of the cube.

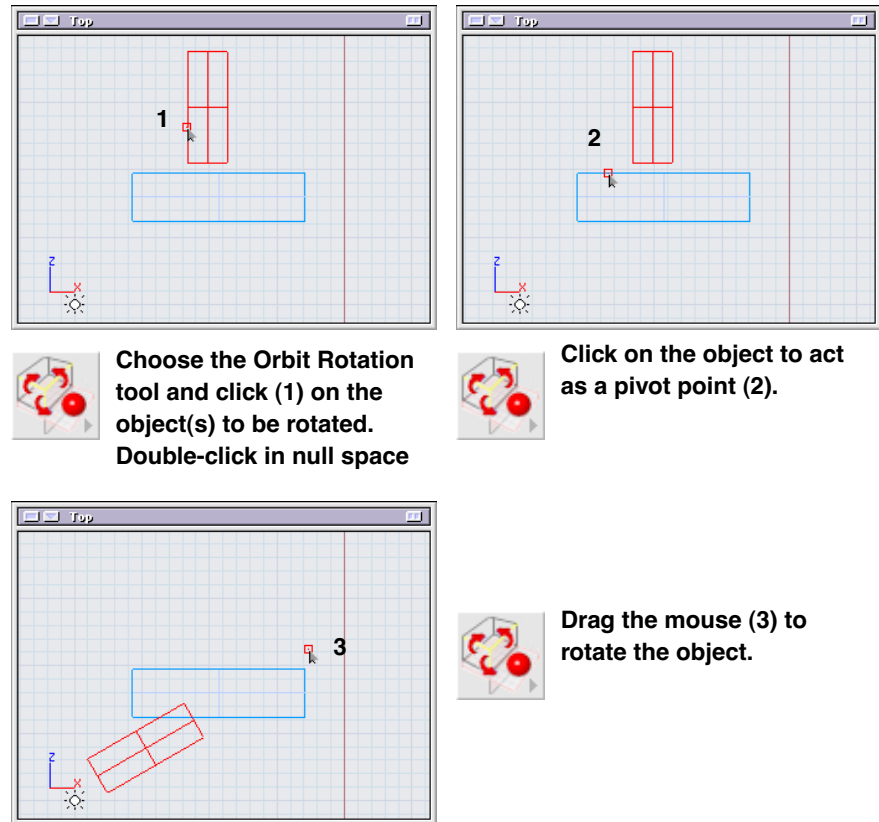
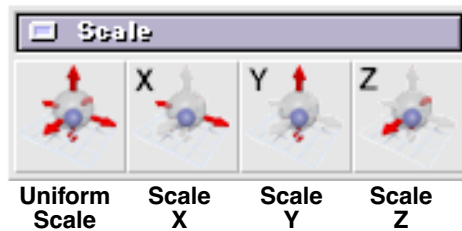


Figure 9.4 — Rotate about an axis

## Scaling Tools

There are two ways of scaling an object, by using the scaling tools and clicking and dragging to approximate the new size, or by typing the new scale percentage in the status bar.



*Figure 9.5 — Scale Tools*

The tool can use both pre-pick and post pick methods. To pre-pick use the Pick tool to select the object or objects to be scaled. Then choose any scale tool and perform the scaling. To use the post-pick method, choose the Scaling tool and click on the object or objects to be scaled. You can click on any number of objects, just like when using the pick tool. Holding down the shift key will allow you to deselect objects.

The Uniform Scale tool can also be used to scale non-uniformly. You can do this by holding down X, Y, or Z for scaling in any of these directions.

To use the numeric input, choose the tool, click on the object to highlight it, change the numeric input in the dialog box and hit the enter or return key. When you start the process the scale will be 1.0. Changing it to 2 would double the size, changing it to 0.5 would halve it.

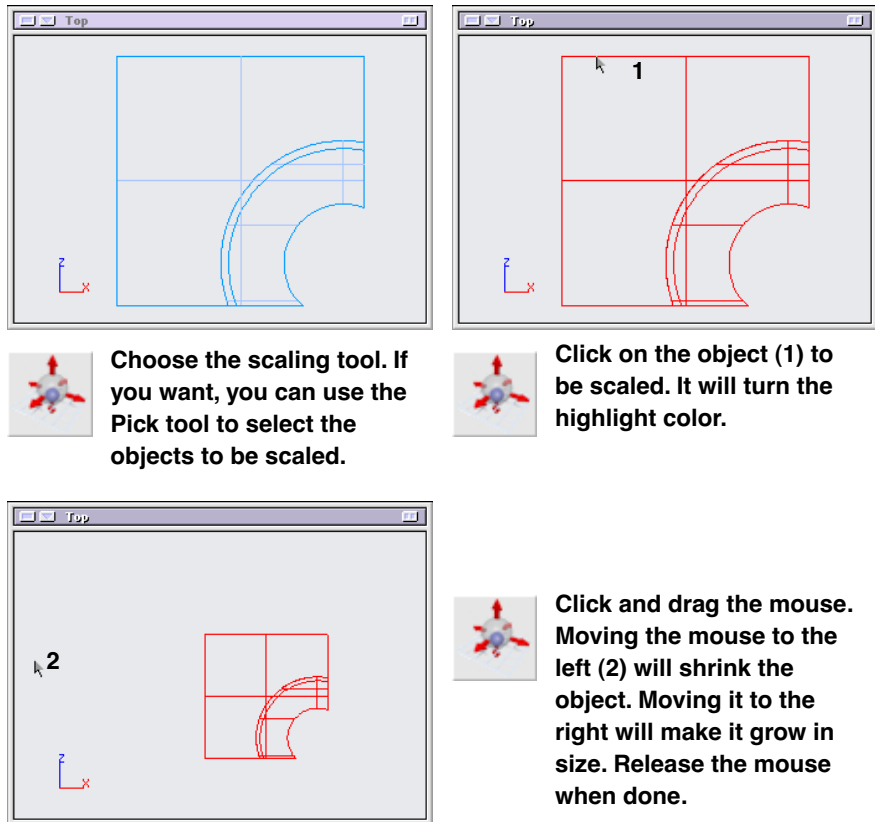


Figure 9.6 — Scaling an object

Holding down the “D” key when scaling will duplicate the object.

### Reflect Tool

A single object or a group of pre-selected objects can be mirrored using the Reflection tool.

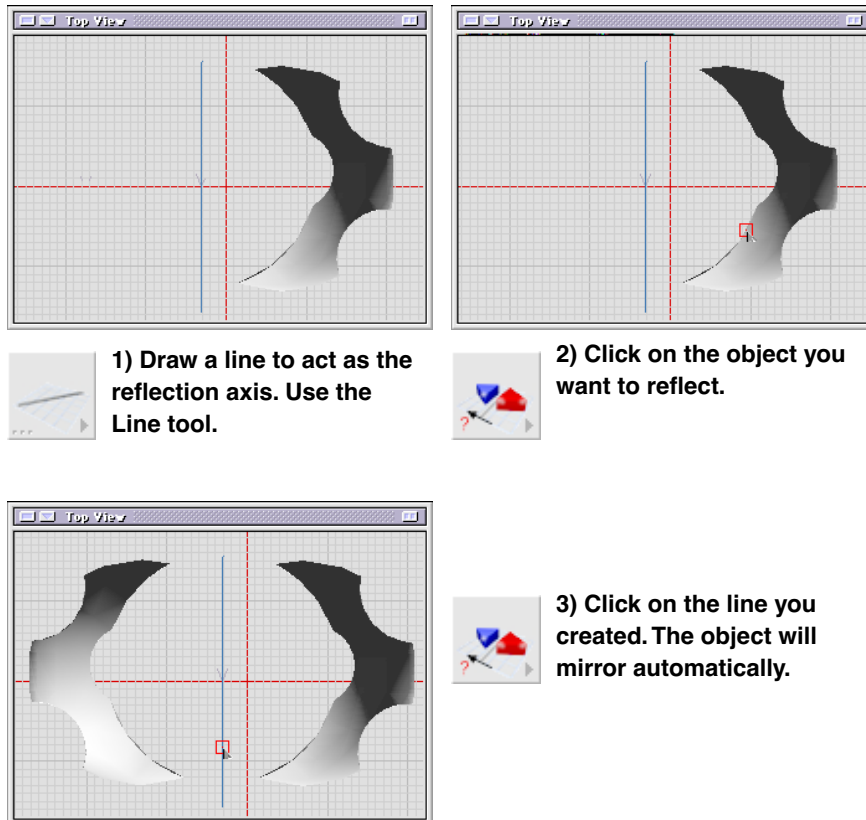
To use this tool, create an axis line to serve as the reflection point of reference. An existing line, such as an edge of an existing shape, also can be used.

The reflect tool always duplicates. The duplicate object will be placed on the originating objects layer.

The reflect tool can come in handy when creating symmetrical objects. By using the Bezier curve tool you can draw half of the curve, reflect it, and use the Join tool to bring the unconnected ends together.

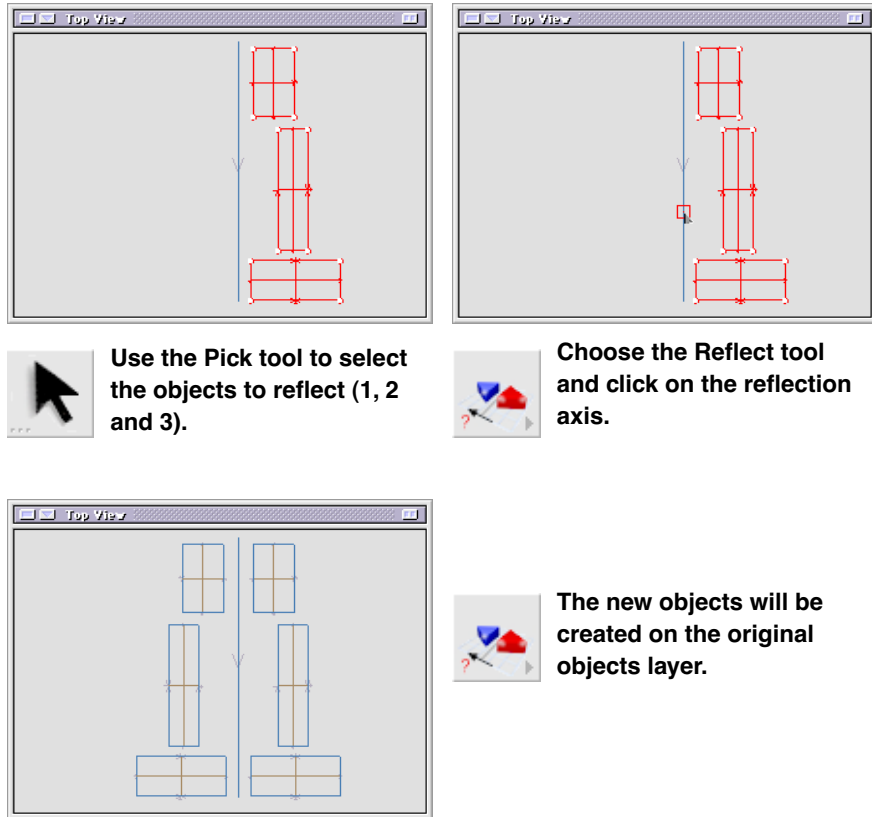
**Note:** *You do not have to draw a line with the line tool to create a reflection line.*

Select the object to be reflected. Double click in the null space. Drag your mouse to create a reflection line. The image is mirrored.



*Figure 9.7 — Reflecting an object*





*Figure 9.8 — Reflecting multiple objects*

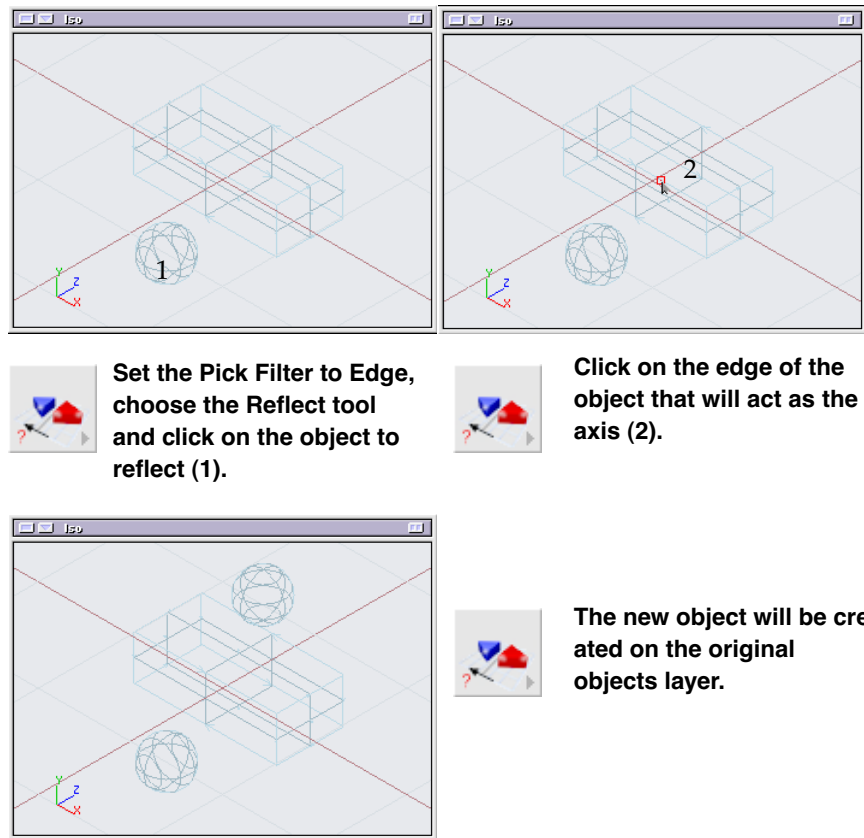
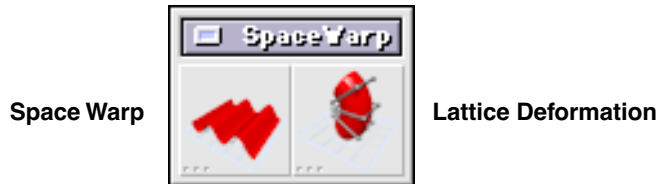


Figure 9.9 — Reflecting an object from an existing object

### Space Warp

There are two tools in the Space Warp toolset. Both affect the surface of the object.

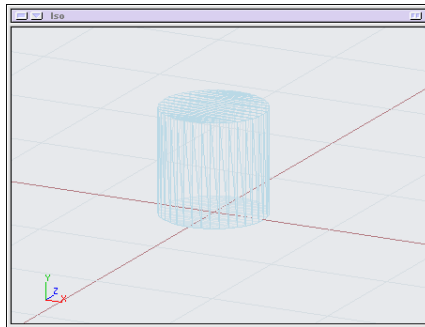


*Figure 9.10 — Space Warp tools*

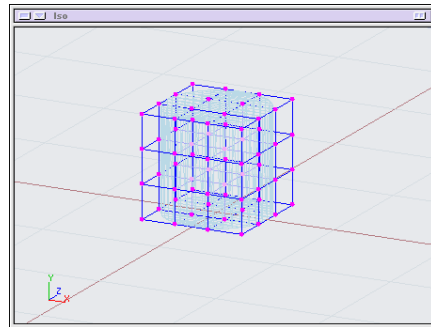
The Space Warp tools allow you to deform surfaces by entering mathematical formulas. Several preset formulas are included. You can save out any formulas you create yourself to a preset for later use or sharing with other users.

### Lattice Deformation

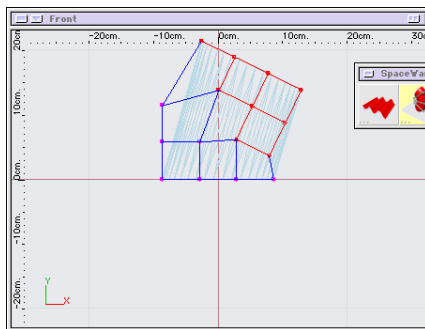
This tool is also called FFD (Free Form Deformation) in other packages. It puts a cage around the object in question, then as you amend the cage the object will be deformed. FFD's are best used to clean up models or amending finished pieces.



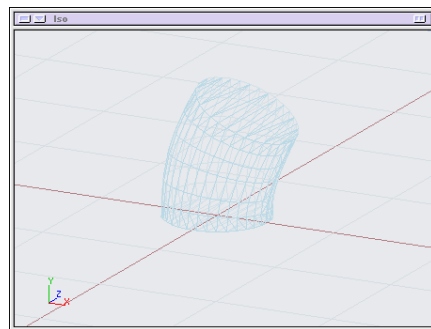
**Start with an object. In this case a cylinder primitive was used.**



**With the Lattice Deformation tool chosen click on the object.**



**Amend the control points of the lattice. In this case, the points are rotated on the Z axis.**



**Double click in null space and the deformation is performed.**

*Figure 9.11 — Performing a lattice deformation on an object*

The Lattice Deformation tool allows you to deform an object by creating a lattice or “cage” around the object. By translating, rotating and scaling the vertices of the cage you can reshape the object. Creating more cage divisions gives you more control in a particular direction.

**Example:** *To turn a simple cylinder into a piece of macaroni, bend the cylinder along the Y axis. You would need more divisions along the Y axis and very few along the X and Z axis.*

### Space Warp Tool



*Figure 9.12 — Space Warp tool*

Space Warp tools are different from Law Surfaces, because they do not create new surfaces; they simply modify the existing one with a mathematical formula. For more information about Laws and Universe Modeler syntax see the “Law Tools Palette” on page 155.

To use the Space Warp tool, select the object you want to warp. Double click on the tool to open the dialog window, and input a Universe Modeler syntax. Some presets are provided.

Space warp tool expects laws to be specified as a vector function transforming one  $x,y,z$  space to another. An identity space warp, one that does nothing to the points, will be  $X = x, Y = y, Z = z$ . In Universe Modeler syntax, this warp will be coded as **vec(x,y,z)** where the value of new  $x, y, z$  are the same as the old. Not of much use, but illustrates at the simplest warp.

Now let us take another warp which twists the space around like a screw.

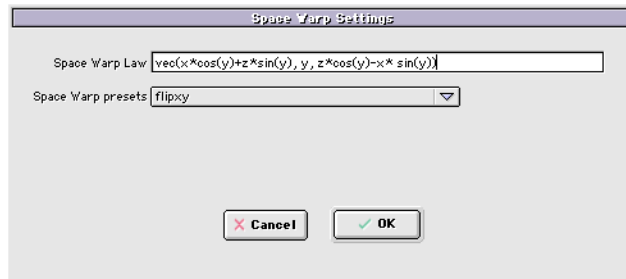
Let the twisting axis be Y-axis centered at the origin. In Universe Modeler notation that translates to a space warp, see Figure 9.13.

$$X = x \cos (y) + z \sin (y)$$

$$Y = y$$

$$Z = z \cos (y) - x \sin (y)$$

*Figure 9.13 — Parametric formula for screw Space Warp*

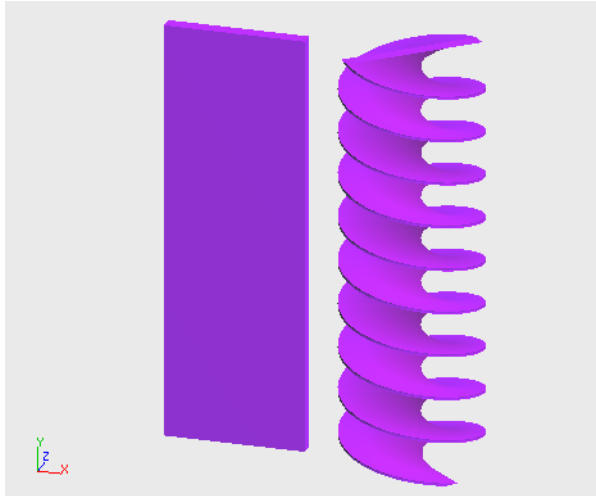


*Figure 9.14 — Space Warp settings window*

$$\text{vec}(x*\cos(y)+z*\sin(y), y, z*\cos(y)-x*\sin(y))$$

*Figure 9.15 — A Space Warp Universe Modeler syntax*

Apply that to a straight block or ribbon standing up on Y axis centered at origin and you will get a twisted ribbon.



*Figure 9.16* — **Space Warp applied to a straight block, twisted ribbon**

The SpaceWarpLaws.txt file is simpler because it does not have any ranges. The space warps are infinite in Universe Modeler.





## The Snapping Palette

Snapping allows for precise alignment or creation of elements, either in relationship with various elements or with the grid. While this is certainly beneficial with the creation of some exact shapes, some tools, like Net Surfaces, require exact vertex placement. The Modeler's Snapping tools allow for the alignment of elements along the main topological building blocks, such as vertices, edges and faces.

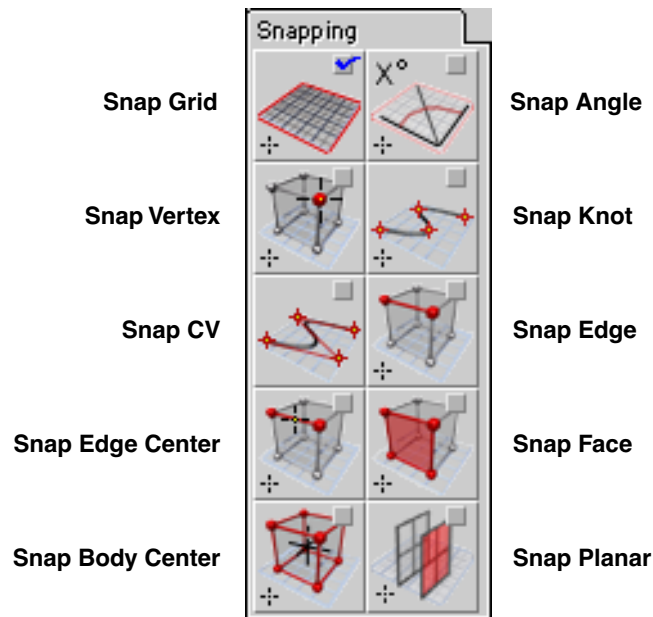


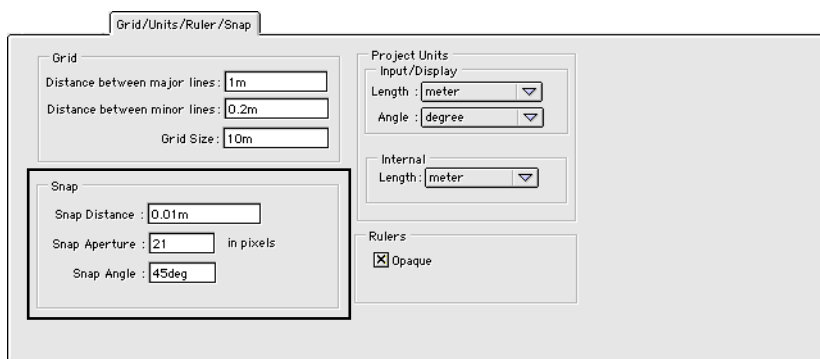
Figure 10.0 — Snapping Palette

To use the tool, click on the icon that represents the kind of snapping desired. When snapping is turned on for any element, a check mark will appear on the upper right-hand corner of the tool icon. Feedback on the

tool can also be seen in the status bar, where, for example, the words “Grid Snapping OFF” or “Edge Snapping ON” can be read.

Holding down the “C” key will keep a snapping element on it’s own plane. If looking down on the XZ plane for instance, and you’re aligning an end-point of a line to a line below it, holding down the “C” will keep the end-point your moving on it’s own plane and not allow it to hop down to the lower vertex.

Snap distance, aperture and angle values can be changed from the Grid / Units / Ruler / Snap tab of the Document Preferences dialog box.



*Figure 10.1 — Snapping Options*

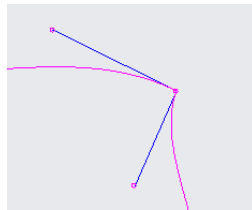
### Grid Snapping

With Grid Snapping turned on, you can precisely create elements that line up, have the same beginning points, or follow a perfect X- or Y-path. One use for this is the Revolve or Reflect tool where you can create an axis by using the Line tool. With Grid Snapping on, you can be assured your axis is perfectly horizontal or vertical. Your cursor will snap to the grid line inter-

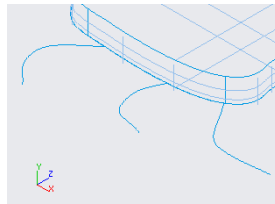
section closest to the point, or hot spot, of the cursor. Grid snapping has three keyboard shortcuts available, F5 toggles on the grid snap, F6 doubles the snap distance and Shift-F6 halves the grid snap distance.

### Angle Snapping

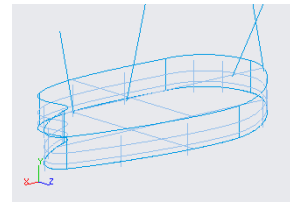
Angle snapping will take effect whenever angles make sense, such as the rotation of an object or the creation of polylines. The angle will be in relationship to the current working window.



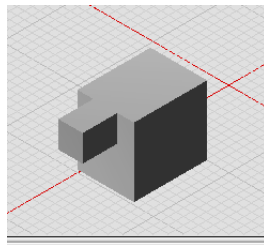
**Vertex Snapping**



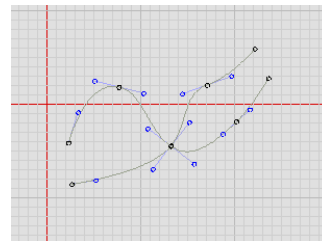
**Edge Snapping**



**Face Snapping**



**Edge Center Snapping**



**Knot/CV Snapping**

*Figure 10.2 — Some commonly used snapping types*

### Vertex Snapping

With vertex snapping on, tools will snap to any vertex that falls into the snap aperture. If you draw a curve and you want to begin at exactly the same point as a previous line, turn the Vertex Snap on, then choose your tool and draw. If you want to align two existing points, make sure Vertex Snapping is on before you go to the line editing tools to reposition the elements. As with all the snapping tools, any time a user inputs a position using the mouse, the clicks will have to be interpreted as 3D positions. Vertex snapping will enable accurate interpretation of the clicks.

### Edge Snapping

When Edge Snapping is on, the cursor position will be attracted by edges in the scene. For the attraction to work, the cursor must be near enough to an edge. For example, some portion of the edges must come within the snap aperture of the cursor.

This tool is useful when objects need to be created on, or moved upon, edges perfectly. For example, when creating a curve that starts from the body of another curve, set Edge Snapping on before creating the second curve. Using your favorite curve tool, click near the first edge. If the first edge is within the snap aperture distance of the click, the program will create the first knot of the new curve on the body of the first edge.

### Face Snapping

With Face Snapping toggled on, any object, such as a wire tool or a solid tool, will be cleanly aligned with any face that falls within the snap aperture. For example, if you wanted to create two block primitive shapes that line up exactly at their faces, create the first block, turn on Face Snapping

and then create the second block, making sure that initial point of the block's creation falls within the snap aperture's distance to one of the faces of the first block.

### Knot Snapping/CV Snapping

When creating curves with either the Bezier or the NURBS tool, Knot Snapping allows you to accurately place knots where other knots exist. Knots need to be defined prior to snapping. CV's can also be snapped to.

### Edge Center Snapping

To create the beginning of a line along an existing edge center, or to place an object squarely on the center of an object's edge, use the Edge Center tool. Edge Center Snapping will also work with the Add Vertex tool in the UberNURBS palette.

### Planar Snapping

Planar Snapping allows you to begin creating a curve in one view, switch to another view and continue creating the curve on the same plane.

**Example:** *Start creating a Bezier curve by clicking the first point down in the front view and switch to top view to continue creating the curve. The segments of the curve, however, will continue to be created in the plane you initially set by clicking in the front view window.*

The planar snap tool is very useful for creating net surface wires and other "multi-planar" topologies where CVs must intersect, but the curve needs to travel into a second planar direction.

### Order of Snapping

More than one kind of snapping can be turned on at any time. If several snaps are on, they function in the order of the following priority:

1. Planar Snapping
2. Snap to Vertex
3. Snap to Knot
4. Snap to CV
5. Snap to Edge Center
6. Snap to Edge
7. Snap to Face
8. Snap to Body Center
9. Snap to Angle
10. Snap to Grid

## Copy Tools

Objects can be copied in a variety of ways. After picking an object, holding down the “D” key while using any transformation tool, move, rotate or scale, will make a duplicate of the object without any use of the copy tools. The number of copies created when using this very basic form of duplication is determined by the number input in the Multicopy box under **System Preferences**. The duplicates created will use the same information used to create the initial duplicate, and will repeat it as many times as input into the Multicopy box. By default the Multicopy box is set at 1, so only a single copy is made each time the “D” key is held down during a transformation.

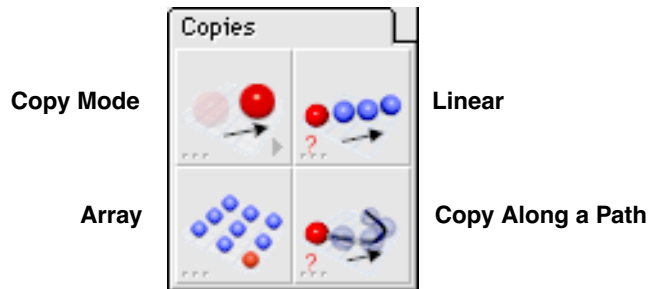


Figure 11.0 — Copies Palette

Under the Copies Palette, a series of duplication tools are available that will aid you in either creating copies of object by positioning them by eye, copying them to exact attributes, or copying them along a path.

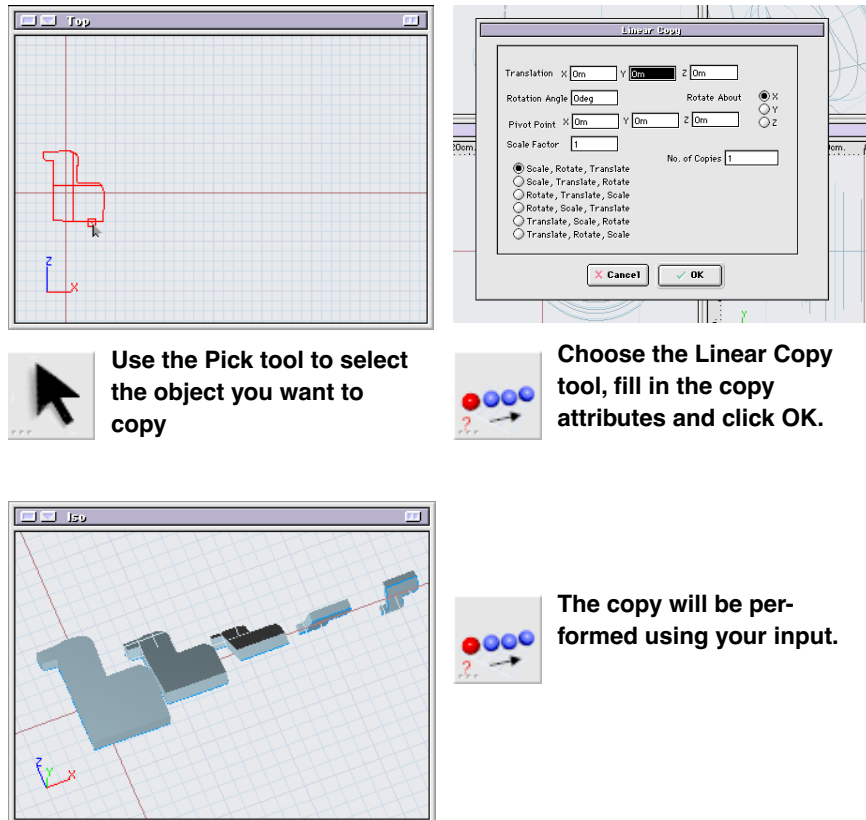
### Copy Mode

This tool toggles ON/OFF the copy mode. It works very similar to holding down the “D” key. If the Copy Mode is on, as will be indicated in the Status area when the tool is clicked on, then any following transformation, (move, rotate and scale), will result in a copy, and the number of copies will be determined by the number input in the System Preferences/Multicopy options box.

### Linear Copies

Copies can follow a rigid set of user defined guidelines including the distance, direction, rotation, pivot, scaling, axis of rotation and number of copies. To use this tool, select an object then double click on the Linear Copies tool. Fill in the numeric input to determine how the copy will be made, and then hit OK. The operation will be performed using your input.





*Figure 11.1 — Linear Copies*

The number of copies is determined by the input in the options box, and will override the number set in the Preferences/Multicopy field.

More than one object can be copied at a time. To do this use the Pick tool to select the objects first, and then perform the copy as described. The objects will pivot around the average point of the combined objects.

On the bottom of the options box is a toggle to determine the order of the operation. Experiment with it to see the difference the order of transformation makes.

Radial arrays can be created by setting the translation component to zeros, and specifying rotation angles.

A helical array can be created by specifying a rotation and a translation along the rotation axis direction.

### **Array Copies**

Copies can be made in more than one plane; the Array Copy tool will create copies in three dimensions. To use this tool, select an object then click on the Array Copies tool. Fill in the numeric input to determine how the copy will be made, and then hit OK. The operation will be performed using your input.

Copies can vary in the distance, scale and number.

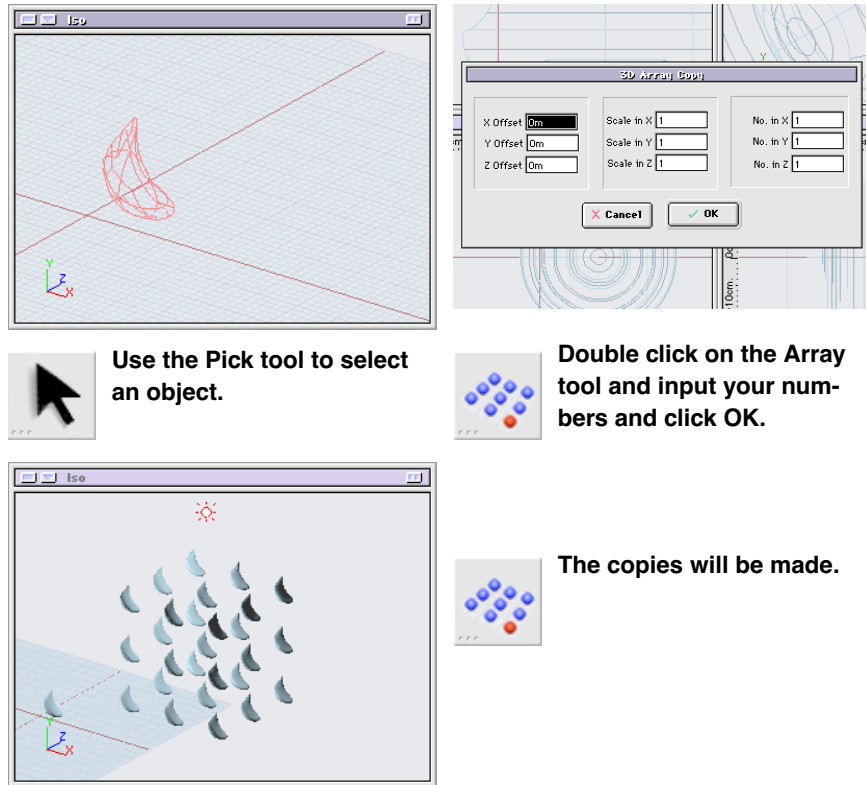


Figure 11.2 — Array Copies

More than one object can be copied at a time. To do this use the Pick tool to select the objects first, and then perform the Array Copy.

### Copy Along a Path

One very powerful copy method is the Copy Along Path tool. Any object, wire, solid, surface or UberNURBS can be copied along a path. If you want several objects copied, they can be joined first; in this manner a fish made

of many different parts could be copied along the path to quickly make a school swimming in line. Since wires can be copied along a path, this tool can be used to quickly make a series of wires that can later be modified and skinned. From snake bodies to twisting cables, this tool wields a great deal of power. The path must be a simple wire, open or closed. Wires with branching paths are not legal and will not work.

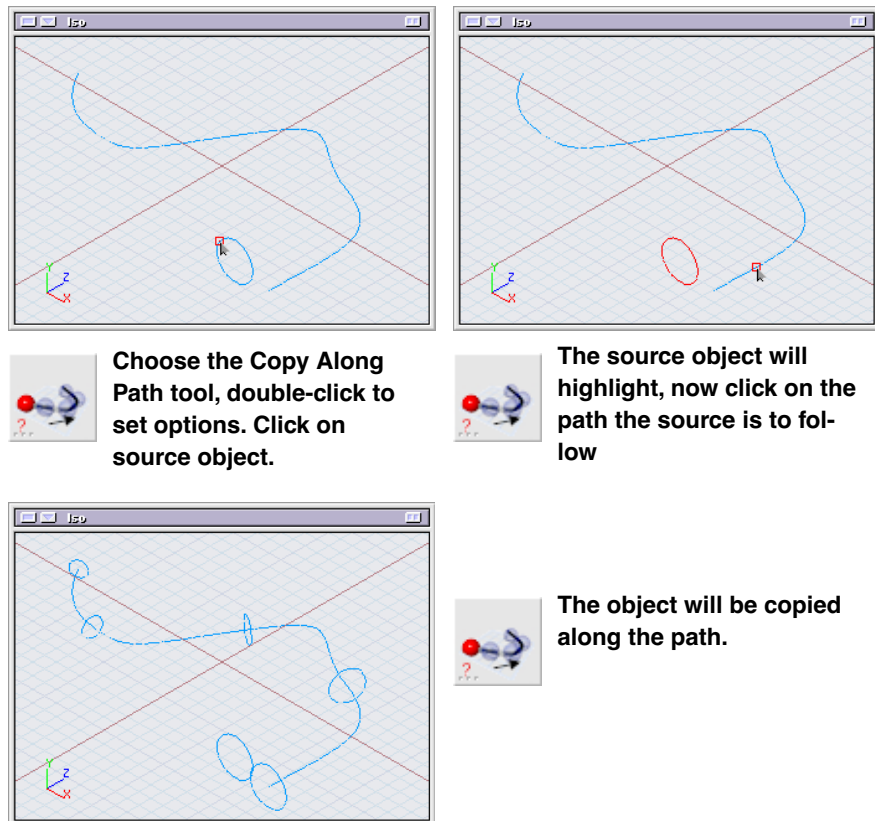


Figure 11.3 — Copy along a path

Several options are available by double-clicking on the tool. The “Direction” input boxes allow you to orient the source object in relation to the path. Choosing the “Align with tangent” option will re-orient the object as it is copied along the path, allowing it to follow the curves.

The number of copies along the path can either be set in the Preference / Multicopy menu, or you can have an object placed only on the knots of a curve.

While the object is being copied, it can be scaled and rotated by entering percentages and angles in the available boxes.



## Curve Primitive Palette

Curve primitives and their extended Bezier and NURBS tools form the foundation for the bulk of the Universe Modeler's architecture. The NURBS curve and the Bezier curve tools also exist in their own palettes and are offered here for easy access. The Line tool can be used to create an axis for either the Revolve tool or the Reflect tool. The Rectangle, Polyline, Regular Polygon, Circle and Ellipse tools by default will create a filled, two-sided surface or sheet body.

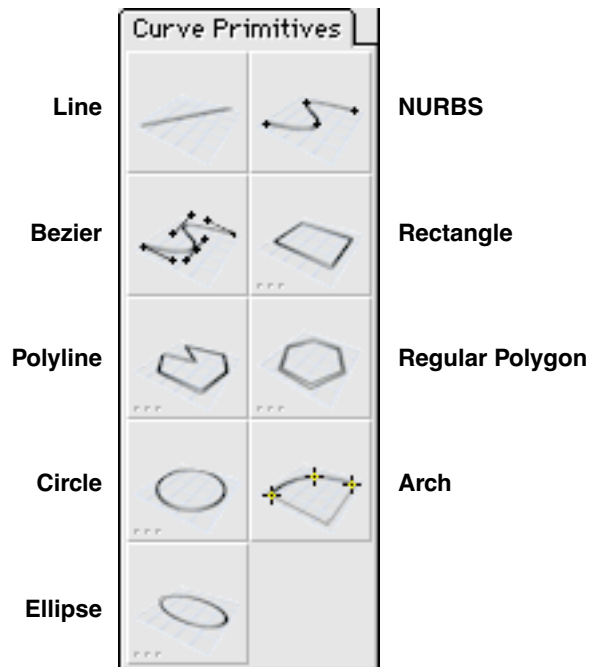
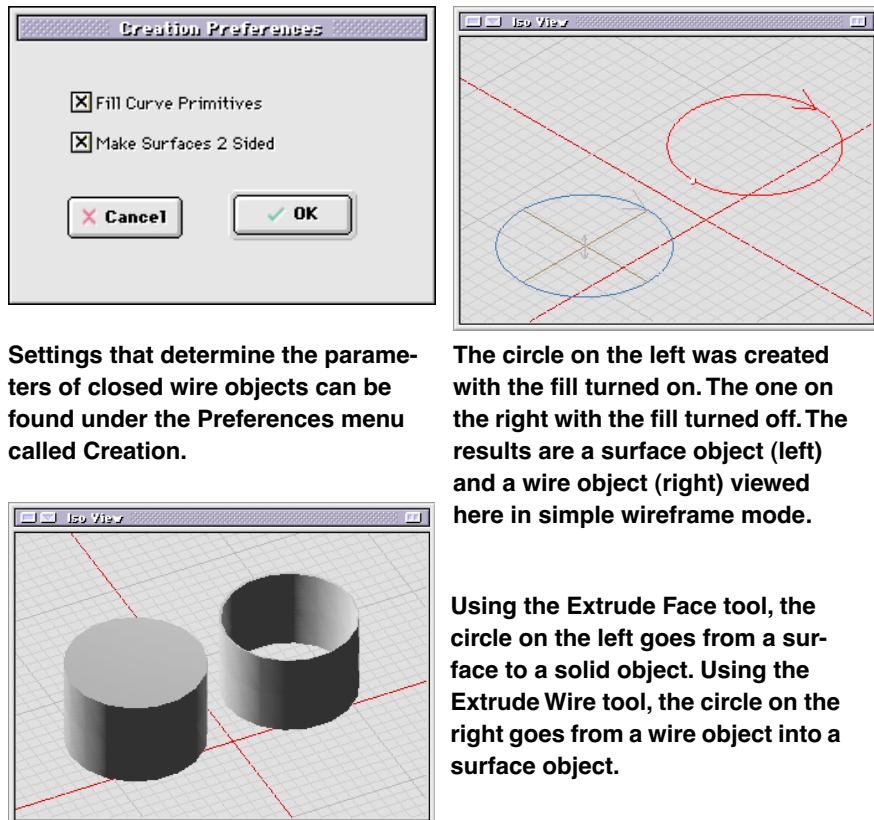


Figure 12.0 — Curve Primitives Palette

If you want just a wire from the tool, go to **Edit/System Preferences** and go to the Creation tab and toggle off “Fill curve primitives.” For more explanation on two-sided and single-sided objects, see “Boolean Operations on Single and Double-Sided Solids” on page 363.



Settings that determine the parameters of closed wire objects can be found under the Preferences menu called Creation.

The circle on the left was created with the fill turned on. The one on the right with the fill turned off. The results are a surface object (left) and a wire object (right) viewed here in simple wireframe mode.

Using the Extrude Face tool, the circle on the left goes from a surface to a solid object. Using the Extrude Wire tool, the circle on the right goes from a wire object into a surface object.

Figure 12.1 — Copy along a path



### Line Tool

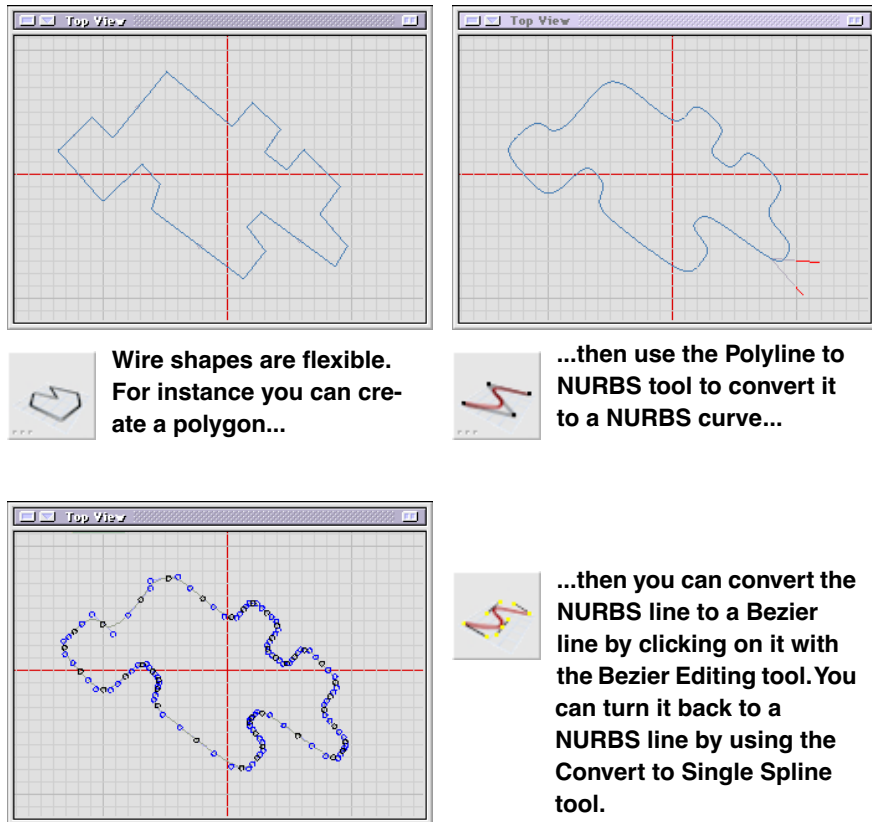
The Line tool can be used to create a single, straight wire that can become part of an edge of an object, or it can be used in the construction of more complex shapes. For example, it can be used as an axis for a revolved object. To use the tool, click to set the start point of the line, and while holding down the mouse button, drag and release the mouse button where you want the line to end.

### Bezier Tool

The Bezier Curve tool works in much the same way as Bezier drawing tools in 2D programs. Unlike the planar Line tool, the Bezier curve can be used to make a fully dimensional wire object by either drawing and using top-, front- and side-views, or by creating a planar Bezier curve and editing it to be 3-Dimensional. The Bezier tool in the Curve Primitives palette is actually only one Bezier tool from a larger, more robust Bezier palette located further down the toolbox.

### Polyline Tool

An open or closed polygonal wire object can be created using the Polyline tool. Use the tool by placing the cursor where you want to start the line and click the mouse. Now move the cursor where you want to place the next point and click again. A straight line will be drawn between the two points. You can continue to click to create as many lines as you want. To finish the line, either double click on the last point to create an open polygonal wire, or click on the first point to close the wire. When closing a wire make sure you click so that the first point falls inside of your pick aperture.



*Figure 12.2 — Flexibility of curves*

Keep in mind that a shape created with the Polyline tool can later be converted to a NURBS curve. Some people may like the control that comes with working with the Polyline tool and choose to convert it later to smooth it off into a curved NURBS curve. To convert the line to a NURBS curve, use the Polyline to NURBS tool. For more information on this See “Polyline to NURBS” on page 476.

### Circle Tool

The Circle tool makes a perfect circle. After choosing the tool, the first click establishes the center of the circle. Holding down the mouse and dragging establishes the radius of the circle. Releasing the mouse finishes the circle.

### Ellipse Tool

The Ellipse tool requires two sets of click-drags to create the finished wire object. The first click establishes the center. Holding down the mouse and dragging establishes the radius of the base circle. After letting up on the mouse button, click again and drag to establish the height of the ellipse. Dragging to the left shortens the ellipse, dragging to the right increases its height. Lifting up on the mouse completes the ellipse.

### NURBS Curve Tool

Like the Bezier tool, the NURBS (Non-Uniform Rational B-Splines) tool can be used to create either complex 2D or 3D wireforms. The NURBS tool in the Curve Primitives palette is actually only one NURBS tool from a larger, more robust “NURBS Surface Palette” on page 302

### Rectangle Tool

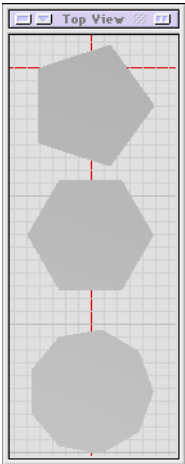
With the Rectangle tool selected, clicking the mouse button establishes a corner of the rectangle, holding down the mouse and dragging adjusts the width and length of the rectangle, and releasing the mouse establishes the final shape of the wire object.

## Regular Polygon Tool

The key for getting the most out of the Regular Polygon tool is to set its options before creating a shape. If you are going to make a stop sign, you will need an eight-sided polygon. By double-clicking on the tool icon, you get the tool's options box where you can input the number of sides. After closing the box, you can click and drag just like a circle to make the shape.



**Regular Polygon option includes input for the number of sides.**

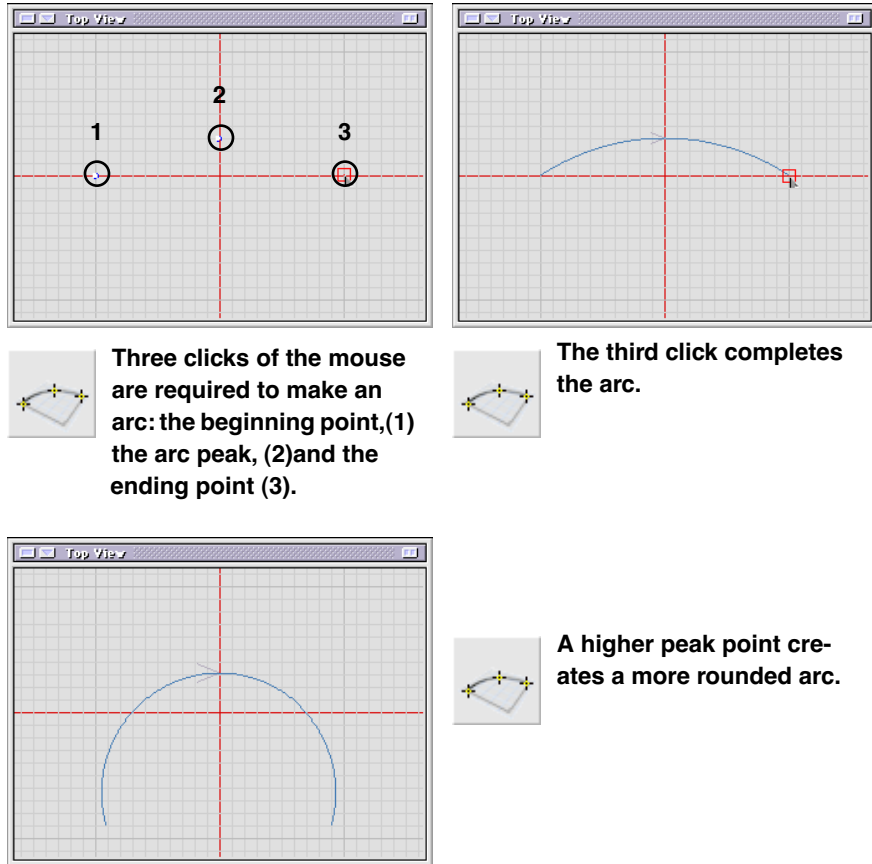


**Surface shapes created with the Regular Polygon with 5, 6 and 9 sides.**

*Figure 12.3 — Regular polygon options*

## Arc Tool

To create an arc, the Arc tool requires three input points: the originating point, the arcs mid point, and the ending point. You can select the three points in any single window: front, top or side to create an arc that lays flat in a given window or you can click on any combination of windows and create an arc that curves through X-, Y- and Z-space.



*Figure 12.4 — Creating an arch*

An arc can head in any direction and can go either clockwise or counter-clockwise, depending on the location of the arc points.



## Solid Primitives Palette

Solid primitives enter the world as solid objects. This means they are completely enclosed, one-sided objects. Solid primitives can be used as foundations for more complex shapes or edited, hacked, knifed, or booleaned into any shape imaginable. Solids can also be converted to UberNURBS.

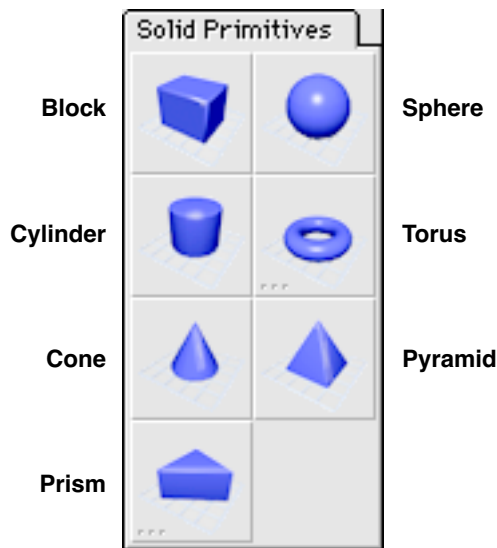


Figure 13.0 — Solid Primitive Palette

With the exception of the Sphere tool, all the solid primitives require a series of click-and-drag operations to define their overall size, width and other parameters. With several shapes, this requires the you to establish one set of parameters in one window and then hop to another window to complete the operation.

## Block Tool

To use the Block tool, click and drag to establish the width and height of the cube. Then use another view to click and drag the depth.

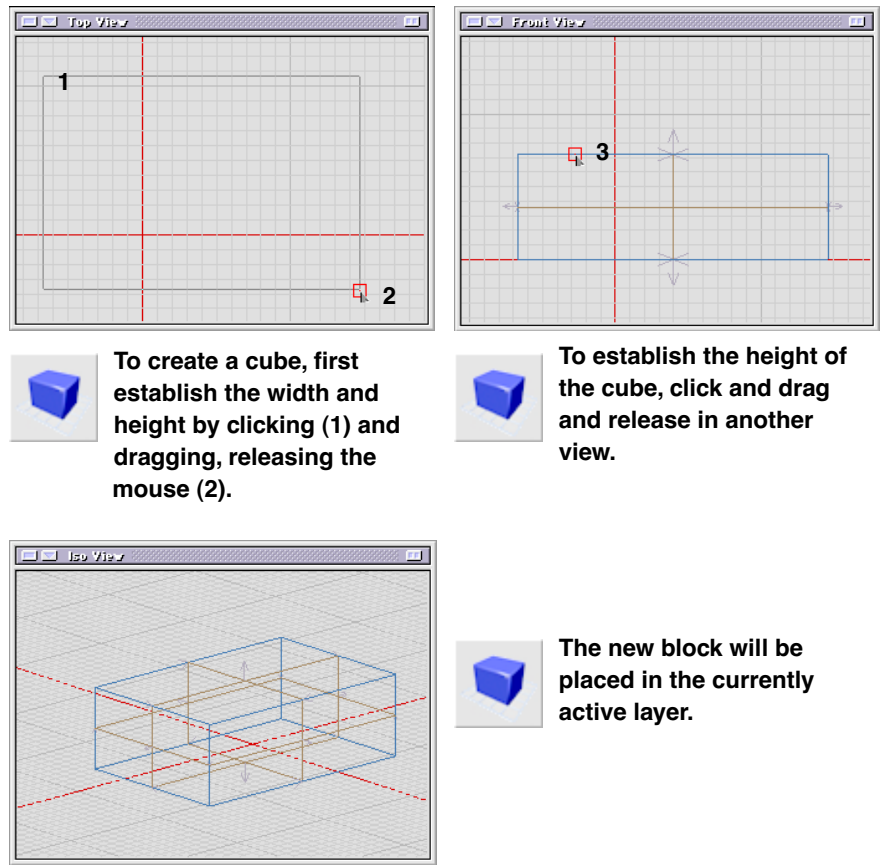


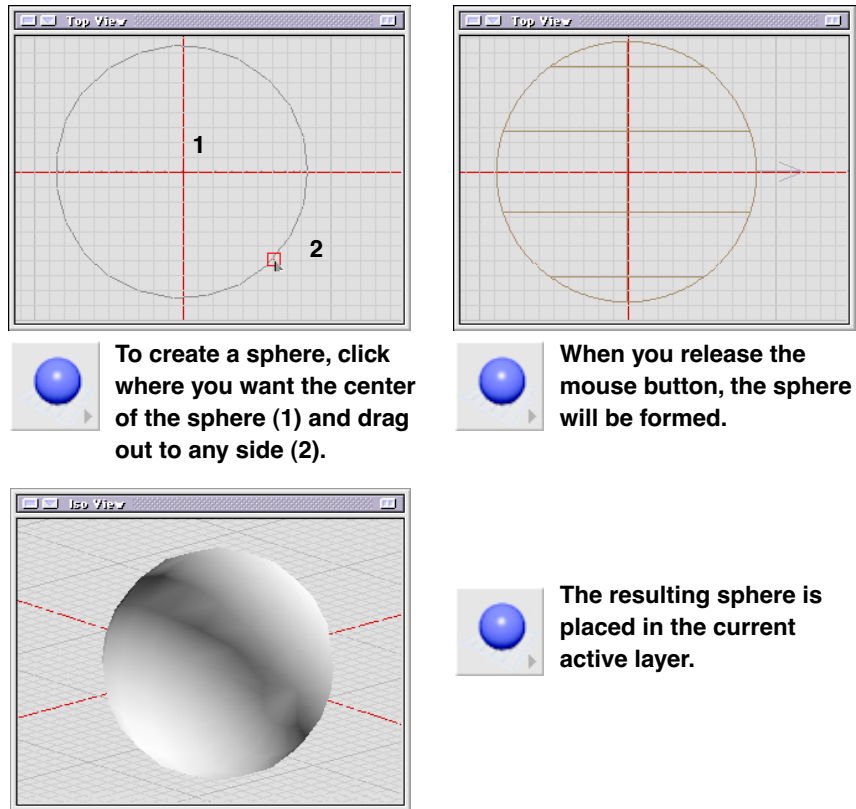
Figure 13.1 — Creating a block



Beyond its obvious uses, the Block serves as a good foundation for using the UberNURBS tool. Keep in mind that any newly created object will be placed in the currently active layer.

### **Sphere Tool**

The Sphere tool is the simplest of the solid primitives and requires only one set of mouse click-drag-release. The created shape is placed in the currently active layer.



*Figure 13.2 — Creating a sphere*

### Cylinder Tool

The Cylinder tool requires two inputs: the cylinder's radius and height. The first step is to click and drag to determine the radius. Then, like the Block tool, click in another window and drag for the height.

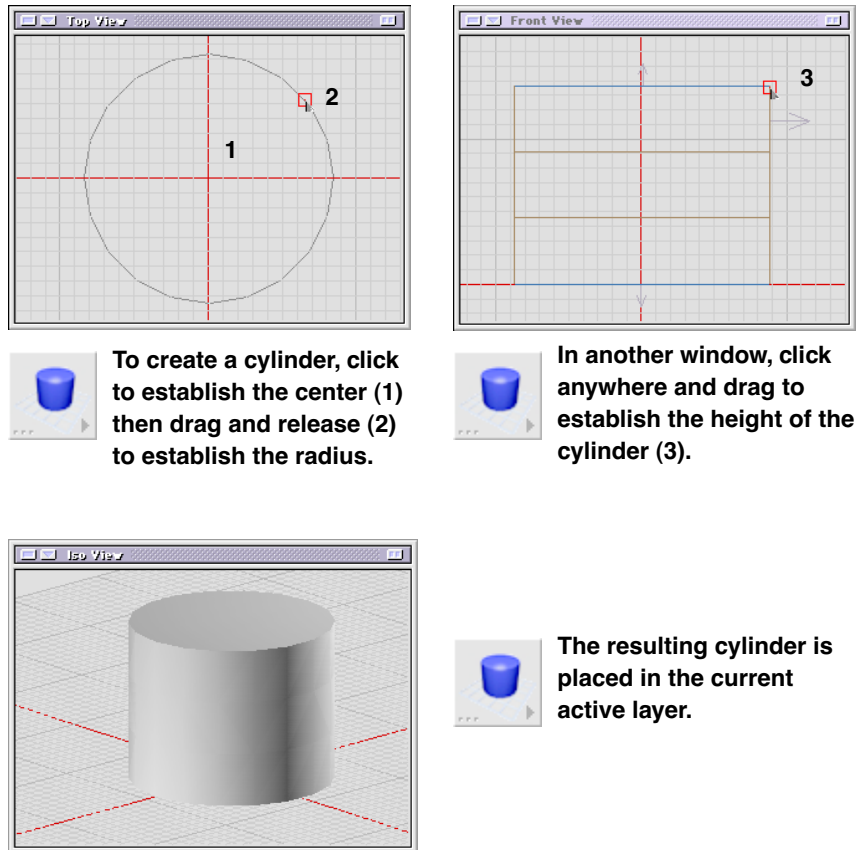
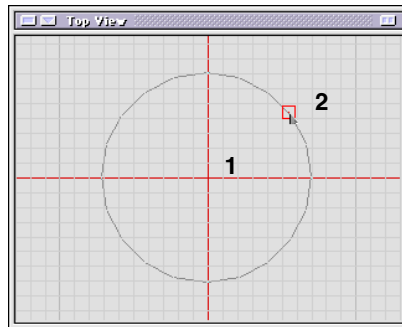


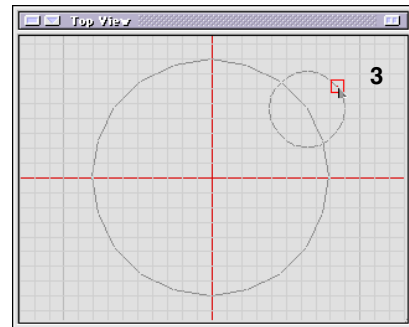
Figure 13.3 — Creating a cylinder

## Torus Tool

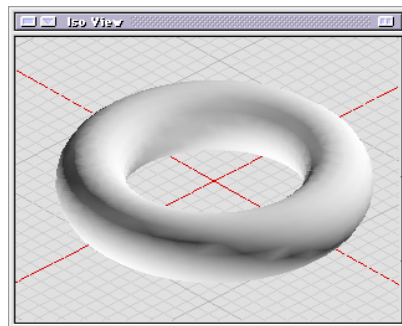
Creating a torus requires the radius of the torus, followed by the radius of the encircling tube.



To create a torus, first click where you want the center (1) and then drag out and release (2) to establish the radius.



Click and drag out again. The small circle that forms indicates the size of the tube that will encircle the first circle. Release when done (3).

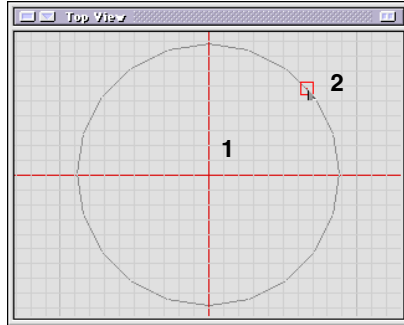


The resulting torus is placed in the current active layer.

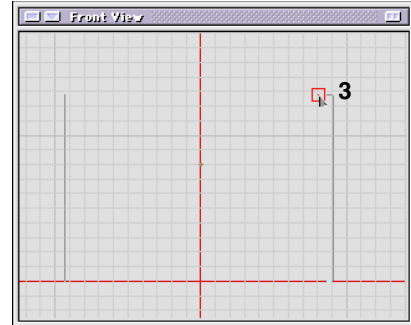
*Figure 13.4 — Creating a torus*

## Cone Tool

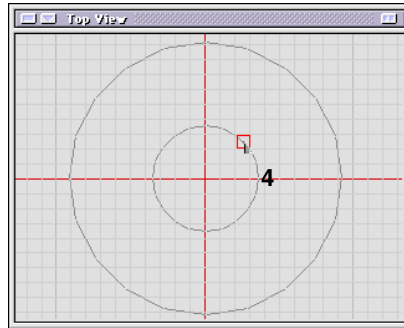
Creating a cone requires information for the base, the height, and the width of the cone top.



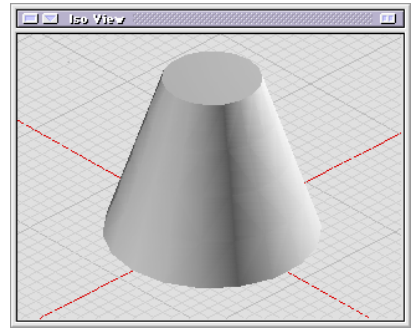
To create a cone, click to establish the center (1), drag and release to determine the radius (2).



In another window, click and drag to determine the height (3).



Return to the first window and click and drag to determine the size of the top of the cone(4).

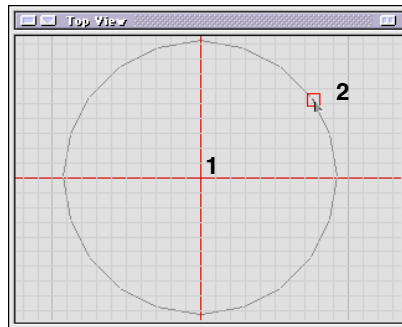


The resulting cone is placed in the current active layer.

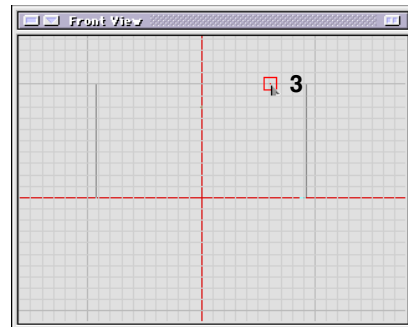
Figure 13.5 — Creating a cone

### Pyramid Tool

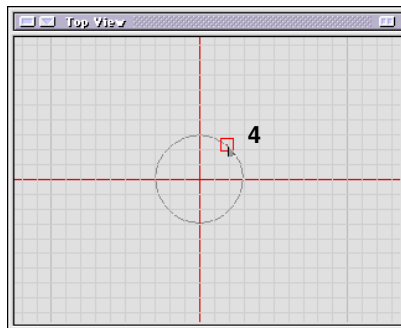
Much like a cone, a pyramid requires information for the base, the height, and the width of the pyramid's top. The number of sides can be controlled by bringing up the options dialog for the tool.



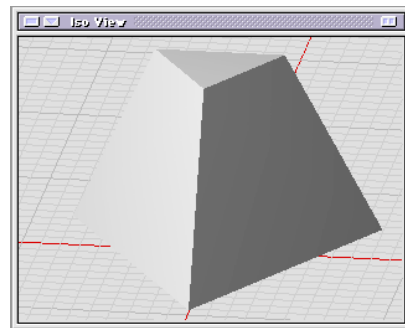
To create a pyramid, click to establish the center (1), then drag and release to determine the radius (2)



In another window, click and drag to determine the height (3).



Return to the first window and click and drag to determine the size of the top of the pyramid (4).



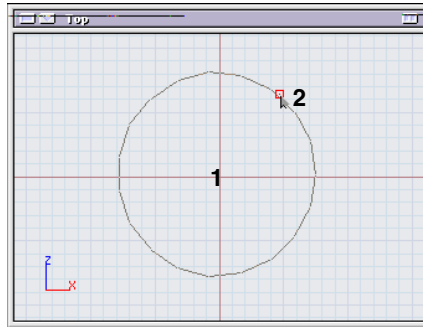
The resulting pyramid is placed in the current active layer.

Figure 13.6 — Creating a pyramid

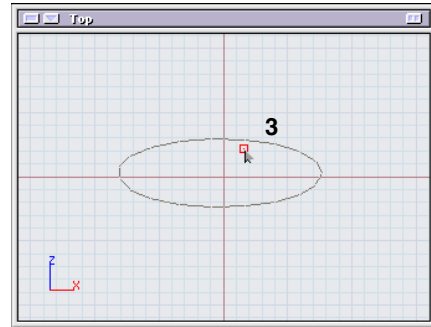
### Prism Tool

Like the previous tools, a series of steps determines the parameters of the prism. The initial click determines the center of the prism, while the following drag will determine how long it will be. On releasing the mouse the next drag and click, in the same window, will determine the width of the prism. The last action will determine how deep it is. To do this use a new window; if the first two actions take place in the top window, for instance, you can determine the depth in a side or iso window.

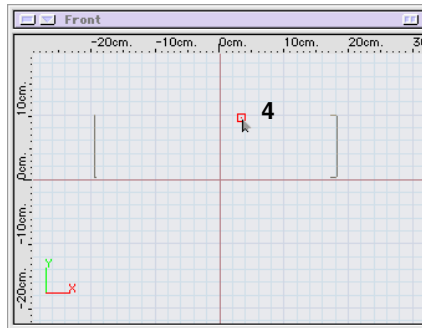




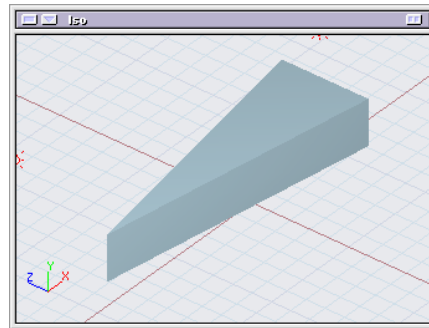
To create a prism, click in the center (1) and drag to create the length (2).



Drag and Click again to determine the width.



Click in another window to determine the depth.

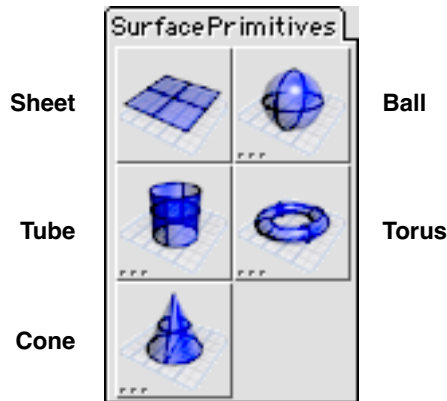


The resulting prism is placed in the current active layer.

Figure 13.7 — Creating a prism



## Surface Primitives Palette



*Figure 14.0 — Surface Primitives Palette*

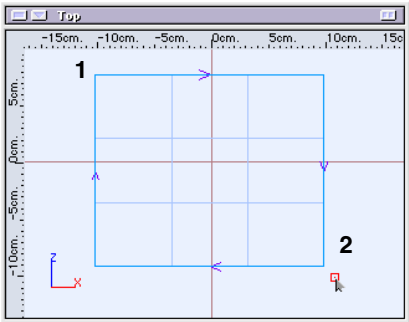
The Surface Primitives palette is a collection of geometric shapes that is similar to the ones in the Solid Primitives palette. The difference however, is that the shapes created are two-sided NURBS surfaces. Another difference is that not all these shapes are completely enclosed - for example, the surface sheet and the surface tube are open objects, and the surface cone can be open at one or both ends.

As with solid primitives, surface primitives can be edited, hacked, knifed, or Booleaned. In addition, these shapes can be edited directly with the tools in the NURBS surface editing tool palette.

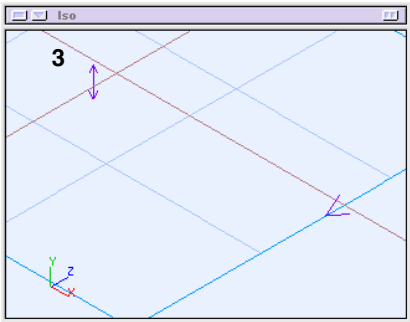
Creation of the Surface Sphere, Tube, Torus and Cone are similar to their counterparts in the Solid Primitives palette. You can create these shapes either interactively, or by specifying the exact dimensions in the options dialog invoked by double-clicking on the tool icon. The Surface Sheet is

created simply by dragging a rectangle with the mouse. All newly created objects are placed in the active layer.

Keep in mind that the Sphere, Tube, Torus and Cone objects are rational NURBS surfaces. If you want to use them in any NURBS surface continuity operations, it would be necessary to convert them to non-rational surfaces (see “Solid Primitives Palette” on page 139).

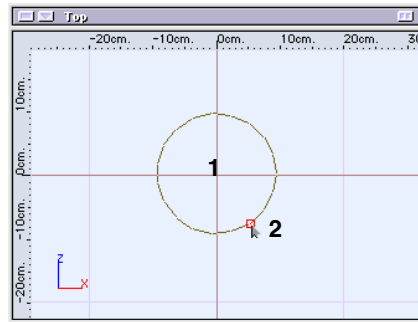


**To create a surface sheet, click (1) then drag and release (2).**

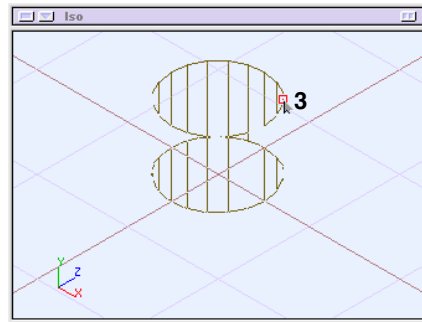


**Note the surface is two sided. (3).**

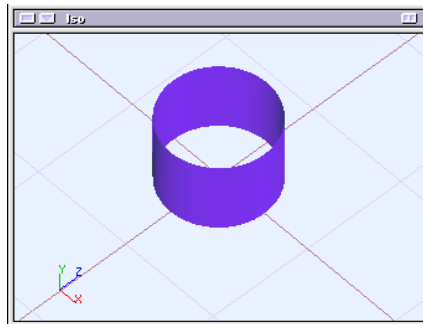
*Figure 14.1 — Creating a surface sheet*



To create a surface tube, click to establish the center (1) then drag and release (2) to establish the radius.



In another window, click anywhere and drag to establish the height of the tube (3).



The resulting surface tube is placed in the current active layer. Note the surface tube is not filled.

Figure 14.2 — Creating a surface tube



## Law Tools Palette

This palette contains tools that allow you to create surfaces and wires from a mathematical formula. Laws are a new feature of the Universe Modeler. They allow the user to do two things:

1. Create surfaces from parametric functions.
2. Create curves from parametric functions.



*Figure 15.0 — Law Tools*

To use either the Face from Law or the Edge from Law tools, simply double click on the tool, and input your formula. Some presets are provided.

The Face from Law and Edge from Law tools are similar in concept to the Space Warp tool. Instead of deforming an existing surface they create a new surface or edge based on a mathematical formula.

As the name implies, the Face from Law tool will create a new face and the Edge from Law will create a new edge.

### 15.0 Parametric Functions

There is a class of mathematical functions called parametric functions. These functions are used extensively in computer graphics as they are very fast and easy to evaluate compared to the other types of functions.

For example, in animation often the channels are represented with function curves. These curves are implemented using a class of parametric functions like Bezier or B-spline. With these functions, it is straightforward to evaluate the value of a channel given a time value. While bezier and b-splines are commonly used parametric functions, they are simply one of the bazillion parametric functions that can be formulated. Actually the number of parametric functions is infinite because you can stack them all in many ways to compose your own function.

### Parametric Curve

If any point on a 3D curve can be computed as a function of a parameter, say  $u$ , we say that the curve is parameterized on the parameter  $u$ , and such a curve is called a parametric curve. In mathematical form, the relationship is represented in Figure 15.1

$$P(x,y,z) = (f(u), g(u), h(u))$$

*Figure 15.1 — A parametric curve mathematical syntax*

### Parametric Surface

If any point on a 3D surface can be computed as a function of two independent parameters, say  $u$  and  $v$ , we say that the surface is parameterized on parameters,  $u$  and  $v$ , and such a curve is called a parametric surface. Figure 15.2, for the mathematical form.

$$S(x,y,z) = (f(u,v), g(u,v), h(u,v))$$

*Figure 15.2 — A parametric surface Universe Modeler syntax*



With the stipulation that we are dealing with parametric functions, let us start with some simple trigonometric functions that we are very familiar with and see how we can massage them in a form that can be dropped into the Laws interface.

Let's say you want to create an edge that is shaped like a sine wave. We all know, that a trigonometric function, such as a sin, gives us that shape. Let us make this sine wave go from  $-\pi$  in  $X$  to  $+\pi$  in  $X$  and have an amplitude of, say 0.5, and a frequency of say 10. We'll make the sine wave be in the  $XY$  plane (front view).

Based on the above description, the math equation we are trying to code is in Figure 15.3

$$Y = 0.5\sin 10X \quad \text{where } x = -\pi, \pi$$

*Figure 15.3 — Sine wave math equation*

In Universe Modeler syntax will be represented as Figure 15.4 and  $u$  will vary from  $-\pi$  to  $\pi$ .

$$\text{vec}(u, 0.5*\sin(10*u), 0)$$

*Figure 15.4 — Universe Modeler syntax for sine wave*

How did we get from  $Y = 0.5\sin 10X$  to  $\text{vec}(u, 0.5*\sin(10*u), 0)$ ?

Our sin function can be thought of as a parametric function where the parameter is  $u$ . Then,  $x$ ,  $y$  and  $z$  are simply described as functions of  $u$ .

$$X = u$$

$$Y = 0.5 * \sin(10 * u)$$

$$Z = 0$$

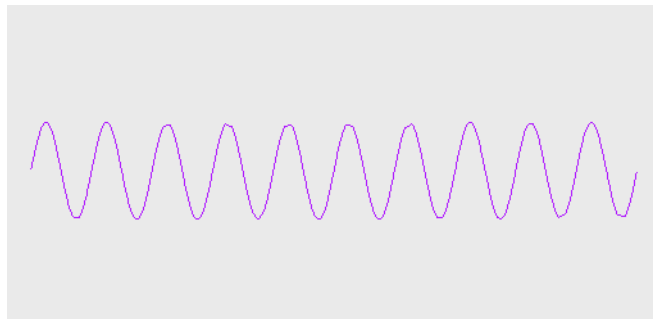
*Figure 15.5 — Parametric function where the parameter is u*

Notice, we explicitly set Z to zero because in Universe Modeler Laws, we are dealing with 3D parametric curves and surfaces, and the law must return a 3D point.

Once you see Figure 15.5 as the way of representing our sine wave, the rest is simply text processing. We take the right hand sides of the above three equations, and put them inside parenthesis, and separate them by commas, see Figure 15.6.

$$\text{vec}(u, 0.5 * \sin(10 * u), 0)$$

*Figure 15.6 — A Parametric Universe Modeler syntax where the parameter is u*



*Figure 15.7 — Sin function*

The `vec` function, as explained later, is really a way of making a vector from arguments. Typical use is making 3D points and 3D vectors, but not limited to that. An example of `vec (0,1,0)` can be used to create a function that is a 3D point at  $((0,1,0))$  or a unit vector along the y-axis.

Notice that often in mathematics, parentheses are omitted when writing function arguments like `sinx`, `cosx`. In Universe Modeler, all function arguments are included inside parentheses, just like you would when writing the same expression in a programming language like C. Also note that the multiplication symbol `*` is explicitly included between two expressions. In Mathematics the multiplication symbol is often omitted. So an expression `xy` in math will be represented as `x * y` in Universe Modeler.

**Exercise** What will be the formula for creating the above sine wave in `z=5` plane? See Figure 15.8 for the answer.

**`vec(u, 0.5*sin(10*u), 5)`**

*Figure 15.8 — Answer for sine wave in `z=5` plane formula*

### Creating A Circle Using The Law Tool

Even though Universe Modeler has a Circle Primitive tool, let us go through the exercise of creating the same circle using the Law tool. Later, we will combine this circle and the above sine wave to create a sine wave along a circle, thereby providing an example on how these simple parametric functions may be combined to create more interesting shapes.

In polar form, a circle is represented as  $r = \text{radius}$ ,  $\text{theta} = 0 \text{ to } 2\text{Pi}$ . Doing the polar to cartesian transformation, we get  $x = r \cos(\text{theta})$ ,  $y = r \sin(\text{theta})$ ,  $r$  is the radius and we'll make  $\text{theta} = u$ .

So, if we want a circle of radius 100, lying in the XY plane i.e.,  $z = 0$

$$x = 100 \cos(u)$$

$$y = 100 \sin(u)$$

$$z = 0$$

*Figure 15.9 — XYZ axis for circle with radius of 100*

Proceeding like we did in the sine wave case, we get the following.

$$\text{vec}(100 * \cos(u), 100 * \sin(u), 0)$$

*Figure 15.10 — Law syntax for circle with radius of 100*

$u$  goes from 0 to  $2 * \text{Pi}$ . If the circle was centered at  $x = 1$ ,  $y = 2$ , Figure 15.10 equations will be as in Figure 15.11.

$$x = 100 \cos(u) + 1$$

$$y = 100 \sin(u) + 2$$

$$z = 0 + 0$$

*Figure 15.11 — Parametric function for a circle centered at (1,2,0)*

To create a sine wave along a circle, make the radius of the circle be the dependent of  $\text{theta}$  and vary with the sine of  $\text{theta}$ .

For example,  $r = \text{radius} + A * \sin(f * \text{theta})$ , where  $A$  is the amplitude of the sine wave, and  $f$  is the frequency. Choosing  $A = 5$  and  $f = 10$ , we get Figure 15.12.

$$x = (100 + 5 * \sin(10 * u)) * \cos(u)$$

$$y = (100 + 5 * \sin(10 * u)) * \sin(u)$$

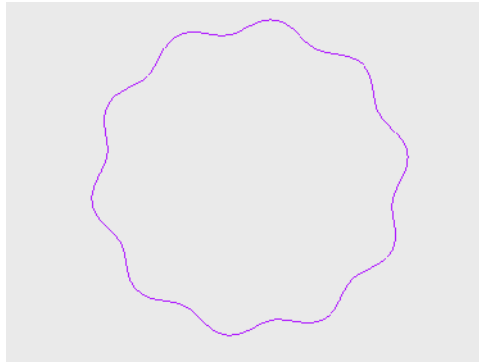
$$z = 0$$

*Figure 15.12* — Parametric function for a sinusoidal circle

Putting this in the Universe Modeler syntax we get, Figure 15.13 and **u** is from 0 to 2 Pi (~ 6.28) to give you a circle with a sine wave ripple.

$$\text{vec}((100 + 5 * \sin(10 * u)) * \cos(u), (100 + 5 * \sin(10 * u)) * \sin(u), 0)$$

*Figure 15.13* — Universe Modeler syntax for sine wave along a circle



*Figure 15.14* — Planar circle with sine wave

### Creating A Spiral Using Law Tools

To make a spiral you start by changing **z** and making it depend on **u**. Since our **u** is varying from 0 to 6.28, we'll make **z = 10 \* u** so that the height of the spiral is roughly 62.8.

$$x = (100 + 5 * \sin(10 * u)) * \cos(u)$$

$$y = (100 + 5 * \sin(10 * u)) * \sin(u)$$

$$z = 10 * u$$

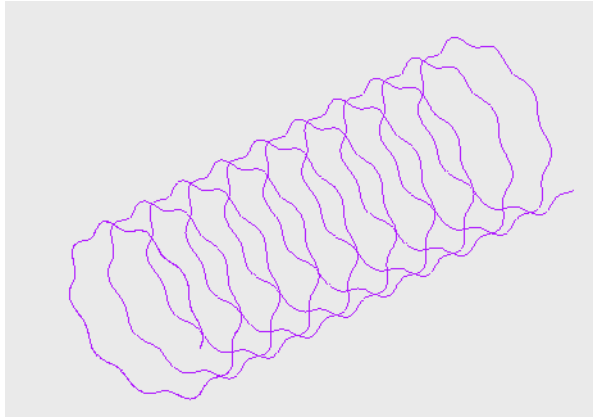
*Figure 15.15 — Parametric formula of a spiral*

Putting this in the Universe Modeler syntax we get, Figure 15.16

`vec((100 + 5 * sin(10 * u)) * cos(u), (100 + 5 * sin(10 * u)) * sin(u), 10 * u)`

*Figure 15.16 — Universe Model syntax for figure 14.15*

If you want more turns to the spiral, simply increase the max **u** from 6.28 to 62.8. 6.28 gives one turn of the spiral. 62.8 will give 10 turns of the spiral.



*Figure 15.17 — Spiral increasing the max u to 62.8*

You have seen how to create syntactically correct Universe Modeler Laws for the Edge from Law tool. Sin and cosine are merely two of the many functions that are supported. A list of many other functions appears at the end of the chapter.

### 15.1 Face from Laws Tool

The Face from Laws tool creates parametric surfaces, which are parameterized on **u** and **v**. Many common shapes such as cylinder, sphere, plane etc. can be described as such functions.

A cylinder is such a function. Its parametric form is often represented as

$$X = 100 * \cos(u)$$

$$Y = 100 * \sin(u)$$

$$Z = \text{Height} * v$$

*Figure 15.18 — Cylinder parametric formula*

$u$  goes from 0 to  $2 * \pi$ ,  $v$  goes from 0 to height of the cylinder. Put this in the UM syntax, we get Figure 15.19.

$$\text{vec}(100 * \cos(u), 100 * \sin(u), v)$$

*Figure 15.19 — Face from Laws syntax*

$u = 0$  to  $6.28$ ,  $v = 0$  to  $200$  will give you a cylinder of height to 200. If you want to give a ripple in the  $z$  direction and modulate the radius with height, you need to change the radius in the  $X$  and  $Y$  to be some function of the  $v$ . We make it  $100 + 10 * \sin(0.1 * v)$

$$X = (100 + 10 * \sin(0.1 * v)) * \cos(u)$$

$$Y = (100 + 10 * \sin(0.1 * v)) * \sin(u)$$

$$Z = v$$

*Figure 15.20 — Formula for cylinder with a ripple in the  $z$  direction*

$$\text{vec}((100+10*\sin(0.1 * v))* \cos(u), (100 + 10*\sin(0.1 * v))*\sin(u), v)$$

*Figure 15.21 — Face from Laws syntax for cylinder with a ripple*

$u = 0$  to  $6.28$ ,  $v = 0$  to  $200$  should give you a rippled cylindrical surface.



## 15.1 Adding More Presets to UM laws.

In the Resources folder of Universe Modeler, there are three text files that define all the law presets in the tools. These are named EdgeLaws.txt, FaceLaws.txt and SpaceWarpLaws.txt. When you open the dialog for the Edge From Law tool, the file EdgeLaws.txt is used to load the presets in the popup menu.

To add more presets to the Edge From Law tool, one simply needs to add the new law to this file. Each line contains one preset. The line below has four fields separated by spaces.

1
2
3
4

Figure 15.22 — 

sine	vec(u,sin(u),0)	0	6.2824
------	-----------------	---	--------

First field states the name of the law as it appears in the interface. In this case it is “sine”. The second field is the law expression in the Universe Modeler syntax as you have seen described above. The third field is the **u** min value. In this case it is a zero. Fourth field is a **u** max value, in this case it is 6.2824

If you want to add more presets to the Edge From Law tool simply append it to the list in the text file. The syntax of presets in the FaceLaws.txt and SpaceWarpLaws.txt are similar. The face laws require two ranges, **u** min to **u** max and **v** min to **v** max, hence the two extra fields as compared to the edge laws.

**Important** *No spaces should be in the law definition itself. Only spaces allowed are in between fields.*

### 15.2 List of functions that may be used in Universe Modeler

#### Operators

+ - \* / ^ for plus, minus, multiply, divide and power

negate (-)

#### Common trigonometric functions and hyperbolic functions

sin(x), arcsin(x), cos(x), arccos(x), tan(x), arctan(x)  
arccosh(x) arccoth(x) arccsc(x) arcsec(x), sech(x) arcsech(x) sinh(x)  
arcsinh(x) arctanh(x)

x can be any law that returns a single value

abs(x) = positive value function.  
exp(x) = e to the power x  
^ exponent operator  
min(x,y,...) minimum of two of more  
max(x,y,...) maximum of two or more functions  
step()  
bend(position, axis, direction, distance)

**Example** `bend(vec(0,0,20), vec(0,1,0), vec(0,0,1), 10)` is a bend law which when applied to an object will bend it with the bend center being (0,0,20), bend axis being the Y axis, the bend directions being the z axis, and the bend radius is 10.

`vec(law1, law2, ....)`

This makes a vector from two or more arguments. It's used mostly to specify 3D points and 3D vectors.

`cross(law1, law2)`

Each of the sub-laws must return three values. Cross law return the vector cross-product.

**Example** `cross(vec(1,0,0), vec(0,1,0))` will evaluate to `vec(0,0,1)`

`dot(law1, law2)`

Dot law returns the dot product of two laws. the two laws can have different dimensions, the smaller one is padded with zeros.

`D(law, variable, n)`

This gives the nth derivative of a law with respect to variable x. So  $D(\cos(x) \times 1)$  is  $\sin(x)$ .

`division (law1, law2)`

same as `law1 / law2`

**Example** `division(sin(x), cos(x))` is `tan(x)` also can be written as `sin(x)/cos(x)`

`E`

Is the natural base of the logarithms.

`equal(law1, law2)`

Returns true if the two laws are equal.

`not_equal(law1, law3)`

Returns true if the two laws are not equal.

`even?(law)`

Returns true of the law.

`ln(law)`

Log to base e.

`log(law, my_base)`

Log to the base my\_base.

`norm(law)`

Normalizes a vector law.

`and (law1, law2)`

`or (law1, law2)`

`not (law)`

`O`

Functional composition as if f of g where f and g are both law mathematics functions.

`PI`

The mathematical constant represented at the accuracy of the system 0.

`piecewise(cond1, law1, cond2, law2, .... default law)`

Permits an operation to be performed in a “piecewise fashion”. depending on the conditions that were established and the laws.

**Example** We will design a space warp law that only deforms space outside  $x > 3$  and  $x < -3$  and uses  $\text{vec}((x-3)^2, y, z)$ , in the region  $x > 3$ ,  $\text{vec}((x+3)^2, y, z)$ , in the region  $x < -3$  and is identity  $\text{vec}(x, y, z)$  elsewhere. Such a law would be:

**`piecewise(x>3,vec(3*(x-3)*2,y,z),x<-3,vec(-3*(x+3)*2,y,z),vec(x,y,z))`**

`term(law, n)`

This gives the  $n$ th term of a multi-dimensional law.

`times(law1, law2)`

This is the same as multiplication symbol `*`.

This list is not a complete list of laws supported by ACIS, but a subset of laws that maybe used inside the Universe Modeler. The laws omitted are the ones that require geometrical input in their arguments. Universe Modeler 3.0 does not allow for laws to be created from the model geometry, and so the comprehensive list would have been meaningless at this point.



# 3D Tools Palette

Up to this point, the toolbox has spent its energy in making shapes and wires. Now it's time for the fun stuff. With the tools palette, a simple wire curve can become a jet engine, a regular polygonal surface can become the deck of an aircraft carrier, and a series of undulating Bezier curves can become a rolling hillside.

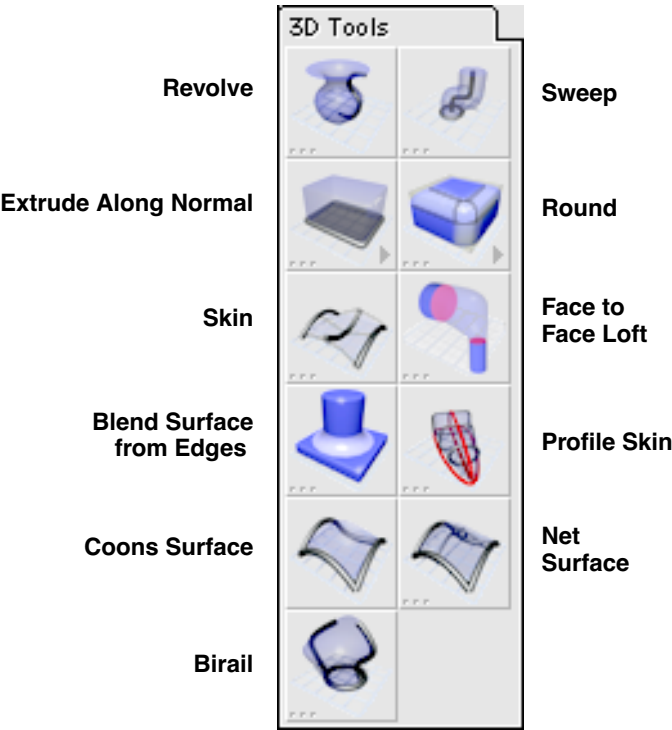


Figure 16.0 — 3D Tools Palette

It's important to note that many of the shapes formed with the tools are derivatives of other shapes, wires, surfaces and even solids. Once the new derivative shape has been formed, you still have the component shapes hanging around. You can select and delete these, or you may place these on a separate layer and make them invisible. Doing so allows you to go back and access the shapes later.

Since all new shapes are placed in the currently active layer, you may want to hop around in your project view while you're building the model. For instance, you could have a layer called **"Parts"** and build your components in this layer. Right before you apply a tool, say a Revolve on a wire, you could create a new layer called **"Shapes."** Before you create a new component shape, click on **"Parts"** to make it the currently active layer. Before you perform any type of derivative function, like Extrude, click on the **"Shapes"** layer.

If you don't want to look at the parts, you can hide either individual elements or everything in the layer by clicking on the visibility dot in front of the element or on the visibility dot in front of the folder. Hidden elements will not be exported when the model is saved as a FACT file, provided **"Export Visible Only"** is selected in the Export Preferences. All the objects within a single folder can be deleted by clicking on the folder and hitting the Delete (PC-Backspace) key on the keyboard.

For a complete explanation of the layers system, see **"Layer View Window"** on page 537

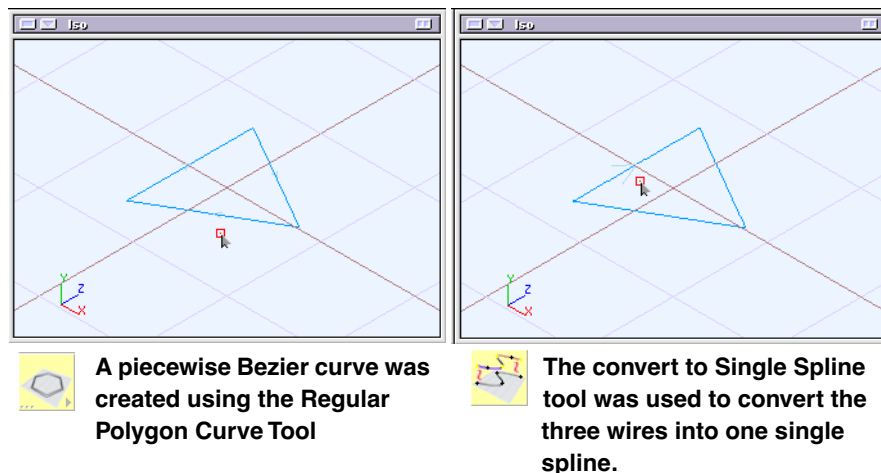


### 16.0 Changing the Topology

If you prefer to use Bezier lines to create shapes, you actually are creating several edges even in a seemingly smooth line. The result may be a shape with more faces than necessary.

To simplify a Bezier wire, use the Convert to Single Spline tool in the Wire Editing palette. This tool will convert a “piece-wise” Bezier line into a single NURBS curve while maintaining the same exact shape. In doing this, a single Bezier which may have had 5 edges will be converted into a single spline. The result is a less complicated wire that uses less computer real estate yet has the same information.

A choice between one or three faces in complex shapes may mean the difference between one face or 20 faces. When creating complex forms, you’ll appreciate the power the Convert to Single Spline tool provides.



*Figure 16.1 — Changing Topology*

The example above also is a good way to point out the difference between topology and geometry. While the number of faces has changed (the topology), the shape remains the same (the geometry).

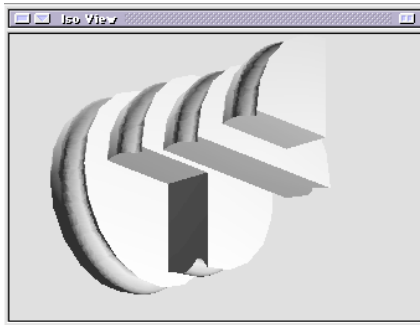
A piecewise Bezier curve is generally only tangent continuous. A NURBS curve created with the NURBS Curve tool creates curvature continuous curves, which give better shading on surfaces.

For a complete description of how the Convert to Single Spline tool works, see “Convert to Single Spline” on page 480.

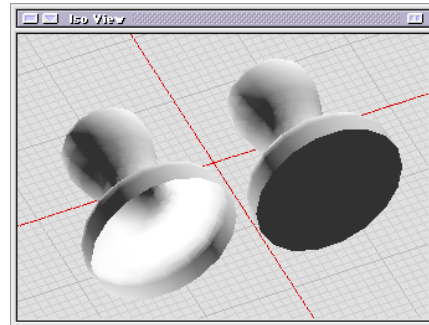
The Convert to Single Spline tool can be applied to any wire that will be extruded, swept, revolved, or used in birail and profile skinning tools. There are restrictions to the type of wires that the tool can be used on. Complex wires, like branches, multi-loops and directionally inconsistent wires should not be converted into single spline wires.

## 16.1 Revolve Tool

An underlying myth of creating 3D images is that the computer does all the work for you. While this is far from true, the lathing process is as close to magic as a computer gets. You create a simple cross-section and, with a little more work, the computer zips out a complete form. Pretty cool.



A wire revolved at 360-, 270-, 180-, and 90 degrees.



A revolved wire as a surface (left) and a solid (right).



Figure 16.2 — Revolve Options

You can revolve any wire shape, NURBS or Bezier curve, polygon, circle, ellipse, rectangle, single face body or arch. To revolve an object, you can create a revolve axis (use Curve Primitive Line tool) or draw the axis interactively.

Before revolving, consider some of the options available in the Revolve tool options box, accessed by double-clicking on the Revolve tool icon.

### **Angle of Revolution**

This is the angle the cross-section will be revolved about the axis, starting from its original position. You can adjust the revolution to a value between 0 and 360 degrees. A full circle is 360 degrees, also the program's default. A quarter rotation, for example, is 90 degrees. A half rotation is 180 degrees and three quarters is 270 degrees.

### **Number of Steps**

A revolve can be a perfect circle, or it can be polygonal in its cross-section. By inputting a number in the “Number of Steps” field you tell the program to create a polygonal revolve based on the number of steps you type in. The default value is zero which creates a smooth shape.

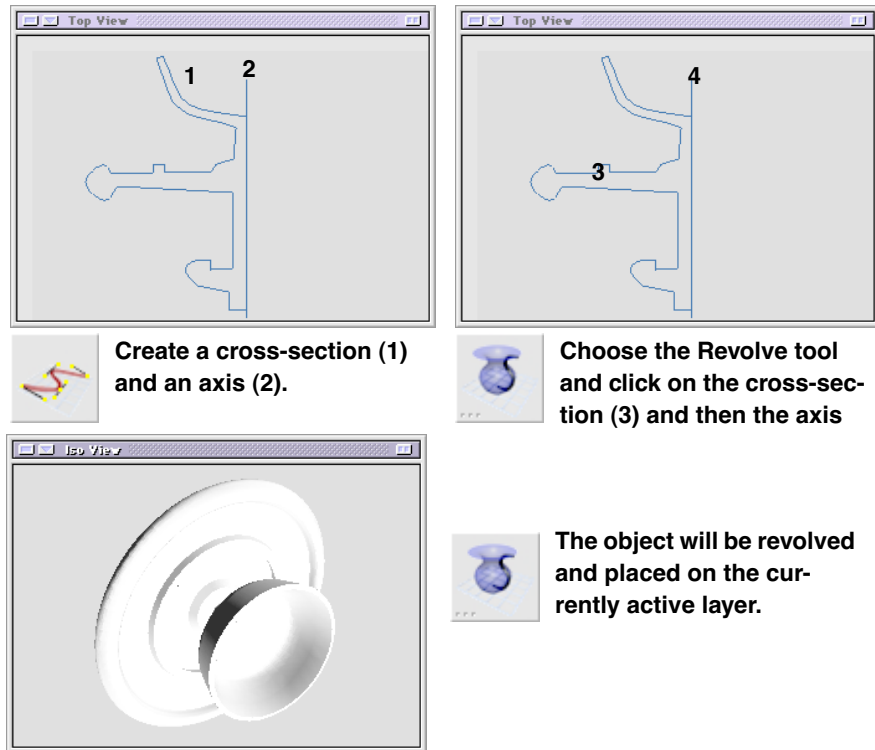


Figure 16.3 — Creating a revolved shape

## 16.2 Solid or Surface

A lathed shape created with the Solid option selected caps any open ends of the object and gives it a single-sided skin, much like a primitive solid. A lathed shape created with the Surface option selected leaves ends open and gives the shape a double-sided surface, like any surface object. You can go back into a shape and remove or add faces and make objects either double-

or single-sided later, but the Solid/Surface option lets you create the kind of form you need right off the bat.

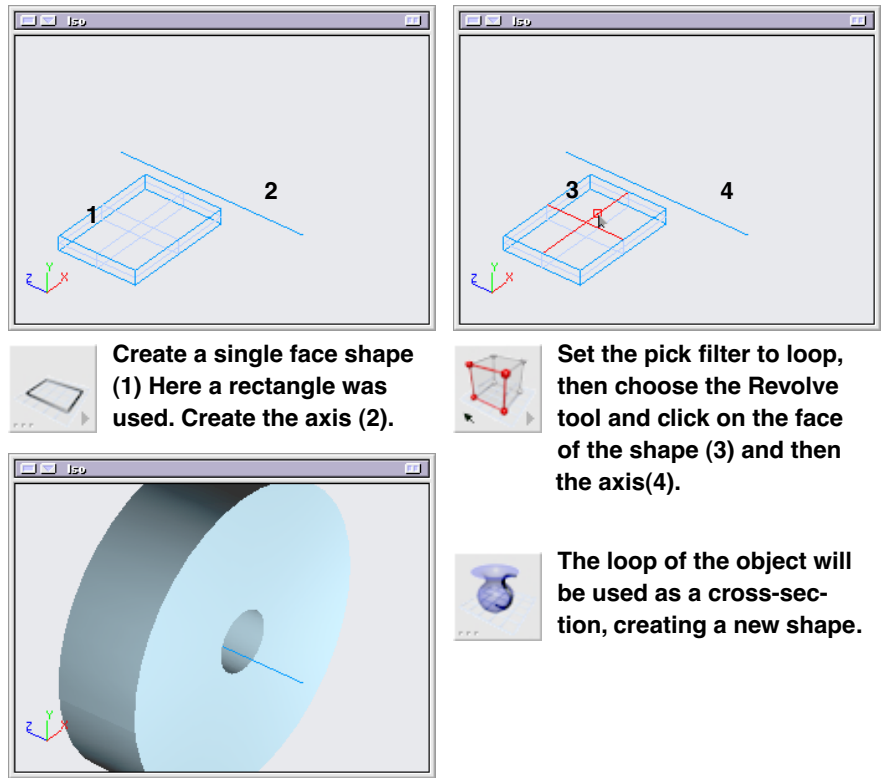


Figure 16.4 — Creating a revolved shape

## **16.3 Sweep Tool**

This tool requires a cross-section and a path. The cross-section can be an open or closed wire, and if it's closed, it can either have a face, or no face. If it has no face, there is a "Caps" option that will create a solid sweep, if so desired. The cross-section must be planar, but the path can be non-planar.

To use the tool, create a cross-section and a path. Choose the Sweep tool, click on the cross-section, and then click on the path. The new swept object will be formed and will be placed on the currently active layer.

Options available for the Sweep tool can be accessed by double-clicking on the tool icon, they include:

### **Oriented Sweep**

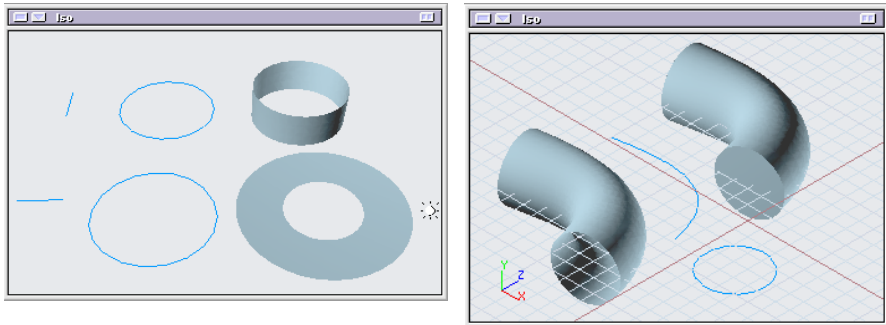
The sweep cross-section is oriented to be perpendicular to the path all along the sweep.

With this option toggled on the sweep will be performed smoothly along the course of the path, rounding out any abrupt changes.

### **Rigid Sweep**

With this option toggled the sweep will follow the path exactly, making sharp corners and not rounding the overall shape.

The cross-section stays rigidly fixed in the same direction as it was when it was picked. This creates varying areas of cross-section.



Orientation of the cross-section will affect how the sweep is performed. The sweeps above use the same cross-section and the same path, the cross-section, however, is rotated 90 degrees different.

Both of these sweeps were created by the open cross-section and path in the center. The sweep on the left used the Caps option toggled off, the one on the right used the Caps option toggled on.



Figure 16.5 — Sweep tool options

### Cap Ends

If the sweep is made from an open cross-section, this option will cap the sweep and create a solid, one-sided object.



## Sweep From Current Position

With this option selected, the sweep will occur from the cross-sections current location and still follow the path defined in the sweep path at the given offset.

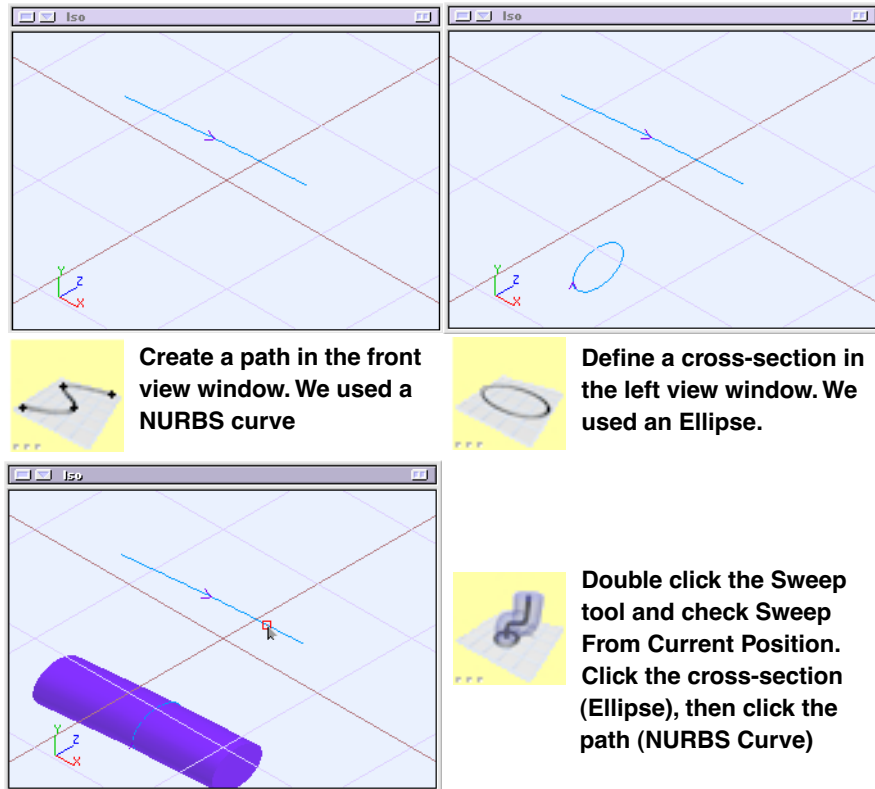


Figure 16.6 — Using the Sweep From Current Position option

### Move The Cross-section To Path

The cross-section will be moved to the start of the path and will be rotated as necessary to perform the sweep.

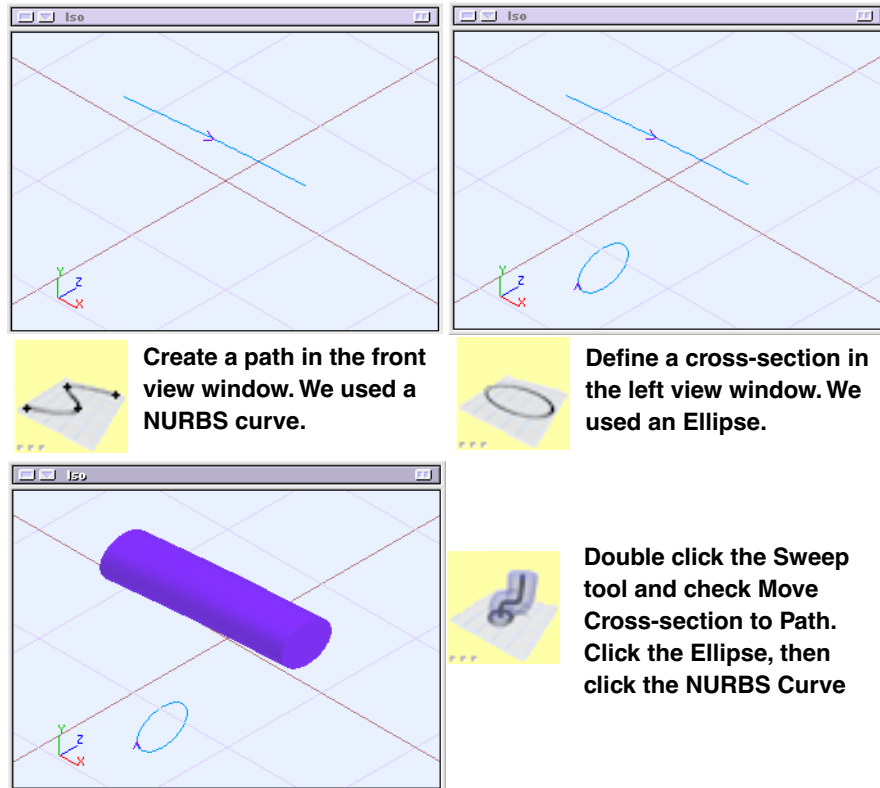


Figure 16.7 — Using the Move the Cross-section to Path option

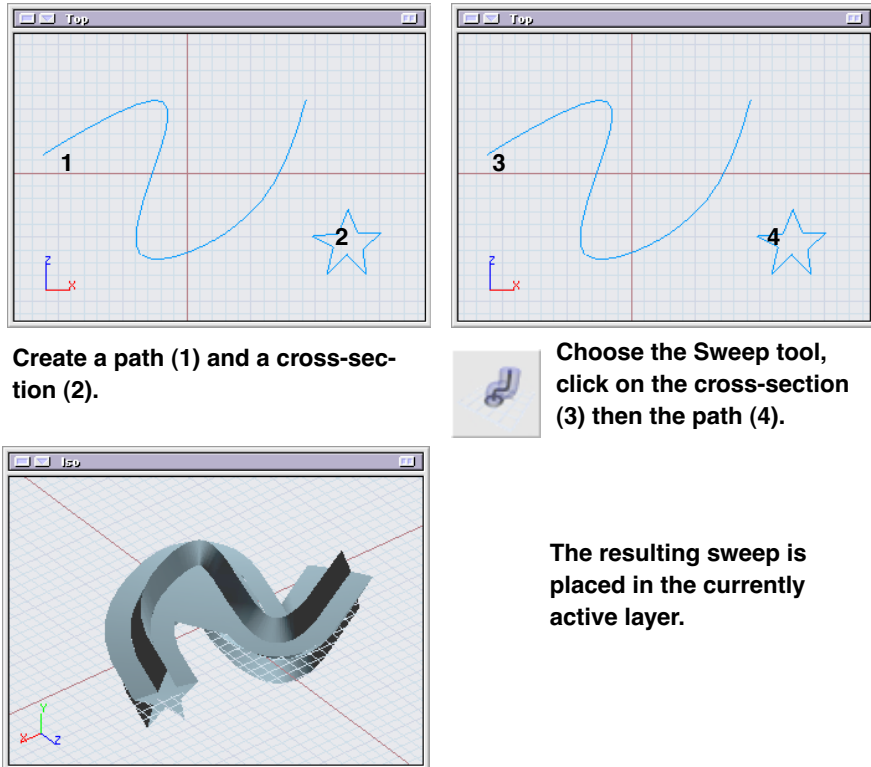


Figure 16.8 — Sweeping a shape along a path

## Cross-section Orientation

The orientation of the cross-section will alter how the sweep is performed. To change the orientation of the cross-section, use the rotation or mirror tools.

### 16.4 Extrude Tool

The Extrude tool takes either a wire or a face and gives it depth. Two ways to extrude an object exist: either along its normal or along a vector. Only planar objects can be extruded. The resulting shape will be a solid object if extruding a filled wire or the capping option is used. The shape will be a surface if the originating object is an open wire, or a closed wire without the capping option.



Figure 16.9 — Extrude tool

#### Extrude a Face Along Normal

First you need a face to extrude. Either create a closed wire shape using any of the wire tools or use a curve primitive. If the “Fill Curve Primitives” option was toggled on under the **Preferences/System** menu under the Creation tab, any shape created with the curve primitives tool will be given a face or filled automatically. If not, go to the Wire Editing toolbox Palette and use the Fill tool. Choose the Extrude Along Normal tool. Click on the face in any view. Move your cursor to a side view of the face and click and drag. Release the mouse at the desired height.

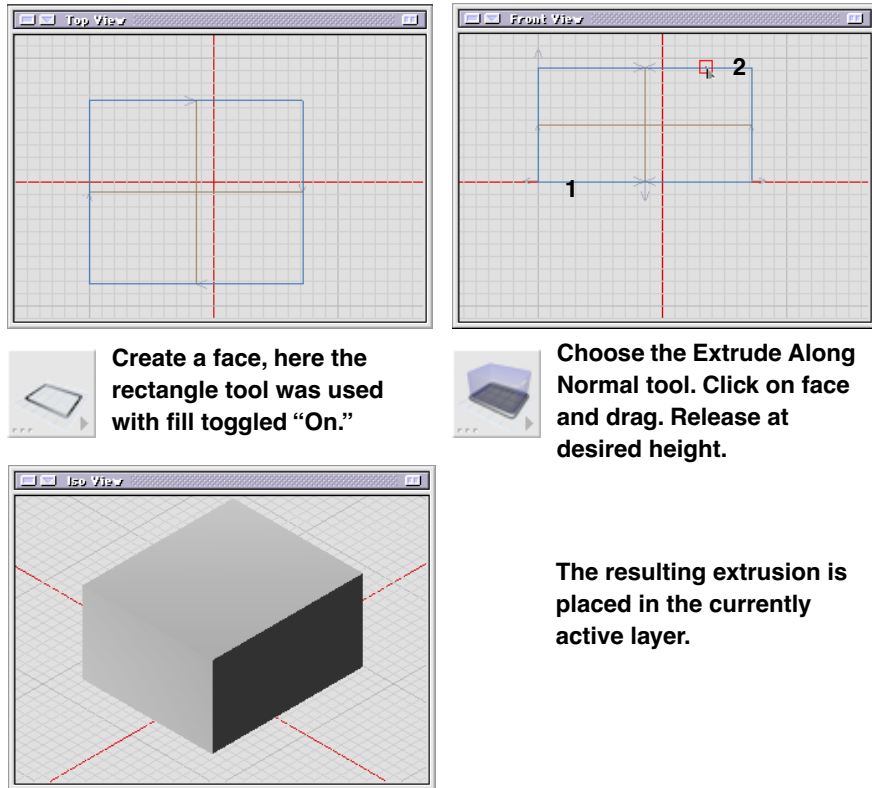


Figure 16.10 — Extruding along a normal

### Extrude a Face Along Vector

The same rule set applies for Extrude Along Vector as Extrude Along Normal. Instead of having a nice perpendicular extrusion, dragging the mouse in any direction determines the height of the extrusion and the shape follows.

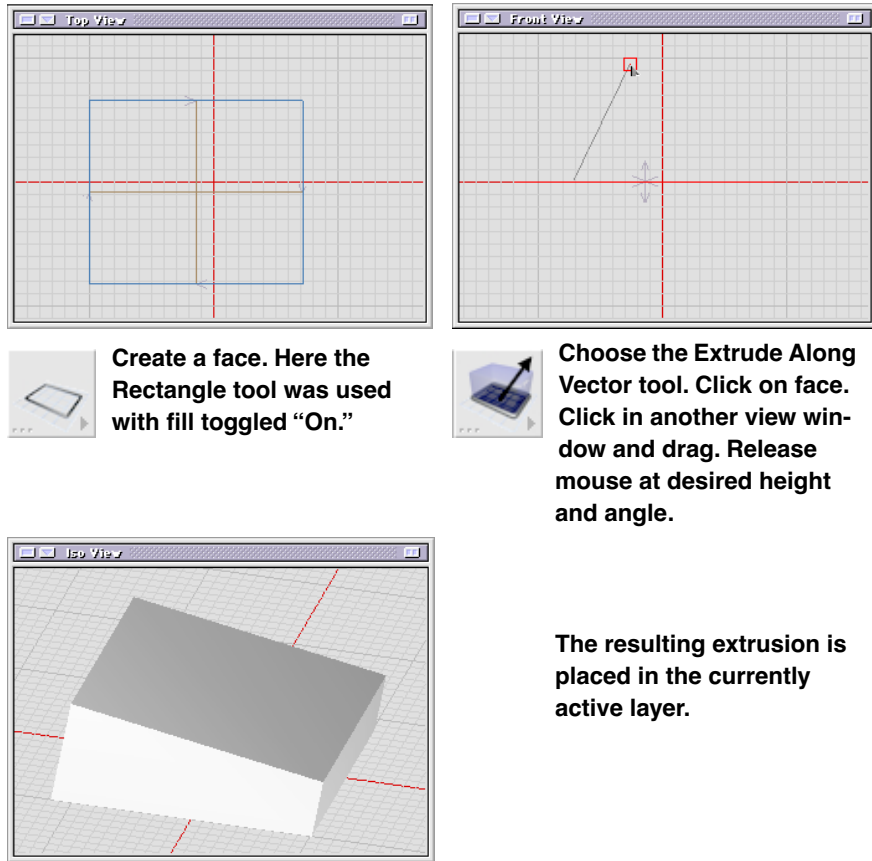
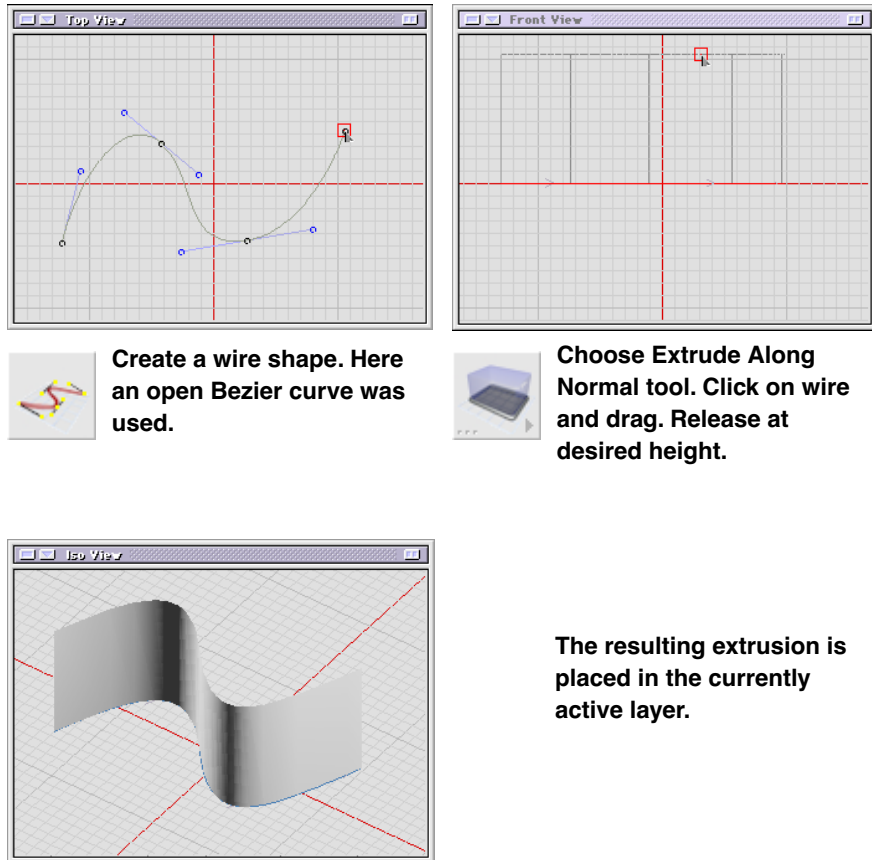


Figure 16.11 — Extruding along a vector

### Extrude a Wire Along Normal

To create a surface shape with the extrusion tool, use an unfilled, closed-wire primitive or an open-wire primitive. These can be created with any of

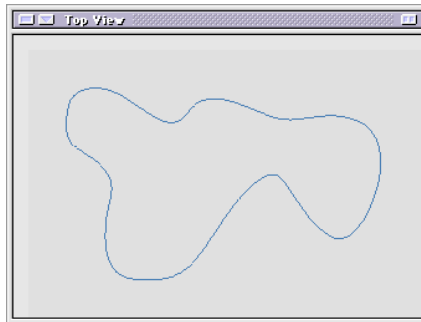
the Curve or Wire Primitive tools. The resulting shape is a double-sided surface shape.



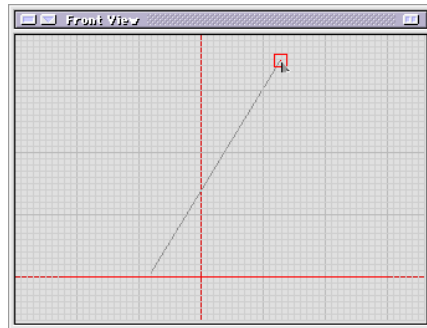
*Figure 16.12 — Extruding a wire along normal*

### Extrude Wire Along Vector

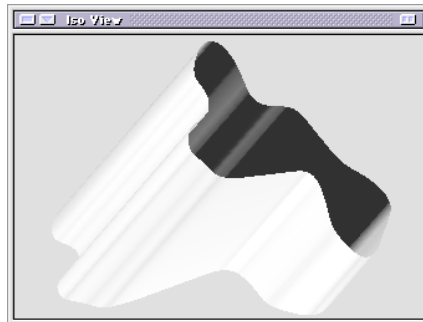
The same tool used to extrude a face along a vector can be used to extrude a wire along a vector. Create the open or unfilled closed wire. Choose the tool, and click on the wire. Drag in any window to achieve the desired height and angle, then release the mouse.



**Create a face, here the Bezier tool was used.**



**Choose Extrude Wire Along Vector tool. Click on the wire and drag. Release at desired height.**

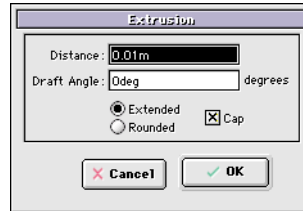


**The resulting extrusion is placed in the currently active layer.**

*Figure 16.13 — Extruding a wire along normal*



**Extrusion Options** Double-clicking on either extrusion tool brings up an options dialog box.



*Figure 16.14 — Extrusion options dialog*

### Extrude Angle

The draft angle determines the taper of an extruded object. With a draft angle of 0 degrees, which is the program's default, an object will extrude directly from the object. With a draft angle input into the field, the object will either grow or shrink as it extrudes. Positive numbers will scale the angle up, and negative numbers will scale the draft angle down.

### Extended/Rounded Corners

As an object is extruded using a draft angle, the corners can either continue as they are, or they can be rounded by using this toggle.

There are two ways to extrude an object at a set distance. Both methods involve input in the extrude Distance and the Draft Angle into the tool's option box.

### Extrude Input Method 1

Create a wire or face to be extruded. Choose the extrude tool and click on the object to extrude, highlighting it. Double-click on the Extrude tool icon

and input your distance and draft angle numbers. When you hit “OK” the extrusion will be performed.

### Extrude Input Method 2

Choose the Extrusion tool and double-click on a face or wire, the program will use the previous extrusion distance.

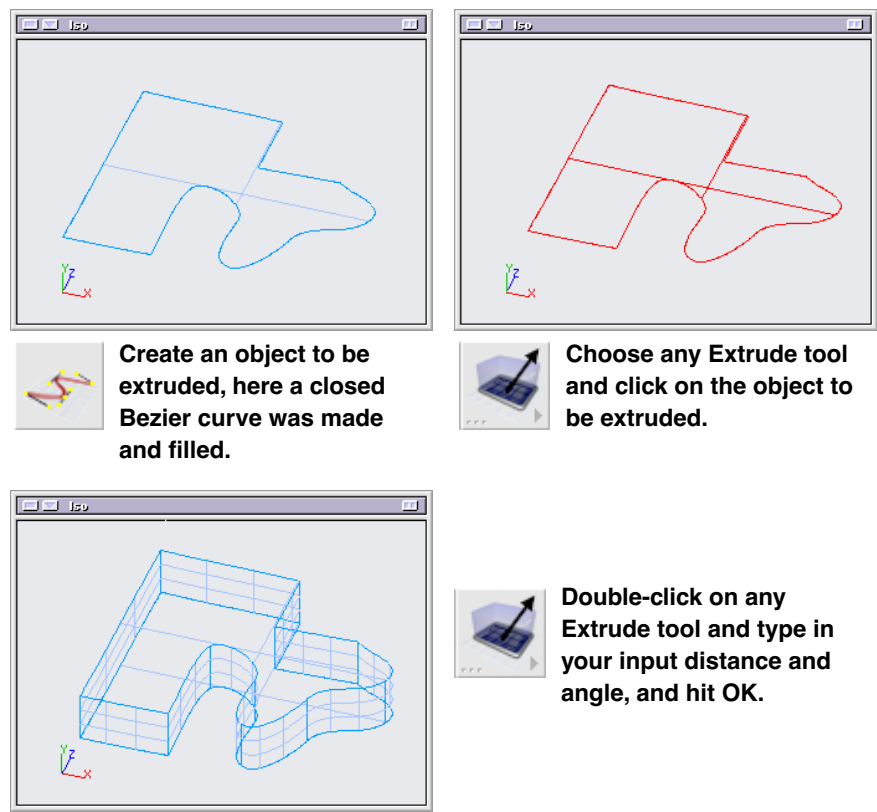
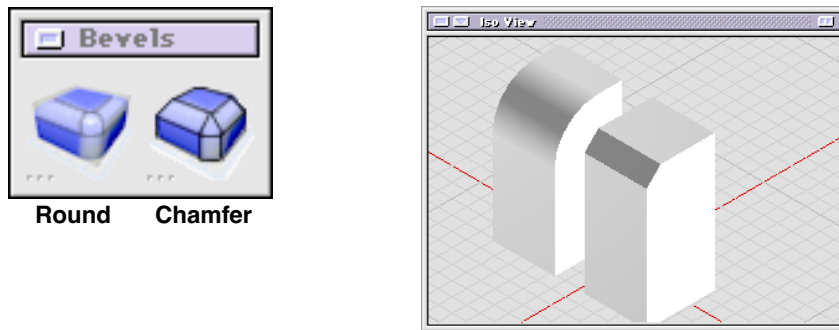


Figure 16.15 — Extruding an object using numeric input

### 16.5 Chamfering and Rounding Tool

The Round and Chamfer tools are used to take the edges off of objects. The tools work by cutting into the existing shape and changing the object's geometry. The tools work hand-in-hand with the Pick Filter tool.

Before using either the Round or Chamfer tool, you have to let the program know exactly what it is you want to round or chamfer. You do so by selecting the proper topology with the Pick Filter tool.

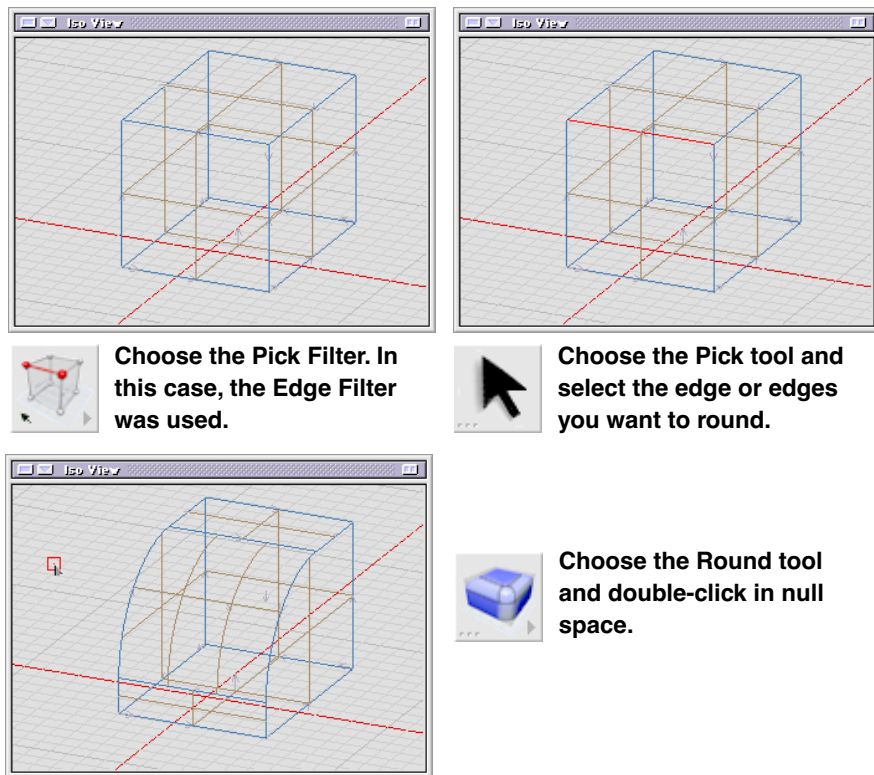


**Figure 16.16 — The block on the left was rounded. The block on the right was chamfered**

The tools work with both the pre-pick and the post-pick method. For pre-picking, use the Pick tool together with the Pick Filter to pre-select all the elements you want to round. Then select the tool and double-click in null space. For post picking, use the Pick Filter to select the type of topology you want to change— an edge for instance. Then choose the tool and select the edge you want to round. Double-click in null space.

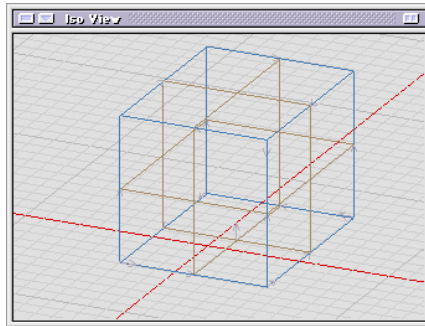
In either case you can double-click on the tool icon to bring up the options box. By entering a number, you can adjust the radius of the rounding or chamfering based upon desired grid measurement system.

While a great many objects can be rounded or chamfered, it is possible to run into edges that don't want to be rounded. One example is on an edge that has already been rounded with the Round tool (A new edge will present itself, but it would be impossible for the program to round this edge.) Another example is entering a radius that is too high, hence asking the program to create a shape which it cannot possibly make. On the other hand, an edge that has been chamfered once can be chamfered again.

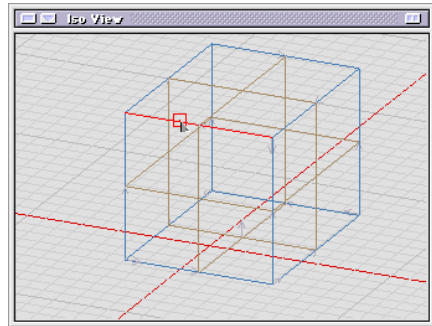


**Figure 16.17 — Using the Round or Chamfer tool with pre-pick.**

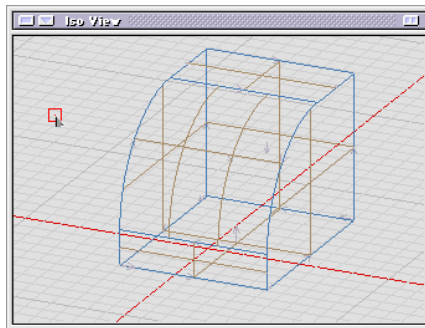
By using the Pick Filter you can round an edge, a face, or an entire body at one time. You also can pick several different items at the same time— for instance, two edges of the same cube or two edges of two different cubes. Once selected, you can round or chamfer by double clicking in null space.



**Choose the Pick Filter. In this case, the Edge Filter was used.**



**Choose the Round tool and click on the edge or edges you want to round.**



**Double-click in null space.**

*Figure 16.18 — Using the round or chamfer tool with post-pick.*

## Chamfering and Rounding Options

Settings that can be altered for the Chamfer/Round tools include the radius and bulge settings, as well as toggles for blending vertices, showing steps and an option for checking the intersections.

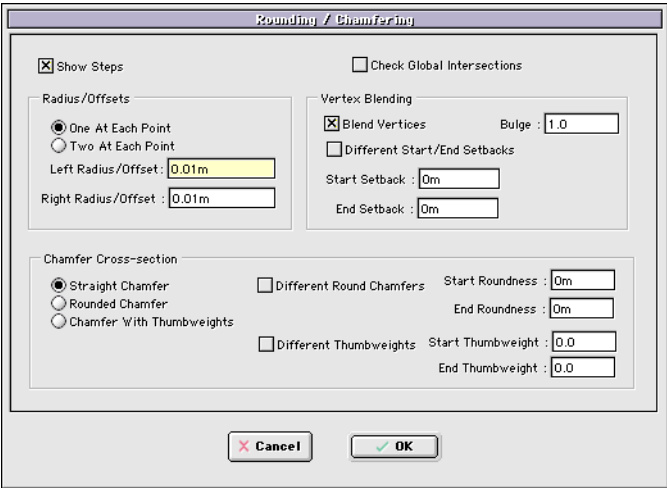


Figure 16.19 — Rounding/chamfering options.

### Show Steps

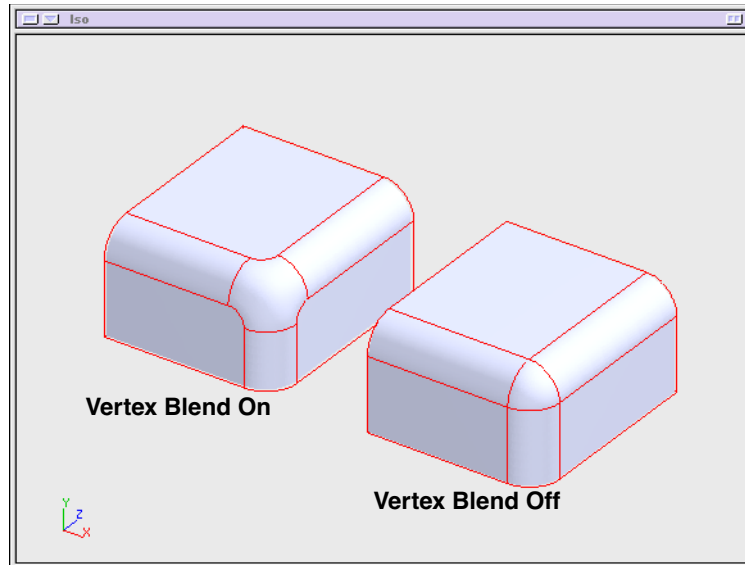
Choosing this option will show the progress of the rounding in the status window, and it also marks blend surfaces so no isoparms or lower tessellation can be done on them.

### Check Global Intersections

This option is normally turned off to speed up the blending process. When this is off, ACIS does not test for surface-intersections with faces that are not involved in the blend, which can be a problem in some cases.

### Blend Vertices

With this option selected, the corners of the body will be rounded, giving six sided vertices on three edge corners.



*Figure 16.20 — Vertex blend example*

### Bulge

Determines what amount of bulge will be created. A setting of “0” will not alter the rounding, a maximum setting of “2” will create a noticeable bulge.

### Vertex Setback

If the blend vertex option is on, the vertices are replaced with surfaces. The point at which this surface starts is controlled using the Vertex Setback, the value is the distance along the edge where the vertex blend starts.

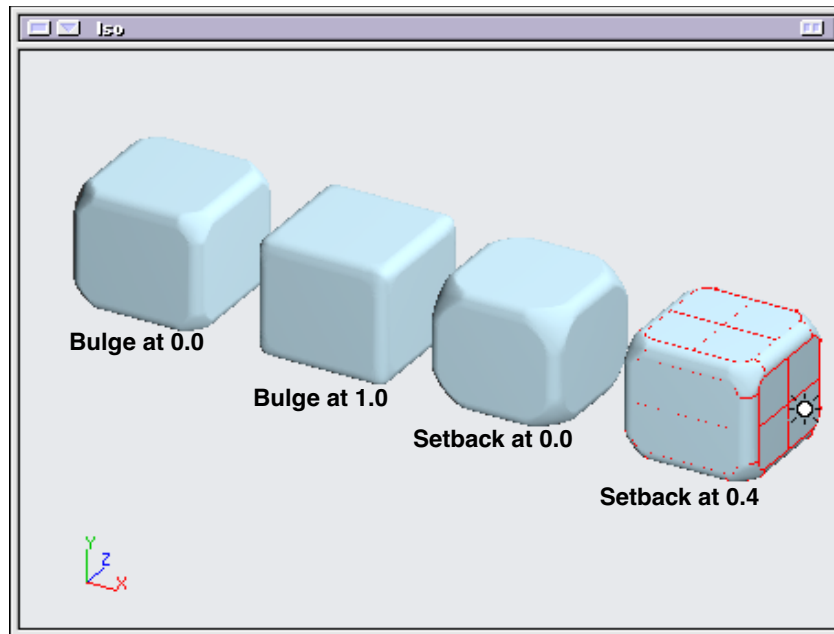


Figure 16.21 — Bulge/setback examples



## Manually Adjusting the Radius

After using the Chamfer or the Round tool a radius circle can be seen in the center of the edges to be effected. By holding down the “E” key and clicking in the center of these circles, the radius can be manually adjusted. When the radius is set, double-click in null space to perform the bevel or chamfer.

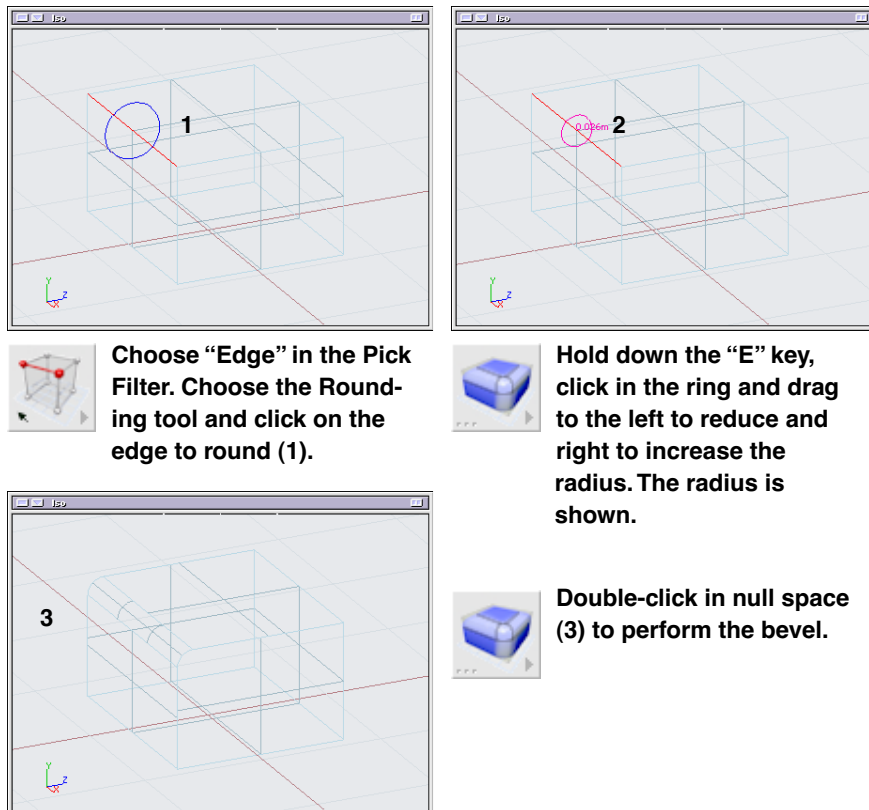


Figure 16.22 — Controlling an edge bevel or round manually

### Oval Rounds

It is possible to choose a separate radius for both sides of the round. This will make the resulting bevel have a different curvature on both sides. To do this, simply hold the “E” key to adjust the radius. But this time, hold the “R” key also to adjust one side, or the “W” key to adjust the other side.

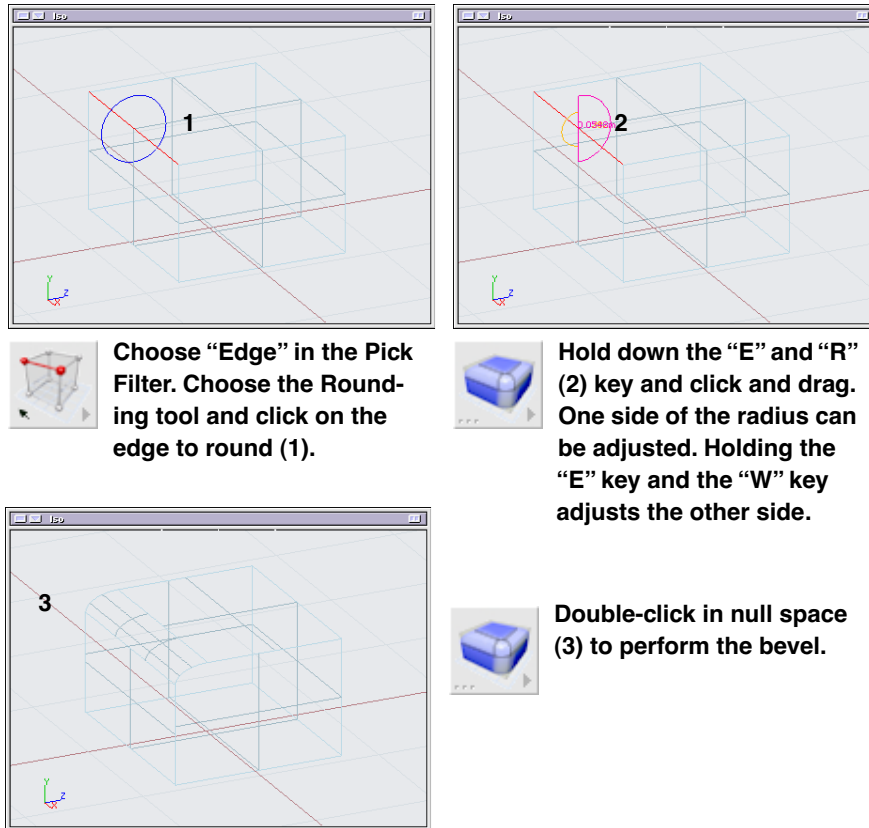
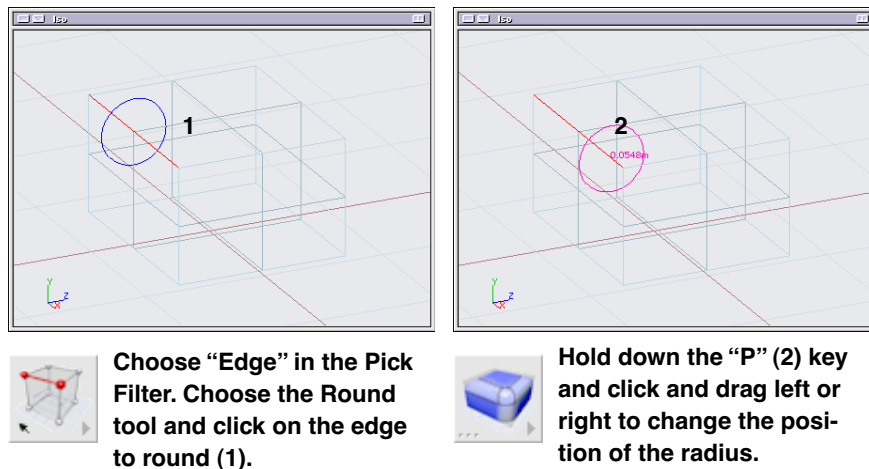


Figure 16.23 — Controlling an oval bevel manually

These settings can be controlled in the options for the tool also, but it is sometimes easier to do them manually. Setting the radius manually works well with the snapping tools too.

### Controlling the Position of the Radius

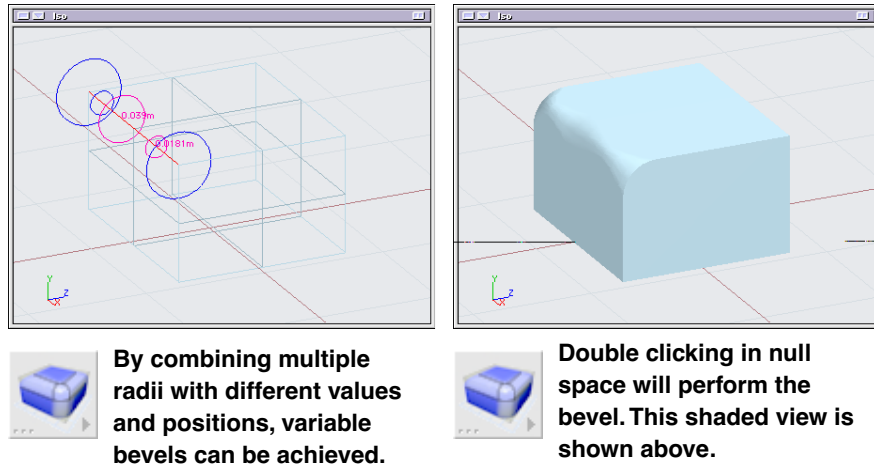
It is also possible to control the position of the radius. To do this, just click the edge to add the radius then hold the “P” key for position and drag. For moving the radius position, dragging right will cause it to move left, and dragging left will cause it to move right. To move it up and down it also needs to be dragged right or left. This is done by design, so the radius and it’s counter are not covered up by the cursor.



*Figure 16.24 — Controlling the position of a bevel manually*

### Variable Radius Blends

It's possible to have an edge with a variable radius. To do this choose the Edge Pick Filter and then the Chamfer or Round tool, and double click on the same edge. On the first click you'll see the regular single radius circle on the center of the edge. On the second click the edge will have two circles, one at each end of the edge. Using the "E" and "R" keys, manually adjust the radius or each circle and double-click in null space.



*Figure 16.25 — Controlling an oval bevel manually.*

By combining multiple radii with different values and different positions, variable blends can be achieved. This is a good way to show wear & tear on an object, or control unique bevels.

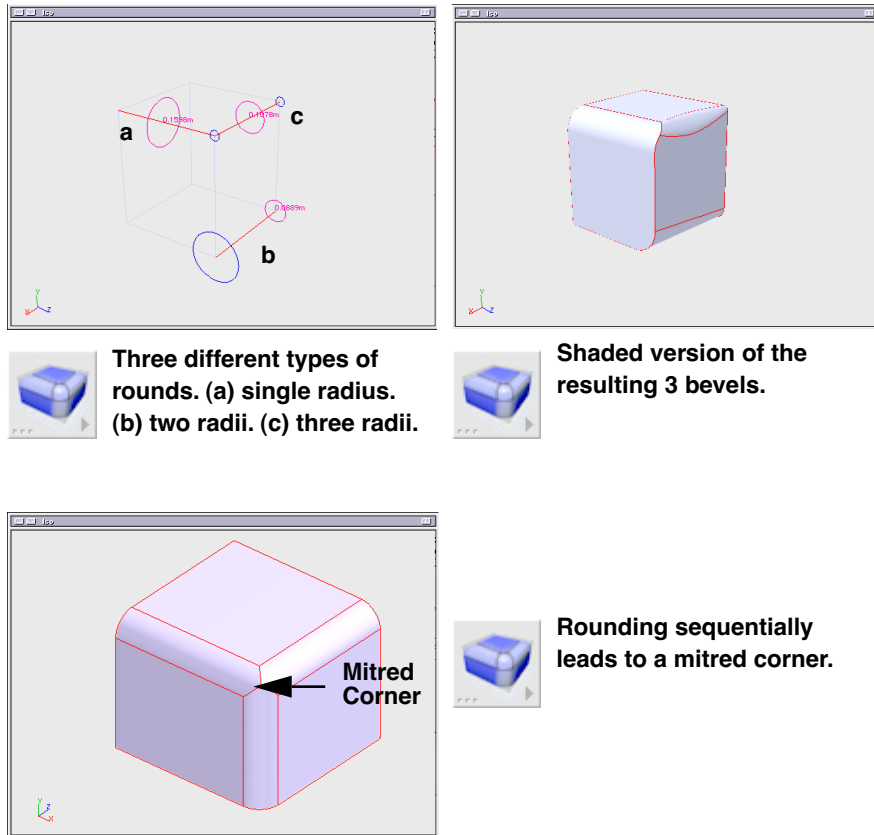
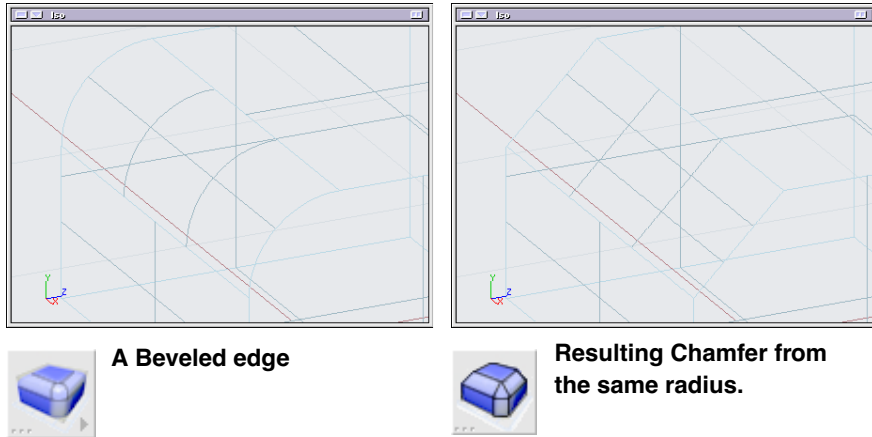


Figure 16.26 — Multiple bevels applied to one object.

### Chamfers

A chamfer is similar to a bevel, except that usually there are no roundings. Instead a very flat rigid edge is created.



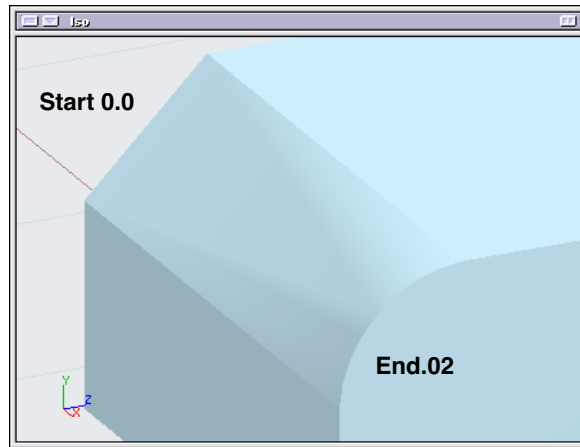
**Figure 16.27 — Comparing a beveled and chamfered edge.**

There are ways of controlling the flatness of a chamfer. One is by using a rounded chamfer and the other is by using thumbweights.

### Rounded Chamfer

The Rounded Chamfer options will allow you to assign a slight round to your chamfer. There is also a check box to use separate values for the edges of the chamfer. Without this selected, the start and end will both be assigned what is listed in the start edit box.

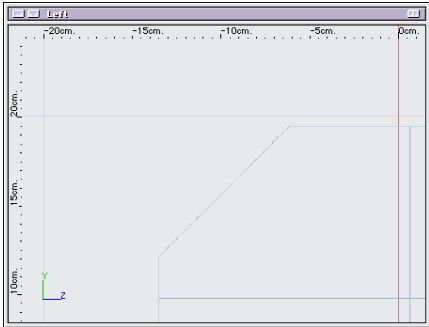
By assigning different values and clicking on the option to use different values you can get a variable chamfer along the edge.



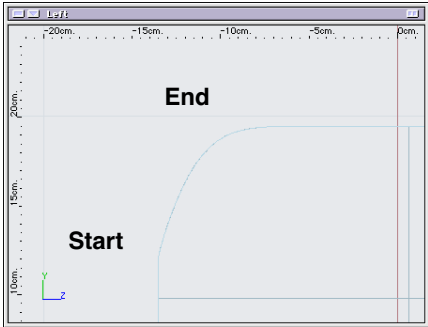
*Figure 16.28* — Chamfer using different rounded start and end settings.

### Chamfers with Thumbweights

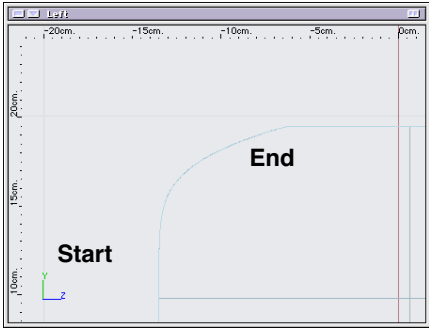
Thumbweights are another way to control a chamfer. This is weighting two pivotal points of a chamfered edge. A thumbweight of zero will be a flat chamfer while a higher number will be more rounded at that point. It is possible to set both start and end points separately in the chamfer options box.



**Chamfer without Thumbweights.**



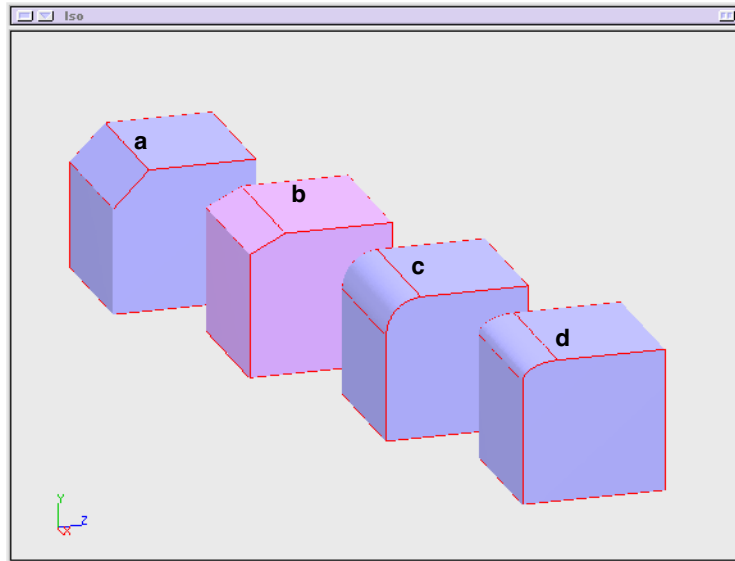
**Chamfer with two separate thumbweights, start is 0.1 and end is 2.0.**



**Chamfer with two separate thumbweights, start is 2.0 and end is 0.1.**

*Figure 16.29 — Controlling an oval bevel manually.*





All are from 1 meter cubes, with one edge was picked.

- (a) Straight chamfer 0.25 meter radius for both left and right.
- (b) Straight chamfer 0.25 meter left, 0.125 meter right.
- (c) Round chamfer 0.25 meter radius left and right.
- (d) Round chamfer 0.25 meter left, 0.125 meter right.

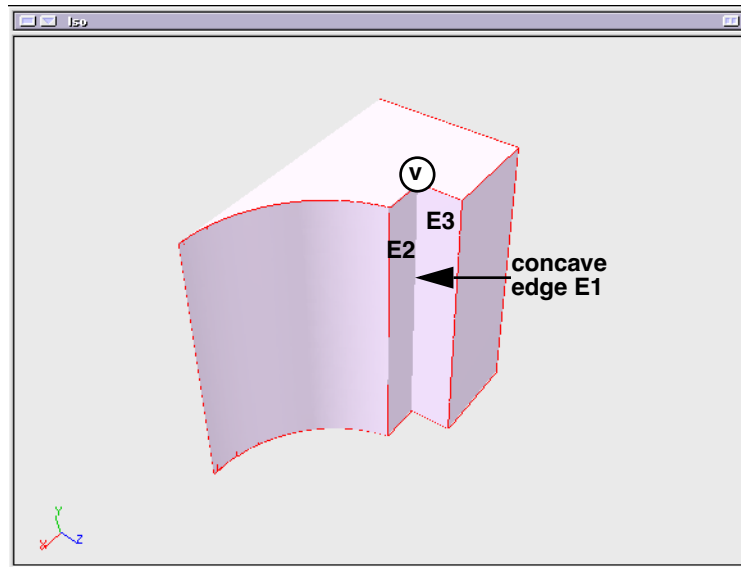
*Figure 16.30 — Chamfering samples.*

### Blending Tips

There are a few things that are important to keep in mind when performing fillets, blends, chamfers and rounds.

1. Pick all edges that connect smoothly together for blending
2. For edges that meet smoothly, the blend radius must be equal at the common vertex

3. If a vertex has three edges coming in, two of which are convex and one concave, blend the concave edge first, then pick the convex edges along with any new edges that were created and meet smoothly connected edges. If two are concave and one convex, blend the convex one first.
4. A vertex with three edges is easier to handle than a vertex with less or more edges. Use boolean tools or knife tools to clean up the topology in the neighborhood of such vertices.
5. Sharper corners are easier to blend than smoother corners.
6. Success of blends depends heavily on the underlying curves and surfaces. If solids are made up with curves that had problems reported in them, the problems will manifest themselves in different ways, such as failure in blending. The shape of the solid may look like something that blending can be done to, but underneath the shape the mathematical definitions may be messed up. These errors are reported by the program in the status window.



**All the edges in this body except one are convex edges. The one concave edge (E1) has its two faces making an angle of over 180 degrees.**

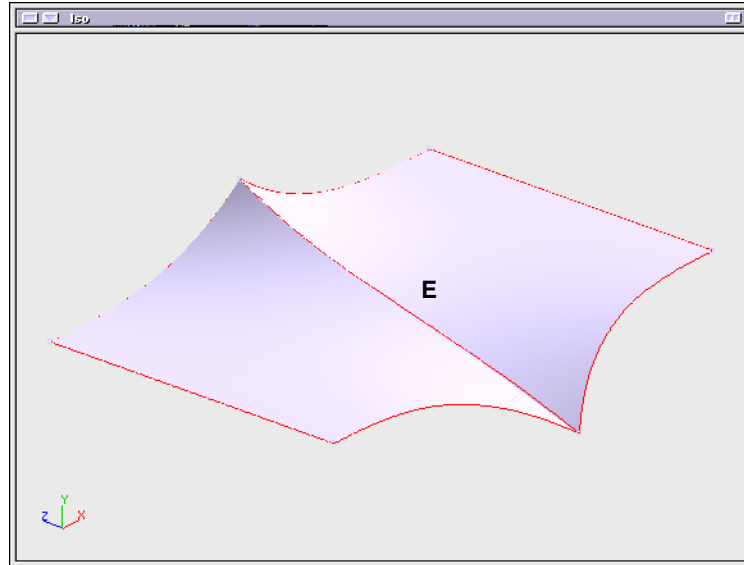
**At vertex (v) there are two convex edges (E2, E3) and one concave edge (E1). rounding the one concave edge before the 2 convex edges succeeds more often. This is called the “odd-man round first rule.”**

*Figure 16.31 — Rounding tips*

Some commonly occurring problems in curves are discontinuities. Discontinuities can be visible as cusps or they can be subtle. The subtle type of discontinuities are a result of coincident CV and multiple knots at the same place. Tips to avoid these include:

1. Do not snap the knots or end points of handles of the beziers to other handles or knots within the curve.
2. Do not zero the lengths of any tangent handle or segment. This is actually a different way of stating the previous point as zero length tangent

- snaps the tangent endpoint to the knot.
3. For non-zero tangent handles, make the handles as big as you can afford to.
  4. Do not create self intersecting Bezier curves.



**Edge (E) is a mixed convexity edge as the convexity changes from convex to concave during the length of the edge. Such edges can occur with free-form surfaces and cannot be rounded.**

*Figure 16.32 — An object like this cannot be rounded*

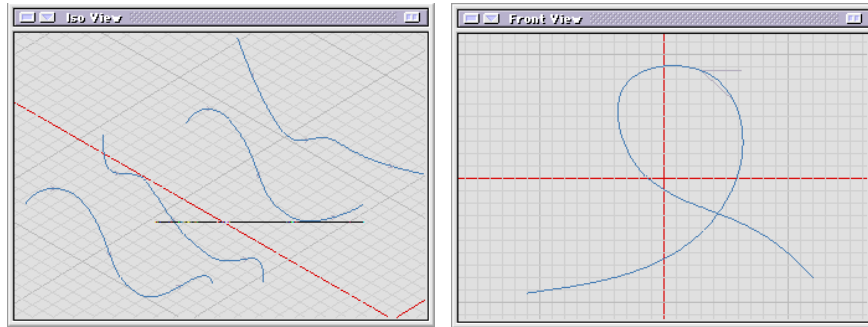
### 16.6 Skin Tool

Skinning or lofting is one of the most powerful ways to create free-flowing, undulating surfaces. These surfaces can be used to build everything from automobile bodies to human bodies. To create a skin, first create a series of ribs to apply the skin surface. Any Primitive Curve tool can be used. The only limitation is that all the ribs need to be open or closed, but no mixture

of the two. Different types of wires can be lofted. A wire can loft to a Bezier curve, then to a NURBS curve. Likewise, you can skin between wire curves that have varying numbers of knots. You can even use edges of existing surfaces as ribs.

An important thing to remember about skinning is that the final loft is greatly affected by the direction of the wires the ribs were created. The direction of each wire is indicated by an arrowhead if the System Preference, Show Edge Direction, is turned on. To have a smooth, untwisting skin, all the wires need to be going in the same direction. If you have a wire with a perfect shape, but the direction is going the wrong way, use the Reverse Direction tool under the Wire Editing Palette.

To create a skin surface on the ribs, choose the tool and click on each rib in sequential order. The skin follows the direction in which you click. After the last rib has been selected, double-click in null space. The new skin surface is placed in the currently active layer.



**Any wire primitive type can be used and even mixed to create the ribs for a skin surface, as long as they are all open or all closed. To avoid unwanted twisting, keep all the wires going in the same direction.**

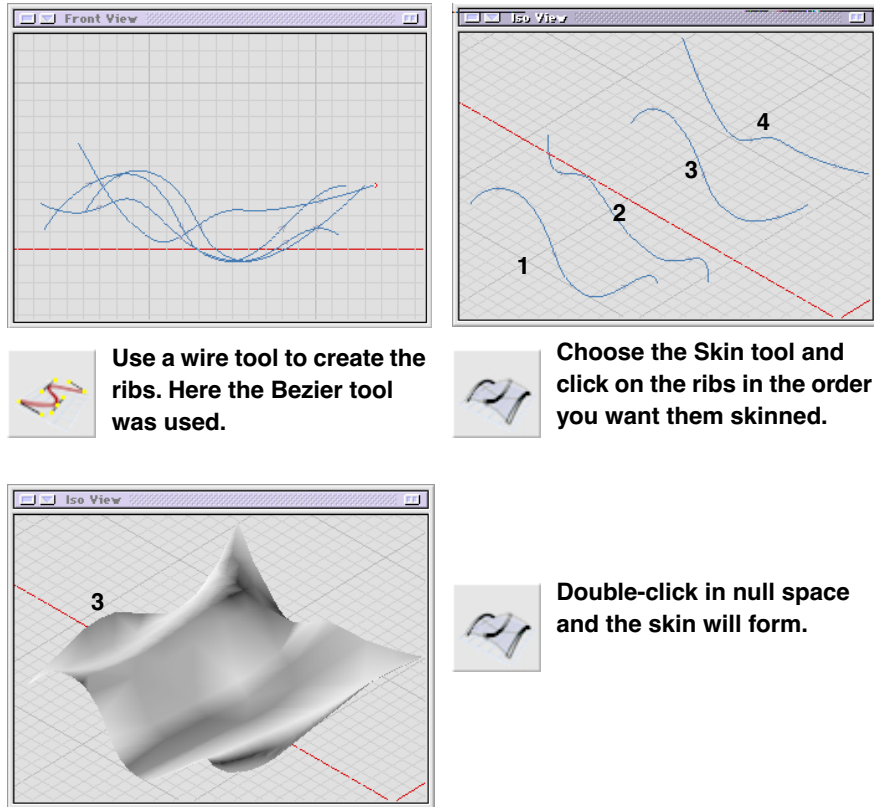
**All the Skin tools require legal wires, meaning that they cannot cross over themselves, like the above example.**

*Figure 16.33 — Legal and illegal rib examples*

If you plan on using a pre existing shape edge, you don't have to set the Pick Filter. The Skin tool knows to look for a wire.

Key points for creating a proper skin:

- Make sure all the lines are going in the same direction.
- Use either all open or all closed wires to make the ribs.
- Select the ribs in the order you want the skin to form.
- Cross-sections with the same number of vertexes skin better.
- The start vertexes should be closely aligned.
- When ribs are closed, start/end vertexes should line up with each other to minimize twist.



**Figure 16.34 — Creating a skin from wire curves.**

Like other derivative tools, when you create a new shape the original parts are still visible. Using the technique for creating different layers that was covered at the beginning of this chapter (page 171) will help keep things organized.

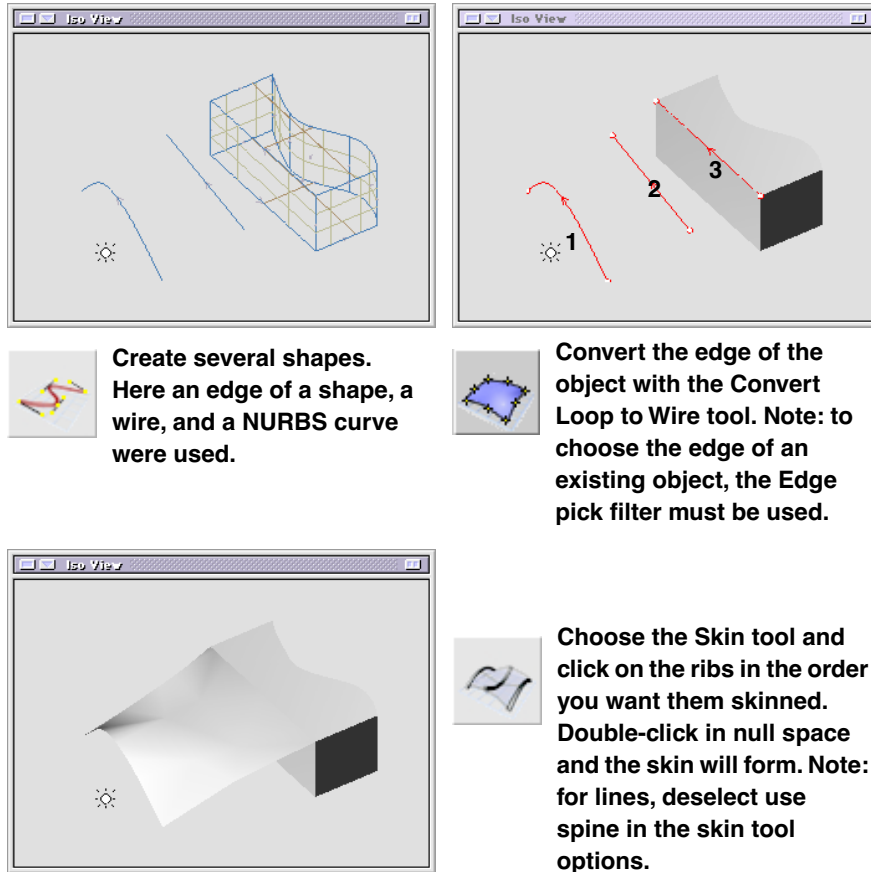


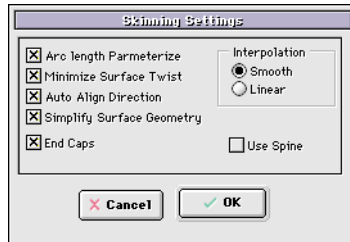
Figure 16.35 — Creating a skin from wire curves

### Creating Optimum Skins

Skinning can sometimes result in surfaces with twists that are hard to get rid of. Twists could be due to direction misalignment, start vertex misalignment and bad parameterization. Sometimes there is no way of getting rid



of these twists, the trick is distributing these faults over the entire surface area so the final result is acceptable.

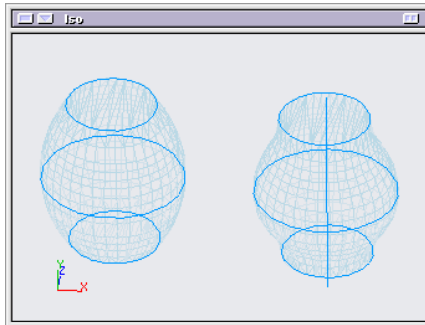


**Figure 16.36 — Skinning options**

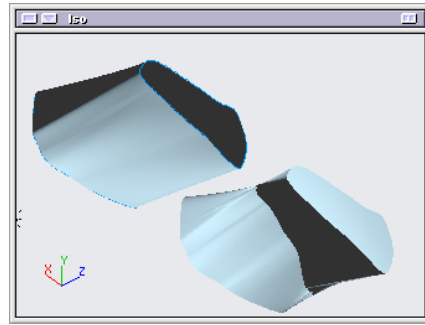
Ideal input to the skin tool are cross-sections that have very similar topologies, meaning they have the same number of vertexes and edges. The start vertexes also need to be closely aligned and so must the direction of the edges. There are some tool options to help produce better skins. Below is a description of the options that are available by double-clicking on the tool icon.

### **Arc-Length Parameterize**

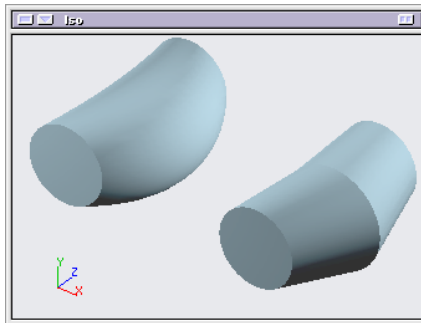
If toggled on, this will parameterize the curves by the length (in animation this means objects travel equal distance in equal time, therefore achieving a constant speed over motion paths). When two cross-section wires have nearly the same lengths but are different curve types or have different topologies, this will be a useful option. For example, a rectangular wire when skinned with a circular wire will work well with this option toggled on.



In the example above, three sets of circle primitives were skinned. On the left the default settings were used. On the right, a line was created to act as a “spine” and the “Use Spine” option was toggled on. The resulting skins show the skin on the right using the spline to align the tangents of the surface.



The starting points on the two cross-sections above were on the opposite sides. By using the Minimize Surface Twist option, the shape on the upper left was properly formed. Without this option, the shape would be twisted, as on the lower right.



Two different interpolation settings were used on identical cross-sections. On the left, Smooth was toggled on, on the right Linear was toggled on.

Figure 16.37 — Examples of different skin options

If the input skin cross-sections have large variations in length, it is better to *not* have this option toggled on, as this will result in large compensations in parameter speed at the end of the cross-section as one curve is much shorter than the other.

### Minimize Surface Twist

This option tries to reduce twists using arc length parameterization or first point alignment. If two cross-sections have nearly the same lengths but have shifted starting points, this option will be useful.

### Auto Align Direction

As the name suggests, this will automatically align the direction of the cross-sections resulting in less twisted surfaces.

### Simplify Surface Geometry

This option will try to convert NURBS geometry back into analytic (sphere, cone, torus) if the resulting surface can be converted. Most algorithms proceed fast when the geometry involved is analytic rather than NURBS. Skinning is an algorithm performed in the NURBS domain, and thus results in NURBS surface. But if you skin two coaxial circles, you could represent the resulting surface with cone surface, which is what this option does.

### End Caps

Toggled on, caps will be placed on the ends of the surface if they are planar.

### Interpolation

With the smooth option the degree of the surface resulting from skinning is typically 2 or 3 (rather than linear) in the direction of skinning. With the linear option, sharp creases may be visible at vertexes.

### Use Spine Option

If selected, after all the ribs have been selected the user will be prompted to pick a path. This path will be used to align the tangents of the surface or curve.

## 16.7 Face to Face Loft Tool

This tool allows the user to create a loft between two faces of two different bodies.

To use the tool, pick one face at a time and after the second face is selected a separate sheet body will be formed between the two faces.

This is similar to the Blend Surface from Edges tool (See the next section), except it requires picking closed faces rather than a set of edges. For surface characteristics, refer to the option settings indicated in the Skin tool. The only additional setting is the perpendicular option.

The loft direction perpendicular option is used to specify the direction of the take-off vector, either perpendicular to the co-edge, or in the loft direction. The default is to use the loft direction, because a perpendicular take-off vector can cause self-intersections in the resulting surface.

The continuing curve created by the Face-Face Loft tool is influenced by the surface direction of both edges that it originated from. To determine the strength of this influence, change the numeric input next in the Tangent 1X

and Tangent 2X boxes. The higher the number, the greater the influence. The Tangents 1 and 2 are determined by the order of edge selection.

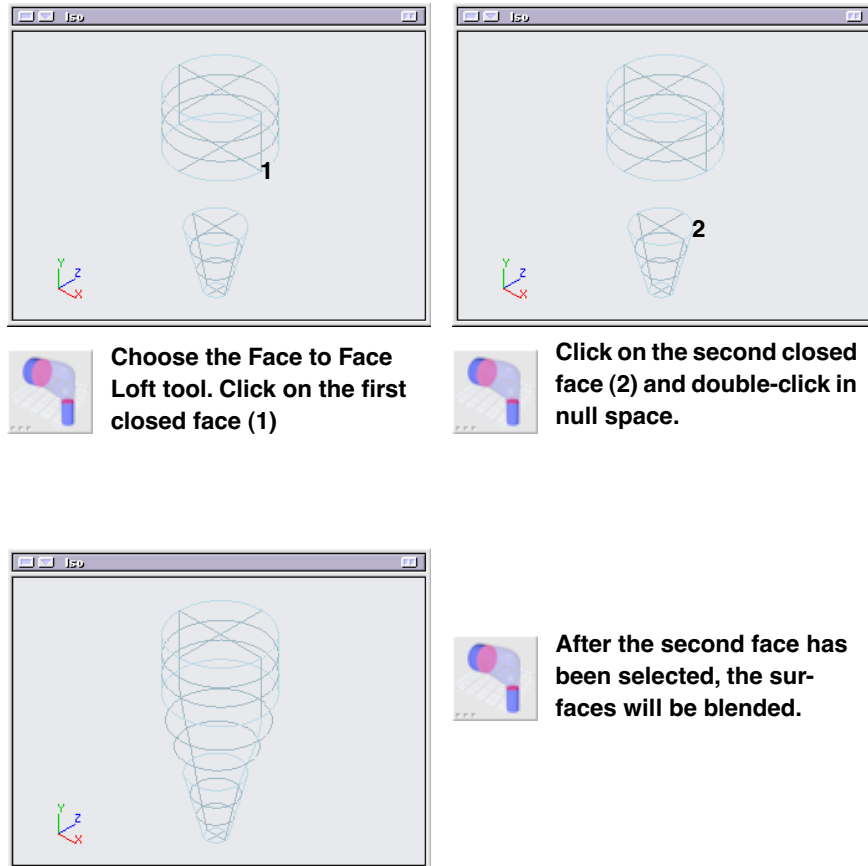


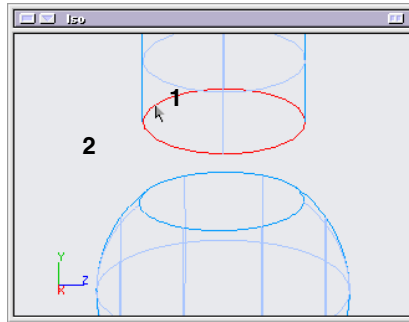
Figure 16.38 — Lofting between two faces.

### 16.8 Blend Surface From Edges (Surface to Surface Blend)

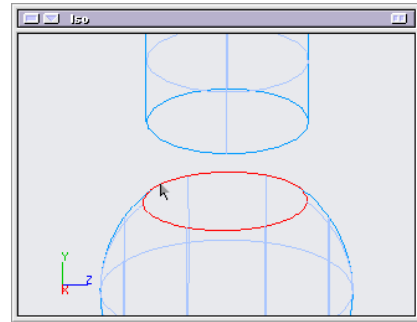
The edges of two surface objects can be blended together smoothly using the Surface-to-surface blend tool. The option box, accessed by double-clicking on the tool, provides options to help ensure a smooth transition between the surfaces.

If you are blending exactly one edge on each side, set the pick filter to Edge, click on the first edge, and then the second. The surface blend will be created immediately after you pick the second edge. If however, you are blending more than one edge on either side, set the pick filter to something else (Body for example). Now choose the tool icon and click on the edges of the first surface to blend, double-click in null space when done to complete the selection. Continue by clicking on the edges of the second surface object, double-clicking in null space when done. The surface will be created after the second edge-set selection is made.

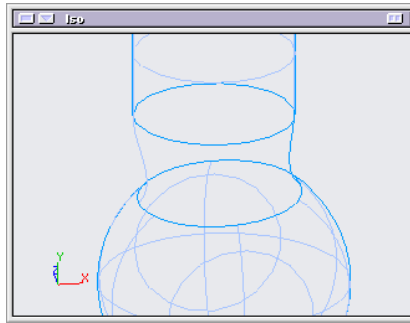
Options are available to clean up parameterization, align the edge directions, simplify the surface, minimize the twist and to loft the direction perpendicular to the edges.



**Choose the Surface Blend tool. Click on the first surface edge (1) and double-click in null space (2).**



**Click on the second surface edge and double-click in null space.**



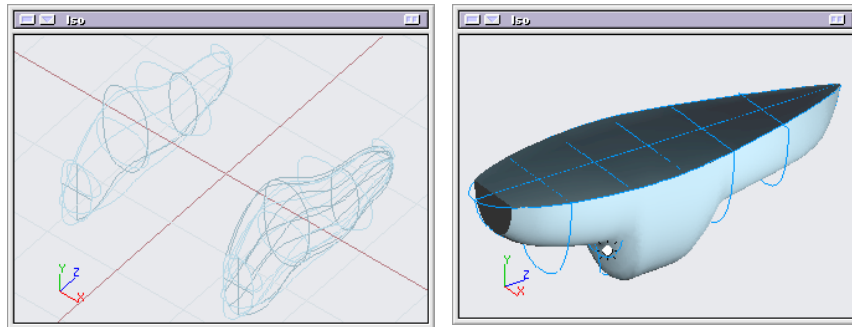
**After the second edge has been selected, the surfaces will blend.**

**Figure 16.39 — Blending between two edges**

The continuing curve created by the Blend Edges tool is influenced by the surface direction of both edges that it originated from. To determine the strength of this influence, change the numeric input next to the Tangent 1X and Tangent 2X boxes. The higher the number, the greater the influence. The Tangents 1 and 2 are determined by the order of edge selection.

### 16.9 Profile Skin Tool

The Profile Skin tool lofts a series of closed ribs over a set of top- and side-profile ribs. The tool can be used to create precise organic shapes based on at least three ribs (top, side and cross-section), but really shows its power when a series of ribs are created to direct the loft. The wires used to create the final object can be made by any of the Curve Primitive tools, including the Polyline, Bezier or NURBS curves. The profiles must be closed, and the cross-sections can be either all open, or all closed.



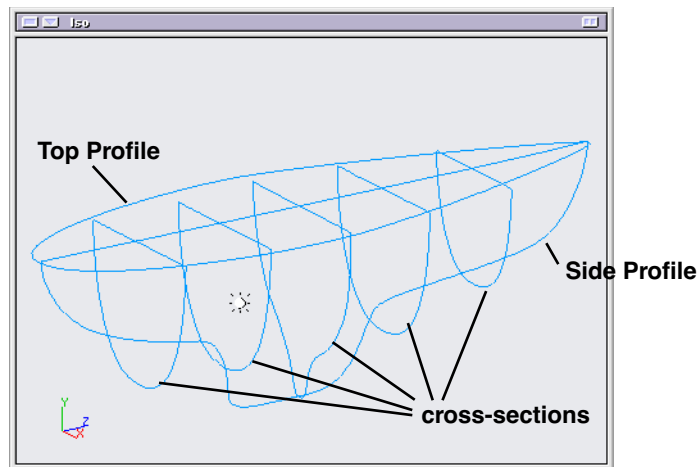
**NURBS curves produce the most efficient surfaces. In the example above the left shape was made with NURBS, the right with Beziers, Notice the number of faces needed for the Bezier form. Converting Bezier curves with the Single Spline tool will simplify the body.**

**The boat hull above was formed using the Profile tool. Note how the cross-sections on the right don't line up exactly with the side section. As the tool is used, the program makes adjustments to the profiles so they match. On the next page is how the hull looks as the profiles are being selected. The computer generates and displays the new "ribs" based on the profiles.**

*Figure 16.40 — Profile skin examples*

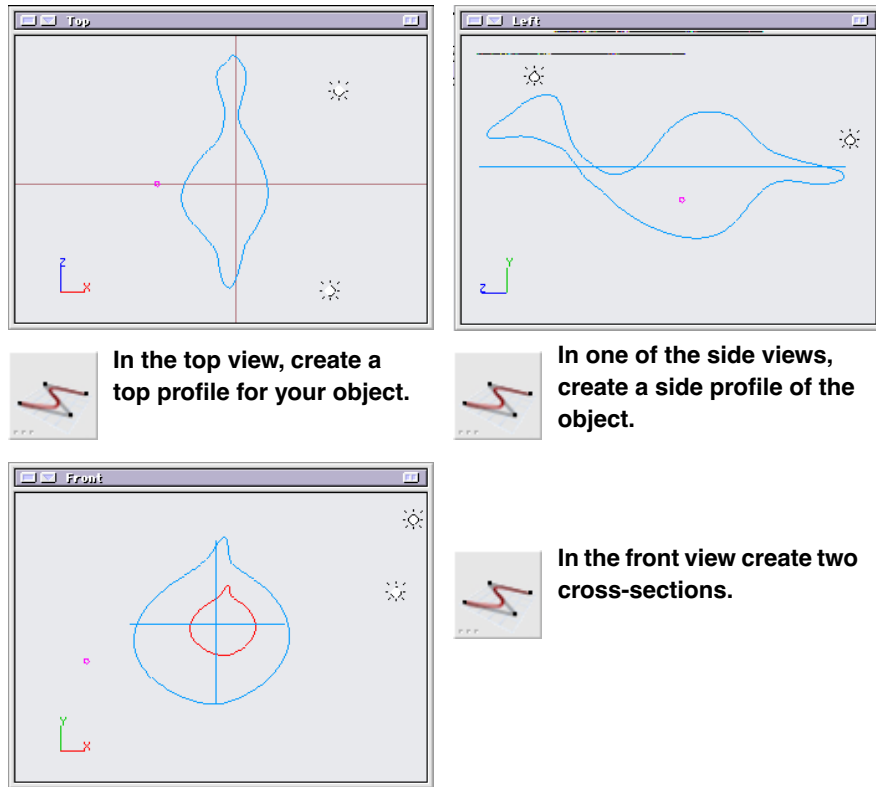


To use the tool, create a top profile in the top view window, then a side profile in either the left or right view windows and a series of cross-sections in the front view window. All of the profiles and cross-sections must be parallel to those views. Choose the tool and click on the profile in the top view, then the profile in the side view, and then each cross-section. After the last cross-section has been picked, double-click in null space.



**Figure 16.41 — Different profiles in a boat hull**

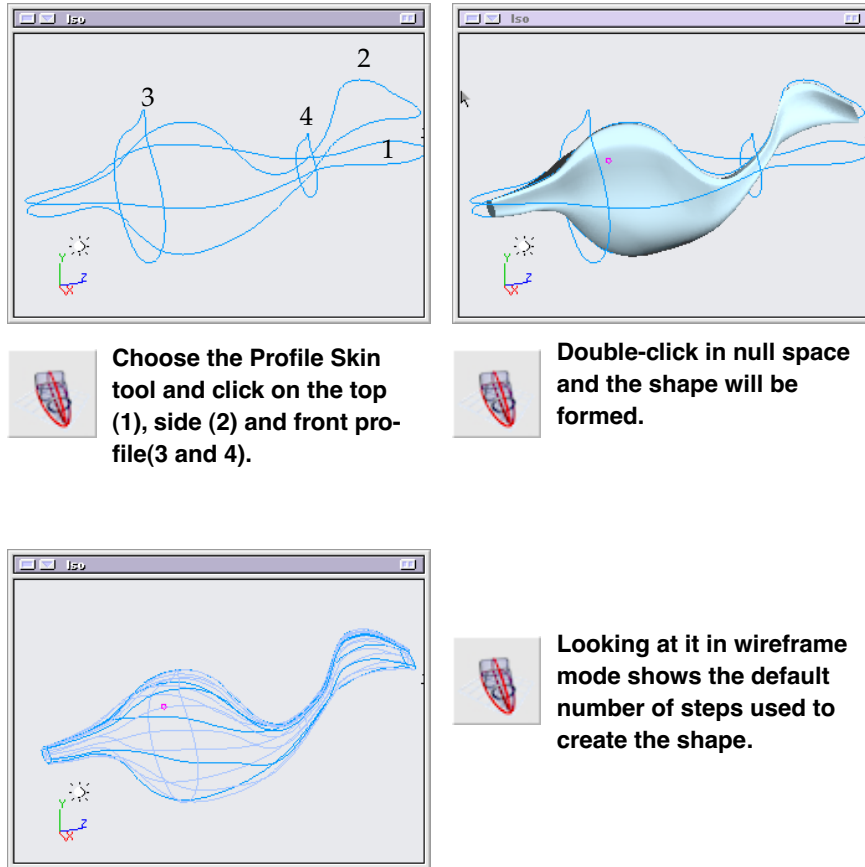
The side profile is scaled in length by the tool to fit the length of the top profile. The scaling is non-uniform. Cross-sections are then scaled to fit both the side and top profiles. Any number of cross-sections can be used, but you need at least one.



**Figure 16.42 — Steps to use the profile skin tool.**

Tips to improve profile skin:

1. Make sure all curves are Planar
2. Change end tolerance if shape generated is erratic near the ends



**Figure 16.43 — Steps to use the profile skin tool**

Special consideration is required at the ends because sometimes the cross-section will get squeezed to a point causing the skinning to fail. Therefore the “End Tolerance” setting is required. This is set to 0.05% default. What this means is that the first and the last generated cross-section will be placed at a distance greater than 0.05% of the length of the profile from the

ends of the profile. You can reduce this tolerance, but it could very well approach the precision limitations of the underlying algorithms and could generate some degenerate cross-sections. If such a cross-section is generated, it will be skipped by the skinning process. You may, in fact, find it beneficial to increase the tolerance to get a better profile skin.

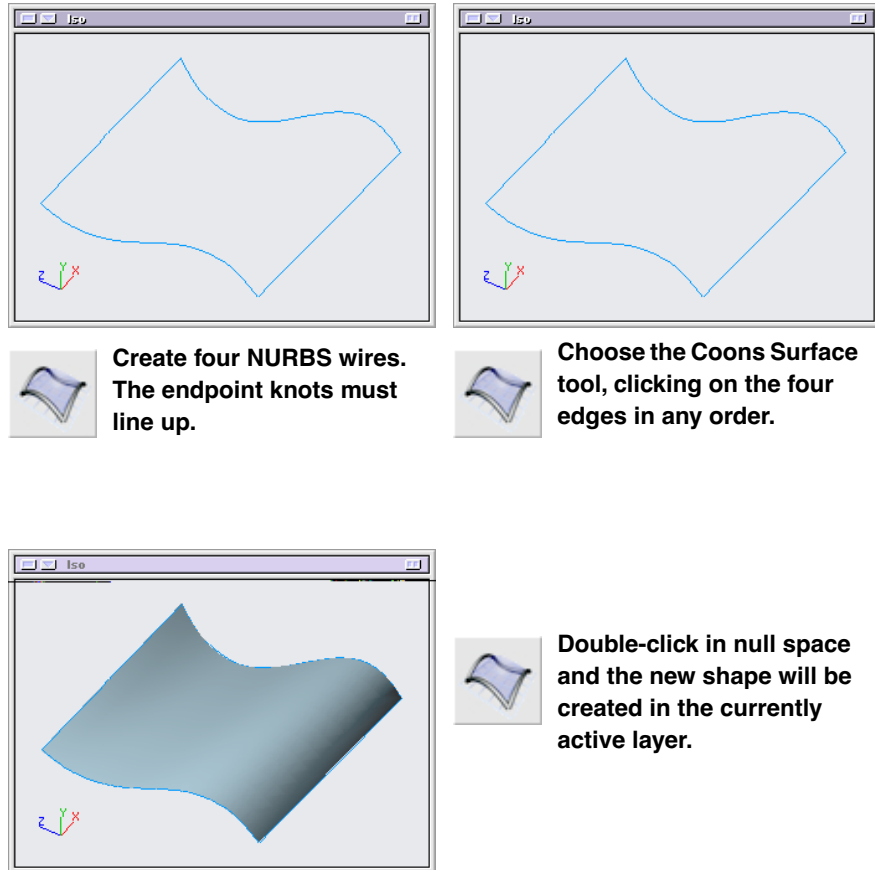
### 16.10 Coons Surface Tool

The Coons tool can create a complex, undulating surface using four edges. Straight lines and arcs may also be used. To use, match each of the four NURBS curves at their end points using snap tools. Each wire can have varying number of knots. Select the tool and click on the four edges in any order. The surface constructed is always a NURBS surface and can be edited with tools in the NURBS Surface palette. The tool will generate a single face sheet body.

A Coons surface is created from four boundary curves. The boundary curves must be edges. The tool will only pick edges, even though they may be partially wires. The edges must all be open.

Key points for creating a Coons Surface:

- Always use four edges. Any of the edges can be the edge of an existing face.
- Endpoints must be touching and form a rectangular “hole.” To have the endpoint vertexes match, either create the original curves with snapping on or set the Snap Filter to Vertex and use the NURBS Editing tool to move the CV’s into position.
- If the four sides are wires with many edges, use the Net Surface tool.



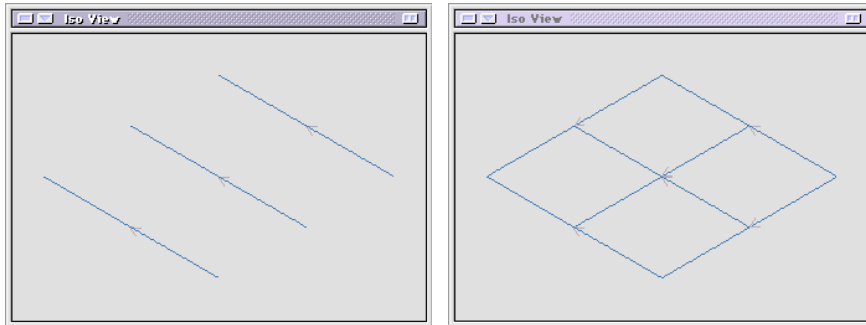
*Figure 16.44* — Example of using the Coons surface tool

### 16.11 Net Surface Tool

Net surfaces can be thought of as skinning in two directions simultaneously. The Net Surface tool is intended to let users create surfaces from a network of curves. These curves are wire bodies that can be created using either Bezier or NURBS curve tools. These curves must form a grid going in U and V directions. U and V can be considered directions which can be configured for an individual model, but not so fixed as having to constrain to world coordinates like X or Y. Obviously curves typically are changing direction along their lengths. Imagine these curves are made of flexible rubberbands. If you straighten, stretch or unfold these curves to result in a rectangular grid, the curve set forms a valid network.

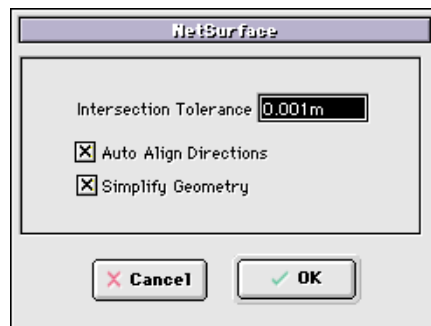
Another requirement is that each U-curve must intersect (touch within tolerance) each V-curve and visa-versa. This is extremely hard to setup without the use of snapping. One can, however, set the distance which is considered close enough to be considered for two curves to be touching.

In other words, even if two curves do not intersect, you can increase the “Intersection tolerance” to make it big enough so that the two curves will be considered as intersecting. Of course, this distance shouldn’t be made too large, in fact, it is recommended that one should change tolerances in a modeling program only when there is no other option.



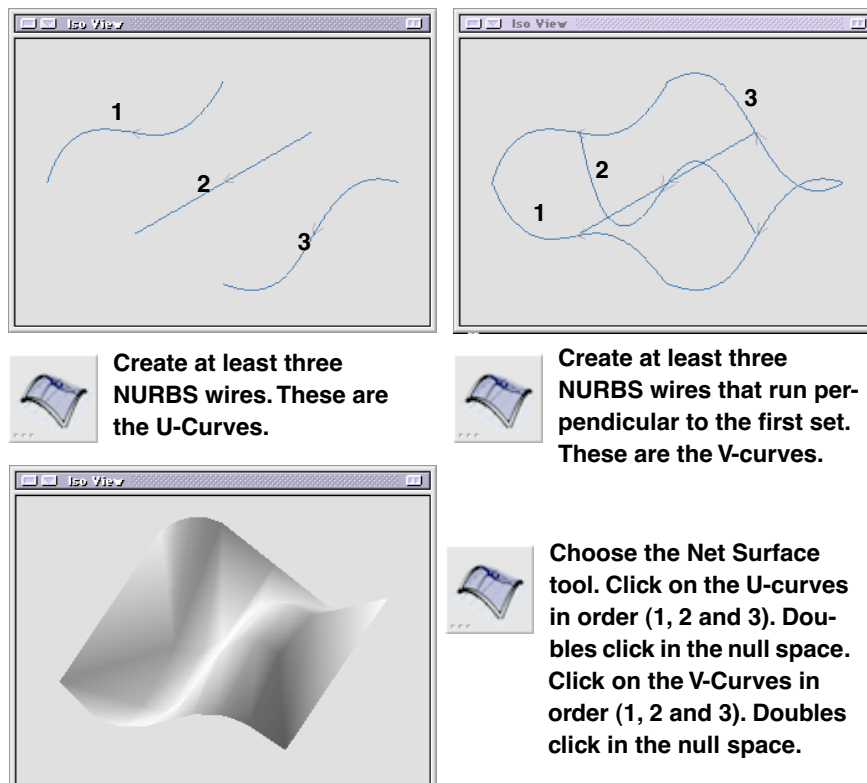
To create a Net Surface, you need at least three wires, each must be going topologically in the same direction. These are called U-Curves.

The second set of curves have the same requirements and run perpendicular to the U-Curves. The second set that completes the net are called the V-Curves.



Double-clicking the icon brings up the Net Surface options box.

*Figure 16.45 — Options and UV directions of the Net Surface tool.*



**Figure 16.46 — Creating an object using the Net Surface tool.**

There are many ways in the Universe Modeler to create curves so that they intersect. When editing a curve with the “Snap to vertex” switch on, one can make the CVs or knots of the curve being edited to snap to the vertexes of the other curve. Curves have knots all along their lengths and at the end points. The end point of edges are also vertexes. These vertexes and knots are not drawn by default. Under the System Preferences menu, display dialog may be used to turn drawing of knots and vertexes ON while creating perfectly snapped curves.

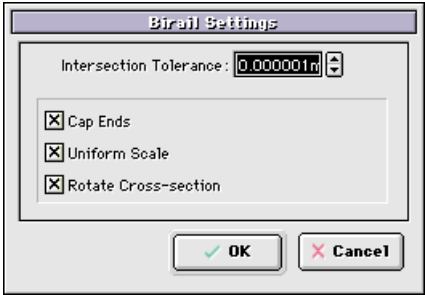


Once you have drawn a network of these curves, the Net Surfaces tool is straightforward. First, you must pick all the curves going in the U direction, double-click in null space and pick all the curves going in the V direction and double-click in null space. If the network is valid, a surface will be generated that passes through (interpolates) all the curves in the network.

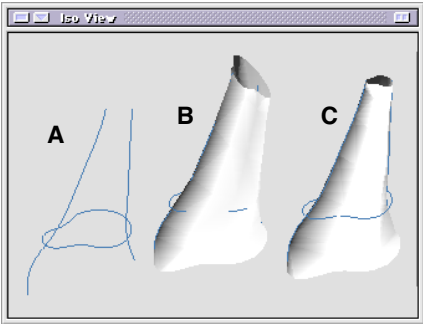
All curves must be at least C1 (parametric tangent-continuous), which means, for example, if the curves used were Bezier, then there were no cusps in the bezier and also the tangent lengths on each side of the internal knots were equal. It also means one can't use polyline networks because polylines are only C0 (positional) continuous. Polyline networks are easier to create, however, and can be converted into NURBS using the "Polyline to NURBS" tool. If you use "as Knots" option of that tool, the points on the polyline will be used as knots, and thus the curves generated will pass through the vertexes of the polyline.

### 16.12 Birail Tool

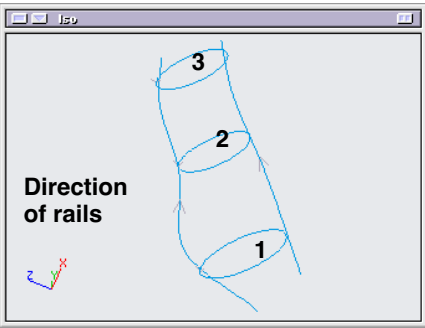
The Birail tool allows for the creation of an object by sweeping cross-sections along two rails simultaneously. The cross-section is scaled, uniformly or non-uniformly, to keep contact with the rails. One of the two rails has greater control over the cross-section, and is designated as the PRIMARY and the other as SECONDARY. The control is exerted in determining the orientation of planar cross-sections along the curves. Planar cross-sections are oriented so their normals are along the tangents of the primary rail. Non-planar cross-sections are valid input, however in that case no orientations will occur.



Double-clicking the icon brings up the Birail options box.



With the Uniform Scaling option, the cross-section can be scaled so that the sweep touches the rails. In the example above “A” is the original wire bodies, “B” is without scaling, “C” is with scaling turned on.



cross-sections must be selected along the direction of the rail.

Figure 16.47 — Birail options and examples.

To set up a birail, you need an open or closed wire that will act as the cross-section and two wire curves that act as the rails. The rails must intersect the cross-section. The rails cannot loop back on themselves.

### Key Points for Creating a Birail

- Rails must go in the same topological direction. Use the Reverse Direction tool if they do not.
- Rails must not cross over themselves.
- The cross-section, also called generator, must make contact with the rails before the sweep can begin. Use of snaps will be generally necessary to meet this requirement.
- The cross-sections must be selected along the direction of the rail. For example, the cross-section nearest to the start of the rail must be the first picked.

Double-clicking on the Birail tool brings up the settings box. The options include:

### Cap Ends

With this option selected, the shape that is created will be a single-sided solid object, with open ends capped off. Without this option the shape will be a two-sided, uncapped surface object.

### Uniform Scale

The cross-section can be scaled uniformly or non-uniformly to maintain contact with the rails.

When planar cross-sections are used, it is the primary rail that determines how the cross-section will be oriented. It is always tangential to the primary curve.

### **Rotate Cross-Section**

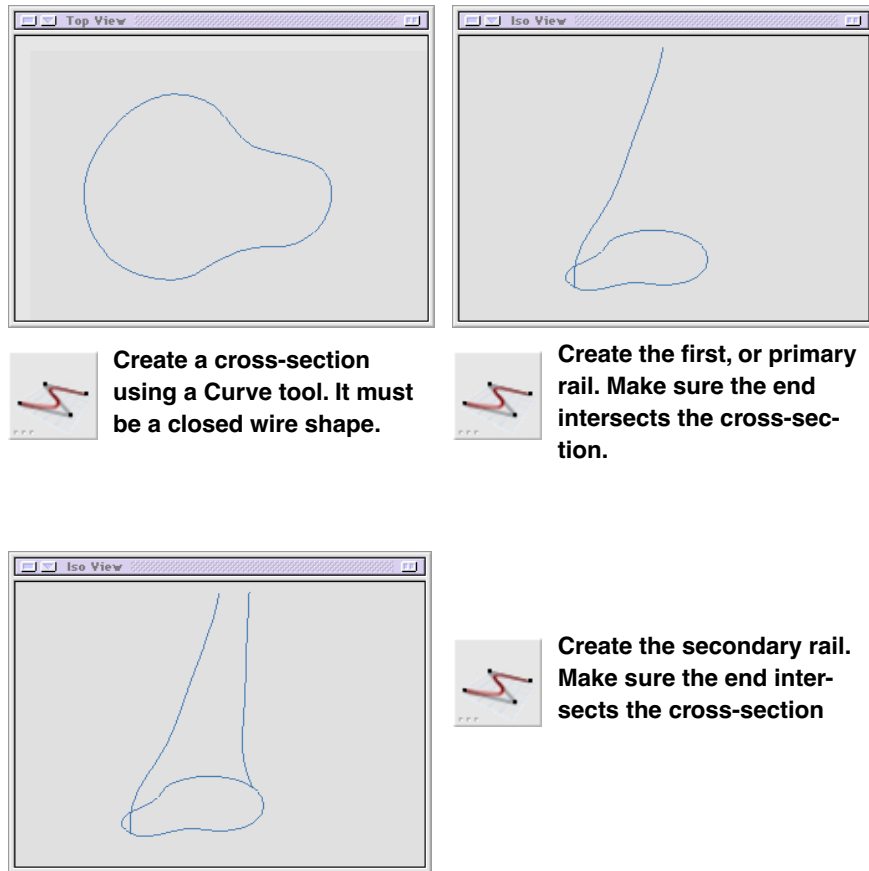
This option will prevent any tilting that may be required to align the normal to the tangent of the primary.

### **Linearly Interpolate**

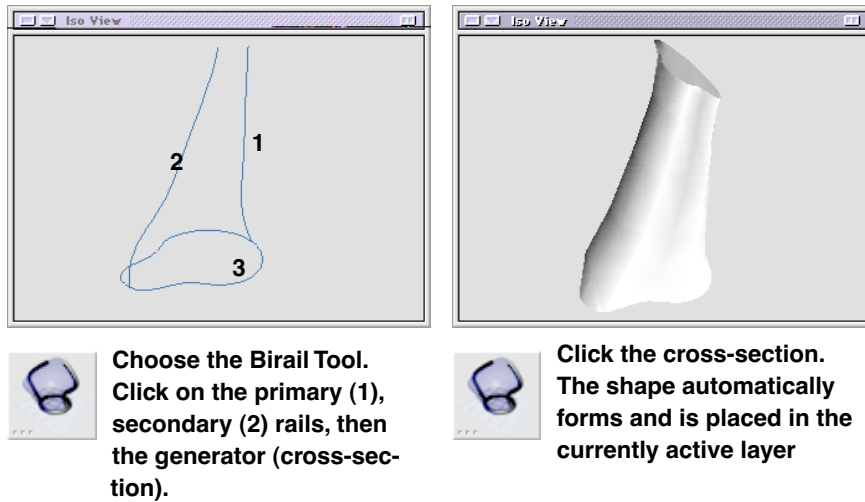
When more than one cross-section is used, this option will force a linear interpolation between them.

### **Intersection Tolerance**

Distance within which to consider two points as coincident.



*Figure 16.48 — Using the birail tool*



*Figure 16.49 — Using the birail tool*

## Polyline Palette

Polylines form a basic, yet important foundation in creating wires that will later evolve into more complex shapes. A polyline is a wire body with each edge having a straight curve definition. The Polyline tools give you the ability to draw straight polylines and then edit them as needed. The finished polyline can be filled and be used as a surface object, or can be used as a path or cross-section for tools like Revolve or Sweep.

Wires created with the Polyline tool don't have to remain polylines. You can use the Polyline to NURBS tool in the Wire Editing palette to convert the line to a NURBS curve. The result will be a largely rounded version of what you started out with. You can also use the Edit Bezier tool on a new polyline. The result will be a wire that can be edited as a Bezier. Even though the initial polyline didn't have any curves in it. Handles can be pulled out to create new curves, or entirely new portions of the line can be added or joined as needed.

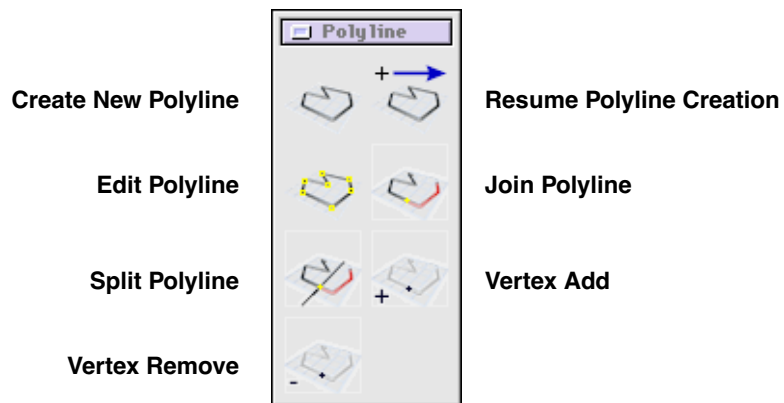


Figure 17.0 — Polyline Palette

## 17.0 Create New Polyline Tool

To create a line you click to set a point, move your mouse, and click again to set the next point. To finish a line either double-click where you want the last point to create an open polyline, or click on top of the first polyline segment to form a closed polyline. Once completed the Polyline Edit tool will automatically toggle on so you can edit the polyline you just created.

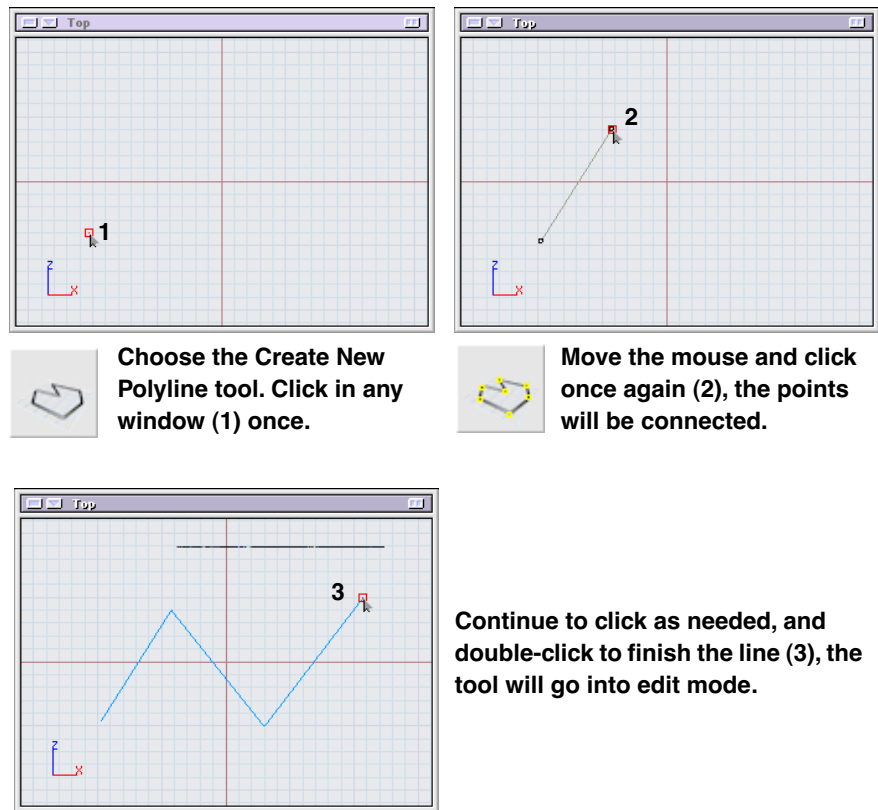
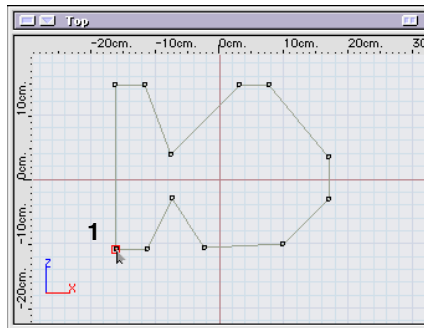


Figure 17.1 — Creating a new polyline

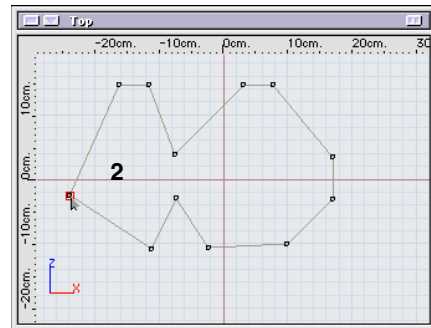


## 17.1 Edit Polyline Tool

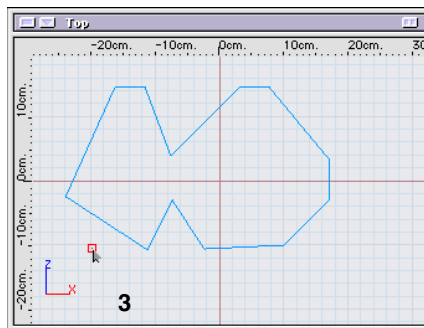
When a new polyline is created this tool is automatically activated. You can also select the tool at any time and click on a polyline to edit it. Double-clicking in null space will take the current polyline out of edit mode. If the “P” key is held down at the time of the double-click, the polyline will also be projected onto the plane of the active workplane.



**Create a polyline, choose the polyline tool and click on vertex (1) or line.**



**Hold the mouse down, release the mouse at the new location (2).**



**Double click (3) in null space to leave edit mode.**

Figure 17.2 — Editing a polyline.

## 17.2 Resume Creation Tool

After creating a polyline, the program switches to Edit mode, after editing, if you want to continue drawing the polyline where you left off, click on this tool and continue the line.

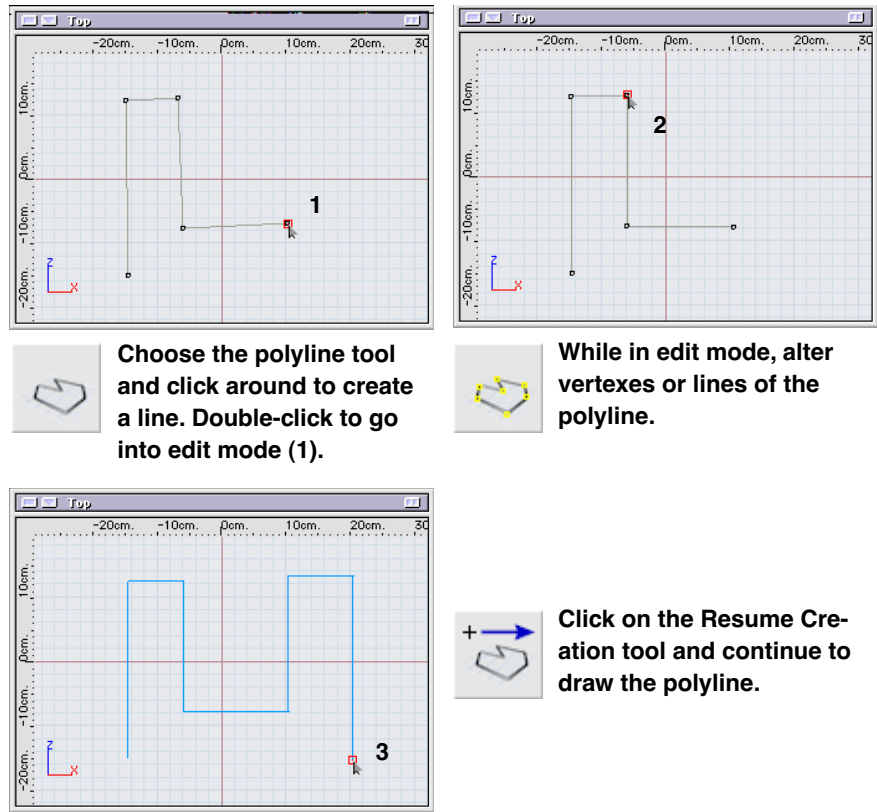
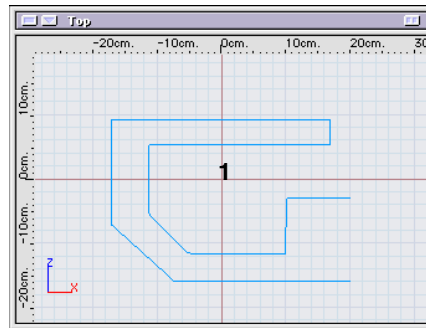


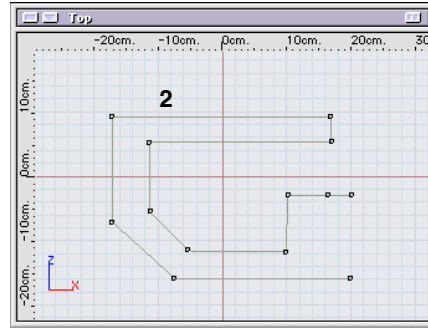
Figure 17.3 — Resume creation of a polyline

## 17.3 Join Polyline Tool

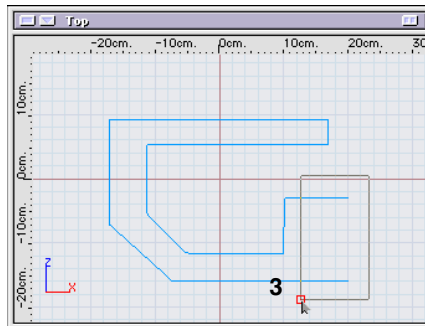
Any two open polyline ends can be joined using this tool. To join lines, choose the Join Polyline Tool and drag a selection rectangle over the points that need to be joined. Polyline ends can be either two separate segments, or two ends of the same line being joined to close the wire. Lines to be joined must be in edit mode.



**Create a new open polyline, or two separate polylines (1).**



**Choose the Edit Polyline tool, click on the lines to join (2) to put them into edit mode.**



**Choose the join tool, click and drag to select (3) the endpoints to be joined.**

*Figure 17.4 — Joining polylines.*

# 17.4 Split Polyline Tool

A polyline can be broken either on a line, or at a vertex. To use the tool, click on the Edit Polyline tool and select the line to be split, then click on the Split Polyline tool and click where needed. The vertexes in Figure 17.5 were moved slightly to make the split more obvious.

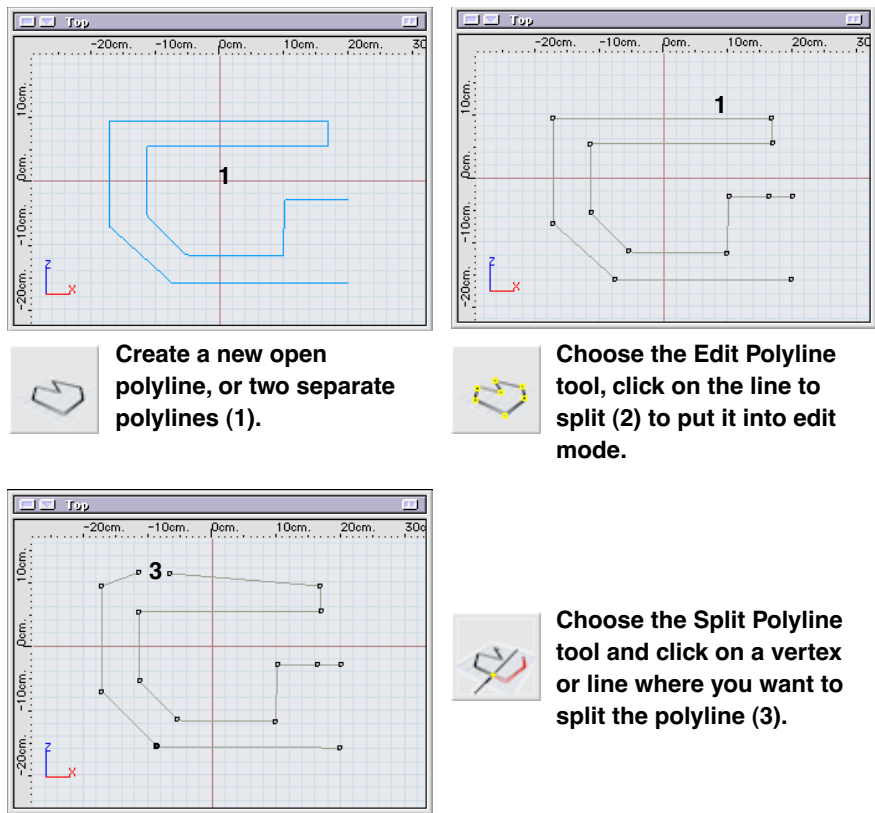
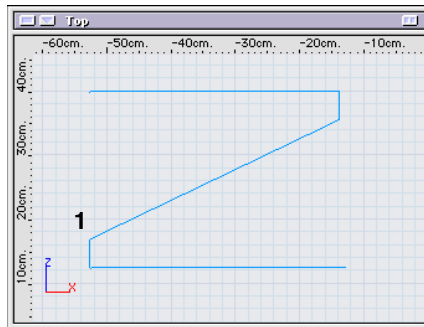


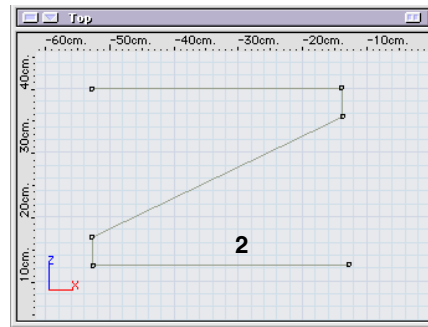
Figure 17.5 — Splitting a polyline

## 17.5 Add Vertex Tool

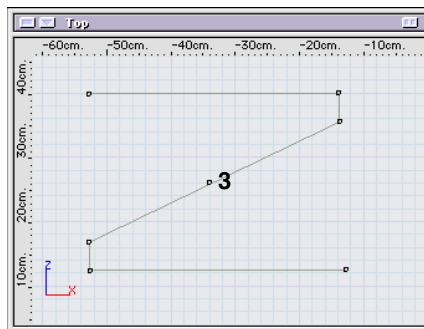
Additional vertexes can be added to a polyline. To use the tool, put the polyline in edit mode, and then select the Add Vertex tool and click on the line where the vertex is to be added.



**Choose the Create New Polyline and draw a polyline (1).**



**Either go into edit mode, or choose the Edit tool and click on the polyline (2).**



**Choose the Add Vertex tool and click on the polyline where you want to add a vertex (3).**

*Figure 17.6 — Adding a vertex to a polyline*

## 17.6 Remove Vertex Tool

To remove a vertex from a polyline, choose the Edit tool and click on the polyline to put it into Edit mode, then choose the Remove Vertex tool and click on the vertex to be removed.

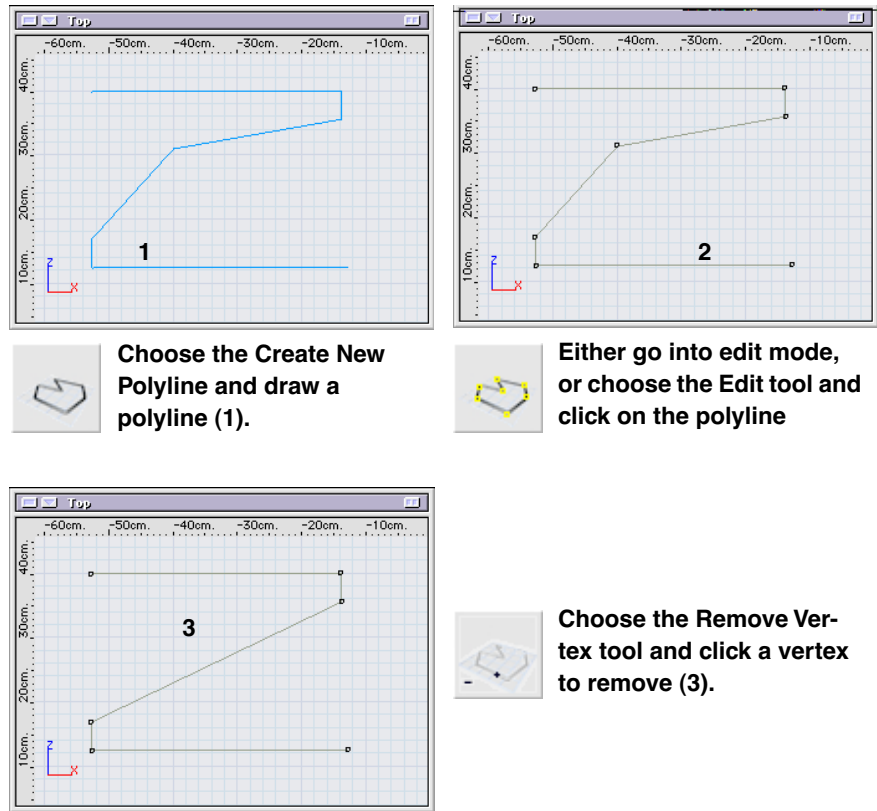


Figure 17.7 — Removing a vertex from a polyline

## Bezier Palette

The Bezier tool gives users the ability to create wires with familiar tools from traditional 2D drawing programs. The Bezier tool creates what is called a “piece-wise Bezier curve.” This means that each curve created by the tool, although a single wire, contains several pieces. When a newly created wire is formed, each piece is given its own direction arrow. This helps you determine whether a wire form is a single NURBS curve or a piece-wise curve.

Cubic and non-rational NURBS curves can be edited with the Bezier Editing tool, but when they are, they’re converted to piece-wise Bezier curves.

Many users may choose to design their original wires using the Bezier tool and convert them to NURBS curves for final shape creation. For more information on the topological differences between piece-wise Bezier curves, see the section on Changing Topology on page 173.

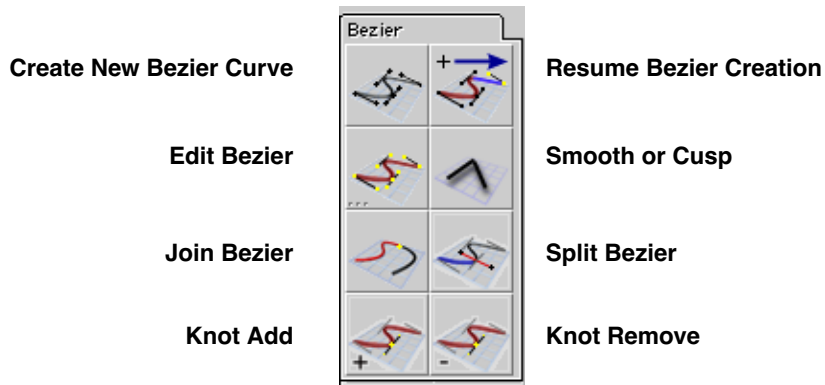
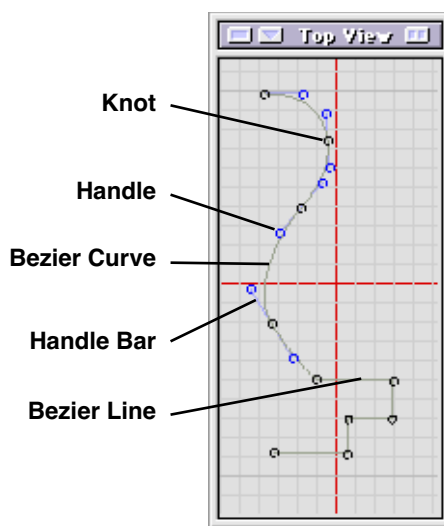


Figure 18.0 — Bezier Palette

### 18.0 Create New Bezier Curve Tool

Bezier line creation is based on the principle that each mouse click adds a new knot. If the mouse is held down and dragged, a curve is created and can be adjusted during this time by moving a handle. If the mouse button isn't held down but immediately released, the user creates a straight, linear point.

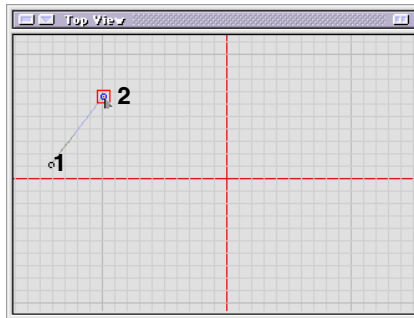


*Figure 18.1* — This wire was created with the Bezier tool— at the top by clicking and dragging and on the bottom by just clicking.

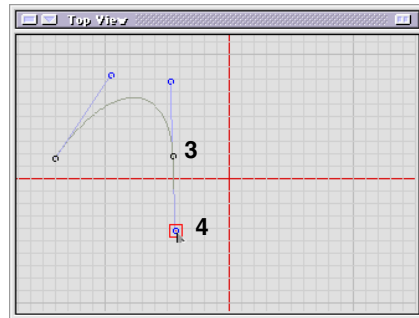
Open or closed wires can be created with the Bezier tool. To create an open wire, double click on your last point. You can also leave the Bezier creation mode at any time and the wire will be formed up to that point. Closed wires are created by returning the mouse to the first point and clicking. If the final click is placed within the size of the aperture setting, the wire auto-



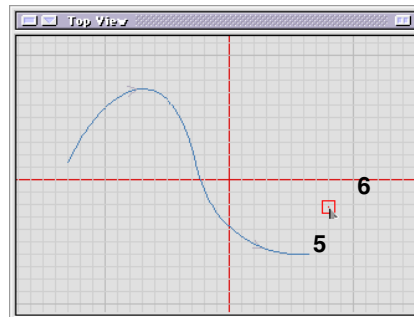
matically closes. Aperture settings are determined by double-clicking on the Pick tool and setting the number of pixels.



**Choose the Create New Bezier Curve tool. Click and hold the mouse down (1) and drag and release (2).**

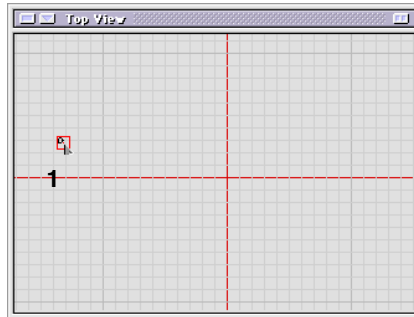


**Click in a new area and hold the mouse down (3). Drag and release (4).**

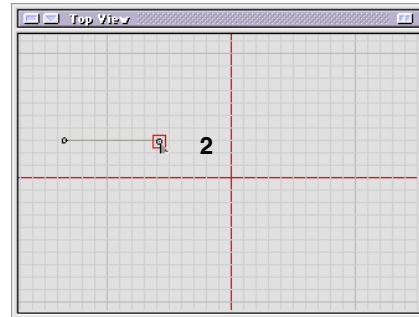


**To end the line, double-click (5). To form the line, double-click in null space (6).**

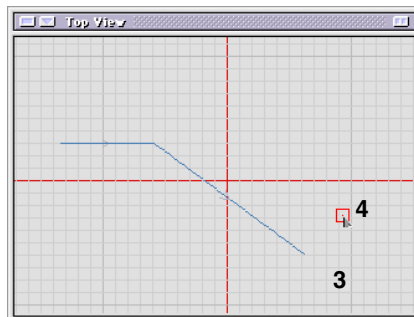
*Figure 18.2 — Creating a Bezier curve*



Choose the Create New Bezier Curve tool. Click and release the mouse button (1).



Click in a new area, and release the mouse button (2).



To end the line, double-click at the last point. To form the line, double-click in null space.

Figure 18.3 — Bezier lines

### Things to Avoid with the Bezier Tool

While it's possible to make complex forms with the Bezier tool, there are some aspects that won't translate well in the 3D world. These include: Bow ties, Cross-overs and Co-resident points. Bow ties and cross-overs look pretty much like they sound, lines that loop over and cross back on themselves. Co-resident points are control points, or vertexes, from the same line that lie directly on top of each other.

### 18.1 Edit Bezier Tool

When a Bezier curve is created, it is not carved in stone. After a curve has been created using the New Bezier tool, the program automatically defaults to the Edit Bezier tool so fine tuning adjustments can be made. You also can choose the tool at any time and go back into lines you've already created and tweak them as needed. To use the tool, click on the toolbox icon and select the line you want to edit.

The Edit Bezier works on both Bezier lines and NURBS curves that are non-rational, where no weight editing has been done. When editing NURBS curves, the line is converted into a piece-wise Bezier. For topological purposes, you may want to turn the curve back into a single spline when you're done. The tool also works on some unfilled wire primitives shapes including the rectangle and line primitives.

The Edit Bezier tool can change a line by affecting four parts of a Bezier curve:

#### The Knot

The tool can reposition knots by click and dragging on them.

#### The Handle

The curve's definition can be changed by clicking on the handle and repositioning it.

#### The Bar

Clicking and dragging on a bar constrains the curve's angle in the direction of the bar.

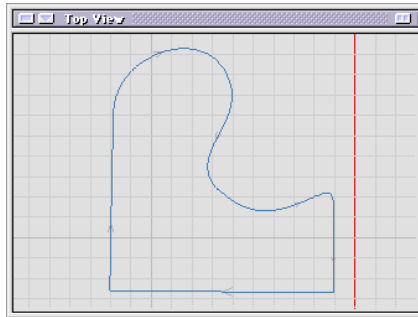
### The Curve

By clicking on the actual curve, you can reposition it without changing the shape of that segment. Adjoining segments will change to adapt to the new position.

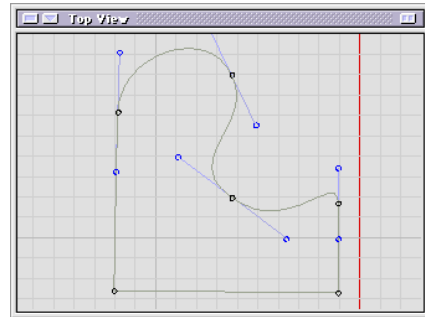
To alter the curve of one segment of a Bezier curve without affecting the other side, hold down the Shift key.

After adjustments have been made to the line, double-click in null space to finish the edit and to form the final line. If the “P” key is held down at the time of the double-click the Bezier curve will be projected onto the active workplane.

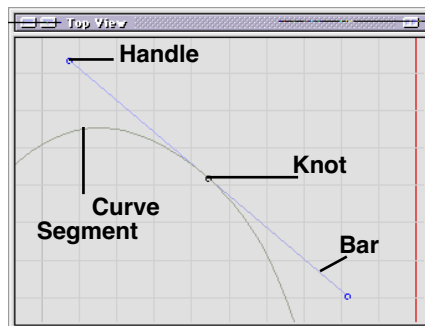
Circles, ellipses and areas converted into NURBS create rational NURBS, hence they can’t be edited with the Edit Bezier tool.



**Fully formed Bezier curve.**  
Note arrows showing direction.

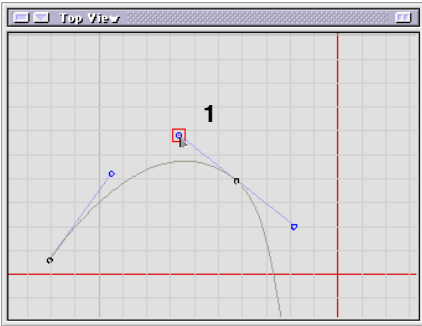


**Bezier curve in edit mode. Arrows disappear and knots and handles are visible.**

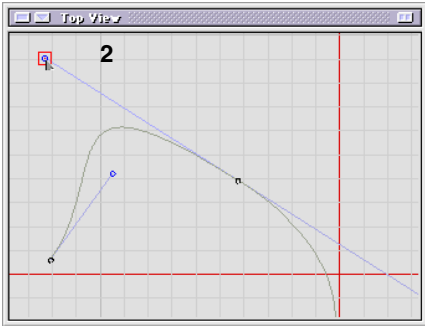


**The Edit tool can affect four aspects of a Bezier curve.**

*Figure 18.4 — Aspects of a Bezier curve*



**Choose the Edit Bezier tool and click on a handle (1).**

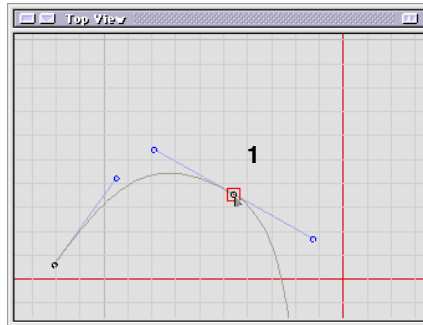


**With the mouse button down, drag the handle and release (2).**

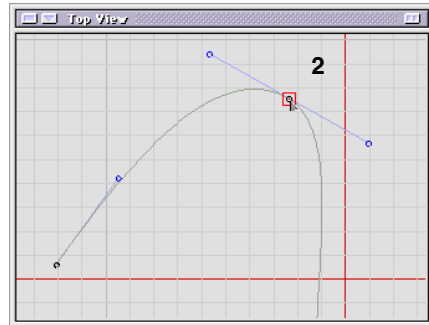


**Double-click in null space to form the newly edited curve.**

Figure 18.5 — Editing a Bezier curve



**Choose the Edit Bezier tool and click on a knot (1).**

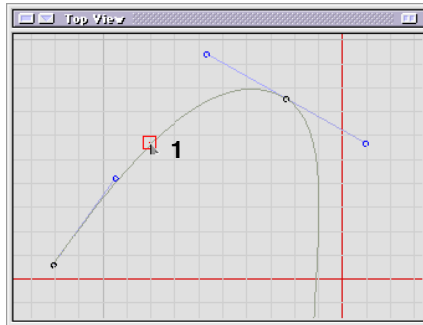


**With your mouse button down, drag the knot and release (2).**

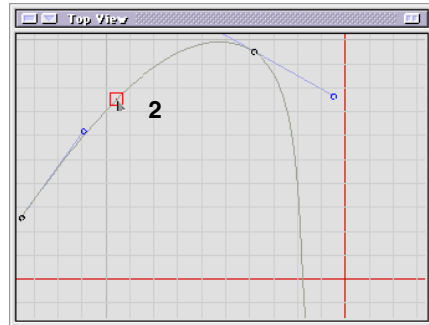


**Double-click in null space to form the newly edited curve.**

*Figure 18.6 — Editing a Bezier knot position*



Choose the Edit Bezier tool and click on a curve segment (1).



With your mouse button down, drag the segment and release (2).



Double-click in null space to form the newly edited curve.

**Figure 18.7 — Editing a Bezier line segment position**

The Edit Bezier tool has a single option, Convert Line Tolerance. This settings value is used to determine if a Bezier curve should be considered to be a straight line when the CVs of the curve are within the specified distance of a straight line. This is useful when importing Bezier curves from other programs that may not be precise. If you want to leave those wires exactly as they are in the source, then set this tolerance to 0.



## **18.2 Join Bezier Tool**

To bring two separate Bezier segments together, use the Join Bezier tool. The tool works on Bezier segments that are in the edit mode, so before using the tool, use the Edit Bezier tool and click on the two line segments to be joined.

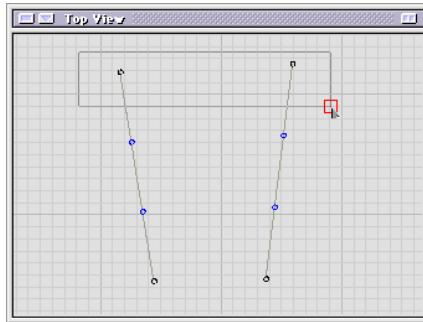
The Join Bezier tool can also be used to close an open Bezier.

To use, choose Join Bezier tool and drag a rectangle over the two end knots to be joined. A new curve segment will be created and continue the curve information from the previous two points.

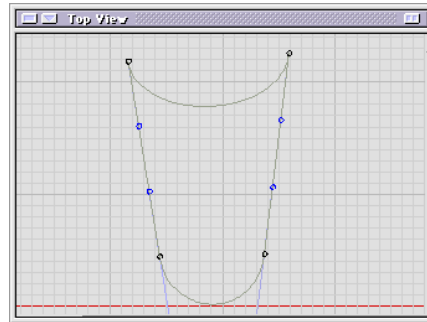
If you have two separate segments that are touching each other and you want to join them, you may have to use the Smooth/Cusp tool in combination with the Join Bezier tool to keep curve information from creating unwanted spikes. To do this, use the Snapping tools to make sure the end-points are aligned. Choose the Join Bezier tool and drag over the two points. If one or both of the segments have a curve to it, a spike will be formed. Choose the Smooth/Cusp tool and click on the junction.

Any type of line that can be edited with the Edit Bezier tool can be joined. For instance, A NURBS curve can be joined with a line made from the Curve Primitives palette. Both only need to be converted to Bezier using the Edit Bezier tool.

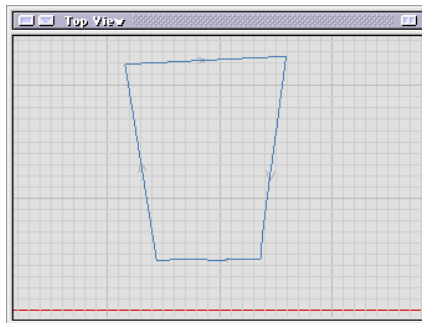
Curves and lines can join regardless of their topological direction, even if the direction of each curve is opposite. Once joined, a single direction will be assigned. If you need the direction of the curve to go the opposite direction, use the Change Direction tool in the Wire Editing Palette.



Two lines from the Wire Primitives can be joined. Here two lines were converted using the Edit Bezier tool then the Join Bezier tool.

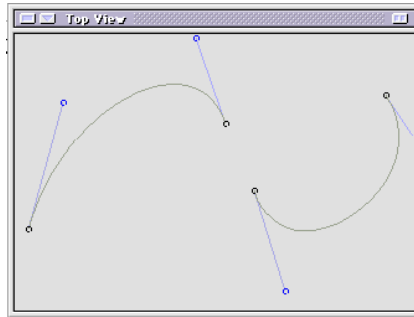


The join created the curved two lines.

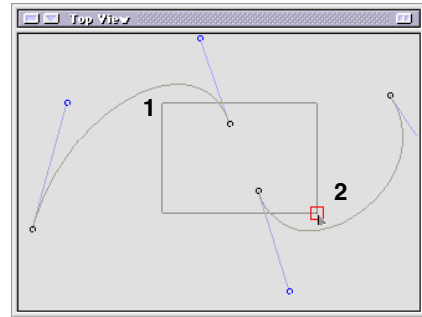


The Smooth/Cusp tool was used to remove the curve information.

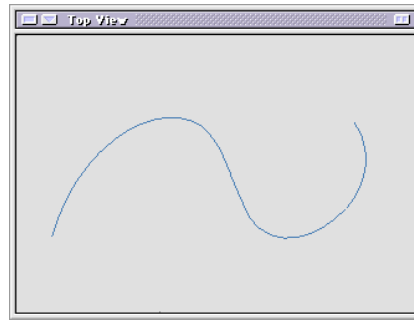
Figure 18.8 — Joining lines with the Join Bezier tool and removing curves with the smooth/cusp tool.



Use the Edit Bezier tool to put the line segments into edit mode.



Click down (1), drag and release (2) to select the two endpoints to join.



The line will be joined with a newly formed segment.

*Figure 18.9 — Joining two separate Bezier segments*

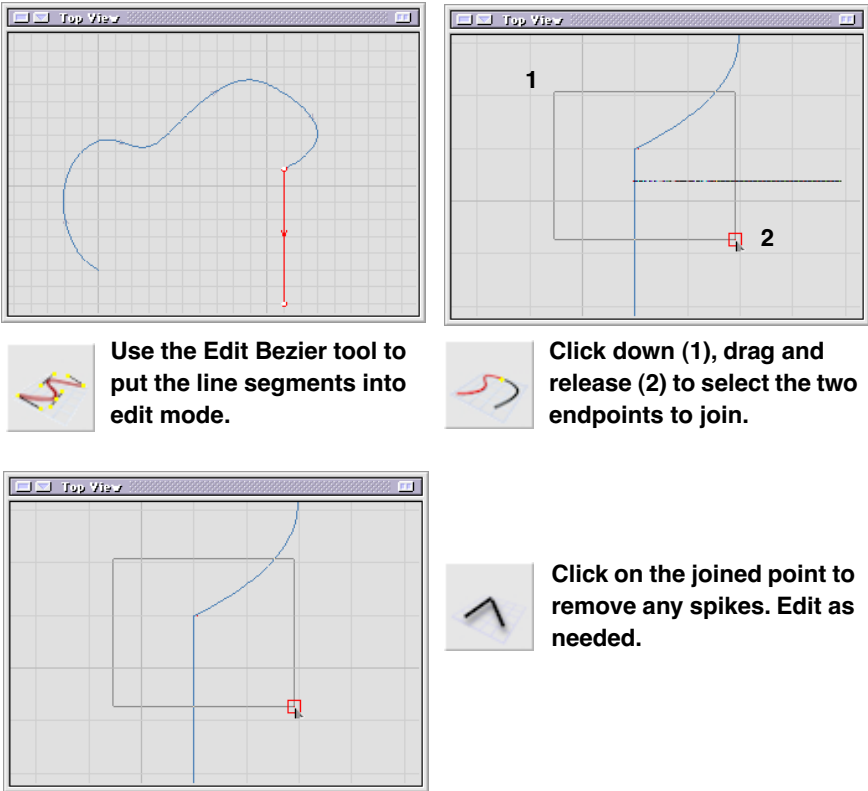
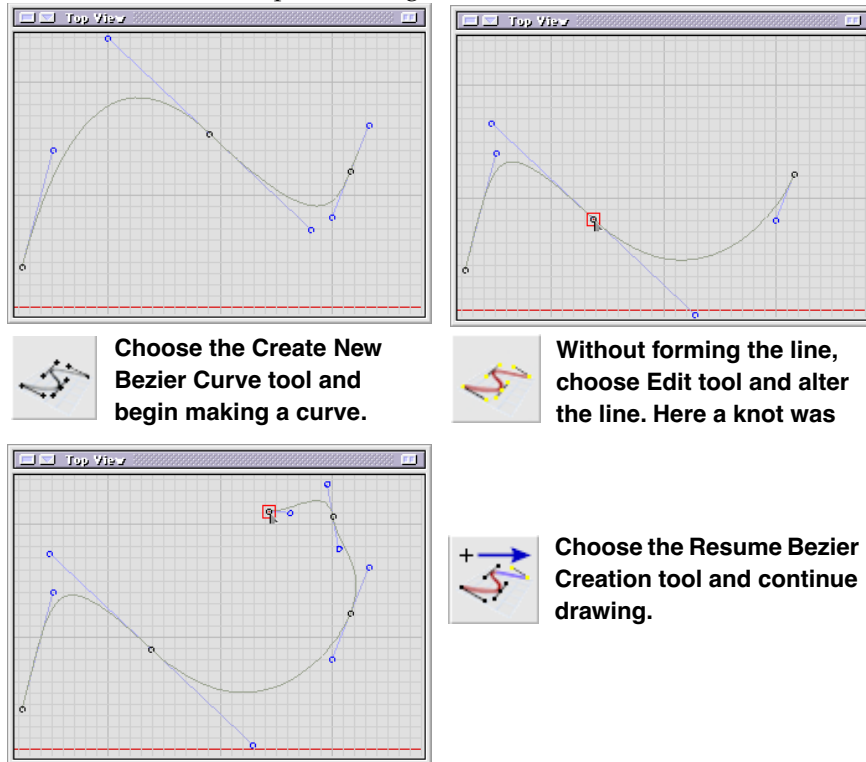


Figure 18.10 — Joining two Bezier segment endpoints that share the same area.

## 18.3 Resume Creation

Line creation can be continued even if you've already gone into editing mode or left the Bezier palette altogether.



**Figure 18.11 — Resuming the creation of a Bezier curve after editing the line**

If you have finished your curve and the tool has reverted to Edit mode and you want to continue your line, click on the Resume Bezier Creation tool to continue to draw the line from the point where you last left. While in the edit mode, move knots or handles and then go to the Resume Bezier Creation tool.

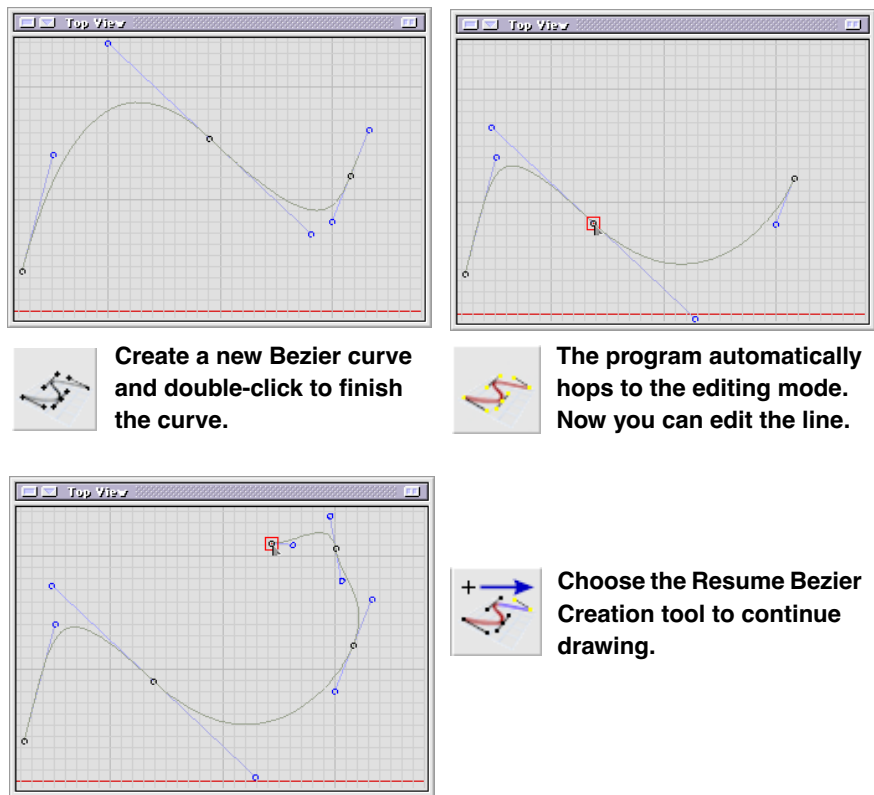


Figure 18.12 — Resuming the creation of a Bezier curve that has been finished

## 18.4 Smooth or Cusp Tool

In some instances, you may want to go from a nice, smooth curving line to a straight line. You can do this by editing the curve's handles, but that requires breaking the tangents, which isn't recommended.

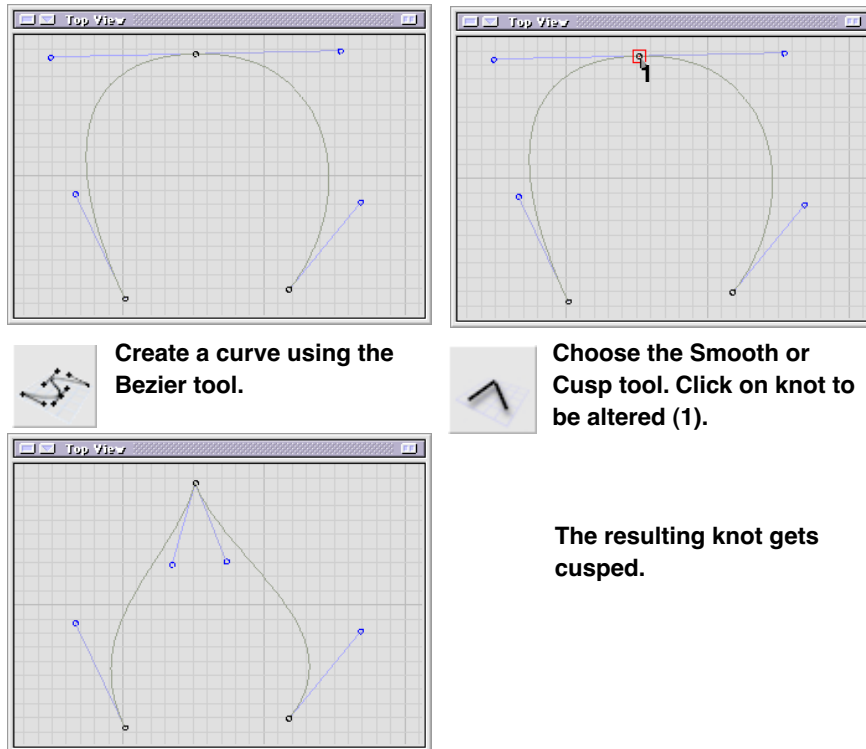


Figure 18.13 — Cusping a knot on a Bezier curve

The Smooth or Cusp tool removes curve information from a knot. A Bezier curve must be in edit mode to use the tool. The curve can be placed back into the knot by continuing to hold down the mouse and dragging.

The tool also can be used to take kinks out of complex Bezier lines.

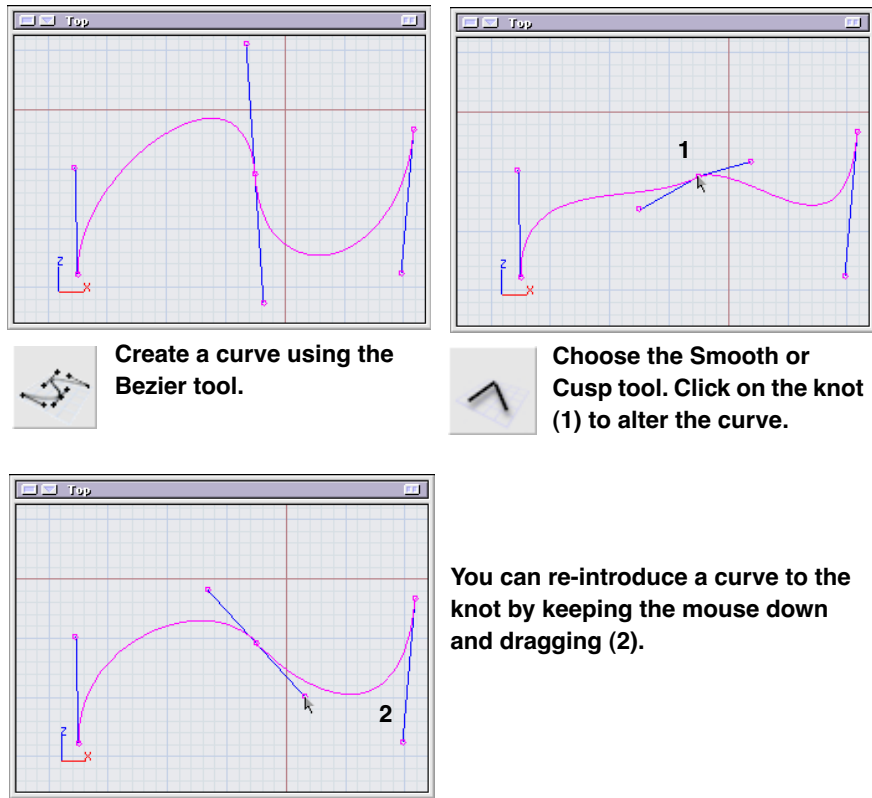


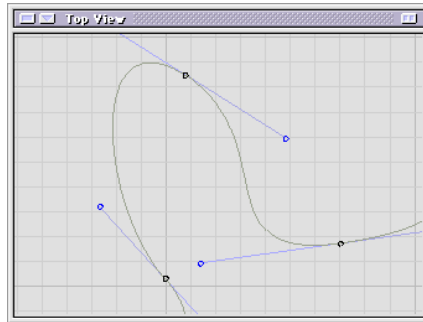
Figure 18.14 — Using the cusp tool to make an abrupt transition

### 18.5 Split Tool

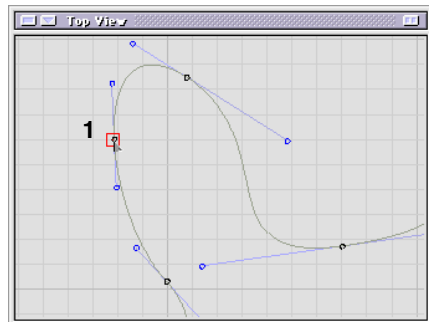
Bezier curves in edit mode can be split into separate wires using the Split Bezier tool. Use the Edit Bezier tool to put a curve in edit mode. Then select the Split tool and click on the curve where you want it to be split.



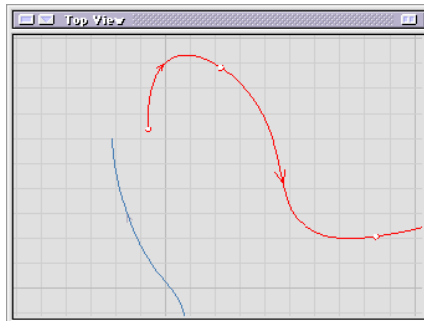
The Split tool is a destructive tool— two completely separate lines are created with no remnant of the first line. If you are editing a Bezier curve and need to use just a portion of the original line for a new shape, copy the original before splitting. This way you still have the original line.



**Create a Bezier curve or use the Bezier Edit tool on an existing curve.**



**Choose the Split tool. Click on a point where you want the line (1) to break.**



**The curve will split and both pieces can be edited as individual curves.**

*Figure 18.15 — Splitting a Bezier curve*

### 18.6 Add Knot Tool

While in the edit mode, a knot can be added to a Bezier curve by choosing the tool and clicking where you want the knot. The shape of the curve will not be altered. The new knot will function like any knot with handles for editing the curve.

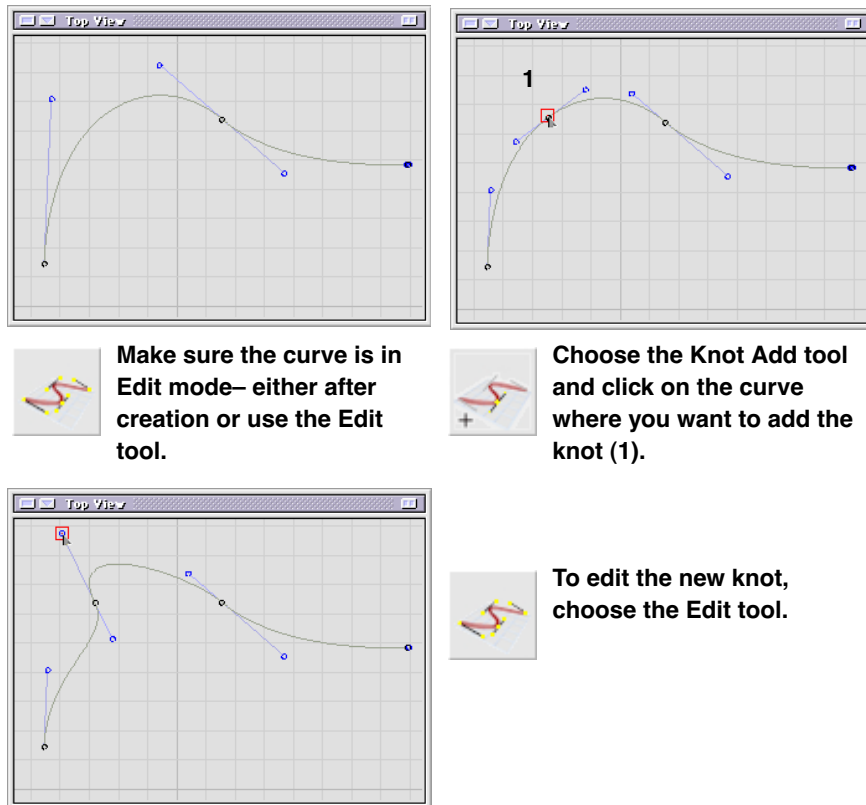


Figure 18.16 — Adding knot to Bezier curve

## 18.7 Remove Knot Tool

While in the edit mode, a knot can be removed. While this can simplify a complex Bezier, it also removes the curve information that was contained in the knot, so the shape of the original line will be changed.

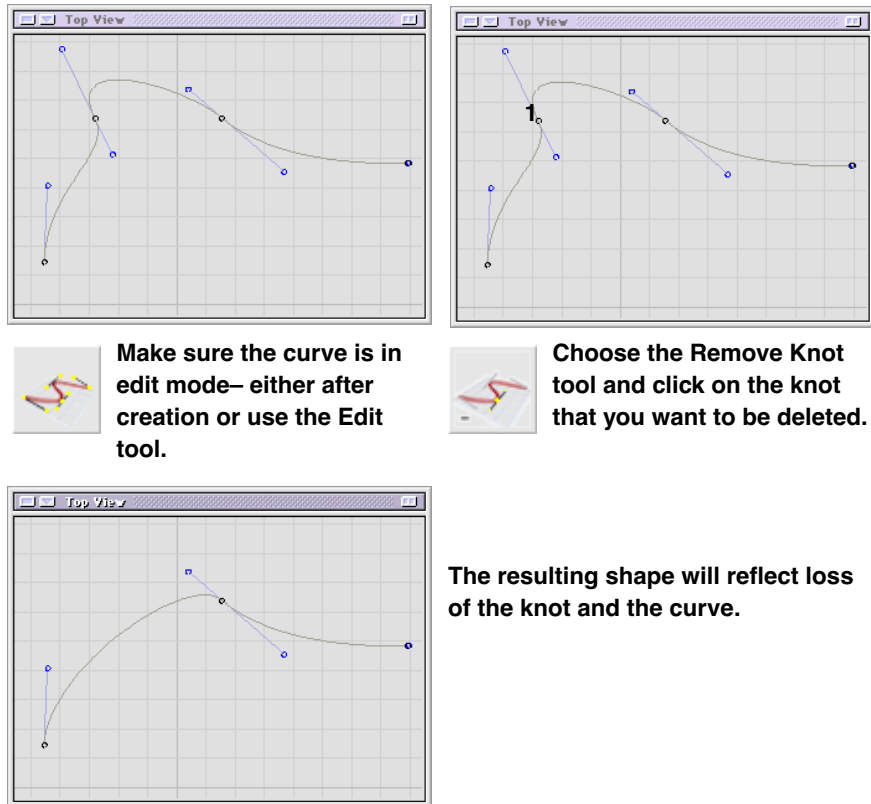


Figure 18.17 — Removing a knot from Bezier curve



## NURBS Curve Palette

### 19.0 Curve Editing

Curves (or wires) form the basis of more complex shapes. For example, a Coons surface is created from four boundary curves. A sweep surface can be created from a wire profile and a sweep path curve. It is important to have a good set of tools to create and manipulate these curves. We have already seen two sets of such tools, the Polyline palette and the Bezier palette. NURBS curve editing is another set of tools that allow manipulation of curves in a manner uniquely different from the first two.

#### What are NURBS Curves?

NURBS curves are single edge wire bodies. Unlike Bezier curves (where the shape of a curve is controlled by handles at each knot), NURBS curves are edited using Control Vertices (CV's) or knots that affect portions of the curve. Each CV can have a different weight which is controlled by holding down the “W” key and dragging to adjust the weight.

In Universe Modeler, you can edit a single curve or many curves. Before we edit any curves, let's take a quick look at the NURBS Curve Palette.

### 19.1 Curve Anatomy

Along with Bezier curves, Universe Modeler provides NURBS for curve shapes. NURBS is an acronym for “Non-Uniform Rational B-splines.”

In Universe Modeler, there are several parts to a curve that you should be aware of, some of which are more important than others. We will use these terms throughout the text, and the program will use them often in dialog boxes and alerts:

- Hull Edge
- Control Vertexes (CVs)
- Knots
- U Parameter
- Weights

#### Hull Edge

A Control hull is the polygonal shape that approximates the shape of the curve. A control hull consists of control vertices (CVs) and hull edges. Once the curve (or curves) is in edit mode, you can move the CVs and edges to change the shape of the curve.

#### CVs

CVs are the points of intersection of the control hull edges. Placement of the CVs determines the shape of the curve. The curve will touch the CVs only at the start and end of the curve. The CVs in between, while defining the general shape of the curve, will not be on the curve. In addition, you can also bias the shape of a curve by adjusting the weight of a CV or hull edge.

### Knots

Knots also affect the shape of the curve. Each curve has a knot vector which is an array of numbers specifying the range of the NURBS function.

A knot point is a point through which the curve will always pass. These knots can be relocated by click-dragging, and the curve will change shape so that it still passes through the knot.

### U Parameter

A curve can be thought of as a point traveling from the start to the end point (through various knots). You can see the direction of a curve by turning on the Show Edge Directions option in the Display tab of Edit/System Preferences. You can also tell where the curve starts by locating the letter *u* next to the second extreme end-point of a curve. One place where you would need to know this information is for resuming creation of a curve - it always resumes at the end of the curve.

### Weights

Weights are applied to a control vertex. As mentioned above, weights bias the shape of the curve.

Excess weight editing should be discouraged unless no other viable method of adjusting shape is successful. Keep the weight editing at a minimum.

Universe Modeler NURBS curves are edited in a curve edit mode. This method structure allows NURBS curves, ACIS solid objects, and Uber-NURBS to co-exist within the same environment.

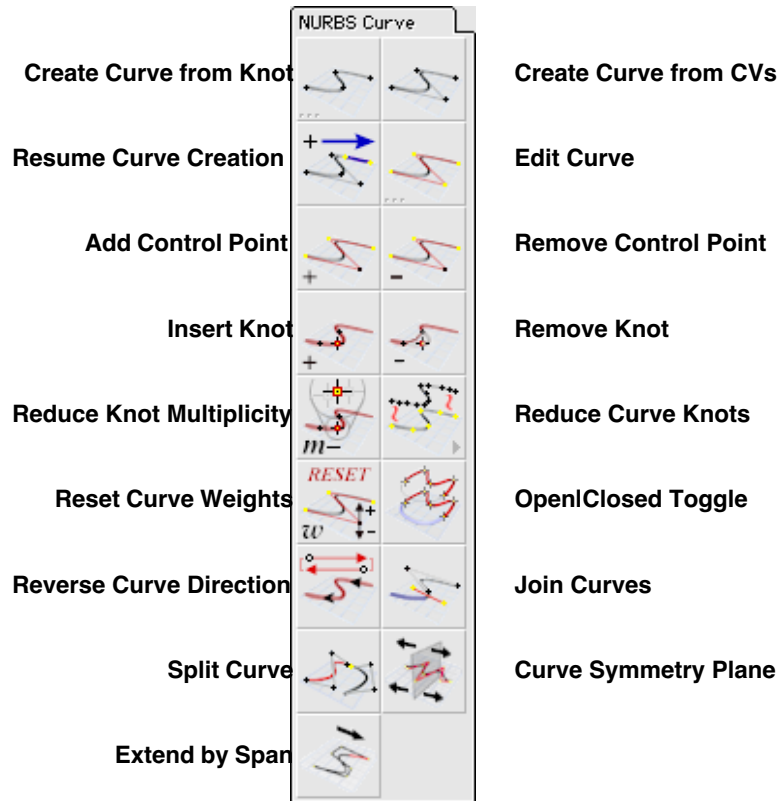


Figure 19.0 — NURBS Curve palette

Figure 19.0 shows the NURBS curve palette with each tool labeled.



### 19.2 Creating NURBS Curves

NURBS curves can be created in two ways. By setting knot points through which the curve will pass, or by setting CV's which defines the overall shape of the curve.

#### Create New NURBS Curve From Knots

To create a curve from knots, use the Create New NURBS From Knots tool. Click and release the mouse button to create the first knot, which acts as an anchor. Then click and drag to create subsequent knots, which can be repositioned by dragging the mouse around before releasing the mouse button. The curve will pass through each knot point placed. After releasing the mouse button, a new segment and its controlling CV's and hulls are formed. To end the creation of the curve, double-click when you set the last knot point, or if you want to create a closed curve, your final knot should be within the aperture range of the first point. This will automatically place the curve in edit mode. From here you can either edit the curve, or you can double-click to finalize the curve. The new NURBS curve is placed in the currently active layer.

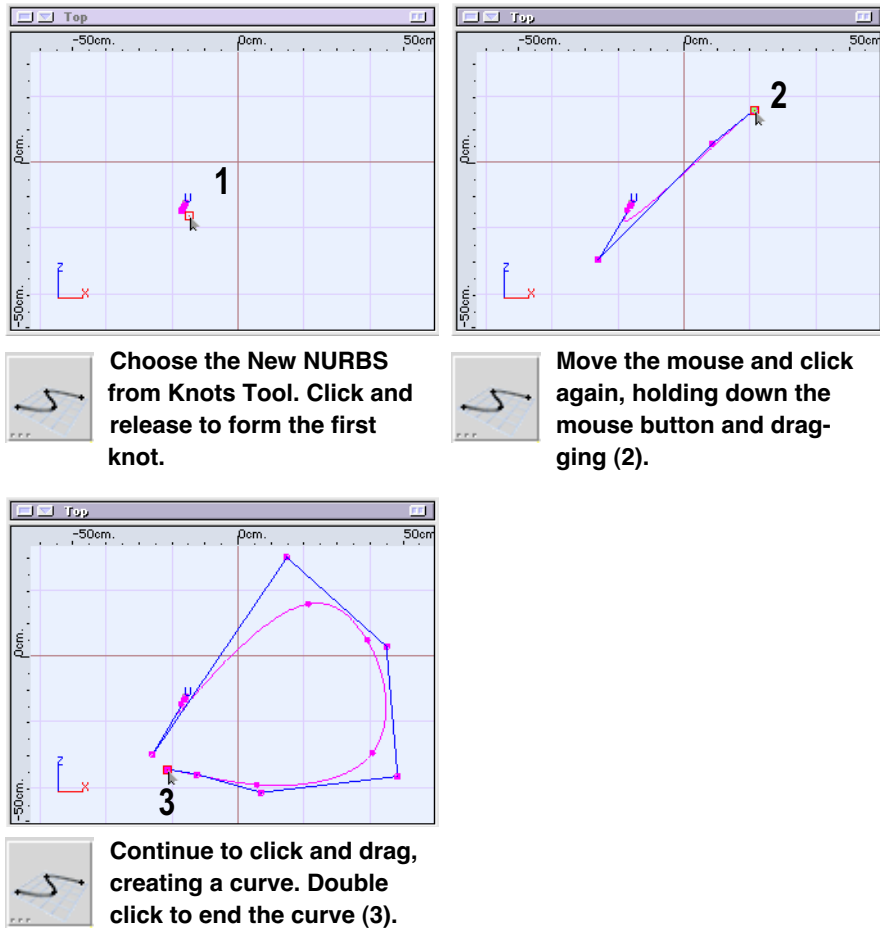


Figure 19.1 — Creating a new NURBS curve from knots

### Create a New NURBS Curve from CVs

If you know the approximate shape of the curve, but don't really need the curve to pass through any particular knot points, then you can use the Create New NURBS Curve From CVs tool.

Choose the tool and click the mouse down to place the first CV. Subsequent clicks will create connecting hulls. Even though you cannot reposition CVs as you are creating the curve, you can use Undo to remove the last CV that you placed. When you reach the final CV, double-click to place that point and finish the curve. If you want to create a closed the curve, simply place the last point on the first, and the curve will be closed with the continuity level set in the Continuity tab of the NURBS Curve Settings dialog. After creation you will be in edit mode, and you can make adjustments to the CVs, knots or hulls as you please till you are satisfied. Double-click to finish editing the curve.

If you leave the NURBS curve creation mode at any time by switching to another tool, the curve creation will end at the last knot or CV that you placed. You can always take the curve into edit mode and edit or resume creation.

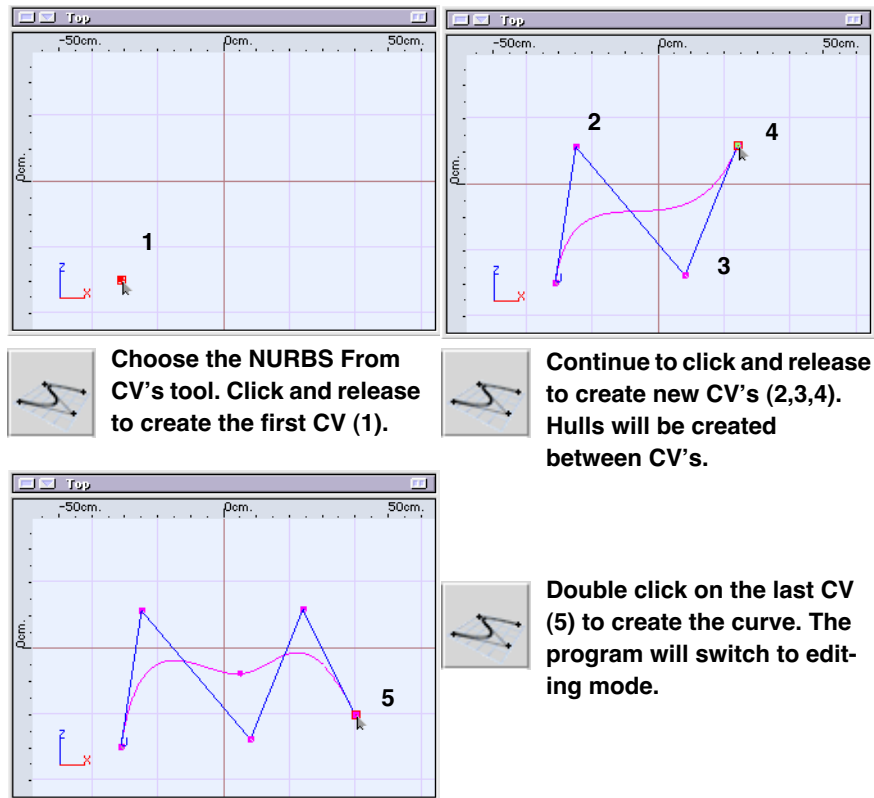


Figure 19.2 — Creating a NURBS curve from CV's

### Resume Creation

If after completing creation of a curve you decide that you want to add a few more knots or CVs to the curve, you can do that with this tool. Click on the Resume Creation tool. It will take all curves out of edit mode and prompt you to pick a NURBS curve to resume creation. The curve you pick will then be extended from the point that you left off.

This tool works with both the Create New NURBS Curve From Knots tool as well as the Create New NURBS Curve From CVs tool.

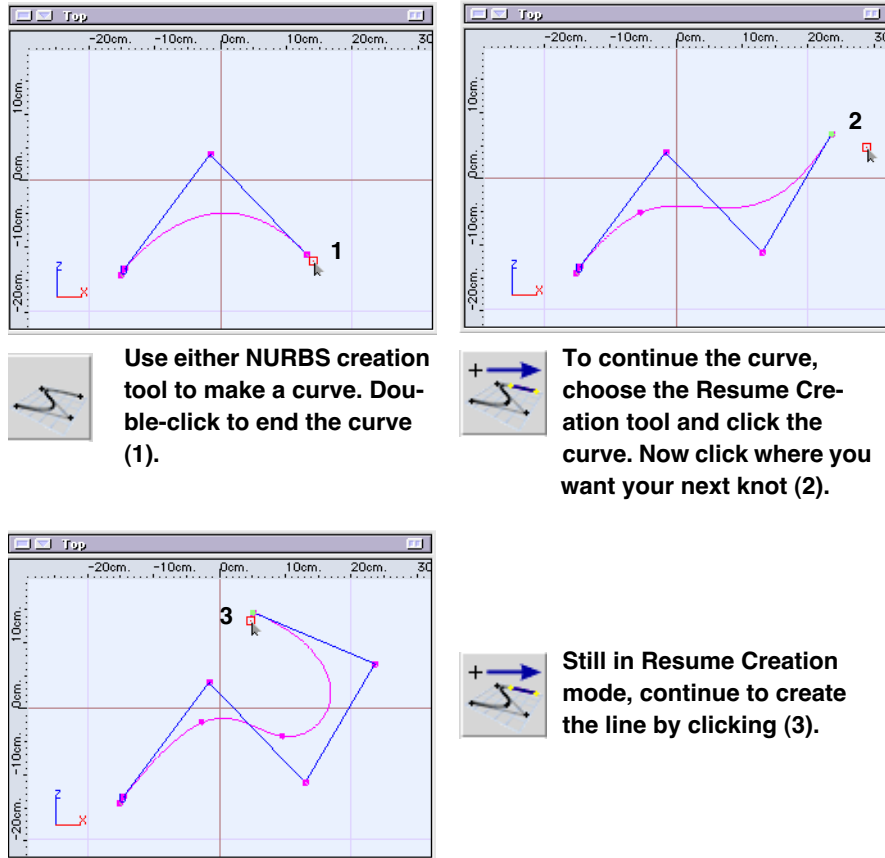


Figure 19.3 — Resuming Creation of a NURBS

### 19.3 Editing Curves

**A curve editing cycle consists generally of the following steps:**

1. Select the curve.
2. Choose the Edit Curve button in the NURBS Curve Palette.

The curve editing environment will now be activated, and the curve will appear with its control hull visible. It is actually the control hull that you will be editing.

3. If you wish to edit another curve along with the first curve you selected, click on the curve with the “n” key depressed.

You should now see both curves with control hulls displayed.

4. Click on a vertex and move it.
5. Double-click in an empty space to exit the NURBS editing environment, and return to normal Universe Modeler operation.

This is the typical curve editing cycle (in a very simplified form).

The NURBS curve editing mode allows you to control the placement of the CVs, knots, or hull edges of a NURBS curve. You can also edit the weights associated with each CV, or you can even change the parameterization of the curve, as discussed later in the rebuilding tools section.

You can reach Edit NURBS Curve tool in two ways: automatically, as soon as you complete creation of a curve, or by selecting the tool and then picking a curve to be edited. You can select more curves for editing by holding down the “n” key and clicking on another curve.

You can reposition CVs, knots or hull edges by selecting and dragging them around. All snaps are honored during creation and edit.

Editing the weights alters the amount of gravity that the CVs has to the curve. To edit the weight associated with a CV, hold down the “W” key and drag the mouse. Moving the mouse pointer to the left will decrease the weight (down to a value of 1), and moving it to the right will increase the weight value. By default the curves created are non-rational, they don’t have weights. If you edit the weights, then the curves are converted to rational curves. This have implications as noted in the rebuilding tools of the NURBS curves and surfaces chapters. You can convert a curve to non-rational after editing weights using the Reset Weights tool.

NURBS curves with all weights equal (or without any weights) are known simply as B-spline curves or NURBS.

You can put many NURBS curves into edit mode at the same time by pre-picking all of the curves with the EDGE filter on. However, NURBS curve editing will only edit single edge wires. Since Bezier curves are usually composed of multiple edges, if you select a Bezier curve for editing with NURBS curve edit, you will see only the picked edge put in edit mode, and will lose the other edges if you make any changes. In that case immediately double-click and first convert the Bezier curve to a single-spline and then use the Edit NURBS Curve tool.

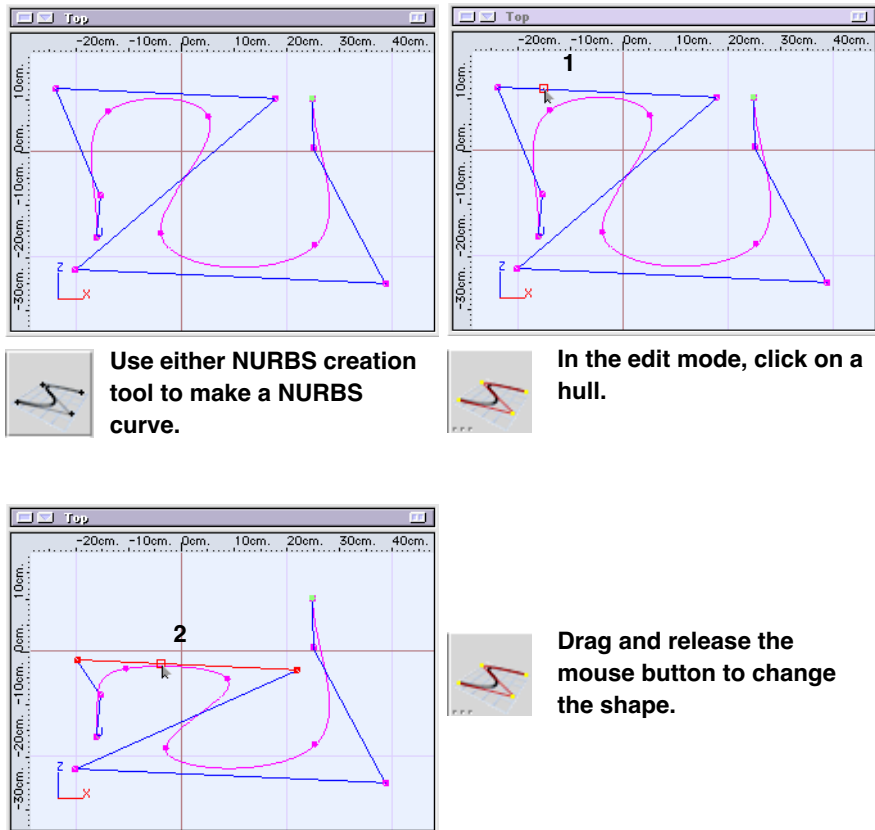
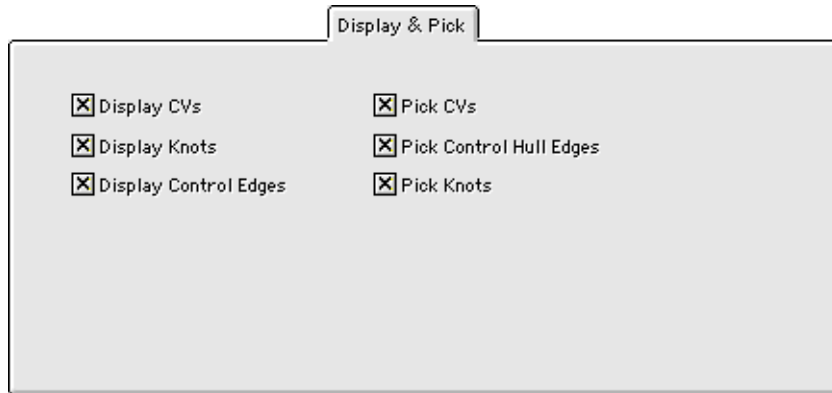


Figure 19.4 — Editing a NURBS Curve



### 19.4 Selecting Curve Entities

Editing a curve starts with the selection of the part of the curve that you wish to edit. Since selection plays a large role in editing, there are naturally several selection options and filters that can help you make curve editing easier. The following information refers to Figure 19.5.



*Figure 19.5 — Curve entity selection filters*

### Curve Entity Selection Filters

You can filter the selection of the following curve entities. Filtering typically means that you can choose to not select an entity type, or in some cases, choose only the desired type. All of these options are available in the NURBS Curve Settings dialog, Display & Pick tab. This dialog is presented when you double-click on the Edit NURBS Curve tool. You can choose to filter the following curve entities:

- CVs
- Knots
- Edges

You can also determine what entities are visible. This will help to eliminate clutter when your curves get more complex. You can enable or disable the following entities for display:

- CVs
- Knots
- Boundary Hull Edges

Entities can also be picked by selecting them directly, or dragging a rectangle around the desired entities. To deselect the entities, shift click on them or shift click in a null area of the screen.

### 19.5 What Can You Edit?

The following options are available as you select editable entities of a curve:

- Translation (x,y,z)
- Rotation (x,y,z)
- Scale (uniform, x,y,z)
- Weight editing per CV or hull edge

In addition to being able to edit the above, you can also edit multiple curves in a single session, split curves, rebuild curves and more.

#### Translate Entities

To translate selected CVs or hull edges, just drag. If you want to limit the translation to a particular axis, depress the desired axis key on the keyboard (x, y or z), and then drag.

For example, if you wish to move a selected CV in the Z direction, just select the CV, hold down the “z” key and drag.

### Rotate Entities

To Rotate selected entities, depress the “r” key. If you wish to limit the rotation to a particular axis, depress the desired axis key on the keyboard (x, y or z), and then drag.

### Scaling Entities

To scale a series of selected entities around their midpoint, depress the “s” key. If you wish to scale on a particular axis (non-uniform scale), just depress the desired axis key, and drag.

### Editing CV and Edge Weights

You can edit the weights of a CV, or a series of CVs, by first selecting the desired CVs, and depressing the “w” key while dragging. Remember that excessive weight editing of CVs on a curve should be avoided. If possible, try using other methods to influence shape, such as adding knots and CV editing, rather than editing weight values.

Now that we have covered what you can edit, let’s look at how to edit your curves. Curve editing is primarily working with CVs and knots, sculpting the curve into the shape that you require. You can push or pull them, rotate and scale them, and change their weights. However, there are times when you will need more (or less) CVs than you have to work with on your curve.

### Add Control Point

You can add a CV (also known as a Control Point) to the curve's control hull. This tool will change the shape of the curve and should be used while the curve's shape is still not finalized. This allows you more control in a particular area where you want to modify the shape further, but no CVs are available for editing.

To use this tool, the curve must be in Edit Mode. Choose the Add Control Point tool and click on a hull edge. A new CV will be placed halfway between the two CVs that define that hull edge. A corresponding knot will also become part of the curve.

Adding CVs or knots is the recommended way of adding detail to the curve rather than elevating the degree of the curve.

### Remove Control Point

You can reverse the effects of adding a CV by using the Remove Control Point tool. Select the tool when a curve is in edit mode, and click on a CV that you want to remove. That CV will be removed from the curve's definition and the two CVs on either side of that CV will be connected directly.

### Insert Knot

Knots can be added to curves in edit mode. Adding a knot will not change the curve's shape, but it will give you an additional knot and CV for manipulation.

To use the tool, the curve must be in edit mode. Choose the Insert Knot tool and click on the curve where you want the new knot to be added. The knot is placed there, and you will see a new control hull with one more CV.

By default Universe Modeler does not show knots on NURBS curves. If you want to see the knots, go to the Display tab under the Edit/System Preferences menu and toggle on “Show knots on curves” option.

Adding knots or CVs is the recommended way of adding detail to the curve rather than elevating the degree of the curve.

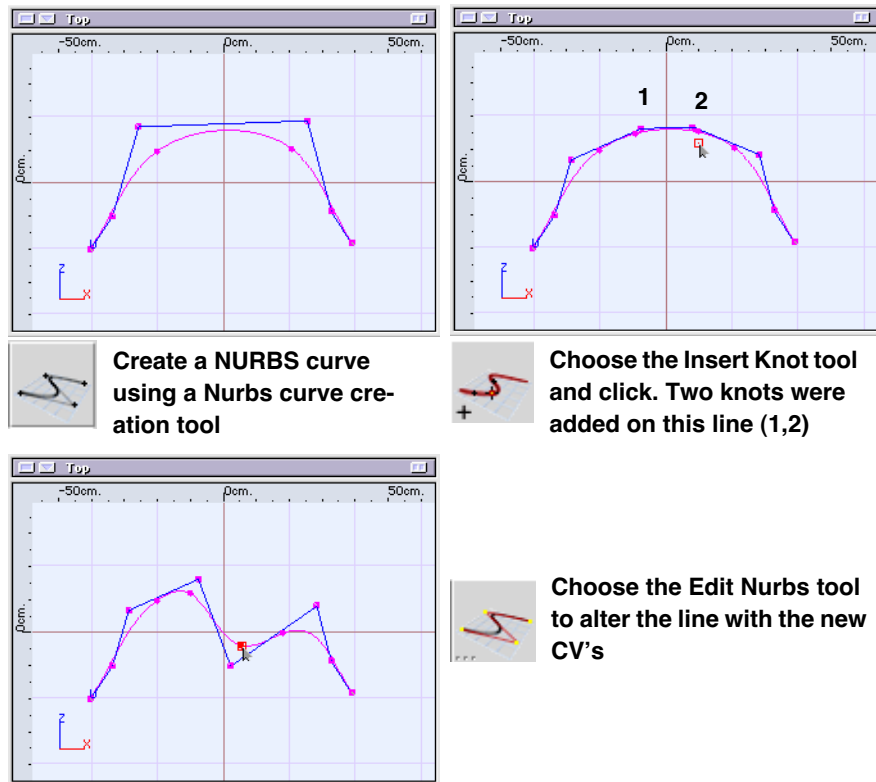


Figure 19.6 — Adding a knot with the Insert Knot tool

### Remove Knot

To reduce the number of knots on a curve that might not be needed, you can use this tool.

Reduce knots will, in general, change the shape of a curve, but within the user specified tolerance, set in the options dialog of the edit NURBS Curve tool, in the Rebuild tab.

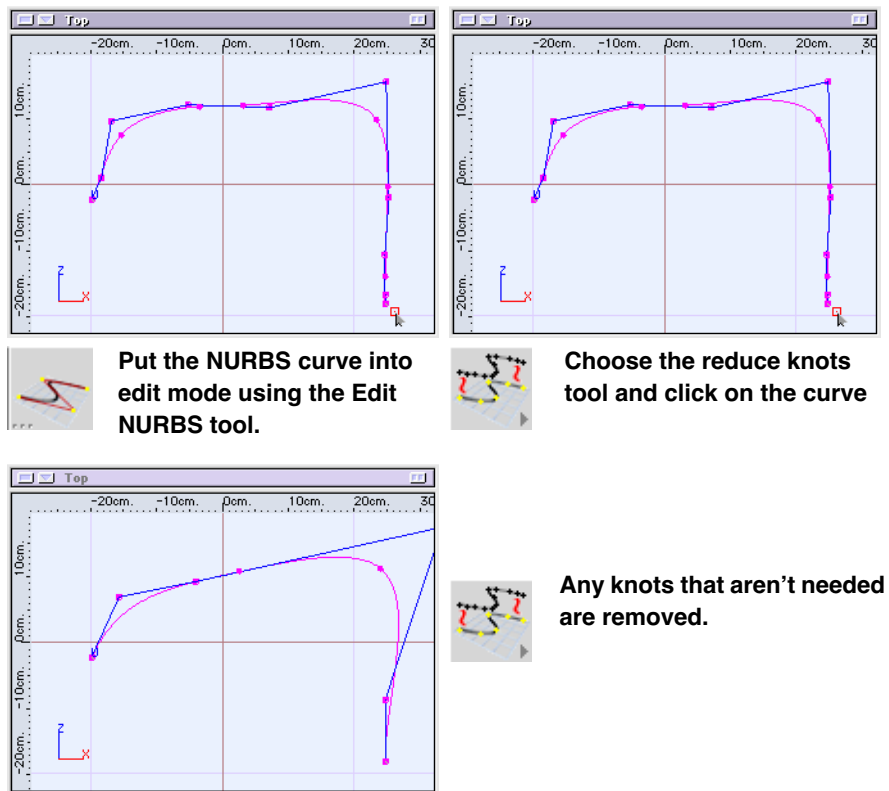


Figure 19.7 — Reducing the Knots on a curve

### Reduce Multiplicity of Knot

It is possible to have more than one knot at the same location on a NURBS curve. This is known as a “multiple knot” or a “knot with multiplicity more than one.” These knots create discontinuities and should be reduced. There are some uses for this, but more often than not, it might cause a problem.

In particular if your NURBS curve shows a knot in bright green, it is a multiple knot. This is always the case at both ends of the curve, which cannot be removed, and is not a problem. However, if you see a bright green knot anywhere in the interior of a curve, you can reduce the multiplicity using this tool. Sometimes this can be done without changing the shape of the curve.

## 19.6 Rebuilding Curves

Sometimes, when working with your curves, you will find yourself needing to simplify them, as they may become too complex to deal with in further modeling operations. You may also find yourself needing to make adjustments to your curves for a variety of reasons, especially if you understand the underlying mathematics upon which NURBS are based.

A separate tool sub-palette called “Rebuild NURBS Curve” has been created to help in these situations. In most cases, you should not need to venture into the sub palette, especially if you are not familiar with NURBS. These tools can be very destructive, and should only be used if absolutely necessary.

The reduce knots tool, on the other hand, should be used to reduce the complexity of a curve as much as needed to keep the CV and knot counts within reason.

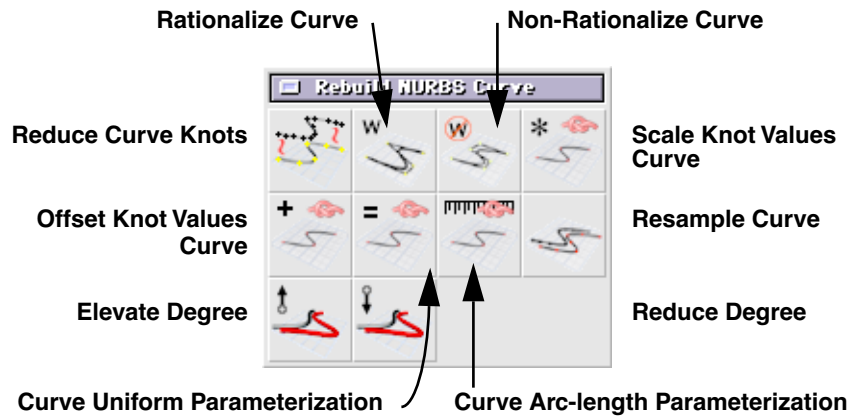


Figure 19.8 — Rebuild NURBS Curve tools

There are ten tools in this palette:

- Reduce Curve Knots
- Rationalize Curve and Non-Rationalize Curve
- Scale Knot Vectors and Offset Knot Vectors
- Curve Uniform Parameterization and Curve Arc Length Parameterization
- Resample Curve
- Elevate Degree and Reduce Degree

### Make Curves Rational and Non-Rational

These two tools control whether or not your curve will have weights that you can edit (rational) or not (non-rational). Overall, we prefer to discourage the practice of weight editing on a curve. This is due to a number of reasons, such as continuity, tessellation and ACIS compatibility.



### **To make a curve rational (add editable weights to a curve):**

1. Make sure the curve is already in editing mode (Rebuild Curve tools function on the last edited curve, the selected curve.)
2. Choose the Make Curve Rational tool from the Rebuild NURBS Curve tools sub-palette.

The make rational tool automatically jumps back to curve edit, as do all of the tools in the rebuild palette

The curve will now be rational, assigning the weight value found in the rebuild tab of the NURBS Curve Settings dialog (available when double-clicking on the Edit icon.) To verify this, make sure that display weights is active (Display & Pick tab of the NURBS Curve Settings dialog.) After “rationalizing” a Curve, weight values should appear for the selected CVs (if no CVs are selected, no weight values will appear.)

### **To make a curve non-rational (remove editable weights from a curve):**

1. Make sure the curve is already in editing mode (Rebuild Curve tools function on the last edited curve, the selected curve.)
2. Choose the Make Curve Non-Rational icon from the Rebuild NURBS Curve tools sub-palette.

The curve will now be made non-rational, removing any weights, which may have been applied to the curve. The curve, however, will be modified by inserting knots to keep the approximate shape of the curve the same prior to operation. To do this, the tool will use the value set in the Reduce Knots Tolerance edit box, found in the Rebuild tab of the NURBS Curve Settings dialog.

If you have a very small tolerance value, you will see that more CVs and knots have been added to maintain the shape of the curve. Larger values will yield less drastic changes to the knot count and the control hull of the curve, but may result in a change of shape from the original curve.

### Scale Knot Vectors of a Curve

You can scale each value in knot vectors of a curve by a specified value in the Rebuild tab of the NURBS Curve Settings dialog. Since this function will not change the locations of the knot values, the curve shape remains the same. However, the tangents at various points on the curve will differ from the original after the operation in their size. The direction of the tangents still will remain the same as before the operation.

You can set a new scale factor for the scale operation.

Scaling knot vectors tool comes in handy when one is dealing with multiple curves that will be involved in some sort of skinning operation. Often curves derived from different sources have very different knot vectors. These different knot vectors cause unpredictable twisting during the skinning process. One way to control that is by explicitly scaling the knot vectors of all the curves to start at the same value and be exactly the same or comparable.

#### To scale the knot vectors of a curve:

1. Make sure the curve is already in editing mode (Rebuild Curve tools function on the last edited curve, the selected curve.)
2. Choose the scale knot vectors icon from the Rebuild NURBS Curve tools sub-palette.

The knot vectors of the curve will now be scaled to the desired value. You can check the knot values by turning on the numeric display from the system preferences dialog box.

### Offset Knot Vectors of a Curve

You can offset each value in the knot vectors of a curve by a value specified in the rebuild tab of the NURBS Curve Settings dialog.

Again, this function is typically used when you have multiple curves that you want all to start from the same knot value. This is usually the first step in making the curves compatible. Curves that are more compatible give less twisting in skinning. Two curves, with the exact same knot vector, will skin with the skin surface having isoparms flowing from the knots of one to the other one-to-one. This makes the skinning process extremely predictable.

#### To offset the knot vectors of a curve:

1. Make sure the curve is already in editing mode (Rebuild Curve tools function on the last edited curve, the selected curve.)
2. Choose the Offset Knot Values Curve icon from the Rebuild NURBS Curve tools sub-palette.

The knot vectors of the curve will now be offset in value by the desired value. The knots do not actually physically move on the curve, just the numeric value is increased (or decreased) by the offset value. You can check the knot values by turning on the numeric display from the system preferences dialog box.

### Reduce Curve Knots

After working with them for a time, curve control hulls can get more and more complex to deal with. They can become very dense, making it difficult to select CVs, or perhaps they begin to develop folds or creases that are unwanted. One way to get curves back under control is to rebuild them, and that is what the Reduce Curves Knots tool is for. Get to know this tool, as it can become one of the best friends that you ever had.

The Reduce Curve Knots tool does its job by deleting all removable knots (which results in a reductions of knots and their associated CVs) from a NURBS Curve to a tolerance you specify in the Reduce/Rebuild tolerance field in the Rebuild tab of the NURBS Curve tools dialog.

#### **To reduce the knots on a Curve with the reduce knot tool:**

1. Make sure The curve is already in editing mode (Rebuild Curve tools function on the last edited curve, the selected curve.)
2. Choose the Reduce Curve Knots knot icon from the Rebuild NURBS Curve tools sub-palette.

The curve will now be rebuilt to the tolerance that you specified. The default value may not be suitable for your needs, so feel free to experiment. Smaller tolerance values will provide less simplified curves, which more accurately represent the curve you started with. Larger tolerance values will less accurately represent the curve you started with but will be better simplified. Depending upon your needs, that result may be desirable. In the case where you need to change values and try again, always undo your earlier attempt, and try a new tolerance value as the results will be more consistent that way.

### Resample Curve

While it may be warranted to rebuild a curve, sometimes even that approach may not yield the result that you require. For those times when you wish an even simpler resulting curve, the best option may be to fit an entirely new curve to a previously existing one.

There are many reasons for this:

- Simplify knot values, CV and knot count
- Smooth out curve irregularities, and so on
- You want a curve with an exact number of knots as another curve.

If you have curves that have unequal number of knots, and you want to make them have the same number of knots, this tool comes in very handy.

The Resample Curve tool accomplishes this by fitting a completely new curve to a set of data points which have been sampled from a selected curve. The number of data points to be sampled is specified in the Rebuild tab in the NURBS Curve Settings dialog in the Fit Curve group. The output will be a smooth curve through those data points.

For closed curves, you can also specify which point sample taken from the original curve shall be used as the first knot of the resulting curve. This enables approximate aligning of the first points of two closed curves, before skinning.

#### **To fit a new curve to another one:**

1. Make sure the curve is already in editing mode (Rebuild Curve tools function on the last edited curve, the selected curve.)
2. Choose the Resample Curve icon from the Rebuild NURBS Curve tools sub-palette.

A new curve will be fitted to the old curve, and the old curve will be deleted. Depending upon your attribute settings, the newly fit curve may not seem to match the original curve all that well. In that case, you might need to increase the sample count after undoing the previous attempt for a better result. More samples you take, the less the new curve will deviate from the original.

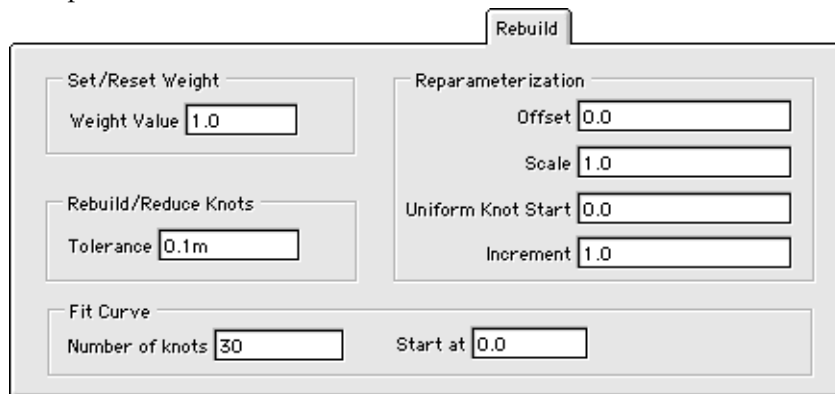
### Curve Parameterization — Uniform and Arc Length

Every point on a NURBS curve has a unique parameter value. The NURBS curve is nothing but a collection of these points. No two points on the NURBS curve have the same parameter value. NURBS are one of the most powerful parametric functions that are used in the modern day computer graphics. Given any parameter value one can quickly evaluate the position of on the curve. The relationship between the parameter values of successive points on a curve is described as its parameterization. Rather than looking at the infinite points on the curves and their parameter values, for the purposes of analysis, it is sufficient to look at the parameter values at the knots. When the length of the segments between two successive knots is proportional to the parameter values at the knots, the curve is said to be arc-length parameterized. When the parameter values are in constant increments at successive knots, irrespective of the length of segments between them, the curve is said to be uniformly parameterized. The process of changing the parameter values of the knots is known as reparameterization.

### To apply uniform parameterization to a curve:

1. Make sure the curve is already in editing mode (Rebuild Curve tools function on the last edited curve, the selected curve.)
2. In the rebuild tab of the NURBS Curve Settings dialog, you can set a start value other than 0 if you wish
3. In the same tab, you can also set an increment value other than 1
4. Choose the uniform parameterization icon from the Rebuild NURBS Curve tools sub-palette.

The curve will now be uniformly parameterized, per your settings in the rebuild tab of the NURBS Curve Settings dialog. You may see changes in the shape of the curve.



*Figure 19.9 — Rebuild options*

### To apply arc-length parameterization to a curve:

1. Make sure the curve is already in editing mode (Rebuild Curve tools function on the last edited curve, the selected curve.)
2. Choose the arc-length parameterization icon from the Rebuild NURBS Curve tools sub-palette.

The curve will now be arc-length parameterized. You may see changes in the shape of the curve.

### **Elevating or Reducing the Degree of a Curve**

In Universe Modeler, curves that you create with the variety of tools available can also have varying degrees. Degree three curves are preferred, as they compute quickly, and can be used to create virtually any shape you will require.

Depending upon the situation, you could find need to adjust the degree of a curve.

#### **To raise the degree of a curve:**

1. Make sure the curve is already in editing mode (Rebuild Curve tools function on the last edited curve, the selected curve.)
2. Choose the Elevate Degree tool from the Rebuild NURBS Curve tools sub-palette

The curve's degree will now be raised by one. The new degree will be displayed in the status window.

#### **To lower the degree of a curve:**

1. Make sure the curve is already in editing mode
2. Choose the Reduce Degree tool from the Rebuild NURBS Curve tools sub-palette

The curve's degree will now be lowered by one.

Degree reduction is not always possible without a potentially drastic change in the shape of the curve. If this is the case, Universe Modeler will display a dialog ahead of time, giving you the option to cancel.



If your goal is to have less knots on a curve, then the reduce degree tool is not the best choice. Instead, use the Remove Knot or the Remove Control Point tool.

### 19.7 More Curve Editing Features

In addition to the previously discussed curve editing features, there are several more features which can make working with curves easier in Universe Modeler:

- Reset NURBS Curve Weights
- Open Close Toggle
- Reverse NURBS Curve
- Join NURBS Curve
- Split NURBS Curve
- Curve Symmetry Plane
- Extend by Span

#### Reset NURBS Curve Weights

When a curve's CVs weights have to be set to a specific value, this tool is the one to use. Typically, you may have intentionally or accidentally tweaked the weights of the CVs of a curve, and you want to reset them. Or, the weights may have become fractional due to interactive editing but you like to see round numbers. This tool is useful under those circumstances. The curve must be a rational NURBS curve, i.e. the CVs must have weights, for this tool to be applicable. Resetting weights may change the shape of the curve, sometimes quite drastically.

**To reset the weight of all CVs of a curve to a specified value:**

1. Make sure the curve is already in editing mode and is rational.
2. Make sure no CV is selected.
3. Choose the Reset NURBS Curve Weights icon from the NURBS Curve palette.

The weights of all the CVs of the curve will be reset to the value set in the Set/Reset Weight field in the Rebuild tab of the NURBS Curve Settings Dialog.

**To reset the weight of a few CVs of a curve to a specified value:**

1. Make sure the curve is already in editing mode and is rational.
2. Select all the CVs for which you want to reset the weights.
3. Choose the Reset NURBS Curve Weights icon from the NURBS Curve palette.

The weights of all the selected CVs of the curve will be reset to the value set in the Set/Reset Weight field in the Rebuild tab of the NURBS Curve Settings Dialog.

### Open Close Toggle

If you need to open a closed curve, or close an open one, you will need to use this tool.

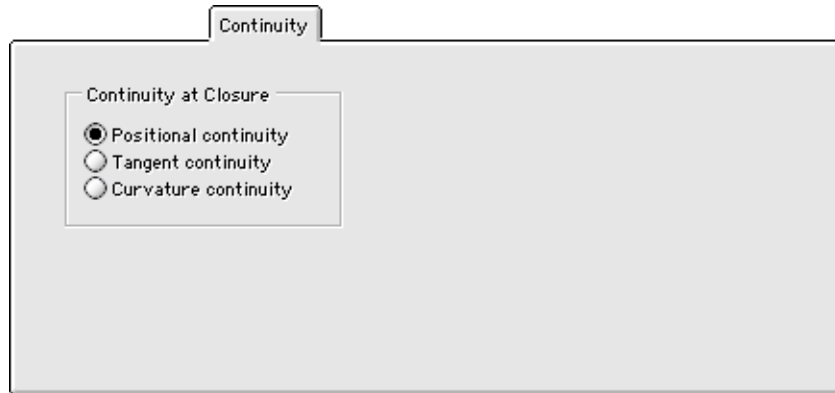
The closure can be made positional, tangent or curvature continuous.

**To close a curve with position continuity:**

1. Make sure the curve is in the edit mode and is open.
2. In the Continuity tab of the NURBS curve settings dialog, set the continuity level to be Positional.

3. Click on the curve that you want to close.

At this point, the open curve will be closed. The closure point will be sharp cusp.



*Figure 19.10 — Continuity options*

**To close a curve with tangent continuity:**

1. Make sure the curve is in the edit mode and is open.
2. In the Continuity tab of the NURBS curve settings dialog, set the continuity level to be Tangent.
3. Click on the curve that you want to close.

At this point, the open curve will be closed. The closure point will be smooth, but could still appear sharp at a coarser scale.

**To close a curve with curvature continuity:**

1. Make sure the curve is in the edit mode and is open.
2. In the Continuity tab of the NURBS curve settings dialog, set the continuity level to be Curvature.
3. Click on the curve that you want to close.

At this point, the open curve will be closed. The closure point will be smoothest.

Curvature closed curves provide better success with rounding of extruded sections.

### **To open a closed curve:**

1. Make sure the curve is in the edit mode and is closed.
2. Click on the curve that you want to open.

At this point, the closed curve will be opened.

## **Reverse NURBS Curve**

The curve has a start and an end and hence a direction. The letter “U” is drawn at the beginning of the curve. The parameter values of the curve always increase from the start to the end. Often times you build shapes without regard to the direction in which you build them. Later on in the modeling process, say trying to do a skin or loft across these curves, you may discover that the curves are in mutually incompatible directions.

Reverse NURBS curve tool is useful to completely reverse a curve, without affecting the shape you built.

### **To reverse a NURBS curve:**

1. Make sure the curve is in the edit mode,
2. Click on the Reverse NURBS curve direction icon from the NURBS curve palette.

This will reverse the curve, and the letter “U” will be at the opposite end of the curve. Closed curves may also be reversed, in that case, the winding of the curve changes.

The Reverse Wire tool in the Wire Editing palette also does reversal, but it does a more “soft” reversal than this tool. By “soft” reversal, we imply that the actual parameter values are not reversed, just a “flag” is flipped. The Reverse Wire tool also works on multi-edge wires, as opposed to the Reverse NURBS curve tool, which works on single edge NURBS curves wires.

### Join NURBS Curves

The Join NURBS Curves tool will join two curves together to form a single curve. Typically you use this to reduce the number of edges in a wire body, so that when that wire body is used in operations such as extrude or revolve, you get fewer faces in the body. Joining two curves always makes the join smoother, so if you want to preserve the sharp corners, you should be using the boolean union tool from the mode.

#### To join two curves together:

1. Make sure both the curves are already in editing mode (use the “n” key to select each additional curve.)
2. Choose the Join NURBS Curves tool from the NURBS Curve palette
3. Click near the end of the first curve.
4. Click on the start of the second curve.

The two curves will now be joined into one continuous curve.

### Split NURBS Curve

The split curve tool will split the selected curve at a point picked on the curve. It can therefore be used to giving two pieces by splitting an open curve, or used to open a closed curve. Splitting a curve does not change the shapes. A curve may be split anywhere, and not necessarily at the knots.

### To split a curve

1. Make sure the curve is already in editing mode.
2. Choose the Split NURBS Curve icon from the NURBS Curve palette
3. Click on the curve at the point where you want the curve to be split.

The curve will now be split into two (or more if you continued the operation.)

### Curve Symmetry Plane

Symmetry is often the holy grail of curve and shape design. It is often easier to create and edit one half of a curve and then have the other half reflect the changes than it is to attempt to build the whole thing by hand. The curve symmetry plane helps to make this easy.

The symmetry plane is temporary; it does not stay established during the editing session, it merely reflects the CV positions and weights from one side of the symmetry line to the other.

**Note** *This is very different from how symmetry works with the UberNURBS tool.*

### To apply symmetry to one side of a curve from the other:

1. Make sure the curve is already in editing mode.
2. Choose the Curve Symmetry Plane tool from the NURBS Curve palette.
3. Choose which view you wish to apply the symmetry.
4. Drag the symmetry line through the curve at the desired location.

The CVs from the middle CV of the curve to the end CV will be reflected about the line (plane) to be symmetrical with the CVs from the beginning to the middle. If the number of CVs is odd, the middle CV is left unchanged.

The knot parameterization is not changed from the original, so, if the knot vector was asymmetric, the curve will not be symmetric. Use the uniform parameterization tool to make the knot vector uniform and then apply symmetry to have a perfectly symmetric curve in all respects.

The snapping grid helps to minimize input error for the plane. It is easy to run the plane through a CV if you are not careful, and this can lead to an undesirable result, we recommend to use the grid snap for this feature.

Feel free to experiment with different plane orientations, as some rather unusual and unexpected (and interesting!) results can be achieved with it.

### Extend by Span

Extend Curve by Span offers the ability to extend the existing curve from the beginning or the end with continuity tangent, curvature or a maximum possible continuity. Any editable curves that are not closed in the direction of extension are candidates for extension. The extension is achieved by simply clicking on the curve near the end you wish to extend. The following options are available:

- Tangent Continuity
- Curvature Continuity

**Tangent Continuity.** When this option is chosen, the extension will be tangent continuous at the point where the original curve ends and the extension begins. There will be multiple knots at the end of the original Curve.

**Curvature Continuity.** To get a curvature continuous (smoother than tangent) extension, use this option. The curve is extended by reflection. Again, there will be multiple knots at the end of the original curve.

### **To extend a curve with Tangent Continuity:**

1. In the continuity tab of the NURBS Curve settings dialog, set the continuity to be Tangent.
2. Make sure the curve you are trying to extend is in the edit mode.
3. Click near the end of the curve.

This would create an extension that is tangent continuous.

### **To extend a curve with Curvature Continuity:**

1. In the continuity tab of the NURBS Curve settings dialog, set the continuity to be Curvature.
2. Make sure the curve you are trying to extend is in the edit mode.
3. Click near the end of the curve.

This would create an extension that is curvature continuous.



## Surface Editing

Universe Modeler allows you to create shapes with solids, UberNURBS or NURBS surfaces. You can use any method that you choose to create a particular shape, and each method can work with each other in Universe Modeler. This section will cover surface creation and editing.

### 20.0 What are Surfaces?

Surfaces are mathematical descriptions of shapes. Surfaces can be combined with other surfaces to create complex objects, such as faces or parts of machinery. Universe Modeler uses NURBS to create and edit surfaces. NURBS are an industry standard method for describing surfaces. In fact, any surface or solid model that you create in Universe Modeler can be exported to other modeling or CAD programs via the IGES file format.

### When Should You Use Surfaces?

When to use surfaces is really a matter of choice. However, here are a few typical guidelines to follow:

- Free form shapes that solids cannot easily duplicate
- Organic shapes
- Aerodynamic shapes
- Anywhere you anticipate significant shape editing

### How To Edit Surfaces

In Universe Modeler, you can edit a single surface or many surfaces — the choice is up to you. Before we edit any surfaces, let's take a quick look at the NURBS Surface Palette.

NURBS Surface Palette

The NURBS surface palette is where all of the NURBS surface tools are located. The illustration below names all of the tools for you. We will define the functions of these tools as we move along.

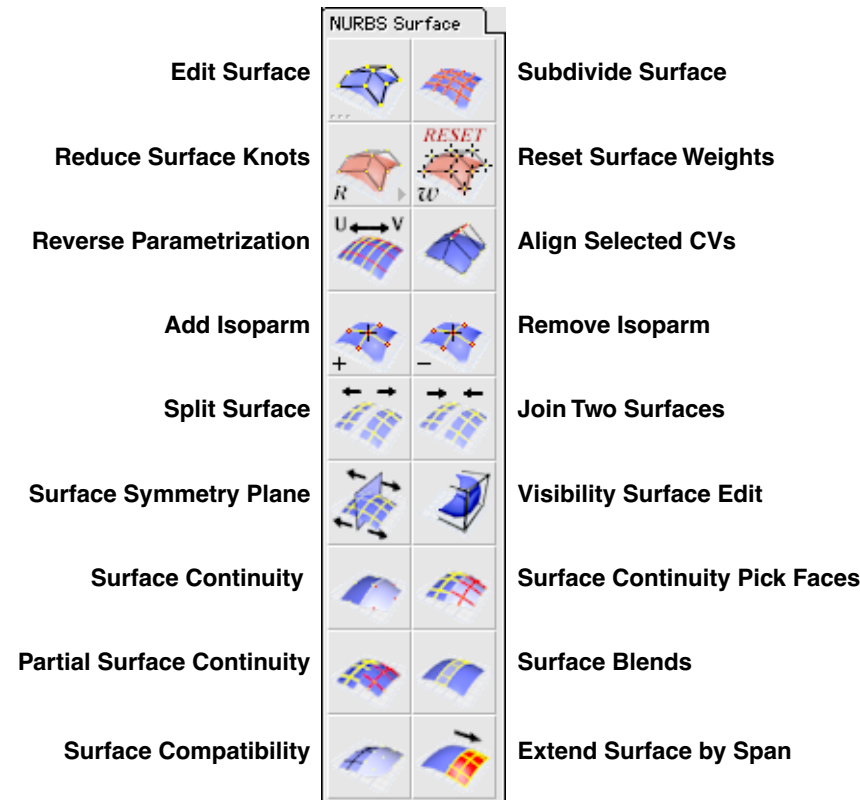


Figure 20.0 — NURBS Surface Palette

### 20.1 Editing Surfaces

You can edit surfaces that you created with the 3D tools, or you can edit the faces of an ACIS object as surfaces. The only restriction is that surfaces with trims, such as those that have been knifed or had boolean operations applied, cannot be edited directly. Instead, the underlying surface (before the trim was applied) will be used. Therefore, it is best to apply knives and booleans after surface editing has been completed.

Universe Modeler NURBS surfaces are edited in a surface edit mode. This is similar to UberNURBS, which also is edited in a separate editing mode. This method structure allows NURBS surfaces, ACIS solid objects, and UberNURBS to co-exist within the same environment.

To illustrate this, the following example will take you through a complete (albeit brief) surface editing cycle.

#### **To Edit a Surface Object:**

1. Select the object
2. Choose the Edit Surface button in the NURBS Surface Palette

*The surface editing environment will now be activated, and the surface will appear with its control hull visible. It is actually the control hull that you will be editing.*

3. If you wish to edit another surface along with the first surface you selected, click on the surface with the “n” key depressed.

*You should now see both surfaces with control hulls displayed.*

4. Click on a vertex and move it
5. Double-click in an empty space to exit the NURBS editing environment, and return to normal Universe Modeler operation.

This is the typical surface editing cycle (in a very simplified form).

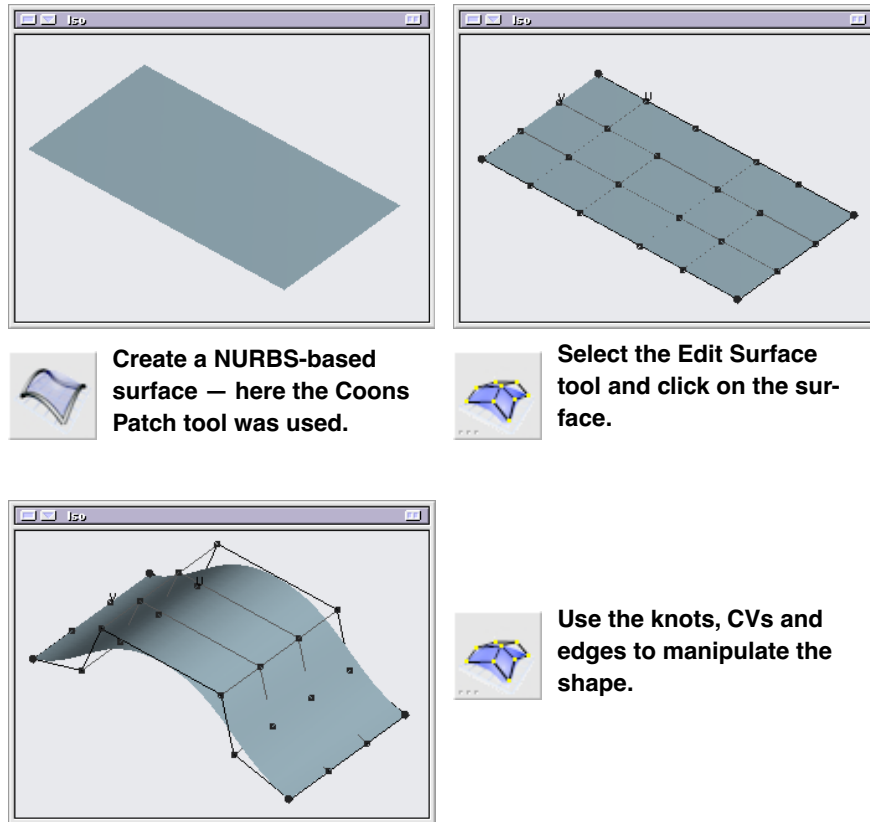


Figure 20.1 — Editing a NURBS surface

## 20.2 Surface Anatomy

Universe Modeler uses NURBS for surface editing. NURBS is an acronym for “Non-Uniform Rational B-splines.”

In Universe Modeler, there are several parts to a surface that you should be aware of, some of which are more important than others. We will use these

terms throughout the text, and the program will use them often in dialog boxes and alerts:

- Control Hulls
- Control Vertices (CVs)
- Weights
- Knots
- U and V Parameters
- Isoparms

### Control Hulls

Control hulls are “nets” that control the shape of the surface. A control hull consists of control vertices (CVs) and edges (hulls). The control hull roughly encloses the shape of the surface itself.

Once the surface (or surfaces) is in edit mode, you can move the CVs and edges to change the shape of the surface.

### CVs

CVs are the points of intersection of the control hulls. Placement of the CVs determines the shape of the surface. In addition, you can also bias the shape of a surface by adjusting the weight of a CV or hull edge.

### Weights

Weights are applied to a control vertex. As mentioned above, weights bias the shape of the surface.

*Excess weight editing should be discouraged unless no other viable method of adjusting shape is successful. This is due to the unpredictability of the affect of*

*excess surface weights on the surface. Keep the weight editing per surface at a minimum.*

### Knots

Like weights, knots also affect the shape of the surface. Each surface has two knot vectors (one each for u and v), which is an array of numbers specifying the range of the NURBS function. Intersecting isoparms indicate knot locations on the surface.

*Knots are a necessary evil in surface editing. In fact, the effects of knot editing are not apparent, so Universe Modeler does not allow you to edit knots directly. Instead, necessary knot manipulation is confined to Add and Remove Isoparm tools and the Rebuild Surface tools sub-palette.*

### U and V Parameters

Each surface has a u and v direction associated with it. You can easily see these directions as a network of intersecting lines (called “isoparms”, which represent the locations of knots) across the surface. You can tell which direction is which by locating the little u or v next to the second isoparm in from the surface edge. You only have to worry about u and v when dealing with surface continuity operations.

### Isoparms

As mentioned previously, isoparms are lines which are drawn within the interior of a surface. More than just decoration, these lines indicate much about a surface. When in edit mode, the isoparm spacing can tell you how a surface will be tessellated, how it is parameterized, and so on, just by a quick glance. You can even add or remove isoparms as you edit.

*Universe Modeler draws isoparms in edit mode in their exact location based on knot value. Once in the normal Universe Modeler (ACIS) environment, however, isoparm location is no longer based on the knot values of the surface, but rather just an even division of the surface based on the value that you specify in the Tessellation tab of the Document Preferences dialog.*

### 20.3 Selecting Surface Entities

Editing a surface starts with the selection of the part of the surface that you wish to edit. Since selection plays a large role in editing, there are naturally several selection options and filters that can help you make surface editing easier. The following information refers to Figure 20.2.



**Figure 20.2 — NURBS surface settings dialog, display & pick tab**

### Surface Entity Selection Filters

You can filter the selection of the following surface entities. Filtering typically means that you can choose to not select an entity type, or in some cases, choose only the desired type. All of these options are available in the

NURBS Surface Settings dialog, Display & Pick tab. This dialog is presented when you double-click on the Edit Surfaces tool.

You can choose to filter the following surface entities:

- CVs
- Edges

You can also determine what entities are visible. This will help to eliminate clutter when your surfaces get more complex. You can enable or disable the following entities for display:

- CVs
- Isoparms (“i” click toggles isoparm display state)
- Boundary Hull Edges
- Weights (displayed only when CVs are selected)
- Surfaces (tab key toggles surface display state)

You can also control the resolution of the shaded surfaces that you are editing separately from the object or global tessellation settings. The value in the surface tessellation edit box will apply a simple uniform tessellation of the shaded surface while in surface edit mode. In the illustration, the surface tessellation value is set to 20, meaning that the surface will be shaded with 400 quads (800 total polygons, as they are triangulated before shading).

In addition to the entity selection filters, you can also select a complete u or v row or column just by depressing the u or v key when clicking on a CV. This is especially helpful when you want to scale a cross section of a revolved surface, for example. Of course, entities can also be picked by selecting them directly, or dragging a rectangle around the desired entities. To deselect the entities, shift click on them or shift click in a null area of the screen.



### 20.4 What Can You Edit?

The following options are available as you select editable entities of a surface:

- Translation (x,y,z)
- Rotation (x,y,z)
- Scale (uniform, x,y,z)
- Weight editing per CV or hull edge

In addition to being able to edit the above, you can also edit multiple surfaces in a single session, join surfaces together with three forms of continuity, split surfaces, rebuild surfaces and more.

#### Translate Entities

To translate selected CVs or hull edges, just drag. If you want to limit the translation to a particular axis, depress the desired axis key on the keyboard (x, y or z), and then drag.

For example, if you wish to move a selected row of CVs in the Z direction, just select the isoparm, hold down the “z” key and drag.

#### Rotate Entities

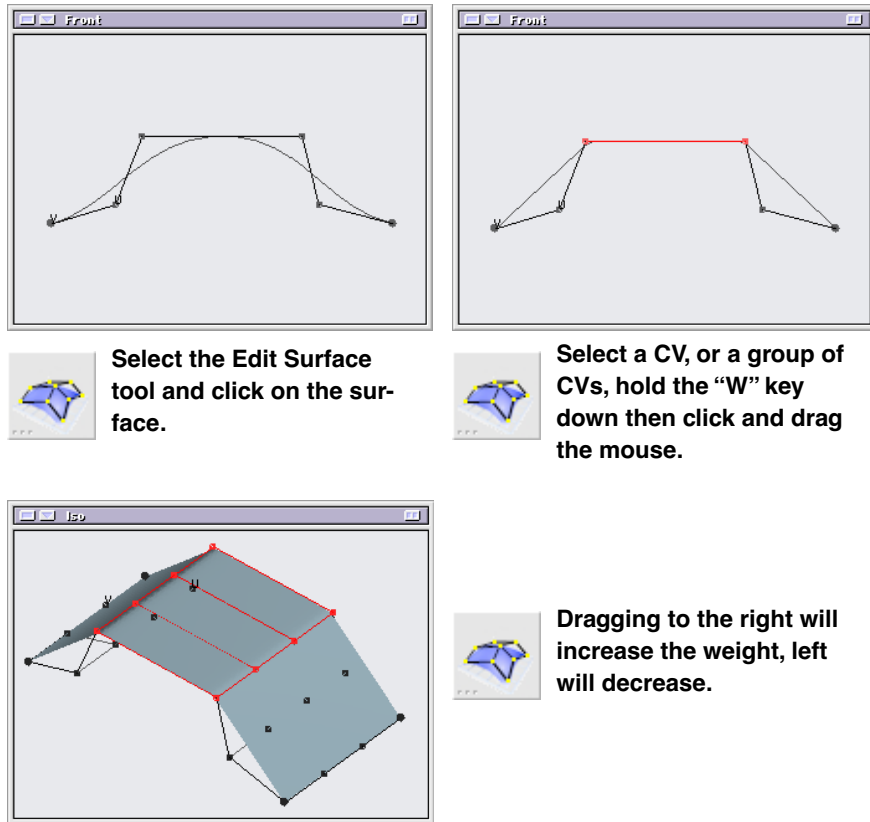
To Rotate selected entities, depress the “r” key. If you wish to limit the rotation to a particular axis, depress the desired axis key on the keyboard (x, y or z), and then drag.

### Scaling Entities

To scale a series of selected entities around their midpoint, depress the “s” key. If you wish to scale on a particular axis (non-uniform scale), just depress the desired axis key, and drag.

### Editing CV and Edge Weights

You can edit the weights of a CV, or a series of CVs, by first selecting the desired CVs, and depressing the “w” key while dragging. Remember that excessive weight editing of CVs on a surface should be avoided. If possible, try using other methods to influence shape, such as adding isoparms and CV editing, rather than editing weight values.



**Figure 20.3 — Changing weights on a NURBS surface**

As you work with weights, you may choose to reset or force a value onto the weights of a surface. This is done by first setting the desired weight value in the reset weights edit field (NURBS Surface Settings dialog, Rebuild tab, see Figure 20.5) and then choosing the Reset NURBS Surface Weights tool in the NURBS Surface tool Palette.

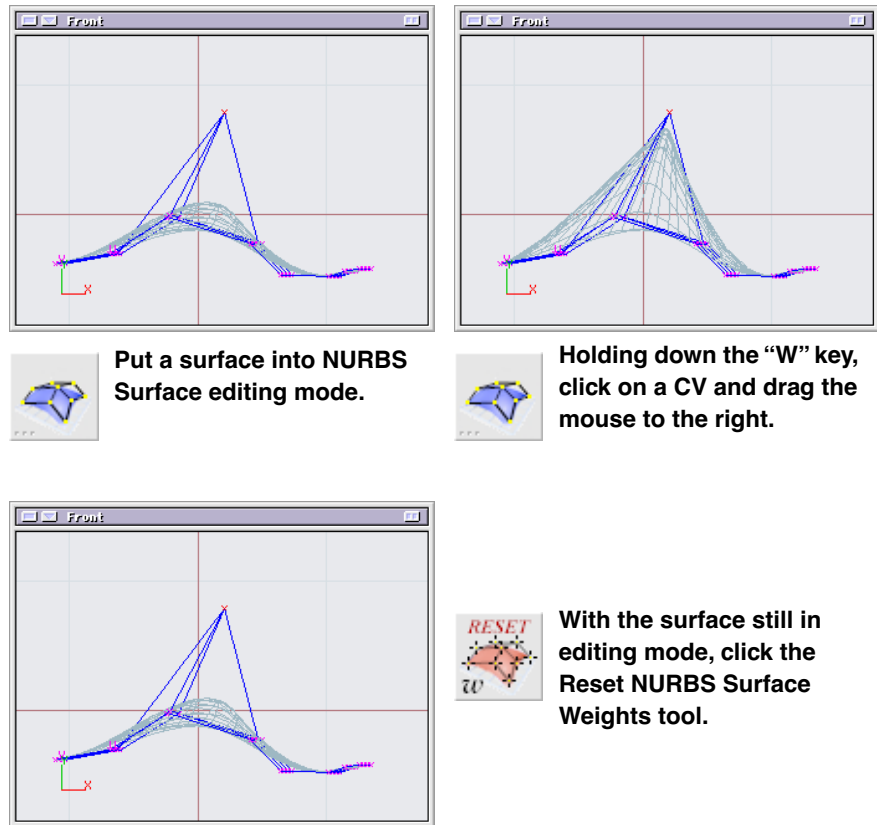
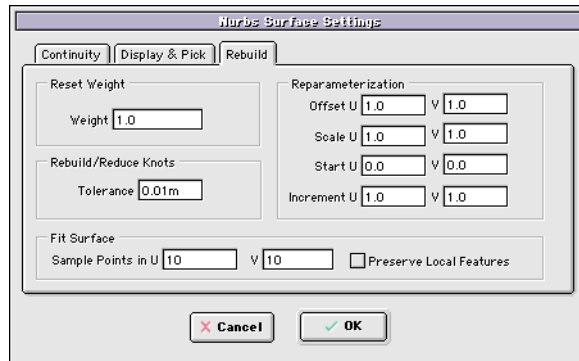


Figure 20.4 — Resetting weights on a NURBS surface

This process is illustrated in Figure 20.4. Remember, this action will reset the weight values for the entire surface, not just the selected CVs.



*Figure 20.5 — NURBS surface settings dialog, rebuild tab*

Now that we have covered what you can edit, let's look at how to edit your surfaces. Surface editing is primarily working with CVs, sculpting the surface into the shape that you require. You can push or pull them, rotate and scale them, and change their weights. However, there are times when you will need more (or less) CVs than you have to work with on your surface.

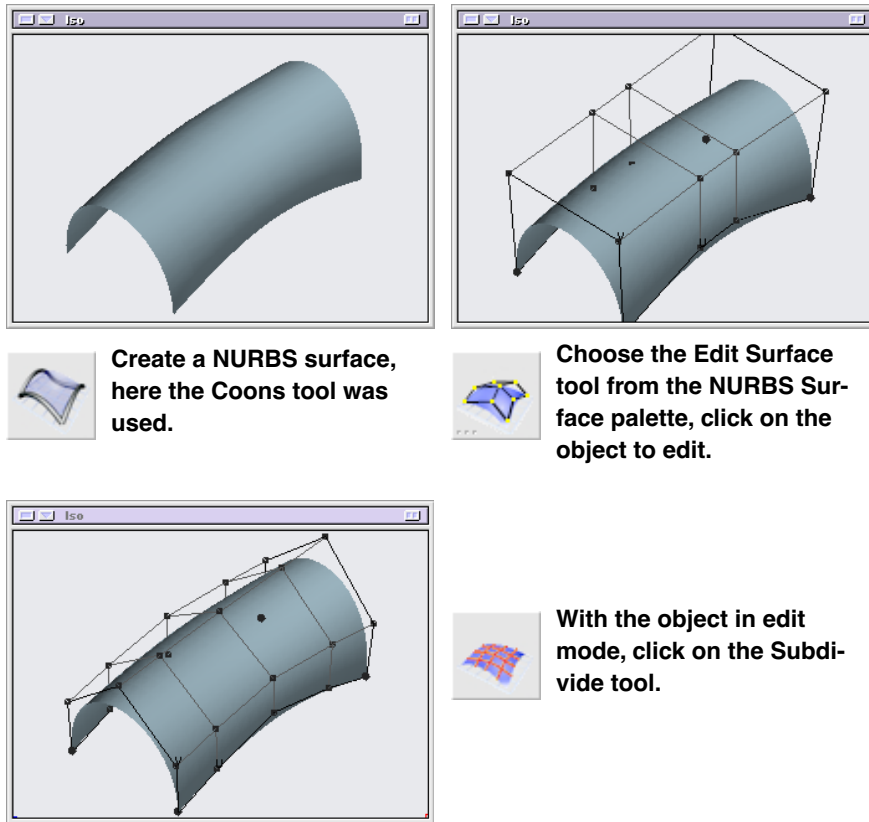
### Subdividing the Control Hull of a Surface

The easiest way to get more CVs to work with is to use the subdivide tool to add more resolution to your control hull mesh. You can subdivide the mesh evenly on both the u and v directions, or you can choose to subdivide just one axis — u, for example.

#### To subdivide an entire surface control hull:

1. Make sure that the surface is already in editing mode
2. Choose the subdivide tool from the NURBS Surface palette

The surface control hull is now subdivided.



*Figure 20.6 — Subdividing a NURBS surface control hull*

**To subdivide a control hull in a single direction (U or V):**

1. Make sure that the surface is already in editing mode
2. Depress the desired direction key (U or V)
3. Choose the subdivide tool from the NURBS Surface Palette

The surface control hull is now subdivided on the desired direction only.

You can subdivide a surface as often as you wish. However, you should only subdivide the surface to the level necessary to get the shape that you require. No need to add unnecessary complexity.

Of course, this method is a fairly radical way to add the necessary resolution to your control hull mesh. There are other, more precise methods that might be of better use.

### Adding and Removing Isoparms

Sometimes, you only want isoparms at a particular location. Why subdivide the entire control hull mesh when all you need to do is add a single isoparm? Fortunately, you don't have to if you don't want to.

#### To add an isoparm to a surface:

1. Make sure that the surface is already in editing mode
2. Choose the Add Isoparm tool from the NURBS Surface Palette

The display will now change. The CVs and interior control hull are hidden, so that only the isoparms are visible.

3. Find the desired location of the isoparm on the target surface
4. Click to add the isoparm

You will now see an isoparm has been added to the surface, in both the u and v directions. Adding an isoparm is essentially adding a knot in the u and v directions. As such, a corresponding row of CVs will also be added to the surface.

Like the subdivide tool, you may need to add an isoparm on a single direction only, as opposed to the entire surface. And, like the subdivide tool, this is also very easy to do.

**To add an isoparm in a specific direction (U or V):**

1. Make sure the surface is already in editing mode
2. Choose the Add Isoparm tool from the NURBS Surface Palette
3. Depress the desired direction key (U or V)
4. Find the desired location of the isoparm on the target surface
5. Click to add the isoparm

You will now see that a single u or v isoparm (knot) has been added to the surface, and that a row of CVs conforming to the knot has been added to the surface control hull.

Now that you know how to add isoparms, you need to know how to remove them.

**To remove an isoparm from a surface:**

1. Make sure the surface is already in editing mode
2. Choose the Remove Isoparm tool from the NURBS Surface Palette
3. Click on the isoparm that you wish to remove

The isoparm will now be removed from the surface. You can see that it is very simple to add and remove isoparms on a surface. However, there are still two more tools, Elevate and Reduce Surface Degree, which can be used to add or remove resolution on a surface. Be warned, these tools affect the complexity of the surface (see page 339 for further information).

### 20.5 Continuity

When creating the overall shape of an object, separate surfaces are often used to represent what would actually be a single complete object in the real world. In order for the object that is constructed on the computer to match the appearance of the real world object, it is often necessary to hide



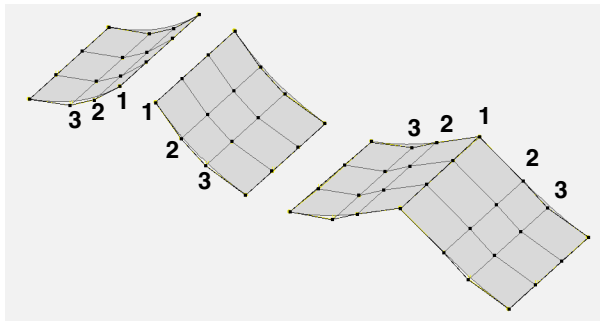
the seams where the separate surfaces come together. Hiding these seams is achieved through surface continuity<sup>1</sup>.

Universe Modeler offers three continuity levels:

- Positional Continuity
- Tangent Continuity
- Curvature Continuity

### Positional Continuity

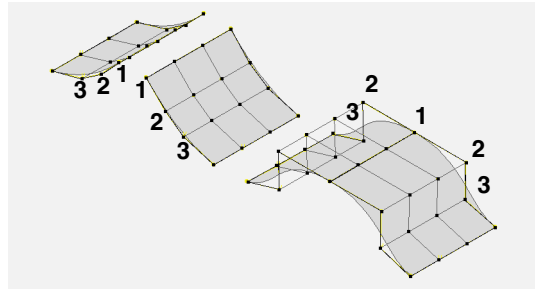
Positional continuity aligns the first row of CVs along the edge of one surface with another. The surfaces will be touching, but will shade as if a hard seam exists between the two. In Figure 20.7, two identical sets of surfaces are shown. The two surfaces on the right have been joined together with positional continuity.



**Figure 20.7 — Positional continuity of two surfaces (both surfaces changed)**

---

1. Surfaces to be made continuous must be non-rational, that is, without weights applied.



**Figure 20.8 — Tangent continuity of two surfaces (both surfaces changed)**

Notice how the first row of CVs along the edge of each surface have moved to join together with each other.

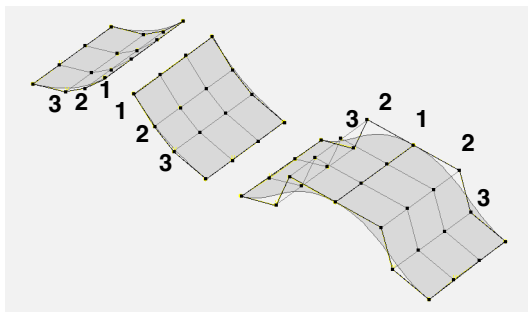
### Tangent Continuity

Tangent continuity aligns the first and second rows of CVs along the two surfaces to achieve a tangency match where the two surfaces meet. The surfaces will be touching, and will shade smoothly, though possibly with a noticeable change in direction depending upon the starting state of the two surfaces. In Figure 20.8, two identical sets of surfaces are again shown. The set on the left shows the two surfaces before tangent continuity was applied, and the two surfaces on the right afterwards.

Notice how the first and second row of CVs along each surface have been adjusted, and how they all join in a straight line.

### Curvature Continuity

Curvature continuity on two surfaces matches the position and tangent continuity of each surface, and matches the curvature of the surfaces as well. Figure 20.9 illustrates this. Unlike tangent continuity, these surfaces will shade completely smooth when rendered.

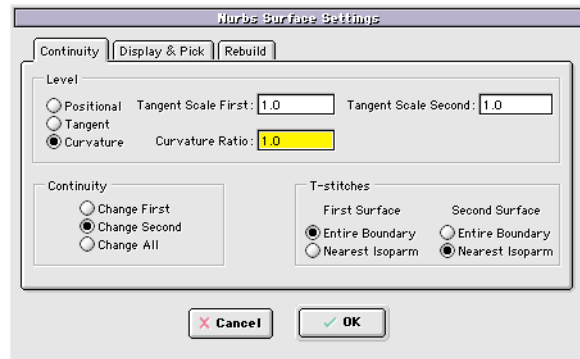


*Figure 20.9 — Curvature continuity of two surfaces (both surfaces changed)*

Notice that the third CV row on either surface has changed to allow for the curvature continuity. Notice also the difference between Figure 20.9 and Figure 20.8. The third row of CVs on each surface extend somewhat more outwards in Figure 20.9 than they do in Figure 20.8 due to the additional curvature continuity being applied. The difference is small, but will have a noticeable effect on these surfaces when rendered.

## Continuity Settings

When you need to join surfaces together, you will need to tell Universe Modeler which continuity level you prefer. This is done via the NURBS Surface Settings dialog, Continuity tab, as shown in Figure 20.10.



**Figure 20.10 — NURBS surface settings dialog, continuity tab**

This tab offers several continuity settings and options. As mentioned in the previous section, there are three levels of continuity that you can apply to surfaces: Positional, Tangent and Curvature. The control settings for these methods are located in the Level section of the tab. We will cover the tangent magnitude and curvature ratio in the following section.

The continuity section allows you to determine which surface will be modified to make the surfaces continuous. There are three choices:

- Change First
- Change Second
- Change All

Change first will only change the surface which is picked first. Change second will only change the surface which is picked second. Change all will attempt to change both surfaces evenly. In some cases of curvature continuity, this cannot always be done.

The T-Stitch section allows you to handle continuity for surfaces which do not meet edge to edge. This junction commonly looks like the letter “T”, hence the name. T-stitches will attempt to match to the first interior isoparm that is encountered to the boundary of the other surface, depending on the settings. Each surface in the operation can have one of the following two settings:

- Entire Boundary
- Nearest Isoparm

*While the program offers this function, you may find that you have more success in the long run by first splitting the surface along the isoparm, and then applying continuity without dealing with the T-stitch.*

### Working with Continuity

Now that we have covered the basics of continuity, let’s get started. In this section, we will illustrate the use of the continuity tools, and discuss the tangent magnitude and continuity ratio options.

#### **To apply positional continuity to two surfaces:**

1. Make sure each surface is already in editing mode (use the “n” key to select each additional surface)
2. Select positional continuity from the NURBS Surface Settings dialog, continuity tab (available when you double-click the Edit Surface icon on the NURBS Surface palette)
3. Choose the desired continuity relationship and close the dialog
4. Select the continuity icon from the NURBS Surface palette

The display of the surfaces will now change. Instead of the control hull mesh previously displayed, you will now see just the boundary hulls of the

surface in different colors, and a corresponding dot in the middle of each hull edge.

*The colors are different to help you more easily identify the boundary hulls for each surface. Do not pick the boundary edges of the same surface, as they cannot be made continuous. Split the surface if that situation arises.*

5. Click on the dot of the first surface edge to make continuous
6. Click on the dot of the second surface edge to make continuous

The two surfaces will now touch edges, and the normal surface edit mode will be restored.

If you wish to permanently join these two surfaces, you will need to use the surface stitch tool in the Boolean palette once you exit surface edit mode. This is necessary in order for the tessellator to produce a “water-tight” polygon mesh when exporting to Universe Animator.

### **To apply Tangent Continuity to two surfaces:**

1. Make sure each surface is already in editing mode (use the “n” key to select each additional surface)
2. Select tangent continuity from the NURBS Surface Settings dialog, continuity tab (available when you double-click the Edit Surface icon on the NURBS Surface palette)
3. Choose the desired continuity relationship and close the dialog
4. Select the continuity icon from the NURBS Surface Palette

The display of the surfaces will now change. Instead of the control hull mesh previously displayed, you will now see just the boundary hulls of the surface in different colors, and a corresponding dot in the middle of each hull edge.

*The colors are different to help you more easily identify the boundary hulls for each surface.*

5. Click on the dot of the first surface edge to make continuous
6. Click on the dot of the second surface edge to make continuous

Once the continuity relationship is established between these two surfaces, you can further edit the tangent magnitude of the surfaces. This is indicated when the boundary handle dots turn orange. Depending upon the continuity relationship you have chosen, you may be able to edit the tangent magnitude for both surfaces (change all) or for a single surface (change first or change second). This would be the same as typing in a value for the tangent magnitude edit boxes in the NURBS Surface Settings dialog, continuity tab.

### **To change the Tangent Magnitude of Tangent Continuous Surfaces:**

1. Depress the “t” key
2. Click and drag on the orange boundary handle for the desired surface
3. Drag to the right to increase the Tangent Magnitude
4. Drag to the left to decrease the Tangent Magnitude

You can edit the Tangent Magnitude interactively only while the continuity tool is active for this operation.

*Once you terminate the edit operation, you will not be able to adjust the Tangent Magnitude of these two surfaces again, unless you undo the continuity operation and start again.*

5. To finish editing the Tangent Magnitude interactively, double-click (which will end the surface edit session as well) or select the Edit Surface icon from the NURBS Surface Palette

The two surfaces will now touch edges, and they will be tangent continuous.

If you wish to permanently join these two surfaces, you will need to use the surface stitch tool in the Boolean palette. This is necessary in order for the tessellator to produce a “water-tight” polygon mesh when exporting to Universe Animator.

### **To apply Curvature Continuity to two surfaces:**

1. Make sure each surface is already in editing mode (use the “n” key to select each additional surface)
2. Select curvature continuity from the NURBS Surface Settings dialog, continuity tab (available when you double-click the Edit Surface icon on the NURBS Surface palette)
3. Choose the desired continuity relationship and close the dialog
4. Select the continuity icon from the NURBS Surface Palette

The display of the surfaces will now change. Instead of the control hull mesh previously displayed, you will now see just the boundary hulls of the surface in different colors, and a corresponding dot (boundary handle) in the middle of each hull edge.

*The colors are different to help you more easily identify the boundary hulls for each surface.*

5. Click on the dot of the first surface edge to make continuous
6. Click on the dot of the second surface edge to make continuous

Once the continuity relationship is established between these two surfaces, you can further edit the tangent magnitude of the surfaces as well as the curvature ratio of the two surfaces. This is indicated when the boundary handle dots turn orange. Depending upon the continuity relationship you



have chosen, you may be able to edit the tangent magnitude and curvature ratio for both surfaces (change all) or for a single surface (change first or change second). This would be the same as typing in a value for the tangent magnitude and curvature ratio edit boxes in the NURBS Surface Settings dialog, continuity tab.

**Why Edit  
Tangent  
Magnitude  
and Curva-  
ture Ratio?**

You may be wondering why both the tangent magnitude and curvature ratio can be edited? The three levels of continuity offered in Universe Modeler are progressive. You cannot have tangent continuity without also having positional continuity. You cannot have curvature continuity without also having positional continuity and tangent continuity. The relationship between each level of continuity is hierarchical and required. As such, when curvature continuity is applied, you can also edit the tangent magnitude to your liking, as well as edit the curvature ratio of the surface.

Continuity operations can often change the surfaces to the point that you need to adjust them to your liking. You can choose to pre-enter the appropriate values in the NURBS Surface Settings dialog, and if precision work is required then by all means do so. More than likely, it is far easier to interactively edit these factors after the continuity operation, as opposed to entering the appropriate values ahead of time.

**To change the tangent magnitude of curvature continuous surfaces:**

1. Depress the “t” key
2. Click and drag on the orange boundary handle for the desired surface
3. Drag to the right to increase the tangent magnitude
4. Drag to the left to decrease the tangent magnitude

**To change the curvature ratio of curvature continuous surfaces:**

5. With the surfaces still in continuity mode, depress the “r” key (for “cur-

vature ratio")

6. Click and drag on the orange boundary handle for the desired surface
7. Drag to the right to increase the curvature ratio
8. Drag to the left to decrease the curvature ratio

You can edit the tangent magnitude and the curvature ratio interactively only while the continuity tool is active for this operation.

*Once you terminate the edit operation, you will not be able to adjust the tangent magnitude or the curvature ratio of these two surfaces again, unless you undo the continuity operation and start again.*

9. To finish editing the tangent magnitude and curvature ratio interactively, double-click (which will end the surface edit session as well) or select the Edit Surface icon from the NURBS Surface palette

The two surfaces will now touch edges, they will be tangent continuous and curvature continuous.

If you wish to permanently join these two surfaces, you will need to use the surface stitch tool in the Boolean palette. This is necessary in order for the tessellator to produce a “water-tight” polygon mesh when exporting to Universe Animator.

### **To apply a T-stitch continuity to two surfaces:**

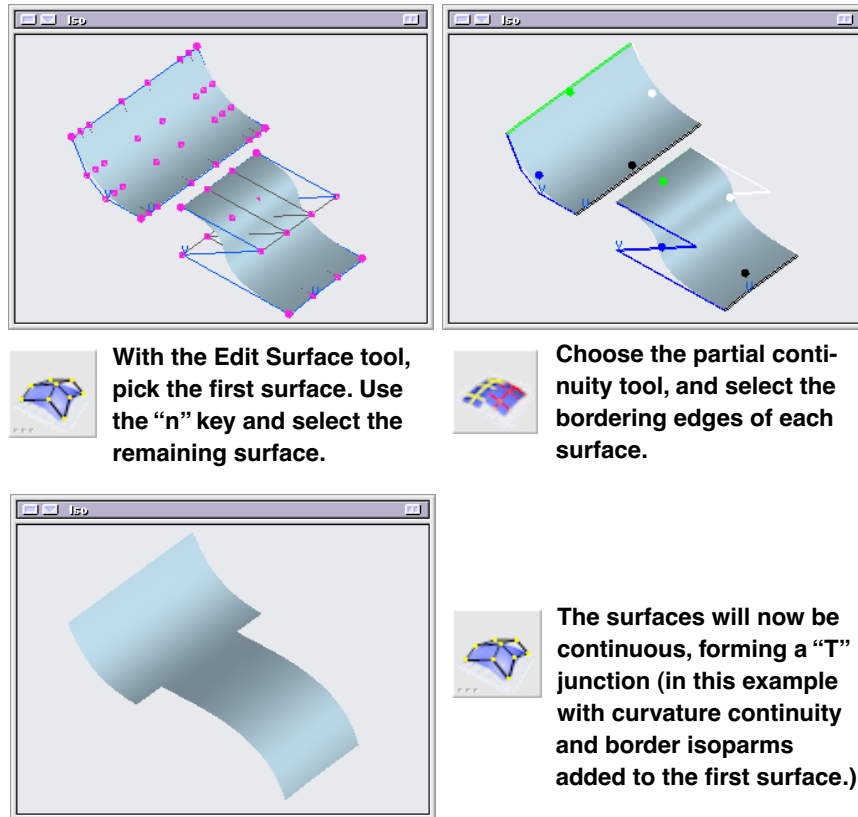
1. Make sure each surface is already in editing mode (use the “n” key to select each additional surface)
2. Select the desired continuity level from the NURBS Surface Settings dialog, continuity tab (available when you double-click the Edit Surface icon on the NURBS Surface palette)
3. Choose the desired continuity relationship and close the dialog
4. Select the partial continuity icon from the NURBS Surface palette

The display of the surfaces will now change. Instead of the control hull mesh previously displayed, you will now see just the boundary hulls of the surface in different colors, and a corresponding dot (boundary handle) in the middle of each hull edge.

*The colors are different to help you more easily identify the boundary hulls for each surface.*

5. Click on the dot of the edge of the first surface to make continuous
6. Click on the dot of the edge of the second surface to make continuous

The surfaces will now be continuous per the chosen continuity level.



**Figure 20.11 — Applying T-stitch continuity to two surfaces**

As mentioned in the previous section, T-stitches can sometimes cause unusual results, such as the following:

- Unless you want the surfaces to flow into each other, you will want to add isoparms on the larger surface that border the smaller surface as closely as possible.
- Given the above, it may not be possible to place the isoparms on the larger surface with exact precision.

- In the cases where you are unsatisfied with the T-stitch, consider splitting the surface along the isoparms that are nearest the smaller surface, and use normal continuity with the resulting split surface instead.
- Alternatively, you can start out with two surfaces that are the same size, and extend the edges of one of the surfaces in order to form the T-stitch.

### 20.6 Rebuilding Surfaces

Sometimes, when working with your surfaces, you will find yourself needing to simplify your surfaces, as they may become too complex to deal with in further modeling operations. You may also find yourself needing to make adjustments to your surfaces for a variety of reasons, especially if you understand the underlying mathematics upon which NURBS are based.

A separate tool sub-palette called “Rebuild NURBS Surface tools” has been created to help in these situations. In most cases, you should not need to venture into the sub palette, especially if you are not familiar with NURBS. These tools can be very destructive, and should only be used if absolutely necessary.

The reduce knots tool, on the other hand, should be used to reduce the complexity of a surface as much as needed (to degree three for both  $u$  and  $v$ ), so as to keep the CV and knot counts within reason.

There are ten tools in this palette:

- Make Surface Rational and Make Surface Non-Rational
- Scale Knot Vectors and Offset Knot Vectors
- Rebuild Surface and Fit Surface
- Uniform Parameterization and Arc Length Parameterization
- Elevate Degree and Reduce Degree

### Make Surfaces Rational and Non-Rational

These two tools control whether or not your surface will have weights that you can edit (rational) or not (non-rational.) Overall, we prefer to discourage the practice of weight editing on a surface. This is due to a number of reasons, such as continuity, tessellation and ACIS compatibility.

#### **To make a surface rational (add editable weights to a surface):**

1. Make sure the surface is already in editing mode (Rebuild Surface tools function on the last edited surface. The Selected Surface.)
2. Choose the Make Surface Rational icon from the Rebuild NURBS Surface tools sub-palette.

*The make rational tool automatically jumps back to surface edit, as do all of the tools in the rebuild palette.*

The surface will now be rational, assigning the weight value found in the rebuild tab of the NURBS Surface Settings dialog (available when double-clicking on the Edit icon.) To verify this, make sure that display weights is active (display & pick tab of the NURBS Surface Settings dialog.) After “rationalizing” a surface, weight values should appear for the selected CVs (if no CVs are selected, no weight values will appear.)

#### **To make a surface non-rational (remove editable weights from a surface):**

1. Make sure the surface is already in editing mode (Rebuild Surface tools function on the last edited surface. The Selected Surface.)
2. Choose the Make Surface Non-Rational icon from the Rebuild NURBS Surface tools sub-palette.

The surface will now be made non-rational, removing any weights which may have been applied to the surface. However, the surface will be modified by inserting isoparms to keep the approximate shape of the surface the

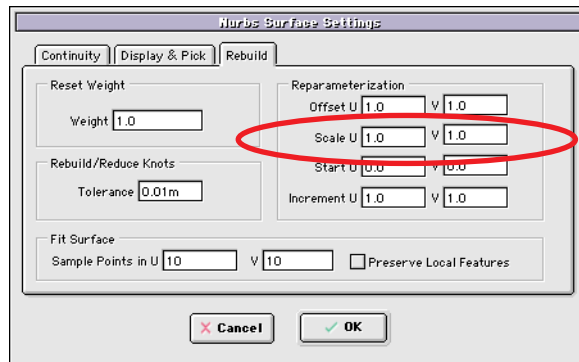
same prior to operation. To do this, the tool will use the value found in the Reduce Knots Tolerance edit box, found in the Rebuild tab of the NURBS Surface Settings dialog. If you have a very small tolerance value, you will see that more CVs and isoparms have been added to maintain the shape of the surface. Larger values will yield less drastic changes to the isoparm count and the control hull of the surface, but may result in a change of shape from the original surface.

### Scale Knot Vectors of a Surface

You can scale each value in the U and V knot vectors of a surface by a specified value in the Rebuild tab of the NURBS Surface Settings dialog. If you want to scale only in one parametric direction, make the other scale value 1. Since this function will not change the locations of the knot values, the surface shape remains the same. However, the tangents at various points on the surface will differ from the original after the operation.

You can set a new start value for the U and the V for the scale operation. These values can be set in the appropriate edit boxes, as indicated in Figure 20.12.

*If you do not understand why you would need to do this, do not be alarmed. You will likely never need to deal with these features. These attributes are presented in the program for those who are familiar with the mathematics underlying NURBS.*



**Figure 20.12 — Scale knot vector attributes highlighted in the rebuild tab of the NURBS Surface Settings dialog**

**To scale the knot vectors of a surface:**

1. Make sure the surface is already in editing mode (Rebuild Surface tools function on the last edited surface. The Selected Surface.)
2. Choose the scale knot vectors icon from the Rebuild NURBS Surface tools sub-palette.

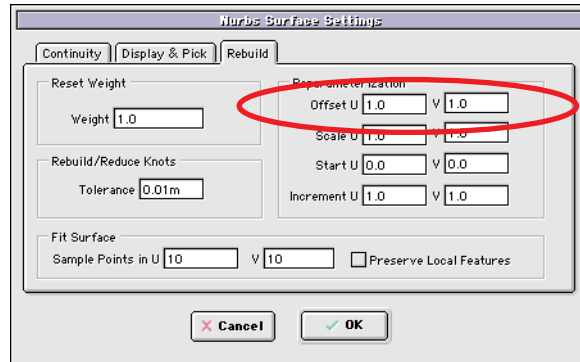
The knot vectors of the surface will now be scaled to the desired value. You can check the knot values by turning on the numeric display from the system preferences dialog box.

### Offset Knot Vectors of a Surface

You can offset each value in the U and V knot vectors of a surface by a value specified in the rebuild tab of the NURBS Surface Settings dialog. If you want to offset the knot vector in only one parametric direction, make the other offset value 0. As with scaling the knot vectors, the surface shape



remains the same, whereas the tangents change as indicated in Figure 20.13.



**Figure 20.13 — Offset knot vector attributes highlighted in the rebuild tab of the NURBS Surface Settings dialog**

**To offset the knot vectors of a surface:**

1. Make sure the surface is already in editing mode (Rebuild Surface tools function on the last edited surface. The Selected Surface.)
2. Choose the offset knot vectors icon from the Rebuild NURBS Surface tools sub-palette.

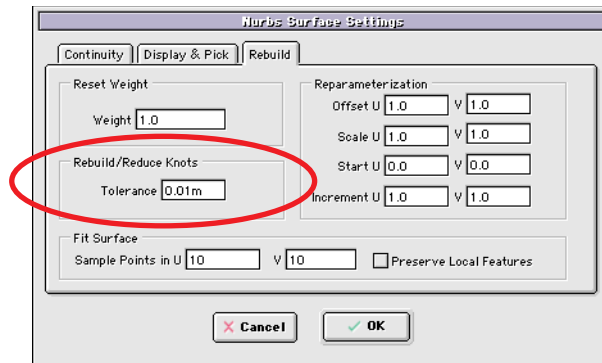
The knot vectors of the surface will now be offset by the desired value. You can check the knot values by turning on the numeric display from the system preferences dialog box.

### Reduce Knots

After working with them for a time, surface control hulls can get more and more complex to deal with. They can become very dense, making it difficult to select CVs, or perhaps they begin to develop folds or creases that are

unwanted. One way to get surfaces back under control is to rebuild them, and that is what the rebuild surfaces tool is for. Get to know this tool, as it can become one of the best friends that you ever had.

The rebuild surfaces tool does its job by deleting all removable knots (which results in a reductions of isoparms and their associated CVs) from a NURBS surface to a tolerance you specify in the rebuild tab of the Rebuild NURBS Surface tools dialog, as highlighted in Figure 20.14.



**Figure 20.14 — Reduce knots attributes highlighted in the rebuild tab of the NURBS Surface Settings dialog**

The reduce knots tool also reparameterizes the surface with respect to arc length.

**To reduce the knots on a surface with the reduce knot tool:**

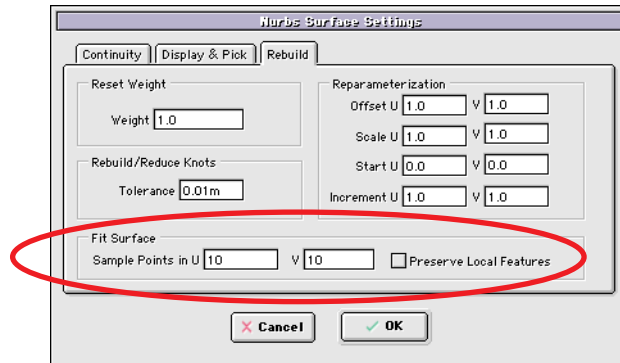
1. Make sure the surface is already in editing mode (Rebuild Surface tools function on the last edited surface. The Selected Surface.)
2. Choose the reduce knot icon from the Rebuild NURBS Surface tools sub-palette.

The surface will now be rebuilt to the tolerance that you specified. The default value may not be suitable for your needs, so feel free to experiment. Smaller tolerance values will provide more complex surfaces which more accurately represent the surface you started with. Larger tolerance values will less accurately represent the surface you started with. Depending upon your needs, that result may not be undesirable. In the case where you need to change values and try again, always undo your earlier attempt, and try a new tolerance value. The results will be more consistent.

### **Fitting a New Surface to Another Surface**

While it may be warranted to rebuild a surface, sometimes even that approach may not yield the result that you require. For those times when you wish an even simpler resulting surface, the best option may be to fit an entirely new surface to a previously existing one. There are many reasons for this:

- Simplify knot values, CV and isoparm count
- Smooth out surface irregularities, and so on



**Figure 20.15 — Fit surfaces attributes highlighted on the rebuild tab of the NURBS Surface Settings dialog**

The Fit Surface tool accomplishes this by fitting a completely new surface to a set of data points which have been sampled from a selected surface. The number of data points to be sampled is specified in the Rebuild tab in the NURBS Surface Settings dialog. The output will be a smooth surface through those data points. However, if you prefer to capture local surface details such as flat-spots or cusps, use the “preserve local features” check box, which will try to capture that in the output surface. A tolerance which specifies maximum deviation of the surface from each data point can be set in the tolerance field.

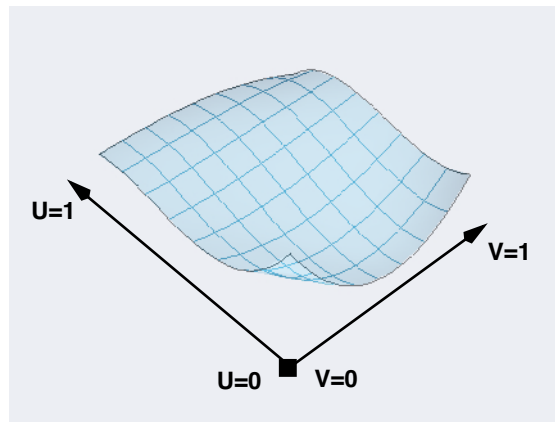
**To fit a new surface to another one:**

1. Make sure the surface is already in editing mode (Rebuild Surface tools function on the last edited surface. The Selected Surface.)
2. Choose the Fit Surface icon from the Rebuild NURBS Surface tools sub-palette.

A new surface will be fitted to the old surface, and the old surface will be deleted. Depending upon your attributes settings, the newly fit surface may not seem to match the original surface all that well. In that case, you might need to increase the resolution of the U and V samples for a better result.

### Surface Parameterization — Uniform and Arc Length

All surfaces have two directions, U and V. The origin (starting corner point) of a surface always has a value of 0, and the directions of the U and V always travel from the origin outwards, away from each other (see Figure 20.16). You can swap the U and V of a surface, and you can change the location of the origin, but you cannot have the direction of the U and V parameters travel in opposite directions. What you *can* do is determine *how* the surface is parameterized.



*Figure 20.16 — Surface parameterization*

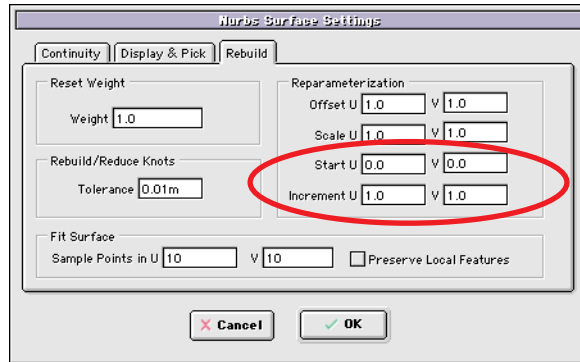
How the surface value varies along the U or V direction is called parameterization. In the illustration above, we see the typical example U and V range of 0 to 1. Uniform parameterization will yield equally-spaced knot values, such as 1,2,3,4,5,etc., regardless of whether or not the isoparms on the surface are evenly-spaced. Arc-length parameterization will yield precise knot values for the surface, which considers the spacing of the isoparms on the surface, such as 0.327, 1.4289, 3.75, etc. Arc-length parameterization is closer to an accurate representation of the actual size of the surface than uniform, which is a more abstract form of parameterization.

The effect of parameterization will be primarily noticeable when tessellating the surface, or texture mapping the surface. Uniform surfaces may occasionally show some distortion. Arc-length surfaces will tend to yield more consistent results in terms of texture distortion and polygon distribution when tessellating.

*In fact, the only time to worry about your parameterization is if you are experiencing problems with either your tessellation or texture map distortion.*

### **To apply uniform parameterization to a surface:**

1. Make sure the surface is already in editing mode (Rebuild Surface tools function on the last edited surface. The Selected Surface.)
2. In the rebuild tab of the NURBS Surface Settings dialog, you can set a start value other than 0 if you wish
3. In the same tab, you can also set an increment value other than 1
4. Choose the uniform parameterization icon from the Rebuild NURBS Surface tools sub-palette.



**Figure 20.17 — Uniform parameterization attributes highlighted on the rebuild tab of the NURBS Surface Settings dialog**

The surface will now be uniformly parameterized, per your settings in the rebuild tab of the NURBS Surface Settings dialog. You will not see any change in the shape of the surface.

### To apply arc-length parameterization to a surface:

1. Make sure the surface is already in editing mode (Rebuild Surface tools function on the last edited surface. The Selected Surface.)
2. Choose the arc-length parameterization icon from the Rebuild NURBS Surface tools sub-palette.

The surface will now be arc-length parameterized. You will not see any change in the shape of the surface.

## Elevating or Reducing the Degree of a Surface

In Universe Modeler, surfaces that you create with the variety of tools available can also have varying degrees, on both u and v. Degree three surfaces are preferred, as calculate quickly, and can be used to create virtually any shape you will require. If you are planning to use corner continuity

with your surfaces, any surface used with that tool must be degree three or it will be rejected by the corner continuity tool. Depending upon the situation, you could find need to adjust the degree of the u or v parameter of a surface.

### **To raise the degree of a surface:**

1. Make sure the surface is already in editing mode (Rebuild Surface tools function on the last edited surface. The Selected Surface.)
2. Choose the Elevate Degree tool from the Rebuild NURBS Surface tools sub-palette

The surface degree will now be raised by one for the u and the v directions. The tool results will be placed within the status window. If the degrees do not match, use either the elevate or reduce degree tool to make them conform.

### **To raise the degree of a single direction of a surface (u or v):**

1. Make sure the surface is already in editing mode (Rebuild Surface tools function on the last edited surface. The Selected Surface.)
2. Depress the appropriate direction key (u or v)
3. Choose the Elevate Degree tool from the Rebuild NURBS Surface tools sub-palette

The surface degree will now be raised by one for the desired direction only.

### **To lower the degree of a surface:**

1. Make sure the surface is already in editing mode
2. Choose the Reduce Degree tool from the Rebuild NURBS Surface tools sub-palette

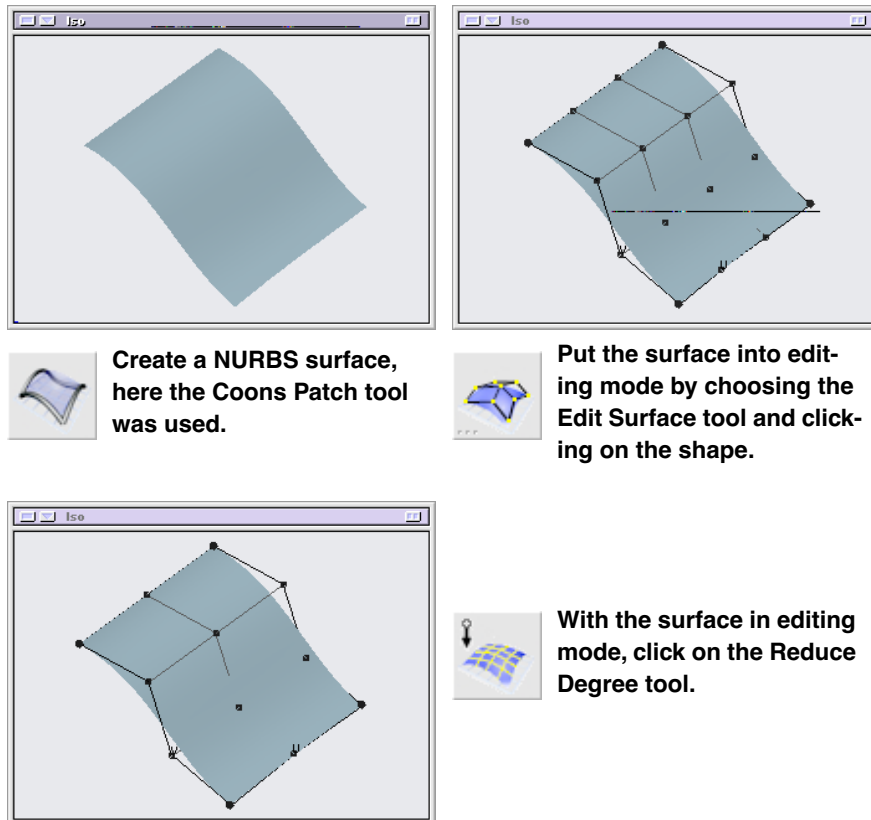
The surface degree will now be lowered by one for both u and v directions.



**To lower the degree of a single direction of a surface (u or v):**

1. Make sure the surface is already in editing mode (Rebuild Surface tools function on the last edited surface. The Selected Surface.)
2. Depress the appropriate direction key (u or v)
3. Choose the Reduce Degree tool from the Rebuild NURBS Surface tools sub-palette

The surface degree will now be lowered by one for the desired direction only. Degree reduction is not always possible without a potentially drastic change in the shape of the surface. If this is the case, Universe Modeler will display a dialog ahead of time, giving you the option to cancel.



*Figure 20.18* — Reducing the degree of a NURBS surface

If your goal is to have less isoparms on a surface, then the reduce degree tool is not the best choice. Instead, use the remove isoparm tool.

## 20.7 More Surface Editing Features

In addition to the previously discussed surface editing features, there are several more features which can make working with surfaces easier in Uni-

verse Modeler. Some of these features have been referred to in other sections, and some will be covered here for the first time:

- Reverse parameterization
- Align Selected CVs
- Split Surface
- Join Two Surfaces
- Surface Symmetry Plane
- Edit Surface Visibility
- Surface Blends
- Surface Compatibility
- Extend Surface by Span

### Reverse Parameterization

Whether a surface is uniform or arc-length, it is sometimes necessary to change the surface origin, swap out which direction is U and which direction is V, and so on.

**To change the origin location of a surface (to the opposite corner):**

1. Make sure the surface is already in editing mode (use the “n” key to select each additional surface)
2. Choose the Reverse Parameterization icon from the NURBS Surface palette.

The location of the origin of the surface will now be placed at the opposite corner, and the U and V directions will be reversed. Reversing either the u or v only will reverse the surface normal. You may need to choose the reverse tool on the Body palette after exiting the surface edit environment to rectify this if necessary. If you reverse both u and v, the normal is left as-is.

### **To reverse the direction of either U or V:**

1. Make sure the surface is already in editing mode (use the “n” key to select each additional surface)
2. Depress the desired direction key (U or V)
3. Choose the Reverse Parameterization icon from the NURBS Surface palette.

The desired direction will now be reversed. Note also that in keeping with the rule that both directions must share the same origin, the other direction will be drawn at a different location than prior to the operation.

### **To interchange the U and V directions (without changing the origin):**

1. Make sure the surface is already in editing mode (use the “n” key to select each additional surface)
2. Depress the “i” direction key
3. Choose the Reverse Parameterization icon from the NURBS Surface palette.

The U and V of the surface will now be swapped, but the origin will remain the same.

## **Aligning CVs**

From time to time, you may want to clean up your control hull, by aligning CVs together with each other. The align tool offers three alignment options:

- Align to a Straight Line (“L” click)
- Align to a Plane (“P” click)
- Align to a Point (“K” click)

*For all of the alignment tools (except for point alignment), it is best to select your CVs one at a time. If you select your CVs by marquee selection (rectangle drag), the results you get will be determined by the program.*

**Line** To align to a line, the first two CVs that you select will determine the line vector. All other CVs you select will conform to the vector established by the first two CVs.

**Plane** To align to a plane, the first three CVs that you select will determine the plane upon which all of your selected CVs will be placed.

**Point** To align to a point, the centroid of the selected CVs will be used. This tool is best used when attempting to attach CVs of one surface to another. If you attempt to align several CVs of the same surface to a single point, ACIS will reject the surface for boolean and rounding operations.

### **To align selected CVs with the Align Tool:**

1. Make sure the surface is already in editing mode (use the “n” key to select each additional surface)
2. Depress the desired alignment key (“L”, “P”, or “K”)
3. Choose the Align icon from the NURBS Surface palette.

The selected CVs will now conform to the alignment that you chose. You do not have to use the align tool directly, however, you can also align during normal CV editing, as indicated below.

### **To align selected CVs during editing:**

1. Make sure the surface is already in editing mode (use the “n” key to select each additional surface)
2. Depress the desired alignment key (“L”, “P”, or “K”)
3. Click.

As in the previous series of steps, the selected CVs will now conform to the alignment that you chose.

### Split Surface

The split surface tool will split the selected surface along the desired U or V isoparm.

#### To split a surface along an isoparm:

1. Make sure the surface is already in editing mode (use the “n” key to select each additional surface)
2. Choose the Split Surface icon from the NURBS Surface palette
3. On the surface, choose the isoparm to divide the surface into separate surfaces
4. Continue on to the next isoparm as desired

The surface will now be split into two (or more if you continued the operation.)

### Join Two Surfaces

The Join Surfaces tool will join two surfaces together to form a single surface. You can use the continuity tool before the join tool to better prepare the two surfaces so that they are compatible with each other.

#### To join two surfaces together:

1. Make sure each surface is already in editing mode (use the “n” key to select each additional surface)
2. Choose the Join Surfaces tool from the NURBS Surface palette

The display of the surfaces will now change. Instead of the control hull mesh previously displayed, you will now see just the boundary hulls of the

surface in different colors, and a corresponding dot in the middle of each hull edge.

*The colors are different to help you more easily identify the boundary hulls for each surface.*

3. Click on the dot on the edge of the first surface edge to join
4. Click on the dot on the edge of the second surface edge to join

The two surfaces will now be joined into one continuous surface. This process is very different from the Boolean Join tool recommended when applying continuity to separate surfaces. That tool will treat the result as a single body with two faces, whereas the join tool will create a single-faced surface. Typically, the boolean join tool will result in more tessellator-friendly results than the Join Surfaces tool will, for a variety of ACIS-specific reasons. However, a topologically simpler surface is often more desirable when using the rounding tools. Experience will tell you what the best approach for you will be.

### Surface Symmetry Plane

Symmetry is often the holy grail of surface and shape design. It is often easier to create and edit one half of a surface and then have the other half reflect the changes than it is to attempt to build the whole thing by hand. Fortunately, the surface symmetry plane helps to make this easy.

The symmetry plane is temporary; it does not stay established during the editing session, it merely reflects the U and/or V CV positions and weights from one side of the symmetry line to the other. This is very different from how symmetry works with the UberNURBS tool, so be aware of that.

**To apply symmetry to one side of a surface from the other:**

1. Make sure the surface is already in editing mode
2. Turn on the snapping grid (F5)
3. Choose the Surface Symmetry tool from the NURBS Surface palette
4. If you wish to limit the symmetry to U or V, depress the desired key
5. Choose which view you wish to apply the symmetry
6. Drag the symmetry line through the surface at the desired location

Depending upon the surface, and the choices you made while using the tool, you should now see a surface which is symmetrical. In the case where the resulting surface appears to be garbage, undo and use the U or V key and try again.

*The snapping grid helps to minimize input error for the plane. It is easy to run the plane through a CV if you are not careful, and this can lead to an undesirable result, hence our recommendation to use the grid snap for this feature.*

Feel free to experiment with different plane orientations, as some rather unusual and unexpected (and interesting!) results can be achieved with it.

### Control Hull Visibility

When your work starts to get cluttered from complex surfaces with many CVs, you can cut down on the clutter with the show/hide tool. Using direct or marquee (rectangle) selection, you can hide or expose various parts of the surface control hull that you are editing.

**To hide parts of a control hull:**

1. Make sure the surface is already in editing mode
2. Choose the show/hide tool from the NURBS Surface palette
3. Select the parts of the control hull that you wish to hide



The portions of the control hull that you have selected will now be hidden from view. They will also be un-selectable and uneditable. Only the remaining visible portions of the surface will be editable.

### **To expose parts of a hidden surface:**

1. Make sure the surface is already in editing mode
2. Choose the show /hide tool from the NURBS Surface palette
3. Depress the shift key
4. Select the parts of the control hull that you wish to expose (you can drag a rectangle around the entire surface if you wish — the visible parts will not be hidden by this operation)

The hidden portions of the control hull will now be visible and editable once again.

*Note that this operation is not the same as hiding the surface display with the tab key.*

## **Surface Blends**

There are times when the shape that you are trying to create cannot be accomplished with one or two surfaces. Take the fins on a 50's era car, for example. You could try and shape them from one sheet, and achieve some success, but you might experience some interesting results when texture mapping. You could try to build the fin with two surfaces, and use continuity to smooth the surfaces together, but the necessary changes that continuity must make to one or both surfaces may distort your fin too much. What to do?

The answer is to use a blend surface. A blend surface is created in the space between two other surfaces. These surfaces act as input surfaces to the blend, and allow you the most control over the shape of the car fin used in

our example (Figure 20.19). You could even add a chrome material property to the blended surface in Universe Animator, and voilà! In short, surface blends are a very powerful construction tool.

**To create a surface blend between two surfaces:**

1. Make sure each surface is already in editing mode (use the “n” key to select each additional surface)
2. Select the continuity level in the NURBS Surface Settings dialog
3. Choose the surface blend tool from the NURBS Surface palette

The display of the surfaces will now change. Instead of the control hull mesh previously displayed, you will now see just the boundary hulls of the surface in different colors, and a corresponding dot in the middle of each hull edge.

*The colors are different to help you more easily identify the boundary hulls for each surface.*

4. Click on the dot on the edge of the first surface edge to blend
5. Click on the dot on the edge of the second surface edge to blend

As is the case with tangent and curvature continuity, you have the opportunity to edit the tangent magnitude, in the case of tangent continuity on your blended surface or tangent magnitude and curvature ratio in the case of a curvature continuous blended surface.

**To change the tangent magnitude of curvature continuous surfaces:**

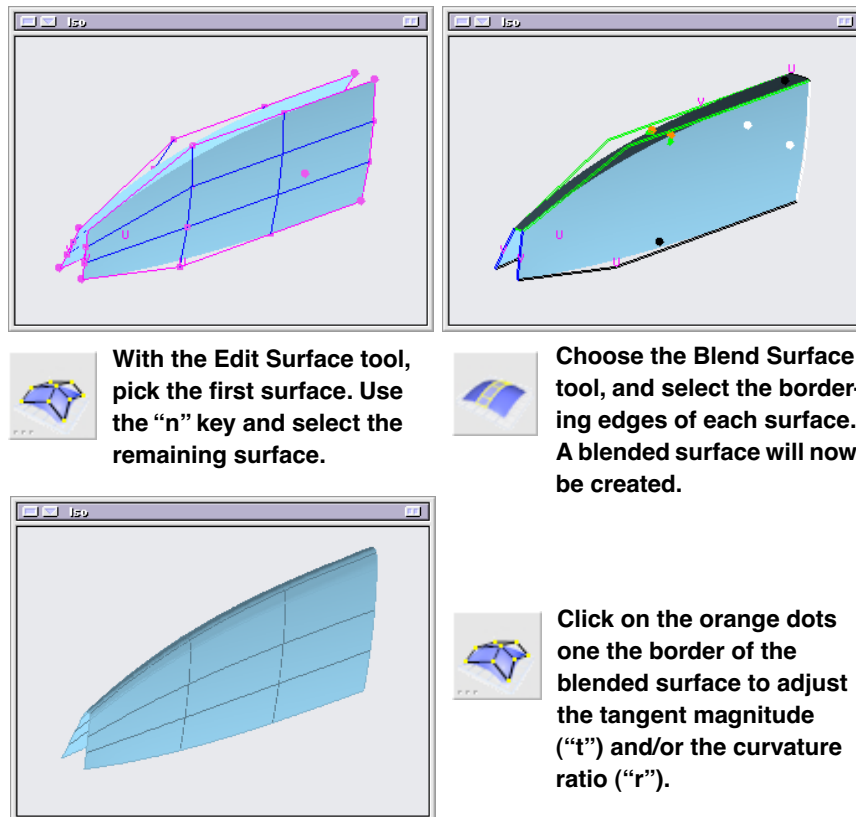
6. Depress the “t” key
7. Click and drag on the orange boundary handle for the blend surface
8. Drag to the right to increase the tangent magnitude
9. Drag to the left to decrease the tangent magnitude

**To change the curvature ratio of curvature continuous surfaces:**

10. With the surfaces still in continuity mode, depress the “r” key (for “curvature ratio”)
11. Click and drag on the orange boundary handle for the blend surface
12. Drag to the right to increase the curvature ratio
13. Drag to the left to decrease the curvature ratio

*You can edit the tangent magnitude and the curvature ratio interactively only while the continuity tool is active for this operation. Once you terminate the edit operation, you will not be able to adjust the tangent magnitude or the curvature ratio of these two surfaces again, unless you undo the continuity operation and start again.*

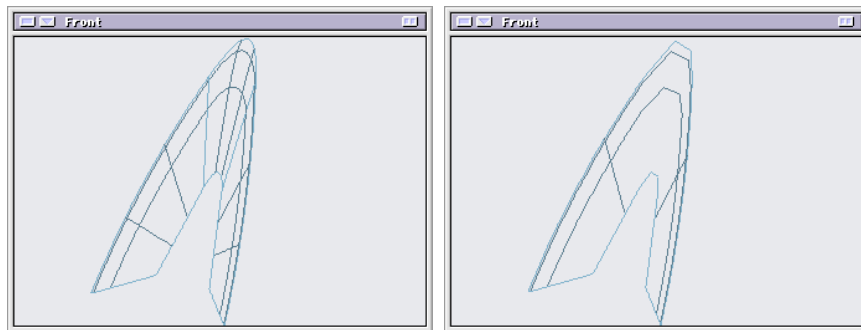
14. To finish editing the tangent magnitude and curvature ratio interactively, double-click (which will end the surface edit session as well) or select the edit surface icon from the NURBS Surface Palette



*Figure 20.19* — Surface blend example of a 50's car fin

You will now have a new third surface between your two original input surfaces. You will find that editing the tangent magnitude and curvature ratio of the blended surface is far easier than attempting to build some shapes as a single surface. The blended surface allows you to avoid dealing with too many CVs, and reduces the amount of time and effort it takes to make cool shapes quickly.

After you have created the surface blend, you will need to decide how you want to treat the surface as an object: do you want it separate, so that you can assign material attributes to it (such as that chrome treatment we were talking about); do you want to join all of the surfaces together into one larger surface (this will make tessellation settings a little more tricky, as leaving these surfaces either separate or as separate faces will allow more precision from the tessellator); or do you want to use the boolean join tool to attach the surfaces together into one body, but as three separate faces (which the tessellator will handle better than the joined surface — see Figure 20.20) The choice is yours and depends upon your unique situation.



**Figure 20.20 — Boolean stitched surfaces on the left and a joined surface of the same data on the right (3 faces vs.1 face)**

### Surface Compatibility

The Surface Compatibility tool is designed to make two surfaces compatible, which includes making the rationality and degrees identical, and their knot vectors span the same range. You probably won't ever need to use this tool, as the continuity, surface blend and surface join tools automatically

make surfaces compatible beforehand. However the difference is that the surface compatibility will make the selected surfaces compatible in both parametric directions, whereas the continuity tools try to be conservative and make surfaces compatible only the direction across which continuity is required.

### **To make two surfaces compatible:**

1. Make sure each surface is already in editing mode (use the “n” key to select each additional surface)
2. Choose the Surface Compatibility tool from the NURBS Surface palette

The surfaces will now be drawn with just the borders visible for clarity, similar to the continuity and blend surface tools.

3. Select the first surface (this surface will set the evaluation parameters for the following surface)
4. Select the second surface

The second surface will now be adjusted to match the parameters of the first surface. The status window will display the results of the Surface Compatibility operation.

From this point, you can continue to make other surfaces compatible, or you can return back to the normal surface edit mode by selecting the Edit Surface icon from the NURBS Surface palette. You can also double-click to exit the surface edit environment.

### **Extend Surface by Span**

Extend Surface by Span offers the ability to extend the existing surface from the beginning or the end of either the u or v direction with continuity tangent, curvature or a maximum possible continuity. Any editable sur-

faces that are not closed in the direction of extension are candidates for extension. The extension is achieved by simply clicking on an isoparm near the end you wish to extend. The following options are available:

- |                   |   |
|-------------------|---|
| <b>Positional</b> | Often it is desirable to not have multiple knots at the end of the original   |
| <b>Continuity</b> | surface. In that case, one can use this option. Each extension will be approximately the proportion of the average length of the last polygonal leg of the control hull in the direction of extension to the complete hull. Note that this method works best for short distances of extension, and it does not guarantee non-intersecting extensions. |
| <b>Tangent</b>    | When this option is chosen, the extension will be tangent continuous at the   |
| <b>Continuity</b> | point where the original surface ends and the extension begins. There will be multiple knots at the end of the original surface.  |
| <b>Curvature</b>  | To get a curvature continuous extension, use this option. The surface is  |
| <b>Continuity</b> | extended by reflection. Again, there will be multiple knots at the end of the original surface.   |

### 20.8 Hints on Using NURBS Surface Tools

*Here are a few tips to keep in mind when working with surface s in Universe Modeler.*

- If you set the tolerance too low you may see Universe Modeler go into an infinite loop trying to achieve that.
- If you see little or no change in the surface after the operation where you did expect to see some change, try relaxing the tolerance.
- If the tolerance is too loose, you may see a big change in the resulting surface; try making the tolerance tighter to keep the surface faithful to its original shape.
- If a body has multiple faces, you must select all the faces before you enter NURBS surface edit mode, otherwise you will lose the faces that were not selected from that body.
- After a series of operations if you end up with surfaces that don't pass the check body test, then you may want to rebuild the surface. A clean-up of the surface usually results in less (multiple) knots, which makes ACIS happy.
- A strict body check may report errors that may not affect further modeling; the warnings are not always indicative of bad data.

Should you wish to further investigate the deeper concepts of NURBS, please refer to “The NURBS Book” by Les Piegl and Wayne Tiller, and published by Springer (ISBN 3-540-55069-0).



# Local Transform

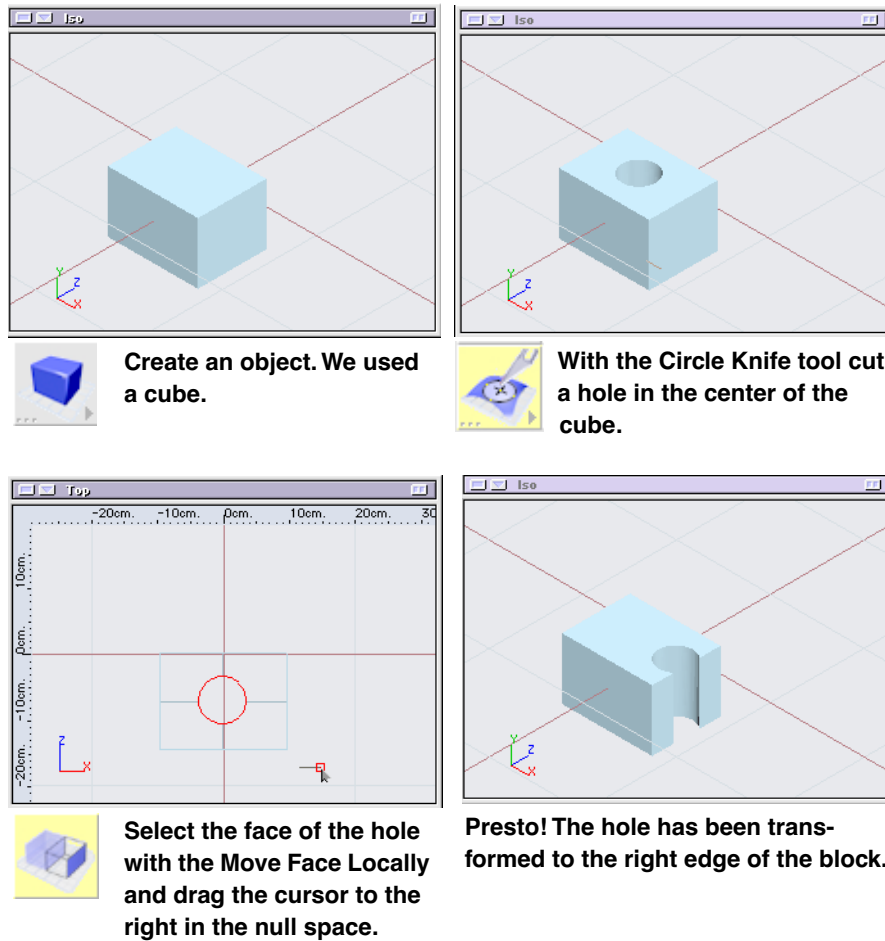
The local transform palette has only one tool in it, Move Face Locally. This tool allows you to grab a face and push or pull it from the model. The model will then be evaluated and its surface attributes will be extended to the new face.

To use this tool, click the Move Face Locally tool to make it active. Choose the face you would like a local transformation performed on by clicking on it. Click and drag in the direction you would like the local transformation performed. The local transformation is then performed.

This tool is great to extend an extrusion that might be too tall, or reduce one that might be too short.

**Note:** *Not all objects can have a local transformation applied to them. If a model's surrounding surface is very complex, then this transformation may fail.*

The Move Face Locally tool is a very powerful tool underestimated to do simple transformations similar to the Extrude tool. While the Move Face Locally tool can certainly extrude objects, it also can recede objects, relocate holes, continue the formation of cones, pyramids, and reform sliced objects such as a sliced torus. The following examples will give you the power of the Move Face Locally.

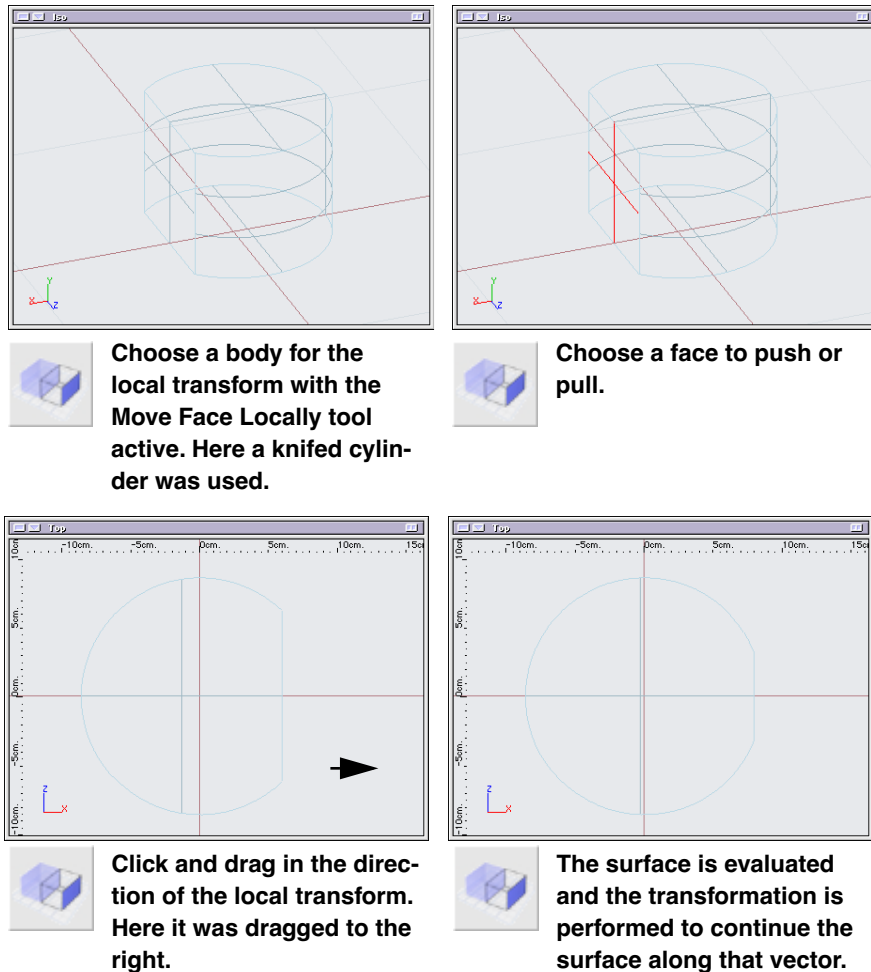


*Figure 21.0 — Using the Move Face Locally tool*

The Move Face Locally is limited by the topology of the object being transformed. For example, if you have a pyramid that was sliced too short off the top, it can only be extended to the point where the four sides would meet. If a semi-hole was cut into a cube at one of its edges, the hole can not

be transformed to enclose the cube. You will have to undo, to the point prior to the hole being created.

The Move Face Locally tool may not be used with UberNURBS.



*Figure 21.0 — Using Local Transform to extend a knifed cylinder*



## Booleans Palette

Boolean functions represent a powerful aspect of the modeling program. The program implements multi-functional Booleans. This means no matter whether objects are wire, surface or solid, all the Boolean tools work on them – mixed or matched.

The standard Boolean tool assortment represented: Union, Intersection, and Subtraction. In addition, four extra features exist: Stitch, Intersection Wires, Imprint and Derive Cross Section. The Stitch Tool unites surfaces along a common edge. The Intersection Wires (non-destructive) creates a wire shape where two shapes meet for further modeling. Derive Cross Section creates a wire on a user-defined cross section.

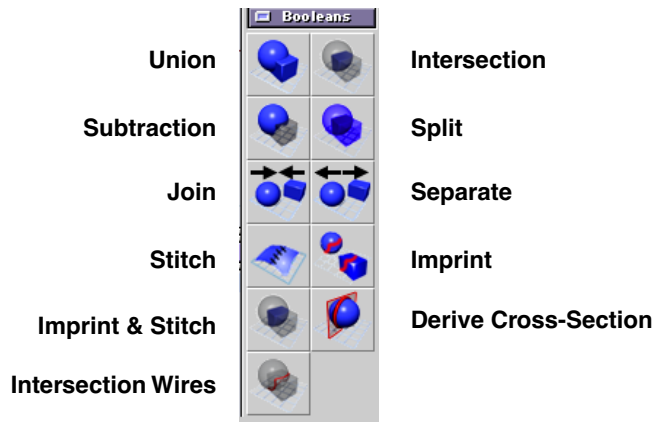
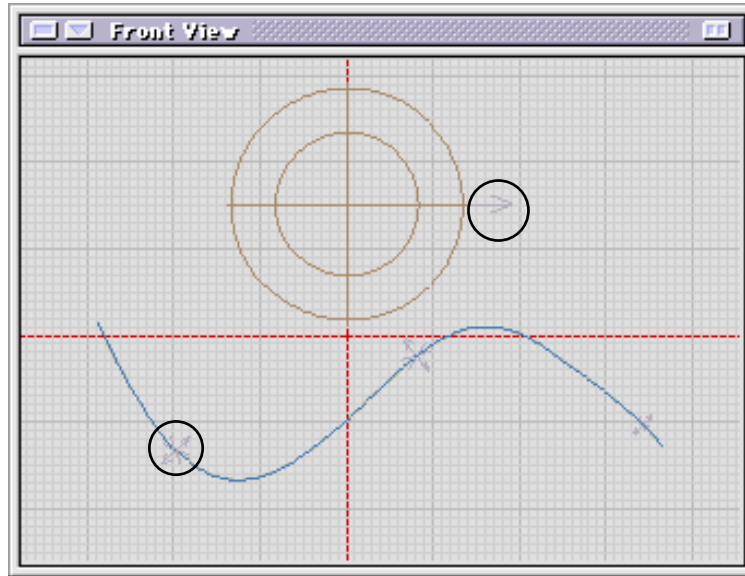


Figure 22.0 — Booleans Palette

The Union, Join, Intersection, Subtraction, Stitch, and Split tools are destructive. If you need a copy of the original object you are about to boolean, it is best to copy and save it on a separate layer.

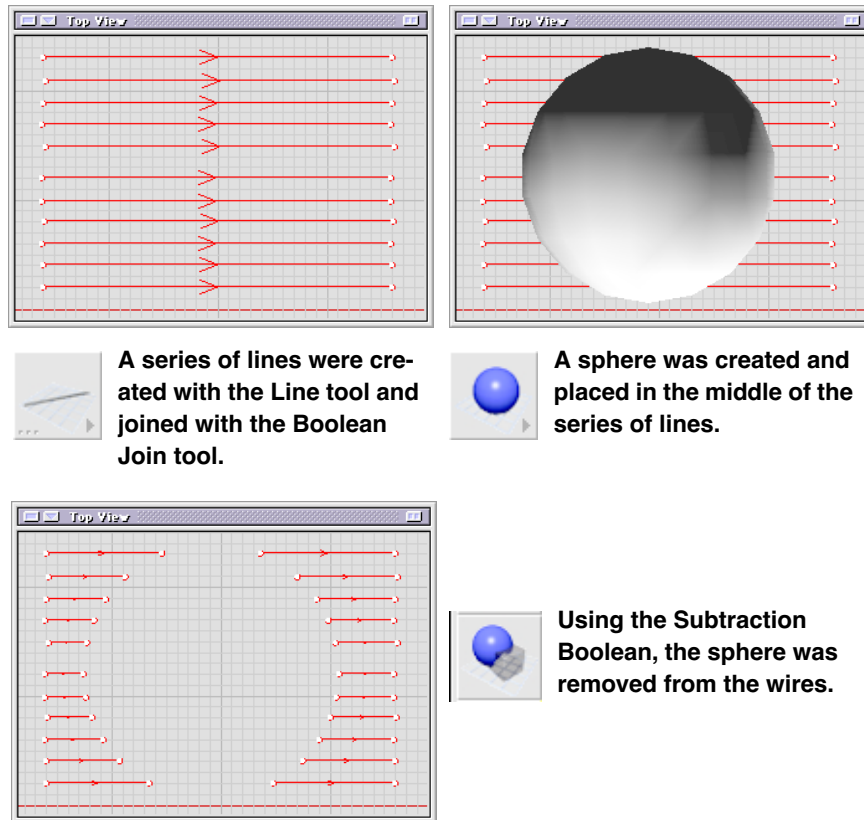


**By default solid objects (top) enter the scene as one-sided objects. New surface objects come in as two-sided. They will not change unless you use body or surface editing tools to change direction. Note the circled grey single- and double-headed arrows that show the normals direction.**

*Figure 22.1 — One sided objects and two sided surfaces*

For instance, using the Boolean Intersection removes the non-intersecting areas and leaves you with only the intersection.

New shapes created with the Boolean tool are placed in the currently active layer.



*Figure 22.2 — Demonstrating the multi-functional Boolean concept*

## 22.0 Boolean Operations on Single and Double-Sided Solids

All solid shapes are inherently one sided. If you remove a one-sided shape from another one-sided shape, it will create a solid one sided shape. However, it is possible to use the Make Two-Sided tool in the Body Editing palette to change a shape from its original one-sidedness and turn it into a two sided object. Before any editing is done, the shape will look the same, but in

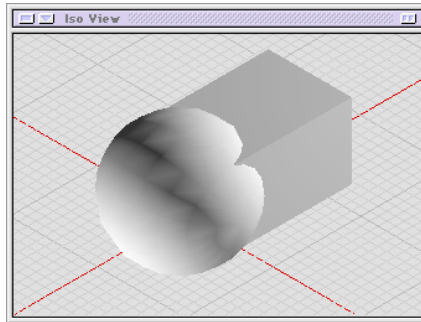
its topology it now has normals on the inside and outside of the shape. In short, the computer now recognizes that it has more to deal with than just the surface of the object.

As shown in Figure 22.3 a Boolean subtraction removes the one-sided sphere from the two-sided block leaving an open surface object.

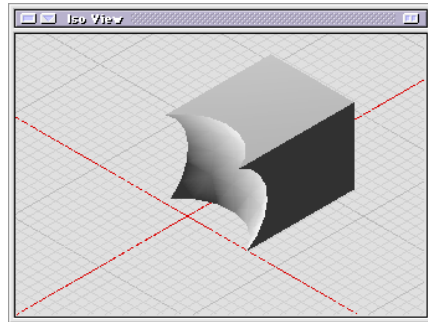
The process works the other way as well. If you remove a two-sided surface object from a solid one-sided object, you can slice into an object. While the immediate effects may not be dramatic, a new topological formation allows for further editing.

The truly hybrid Boolean tools give you the freedom to edit shapes as you wish, without the restrictions of object types. For example, you could remove a wire from a solid object. While this is not practically useful, the point is that you can mix and match Boolean functions without limitations. You can, for instance, trim your wires to the boundaries of a shape by subtracting the shape from the wire. See “Demonstrating the multi-functional Boolean concept” on page 363.

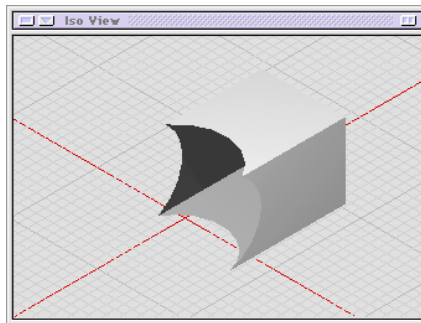




**Two solid primitives, before any side-changing or Booleaning is performed.**

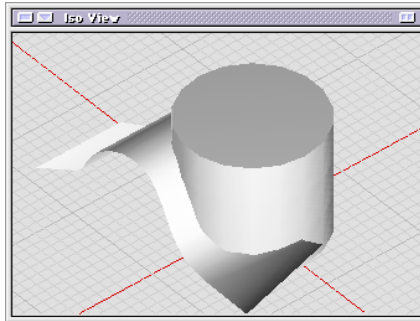


**Without altering the body topology, both sides are inherently one-sided. A subtraction Boolean shows how the two would interact. The result is a solid shape.**

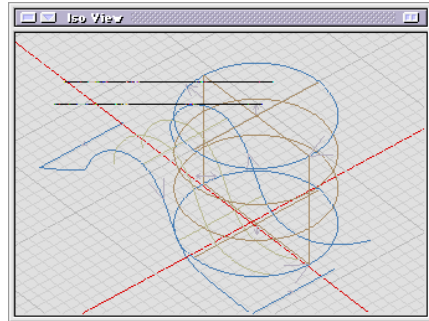


**By making the cube two-sided, it now has an “inside.” The subtraction of a solid one-sided shape shows the resulting surface object.**

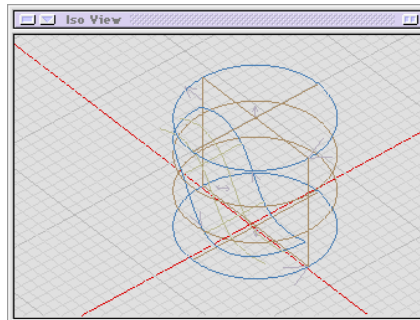
*Figure 22.3 — Booleans are one and two-sided primitives*



The surface object, like all surface objects, is two-sided. The solid object is one-sided.



The surface object, like all surface objects, is two-sided. The solid object is one-sided.



The resulting shape is a one-sided object with a “slit” cut into it.

*Figure 22.4 — A Boolean Subtraction between two-sided surface objects*

### 22.1 Boolean Union Tool

Two objects can be melded together as one using the Boolean Union tool.

Reasons to Boolean two objects together:

- Create a more complex shape from simple primitives
- A single object can take a single, continuous texture map.
- By using the Blend tool after the Boolean, the area around the unifica-

tion can be altered to show a smooth transition between the two objects.

- Two or more objects that have been unified can later have other Boolean and Knife functions performed to the group as a whole.

Any number of objects can be unified. Choose the tool and begin clicking on the objects. As each is selected, they will turn into the selection highlight color. When you have clicked on all the objects you want to unite, double-click in null space.

The new shape is placed in the first picked object layer.

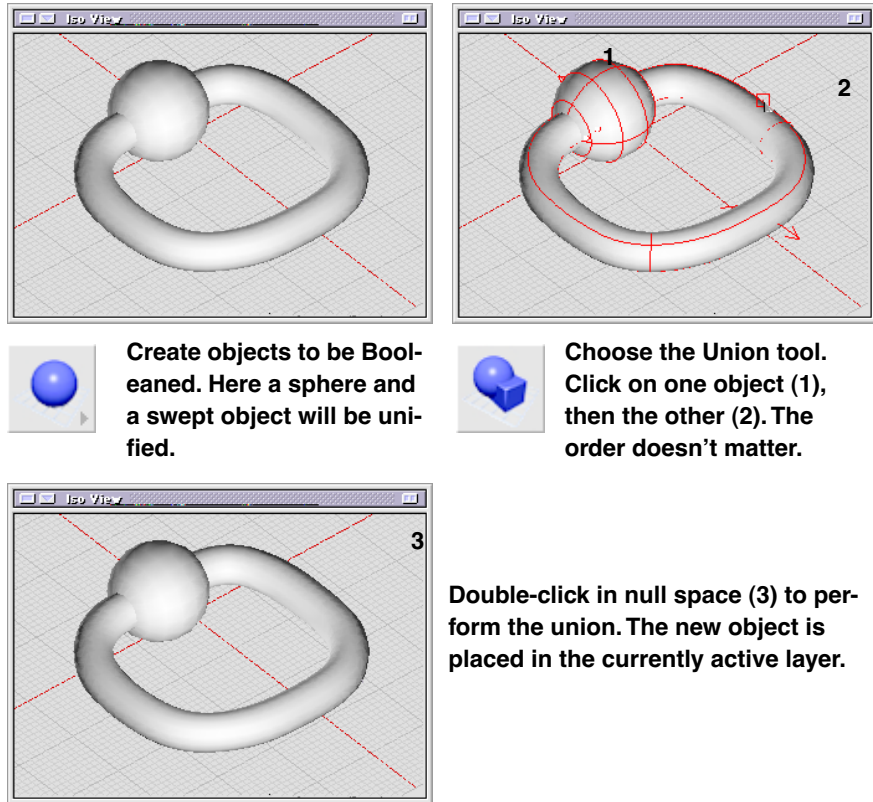


Figure 22.5 — Melding two objects with the Boolean Union

## 22.2 Boolean Subtraction Tool

Chop, hack and whittle away at shapes using other shapes with the Subtraction tool. Any type of object can be removed from any other type of object without restriction. Any number of objects can be removed from another object at the same time.

To remove one shape from another, click on the base shape (the shape you want to edit) first, then the editing shape (the shape that will soon become a hole in the object). Double-click in null space to perform the function.

To remove a series of objects from an object, click on the base shape and click on all the editing shapes, one at a time. When finished with your selection, double-click in null space.

In either case, the new shape is placed in the currently active object and the editing shape disappears. If you need the editing shape, make a copy of it before doing the Boolean. For example, if you had a bolt and wanted to use the bolt to remove the insides of a nut to get the same threads, make a copy of the bolt before the Subtraction.

Keep in mind the one-sided / two sided issues (“Boolean Operations on Single and Double-Sided Solids” on page 363) when performing the Subtraction boolean.

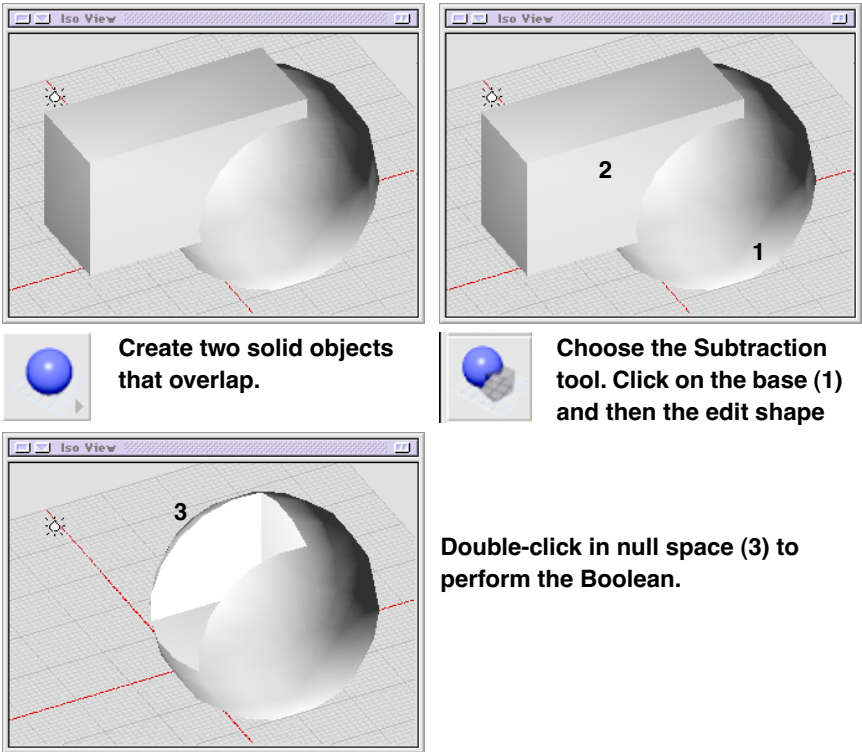
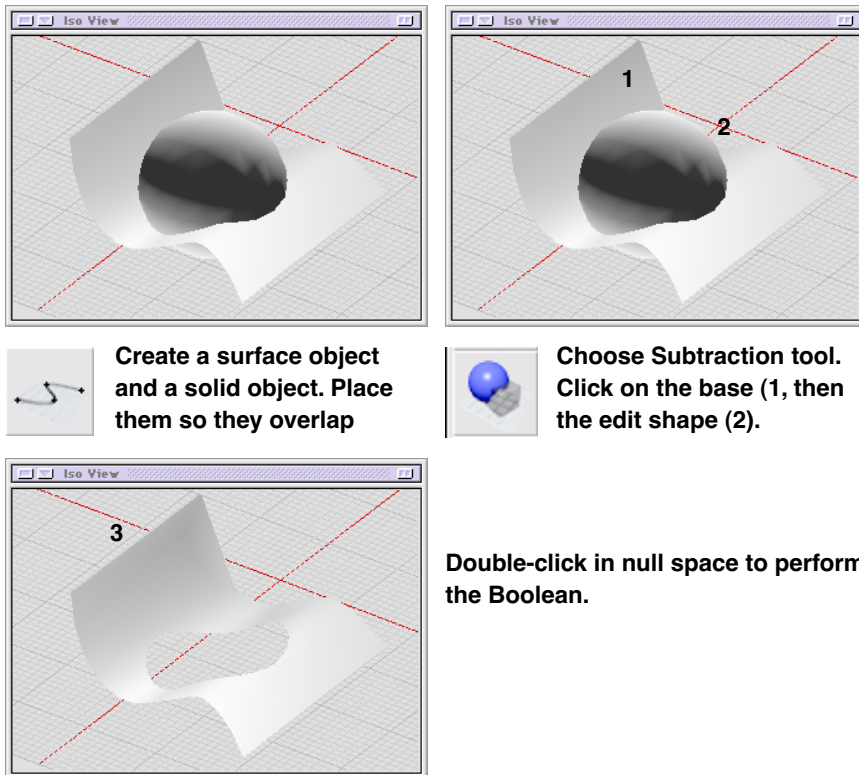
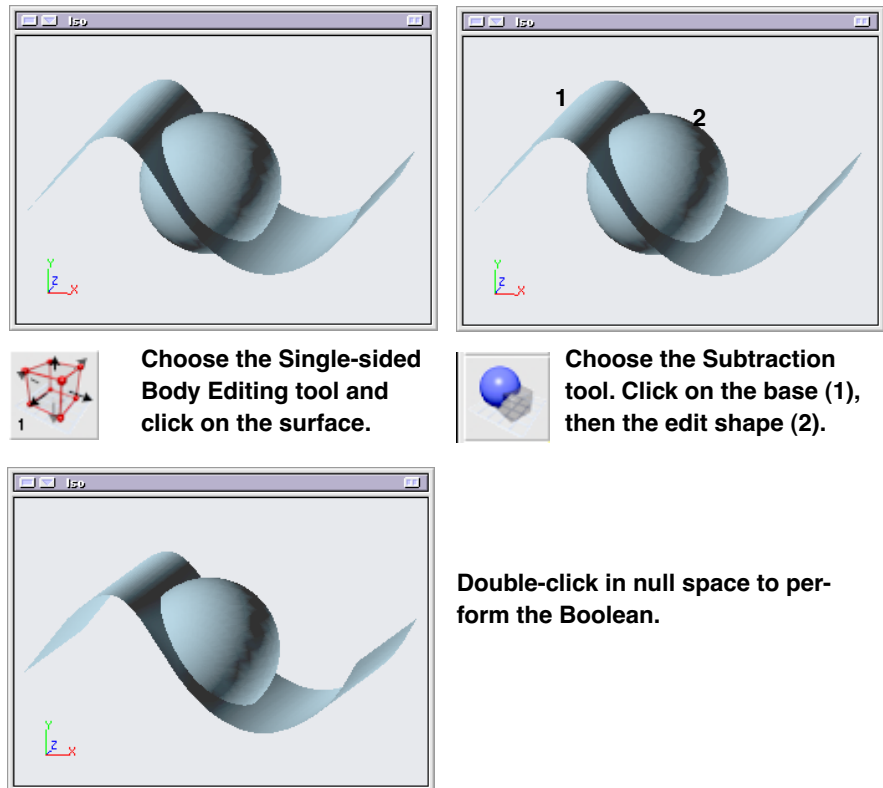


Figure 22.6 — Subtracting a one-sided solid object from a one-sided solid

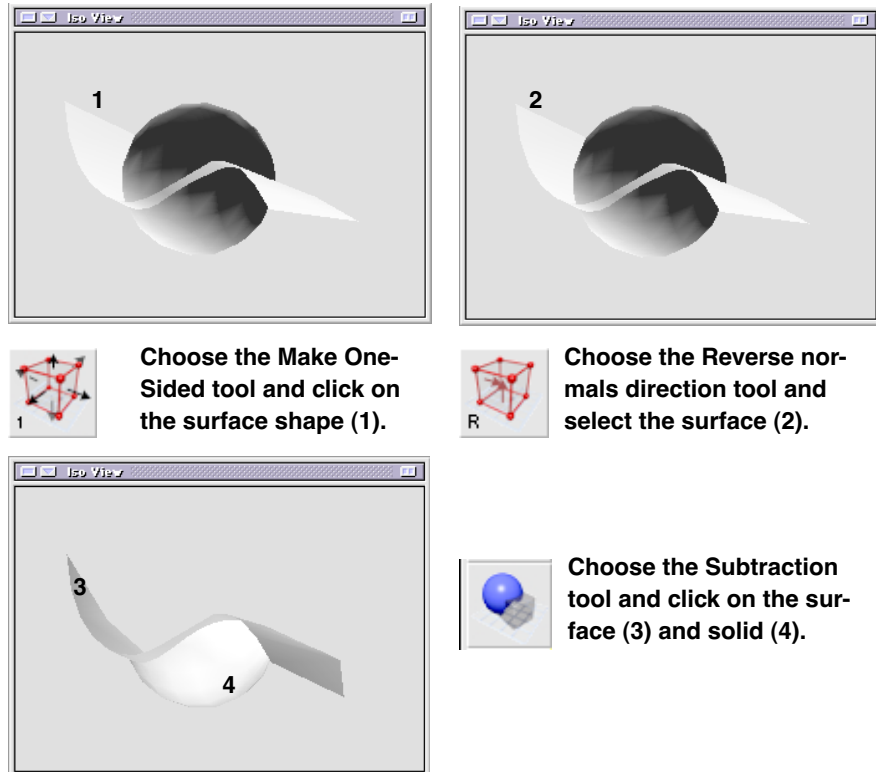


*Figure 22.7 — Subtracting a one-sided solid from a two-sided surface*



**Figure 22.8 — Subtracting a solid object from a one-sided surface with normals pointing down**

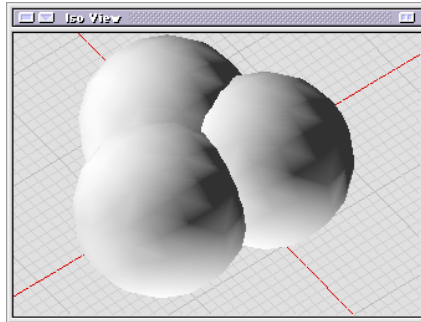




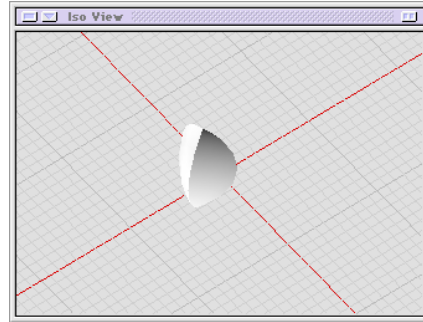
*Figure 22.9* — Subtracting a solid object from a one-sided surface with normals pointing up.

### 22.3 Boolean Intersection Tool

The Intersection tool answers the question “what part of these objects overlap?” Two or more objects can be used to create the intersection. This includes solid, surface, and wire objects.



Several objects can be used when working with the Boolean Intersection tool.

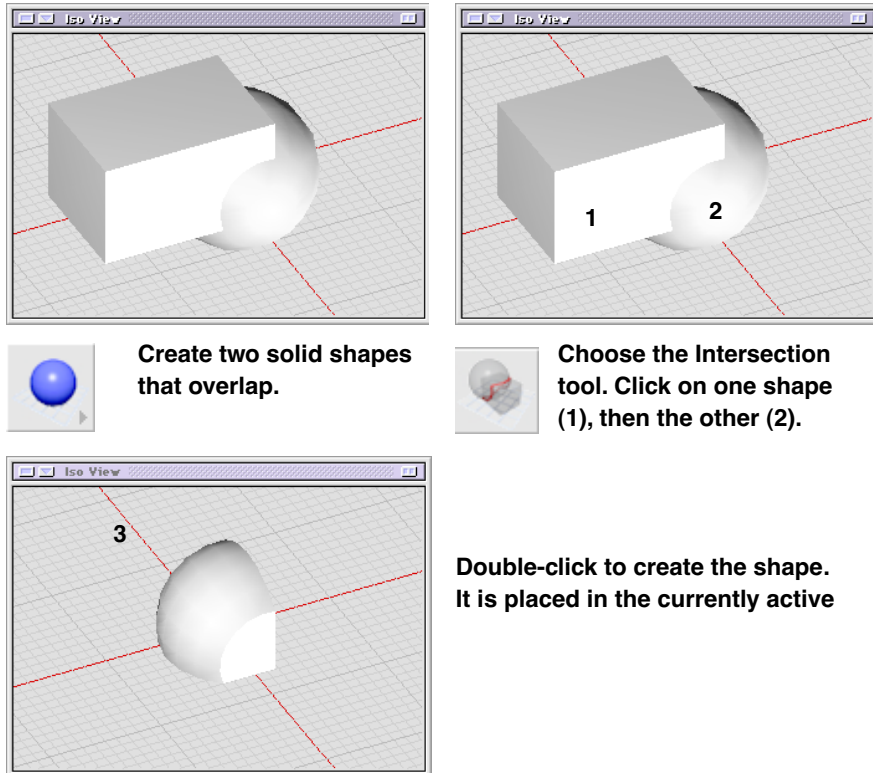


Using the Intersection tool, only the area that is overlapped by the three spheres remains.

*Figure 22.10 — Boolean intersections*

To use, choose the Boolean Intersection tool and click on the objects to be included in the intersection. After clicking on the last object, double-click in null space. The order of selection doesn't matter, nor does the number of objects used to create the intersection. The new shape is placed in the currently active layer. The original shapes will disappear.

If you need the original shapes, make a copy of them before doing the Boolean or you may want to use the Split tool. This tool gives you the same resulting shape as the Intersection tool, but does not remove the non-intersecting portions of the shape.



*Figure 22.11* — Creating an intersecting shape from two solids.

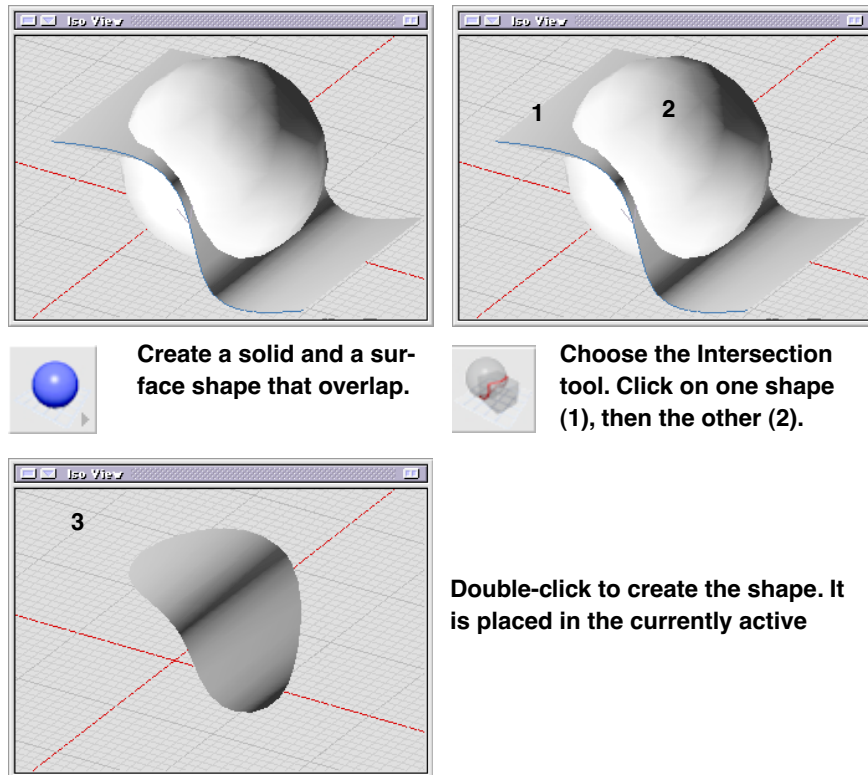


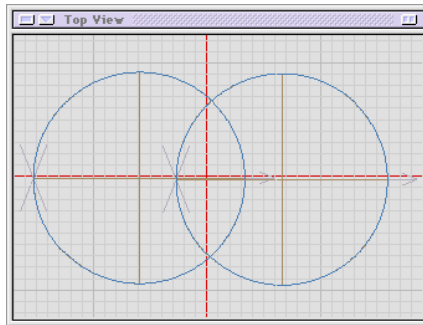
Figure 22.12 — Creating an intersecting shape from a solid and a surface

### 22.4 Boolean Split Tool

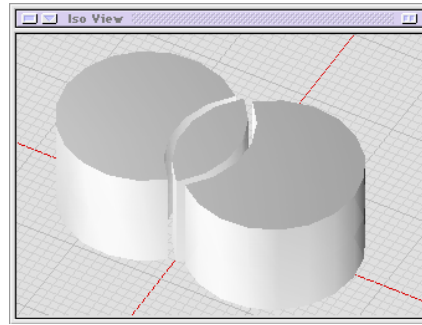
The Boolean Split tool works similar to the Boolean Intersection, except the original shapes remain. The Split works only with two shapes. If you want a complex shape made from several different objects, Union them first.

To use the Boolean Split tool, create two shapes that overlap. Choose the tool and click on one, then the other object. Double-click in the null space to

perform the Boolean. The order of selecting the objects does not matter. The result is three separate, new shapes. They are placed in the first picked object layer.



**Two overlapping cylinders shown before the Boolean operation.**



**After the Boolean split is performed, three new shapes are formed. (NOTE: The shapes have been moved slightly to make the concept clearer.) The split is reflected in the layers (above).**

*Figure 22.13 — Boolean split layer*

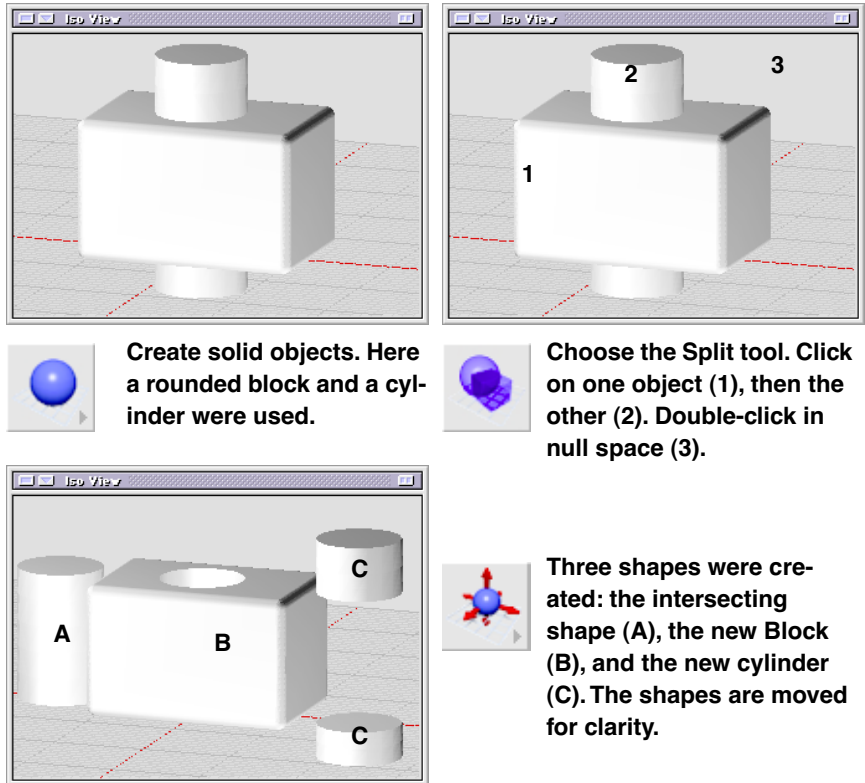


Figure 22.14 — Splitting solid objects using boolean split tool

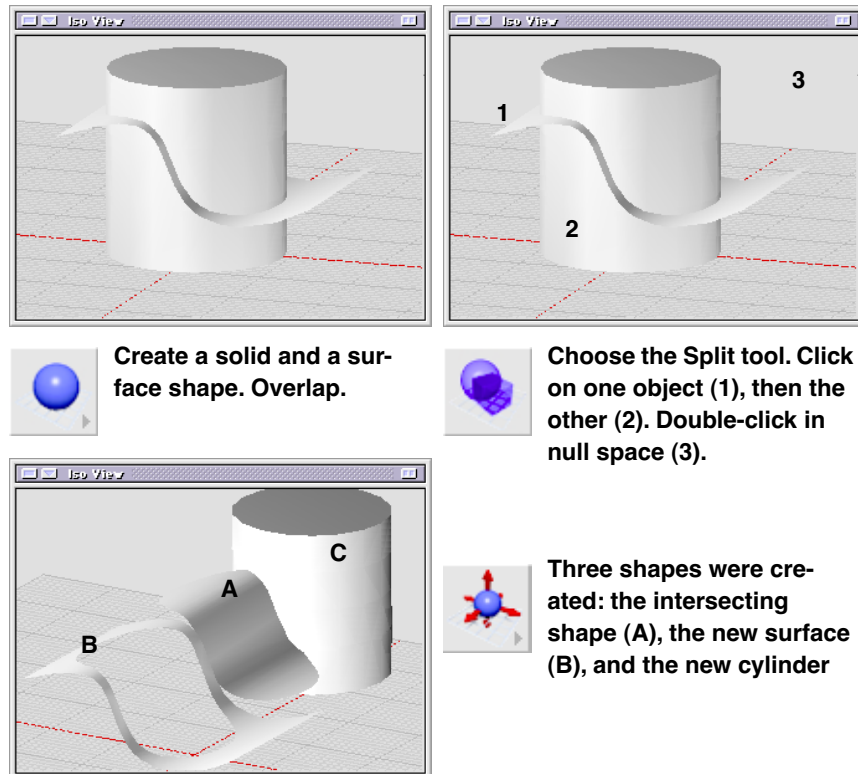


Figure 22.15 — Splitting a solid and a surface object using the Boolean Split tool

## 22.5 Boolean Join Tool

Boolean Joining objects is a non-destructive way to unite two or more objects without losing their original topology. Joining is like “grouping” in other programs. While the geometry of two objects that are joined would be identical to that of two objects that were “Unioned,” the information about the separate shapes remains intact. Because of this, you can later separate them, unlike the Union Boolean.

This makes the Boolean Join an excellent organizational tool for combining elements that will have a similar fate, such as placement in the scene or the same texture map. However, because of the topological differences, you need to be aware of the differences between the Join and Union tools. Joined objects cannot have their “joining” edges blended like a Unioned shape. Joined objects will not perform other Boolean functions together, like a Unioned shape. Any number of objects can be Boolean joined at the same time.



*Figure 22.16 — Boolean Join tool*

**The user can pick, or pre-pick with either tool. To pre-pick, select the Pick tool and choose all the objects to be joined or separated. Then select the tool and double-click in null space.**

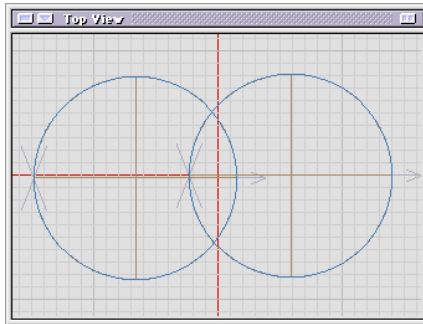
### 22.6 Boolean Separate Tool

Choosing the Boolean Separate tool and clicking on an object created by the Boolean Join tool separates the objects back into their original shapes. If a series of objects were created with several joins, all the joins will be broken at the same time. Objects created with either tool are placed in the same layer as the joined object.

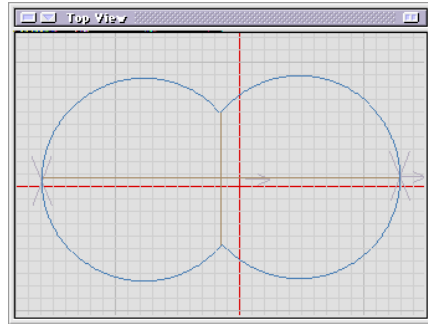


*Figure 22.17 — Boolean Separate tool*

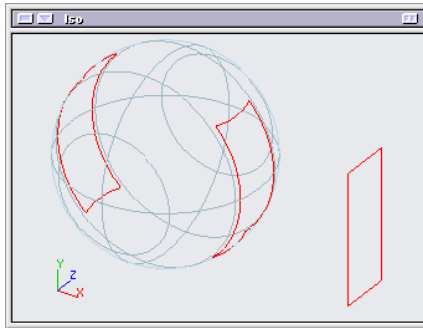




The two cylinders above were brought together as a single object using the Join tool. They retain their topology.



The two cylinders above were brought together as a single object using the Union tool. Their original topology is lost and a new shape is created.



The Boolean Separate tool can also be used when various construction tools, like the wire projection, creates a body with two lumps. The two projected wires can be disconnected from each other using the Boolean Separate tool.

Figure 22.18 — Using the join tool, union tool, and separate tool

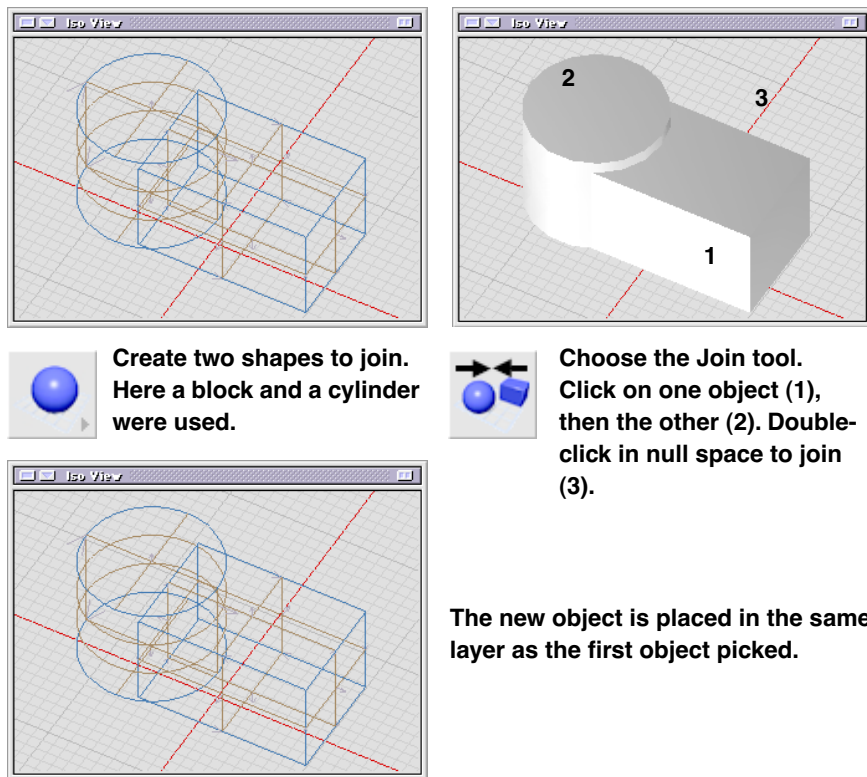
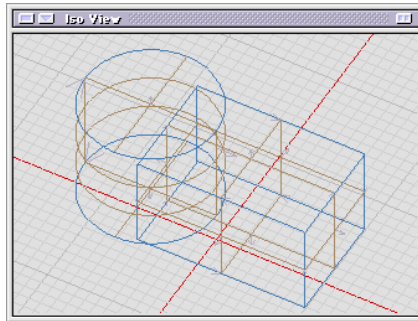
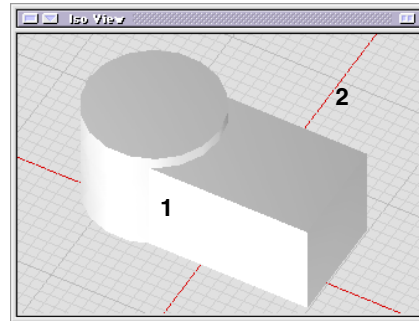


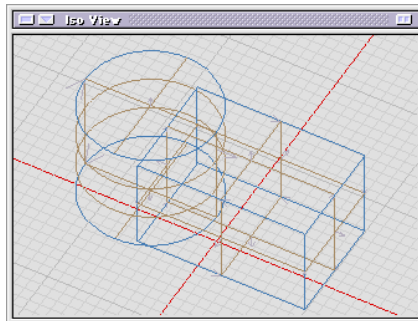
Figure 22.19 — Joining two objects together



**Choose the Separate tool.**



**Click on the object to be separated (1), Double-click in null space (2).**



**The original objects are placed in the currently active layer.**

*Figure 22.20 — Separating objects*

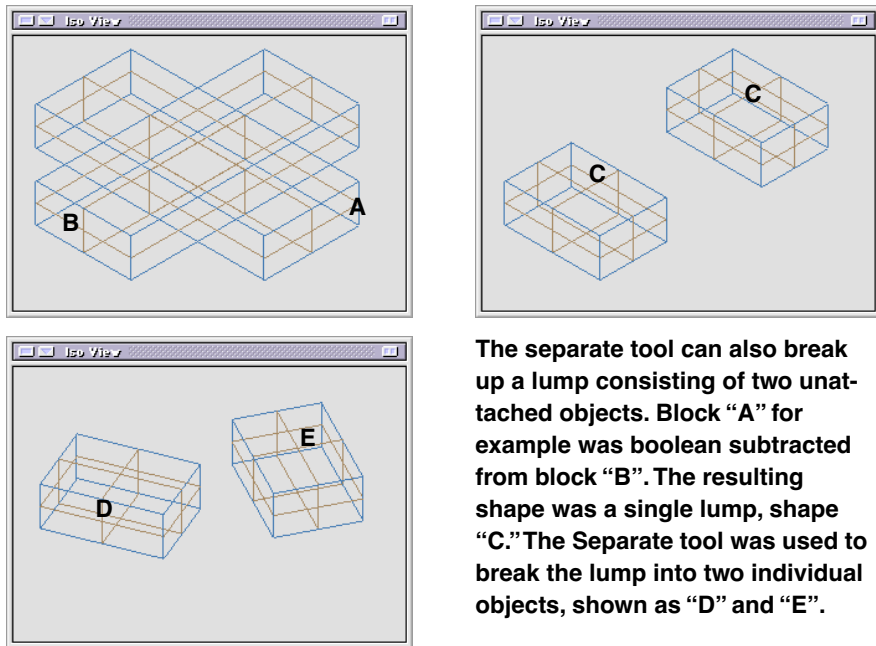


Figure 22.21 — Separating booleans

### 22.7 Boolean Stitch Tool

The Boolean Stitch tool was designed to bring together objects with a shared edge. Any number of surface objects can be brought together.

The trick here is the edges to be stitched need to be exactly the same. Fortunately, the Skin tool can use existing edges of other surfaces, including other skinned surfaces to create a skin, hence creating co-edges that match up perfectly. As long as the edges match, a solid object can be stitched with a surface object.

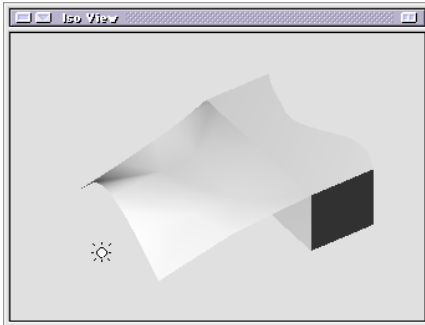
Stitched objects become multiple faces of one body, and the original surface shapes are lost. Be sure to duplicate the original objects if you think you'll need them again in the future.

**Note** Single sided faces that are being stitched must be facing the same direction.

Using the pre-pick mode is not recommended with this tool. Instead, select surfaces that share common edges consecutively after activating the tool.

To use the tool on two objects, choose the Stitch tool, click on one surface, then click on the other. Double-click in null space and the Stitch will be performed. The new shape is placed in the same layer as the first object picked.

Stitching does not change the tangent or curvature of the stitched faces. Stitching is important when faceting is required to match polygon vertexes on either side. Two stitched faces, when faceted, will give matched vertexes. The faceter knows that two faces share an edge. If it doesn't, it will produce facets for each face independently and seams will show when shaded.



A skinned object that was created using the edge of a solid object can be stitched together.



Single-sided faces that are to be stitched must be facing the same direction, the two surfaces above can't be stitched because they are facing different directions.

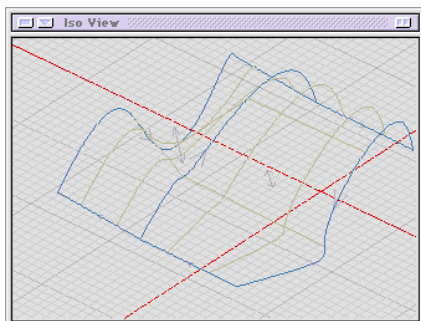


Boolean Stitch tool

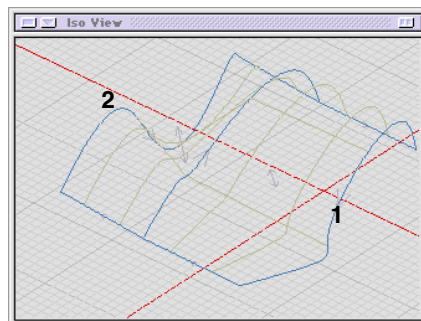


Left, the Reverse Face tool was used to correct the problem.

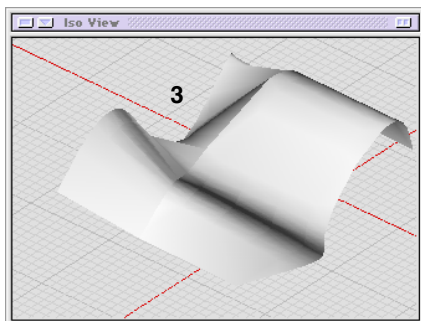
Figure 22.22 — Boolean Stitch tool



Create two surface shapes with the same edge. Here the Skin tool was used.



Choose the Stitch tool. Click on one surface (1), then the other (2).



Double-click in null space (3). The stitch is performed.

Figure 22.23 — Stitching two surface objects

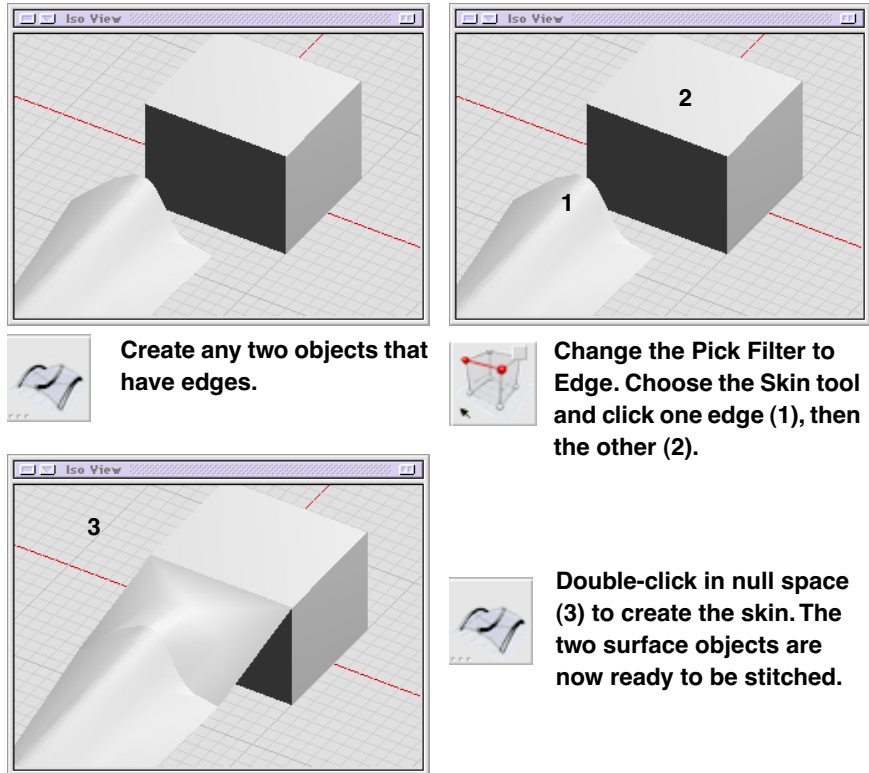


Figure 22.24 — Creating matching edges from existing shapes.

## 22.8 Boolean Intersection Wires Tool



Figure 22.25 — Boolean intersection wires



This tool is a non-destructive tool whose job is to create a wire body at an intersection point of two objects.

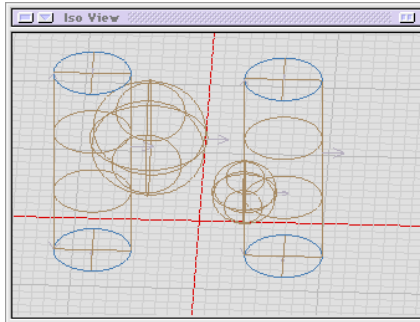
The tool can be used to create wire shapes that otherwise would be difficult or impossible to generate. Wire bodies created with this tool can go on to be the basis of any other editing feature, such as skinning, lofting, or extrusion.

While only two objects can be used, they can be either surface or solid shapes – in any combination. Solids and solid, solid and surface, or surface and surface combinations work.

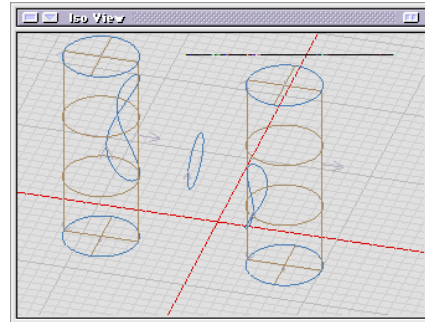
To use the tool, create two objects with some overlap. Choose the Intersection Wires tool. Click on one object, then the other. On the second selection, the wire is formed automatically. Neither shape used in creating the intersection is altered.

The wire created by the intersection can be treated like any other wire object and can be edited using the Bezier or NURBS editing tools.

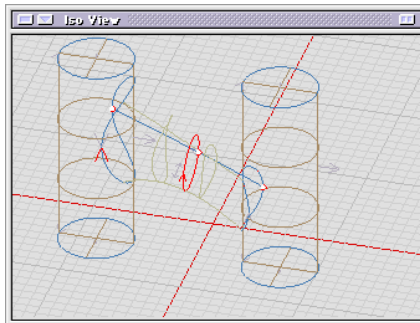
**Note** *Figure 22.26 an example of how the Intersect Wires tool can be used in conjunction with other tools to create custom shapes.*



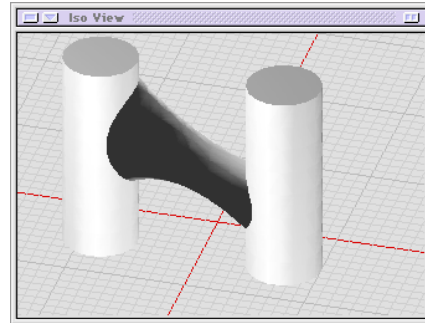
To create a connecting form between two cylinders, two spheres were created that overlapped the cylinders.



The Intersection Wires tool was used, creating wire bodies on each cylinder. An additional circle wire body was created between the two cylinders.

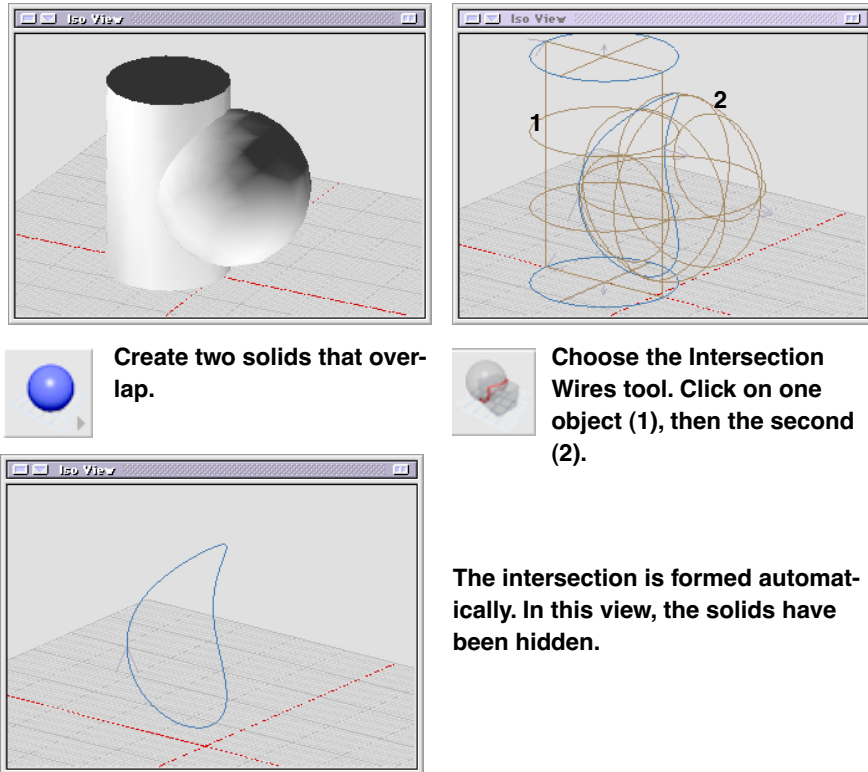


The Skin tool was used to create a skin object between the three wire bodies.



The resulting shape is an organic connecting shape that was designed around the contours of the original cylinders.

*Figure 22.26 — Using intersection wires tool to connect shapes*



*Figure 22.27 — Creating an intersection wire from two solids*

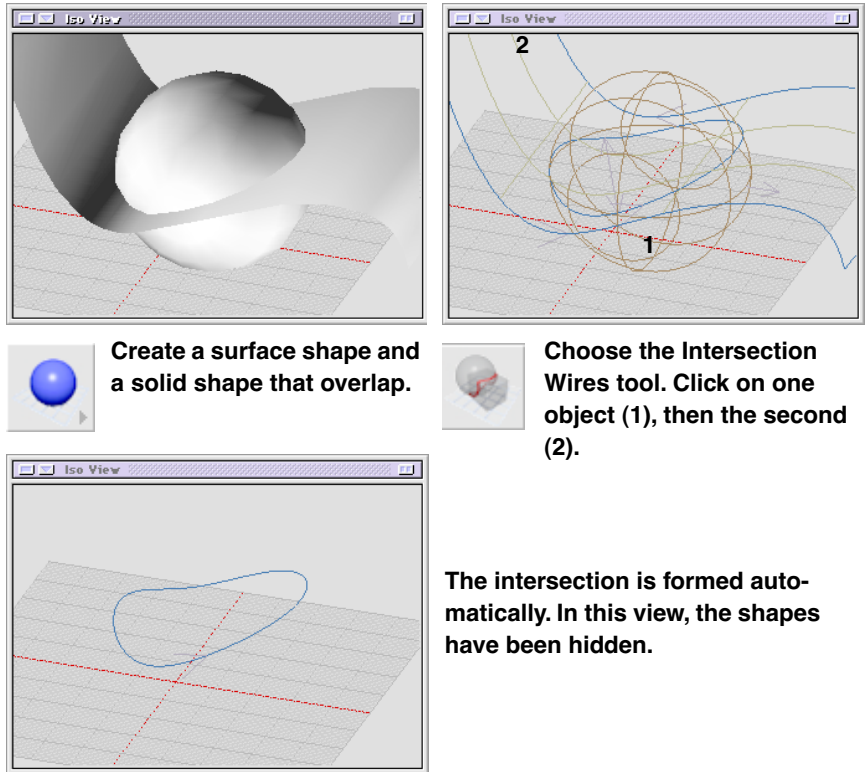


Figure 22.28 — Creating an intersection wire from a solid and a surface

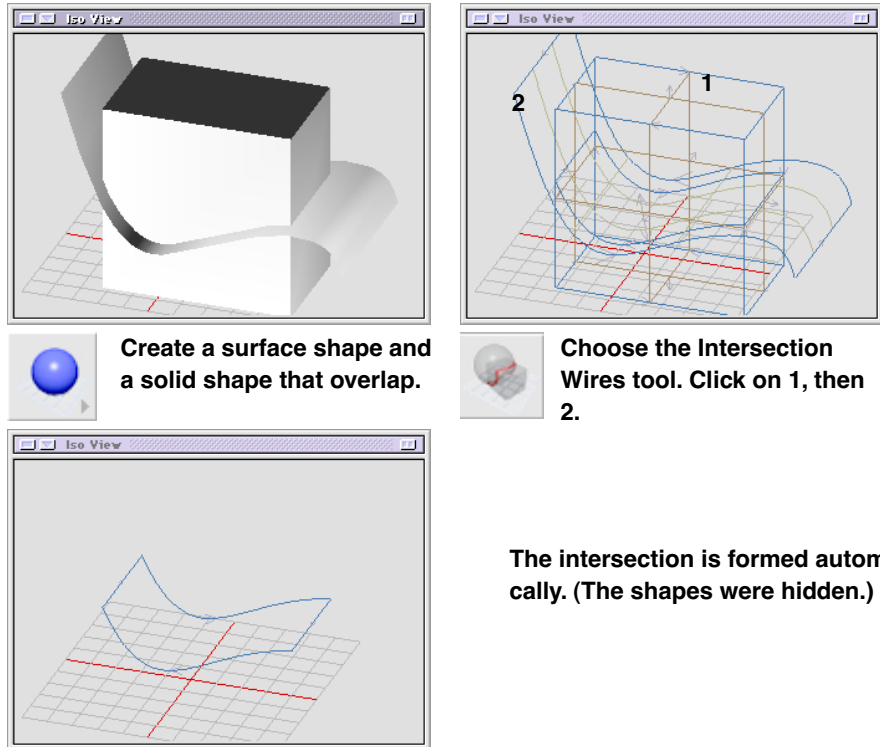


Figure 22.29 — Creating an intersection wire from a solid and a surface

### 22.9 Derive Cross-Section Tool

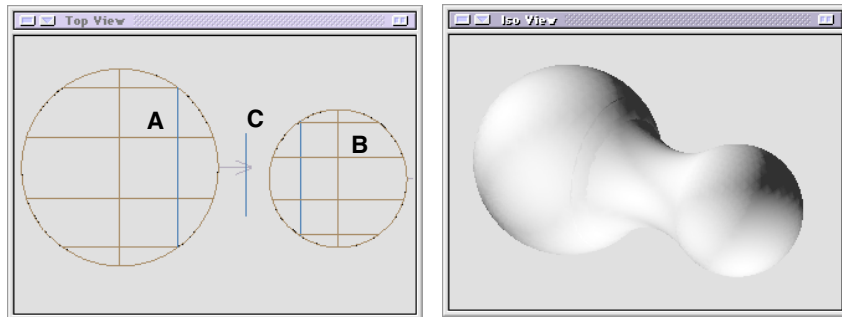
This tool creates a cross section of a solid or surface object along a plane that the user defines by drawing a line through that object. The resulting cross section is a wire object that can be used either for shape analysis or for continued construction.

As an analytical tool, the Derive Cross-Section tool can map out exactly what a shape is doing at any given point along its surface.

As a construction tool, the resulting wire can be edited with the Bezier or NURBS editing tool, or it can be used with other tools such as extrude, fill, or skin.

To use, choose the tool and click once on the object that will be used to determine the cross section. Next, click down on the mouse and drag across the object without releasing. Release the mouse when the cross section is done. A new wire shape is formed automatically and placed in the currently active layer. The new wire is then ready for editing.

The tool can be used to meld two objects together. In Figure 22.30 two spheres were used with the tool. A copy of one of the new wires was scaled down between the two and the resulting wires skinned.

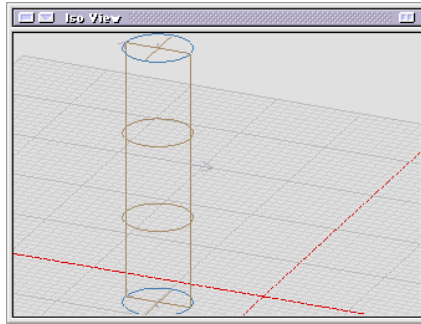


The Derive Cross-Section tool was used on the two spheres, creating 2 new wires (A,B). A copy of a wire was scaled in the middle (C).

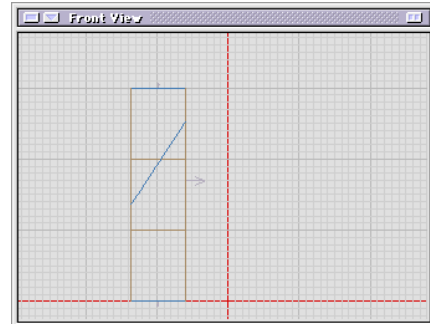
The wires were then skinned.



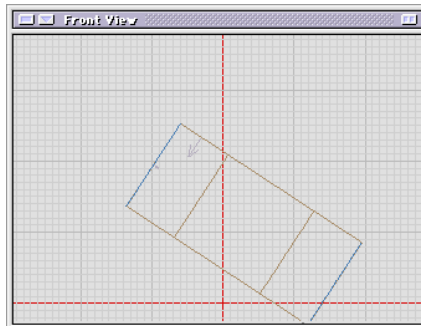
*Figure 22.30 — Derive Cross-Section*



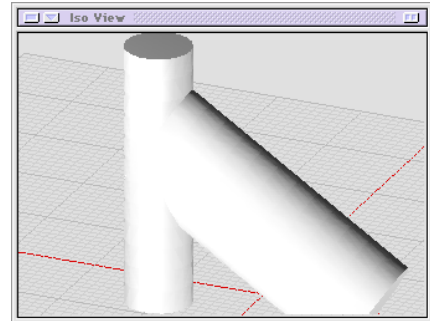
The goal is to create a pole that has a diagonal brace. The Cylinder Primitive tool is used to create the primary pole.



From the front view, the Derive Cross-Section tool is used, creating a strong diagonal wire element.



With the original cylinder hidden, the new wire is given a face by using the Fill tool in the Wire Editing palette.



The resulting shape is a brace that fits exactly with no sloppy edges.

*Figure 22.31* — Creating a brace with the Derive Cross Section tool.



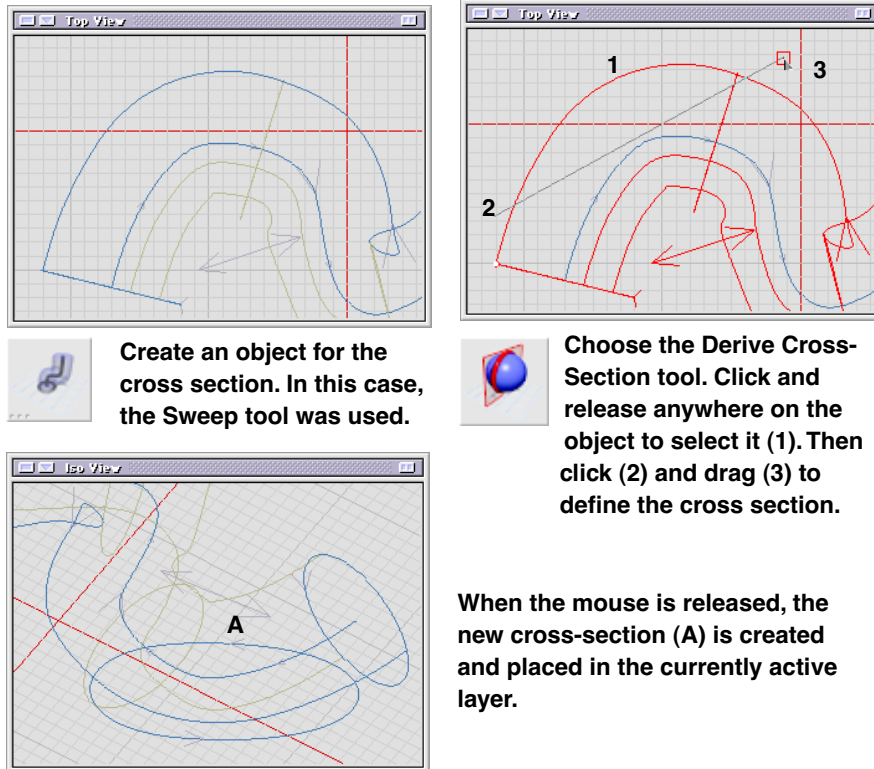


Figure 22.32 — Creating a cross-section on a solid object

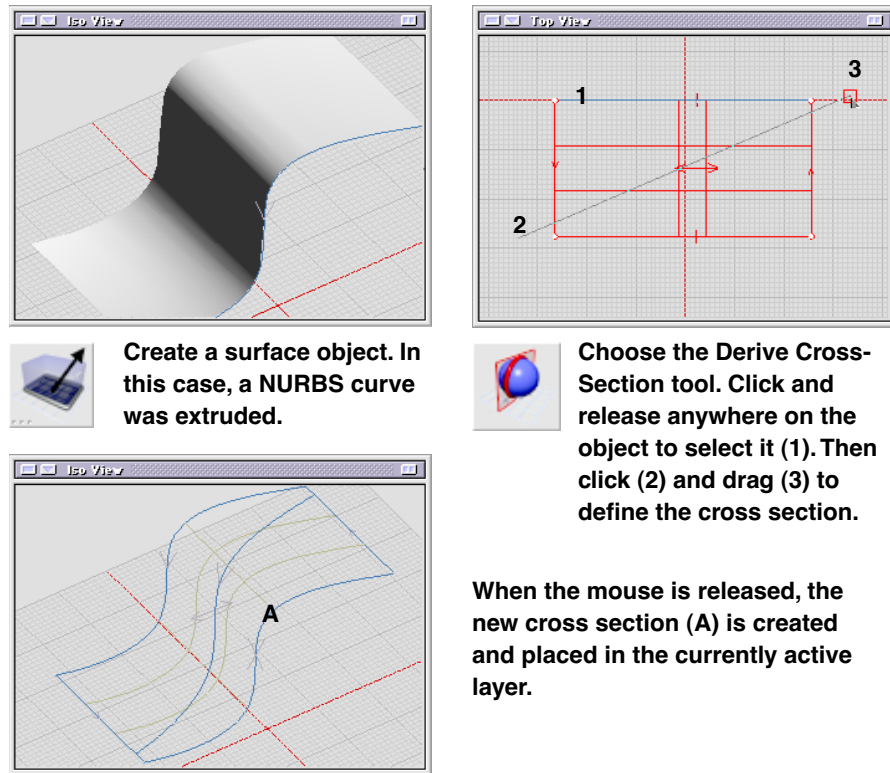
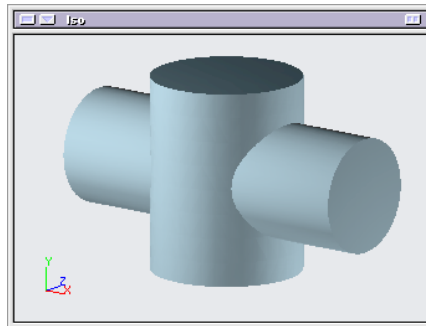


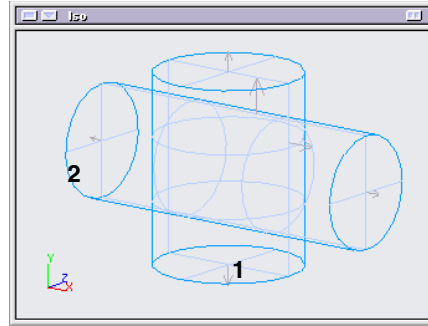
Figure 22.33 — Creating a cross section on a surface object

### 22.10 Imprint Tool

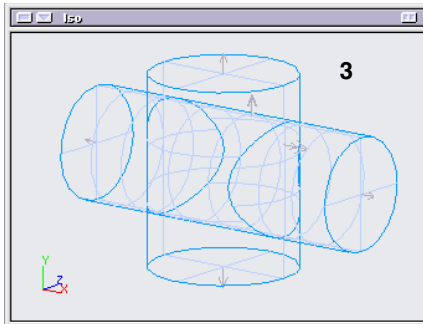
The Imprint tool uses existing shapes to add new information. In this case, the new addition isn't an additional wire object, but is an alteration of the topology of the interacting shapes. Two overlapping shapes using this tool will gain a new loop at the point of intersection, but without altering the overall geometry of the shape.



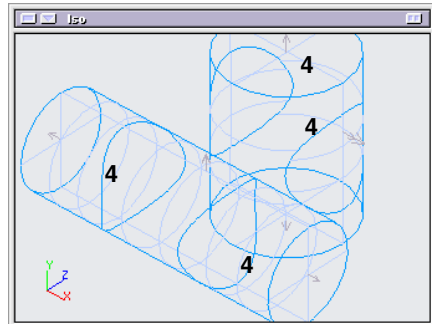
Create two solid objects that overlap. Here two cylinders were made.



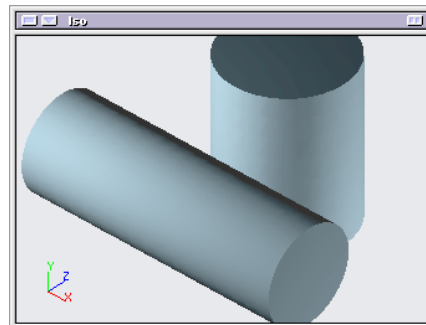
Choose the imprint tool, click on the first object (1) and then the second (2).



Double-click in null space (3) to perform the Boolean.



Use the move tool and separate the objects, you'll see the new loops (4).



While the shapes are now topologically altered, geometrically they are the same.

Figure 22.34 — Changing topology with the imprint tool.

### 22.11 Imprint & Stitch Tool



*Figure 22.35* — **Boolean imprint & stitch tool**

The Imprint & Stitch tool uses existing shapes to add new information. In this case, however, the result isn't an additional wire object, but is an alteration of the topology of the interacting shapes. Two overlapping shapes using this tool will gain a new loop at the point of intersection.

The two separate elements become multiple faces, but the original surface shapes are lost. Be sure to duplicate the original objects if you think you'll need them again in the future. Any number of surface objects can be brought together.

The trick here is the edges to be stitched are created from the imprint and are exactly the same. Fortunately, the Skin tool can use the imprint edges of the surfaces, including other skinned surfaces, to create a skin, hence creating co-edges that match up perfectly. As long as the edges match, a solid object can be stitched with a surface object. Single sided faces that are being stitched must be facing the same direction.

To use the tool on two objects, choose the Imprint & Stitch tool, click on one surface, then click on the other. Double-click in null space and the Imprint & Stitch will be performed. The new shape is placed in the currently active layer.

## UberNURBS and UberMeshes

Universe Modeler allows you to create shapes with solids, UberNURBS or surfaces. You can use any method that you choose to create a particular shape, and each method can work with each other in Universe Modeler. This section will cover UberNURBS.

### 23.0 What are UberNURBS and UberMeshes?

UberNURBS could be considered a modeling program within a modeling program. More than just a tool, UberNURBS is an entire shape creation system. UberNURBS allows for the deformation of shapes with an incredible amount of flexibility. Users can define which areas of an object they can push and pull and create complex organic shapes, like virtual modeling clay.

UberNURBS is the term used by Universe Modeler to describe the modeling technique known as “subdivision surfaces.” This technique allows you to create a fairly coarse control cage, and interpolate a smoother, higher-level shape. This is a very popular technique for quickly and easily creating very complicated shapes such as characters, with a minimum of modeling time. UberNURBS is a resolution independent object format, and can easily be converted into ACIS class objects for further editing, such as booleans or rounding.

UberMeshes are partially-implemented Catmull-Clark meshes, made popular by “Geri’s Game,” a computer animated short film by Pixar. These meshes are polygonal mesh objects, and as such are not compatible with certain ACIS tools, such as booleans or rounding (if you need these functions, just convert the UberMesh object back to an UberNURBS object).

UberMeshes are primarily for export, although they can be edited by the UberNURBS toolset at any time.

### **UberNURBS and Universe Modeler (ACIS) Environment**

UberNURBS can co-exist in the Universe Modeler (ACIS) environment of the Universe Modeler with other ACIS objects. Normally, UberNURBS objects are not stored as ACIS objects. If you perform an operation that requires the UberNURB body to be fully converted into an ACIS object (for booleans and rounding, etc.), a dialog box will appear to inform you that a conversion is about to take place, and will let you cancel the operation if you want. Once you perform a boolean, knife, round or any other ACIS operation which changes the topology of the UberNURBS object, you may be prevented from any further UberNURBS editing of the object.

### **UberMesh and Universe Modeler (ACIS) Environment**

UberMesh objects are not compatible with any Universe Modeler mode (ACIS) operation that would change the topology of the UberMesh object. If you need those ACIS operations, you can go back into UberNURBS mode, and change the object into an ACIS object from there.

You can use ACIS operations to move, rotate, scale and mirror your UberMesh objects. This is especially handy when you are building an UberMesh or UberNURBS object with “sub-patch” modeling techniques (where you build an eye or an ear in separate pieces, and then join them together later.)

### **When Should You use UberNURBS?**

UberNURBS are great for any modeling task that can best benefit from subdivision surfaces. There are many books and articles which cover modeling

with subdivision surfaces, and we recommend that you seek them out for reference. In the meantime, here are a few suggestions for modeling with UberNURBS:

- Characters and Anatomy
- Shapes friendly to facial animation
- Sculptural shapes
- Shapes that are hard to make with solids or NURBS surfaces

### When Should You use UberMeshes?

UberMeshes are best used when the output of UberNURBS is not to your liking. UberMeshes tends to smooth out little creases that can sometimes occur with UberNURBS models. As mentioned previously, UberNURBS and UberMeshes are compatible, and can easily be converted from one to the other.

### Sub-Patch Modeling

Sub-patch modeling is a modeling technique that allows you to construct your model in parts or pieces, joining together various parts at the appropriate time. This technique allows you to reuse favorite parts, or to edit a pre-existing part into something that you need. UberNURBS and UberMesh allow you to model in this fashion.

### How to Create and Edit UberNURBS

In Universe Modeler, you must first start off with a suitable ACIS object, and then convert it by choosing the UberNURBS edit tool from the UberNURBS palette. Before we start, let's take a quick look at the UberNURBS Palette.

## UberNURBS Palette

The UberNURBS Palette is where all of the UberNURBS tools are located. The illustration below names all of the tools for you. We will define the functions of these tools as we move along.

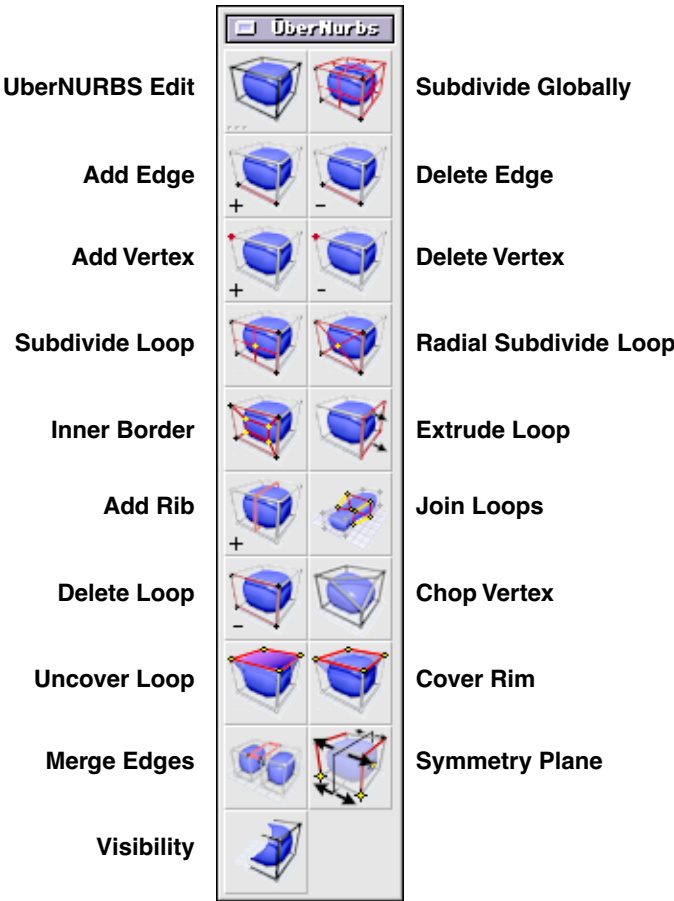


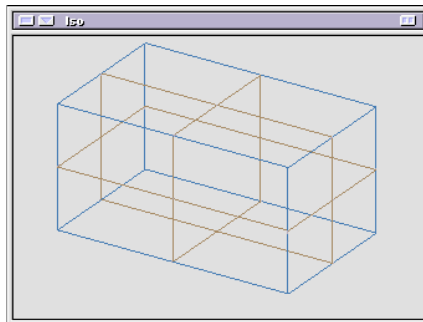
Figure 23.0 — UberNURBS Palette



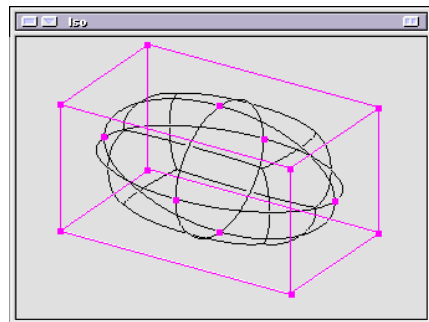
UberNURBS can be created from ACIS bodies that are solids (enclosing a volume) and 4-sided planar rectangles (you can have more complex planar geometry, but you will need to add edges once in UberNURBS edit mode to make sense of the result). The following shapes are acceptable as UberNURBS base objects:

- Surface Sheet or Rectangle
- Block
- Pyramid
- Prism

Shapes such as those which start off as a block and then are knifed with straight lines or rectangles are acceptable, although the caution regarding planar objects in the previous paragraph will apply. Once you have a suitable starting object, you can use the UberNURBS tools.



**Simple primitives are the best starting points for UberNURBS shapes.**



**The initial UberNURBS conversion of the block.**

*Figure 23.1 — Converting an ACIS object into UberNURBS*

To edit an object as an UberNURBS object:

1. Select the UberNURBS Edit icon from the UberNURBS palette
2. Click on the object you wish to edit as an UberNURBS object

*You will now see the object converted into an UberNURBS object. The display will change, showing a different shape from the original shape. It will be surrounded by a control cage that is similar to the control hulls when editing surfaces.*

3. To exit from UberNURBS edit mode, double-click in an empty space

Universe Modeler's normal ACIS-based editing environment will now be restored, and the new UberNURBS object will have replaced the original object.

### 23.1 UberNURBS Anatomy

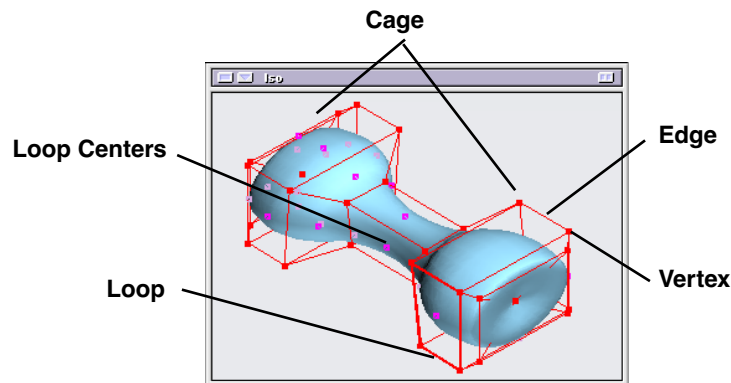


Figure 23.2 — Parts of an UberNURBS object

When the UberNURBS tool is initially used on an object, a control cage immediately appears and surrounds the object. The object will change

shape, due to the interpolation used for subdivision surfaces. The control cage can contain the following elements:

- Vertexes (similar to CVs for surface editing)
- Edges
- Loops
- Loop Centers
- Ribs
- Boundary Rims
- Vertex Weights

Figure 23.2 shows the parts of an UberNURBS object labeled (with the exception of ribs and vertex weights). Some of these elements will appear familiar to you if you have edited NURBS surfaces in Universe Modeler. There are also differences: UberNURBS do not have editable U and V directions as NURBS surfaces do, although U and V coordinates are assigned to UberNURBS faces upon export for texture mapping purposes.

### Vertexes

Vertexes are the corner points of a control cage. You can change the position of a vertex at any time. You can add or remove vertexes with the Add Vertex or Remove Vertex icons in the UberNURBS palette. A vertex will always be placed at the intersection of two edges. Weight editing is accomplished at the vertex level of the control cage.

### Edges

Edges are the lines drawn between each vertex. You can change the position of an edge by dragging at any time. You can rotate (“r”) and scale (“s”) edges at any time by holding down the appropriate keys as indicated. You can add or delete edges with the Add Edge or Delete Edge icons found in

the UberNURBS palette. You can add an edge at any location on the control cage. If you add an edge at the location of a pre-existing vertex, the edge will begin at that vertex, and end at the location you specify. If you add an edge to the interior sections of another edge, a vertex will be placed at the intersection point for you. Should you decide you do not want to place an edge at that location, you will need to “undo” at least twice: once for the edge, and once for each vertex that was added.

### Loops

Loops are closed “rings” of edges covered by four connecting surfaces, and contain a “loop center” vertex for selection purposes. Loops are different from “ribs” which are a ring of edges with no surface covering them. Loops can be subdivided by several subdivide tools in the UberNURBS palette. You can also move, rotate and scale loops as desired. In Figure 23.2, the loop in the example has been drawn with thicker lines.

*Each loop must have a minimum of 3 edges. It is not uncommon for complex UberNURBS shapes to contain as many as 2000 of these loops. This can be a memory and file size issue when converting an UberNURBS object to an ACIS object.*

### Loop Centers

Loop Centers are the centroid vertex of each loop. Think of these as “selection handles” for loops, as the loop center is used to select a loop by clicking on it. You also use the loop center to subdivide the loop. This is done by selecting a subdivide tool and clicking on the loop center.

### Ribs

Ribs are connected edges that are not covered by surfaces. You cannot select a rib directly, meaning that there is no analogy for a “loop center” for ribs. You can select ribs as you would normally select edges. You can add a complete rib with the Add Rib icon in the UberNURBS palette. You cannot directly delete ribs. To delete ribs, you must delete each edge of the rib instead.

### Boundary Rims

Boundary rims are uncovered loops that form the boundary with objects. Think of them as ribs without faces or adjoining edges. Open boundary UberNURBS typically start out as rectangles before becoming UberNURBS.

### Vertex Weights

Vertex weights allow you to add additional tension or slack to a point. This can save you from having to add more edges or loops for additional details, such as facial creases or skin folds. Unlike weights in NURBS surface editing, which are discouraged but allowed, you can freely use weights in UberNURBS. Weights are added by selecting a vertex (or series of vertexes) and depressing the “w” key, and dragging to the left or right. Weights can be reset in the **UberNURBS Settings dialog**, available by double-clicking the UberNURBS Edit icon in the UberNURBS palette.

## 23.2 Returning to Universe Modeler (ACIS) Mode

When shapes are completed, they are processed, and become solid shapes that can be edited like any other ACIS object. You can take the object back into the UberNURBS environment at any time. However, if you have per-

formed any operation which changes the topology of the object, such as a boolean or rounding operation, the original UberNURBS data will be lost. Instead, UberNURBS will use the existing topology as a new cage, resulting in a very different shape. Depending upon your needs, this behavior could be valuable to you. If you determine that you need to use ACIS tools and you wish to maintain your UberNURBS shape for as long as possible, then save the ACIS topology operations until the very end of building your model.

### 23.3 What Can You Edit?

The following options are available to you as you edit the UberNURBS control cage elements discussed in “UberNURBS Anatomy” on page 406:

- Translation (x,y,z)
- Rotation (x,y,z)
- Scale (uniform, x,y,z)
- Weight editing per vertex

#### Translate Control Elements

To translate selected vertexes, edges, or loops, just drag. If you want to limit the translation to a particular axis, hold down the axis key on the keyboard, and then drag.

For example, if you wish to move a selected loop in the X direction, just select the loop, hold down the “x” key and drag.

### **Rotate Control Elements**

To Rotate selected elements, depress the “r” key. If you wish to limit the rotation to a particular axis, follow the example above for translating entities.

### **Scale Control Elements**

To scale a series of selected elements around their midpoint, depress the “s” key. If you wish to scale on a particular axis (non-uniform scale), just depress the desired axis key, and drag.

### **Edit Vertex Weights**

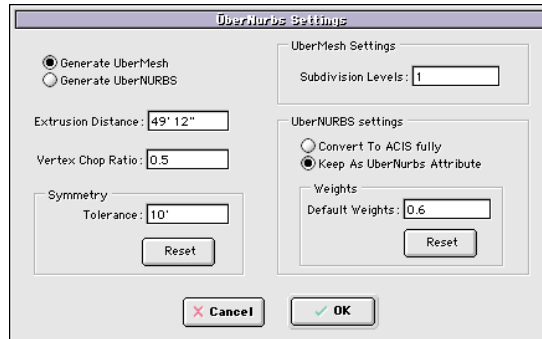
You can edit the weights of a vertex or series of vertexes, by first selecting the desired vertexes, and depressing the “w” key while dragging. Dragging to the left will increase the weights to a maximum value of 1. Dragging to the right will decrease the value of the weights to 0.

Action	Activation Key	Axis Limit Key	Optional Keys
Translation	None	x, y, z	n (normal)
Rotation	r	x, y, z	n (normal)
Scale	s	x, y, z	
Vertex Weight Edit	w		
Flip the normal of the object	v		
Disable Reflection Symmetry (temporarily)	k		
Show/Hide Control Cage	t		
Show/Hide Surface	Tab		
Show/Hide Vertexes and Loop Centers	Shift Tab		
Merge UberNURBS together	m r		

*Table 1: UberNURBS keyboard Shortcuts*

As you work with weights, you may choose to reset or force a value onto the weights of an UberNURBS object. This is done by first setting the desired weight value in the Default Weights edit field (UberNURBS Settings dialog, see Figure 23.3) and then choosing the Reset button.





**Figure 23.3 — UberNURBS Settings Dialog Box**

Now that we have covered what you can work with, let's look at how to edit an UberNURBS object. UberNURBS editing is primarily pushing and pulling vertexes, edges and loops. The purpose is to build a control cage that is subdivided into the shape that you require. Tools are provided to allow you to add and remove vertexes, edges and ribs, subdivide all or parts of your UberNURBS object, and rotate and scale parts to fit your needs.

### 23.4 Globally Subdividing an UberNURBS Object

If you are starting out with a plain block, you are going to need to get more resolution into your control cage before you can do anything of use. The easiest way to increase the resolution of your UberNURBS cage is to use the Global Subdivide tool to add more resolution to your control cage. You have a choice of two subdivision methods: Subdivide, which will subdivide the cage and will result in a significant change to the UberNURBS object; or Subdivide and Average, which will result in no significant

changes to the object, but which will change the shape of the control cage quite a bit.

### **To globally subdivide an UberNURBS object:**

1. Make sure that the object is already in UberNURBS edit mode
2. Choose the Global Subdivide icon from the UberNURBS palette
3. Depress the “shift” key
4. Click on the object

The control cage is now subdivided, and the UberNURBS object will have changed significantly. This method of subdivision is likely to be objectionable, as it does change your object quite a bit. You may find little use for this particular method.

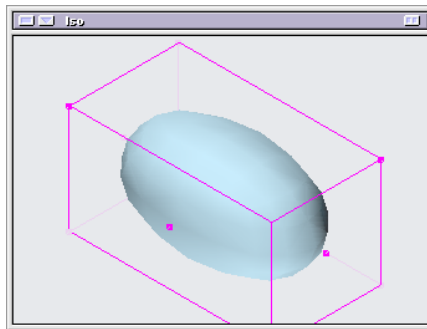
### **To globally subdivide and average an UberNURBS object:**

1. Make sure that the object is already in UberNURBS edit mode
2. Choose the Global Subdivide icon from the UberNURBS palette
3. Click on the object.

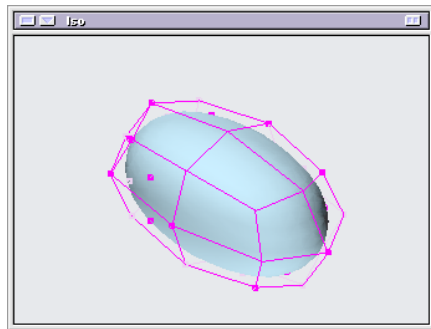
The control cage is now subdivided. The UberNURBS object will not have changed significantly, but the control cage will have (see Figure 23.4)

This method of subdivision is likely to be preferred over the other, as it doesn't change the shape of the object much at all, and still gives you the resolution you need.

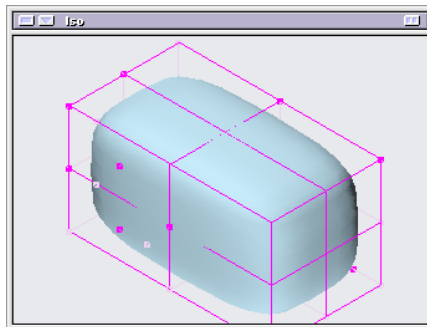
You can subdivide an UberNURBS object as often as you wish. However, you should only subdivide the surface to the level necessary to get the resolution that you need to work with. Once global subdivision serves its purpose, there are more precise tools available for detail work.



**A box has been converted to an UberNURBS for editing.**



**The box with global subdivision and averaged.**



**The same box with global subdivision only (shift key depressed when tool was applied.)**

**Figure 23.4 — Global subdivision methods for UberNURBS**

### 23.5 Working With Vertexes and Edges

Vertexes and edges are foundation elements for creating shapes with UberNURBS. As powerful as the subdivision tools are, in the end you will need to be able to deal with vertexes and edges, as they offer the most control over your shape. Vertexes and edges also give you the best tools to fix problems, such as orphaned vertexes which can cause “peaks” on your shape. There are five tools that facilitate working with vertexes and edges:

- Add Vertex
- Delete Vertex
- Chop Vertex
- Add Edge
- Delete Edge

#### Adding Vertexes

If you think of an UberNURBS control cage as a “ball and stick” model, the vertex would be the ball, and the edge would be the stick. You cannot have an edge without vertexes at each end, just as you cannot have a stick without a ball. Most of the time, you will find that the edge tool will suffice in most instances, but there will be times when you must work at the vertex level.

**To add a vertex to an UberNURBS control cage:**

1. Make sure that the object is already in UberNURBS edit mode
2. Choose the Add Vertex icon from the UberNURBS palette
3. Click on the location for your vertex
4. Continue to add as many vertexes as desired. Choose the Edit icon from the UberNURBS palette to terminate the Add Vertex operation

A vertex will be added to the control cage. If you wish to precisely place your vertexes, you can use the appropriate snap settings. For example, if you wish to have a vertex in the exact center of an edge, activate the Snap to Edge Center icon in the Snap palette, and then add the vertex. It will be added to the middle of the edge.

A single vertex which doesn't form the intersection of three or more edges will have a "spike" effect on the subdivided surface. While the weight of the vertex can be adjusted, it will always draw a point of the subdivided surface to it. To correct this, you will need to add an edge to the vertex, or to remove the vertex.

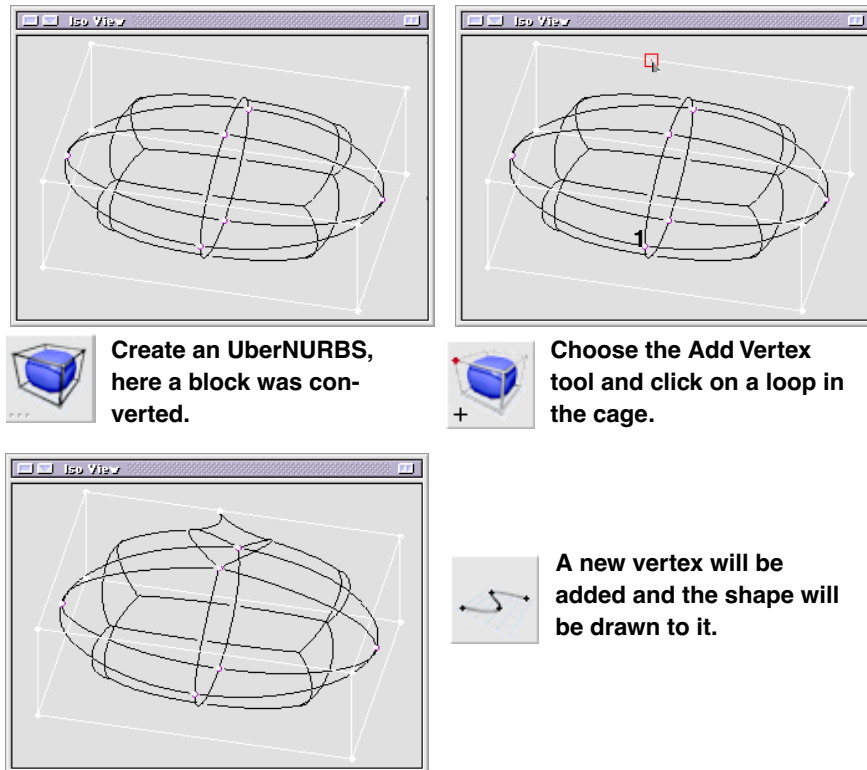


Figure 23.5 — Adding a vertex to a cage

### Removing Vertexes

As noted previously, removing vertexes is typically done to clean up problems with the UberNURBS control cage. Extra vertexes can be apparent if you delete edges without deleting the vertexes associated with the edges. They can also be apparent if you undo an Add Edge operation without

undoing the vertex creation as well. Any vertex in an UberNURBS control cage can be deleted. There is no restriction.

**To delete a vertex from an UberNURBS control cage:**

1. Make sure that the object is already in UberNURBS edit mode.
2. Choose the Delete Vertex icon from the UberNURBS palette.
3. Select the vertex to delete.
4. Continue to delete as many vertexes as necessary. Choose the Edit icon from the UberNURBS palette to terminate the Delete Vertex operation.

**To delete multiple vertexes at the same time:**

1. Select all vertexes.
2. Click on the Delete Vertex icon.
3. Click on the world view.

The program will delete all selected vertices. This is useful for extracting parts from previously modeled UberNURBS.

As you delete vertexes, you may see edges collapsing, and eventually removed. Once a vertex is removed from the UberNURBS control cage, the edges that were attached to that vertex are no longer viable and are removed. To understand this effect, create a block and convert to UberNURBS. Use the tab key to hide the subdivided surface, and then choose the Delete Vertex tool to remove one of the top corners of the cage. It will appear as if the corner of the cube was cut away (because it effectively was.) If you continue by removing the bottom corner below the first corner, your cube has now been changed to a prism.

### Chopping Off Vertexes

The Chop Vertex tool deletes an existing vertex, but unlike the Delete Vertex tool it does not remove the surrounding edges. The Chop Vertex tool will instead subdivide the bordering edges of the vertex, adding new edges in-between, and then remove the vertex.

#### **To chop a vertex from an UberNURBS control cage:**

1. Make sure that the object is already in UberNURBS edit mode.
2. Choose the Chop Vertex icon from the UberNURBS palette.
3. Select the vertex to chop.
4. Continue to chop as many vertexes as necessary. Choose the Edit icon from the UberNURBS palette to terminate the Chop Vertex operation.

The Chop Vertex tool can be used to better define a given point of an UberNURBS. Once the vertex area has been redefined, further editing with Subdivide Loop, or inset loop can be applied to the new loop.



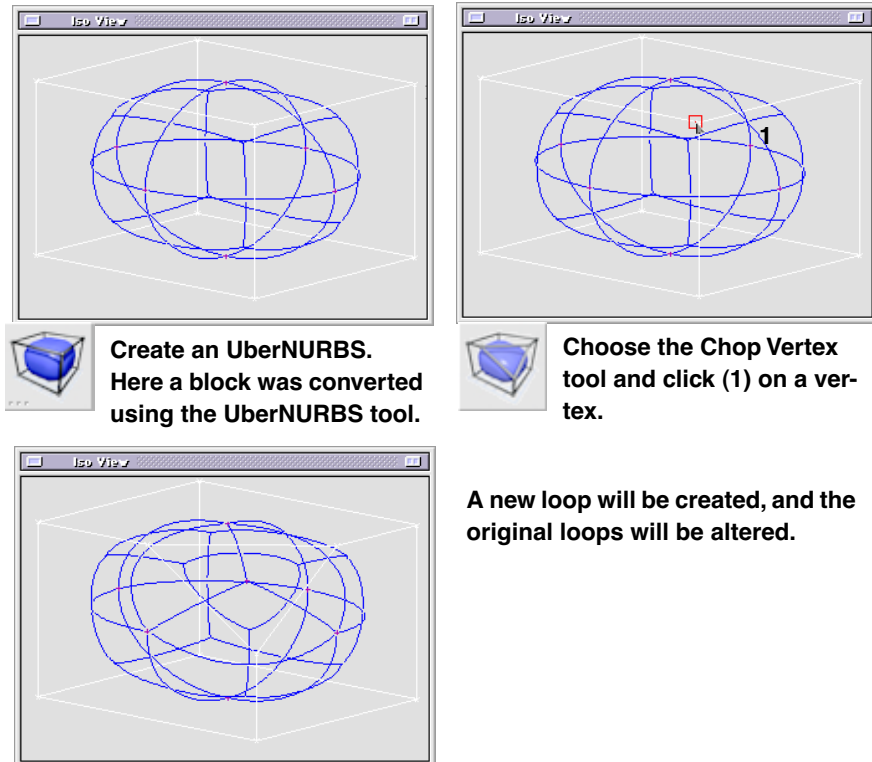


Figure 23.6 — Chopping a vertex on an UberNURBS

### Adding Edges

As mentioned previously, vertexes and edges resemble the “ball and stick” model. In fact, if you view the UberNURBS control cage with the subdivision surface hidden (tab key), the cage will definitely resemble the ball and stick metaphor. And, if vertexes are the balls, then edges are the sticks.

Edges cannot exist without vertexes at either end of the edge. If you remove an end vertex from an edge, you effectively destroy the edge.

You can place an edge anywhere with the exception of an edge which would cross through a loop, as this would place a loop bisecting the volume of the subdivision surface, which is not allowed.

**To add an edge to an UberNURBS control cage:**

1. Make sure that the object is already in UberNURBS edit mode
2. Choose the Add Edge icon from the UberNURBS palette
3. Use the tab key to hide the subdivision surface and the “o” key to orbit in the iso view to get a better look at your model

*The surfaces can clutter the view, and obscure the edges to delete. The tab key removes the guess work there. Orbiting in the view will also make edges easier to see.*

4. Click on the edge or vertex to use as your starting point
5. Click on the edge or vertex to use as your end point

If you click on an edge, a new vertex will be created as a starting point. If you click on a vertex, that vertex will be used instead of creating a new vertex. Be careful, as you can accidentally create a vertex next to another vertex without realizing it.

6. To continue adding edges, follow steps 4 and 5. Remember to select the first edge, and then the last edge each time
7. To exit the Add Edge tool, choose the Edit icon from the UberNURBS palette

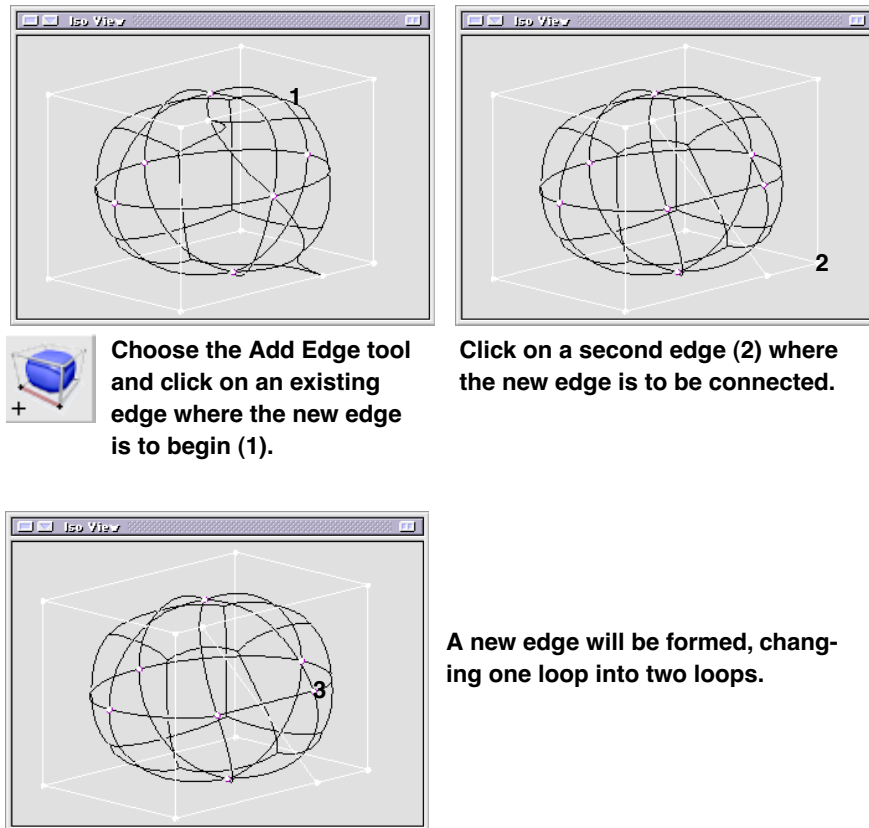


Figure 23.7 — Creating a new edge

### Removing Edges

Any edge can be deleted at any time. If additional edges need to be deleted to preserve the integrity of the cage, they are done so automatically. Removing an edge will not remove the vertexes on the ends of the edge. If

you need to remove these as well, you must also use the Delete Vertex tool. This is necessary as the remaining vertexes are attached to other edges.

**To delete an edge from an UberNURBS control cage:**

1. Make sure that the object is already in UberNURBS edit mode
2. Choose the Delete Edge icon from the UberNURBS palette
3. Use the tab key to hide the UberNURBS surfaces and the “o” key to orbit in the iso view to get a better look at your model

*The surfaces can clutter the view, and obscure the edges to delete. The tab key removes the guess work there. Orbiting in the view will also make edges easier to see.*

4. Select the edge to delete

The edge will be removed from the control cage. If you wish, you can continue to remove edges as needed.

5. To exit the Delete Edge tool, choose the Edit icon from the UberNURBS palette.

As edges are removed from the cage, the underlying UberNURBS shape will change accordingly. If you accidentally remove an unintended edge, use the undo key shortcut, command z (PC- control z).

### 23.6 Working With Loops

Loops are used to form all of the subdivision surfaces created with UberNURBS. There are several tools which allow you to modify loops and make them more complex, or inset more loops into previous loops, delete loops, expose boundary rims, and so on. These tools are:

- Subdivide Loop
- Radial Subdivide Loop
- Inset Loop
- Extrude Loop
- Add Rib
- Delete Loop
- Uncover Loop
- Cover Boundary Rim

### Subdividing a Loop

Subdivide Loop breaks up a single loop and makes several new smaller loops. The number of loops created depends on the number of sides the loop had originally. A subdivided six-sided loop will create six new loops. Each of these loops comes with a new set of vertexes, and that means more control over the shape. Subdivide also creates equidistant edges, which can look “too regular” on your object. This tool is typically used as a starting-off point for more detailed work, sometimes by further addition of vertexes and edges.

**To subdivide a loop on an UberNURBS control cage:**

1. Make sure that the object is already in UberNURBS edit mode.
2. Choose the Subdivide Loop icon from the UberNURBS palette.
3. Select the loop to subdivide.
4. Continue to subdivide as many loops as necessary. Choose the Edit icon from the UberNURBS palette to terminate the Subdivide Loop operation.

The Subdivide Loop tool will place a new vertex at the center of the loop, and split each loop edge in two. As such, you will end up with extra vertexes on the bordering loops which you may need to rectify.

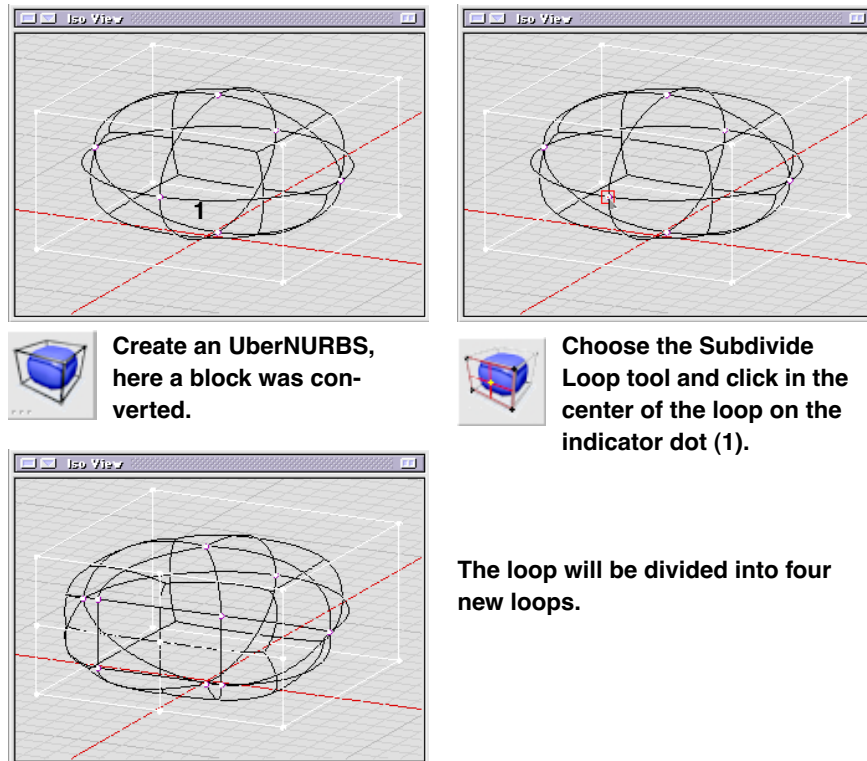
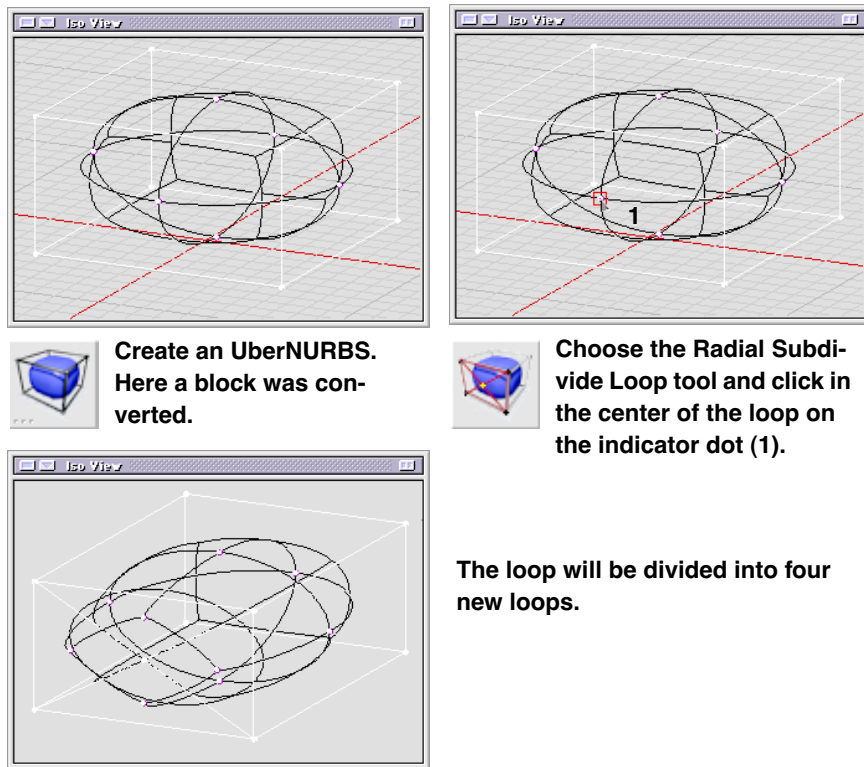


Figure 23.8 — Subdividing an UberNURBS loop

### Subdividing a Loop Radially

The Subdivide Loop tool breaks the loops by subdividing the bordering edge of the original loop. The Radial Subdivide Loop breaks the loop by creating edges from the center of the original loop to the corner vertexes of the original loop. This method will remove any need to further rectify unattached vertexes, as there won't be any.



**Figure 23.9 — Radial subdividing an UberNURBS loop**

**To radially subdivide a loop in an UberNURBS control cage:**

1. Make sure that the object is already in UberNURBS edit mode.
2. Choose the Radial Subdivide Loop icon from the UberNURBS palette.
3. Select the loop to subdivide radially.
4. Continue to subdivide as many loops as necessary. Choose the Edit icon from the UberNURBS palette to terminate the Radial Subdivide Loop operation.

### Inset Loop

Unlike the two previous Subdivide tools, the Inset Loop tool creates a new loop in the inside of an existing loop, and attaches it at the corners by a series of edges, each creating a new loop. This way the inside of the initial loop can be edited without affecting the overall shape of the object.

The number of edges that the new inset loop will have is determined by the number of edges that the original loop had. A loop with six edges that has the inset loop applied to it will make a six sided inner border with six new surrounding faces.

#### **To inset a loop in an UberNURBS control cage:**

1. Make sure that the object is already in UberNURBS edit mode.
2. Choose the Inset Loop icon from the UberNURBS palette.
3. Select the loop to inset.
4. Continue to inset as many loops as necessary. Choose the Edit icon from the UberNURBS palette to terminate the Inset Loop operation.



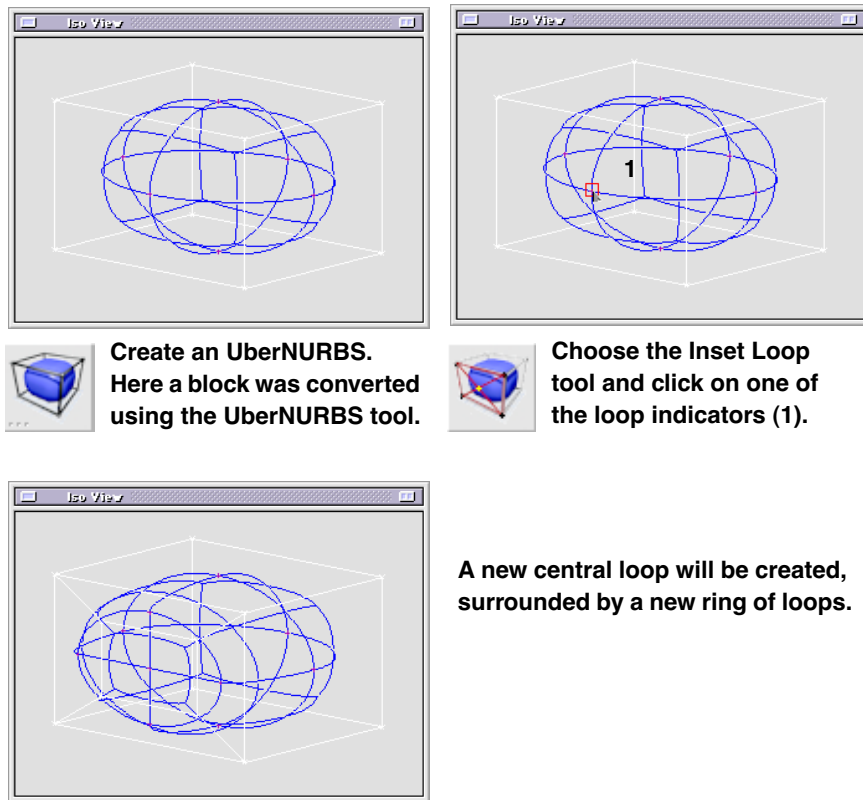


Figure 23.10 — Creating an inset loop on an UberNURBS

### Extruding a Loop

Extruding a loop is a quick and easy way to add extensions to your original UberNURBS control cage. With Extrude Loop, you can create appendages for arms and legs, handles, and so on. The Extrude Loop tool will extrude the loop at a preset distance, which you supply in the **UberNURBS settings dialog** (extrusion settings edit box.)

To extrude a loop in an UberNURBS control cage:

1. Make sure that the object is already in UberNURBS edit mode.
2. Choose the Extrude Loop icon from the UberNURBS palette.
3. Select the loop to subdivide radially.
4. Continue to extrude as many loops as necessary. Choose the Edit icon from the UberNURBS palette to terminate the Extrude Loop operation.

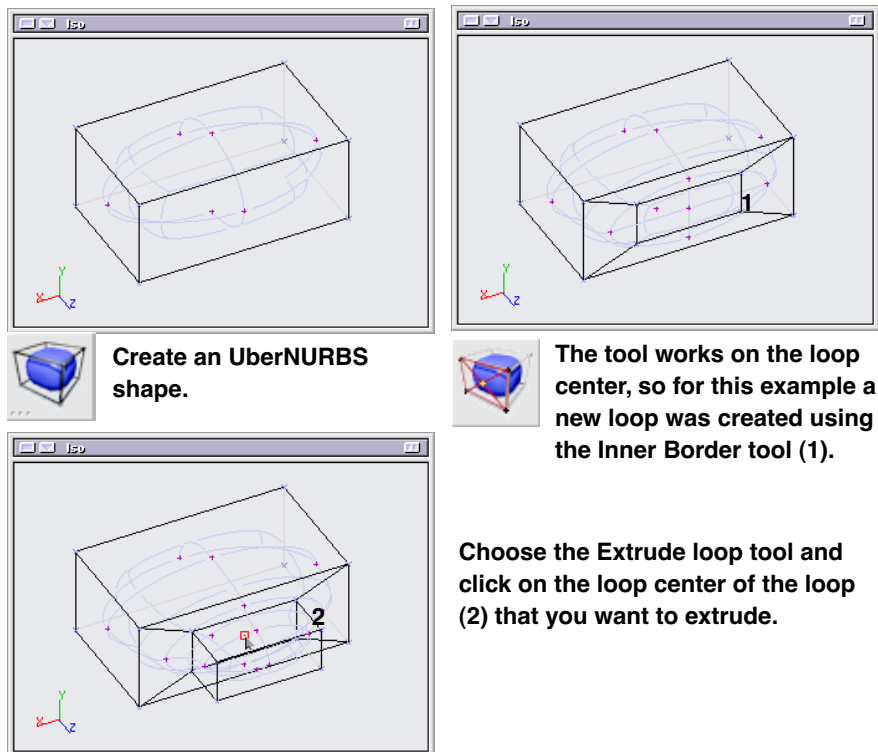


Figure 23.11 — Extruding a loop on an UberNURBS

### Adding a Rib

As you create your UberNURBS shapes, you will find need to add more edges to your control cage in order to add more detail to your work. In many instances, you may find that adding edges, while it works, is tedious. In those cases, you should try the Add Rib tool. The Add Rib tool creates a ring of edges along a plane that you specify.

Parts of the cage that are hidden using the visibility tool are not affected by the Add Rib tool. If the plane line cuts across a single edge, a vertex is added in the same manner as if you used the Add Vertex tool.

**To add a rib to an UberNURBS control cage:**

1. Make sure that the object is already in UberNURBS edit mode.
2. Choose the Add Rib icon from the UberNURBS palette.
3. Drag a line in the appropriate view window that crosses through the area in which you want the rib (similar to the Straight Knife tool).
4. Continue to add as many ribs as necessary. Choose the Edit icon from the UberNURBS palette to terminate the Add Rib operation.

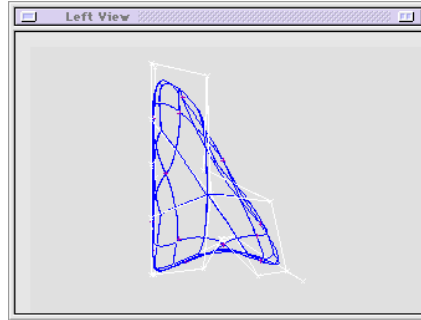
### Delete Loop

Every so often you will find the need to delete a loop. This can happen for a variety of (primarily artistic and subjective) reasons. You have two options in this case: the aforementioned Delete Edges tool, which you will need to use on every edge of the loop, but offers more control, or the Delete Loop tool. Remember, as loop edges are shared, deleting loops will have a drastic effect on neighboring loops as well.

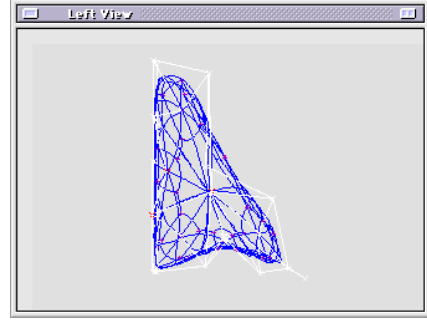
**To delete a loop from an UberNURBS control cage:**

1. Make sure that the object is already in UberNURBS edit mode

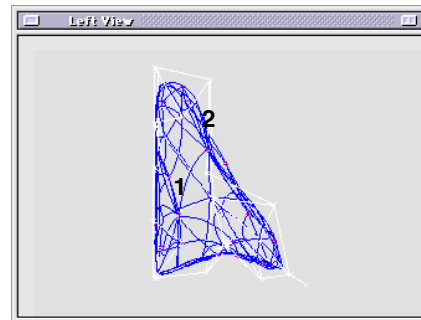
2. Choose the Delete Loop icon from the UberNURBS palette
3. Select the loop that you want deleted
4. Continue to delete as many loops as necessary. Choose the Edit icon from the UberNURBS palette to terminate the Delete Loop operation



In creating this nose, more definition was needed, so the UberNURBS was subdivided.



Subdividing added more faces to the top as well as the bottom, where most of the definition was needed.



So the Delete tool was chosen and loops at the top of the nose were deleted (1 and 2).

Figure 23.12 — Deleting a loop in an UberNURBS

### Uncovering a Loop

When you remove a face from an UberNURBS object, you are uncovering a loop. When you uncover a loop, you are creating a boundary rim. Use this tool if you desire an open face on your UberNURBS object (artist and über modeler Paul Sherstobitoff uses open-loop UberNURBS to easily blend ACIS objects with his UberNURBS work, with spectacular results).

**To uncover a loop of an UberNURBS control cage:**

1. Make sure that the object is already in UberNURBS edit mode
2. Choose the Uncover Loop icon from the UberNURBS palette
3. Select the loop you wish to uncover
4. Continue to uncover as many loops as necessary. Choose the Edit icon from the UberNURBS palette to terminate the Uncover Loop operation

### Covering a Boundary Rim

Covering a boundary rim is necessary to create a closed UberNURBS object. It can become necessary to cover a boundary rim if you wish to extend the UberNURB control cage beyond the boundary. You cannot use the subdivide tools, as there is no loop to subdivide. You would first need to cover the boundary rim, and then subdivide the result. Covering a boundary rim also allows the use of the Join Tool described later in the manual.

**To cover a boundary rim of an UberNURBS control cage:**

1. Make sure that the object is already in UberNURBS edit mode
2. Choose the Cover Boundary Rim icon from the UberNURBS palette
3. Select the boundary rim that you wish to cover

4. Continue to cover as many boundary rims as necessary. Choose the Edit icon from the UberNURBS palette to terminate the Cover Boundary Rim operation

### 23.7 Merging UberNURBS Objects Together

In addition to being able to edit the above, you can also join UberNURBS objects together (for sub-patch modeling). This allows you to build parts separately, such as ears and noses, and reuse them whenever you need. You can also merge a multiple-lumped body together in UberNURBS. There are two tools which allow you to merge UberNURBS objects:

- Join Loops
- Merge Edges

In order for you to join two loops of an object into one, or to merge together two separate UberNURBS shapes, you first need to get the desired objects into the UberNURBS edit environment.

**To add another UberNURBS object into the editing environment:**

1. Select the Edit icon from the UberNURBS palette
2. Click on the first object you wish to edit as an UberNURBS object
3. Depress the “m” and “r” keys (for “merge”)
4. Click on the second object that you wish to edit
5. Continue until you have all of the objects needed in the environment

You can now edit these objects separately, or join them together.

*Once you have merged UberNURBS together with the above procedure, all of the objects will now be considered a single object. Further, UberNURBS will add all of the objects into the first object selected for UberNURBS editing. The subsequent objects that you add will be copied into the first object. This will result in leaving*

*the originals of the subsequent objects intact as separate bodies in the Universe Modeler (ACIS) environment.*

### Joining Two Loops Together

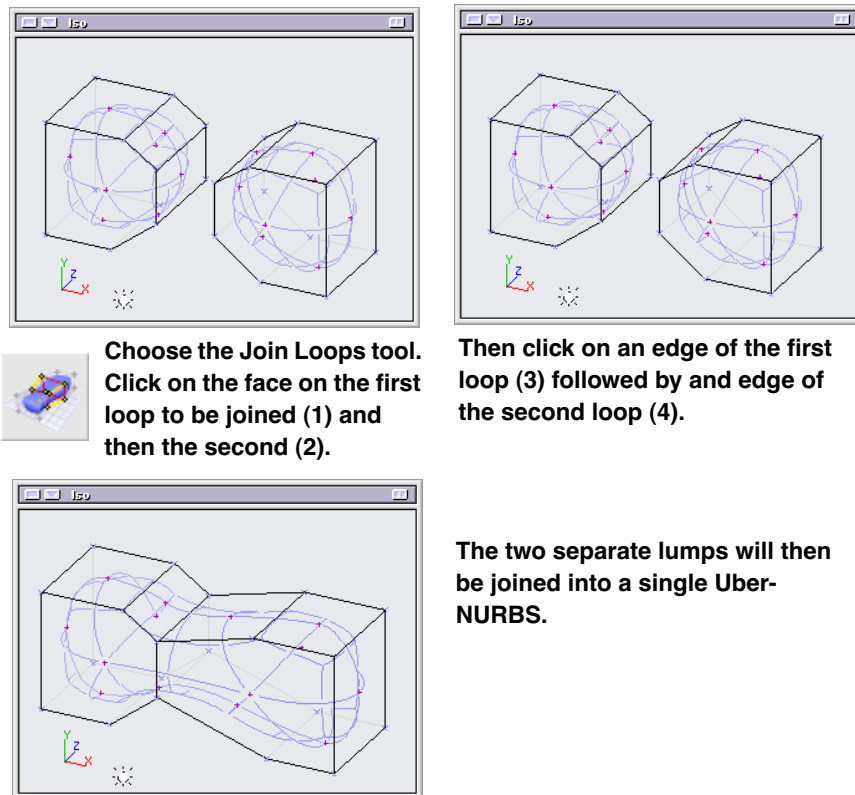
When you have more than one UberNURBS object that you would like to connect, and neither object has an open boundary rim that you would like to connect to, then the Join Loops tool is the tool you need. Join Loops can work on either separate objects, or you can join separate loops of the same object. In both cases, the two loops must have the same number of edges.

**To join two loops of different UberNURBS control cages together:**

1. Make sure that the objects have been merged and are already in UberNURBS edit mode, as described in the previous section
2. Choose the Join Loops icon from the UberNURBS palette
3. Select the loop on the first object to join
4. Select the loop on the second object to join
5. Per the prompt in the status window, pick the first edge of the first loop
6. Pick the first edge of the second loop

The loops will now be joined together. You must first pick the loops, and then the edges in order to give UberNURBS the information to properly stitch the loops together.

7. Choose the Edit icon from the UberNURBS palette to terminate the Join Loops operation



*Figure 23.13 — Joining loops on separate UberNURBS*

In addition to joining loops of separate bodies, you can also join separate loops in the same body. This can come in handy to create bridges or handles from one part of the UberNURBS object to another.



**To join two separate loops of the same UberNURBS control cage together:**

1. Make sure that the object is already in UberNURBS edit mode
2. Choose the Join Loops icon from the UberNURBS palette
3. Select the first loop to join
4. Select the second loop join
5. Per the prompt in the status window, pick the first edge of the first loop
6. Pick the first edge of the second loop

The loops will now be joined together. It is your responsibility to insure that the loops do not cross into the space occupied by other loops. UberNURBS will not reject this condition, but you probably will.

7. Choose the Edit icon from the UberNURBS palette to terminate the Join Loops operation

### Merging Edges Together

Merging edges is similar to joining loops together, except that merging edges works with open boundaries only (all other edges are rejected by the tool). If you wish to use the merge edges tool on objects with closed loops, you will need to first uncover the loops, and then use the tool.

*This tool will produce the same effect as joining loops. Merge Edges is designed primarily for “open boundary” UberNURBS, as is popular with sub-patch modeling.*

**To merge together edges of separate UberNURBS objects:**

1. Make sure that the objects have been merged and are already in UberNURBS edit mode, as described in the previous section
2. Choose the Merge Edges icon from the UberNURBS palette
3. Select the first edge of the first boundary rim

4. Select the first edge of the second boundary rim
5. Select the second edge of the first boundary rim
6. Select the second edge of the second boundary rim
7. Continue for the remaining edges of each rim

The boundary rim edges will now be joined together. In the case where two edges are shared, as will always be the case on the last remaining two edges, UberNURBS will be able to see both edges, so you only need to pick once. A confirmation message will appear in the status window.

8. Choose the Edit icon from the UberNURBS palette to terminate the Merge Edges operation

In addition to merging edges of separate bodies, you can also merge boundary rim edges in the same body.

**To merge separate boundary rims of the same UberNURBS control cage together:**

1. Make sure that the object is already in UberNURBS edit mode
2. Choose the Merge Edges icon from the UberNURBS palette
3. Select the first edge to join of each boundary rim
4. Select the second edge join of each boundary rim
5. Continue for the remaining edges of each rim

The boundary rims will now be joined together. It is your responsibility to insure that the rims do not cross into the space occupied by other loops. UberNURBS will not reject this condition, but you probably will.

6. Choose the Edit icon from the UberNURBS palette to terminate the Merge Edges operation

### 23.8 More UberNURBS Features

There are two UberNURBS tools which help to make editing UberNURBS a little easier. These tools are:

- Symmetry Plane
- Visibility

We'll cover these tools in depth in the following sections.

#### Symmetry Plane

There will be times when you want to create an UberNURBS shape that will be the same on both sides, such as a character's face. The best way to accomplish this is to use the Symmetry Plane tool.

The Symmetry Plane tool allows you to define a plane for the UberNURBS object that acts as a reflecting mirror. Vertexes and loops moved on one side of the plane will be reflected to their opposites on the other side of the plane, within a tolerance that you specify. The following operations are supported by the symmetry plane:

- Translation
- Rotation
- Scale
- Vertex Weight Editing

No other operations are supported for the Symmetry Plane. If you add an edge, for example, you will not see a corresponding edge add to the other side of the plane.

The UberNURBS Symmetry Plane tool does not work like the Symmetry Plane for NURBS surfaces. UberNURBS are arbitrary meshes which are randomly created by you, whereas control hull editing for surfaces will always result in an ordered control hull. Therefore, there is no tolerance value required for NURBS surfaces as there is for UberNURBS.

The tolerance value is very important to your success with the Symmetry Plane in UberNURBS. It is used to capture vertexes on the opposite side of the symmetry plane, and track them with vertexes on the primary side of the plane. If your tolerance value is too small, you will not see any result. If your tolerance value is too large, you will not see a mirror effect; instead, you may see clusters of vertexes responding to the motion of one on the other side of the plane.

### **Adding a symmetry plane to your UberNURBS control cage:**

1. Make sure that the object is already in UberNURBS edit mode
2. Set your symmetry tolerance value in the **UberNURBS Settings dialog**
3. Choose the Symmetry Plane icon from the UberNURBS palette
4. Activate the snapping grid (F5) for precise placement (optional)
5. Click to set the first point of the plane, and drag to set the second point

The symmetry plane will now be set. It will remain set until you deactivate it. The symmetry plane is an attribute of the UberNURBS control cage for each object, so the plane will remain active even through multiple UberNURBS edit sessions of the same object.

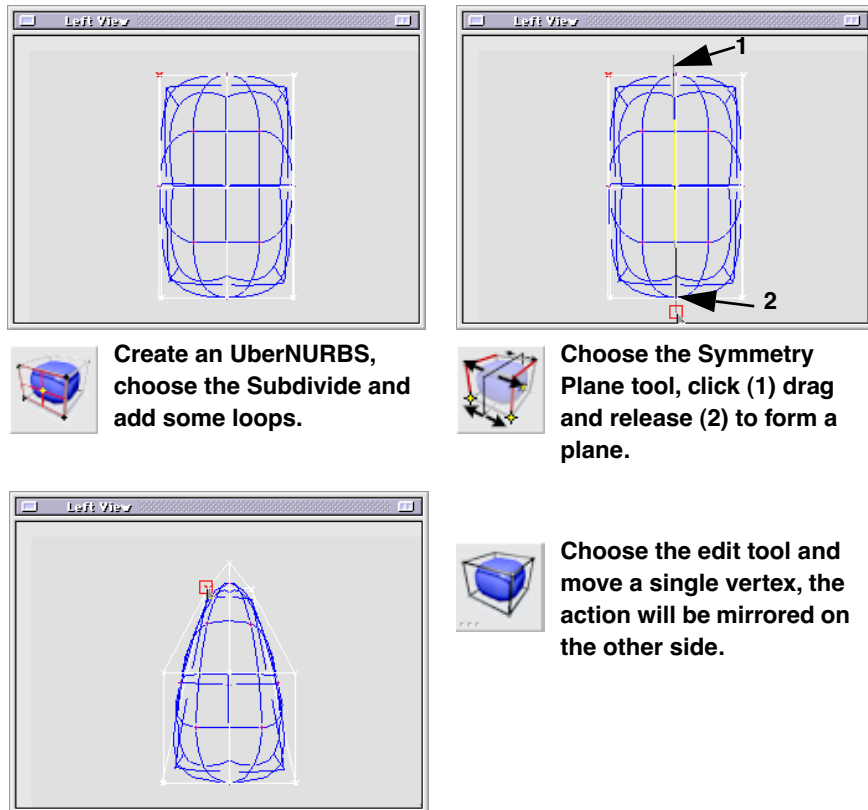


Figure 23.14 — Symmetrically editing an UberNURBS

To deactivate the symmetry plane of an UberNURBS control cage:

1. Make sure that the object is already in UberNURBS edit mode
2. Set your symmetry tolerance Reset button in the **UberNURBS Settings** dialog

or, as an alternative,

3. Anywhere outside of the control cage, set the first point, then drag and click to set the second point

The symmetry plane will now be deactivated. If you want to reactivate the plane later, you will need to add a new symmetry plane.

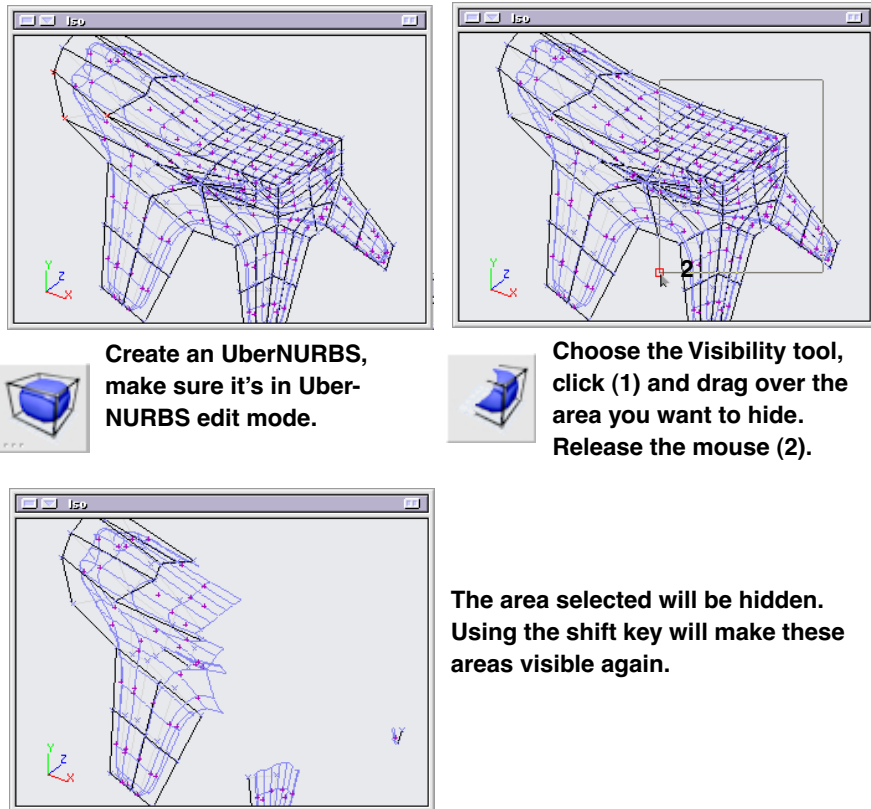
### Visibility

UberNURBS is a powerful tool that allows for extensive editing of organic shapes, but the level of control afforded can come at a price — complex shapes can get a little overwhelming. The program will dim the back edges and vertexes on the other side of the object for clarity, but that may not always help you to see the structure of your creation. To give you control over this, the Visibility tool can be used to select, vertexes, edges and surface portions to make them visible or invisible. The tool effects both the construction elements, as well as the body itself.

#### **To make portions of an UberNURBS object invisible:**

1. Make sure that the object is already in UberNURBS edit mode
2. Choose the Visibility icon from the UberNURBS palette
3. Drag a rectangle around (or individually select) the elements to hide

The elements selected will now be hidden from view, and locked from selection or other manipulation from you. You can continue to hide elements at any time during the UberNURBS edit session.



*Figure 23.15 — Making parts of the UberNURBS invisible*

**To restore the visibility of hidden UberNURBS elements:**

1. Make sure that the object is already in UberNURBS edit mode
2. Choose the Visibility icon from the UberNURBS palette
3. Depress the “shift” key
4. Drag a rectangle around the affected area or around the entire object

The hidden elements of the UberNURBS object will now be restored. To continue editing the object, select the Edit icon from the UberNURBS palette.

### 23.9 Finishing an UberNURBS Edit Session

Once you get to the end of your editing process, you need to ask yourself a few questions:

- Are you going to use UberNURBS on this object again?
- Are you going to use other modeling tools such as Booleans, Knives or deformations?
- Are you planning on making morph targets?
- What tessellation and export settings should you use?
- Are UberMESHes better than UberNURBS for your use?

Let's explore some of these topics in the following sections.

#### Keep as UberNURBS or Convert to ACIS?

The first two questions, “are you going to use UberNURBS again?” and “are you going to use modeling tools such as Booleans, Knives or deformations?” will be answered in this section. UberNURBS bodies can be either kept as UberNURBS, or they can be converted to ACIS bodies. Converting to ACIS bodies creates huge data structures with a lot of memory requirements. In the **UberNURBS Settings** dialog you will see an option to either “Convert to ACIS fully” or to “Keep as UberNURBS Attribute.”

Many times conversion to ACIS is not necessary. If future editing is planned with the body, such as using the knives or booleans, the ACIS conversion is required, but will be automatically handled by the particular ACIS tool on demand. If the shape is going to be used as a stand-alone



object and you're going to continue to sculpt the object as an UberNURBS, you do not need to convert.

### Creating Morph Targets with UberNURBS

If you plan to do any facial or character animation with the objects that you create with UberNURBS, then it is likely that you will be dealing with morph targets. Morph targets are shapes which are intended to be blended together as you animate a time line, perhaps in sync with a dialog or sound effects track in Universe Animator.

It is very easy to create morph targets with UberNURBS. there are several simple points to consider in order to create successful morph targets:

- Your first version of the object should be a neutral pose or expression.
- You should not begin to create your morph targets until you are completely satisfied with the shape of your objects.
- Universe Animator morphs complete objects — you may create separate model files on disk, or in the same Universe Modeler file, but you must export morph targets as separate files.
- Each morph target must have the same level of tessellation.
- Special considerations are required if you intend to use outline shading with your UberNURBS models.

Editing morph targets is not as time consuming as creating your model, but you should still put some thought into it. Only changes to the mesh topology are effected by morphing, not location of the mesh. You can, therefore, model your targets side-by-side as is popular in some programs, or as separate files. The choice is yours.

One little-known feature of UberNURBS is a rudimentary implementation of selection sets for vertexes. While not useful for model construction, you may find selection sets useful when creating morph targets. For example, you may choose to define the upper lip region as one selection set, and the lower lip/jaw region as another.

Currently, selection sets are referenced as a set from one to ten, and are created linearly. You cannot name selections. Each set must be created following the last. Only after all ten selection set slots have been filled can you replace one. You can only replace ( $n^{\text{th}}$ ) selection after it has been created, say your current selection (unassigned) into slot 5.

### **To create a selection set in UberNURBS:**

1. Make sure that the object is already in UberNURBS edit mode
2. Select the vertexes that you want in a selection set
3. Depress “shift”, “command (PC- control)”, “w” to save the selection in the first available slot
4. Write down which slots are filled with what sets (remember, this implementation is rudimentary, so no tracking mechanisms are provided)
5. Continue for desired selection set

Once you have a series of selection sets created, you will be able to recall them when necessary. For example, if you are animating the brow area of a face, and you have a selection set for that area already created following the preceding directions, you can then recall the brow set and create frown or surprises poses.

### **To recall a stored selection set in UberNURBS:**

1. Make sure that the object is already in UberNURBS edit mode
2. Depress the “control” and number keys (the number in which the set was stored, such as “3”)

The stored selection set will now be recalled. As you work with your sets, you may need to refine your sets, or replace an existing set for some reason.

### **To replace an existing selection set with a new set in UberNURBS:**

1. Make sure that the object is already in UberNURBS edit mode
2. Determine the number of the set that you wish to replace
3. Select the vertexes that you want in the replacement selection set
4. Depress the “control”, “option (PC- alt)”, and number keys (of the set to be replaced)

The old selection set will now be replaced with a new selection set at the previous location.

## **UberNURBS vs. UberMesh Models**

Once you have created your UberNURBS object, and any morph targets required, you should determine how you want to tessellate your work. Tessellation reads your UberNURBS data and makes a copy of polygons for export into an external model format such as FACT. UberMESH data is passed directly through to the FACT format. FACT (short for “facet”, another name for polygon in computer graphics) is the format used by Universe Animator.

You actually have a choice between UberNURBS and UberMESH to represent your model. UberNURBS are true surfaces and are tessellated only upon export, whereas UberMESH objects are Catmull-Clark subdivision surfaces, and exist only as polygons.

Often you may find some creases on UberNURBS data that you find objectionable, or not able to hide with modeling techniques or texture maps. In these instances, a better choice would be UberMESH. Due to the superior tessellation of the Catmull-Clark subdivision surfaces, your UberMESH

models will likely appear smoother than your UberNURBS models, even if the tessellation settings for the UberNURBS models were higher. However, you have more control over the amount of polygons generated by the UberNURBS models than you do with UberMESH models, as the difference between a subdivision value of 2 versus 3 is four (4) times.

### UberNURBS Tessellation

UberMESH tessellation is controlled in the UberNURBS Settings dialog, and UberNURBS tessellation is controlled in the **Object Information Window, Tessellation Tab**.

**To set the tessellation for an UberNURBS object:**

1. Select the object
2. Choose **Tessellation>Object Export Tessellation Settings** from the menu bar

The Object Information Window will appear, and the tessellation tab will be active.

3. Uncheck the Use Project's Setting check box
4. Set the tessellation Type popup menu to Uniform
5. Enter the desired number in the U and V divisions edit boxes
6. Click on the Apply New Settings button

The values entered in the U and V edit boxes will now be used whenever the UberNURBS object is exported. You can also copy these settings to the display record for the object, by clicking on the Copy Export to Display button. A good way to check your work would be to apply your export settings to display, and then shading your model in the interface (you can also choose to display the mesh to see the actual polygons).

*You are allowed to set two separate tessellation settings, one for display and one for export, to enable faster interaction in the program interface of Universe Modeler.*

The U and V values will be applied to *each face* of the UberNURBS object. If you have chosen a value of 3 by 3, each face will have 9 quadrilateral polygons, which will then be tessellated into 18 triangles. If you have 1200 faces, that's 21,600 triangles.

Uniform tessellation will always apply an even mesh to your object. It will not consider curvature, as adaptive triangulation does. Therefore, if you need more resolution to provide a better silhouette to your model, you will need to increase either the U or V or both.

### UberMesh Tessellation

UberMesh tessellation differs from UberNURBS tessellation to a degree. When tessellating an UberNURBS object, the tessellation settings apply to the amount of tessellation applied to each face (3 by 3, for example). However, the tessellation settings applied to UberMesh models represent how many times a face is subdivided. That means U and V values of 3 by 3 for an UberNURBS object will have fewer polygons than a subdivision value of 3 for an UberMesh object.

#### **To set the tessellation for an UberMesh object:**

1. Double-click on the Edit Icon in the UberNURBS palette.
2. In the UberNURBS Settings dialog, choose the “Generate UberMESH” option.
3. Set the subdivision levels to the desired number (there is no separate U and V setting for UberMesh).
4. Click the OK button to accept the settings and close the dialog.
5. Choose the desired UberNURBS or UberMesh object.

The object will now be placed into UberNURBS edit mode.

6. Double-click to exit the UberNURBS edit mode.

Caution: Too high values can cause memory requirements to be excessive.

The UberMESH object will now appear in the Universe Modeler (ACIS) environment. It is drawn in polygons or shaded polygons, depending upon your display settings, accessible via control-click (*PC*- right-mouse click).

### Outline Shading for UberNURBS and UberMESH Models

When exporting UberNURBS and UberMESH the last item to consider is whether or not you intend to use outline shading in Universe Animator. If you do, then you need to make sure that the Weld Vertices check box in the Global tessellation settings dialog is active. Otherwise, the individual faces will appear outlined in your renderings.

### 23.10 Hints on Working with UberNURBS

- As your UberNURBS gets more complex and loop dots begin to get lost with all the vertexes, you'll want to zoom in closer and use the orbit feature "O" to zero in on your loop.
- The size of your pick aperture will decide how much of the area round where you pick will be included in the subdivide, so you may want to tighten up the aperture setting. Do this by double-clicking on the Pick icon.
- If you're not sure if you have the right loop indicator dot, use the Edit Cage tool and click on the dot. All the vertexes of the loop will highlight and let you know if you have the right one or not. If you do, then switch back to your Subdivide Loop tool and click on that loop indicator.
- UberNURBS models can have creases. You can minimize creases by minimizing the occurrences of loops with less than or greater than four edges or keeping the amount of loops which share a vertex down to four. If this is not possible, you can hide the creases in natural skin fold areas, or use UberMESH models instead.
- Don't overuse the subdivision tools. They can leave a tell-tale "foot-print" if not managed well, due to the fact that most of the subdivision tools will produce planar faces which will "pull" on the underlying mesh, producing flat spots. Also, tools such as Inset Loop will produce near-perfect rings, which will also be very obvious unless you modify them.
- Save small details, such as subtle wrinkles or creases for texture maps. If you build these details into your model, and need to create morph targets down the line, you will be creating far more work for yourself than you need to.
- Successive operation of Subdivided Loop followed by Chop Vertex can reduce the area controlled by smaller regions



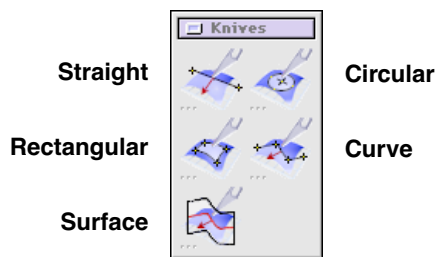


## Knives Palette

The knives represent a destructive editing tool that can cut or split any object type: solid, surface, or wire. All knives can either hack away indiscriminately and remove portions of the shape or, they can cleanly slice through an object without removing any portions of the edited object, depending on the option selected.

The newly created shapes are placed in the currently active layer. If your original shape was on Layer “A” and you cut it with a knife while the active layer was “B,” the edited object will appear on layer “B.”

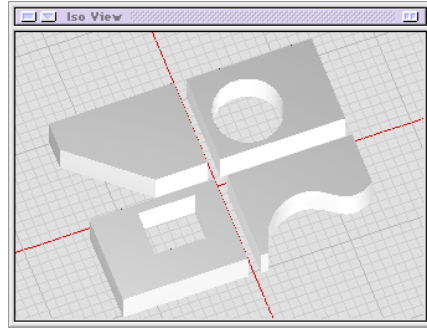
The Knife tools work on single objects only. If you want to slice through several objects, use the Boolean Union tool to bring them together and then cut or separate. When using the Union tool, the objects don’t have to be touching, they can be completely separate and still be affected by the knife, as long as it passes through them. Boolean Join tool objects cannot be cut or separated with the Knife tool.



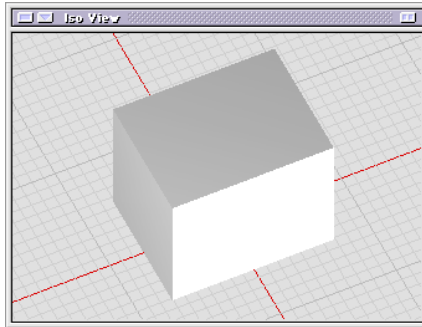
*Figure 24.0 — Knives Palette*

Knife tools used on a block, clockwise from upper left hand block: Straight, Circular, Curve, and Rectangular. In addition to these, the Surface knife will allow a face to be used as the editing shape.

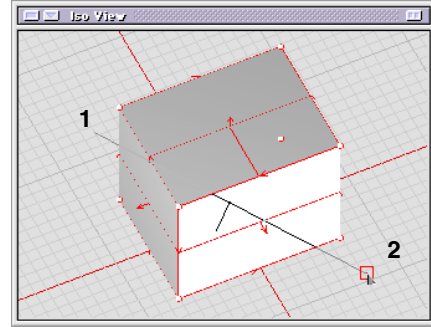
These shapes were cut in the top view and then angled to show the results.



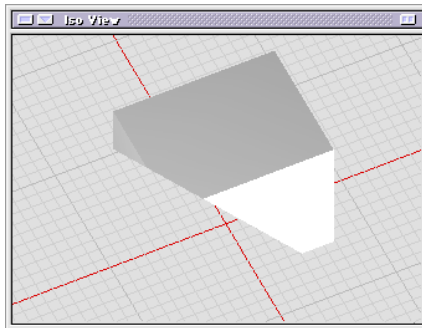
*Figure 24.1* — Straight, Circular, Curve, and Rectangular Knife tools.



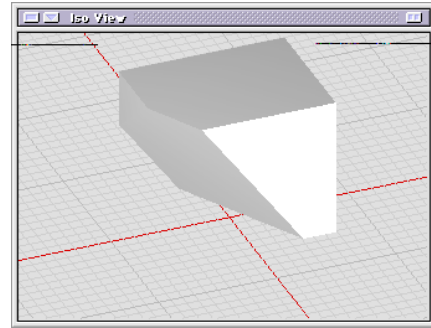
All the Knife tools work based on the view used. If the Iso view is used, the cut is in the line of site.



Here, the Straight Knife tool is used by clicking (1) and dragging (2) to establish the cutting line.



With the knife performed, the cut initially looks flat.



Rotating the object or the view demonstrates the line-of-site cut.

*Figure 24.2 — Knifing in the line of site*

Knife tools are specialized booleans, and as such obey all the limitations and features of the booleans. They can be used on wires, surfaces and solids.

The knife direction always is dictated by the view in which the operation is being performed. If the knife is used in the Iso view window, the cut will be performed along a line of site from the viewing angle.

To toggle on the split/cut option, the user can select the object, begin the cut, hold down the Shift key, and the choice opposite of what has been chosen in the option box will be performed. If “Cut” is the setting, for instance, holding down the shift key when using the knife will force the knife to “Split” instead.

**Every knife tool has the same options available by double-clicking on the tool. The object to be cut can either be split, leaving the two portions, or can be cut, removing one portion. The default setting is to split.**



*Figure 24.3 — Knife setting dialog box*

### 24.0 Straight Knife Tool

The Straight Knife tool cuts through an object with a user-defined path. The tool is destructive and permanently alters any object for which it is used.

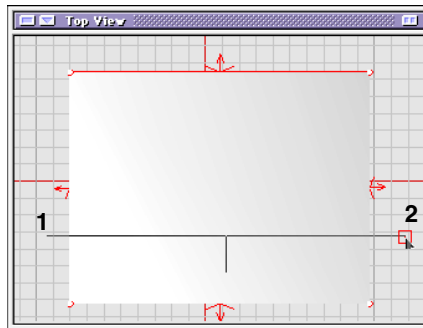
To use, choose the Straight Knife tool and click on the object to be cut. The object turns the highlight color. Next, click and drag the mouse along the path to be cut. When the mouse button is lifted, the cut is performed. Complex cuts may take a few seconds to complete.

The area to be cut is determined by the slice direction. Dragging the mouse left to right removes all the items below the cut line. Dragging from right to

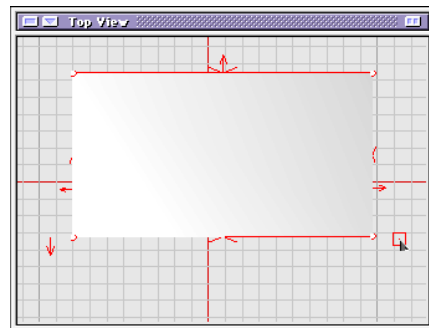
left removes all the items above the cut line. Dragging from the top to the bottom removes the part of the shape on the left. Dragging from bottom to top removes to the right. The results are placed on the currently active layer.

### Modifier Keys

Holding down the shift key when using any of the Knife tools performs a cut/split opposite of what is toggled in the option box. The “R” key will reverse the direction of the area to be removed if the “Cut” is used.



**The Straight Knife tool dragged from left (1) to right (2) cuts below the line. Note the indicator line that runs perpendicular to the cut line. This indicator points to the area that will be cut.**



**As shown above, the area below the cut line was removed.**

*Figure 24.4 — Cutting areas with the straight knife tool.*

The diagram at right shows the area that's removed by the Straight Knife. Initial click is shown as (1), the drag and release is shown by (2). The (X) indicates the area that will be removed.

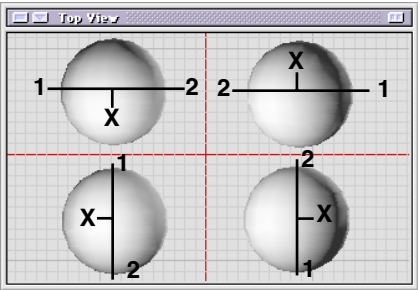
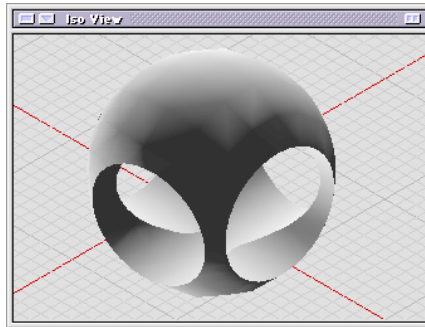
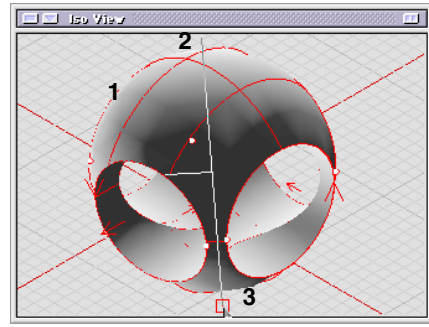


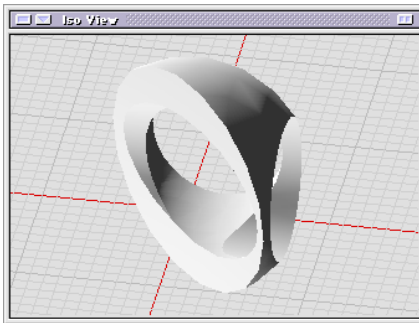
Figure 24.5 — Straight knife cut directions.



Create a solid object. Here a Booleaned sphere was used. Double click on the Knife tool and choose the “Cut” option.



Choose the Straight Knife tool. Click on the object (1), then click at the originating point of the cut and drag to indicate the cut area.



The cut is performed automatically. The view above was slightly rotated to see the cut better.

Figure 24.6 — Cutting an object

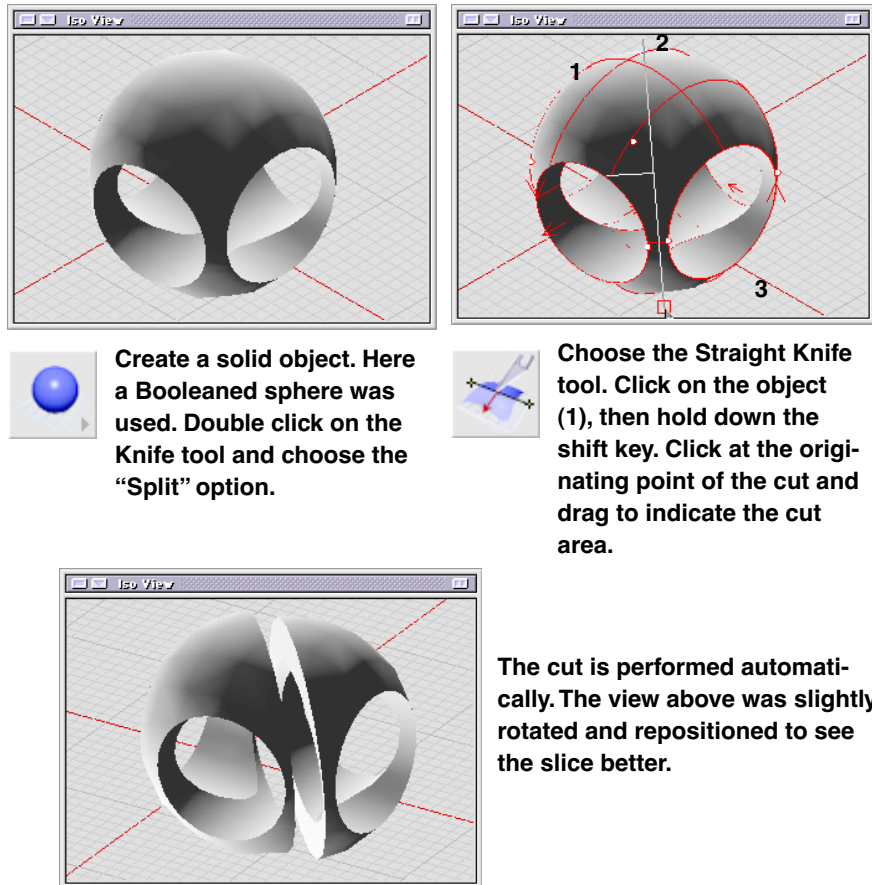


Figure 24.7 — Slicing an object

### 24.1 Circular Knife Tool

The Circular Knife tool cuts a perfect circle either into or out of any object. The tool’s default setting removes the inside of any area within the circle.



### Modifier Keys

Holding down the shift key when using any of the Knife tools performs a cut/split opposite of what is toggled in the option box. The “R” key will reverse the direction of the area to be removed if the “Cut” is used.

**Important:** *When using a modifier key, hold down the key until the operation has been completed!*

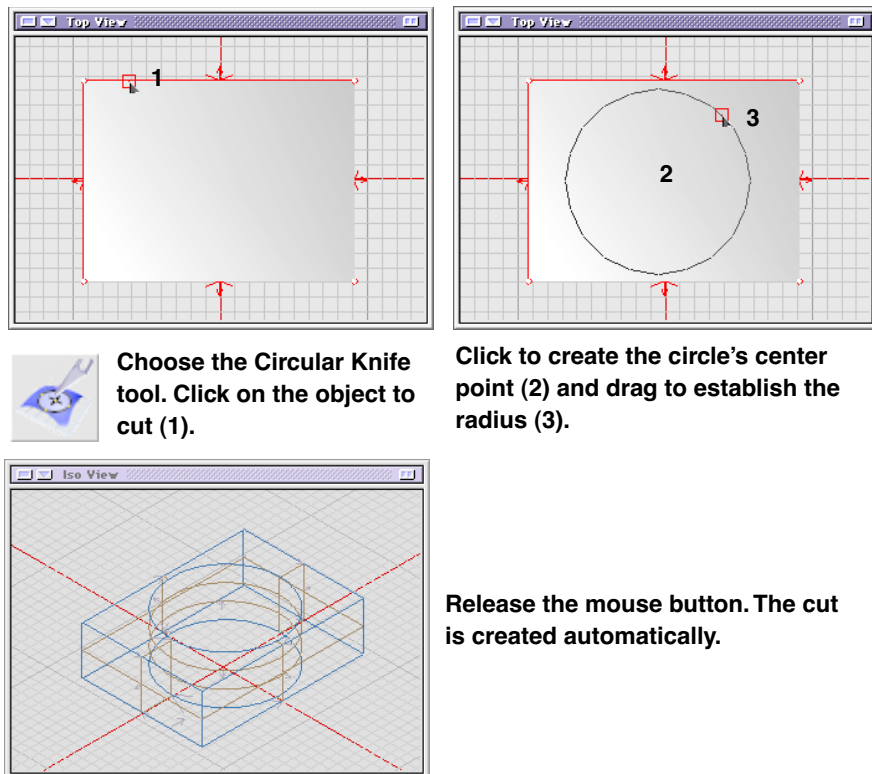


Figure 24.8 — Cutting a circle into an object

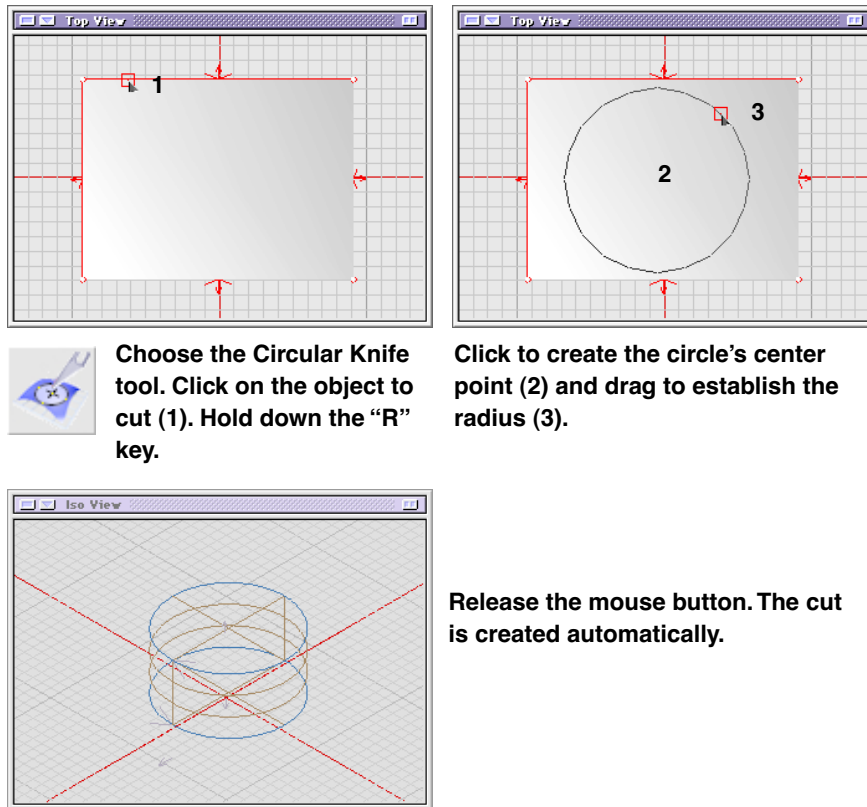


Figure 24.9 — Cutting a circle out of an object

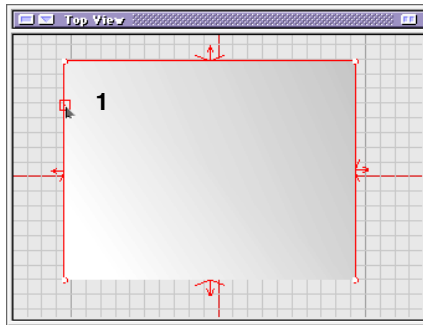
### 24.2 Rectangular Knife

The Rectangular Knife tool cuts a rectangle either into or out of any object. The tool's default setting removes the inside of any area within the rectangle.

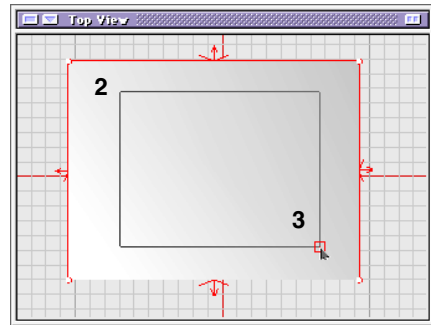
Holding down the shift key when using any of the Knife tools performs a cut/split opposite of what is toggled in the option box.

The “R” key will reverse the direction of the area to be removed if the “Cut” is used.

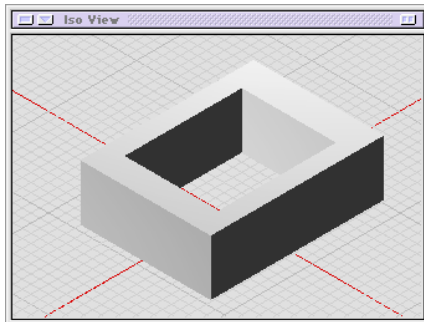
**Important:** *When using a modifier key, hold down the key until the operation has been completed!*



**Choose the Rectangular Knife tool. Click on the object to cut (1).**



**Click to start the Rectangle (2) and drag to establish the size (3).**



**Release the mouse button. The cut is created automatically.**

*Figure 24.10 — Cutting a rectangle into an object*

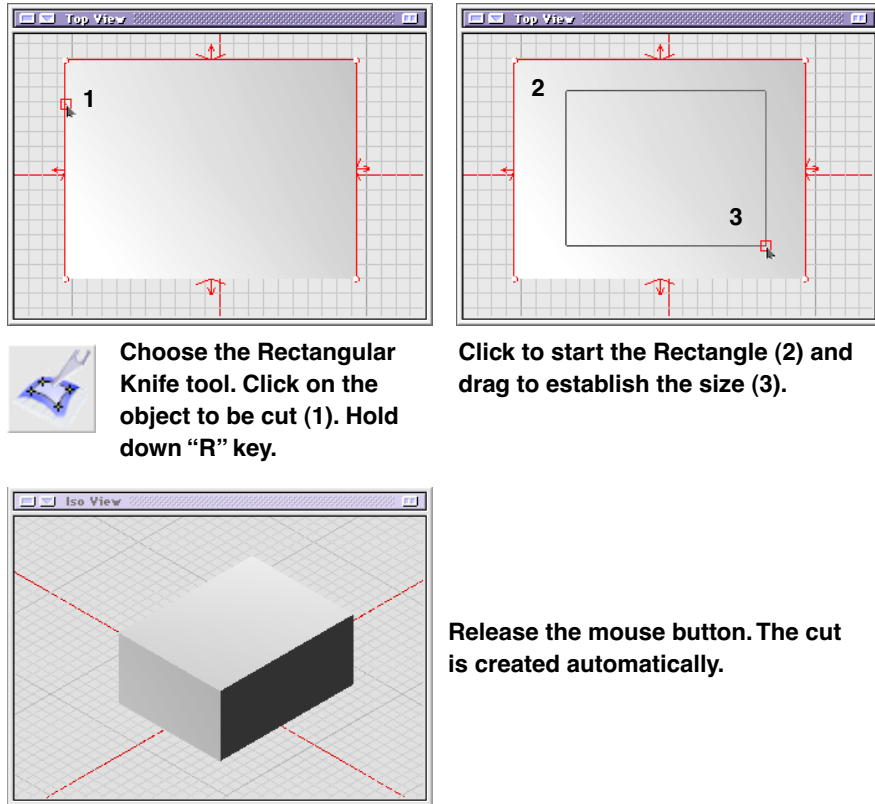


Figure 24.11 — Cutting a rectangle out of an object

### 24.3 Wire Knife

The Wire Knife tool works similar to the Straight Knife tool, but the Wire Knife tool allows the user to create a curve to be used as the cutting mechanism. The wire direction determines the portion of the object that will be cut, following the same rules as the straight cut (page 456).

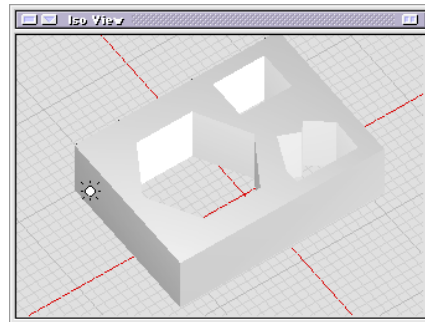
To use the tool, create an object and a wire to cut it. Choose the Wire Knife tool. Click on the object, then the wire. The cut then is performed automatically.

Either the Bezier or the NURBS Curve tools can be used, but a curve must be “legal,” meaning that it cannot loop back onto itself. The curves can be open or closed. Any of the Wire Primitive tools can be used as well. For instance, the circle wire primitive may be used instead of the Circular Knife tool, allowing the user to precisely place, reposition, and scale the circle to be cut before the operation is performed.

### Modifier Keys

Holding down the shift key when using any of the Knife tools performs a cut/split opposite of what is toggled in the option box. The “R” key will reverse the direction of the area to be removed if the “Cut” is used.

**The Wire Knife tool also can be used with Wire Primitive tools. Here the Regular Polygon, the Rectangle, and the Polyline tools were used to make these cuts.**



*Figure 24.12 — Using the wire knife tool with wire primitive tools.*

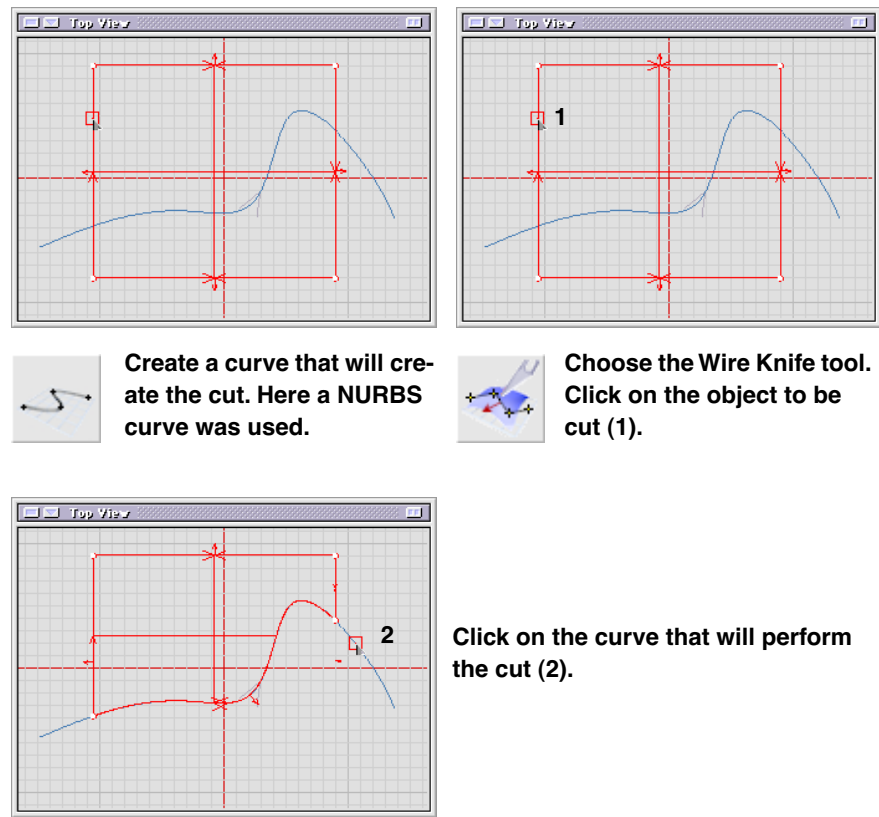


Figure 24.13 — Cutting curves with the knife tool.

## 24.4 Surface Knife

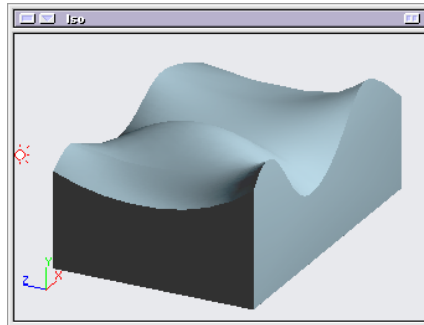
Not all cuts have to be straight lines. A face can be used as the cutting element using the Surface Knife tool. The face doesn't have to be planar and can be an undulating surface.

To use the tool, create two shapes, making one of them a surface shape. The surface shape can be created using any of the tools, such as Skin, or it can be extracted from a face of an existing object. Place the surface shape where you want the cut to be performed in relationship to the body that will be edited. Choose the Surface Knife tool, click on the body, and then on the surface. The knife will process automatically after clicking on the surface.

Surface shapes can be used to knife any type of shape, solid or surface, UberNURBS or wire.

### Modifier Keys

Holding down the shift key when using any of the Knife tools performs a cut/split opposite of what is toggled in the **option box**. The “R” key will reverse the direction of the area to be removed if the “Cut” is used.



*Figure 24.14* — Using a surface to edit a solid shape can create a complex and otherwise difficult to create shape.

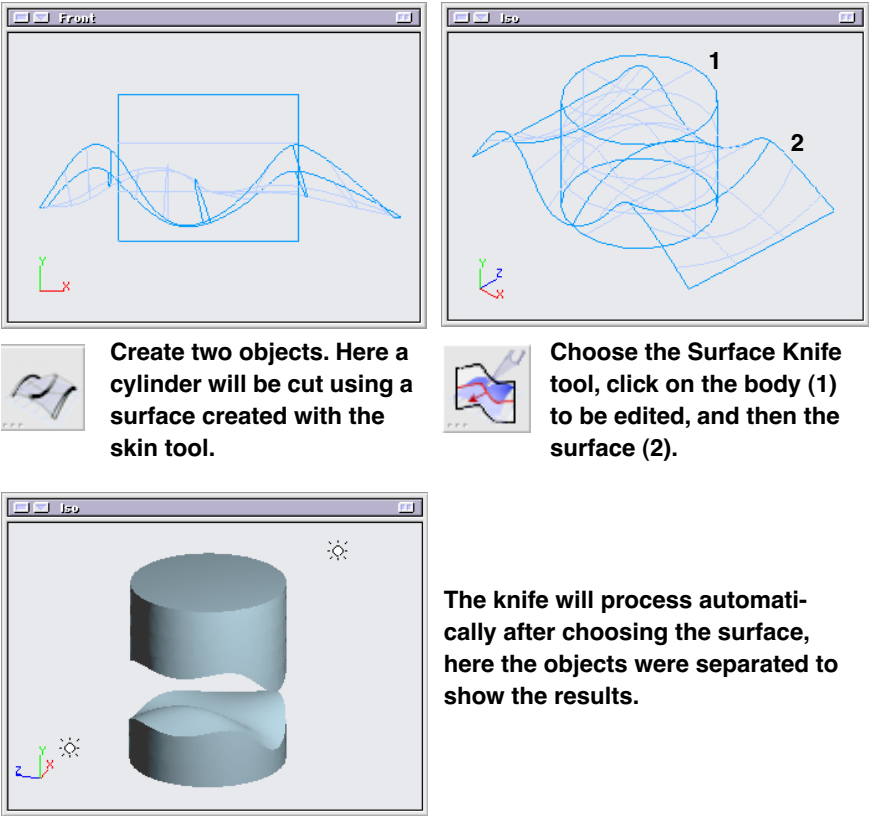


Figure 24.15 — Knifing an object with a surface



## Wire Editing Palette

Wires are a group of edges with no faces. If a solid is created, and all the faces are uncovered, you will be left with a wire body. All curve primitives that are unfilled are wire bodies. NURBS and Bezier curves are wire bodies. As the interaction between objects gets more complex and the need to alter the wires in objects arises, Wire Editing tools come in handy. For instance, Skinned objects like having the “ribs” all going in the same direction. If they’re not, then the Reverse Direction Tool can set you right.

The Wire Editing tools play an important part in wire editing, such as Polyline to NURBS, Convert to Single Spline, and Reverse Direction. They also are important for creating new wires from existing objects, such as Convert Loop to Wire, and Offset Planar Wire.

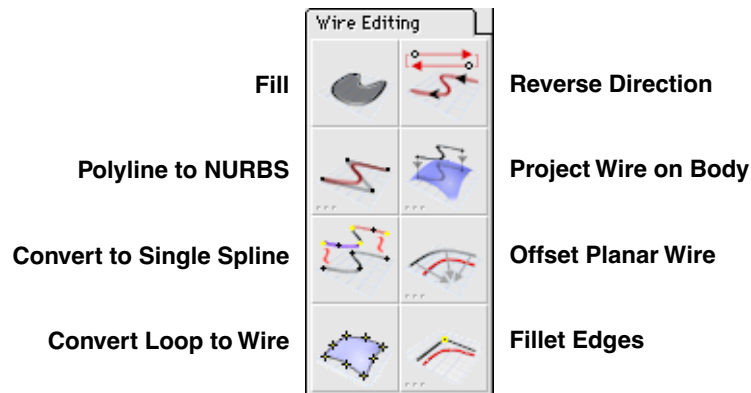


Figure 25.0 — Wire Editing Palette

Since Wire Editing tools work on wires, the program knows that you are looking for wires, so you don’t have to switch the **Pick Filter** when using the tools. In the event that you have several wires lined up on top of each

other, you might want to use the Iso view and Orbit feature to find an angle where you can click on the edge you want to pick.

### 25.0 Fill Tool

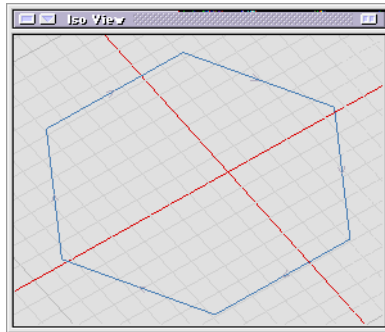
To place a face on a closed wire object, use the Fill tool. Closed wire primitives can be created with or without fills, depending on the setting in the **System Preferences/Creation menu setting**. Closed Bezier and NURBS curves are the exception. After creating one of these, you have to manually fill in the wire if you want it to have a face. Only planar wires can be filled.

A closed wire given a face becomes a two-sided surface object. A two-sided surface object that is extruded, revolved or swept becomes a one-sided solid. Whether a body is made 2-sided or single sided depends on the preference **“Make 2-sided”** accessible in most **curve-primitive dialog boxes**.

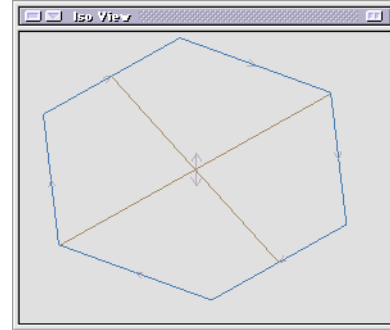
If the wire was not given a face and then was extruded, it would become a two sided surface shape.

To use the Fill tool, create an unfilled wire object using either the Wire Primitive tools, or the Bezier or NURBS curves. Choose the Fill tool and click on the wire.

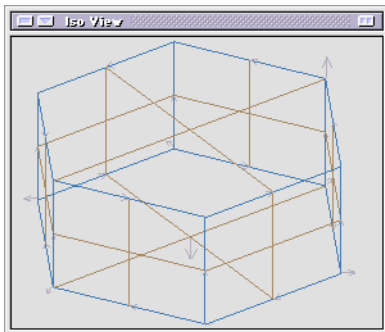
The Fill tool is non-destructive and creates a new object, leaving the original wire behind. The new object is placed, you guessed it, on the currently active layer.



**An unfilled Regular Polygon...**

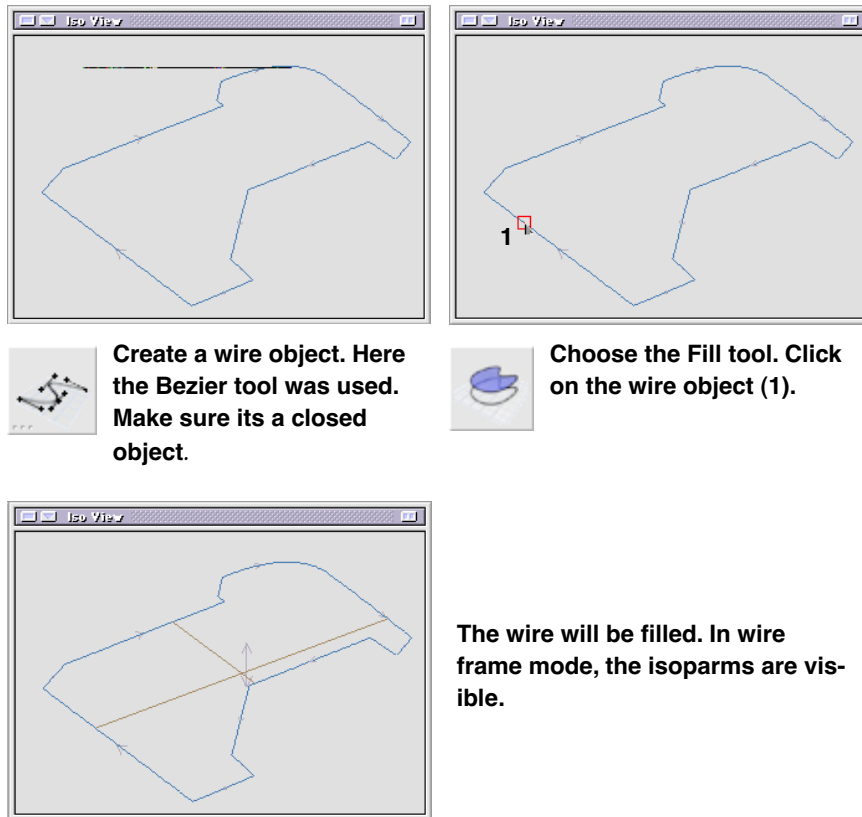


**... has a face applied and becomes a two-sided surface object...**



**...when extruded, it becomes a single-sided solid object.**

**Figure 25.1 — Changing two-sided surface objects to single-sided solid**



*Figure 25.2 — Filling a closed wire object*

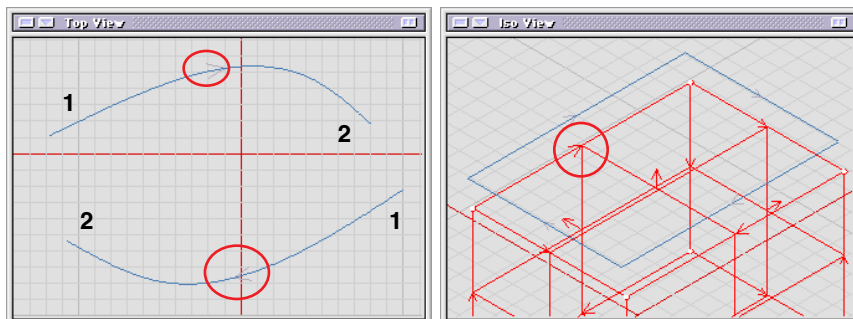
### 25.1 Reverse Direction

All wires have a direction. Sometimes the direction is created automatically by the computer when creating a form, like a wire primitive. Other times, the direction is determined by the actual direction a line is drawn.

In many cases the direction of the wire won't affect you at all, but, in some instances, the direction of the wire makes all the difference in the world. For example, if you're creating a skinned object and the direction of the wire ribs aren't in the same direction, the shape will twist. If you create all your ribs from scratch, you can control the direction by the way you draw the lines (see diagram at left). But if you are building from existing wires and the wires aren't jiving, the Reverse Direction tool comes to the rescue.

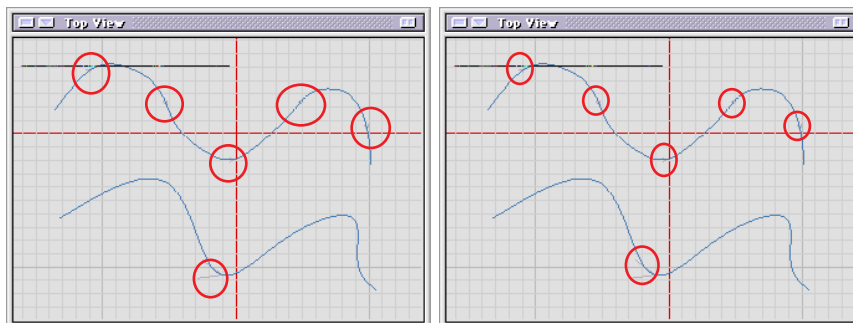
To use, select the Reverse Direction tool. Click once on the wire or loop you want to alter. Small arrows on the wire or loop indicate the direction of the line. If the arrows are too small for you to see, zooming in enlarges them. When you click on the wire or loop, you will see the arrows change direction.

The tool works on any wire, including the Curve Primitives, Bezier Curves, NURBS Curves. Reversing a wire reverses the direction of each edge that makes up the wire.



When you draw a line, the direction is determined by the way you draw it. The lines above begin at (1) and end at (2). The direction, indicated by the small grey arrow, goes in the direction of creation.

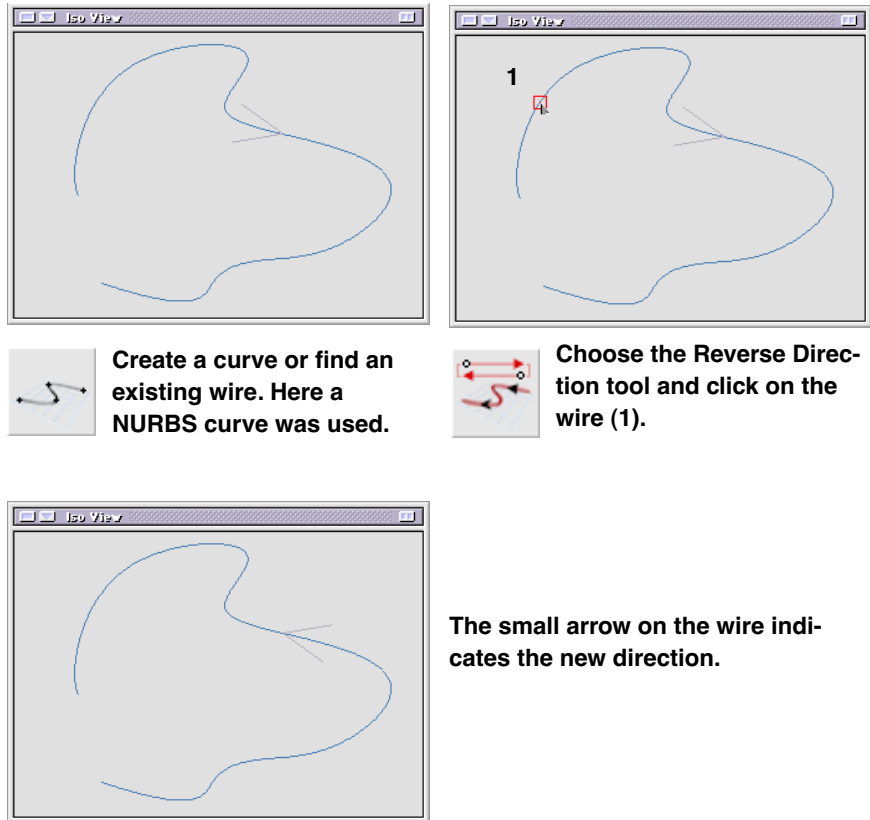
The block (above) had a wire made from the top loop, it was moved up slightly to view the direction of the loop better. This loop's direction goes clockwise.



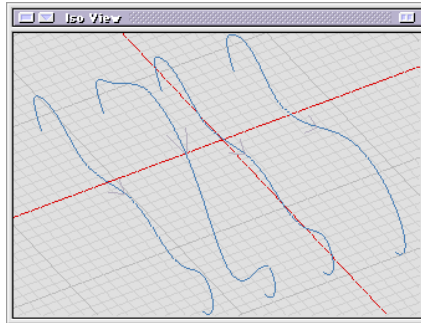
The piece-wise Bezier curve on the top has each segment marked by the direction arrows. The NURBS curve below is a single spline with one direction arrow.

Using the Reverse Direction tool on the Bezier curve reverses the direction of all the segments. On the NURBS curve, it reverses the single segment.

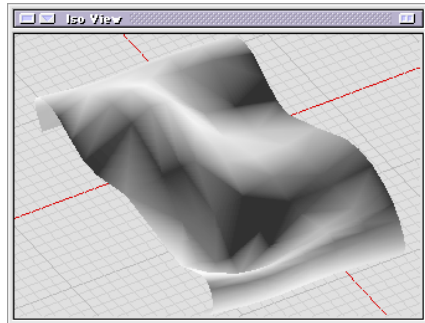
*Figure 25.3 — Reversing direction of segments on NURBS curve.*



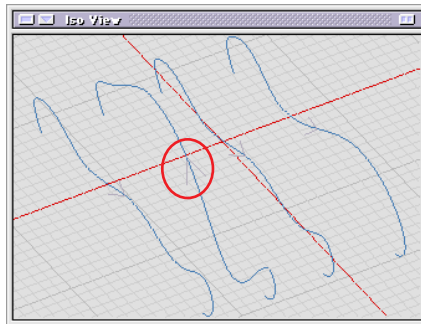
*Figure 25.4 — Changing wire direction*



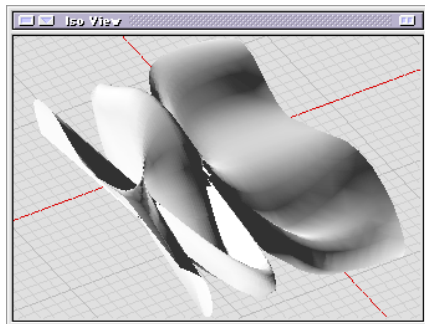
**Wire ribs with all the wires in the same direction...**



**...result in a smooth skin. The skinning operation demonstrates the importance of the wire's direction.**



**Wire ribs with one in the wrong direction...**



**...results in a twist.**

*Figure 25.5 — Wire direction*

### 25.2 Polyline to NURBS

This tool converts a polyline created with a wire primitive into a NURBS curve. The original wire remains and a new curve is created. The tool works with unfilled polylines, rectangles and regular polygons.



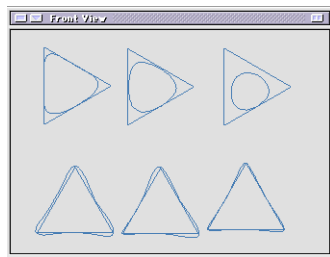
The new shape loses its edges and rounds out all the corners. If specific sharp corners are still needed, the Edit NURBS tool can be used. By holding down the “W” Key, the weight is increased to pull the line back into its original shape.

To use the Polyline to NURBS tool, choose the tool. Click on the polyline.

### Options

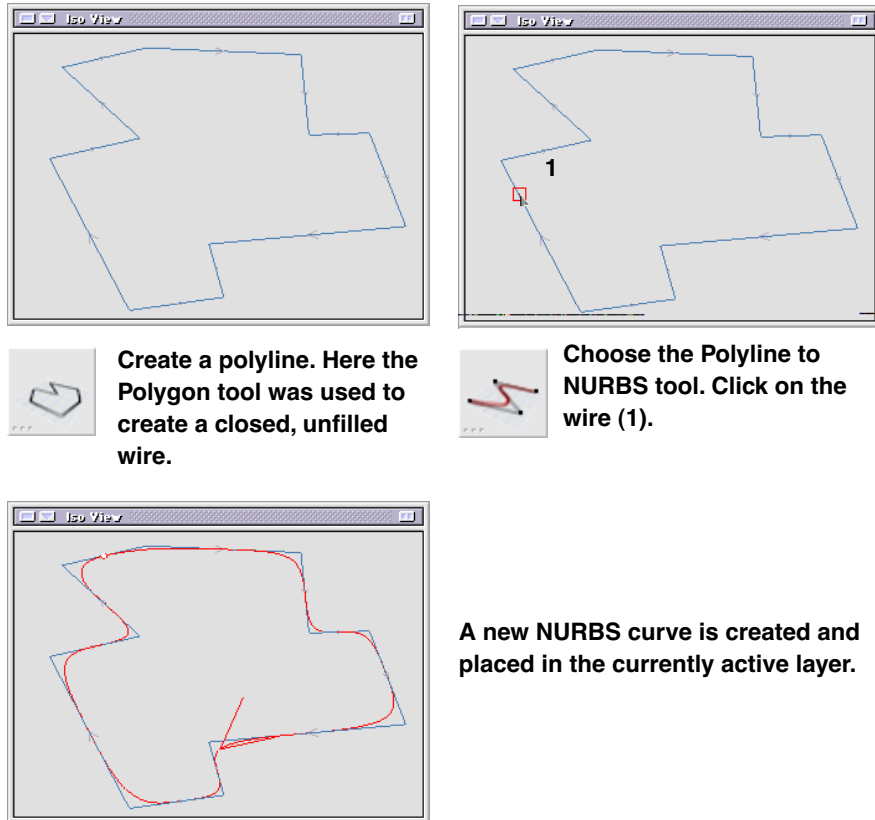
By double-clicking on the tool icon, you can set whether the polyline is used as CVs hulls or the knot position for the NURBS spline that will be created. You can also alter the degree of the NURBS curve. For the **“Use as CV” option** a lower setting will result in a tighter curve that follows the original polyline closer. A higher degree setting will result in a smoother curve. With the “Use as knot” setting the higher the degree setting, the tighter the curve in relation to the original polyline.

Instead of interpolating (passing through) each point of the polyline, it is possible to fit a curve within a certain tolerance. The parameterization of the curves is an important aspect in the design of a good surface.



Double clicking on the tool brings up the tools options box where the degree and how the polyline is used can be altered. The first three triangular polylines were used as CVs, with degree settings of 2, 4 and 6. The second row used the polyline as knots, with degrees set at 2, 4 and 6.

*Figure 25.6 — Using cv options in polyline to NURBS tool.*



*Figure 25.7 — Converting polyline into NURBS curve*

### 25.3 Project Wire on Body

A useful construction tool, the Project Wire on Surface allows the user to re-create the face of a surface or solid object by projecting a wire onto it. The new object, while geometrically identical, has been topologically altered.

No new shape is created. The original surface shape is altered, so copy it if you plan to use an unaltered version of the surface.

Any curve, Bezier, or NURBS can be used as the projection wire, as well as any unfilled wire primitive.

To use Project Wire on a Surface, create a wire primitive and an object to be edited. Choose the tool. Click on the wire, then select the object. Drag a projection vector to indicate the distance and the direction of the projection. If you want to put in numeric input, double click on the tool icon and put in your settings and click OK. The new face will be created after the mouse button is released.

The Projection tool also works on solid shapes. The wire is projected on all the faces that the vector/drag reaches.

### Options

The user has the option to **“Leave as wire”** which will create the projection as a new wire on the body, but will not create a face on the object.



The Project Wire dialog box gives the user control over the direction and length of the projections. This can be done manually as well by clicking and dragging. The destiny of the projection can be altered by either letting the projection remain as a wire, or making it a face.

*Figure 25.8 — Project wire dialog box*

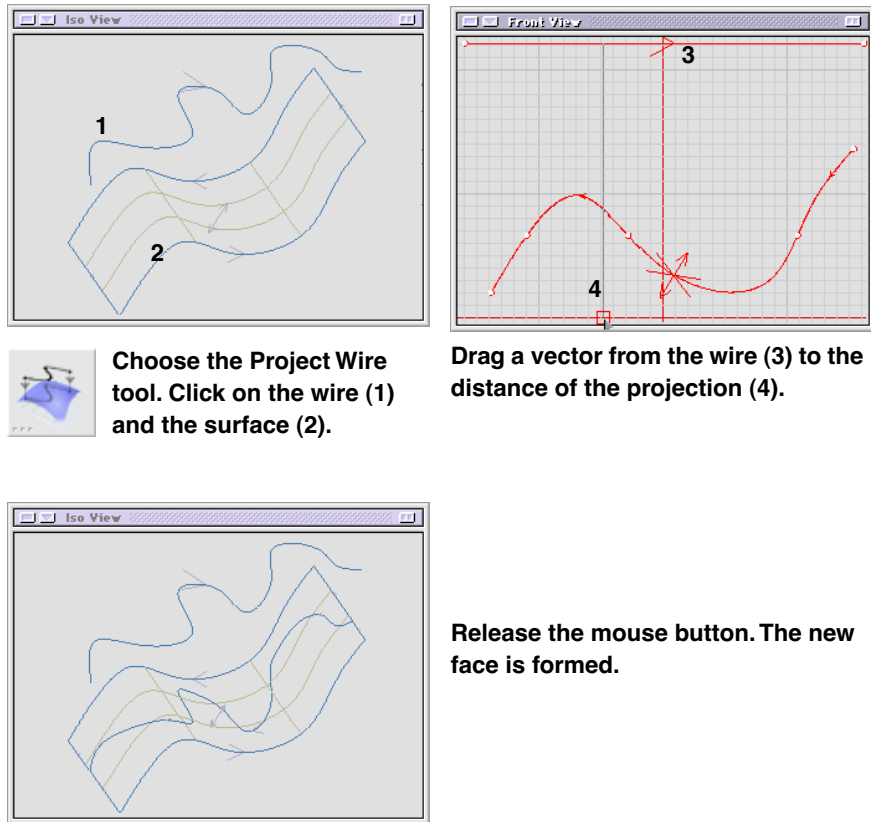


Figure 25.9 — Projecting a wire onto a surface.

### 25.4 Convert to Single Spline

Complex wire shapes make for complex models, which in turn make for more unwieldy models, longer render times, and longer screen redraws.

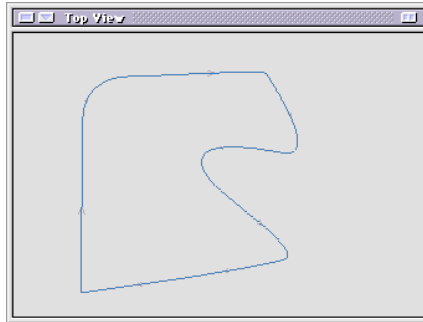
One way to simplify objects is to reduce the amount of information making the initial wire before extrusions, lathes, skins or other shapes are formed.

The Single Spline tool converts a piece-wise Bezier curve, which may have any number of segments, into a single NURBS curve. Each segment of a piece-wise Bezier curve adds a face to an object. A single NURBS curve gives a single face to an object, reducing the size of the file and the complexity of the object without altering its shape.

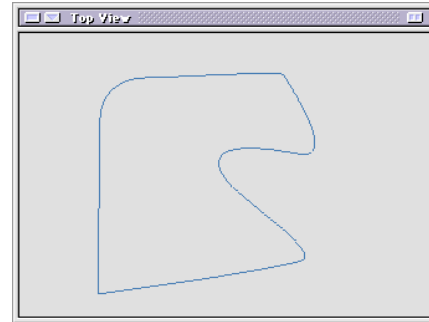
There are restrictions to the type of wires that the tool can be used on. Complex wires, like branches, multi-loops and directionally inconsistent wires should not be converted into single spline wires.

To use, choose the Single Spline tool and click on a piece-wise wire, like a Bezier curve or wire primitive. The wire will convert, altering the original wire, but not creating any new objects.

For more information on how using the Single Spline alters the topology of an object, see page 173.



**This piece-wise Bezier has eight segments.**



**Using the Single Spline tool, the Bezier is converted to a single NURBS curve.**

*Figure 25.10 — Single Spline NURBS*

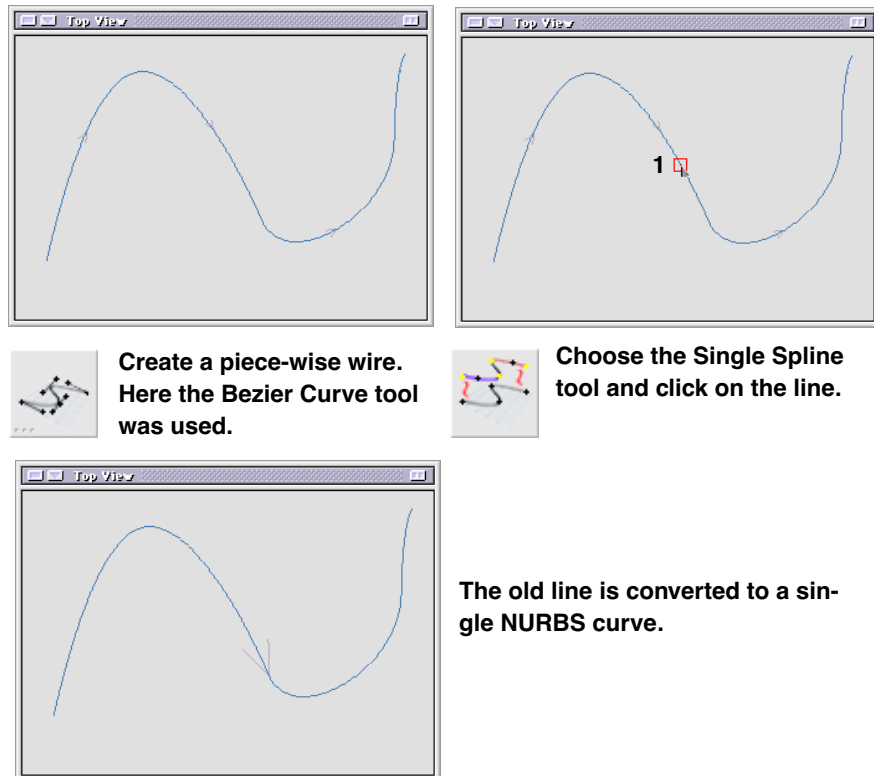


Figure 25.11 — Converting a piece-wise Bezier curve into a single NURBS

### 25.5 Offset Planar Wire

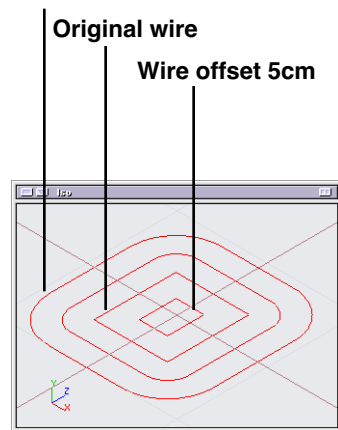
To create a wire that runs exactly parallel, but offset to an existing wire, use the Offset Planar Wire Tool. The resulting wire is a perfect offset that is impossible to create using the Scaling tool, as the new wire follows the original exactly.

By double-clicking on the tool icon, the tool settings dialog is invoked, where the offset value can be entered. Positive numbers offset below and to the right, negative numbers offset above and to the left. The amount of the offset is determined by the user defined measuring system.

After entering the desired offset distance, the new wire will be created in the currently active layer.

The tool works on any wire that is planar.

### Wire offset -15cm



Double-clicking on the tool icon brings up the option box which allows for the input of offset numbers. If the line is a closed rectangle, a negative number would result in a larger, rounded-corner wire. A positive number would create a smaller rectangle.

*Figure 25.12 — Wire offsetting*



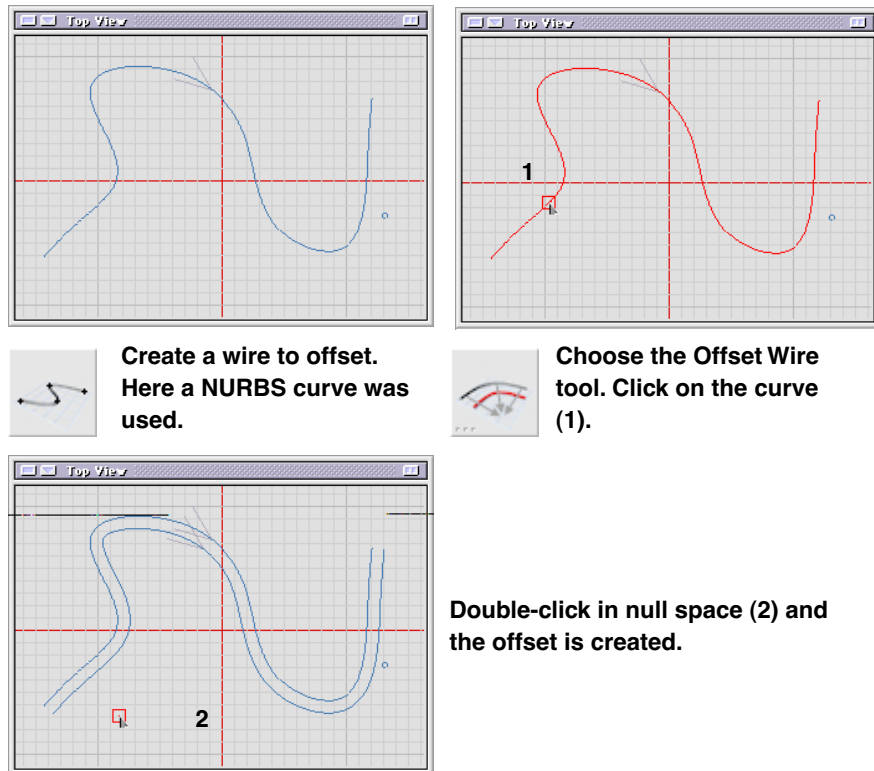


Figure 25.13 — Creating an offset wire

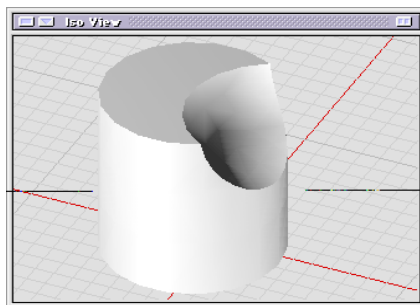
### 25.6 Convert Loop to Wire

The loop of any face can be used to create a wire, that can then be used for future shape creation. Wires can be made from any body that has a loop, which covers pretty much any kind of solid or surface object. If the edge filter is set, wires can be made from edges of a body.

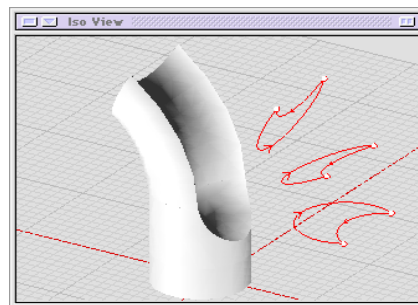
The creation of the wire does not affect the existing shape. The existing shape remains untouched and on its current layer, while the new wire created from the loop is placed on the currently active layer.

In the example below, a sphere Booleaned from a cylinder created a unique shape on the top. To create a continuation of this shape, the Convert Loop to Wire tool is used, generating a new wire shape. This wire shape is then duplicated, repositioned, and rotated. The ribs created from the original face are then skinned, creating a smooth arch of the same shape with no transition. The ribs were then placed on a “**Parts**” layer and hidden for future reference.

If you use other wires in conjunction with the converted loops in operations such as skinning, keep in mind that the wires need to be going the same direction. For more about wire directions, see page 472.



The loop at the top of this shape (crescent) was converted...



...and the resulting wire duplicated and repositioned to create this skin.

*Figure 25.14 — Converting a loop to wire*

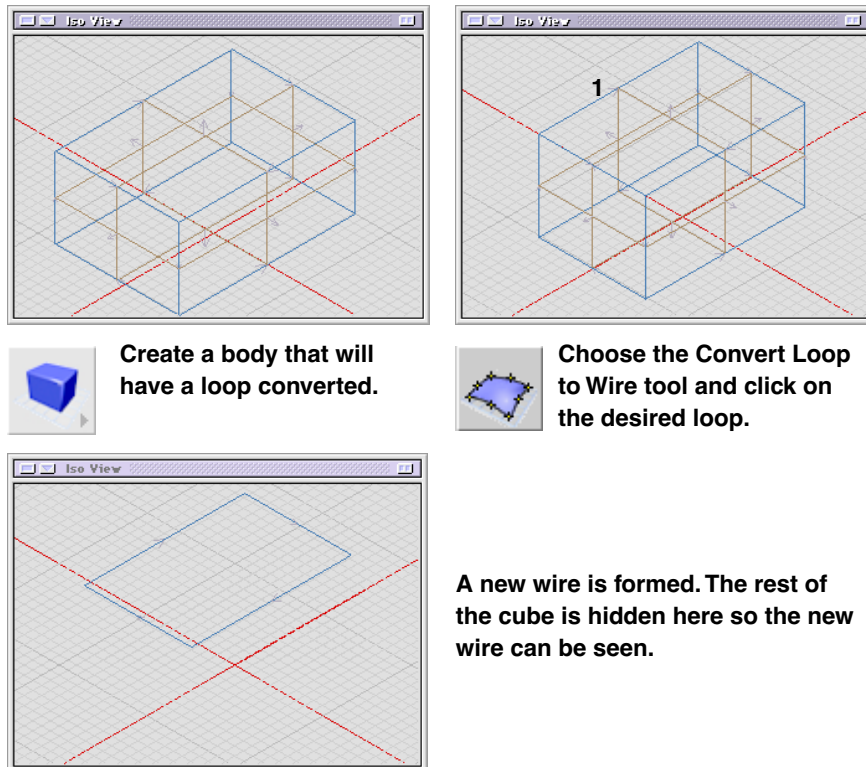


Figure 25.15 — Creating a wire from a loop

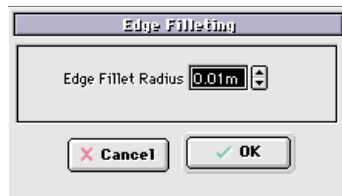
### 25.7 Fillet Edges

This tool is used for rounding corners in simple wire bodies that are opened or closed.

To use, choose the tool and pick the edges that meet each other at the vertex. After the second edge is selected, a rounded corner is created.

### Options

The radius of the fillet is specified by the user in the **options box** for this tool.



Double-clicking on the tool icon brings up the tool's option box. Here the radius for the fillet is entered.

*Figure 25.16 — Edge filleting option box*

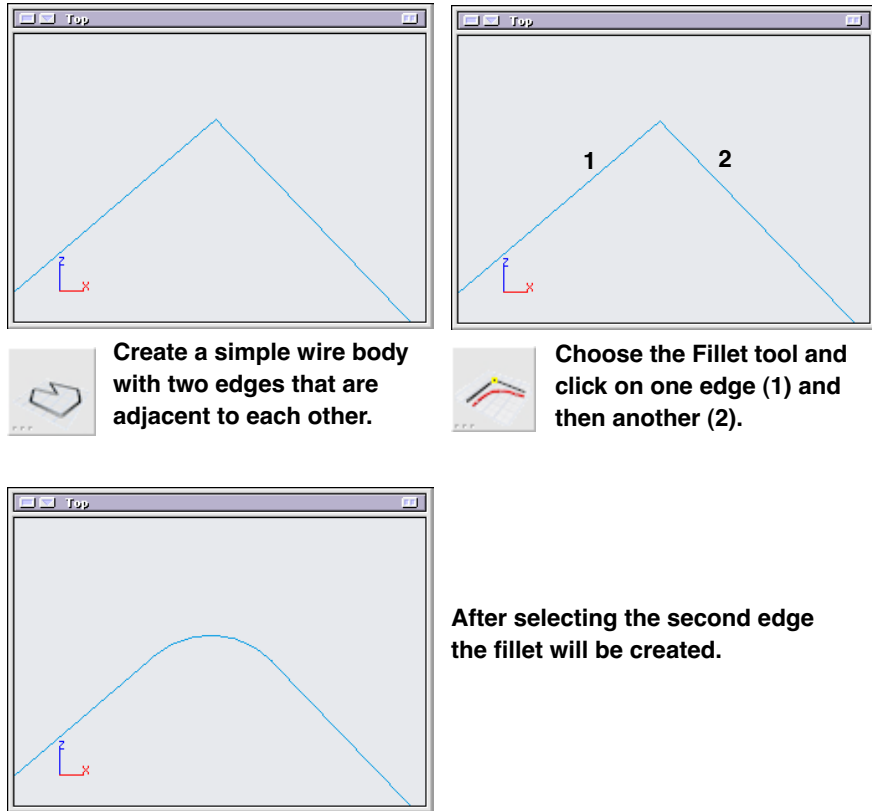


Figure 25.17 — Creating a wire from a loop

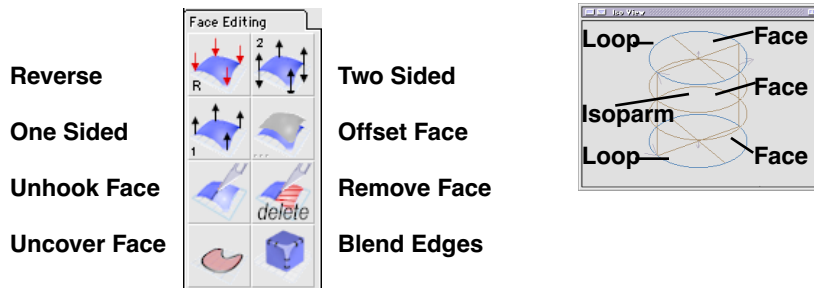


## Face Editing

Both surface and solids have faces. They're the non-wire elements that form an object's shell. The Face Editing tools affect either the physical attributes of the face or the direction of the face's normals.

By default, surface objects are double-sided and solid objects are single-sided. However, this can be changed and will greatly affect how objects are created, specifically using the Boolean tools. See "One sided objects and two sided surfaces" on page 362 for details.

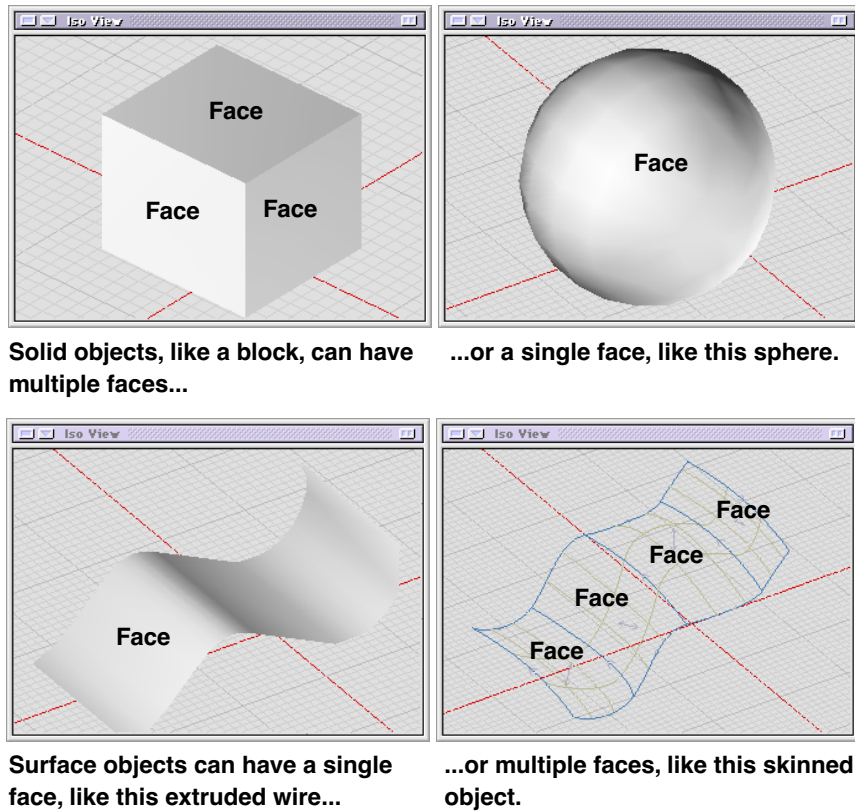
Faces can be selected by clicking on the isoparm of an object. Remember, the isoparm is a representation of the surface area and not a topological element like a loop or a face. To choose a face for editing, you may want to use a camera or Iso view so you can select a specific face.



All faces are created by loops. This cylinder primitive has three faces: the top circle, the bottom circle, and the side— which is formed by the top and bottom loops. The side shares the loops, making them co-edges. While the light grey isoparms may resemble faces, they are just indicator's of the surface area.

Figure 26.0 — The Face Editing Palette

**Important!** *If you click on a shared edge, all faces are picked, however if the edge is a boundary, the face will get picked.*



*Figure 26.1 — Solid and surface faces.*



## 26.0 Reverse Face Tool

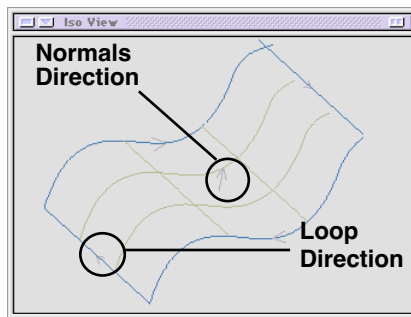
In the process of building, editing, Booleaning, and generally messing around with surfaces, it's possible to come up with a single-sided surface with normals pointing in the wrong way. The Reverse tool remedies this.

To use the Reverse tool, choose the tool and click on either the loop or an isoparm of the face. The up/down normals arrow switch positions. The direction of the loop of the face also is reversed.

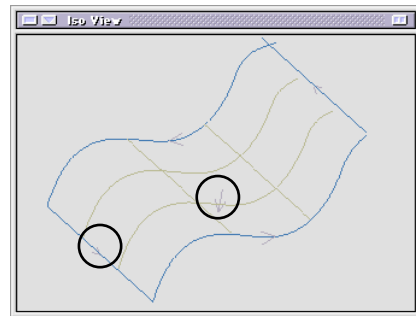
It's possible to reverse just one face on a multi-sided solid object. For instance, a cube by default has six single-sided faces pointing outward and can be altered so one of the faces is pointing inward.

If you're dealing with a surface object and the shading of the face doesn't look right, you may want to try to reverse the face.

To see the direction of the faces, go to menu **Edit/System Preferences** and look under the **Display** tab and toggle on "**Show Face Directions.**"



Using the Reverse tool on a face...



...alters the normals and the direction of the loop.

*Figure 26.2 — Changing normals direction*

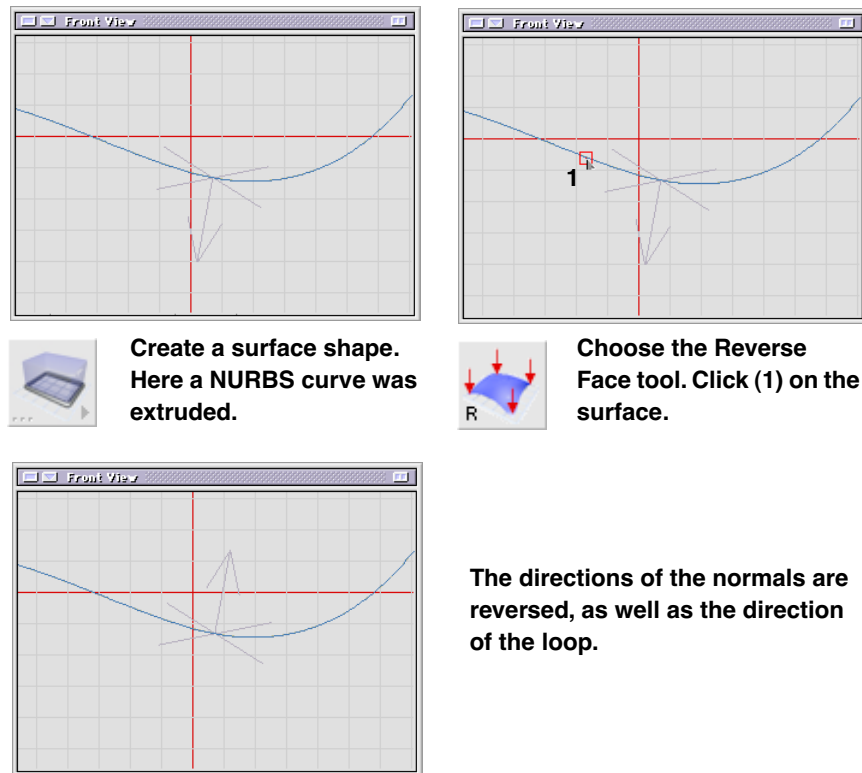


Figure 26.3 — Reversing the direction of the normals on a face

### 26.1 Make Face Two-Sided

Newly created surfaces enter the world as two-sided objects, meaning they have normals on each side. Solid objects, however, enter the world by default as single-sided objects. In the process of editing two-sided objects, such as unhooking faces, it is possible to create a surface object that is one-sided. The Two-Sided tool remedies this by altering the shape to become two-sided, with normals on each side.

The way solid objects interact with surface objects when Booleaned also can be altered if an object is single-sided or double-sided. More details on this can be found in the Boolean section on See “Subtracting a one-sided solid from a two-sided surface” on page 371.

To use, choose the Two-Sided tool and click on either a loop or an isoparm of the surface object. The normal direction arrows in the center of the shape will change from a single arrow to a double arrow.

To see the direction of the faces, go to menu **Edit/System Preferences** and look under the **Display tab** and toggle on “**Show Face Directions.**”

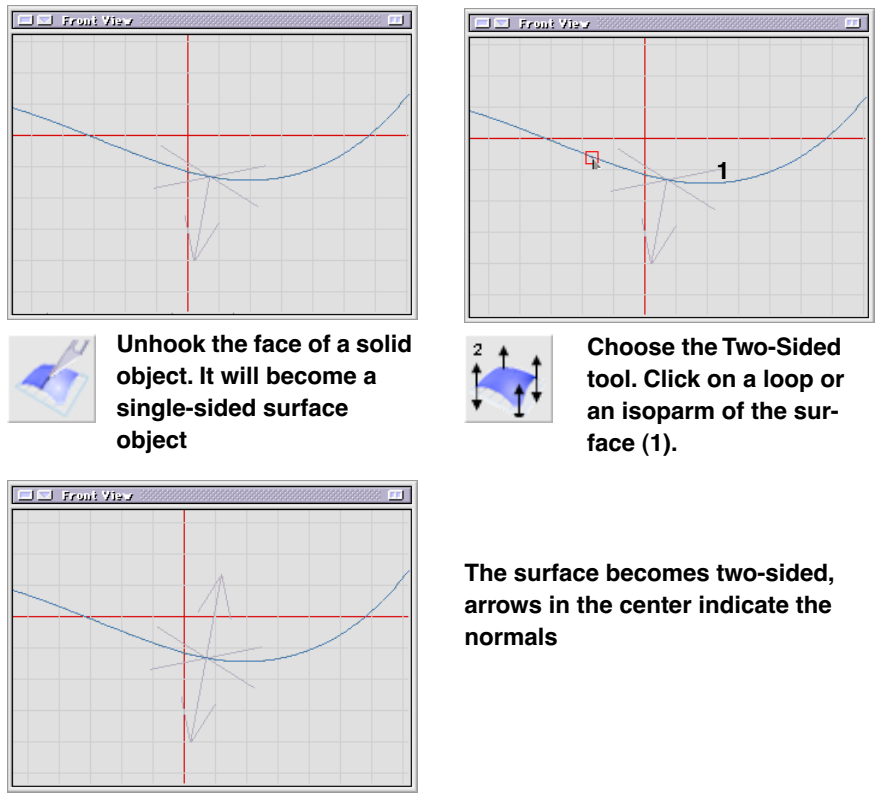


Figure 26.4 — Making a single-sided face two-sided

## 26.2 Make Face One-Sided

Newly created surfaces enter the world as two-sided objects, meaning they have normals on each side. Solid objects, however, enter the world by default as single-sided objects. In the process of editing surface objects, such as stitching, it is possible to create a solid object that is two-sided. The

One-Sided tool remedies this by altering the shape to become one-sided with normals on a single side.

If normals are not pointing in the proper direction after the One-Sided tool has been used, use the Reverse tool to correct it.

To use the One-Sided tool, choose the tool and click on either a loop or an isoparm of the surface object. The normal direction arrows in the center of the shape will change from a double arrow to a single arrow.

The way solid objects interact with surface objects when Booleaned also can be altered if an object is single-sided or double-sided. More details on this can be found in the Boolean section on “Subtracting a one-sided solid from a two-sided surface” on page 371.

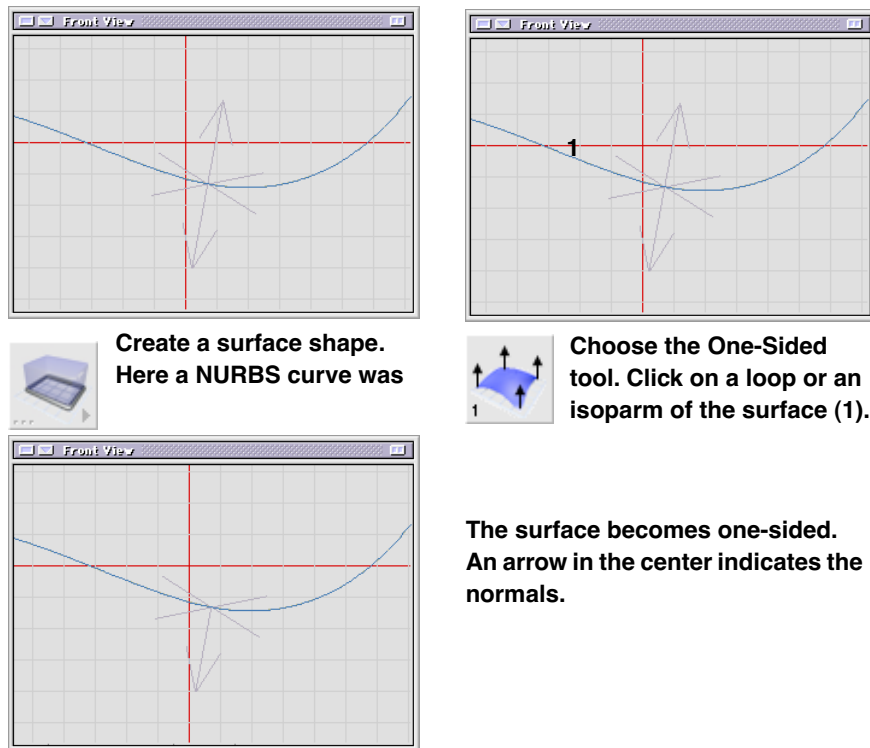
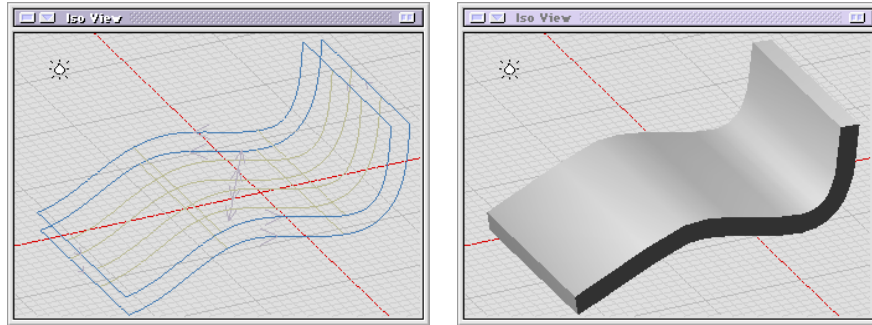


Figure 26.5 — Making a surface one-sided

### 26.3 Offset Face

This tool creates a new face that runs parallel to the original. The distance of the offset is set by double-clicking on the tool icon and inputting a distance for the offset in the **option box** that comes up. Negative numbers will go in the direction of the normal. Two sided faces also have a preferred normal. If you convert a surface from 2-sided to 1-sided, the side picked is the preferred normal side.



**Figure 26.6** — The bottom surface was offset with a positive number, creating the 2nd surface above it. The skin tool was used to create new sides.

**Important!** *The surface of the underlying face must be tangent continuous (a G1 curve).*

The new surface will have the same loop and normals directions. If you want to use the skin tool to create a face to connect the two, all you have to do is to set the Pick Filter to Edge, choose the skin tool and click on one edge, then the next.

The new surface will be placed in the currently active layer.

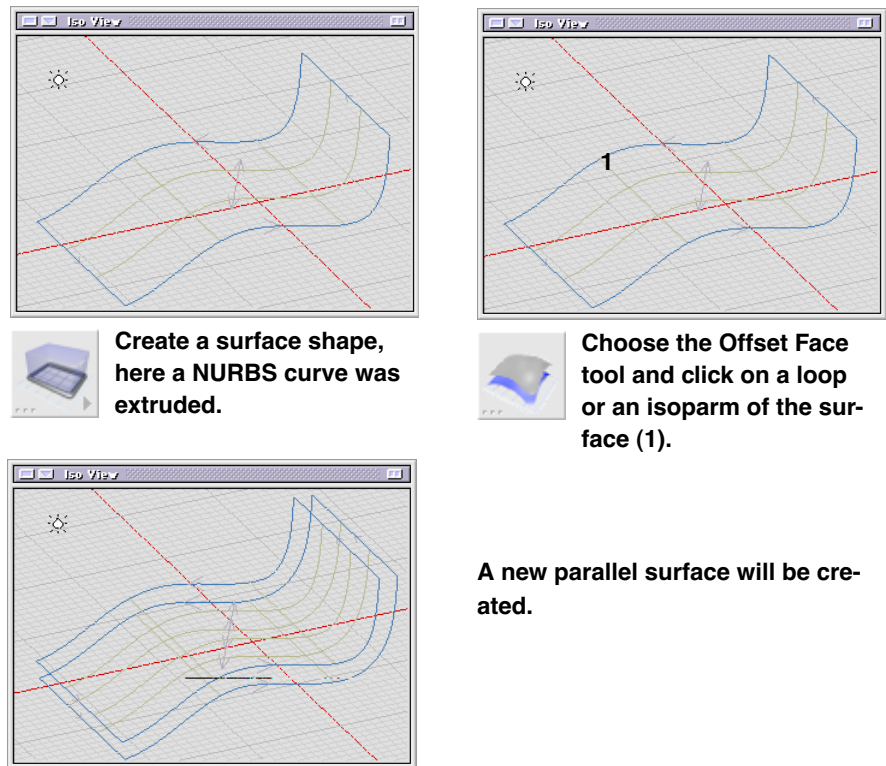


Figure 26.7 — Creating a duplicate offset face

## 26.4 Unhook Face

Faces can be detached from solid or multi-faced surface objects using the Unhook Face tool.

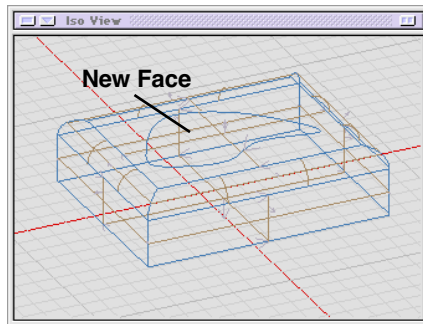


In the case of a single-sided solid object, the new face will remain single-sided, and the portion of the original solid left behind will also be single-sided.

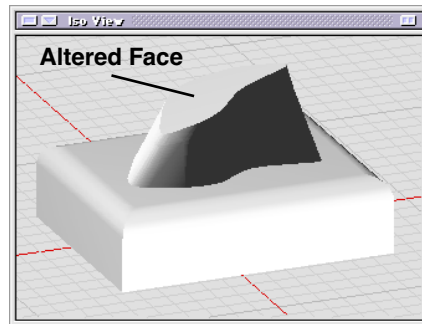
To use, choose the tool and click on the face of an object. The easiest way to do this is to click on the isoparm of an object. The unhooked loop will become a new object and will be placed on the currently active layer.

The direction of the loop on the unhooked face will reverse, so if you are planning to use the face in conjunction with future editing, like skinning, you may want to reverse the direction of the loop.

The Unhook face can be used handily with other tools that edit the topology of faces. The Project Wire tool, for instance, can be used to add a new face to a single face of a solid object, then that face can be unhooked.

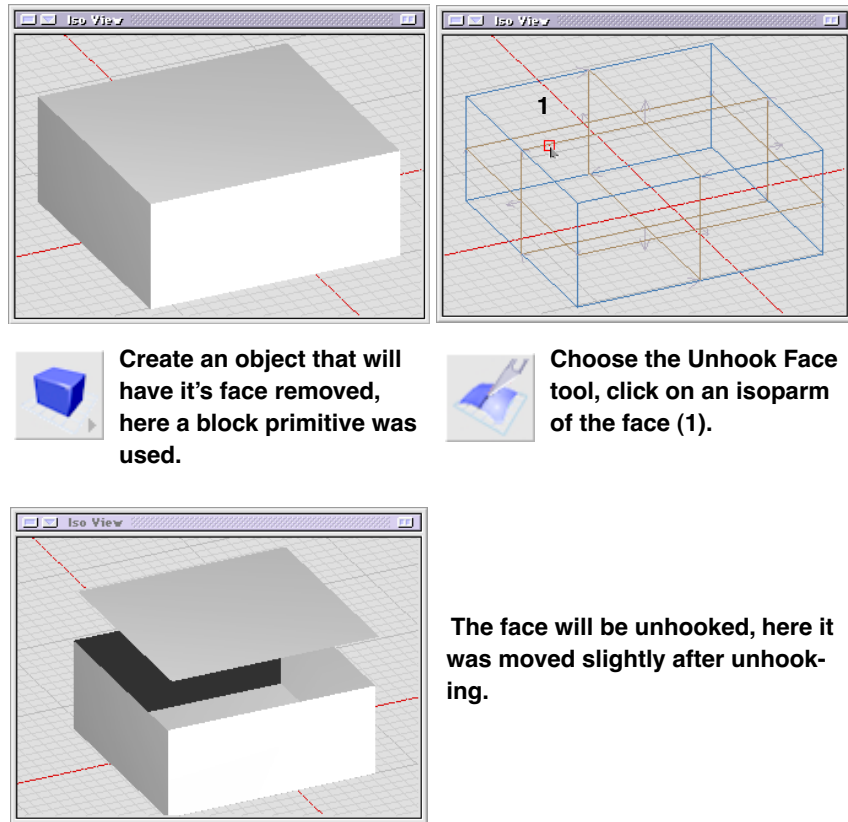


**The Project Wire tool was used to create a new face.**



**The new face was unhooked, scaled, rotated, repositioned and then**

*Figure 26.8 — Unhook Face*

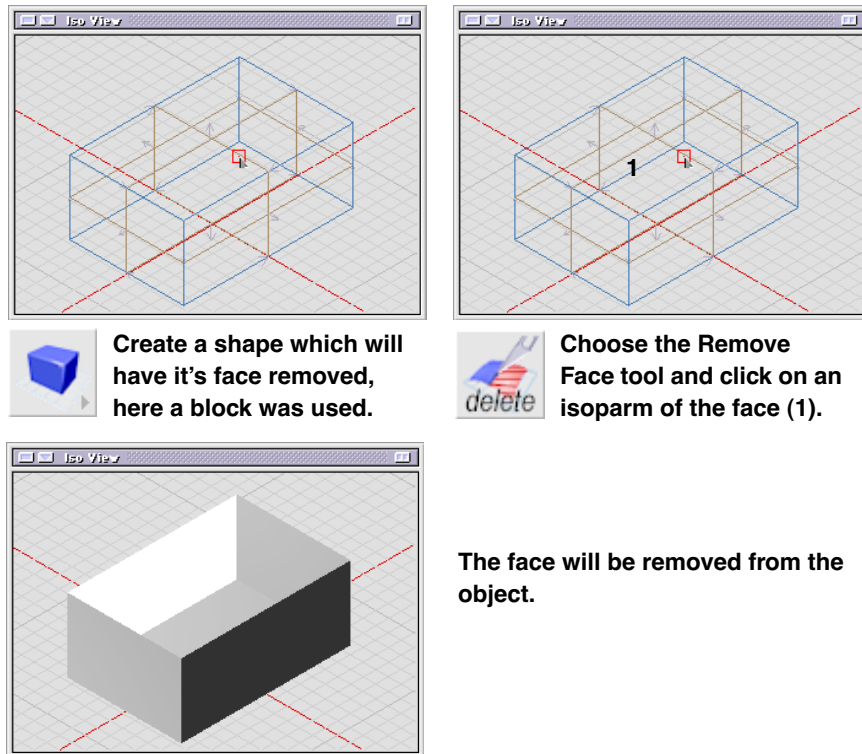


*Figure 26.9 — Unhooking the face on an object*

### 26.5 Remove Face

Faces can be deleted from solid or multi-faced surface objects using the Remove Face tool. Removes both face and wires.

To use, choose the tool and click on the face to be removed. The isoparms allow for easy selection of a face in a complex model.



*Figure 26.10 — Unhooking the face on an object*

## 26.6 Uncover Face

This tool will uncover the face, but will leave co-edge definition so the object can be covered back again. To use, choose tool and click on an iso-  
pirm of the face.

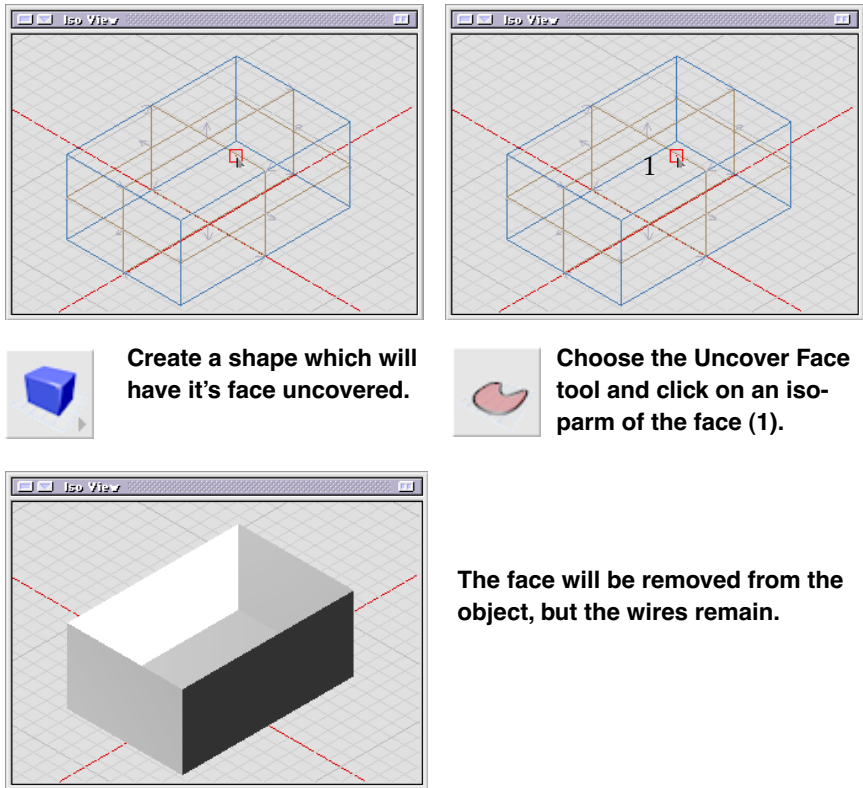


Figure 26.11 — Uncovering the face from an object

### 26.7 Enclose Void Starting at Face



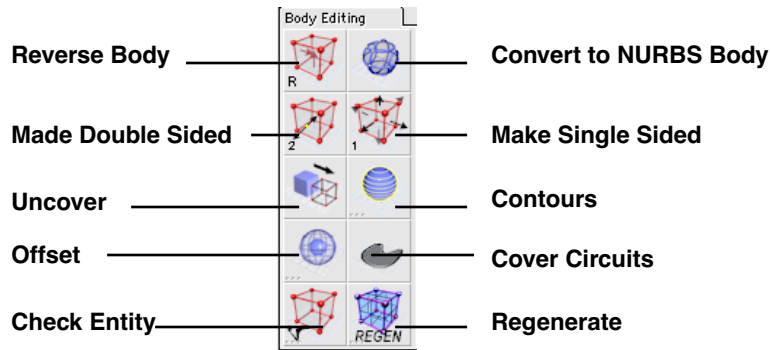
*Figure 26.12* — **Enclose Void Starting at Face tool.**

This tool is used to help make a model water-tight. One of the great things about having a solids modeler like Universe Modeler is your models will not break seams even if built out of several surfaces. Sometimes however even when the surfaces are stitched there can be a little gap or overhanging edge where they meet. This tool is used to enclose that gap and remove the unneeded edges. It is probably not going to be used too often, but it's nice to have anyway, just in case it is needed.



## Body Editing Palette

Using the tools in the Body Editing palette will change the attributes of the entire body, the tools can be used for re-configure the normals, altering the body topology.

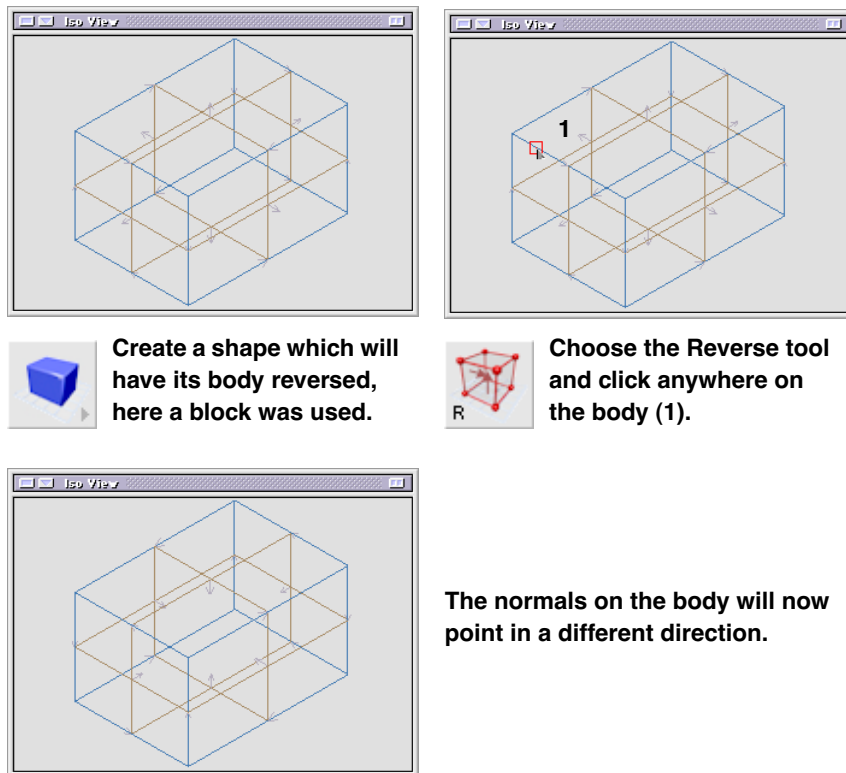


*Figure 27.0 — Body Editing Palette*

**Note:** *Inside out solids act like holes in space. When booleaned these “inverted” solids do the reverse of what normal solids do.*

### 27.0 Reverse Body Tool

Using this tool reverses the direction of the normals on a body. In the process of creation it is possible to develop a body that is “inside out.” This tool will fix the problem. There may be times when you want the normals on the inside of a solid object, if you have your camera inside the shape, for instance. To use, choose the tool and click once on the body.



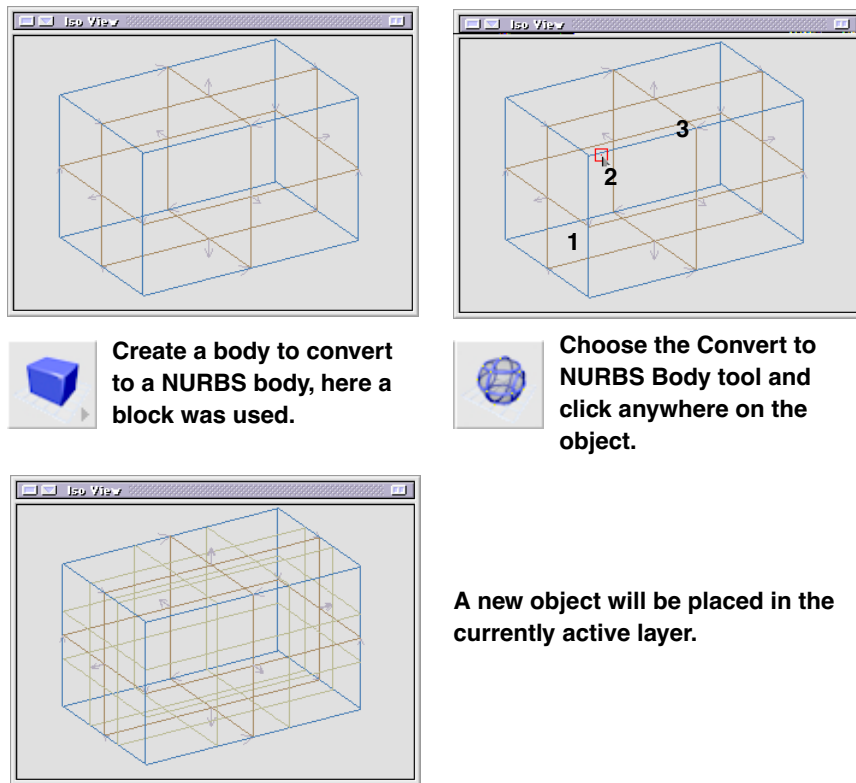
*Figure 27.1 — Reversing the normals on a body*

### 27.1 Convert to NURBS Body Tool

When you want a tool to work like the NURBS Surface editing tools, you need a NURBS body to work with. This tool will convert a body to be compatible with tools that require NURBS information.

To use the tool, select the tool and click on the body that you want to convert.





*Figure 27.2 — Converting an object to a NURBS body*

The original body will be replaced with the NURBS body.

### 27.2 Make Double-Sided Tool

Solid objects are inherently single sided, with the normals facing outward. The Make Double Sided tool will put normals on both sides of a face, making once solid objects behave like thin shells in all boolean operations. To use, choose the tool and click anywhere on an object.

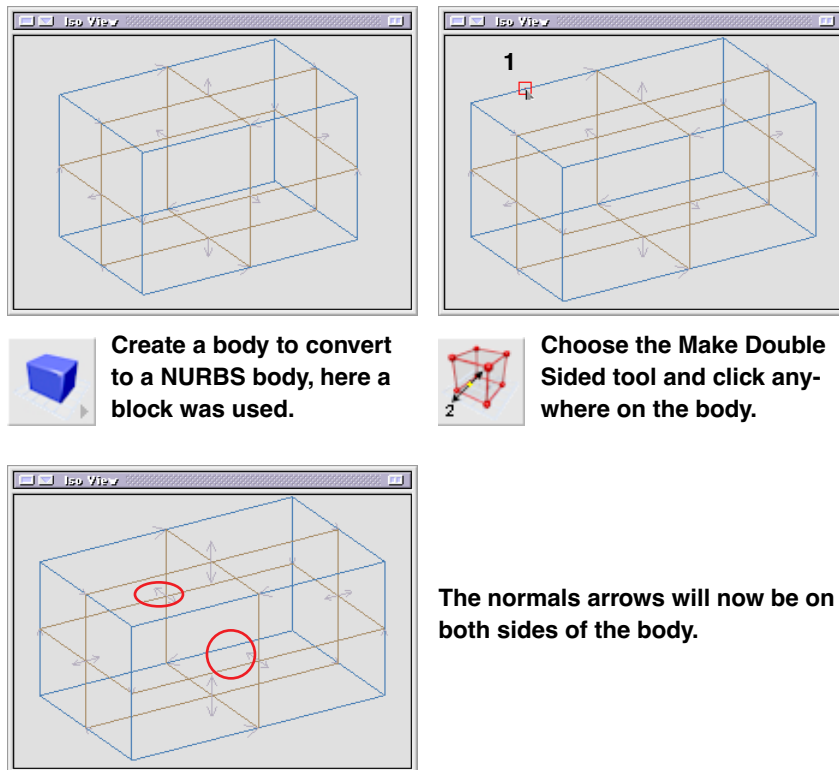


Figure 27.3 — Changing a body to a double-sided object

### 27.3 Make Single-Sided Tool

This tool will force a body into being a single-sided object.

To use, choose the tool and click anywhere on an object.

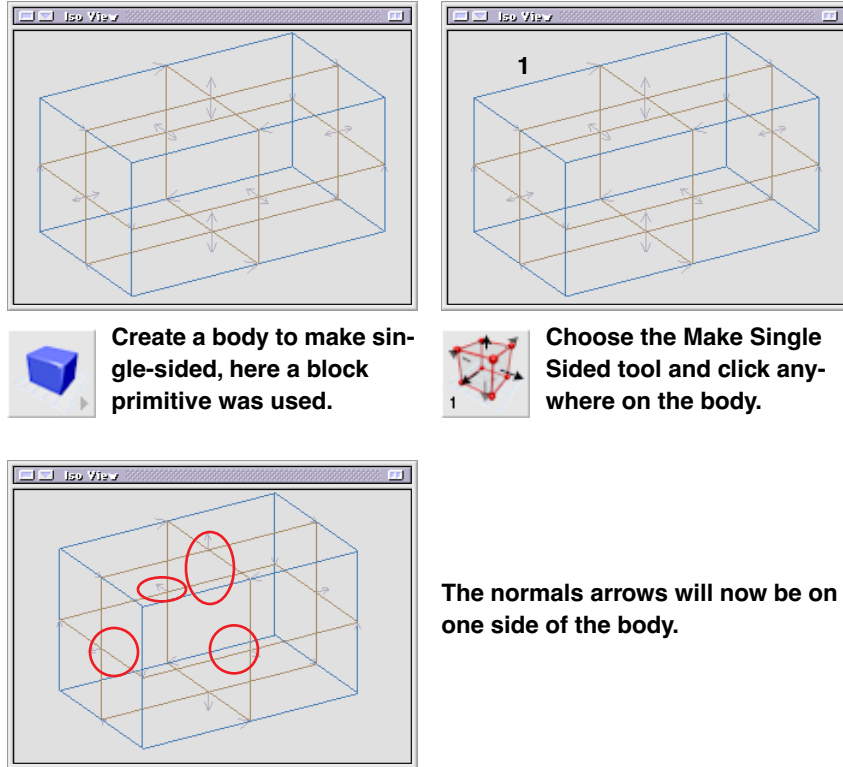


Figure 27.4 — Changing a body to a single-sided object

### 27.4 Uncover Circuits Tool

This tool will uncover all the faces of a body, but will leave the underlying wire structure.

To use, choose the tool and click anywhere on an object.

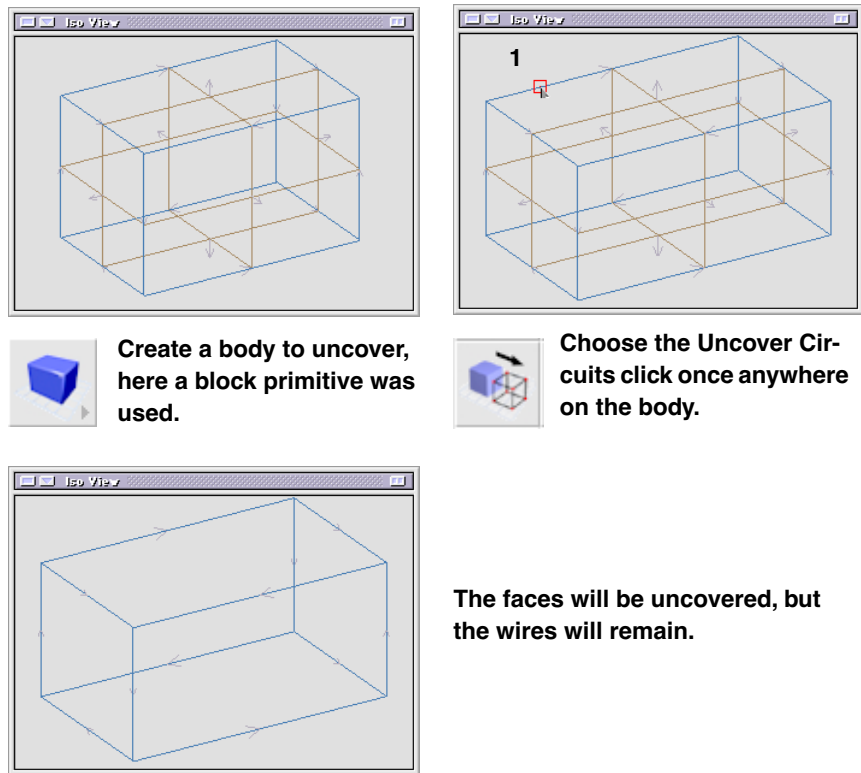


Figure 27.5 — Uncover the faces on a body

### 27.5 Contours Tool

This tool creates contours along the surface of an object. These contours can be used for skinning, as edges for use in other tools, or just to create a contour object.

The operation is simple. First select the Contour tool, then select the body you would like contours from. Next drag a vector to indicate what direction you would like the contours created in.

**Options** *Options for this tool include, the number of contours, and if you would like the contours in one object or each separate contour as a separate object.*

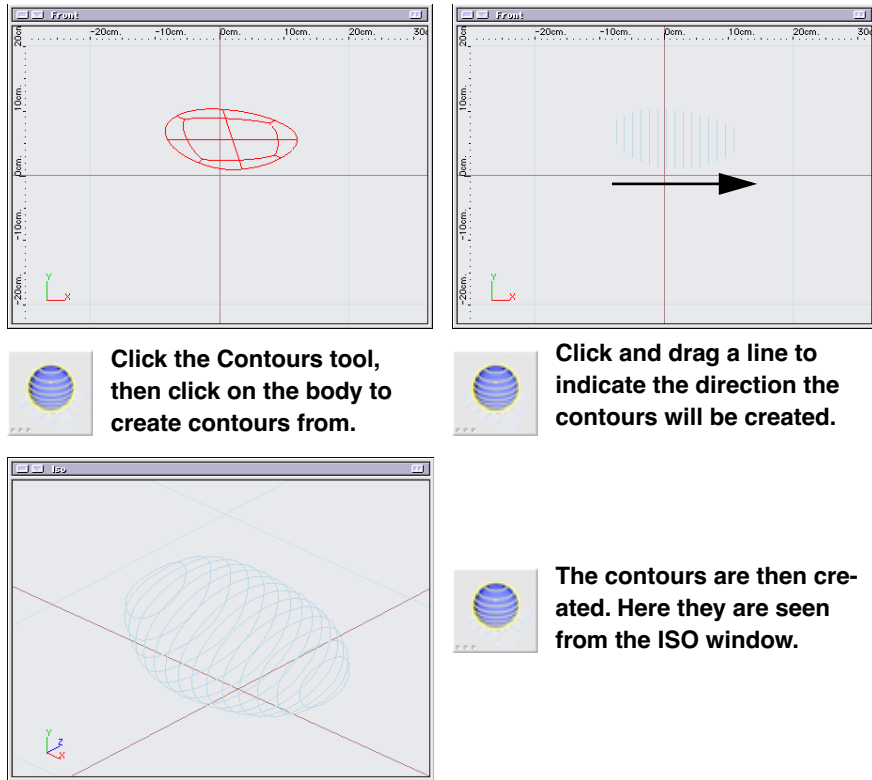
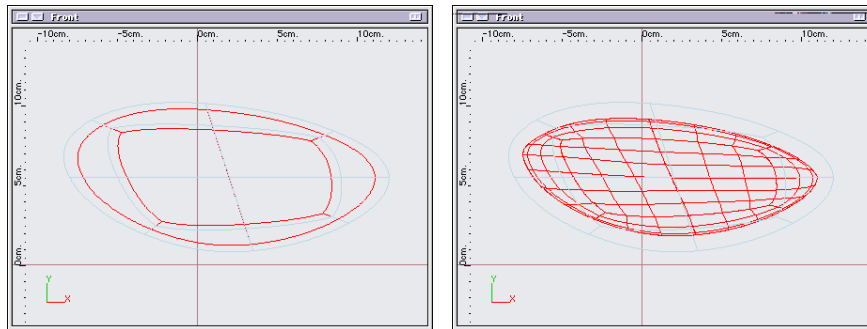


Figure 27.6 — Body contours

### 27.6 Offset Tool

Body Offsets takes an object and creates a new object whose surface is a defined distance away from the original. This process is also often called Onion Skinning or Shelling.

To use this tool, simply click the tool to highlight it, then click the body you would like to offset. The new offset body will then be created.



This example shows a scaled copy of our object. Note how the scaled version does not follow a uniform distance.

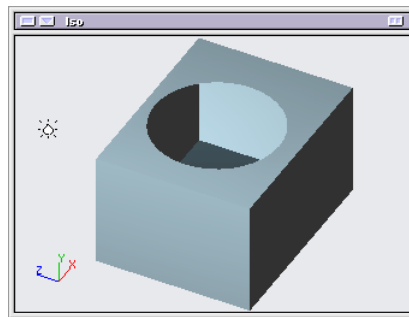
This is a negative offset of our object.

*Figure 27.7 — Offset vs. Scale*

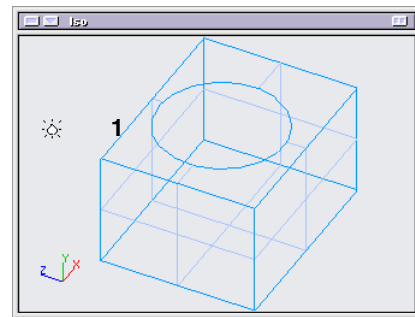
**Options** The only option for this tool is a numeric offset. It can be positive or negative.

### 27.7 Cover Circuits Tool

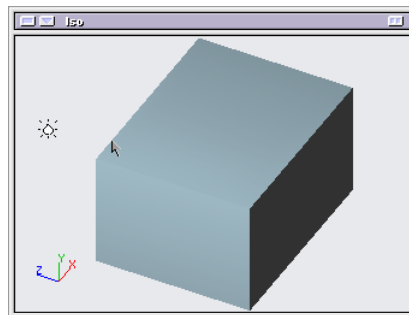
This tool covers circuits of single sided bodies. Faces previously uncovered may be filled using this tool. If two adjacent faces are “uncovered” they cannot be covered back. The tool can fill holes, not faceless wireframes.



A block with a circular hole is shown here in shaded mode.



Choose the Cover Circuits tool and click anywhere on the body (1).



Any holes on the body will be covered.

*Figure 27.8 — Cover the wires on a body*

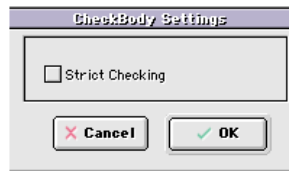
### 27.8 Check Entity Tool



*Figure 27.9 — Check Entity tool*

This tool allows users to check objects to make sure the construction of the object is proper.

To use, select the Check Entity tool and click on any body in the scene. Instant feedback on the construction of the body can be seen in the status bar on the bottom of the screen. At the same time a text document called “checkBody” will be generated and placed in the same directory as the modeling program.



*Figure 27.10 — Double-clicking on the tool brings up a dialog box. With this option selected, more thorough checking is done. When set it will analyze the geometric definitions of all the edges and faces of a body and point out any discontinuities in the underlying curves and surfaces.*

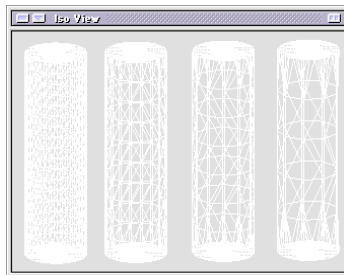


## 27.9 Regenerate Tool



*Figure 27.11 — Regenerate tool*

The tessellation of an object can be altered at any time in the model building process. Some objects that may need more or less detail can be changed by re-inputting new values in the **Preferences/Tessellation dialog box** and then choosing the REGEN tool and clicking on the object. The objects will be recomputed using the new values.



The cylinders (left) were all created with the same tessellation setting, then altered using new tessellation values and then selected with the Regeneration tool. The values, left to right, are 10, 20, 30 and 40.

*Figure 27.12 — Setting tessellation and using the Regeneration tool.*

By viewing an object in Mesh Wireframe mode, you get a feel for how the object will be faceted when the model is exported. By altering the Normal Tolerance you can adjust the mesh density of the object. The higher the number, the more course, and less intense the mess will be. The lower the number, mesh becomes finer and more intricate.

Using the Regenerate tool it is possible to have a scene with a large variety in the tessellation settings of many object. Users may choose, for instance, to have objects that will be near the camera, or that will be large, to have a

lower tolerance setting resulting in finer models, and smaller and more obscure objects can have higher, courser shapes. This manipulation of tessellation settings will result in faster rendering times and less bulky scenes for the final product, but will not affect the topology of an object.

For more on tessellation settings, see page 65.

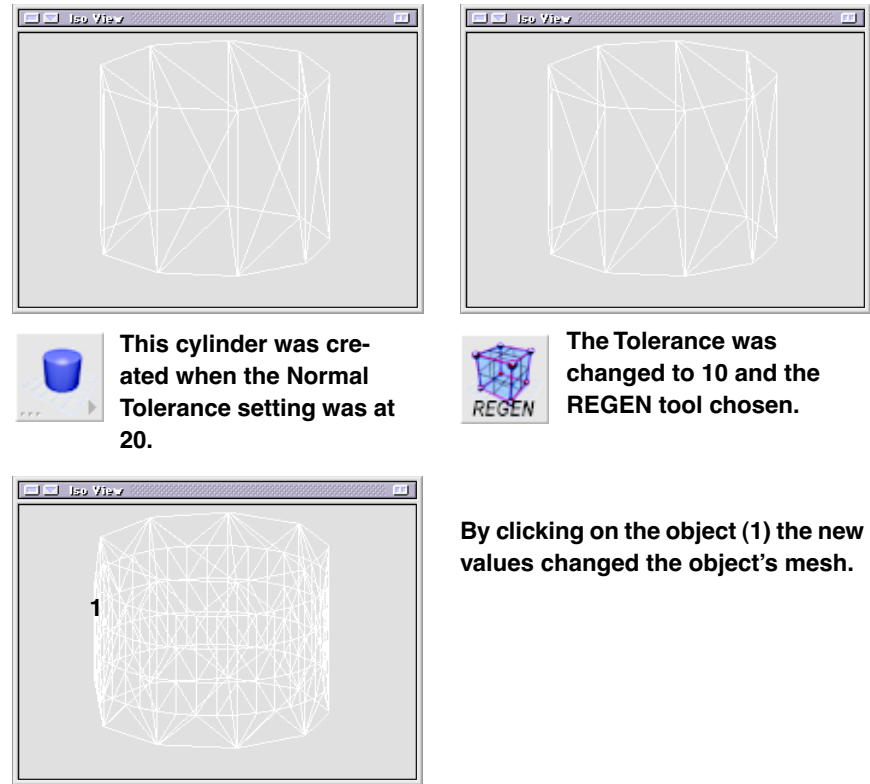


Figure 27.13 — Changing the tessellation settings of an individual object.

# Workplanes Palette

## 28.0 Introduction

A workplane is a reference grid used by Universe Modeler to determine the orientation of the world. Universe Modeler creates the “world” workplane for the top and iso views automatically (they share the same plane, by default), as well as the XY (front) and YZ (side).

**Note** *When the World plane is set, the work planes for the front and side views are automatically (and transparently) activated, so that the behavior that you would expect in those views is consistent.*

For the most part, you may find that you do not need to set up a workplane of your own. However, you will likely run across an occasional situation where creating a custom workplane is necessary.

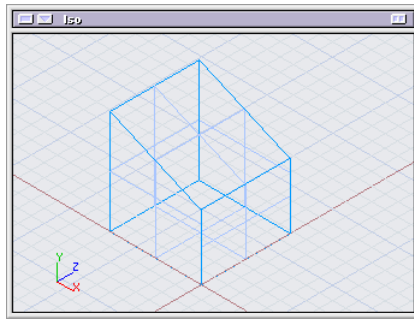
The overused, but accurate, example of a roof on a house comes to mind. In a typical A frame roof, the planes of the roof will be different than the construction of the rest of the house. You may wish to add features to the roof, using the plane of the roof as a construction plane. We call these “user defined workplanes.”

The tool used to create user defined workplanes can be found in the Workplanes Palette and managing workplanes is done in the Workplanes List Window.

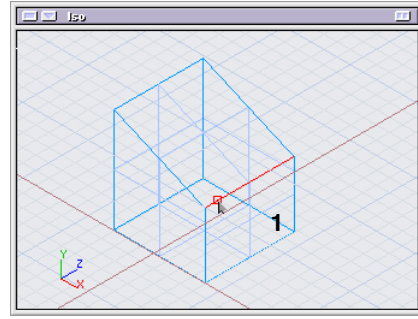
## 28.1 Adding a Workplane to a Project

There are two ways to add a workplane to the project. One is to click on the Add Workplane tool icon and then click on the edges of an object in the scene to define the horizontal and vertical directions of the workplane or to

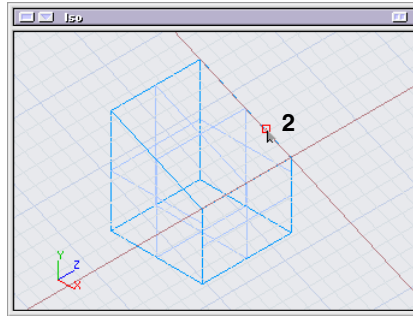
click on the **“Create”** button in the Workplane List Window and follow the same steps as if you clicked on the tool icon.



**Choose the Add Workplane tool.**



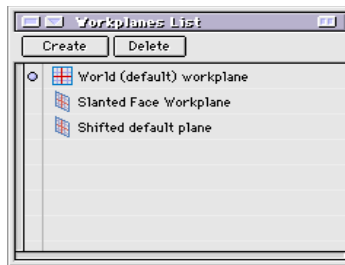
**Click on an edge that will define the horizontal direction of the new workplane.**



**Then click on the edge to define the vertical direction, the new grid will appear.**

**Figure 28.1 — Adding a workplane to the project.**

The new workplanes will be represented with a slanted icon in the Workplanes List, and the currently active workplane will have the round bullet in front of it.



*Figure 28.1* — The workplanes are managed in the Workplanes List Window.

## 28.2 Managing Workplanes

Workplanes are managed in the Workplane List window. You can activate the window by choosing **Windows>Show Workplanes Window** in the main menu bar. Entries in list correspond to User-defined Coordinate Systems or UCS for short. Any drawing window can display a **UCS View** by clicking on the triangle in the window bar and pulling down to **UCS View**.

The active workplane will have a round bullet icon to the left of its name. There can be only one active workplane at any time.

Workplanes can be renamed by selecting the workplane in the Workplane List and hitting the Enter or Return key and typing in the new name, then hitting Enter or Return again to save the name.

Workplanes that are no longer needed can be deleted by selecting them in the Workplane List, then clicking on the "Delete" button.

### 28.3 Aligning a Window to the Workplane

You can align any window to the active workplane by holding down your Control-key and clicking on a window, drag the pop up window to the Orientation menu and choose “Workplane.”

**Note** *When a new view is created it is automatically aligned to the active workplane. You can use the View Manager to save windows with specific views.*

# Element Info Window

## 29.0 Introduction

The Element Information Window provides you with data concerning a selected object. Information regarding the layer in which the object resides, its position in space, and any specific values such as color, tessellation, and so on are provided. Element Information Windows are offered for:

- Objects
- Lights
- Background Images

## 29.1 Activating the Element Information Window

You can activate the Element Information window in the following ways:

- Selecting the object and choosing **Windows>Element Info Window** from the main menu bar
- Selecting the object and pressing command (Mac) or control (PC and Sun) “i”.
- Double-clicking on the object in the Layer view

## 29.2 Objects

The Object Element Info Window is presented in Figure 29.0. You can use the window to manage the following data for objects (bodies):

- Object Name
- Object Layer
- Object Position
- Object Color
- Object Tessellation

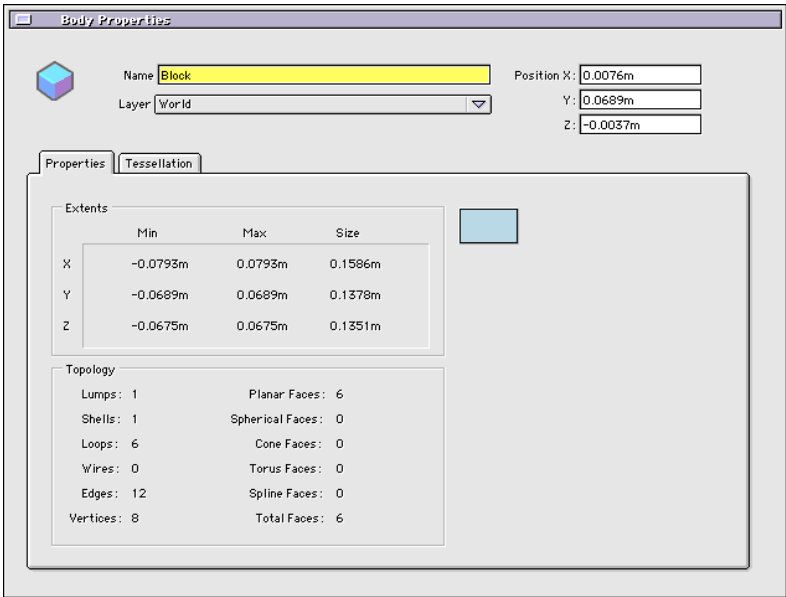


Figure 29.0 — The Object Element Information Window

### 29.3 Object Name

The object name is contained within the name field of the window. To change the name of the object, double click in the field (to select the entire name) and type in the new name.

### 29.4 Object Layer

You can see the current layer in which the object is residing by viewing the Layer popup menu. If you wish to change the layer of the object, click on the menu and select a new layer from the list.



**Note** Only pre-existing layers will be listed in the menu. If you wish to add a new layer, you will need to do so in the Layer Window.

### 29.5 Object Position

The position of the object is stored in three edit boxes (X, Y and Z) to the right of the name field. From the Object Element Information Window, you can change the position of the object by typing in new values into the fields.

### 29.6 Object Color

The color of an object can be set by clicking on the color swatch in the Element Info Window. The color swatch is located in the Properties tab. Click on the color swatch to activate the color picker. See “Color Picker” on page 90.

### 29.7 Object Tessellation

The Tessellation tab allows you to set custom tessellation settings per object. You can set a custom tessellation setting for export or for display purposes (or both.) By default all objects get their tessellation settings from the **Edit > Document Preferences > Tessellation** menu. To set tessellation individually, uncheck the “Use Project Setting” box. Once you’ve unchecked this box you will see the Adaptive Tessellation settings become active. These settings are exactly the same as they appear in the **Document Preferences > Tessellation** menu. (See “Tessellation” on page 65. for a discussion of the settings.)

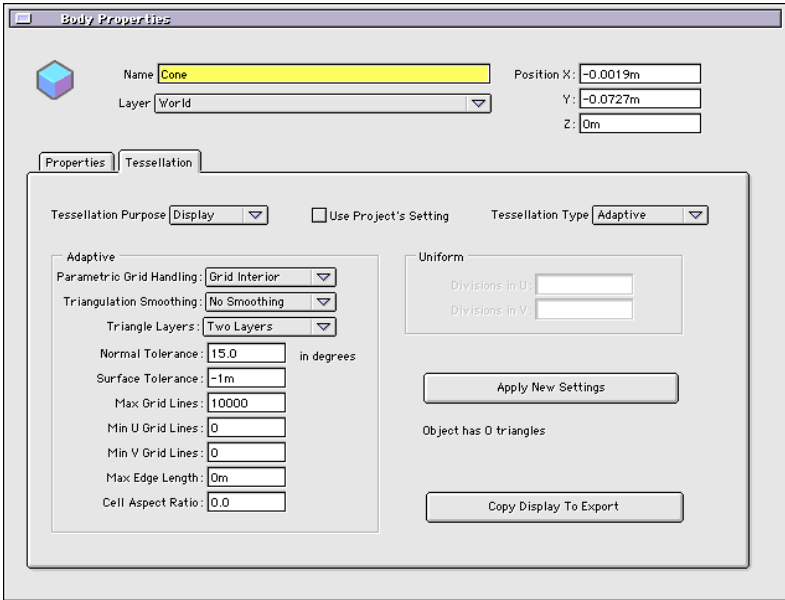


Figure 29.1 — Tessellation window

**Tessellation Purpose.** You can set a custom tessellation setting for display purposes (wireframe mesh, flat shaded, or smooth shaded) or for Export (or both). Selecting your choice from the Tessellation Purpose drop down menu will make one or the other active.

You can copy display and export settings to one another as needed using the copy button on the lower right of the palette.

As a general workflow rule it's often easier to begin setting a custom tessellation setting by choosing the "Display" option and changing the display mode in one of your 3D views to "wireframe". This will allow you to see the results of your changes interactively as you make them. Once you have

a mesh that suits your needs you can use the “Copy Display to Export” button to copy these settings to the export settings.

Often setting a lower display tessellation for large objects will speed up shaded displays in modeler.

**Adaptive or Uniform.** You can choose whether to adaptively tessellate your object or to use Uniform Tessellation. Uniform Tessellation will only be available with UberNURBS objects, for ACIS geometry Adaptive is the only option. (You can use Adaptive on UberNURBS as well if you like.)

### Applying Tessellation Settings

Once you've adjusted the tessellation settings to your liking you can press the Apply New Settings button to see your object updated with the new settings. Be patient. If your adjustments require the creation of more triangles it may take longer to generate the new display each time you make a change.

Under the Apply New Settings Button is a triangle count that will be updated each time you make a change. This count only becomes active when you uncheck the “Use Project Setting” box.

### Reverting to Project Settings

You can revert the object level tessellation settings back to the project tessellation settings at any time by checking the Use Projects Settings box back on.

### Why use Object Level Tessellation?

The main reason to use object level tessellation is to speed shaded redraws (display tessellation) on large objects in a scene, or to reduce triangles on certain objects that are out of scale with the project level tessellation settings.

Imagine you were modeling an old fashioned steam engine train. Most parts of the train would be roughly within the same scale. The engine, the smoke stack, the cab walls, etc. However you might also have modeled some very small bolts to give it more realistic detail. These bolts are much, much smaller than the engine is.

This creates a dilemma. If you set up your project tessellation settings for the engines scale you get a nice, efficient mesh, but your bolts show up as jagged little polygons because they're too small to be properly tessellated.

If you set up your project tessellation settings for the bolts, then your large engine will have too many polygons in it.

The answer is object level tessellation. You can set your project up for the lowest common denominator, which in this case might be the scale of the engine. Then you can select the bolts and use the object level tessellation settings to give them a little more detail.

## 29.8 Lights

The Light Element Information Window is used to manage the following light-specific data:

- Light Name
- Light Layer
- Light Position

- Light Color
- Light Intensity

### 29.9 Background Images

Background images are used to aid in the construction of models. For example, you may wish to load blueprints into the background of a window so that you can accurately build your model. Background images can be any size or color depth, but they must be in the Electric Image, JPEG, TIFF, or PNG formats. Solaris only supports the Electric Image format.

The Background Image Element Information Window is used to manage the following background image-specific data:

- Image Name
- Image Layer
- Image Position
- Image Polygon Size
- Image Polygon Rotation

#### Image Polygon Size

Background images are texture mapped onto polygons. The size of the polygon is presented in the Image Polygon Size edit boxes. There are two values, Width and Height. Note that you can move, rotate and scale the image polygon as you would any other object type in Universe Modeler.

#### Image Polygon Rotation

Background images can be rotated in any direction. The Element Information window offers three edit boxes, X, Y and Z in degrees for the purposes

of editing the rotation of an image polygon. You can also use the Object rotation tool to change the rotation of the image polygon.

# View Manager

## 30.0 Introduction

The View manager allows you to create and save a variety of window views and orientations. By saving your favorite views of your model, you can easily go back to them without having to recreate them from scratch.

**Note** *Views can be saved and recalled by any Universe Modeler project.*

For instance, you might have a camera setting that reflects a wide-angle view that encompasses your whole scene, and you might have another view that is a telephoto angle of a small portion of your model. The view manager will let you hop between these two views by saving their settings to a view list and letting you apply them as you need them.

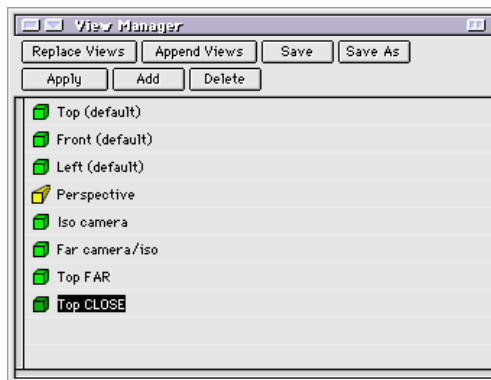
An example of how the View Manager can help you work would be the construction of a complex model, like a car. You might have one Top view that includes the entire car, and a series of other Top views that focus on various parts, such as the engine, wheels and steering. By using the View Manager you can save these window settings to keep you from having to pan to a new position every time you work on a different area.

## 30.1 Activating the View Manger

The View Manager window can be accessed in the following two ways:

- By clicking on the small triangle in any drawing window bar, and selecting View Manager from the menu.
- By choosing **Windows>Show View Manager Window** from the main menu bar.

When you activate the View Manager, you will see the following window, as shown in Figure 30.1. If you do not have any views stored, the list will not contain any view items. The view manager consists of two parts, the function buttons at the top, and the view list in the window below.



*Figure 30.1 — The View Manager*

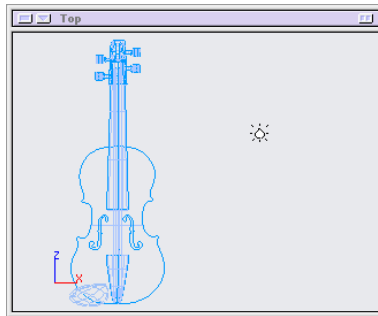
The first step in using the View Manager is to create a view in any 3D window, the top, left, right or camera for instance, and then open up the View Manager. Click on the ADD button and then click on the view you wish to save. The name of the view will appear on the view list. This view can now be either saved for future reference, or applied to any 3D window.

You can change the name of the default setting by clicking on the name in the View Manager and hitting the Return or Enter key, typing the new name and then hitting the Return or Enter key to lock in the name.

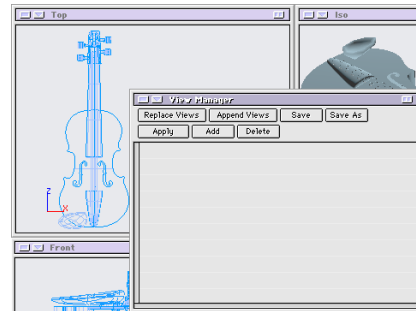
To apply the view settings you added to the list to any window, click on the name of the view you created in the view list, click on the Apply button and click on the window that the view is to be applied to.



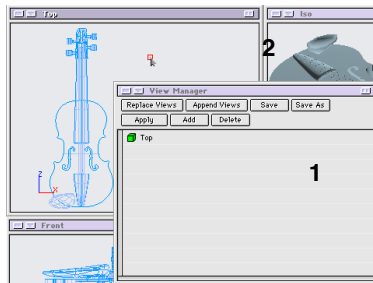
View settings that can save, pretty much include anything you can do to a 3D window, including whether a view is parallel or perspective as well as all the variables in the Camera Parameters settings.



**Set up a window with the view settings you want to save.**



**Create a new window and click and drag on the small triangle to change it to the View Manager.**



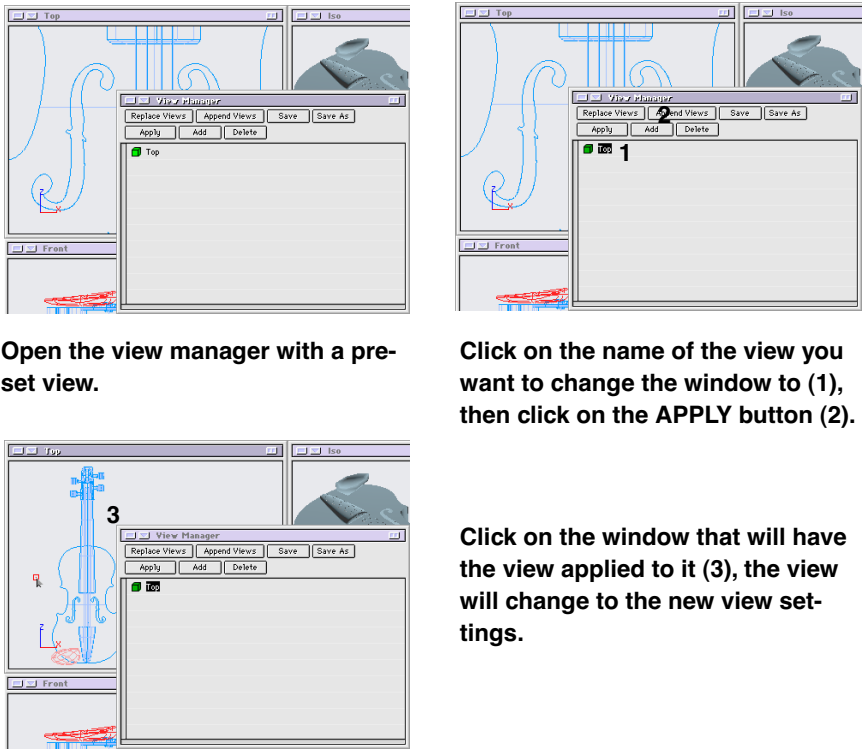
**Click on the ADD function button (1) and then on the view you wish to save (2). The name of that view will appear in the view list.**

**Figure 30.2 — Adding a view to the view manager list**

You can save your view settings so you can access them as a disk file at any time. You can also add old view settings to your existing list. The function buttons at the top of the View Manager control the items in the list.

## 30.2 Replace Views

Removes all existing views in the list and replaces them with a view list that has already been created and saved to your disk. Clicking on the button will bring up a **FIND FILE dialog box** and only saved view lists can be seen.



Open the view manager with a preset view.

Click on the name of the view you want to change the window to (1), then click on the **APPLY** button (2).

Click on the window that will have the view applied to it (3), the view will change to the new view settings.

Figure 30.3 — Applying a view from the view manager to a window.

### 30.3 Append Views

You can add views you've created and saved before to your existing list. Clicking on this will bring up a FIND FILE dialog box and the selected view list will be added to your own.

**Save.** After creating a view list you want to save. Clicking on this button will bring up a standard SAVE dialog box. Files saved can be opened later using the Replace Views or Append Views buttons.

**Save As.** If you want to save your current file under a different name, use the SAVE AS button.

**Apply.** After you've added views to your list, they can be applied to any 3D window by clicking on the name of the view in the list, then the Apply button, and then the window that you want the view applied to.

**Add.** To add a view to the list, create a custom view in one of your windows, then click on the Add button and the view that has the settings you wish to add to the view list. The name of the view will pop up in the view list.

**Delete.** Any view in the list can be removed by selecting the view, and clicking on the Delete key.

Throughout the process the status bar will give you directions on what to do.



## Layer View Window

### 31.0 Introduction

Once you begin to work with Universe Modeler, you will find yourself creating a tremendous amount of data very quickly. Suddenly, selecting objects become more complicated, as you have to wade through all of those edges and isoparms. You begin to think to yourself, “there has to be a better way.” And there is. It’s called the Layer View.

The layer view allows you to place objects into layers, and organize those layers within your project. You can create as many layers as you like, and you can nest layers within other layers. You can use layers to lock down whole groups of objects or just one at a time. You can use layers to organize your data, perhaps background images are put into a separate layer, the same for curves and surfaces and so on. The point is, how you choose to use layers is pretty much up to you.

### 31.1 Activating the Layer View Window

When the program is started, a standard four-window setup and the toolbox come to the screen. To get to the Layer View you have three options:

1. Click on **World View** toggle of an existing window and drag down to Project View to access the layering system, or
2. Go to the **Windows menu** and drag down to New Window, creating a new window. By default, this window is created in the Top View setting. Click on the World View toggle and drag down to Project View to access the layering system.
3. Using the Command-L or go to the **Windows menu** and pull down to “**Show Layer Window.**” Either way will bring up a new layer view.

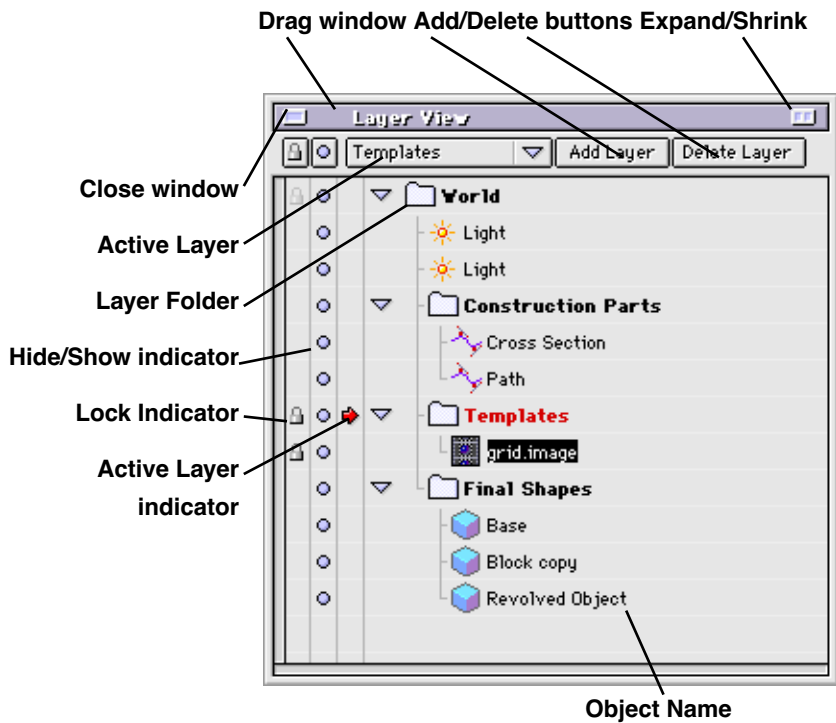


Figure 31.1 — Layer View

**Important** *The objects in the layer view are hierarchical, allowing FACT exports to have the layer's hierarchy.*

In the Layer View, you see all the elements used in the scene. If you haven't introduced any elements yet, you will see the default layer with just the lights. A quick run-through of the elements in the Layer View include:

### Close Window

If you want to close the Project View window, click on the rectangle in the top left. Closing the window will not remove any of the layers created or any of the changes made to the Project View. The next time you open up the Project View, all your settings will be there.

### Layer Folder

Layers are handled by folders and work very similar to the desktop metaphor. Arrows next to the folders show what is in the folder or close the folder to save space.

### Active Layer Indicator

The red arrow to the left of a folder indicates that it is the currently active layer. Any new items created are placed in this folder. Clicking on the column in front of any folder causes that folder to become the active layer.

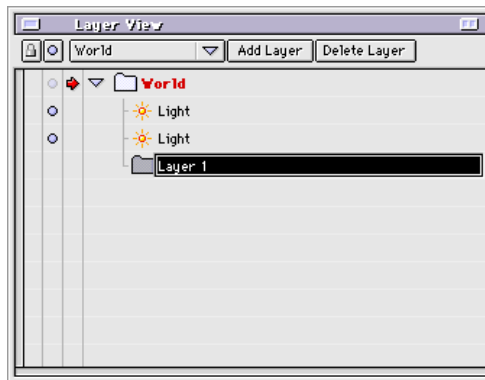
### Object Name and Icon

Each time an object is created, it is placed in the currently active layer. Objects can be moved around by clicking and dragging on them. Objects can be renamed by hitting the return or enter key, typing the new name, and then hitting the return or enter key again.

### Hide/Show Indicator

When a new object is created, it is visible by default. If you want to hide an object or all the objects in a folder, click on the Hide/Show indicator dot in front of it. By clicking and dragging, a whole series of objects and folders can be hidden quickly. Hiding an object does not remove it. Hidden objects

can be made visible again by clicking on the objects. If there is a hidden object in a layer, the layer folder will indicate this by ghosting slightly.



**Figure 31.2 — Changing the name of an object is done by clicking on the object, hitting the return or enter key, typing the new name and hitting the return or enter key again.**

### Lock/Unlock Indicator

Any object or layer folder can be locked by clicking on the “Locking” column in front of the object or folder. A locked object cannot be changed in position, scale or orientation. In addition, a locked object cannot be selected.

## 31.2 Setting up the Layer View

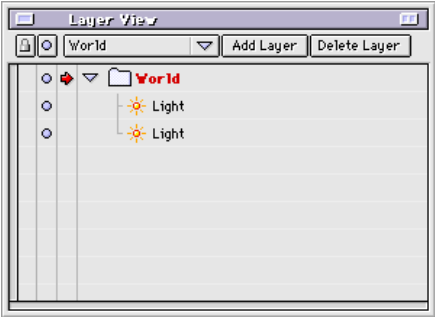
Most people will want to establish a layering system that helps organize their project as it grows. Key to this is creating a customized set of layer folders.



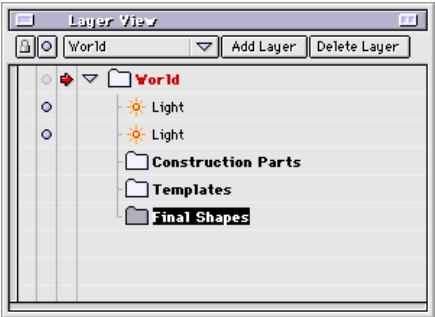
The first step is to bring up the Layer View, the easiest way to do that is to do a Command-L. A new window will appear that is labeled “Layer View.” Like the 3D windows, this new window can be repositioned and rescaled as needed. A two-screen setup is worth its weight in gold here, instead of having the Layer View take up valuable screen real estate, if you have a two-screen setup you can toss your Layer View window onto your second screen and scale it to fit.

When you create a new document and go to the Layer View, the “World” will be the only existing folder. Everything you create will go into this layer folder.

Before you start modeling you might want to get a jump on your organization, creating layers for various elements in your scene. A good outline might include a folder for your construction parts which would contain the parts and pieces used to create your shapes like cross sections and paths. You would also want a layer for your final shapes. If you’re planning on bringing in background images to use as templates, you might create a Template layer.



When the Layer View is first brought up, the default setup will be seen. Without any alteration to the default layer setup, all objects will be placed in the World layer.



When building models it is important to keep your object organized, this will save you time as the scene gets more complex. Key to organization is creating layers to “store” your various elements. With the World still being your default layer, click on the Add Layer button. Change the names of your layers to reflect their purpose in life.

*Figure 31.3 — The layer view window*

Keep in mind that once a layer is created, all the elements in that layer can be quickly locked or hidden. If you have a layer of parts, for instance, you can hide all the parts at once by clicking on the “Hide” column in front of

layer. With this in mind you can organize your layer based on elements you might want to hide, lock or delete quickly in the future. Background images, for instance, behave like other objects in a scene, you can click on them, move them, rotate them and manhandle them as needed, but this can also be a problem if you want to select an object, but the background keeps getting in the way. By putting all your backgrounds in a layer, you can quickly lock the entire group.

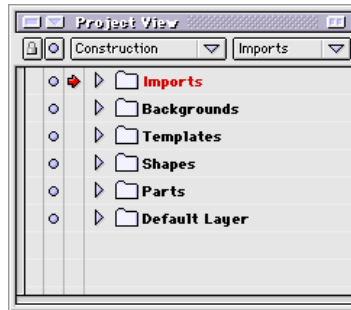
Once you've created your own layering system, you can put objects into it one of two ways. You can click on the third column in front of the layer you want to make as your active layer, and the red arrow will hop down in front of it (you can also click on the “Active **Layer popup menu** and drag to the file you want) and with the layer selected as active, any newly created shape will be placed into that layer. You can also select any existing object that is in any other layer and drag it into the layer you want.

Layers can be nested into other layers. They can either be created by clicking on the Add Layer button while a layer you've created other than the World layer is the currently active layer, or you can drag an entire layer folder from one layer to another.

### 31.3 Creating Layers

To the right of the Active Layer pop-up menu is the **Add Layer button**. Clicking on this will create a new layer in the currently active layer. Creating a new layer does NOT make it the currently active layer, so if you're creating a layer and plan to create objects to put into it, make sure that you designate it as the currently active layer before returning to the 3D workspace.

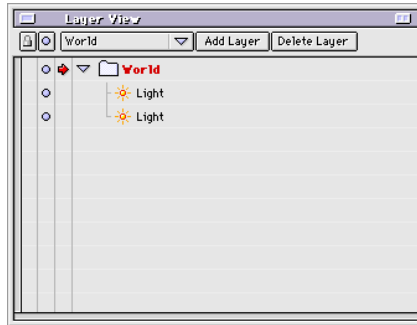
By default, all new layers will be placed into the World Layer until another layer is designated as the currently active layer.



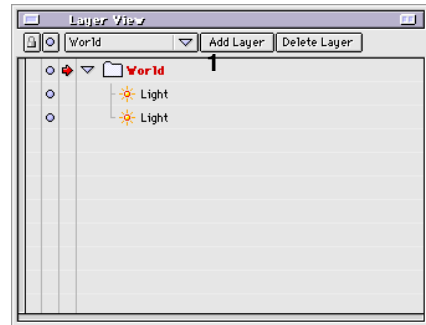
**Figure 31.4** — Once a layering system has been made, layers can be added to it. In this example, the layer system is called construction. Layers have been created for imports, backgrounds, templates, shapes and parts.

### 31.4 Deleting Layers

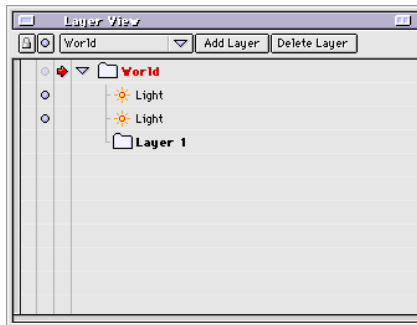
To remove a layer, click on it to highlight it, the layer does NOT have to be the currently active layer, it just needs to be highlighted. Click on the Delete Layer button and the layer will go away. The contents of the layer will not be removed, they will be placed in the same layer that the deleted layer resided in. To delete objects in the scene using the layer view, click on the objects to highlight them and then hit the Delete key (*Backspace for PC*) on the keyboard. Only one layer can be deleted at a time, nested layers inside a deleted layer will be untouched.



**Bring up the Layer Window by doing a Command-L.**



**Click on the Add Layer button (1).**



**A new layer will be created in the currently active layer.**

*Figure 31.5 — Adding a new layer.*

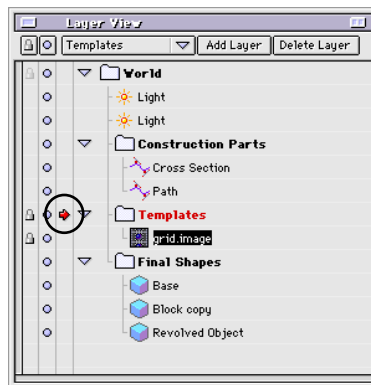
### 31.5 Working With Layers

Now it's time to put the layers to work. Any new object created in the World View is going to be placed in the currently active layer. Make a layer active by clicking in the Active Layer column where the red arrow resides. The red arrow indicates the currently active layer. Clicking in the column makes the arrow hop to the selected layer folder. Clicking on the folder

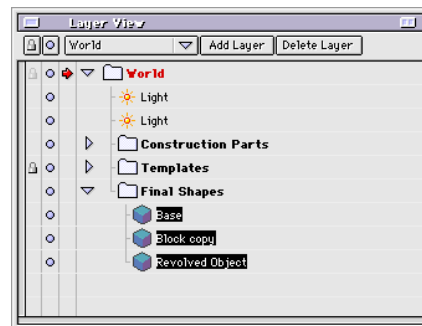
itself will not make the layer active, you must click the column directly to the left of the folder.

Objects can be moved from one layer to another layer by clicking on the object's icon and dragging it to a new layer. If you start building a model without thinking about layers and your scene gets complex, you can get organized retroactively by establishing a Layering System. Simply create layers and drag all your objects into their proper layer folders.

Several objects can be selected at one time by holding down the shift key to make multiple selections. To deselect an object, hold down the shift key and click individually.



**Any object introduced into the World View is placed in the currently active layer. The active layer is indicated by a red arrow in the column next to the layer folder.**



**Several objects can be selected at the same time by holding down the shift key and making multiple selections.**

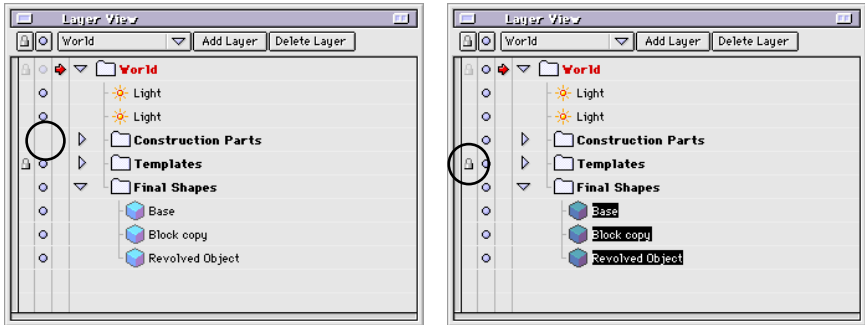
*Figure 31.6 — Selecting objects in the layer view window.*

### 31.6 Hiding Objects

A common construction approach is to create a layer to hold all the parts and pieces used during object creation. Doing this allows you to hide all the wire bodies used to construct shapes in case you need to reference them again. An entire folder can be hidden by clicking on the Hide column, directly to the left of the layer. Hidden elements maintain their position and scale in the world, but will be hidden until needed later. Objects also can be hidden individually, regardless of the layer they reside.

Visible objects dropped into a Hidden layer are still visible. You have to either hide each object individually or toggle the whole layer off. A layer with a mixture of visible and hidden objects has a ghosted visibility button, but will hide everything when clicked on once.

Altering the visibility of an object can be a great aid in construction. For instance, if you are creating the ribs of a skinned object it is advised that you copy the first rib so it can act as the last rib as well. By doing this, you can hide the ribs alternately. To begin, copy the first rib and hide the copy. Choose the skin tool and click on the original rib. Work your way around to the other ribs, until you are near the end. Then hide the original rib, reveal the copied (last) rib and click on it. The skin will be formed that goes back to the original curve.



To hide an object, click on the Hide Indicator button. In the example (above), the “Construction Parts” layer was hidden (circled). Both individual elements and entire layers can be hidden.

To lock an object, click on the Lock column. In the example (above), the “Templates” layer was locked (circled). Both individual elements and entire layers can be locked

Figure 31.7 — Hiding objects using the layer view window.

There also may be times when you have a complex scene and you just do not need everything on the screen. Hiding complex objects allows you to navigate faster and reduce screen redraw time.

## 31.7 Locking Objects

You may have an object that is in the way or one that you just do not want moved. By clicking on the Locking column to the left of the layers window, you can lock an object. A locked object cannot be selected or altered.

If you need an object visible for positioning other objects around it, but you do not plan to alter any time soon, lock it in place. This aids in selecting other objects around the object without also selecting the object itself.



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