

Rhinoceros 5

User's Guide



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Section 1: Introduction



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Chapter 1: Working in 3-D

Working in 3-D on a computer requires that you visualize three-dimensional objects drawn on a two dimensional medium—the computer screen. Rhino provides tools to help you do this.

You can manipulate the views and look at your model from different angles easily by dragging with your right mouse button. You can do this in both a wireframe view and a shaded view.

Why NURBS modeling

NURBS (non-uniform rational B-splines) are mathematical representations that can accurately model any shape from a simple 2-D line, circle, arc, or box to the most complex 3-D free-form organic surface or solid. Because of their flexibility and accuracy, NURBS models can be used in any process from illustration and animation to manufacturing.

NURBS geometry is an industry standard for designers who work in 3-D where forms are free and flowing; where both form and function is important. Rhino is used in marine, aerospace, and automobile interior and exterior design. Makers of household and office appliances, furniture, medical and sports equipment, footwear, and jewelry use Rhino to create free-form shapes.

NURBS modeling is also widely used by professional animators and graphic artists. The advantage over using polygon modelers is that there are no facets. The models can be rendered at any resolution. A mesh can be created from the model at any resolution. For more information about the mathematics of NURBS, see the Rhino Help topic.

The Rhino interface

In the exercises, you will use Rhino's commands, navigation tools, shaded modes, render, and use some basic object manipulation.

Open the tutorial model

1. Review the Rhino [Help](#) topic [The Rhino Window](#).
2. From the Rhino **File** menu, click **Open**.
3. Browse to the **Tutorial Models** folder that you downloaded with the *User's Guide*.
4. Select the file **Start.3dm** and click the **Open** button.

This model contains four objects: a box, a cone, a cylinder, and a sphere.

Menus, buttons, and command prompt

With Rhino, you can work with the menus, the toolbar buttons, or by typing commands at the command prompt. This gives you the flexibility to work in a way that is comfortable for you.

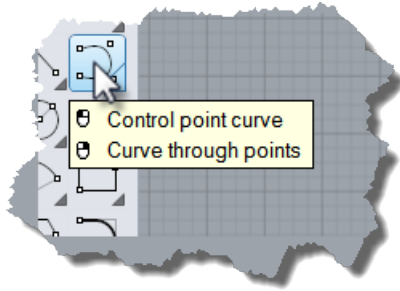
Rhino's toolbars also provide a way to create your own custom environment. Drag toolbars to float them anywhere on the screen, change their shape, or dock them at the edge of the Rhino screen.

Note: Hold the Ctrl key while moving a toolbar to prevent docking.

Many toolbar buttons have a second command that you can access by right-clicking the button.

To display a button tooltip

- ▶ Hover your mouse over the button.
The command names that are activated with the left and right mouse clicks display.



Change the view

Before starting the next command, use Rhino's mouse navigation to change the view.

Set up the view

1. Click in the **Top** viewport, zoom out a little (scroll the mouse wheel).
2. **Pan** (click the right mouse button and drag) to give yourself some room).

Start a command from the menu

Most Rhino commands are arranged in the menus.



Start the Cone command

1. Review the Rhino [Help](#) topic for the **Cone** command.
2. From the **Solid** menu, click **Cone**.

Draw the cone

1. At the **Base of cone...** prompt, in the **Top** viewport, click with the mouse to pick the center point for the base of the cone.
2. At the **Radius...** prompt, in the **Top** viewport, drag the mouse and click to draw the cone's base.
3. At the **End of cone** prompt, in the **Front** viewport, drag the mouse and click to draw the cone's point.

Start a command from a toolbar button

Toolbars provide a graphical interface to the commands.



Start the Box command

1. Review the Rhino [Help](#) topic for the **Box** command.
2. From the **Main1** toolbar, click **Box**, and then click **Corner to Corner, Height**.
Note: The **Main1** toolbar is normally docked on the left side of the Rhino window.

Draw the box

1. At the **First corner of base...** prompt, in the **Perspective** viewport, click the mouse to pick a corner point for the box.
2. At the **Other corner of base...** prompt, click the mouse to pick the opposite corner for the box.
3. At the **Height...** prompt, in the **Front** viewport, click the mouse to pick the height for the box.

Admire your work

1. Click in the **Perspective** view.
2. On the **View** menu, click **Shaded**.

3. Drag with the right mouse button to rotate the **Perspective** view.

Start from the command line

You can start a command by typing the name at the command prompt.

Start the Sphere command by typing

1. Review the Rhino [Help](#) topic for the [Sphere](#) command.
2. At the command prompt, type **Sphere**.
When you type the first letters of a command, a list of possible commands appears.
3. Select the **Sphere** command from the list.

Draw the sphere

1. At the **Center of sphere...** prompt, in the **Perspective** viewport, click the mouse to pick the center point for the sphere.
2. At the **Radius...** prompt, in the **Perspective** viewport, move the mouse to draw the sphere and click.

Start modeling

Rhino is a "command driven" program. Actions are activated by commands. Commands are accessed by typing at the command prompt, through the *menus*, or the toolbars.

You have already used the menus to start commands. Before we go further, explore using the typed command names and the toolbar buttons. You may find one method easier than another. The choice is yours and there is no preference for one method over another.

To start your first Rhino model

1. Start **Rhino**.
2. From the **File** menu, click [New](#).
3. In the **Open Template File** dialog box, select **SmallObjects - centimeters.3dm** and click **Open**.

In this session you will

- Start a command by typing at the command line.
- Start a command by choosing from the menu.
- Start a command by choosing from a toolbar button.

Undo a mistake

If you did something you did not want to do, you can undo your actions.

Undo a command

- ▶ From the **Edit** menu, click [Undo](#).
The prompt displays the action you are undoing.
You can undo a series of commands.
You can also Redo the Undo if you go too far.

Redo commands

- ▶ From the **Edit** menu, click [Redo](#).
- ▶ **Note:** To cancel a command any time, press the **Esc** key.

Command history window

The command history window contains the command line and the command history. The window is normally docked at the top of the Rhino window, but you can dock it at the bottom of the screen or float it just like any toolbar.

The *command prompt* appears on the command line and is where you can type command names, set options, enter distances and angles for drawing, and read the prompts for the commands.

The command prompt displays options for the command in parentheses. To activate an option, click the option with the mouse or type the option name or accelerator letter (the underlined letter in the option).

Options give you alternate methods for using the command. Prompts are messages to you to select objects to act on, to enter information, or to enter a point on the screen. As you start to build objects, prompts become increasingly important, since they give you valuable feedback about what to do next.

Because of the autocomplete feature and command list display, many people find that once they learn the program, they can type the command names faster than they can find them on the menus or toolbars.

Command-line options

Command options change how a command acts. For example, when you draw a circle, the circle is normally drawn on the active construction plane. The **Circle** command has several options including **Vertical** and **AroundCurve**. Command options appear in parentheses at the prompt.

To use a command option, click the option name or type the underlined letter of the option or the whole option name.

Choose a command option

1. At the command prompt, type **Circle**.
Or, type **Ci**.
As soon as you have typed enough letters to uniquely identify the command, the **Circle** command automatically completes at the prompt. Press **Enter** or click the command name.
2. At the command prompt, the prompt for the **Circle** command appears:
Center of circle (Deformable Vertical 2Point 3Point Tangent AroundCurve FitPoints)
3. To draw a circle vertical to the active construction plane, use the **Vertical** option.
4. Click **Vertical** or, type **V**.

Select objects

Most operations you will do in Rhino require that you select one or more objects. One method of selecting is to click anywhere on the object. Click away from the object to deselect it. This method allows you to select one object at a time.

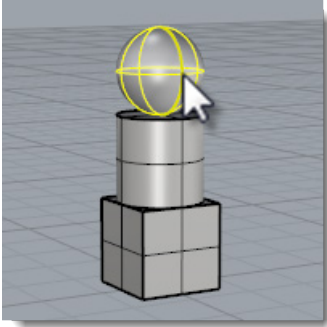
- ▶ To select additional objects, hold the **Shift** key while clicking the objects.
- ▶ To remove objects from the collection of selected objects, hold the **Ctrl** key and click the objects again.
- ▶ To cancel the selection, click away from the objects or press the **Esc** key.

Open the tutorial model

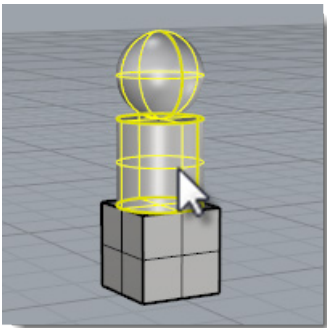
1. From the Rhino **File** menu, click **Open**.
2. Browse to the **Tutorial Models** folder that you downloaded with the *User's Guide*.
3. Click **Select Objects.3dm** and then click the **Open** button.

Practice selecting objects

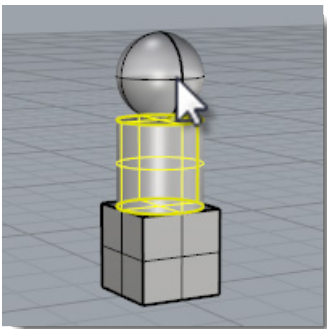
1. In the **Perspective** viewport, click to select the sphere.



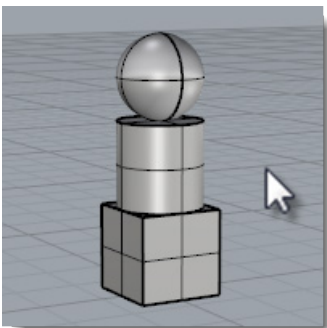
2. Hold the **Shift** key and select the cylinder.
The cylinder is added to the selection.



3. Hold the **Ctrl** key and select the sphere again.
The sphere is removed from the selection.



4. Click away from the objects, or press the **Esc** key.
The selection is canceled.



Select objects with windows

Another method is to make a window selection or a crossing selection to select multiple objects in one operation.

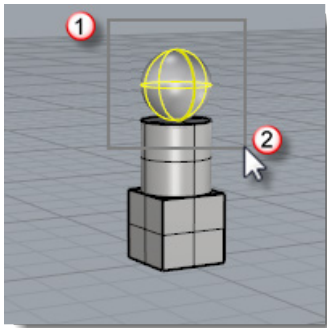
You can click in an open area of the screen and drag to create a selection window. To make a window selection click in an open area of the screen and drag to the right. To make a crossing selection click in an open area of the screen and drag to the left.

A window selection, selects all objects completely enclosed by the window. A crossing selection, selects all objects that are enclosed by the window or any object the window crosses.

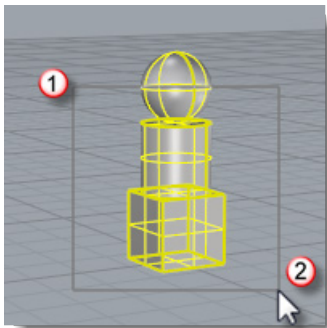
To add objects, hold the **Shift** key while making a window or crossing selection. To remove objects, hold the **Ctrl** key while making a window or crossing selection.

Practice window and crossing select

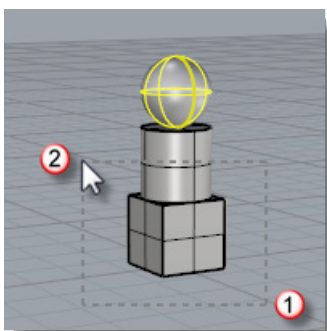
1. In the **Perspective** viewport, click and drag a window around the sphere.



2. In the **Perspective** viewport, hold the **Shift** key then click and drag a window around the box and the cylinder. The cylinder and box are added to the selection.



3. In the **Perspective** viewport, hold the **Ctrl** key then click and drag a crossing over the box and cylinder. The cylinder and box are removed from the selection.



Repeat the last command

Many tasks in Rhino are repetitive. You might want to move or copy several objects, for example. Several methods are provided for repeating commands.

To repeat the last command

- ▶ Press the **Enter** key when no command is active.
- ▶ In addition to pressing the **Enter** key on your keyboard, you can press the spacebar or click the right mouse button in a viewport. These all perform the same function.
Some commands, such as **Undo** and **Delete** do not repeat. Instead, the command prior to these commands is repeated. This prevents you from accidentally undoing too many commands or deleting objects accidentally. In addition, you often want to repeat the command you were using before undoing a mistake. You can define the list of commands that do not repeat to suit your own way of working.

Get help any time

To get answers to your questions about the mathematical principles involved in 3-D modeling, see: <http://www.mathopenref.com/>.

The Rhino **Help** file is the major resource for detailed information on specific commands.

To get help on a specific command

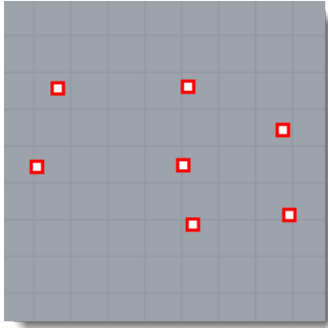
- ▶ To get Help for a command, press **F1** while the command is running.
- ▶ On the Rhino **Help** menu, click **Command Help**.
The Rhino **Help** will display in a dockable window.
- ▶ Click **Auto-Update** to display the Help topic for the current command.
The Help window appears with the specific command topic visible.
- ▶ Find the answers to frequently asked questions at: <http://www.rhino3d.com/support.htm>.

Chapter 2: Rhino Objects

The fundamental geometric objects in Rhino are points and NURBS curves, surfaces, polysurfaces and polygon mesh objects.

Points

Point objects mark a single point in 3-D space. They are the simplest objects in Rhino. Points can be placed anywhere in space. Points are most often used as placeholders.



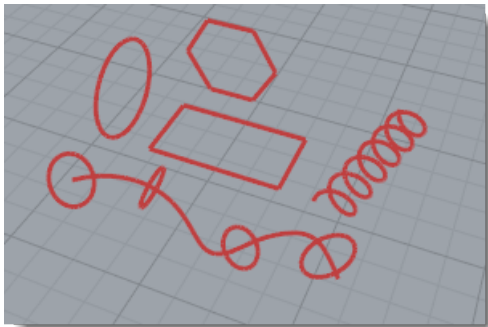
Curves

A Rhino curve is similar to a piece of wire. It can be straight or wiggled, and can be open or closed.

A *polycurve* is several curve segments joined together end to end.

Rhino provides many tools for drawing curves. You can draw straight lines, polylines that consist of connected line segments, arcs, circles, polygons, ellipses, helices, and spirals.

You can also draw curves using curve *control points* and draw curves that pass through selected points.

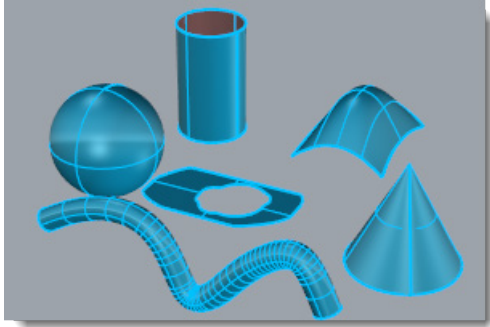


Curves in Rhino include lines, arcs, circles, free-form curves, and combinations of these. Curves can be open or closed, planar or non-planar.

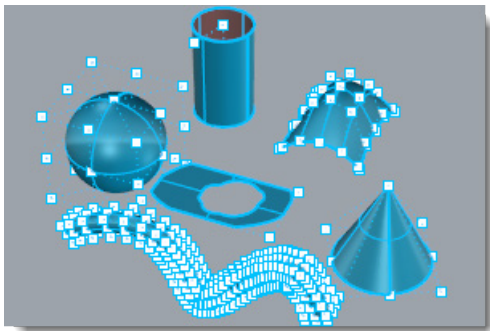
Surfaces

A surface is like a rectangular stretchy rubber sheet. The NURBS form can represent simple shapes, such as planes and cylinders, as well as free-form, sculptured surfaces.

All surface creation commands in Rhino result in the same object: a NURBS surface. Rhino has many tools for constructing surfaces directly or from existing curves.



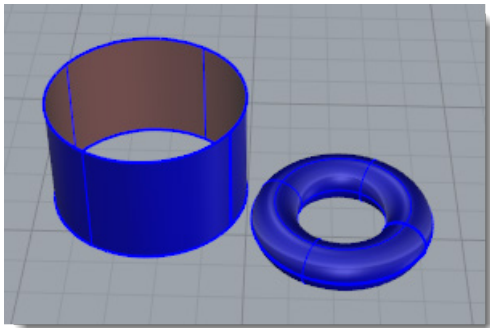
All NURBS surfaces have an inherently rectangular organization. Even a closed surface such as a cylinder is like a rectangular piece of paper that has been rolled up so two opposite edges are touching. The place where the edges come together is called the *seam*. If a surface does not have a rectangular shape, either it has been trimmed or the control points on the edges have been moved.



Closed and open surfaces

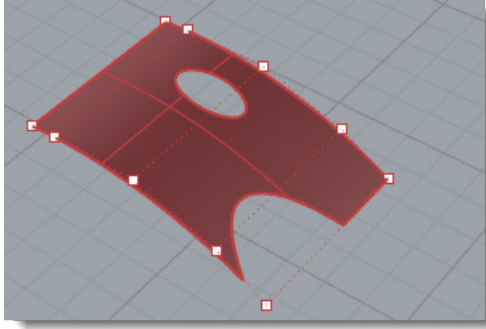
A surface can be open or closed. An open cylinder is closed in one direction.

A torus (donut shape) is closed in two directions.



Surface control points

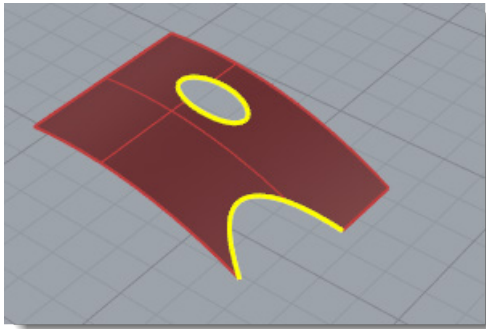
The shape of a surface is defined by a set of control points that are arranged in a rectangular pattern.



Trimmed and untrimmed surfaces

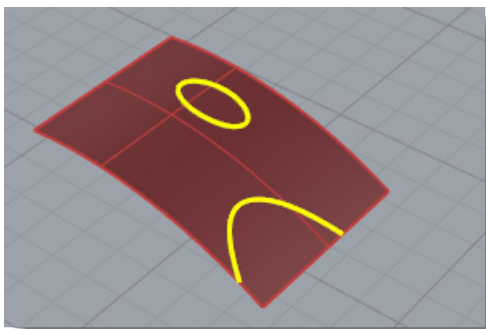
Surfaces can be trimmed or untrimmed. A trimmed surface has two parts: a surface that underlies everything and defines the geometric shape, and trimming curves that mark sections of the underlying surface that are removed from view.

Trimmed surfaces are created with commands that trim or split surfaces with curves and other surfaces. Some commands create trimmed surfaces directly.

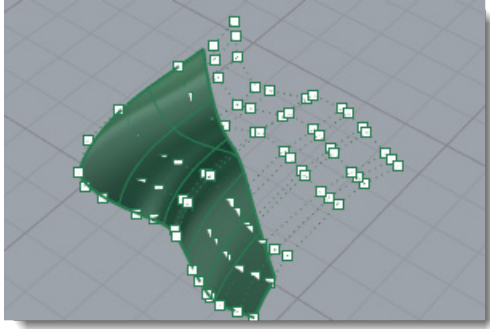


Since it can be important for you to know if a surface is trimmed, the **Properties** command lists the trimmed or untrimmed state of the surface. Some Rhino commands work only with untrimmed surfaces and some software does not import trimmed NURBS surfaces.

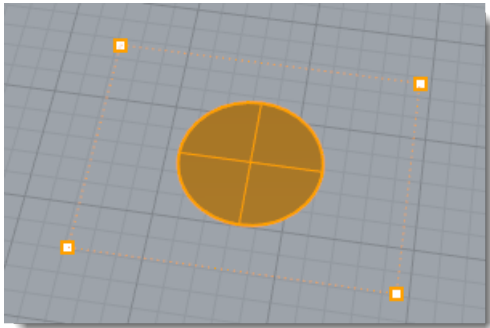
Trimming curves lie on the underlying surface. This surface may be larger than the trim curves, but you will not see the underlying surface because Rhino does not draw the part of the surface that is outside the trim curves. Every trimmed surface retains information about its underlying surface geometry. You can remove the trimming curve boundaries to make the surface untrimmed with the **Untrim** command.



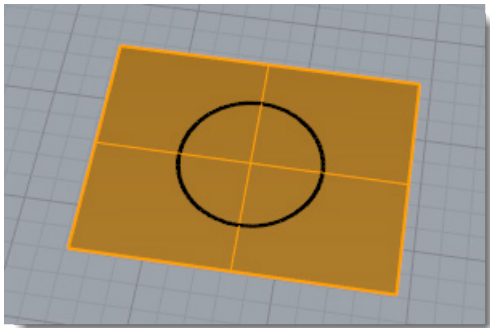
If you have a trim curve that runs across a surface, the trim curve itself does not have any real relationship to the control point structure of the surface. You can see this if you select such a trimmed surface and turn its control points on. You will see the control points for the whole underlying surface.



If you create a surface from a planar curve, it can be a trimmed surface. The illustrated surface was created from a circle. The control points display shows the rectangular structure of the surface.



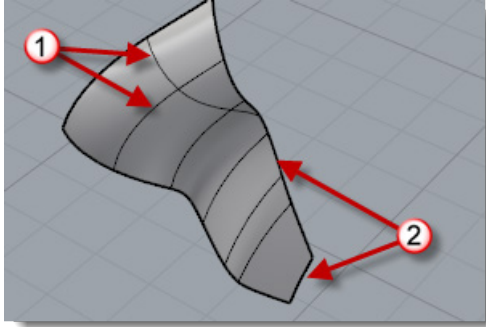
The **Untrim** command removes the trimming curve from the surface to get back to the underlying untrimmed rectangular surface.



Surface isoparametric and edge curves

In wireframe view, surfaces look like a set of crossing curves. These curves are called *isoparametric curves* or *isocurve*. These curves help you visualize the shape of the surface. Isoparametric curves do not define the surface the way the polygons do in a polygon mesh. They are merely a visual aid that allows you to see the surface on the screen. When a surface is selected, all of its isoparametric curves highlight.

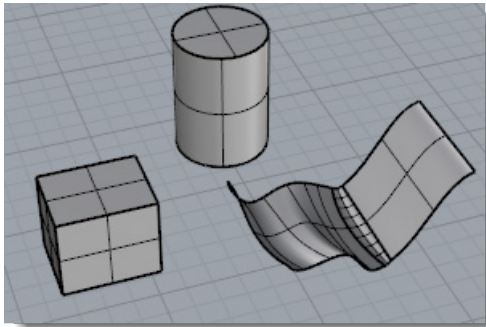
Edge curves bound the surface. Surface edge curves can be used as input to other commands.



Isocurves (1), Edge curves (2).

Polysurfaces

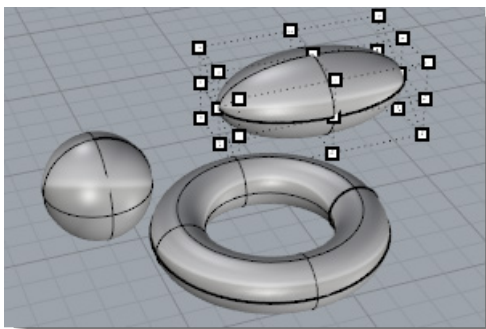
A polysurface consists of two or more surfaces that are joined together. A polysurface that encloses a volume of space defines a solid.



Solids

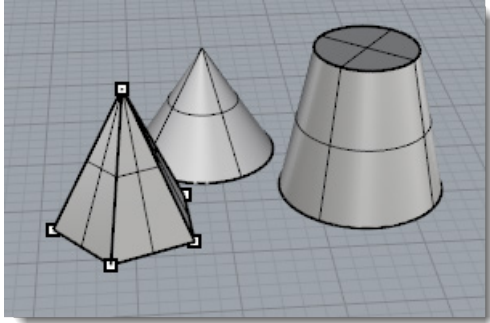
A solid is a surface or polysurface that encloses a volume. Solids are created anytime a surface or polysurface is completely closed. Rhino creates single-surface solids, polysurface solids, and extrusion solids.

A single surface can wrap around and join itself. Example commands include **Sphere**, **Torus**, and **Ellipsoid**. Control points can be displayed on single-surface solids and moved to change the surface.



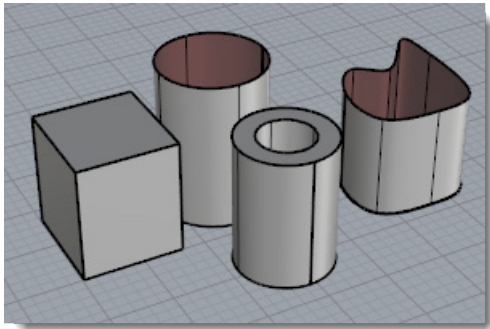
Some Rhino commands create polysurface solids. **Pyramid**, **Cone**, and **TCone** are examples of commands that create polysurface solids.

The **SolidPtOn** command turns on grip points for polysurfaces, which act like control points.



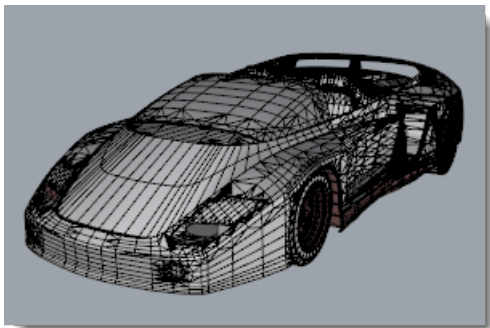
Extrusions

Light-weight extrusions are objects created using only a length and a profile curve. The **Box**, **Cylinder**, **Tube**, and **ExtrudeCrv** commands create extrusion objects. Extrusions can be closed with a planar cap or open. These objects can be converted to polysurfaces if necessary to add additional information for editing.



Meshes

Because there are many modelers that use polygon meshes to represent geometry for rendering and animation, stereolithography, visualization, and finite element analysis, the **Mesh** command translates NURBS geometry into polygonal meshes for export. In addition, the Mesh creation commands **MeshSphere**, **MeshBox**, **MeshCylinder**, etc., draw mesh objects.



Note: There is no easy way to convert a mesh model into a NURBS model. The information that defines the objects is completely different. However, Rhino has a few commands for drawing curves on meshes and extracting vertex points and other information from mesh objects to assist in using mesh information to create NURBS models.

Chapter 3: Viewports

The viewport title has some special functions for manipulating the viewport.

- Click the title to make the viewport active without disturbing the view.
- Drag the viewport title to move the viewport.
- Double-click the viewport title to maximize the viewport. Double-click again to restore the size to normal.

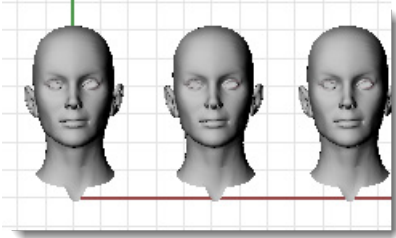
Viewport projection

Viewports can have one of three projections: parallel, perspective or two-point perspective.

Right mouse navigation works differently in the two viewport styles. In parallel views, right-mouse dragging pans the view. In perspective views, right-mouse dragging rotates the view. In the usual four-viewport layout, there are three parallel viewports and one perspective viewport.

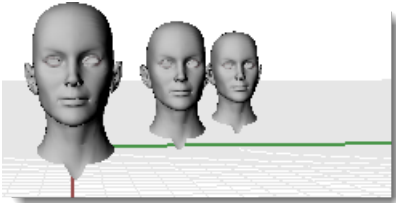
Parallel

Parallel views are also called orthogonal views in some systems. In a parallel view, all the grid lines are parallel to each other, and identical objects look the same size, regardless of where they are in space.



Perspective

In a perspective view, grid lines converge to a vanishing point. This provides the illusion of depth in the viewport. Perspective projection makes objects farther away look smaller.



Viewport navigation

Rhino's easy navigation helps you to visualize your model.

The simplest way to change the view is to drag the mouse with right mouse button held down. This pans the view in parallel views and rotates the view in perspective views.

You can change your view in the middle of a command to see precisely where you want to select an object or choose a point.

To zoom in and out, hold down the Ctrl key and drag up and down with the right mouse button held down. If you have a mouse with a wheel, use the wheel to zoom in and out.



Pan

In parallel viewports (for example: **Top**, **Front**, and **Right**), drag with the right mouse button.

In perspective viewports, Shift + drag with the right mouse button.

**RotateView**

In parallel viewports (for example: **Top**, **Front**, and **Right**), **Ctrl + Shift + drag** with the right mouse button.

In perspective viewports, drag with the right mouse button.

**Zoom**

Ctrl+ drag the right mouse button up and down

Rotate the mouse wheel.

Mouse navigation

Working in 3-D on a computer requires visualizing three-dimensional objects drawn on a two-dimensional medium—the computer screen. Rhino provides tools to help do this.

Drag with the right mouse button to easily manipulate the views to look at the model from various angles. Use the right mouse button view manipulations in both wireframe and shaded views.

**To pan in a viewport**

- ▶ In the **Top** viewport, drag the mouse with the right mouse button held down to pan the view.

**To zoom in and out with the right mouse button**

1. In the **Top** viewport, hold the **Ctrl** key, click and hold the right mouse button, and drag the mouse up and down.
2. Drag up to zoom in.
3. Drag down to zoom out.

**Restore a view**

If you get lost, there are several ways to get yourself reoriented:

- ▶ Press the **Home** key to step back through your view changes.
- ▶ Press the **End** key to step forward through your view changes.
- ▶ To set your view so you are looking straight down on the construction plane, use the **Plan** command.
- ▶ To bring all your objects into view, use the **Zoom** command **Extents** option.

Practice rotating the view

1. Click the left mouse button in the **Perspective** viewport to make it active.
An active viewport is the viewport where all your commands and actions take place. The active viewports title highlights so you can easily see which viewport is active.
2. In the **Perspective** view, drag the mouse with the right mouse button held down to rotate the view and see the objects from a different angle.

Viewport title menu

Right-click or click the down arrow on any **viewport title** to display a menu. From this menu, pan, rotate, zoom, set one of the standard views, set a construction plane, set the camera and target locations, choose a shading option, set the grid, and set other viewport properties.

Viewport display modes

You can view your model in a variety of ways that depend on your own needs. Wireframe display usually offers the fastest display speed, shaded modes offer the ability to view surfaces and solids with shading to help you visualize the shapes.

Standard and customized shaded modes allow easier visualization of surfaces and solids.

Wireframe display

In wireframe view, surfaces look like a set of crossing curves. These curves are called *isoparametric curves* or *isocurves*.

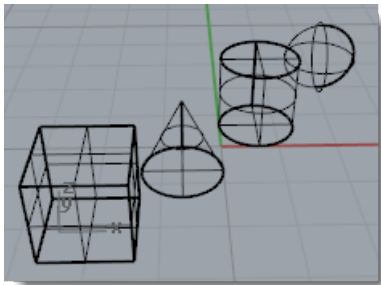
Isocurves do not define the surface the way the polygons do in a polygon mesh. They are merely a visual aid.

The **Wireframe** mode sets the viewport display to wireframe.



Wireframe mode

1. Click a mouse button in the **Perspective** viewport to make it active.
An active viewport is the viewport where all your commands and actions take place.
2. On the viewport title menu, click **Wireframe**.



Shaded display

Shaded modes display surfaces and solids with the surfaces shaded using their layer, object, or custom color. You can work in any of the shaded modes. The surfaces are opaque or transparent.

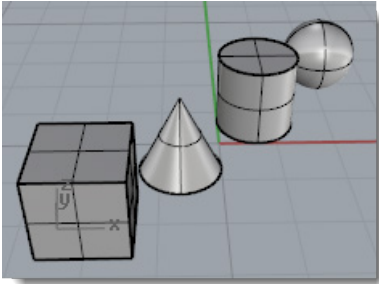
The **Shaded** mode sets the viewport display to shaded mode.





Shaded mode

1. On the viewport title menu, click **Shaded**.



2. Rotate your view by holding down the right mouse button and dragging from the bottom of the view toward the top.

You are now under the objects looking up.

The construction plane grid helps you stay oriented. If the objects are behind the grid, you are looking at the bottom of the construction plane.

3. Press the **Home** key to undo your view changes.



Rendered display

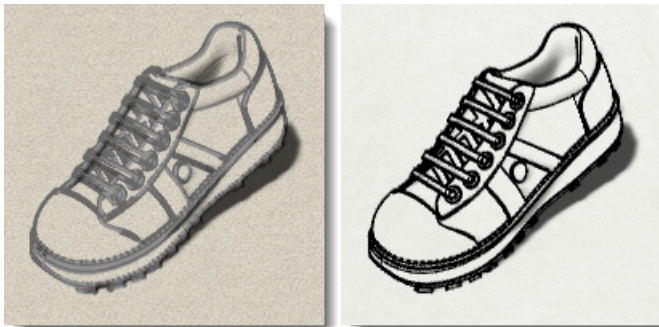
Rendered viewport display shows the objects with lighting and render materials applied

The **Rendered** mode sets the viewport display to rendered mode.



Other shaded modes

Other display modes and custom settings are described in the Rhino Help.



Artistic (left) and Pen (right) display modes.

Viewport title

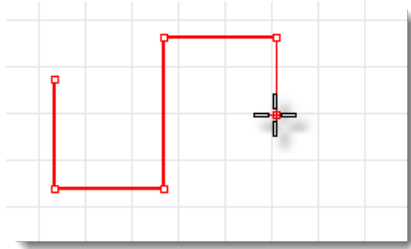
The viewport title has some special functions for viewport control.

- ▶ Click the title to make the viewport active without disturbing the view.
- ▶ Drag the viewport title to move the viewport.
- ▶ Double-click the viewport title to maximize the viewport. Double-click again to restore the size to normal.

Viewport title menu

- ▶ Right-click the viewport title or click the down-arrow on the menu to display the viewport title menu.

From the viewport title menu you can maximize the viewport, pan, rotate, zoom, set one of the standard views, set a construction plane, set the camera and target locations, choose a shading option, set the grid, and open the **Viewport Properties** dialog box.



Ortho on.

If you only need a different angle for a single operation, angle constraint is faster to use. Enter a specific angle for one operation instead of changing the ortho angle and then changing it back.

Object snaps

Object snaps constrain the marker to specific points on an object. When Rhino asks you to specify a point, you can constrain the marker to specific parts of existing geometry. When an object snap is active, moving the cursor near a specified point on an object causes the marker to jump to that point.

In this chapter you will learn:

- How to use object snaps to find specific points on geometry.
- How to set, clear, and suspend persistent object snaps.
- How to use one-time object snaps.
- How to use object snaps in combination with other modeling aids.

Object snaps can persist from pick to pick, or can be activated for one pick only. Multiple persistent object snaps can be set from the status bar. All object snaps behave similarly, but snap to different parts of existing geometry. In addition, there are special object snaps that work for one pick only.

Persistent object snaps

Use persistent objects snaps to maintain an object snap through choosing several points. Since persistent object snaps are easy to turn on and off, you can set them and leave them on until they get in your way. You can then set a different one or just disable them.

Sometimes object snaps interfere with each other and with grid snap or ortho. Object snaps normally take precedence over grid snap or other constraints.

There are other situations where object snaps work in conjunction with other constraints. You will see examples of this in this chapter. For more information including video demonstrations, see the Help topic "Object Snaps."

To turn persistent object snaps on and off

1. On the **status bar**, click the **Osnap** pane.
2. In the **Osnap** toolbar, check or clear the desired object snaps.

To suspend all persistent object snaps

- ▶ In the **Osnap** toolbar, click the **Disable** button.
All persistent object snaps will be suspended, but remain checked.

To clear all persistent object snaps

- ▶ In the **Osnap** toolbar, click the **Disable** button with the right mouse button.
All persistent object snaps will be cleared.

To turn on one object snap and turn all others off with one click

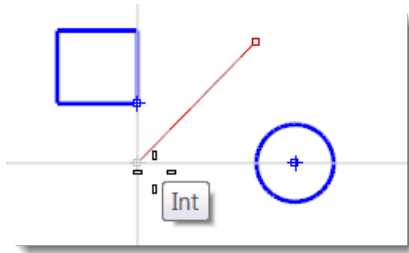
- ▶ In the **Osnap** toolbar, right-click the object snap you want to turn on.

SmartTrack

SmartTrack™ is a system of temporary reference lines and points that is drawn in the Rhino viewport using implicit relationships among various 3-D points, other geometry in space, and the coordinate axes' directions.

Temporary infinite lines (tracking lines) and points (smart points) are available to object snaps very much like real lines and points.

You can snap to intersections of the tracking lines, perpendiculars, and directly to smart points as well as intersections of tracking lines and real curves. The tracking lines and smart points are displayed for the duration of a command.



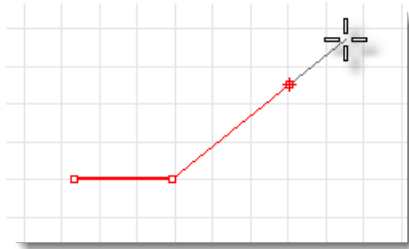
Cursor constraints

When entering points, you can constrain the marker to a distance or angle from the previous point. Once you have set the distance, drag the line around to any angle. You can also use further snaps to point the line in a specific direction.

Distance constraint

During any command that requires two points, such as the **Line** command, place the first point. Then at the next prompt, type a distance and press **Enter** or the **Spacebar**.

The marker will be constrained to the specified distance from the previous point. Drag the cursor around the first point and then pick a point.



Angle constraint

Angle constraint is similar to ortho, but you can set any angle and it is a one-time setting.

The < symbol is used because it is similar to the symbol used in geometry to indicate an angle.

The marker will be constrained to lines radiating from the previous point separated by the specified angle, where the first line is the specified number of degrees counterclockwise from the x-axis. If you enter a negative number, the angle will be clockwise from the x-axis.

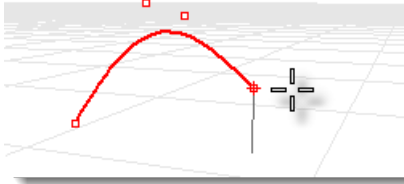
Distance and angle together

Distance and angle constraints can be used at the same time. Type the distance at the prompt and press **Enter**, and then type < and then the angle and press **Enter**. The order of the distance and angle does not matter. The marker will drag around your original point at angle increments at the specified distance.

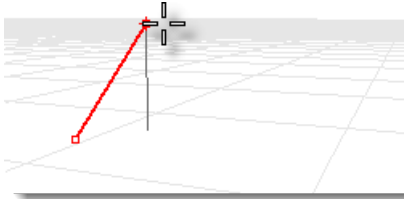
Elevator mode

Pick a second point to specify the z-coordinate of the desired point. It is easiest to see this in a different viewport or use the Perspective viewport. Drag the mouse cursor around to see the marker move vertically from the base point along the tracking line.

Pick the point with the mouse or type the height above the construction plane. Positive numbers are above the construction plane; negative numbers are below it. You can use further constraints like coordinates, object snaps or grid snap for the first point, and you can use object snaps for the height.



To move the marker in the construction plane z-direction, hold the **Ctrl** key and click a point on the construction plane, and then drag vertically from the construction plane and click to pick a point. This constraint is called elevator mode. Using elevator mode to move your pick point vertically from the construction plane lets you work more in the Perspective viewport.



Chapter 5: Coordinate Systems

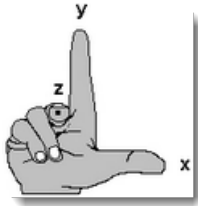
Rhino uses two coordinate systems: construction plane coordinates and world coordinates. World coordinates are fixed in space. Construction plane coordinates are defined for each viewport.

Cartesian coordinates

When Rhino prompts you for a point, if you type x and y Cartesian coordinates, the point will lie on the construction plane of the current viewport. For more information about coordinate systems and numeric constraints, see the Rhino Help topic, Unit Systems.

Right-hand rule

Rhino follows what is called the *right-hand rule*. The right-hand rule can help you determine the direction of the z-axis. Form a right angle with the thumb and forefinger of your right hand. When your thumb points in the positive x-direction, your forefinger points in the positive y-direction, and the palm of your hand faces in the positive z-direction.



World coordinates

Rhino contains one world coordinate system. The world coordinate system cannot be changed. When Rhino prompts you for a point, you can type coordinates in the world coordinate system.

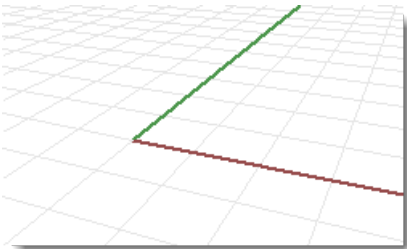
The arrow icon in the lower left corner of each viewport displays the direction of the world x-, y-, and z-axes. The arrows move to show the orientation of the world axes when you rotate a view.



Construction plane coordinates

Each viewport has a construction plane. A **construction plane** is like a tabletop that the cursor moves on unless you use coordinate input, elevator mode, or object snaps or a few other instances where input is constrained. The construction plane has an origin, x- and y-axes, and a grid. The construction plane can be set to any orientation. By default, each viewport's construction plane is independent of those in other viewports.

The construction plane represents the local coordinate system for the viewport and can be different from the world coordinate system.



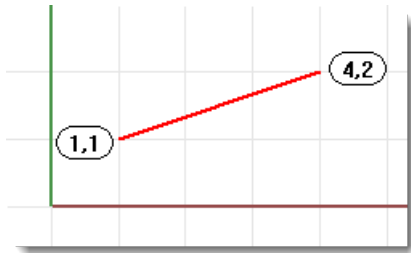
Rhino's standard viewports come with construction planes that correspond to the viewport. The default **Perspective** viewport, however, uses the world Top construction plane, which is the same construction plane that is used in the Top viewport.

The grid lies on the construction plane. The dark red line represents the construction plane x-axis. The dark green line represents the construction plane y-axis. The red and green lines meet at the construction plane origin.

To change the direction and origin of a construction plane, use the **CPlane** command. Preset construction planes (World Top, Right, and Front) give you quick access to common construction planes. In addition, you can save and restore named construction planes and import named construction planes from another Rhino file.

2-D construction plane coordinates

- ▶ At a prompt, type the coordinate in the format **x,y** where **x** is the x-coordinate and **y** is the y-coordinate of the point.



A line from 1,1 to 4,2.

3-D construction plane coordinates

- ▶ At a prompt, type the coordinate in the format **x,y,z** where **x** is the x-coordinate, **y** is the y-coordinate, and **z** is the z-coordinate of the point.
There are no spaces between the coordinate values.
- ▶ To place a point 3 units in the x-direction, 4 units in the y-direction, and 10 units in the z-direction from the construction plane origin, type **3,4,10** at the prompt.

Note If you enter only x- and y-coordinates, the point will lie on the construction plane.

Relative coordinates

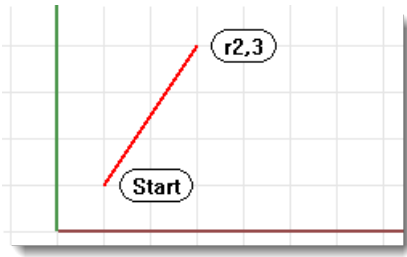
Rhino remembers the last point used, so you can enter the next point relative to it. Relative coordinates are useful for entering a list of points where the relative locations instead of absolute locations of the points are known. Use relative coordinates to locate points according to their relationship to the previous active point.

Relative coordinates

- ▶ At a prompt, type the coordinates in the format **rx,y** where **r** signifies that the coordinate is relative to the previous point.

For example

1. Start the **Line** command.
2. At the **Start of line...** prompt, click to place the first end of the line.
3. At the **End of line...** prompt, type **r2,3**, and press **Enter** or the spacebar.
The line is drawn to a point 2 units in the x-direction and 3 units in the y-direction from the last point.



Chapter 6: Create Surfaces from Curves

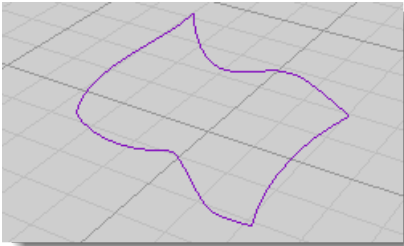
A common way of working in 3-D is to draw curves that represent edges, profiles, cross-sections, or other surface features and then to use surfacing commands to create surfaces from those curves.

Surface from edge curves

You can create a surface from three or four curves that form the sides of the surface.

Open the tutorial model

1. From the Rhino **File** menu, click **Open**.
2. Browse to the **Tutorial Models** folder that you downloaded with the *User's Guide*.
3. Click **EdgeSrf.3dm** and then click the **Open** button.



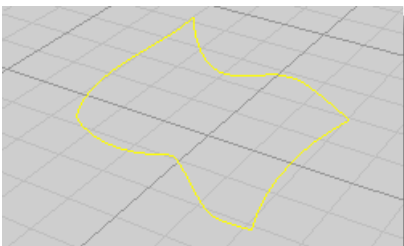
Review the Help topic

- ▶ Review the **Help** topic for the **EdgeSrf** command.



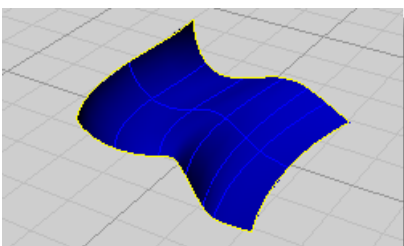
Create a surface from edge curves

1. From the **Surface** menu, click **Edge Curves**.
2. **Select** the curves.



Objects change to yellow when you select them.

A surface is created from the curves that form its edges.

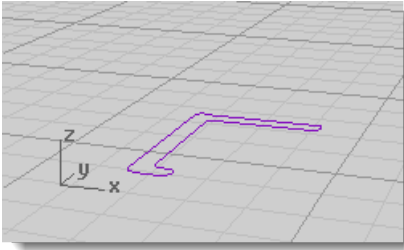


Extrude a curve

Open the tutorial model

1. From the Rhino **File** menu, click **Open**.

2. Browse to the **Tutorial Models** folder that you downloaded with the *User's Guide*.
3. Click **Extrude.3dm** and then click the **Open** button.



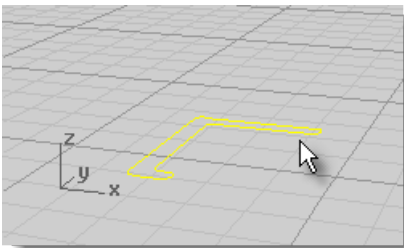
Review the Help topic

- ▶ Review the [Help](#) topic for the [ExtrudeCrv](#) command.

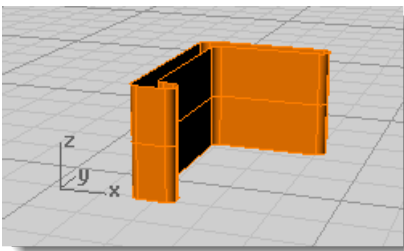
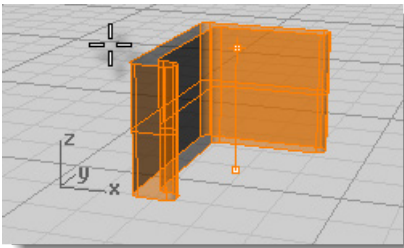


Create an extruded surface

1. From the **Surface** menu, click **Extrude Curve**, and then click **Straight**.
2. **Select** the curve.



2. At the **Extrusion distance** prompt, drag a distance with your mouse and click.



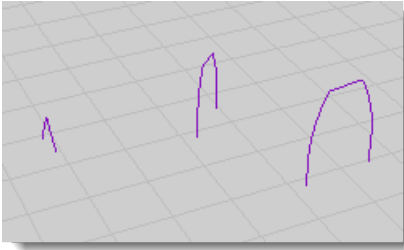
Loft through curves

Lofting creates a smooth surface that blends between selected shape curves. This surface looks similar to the Sweep a curve with two rails example, but is created without rail curves. Instead, the edges of the surface are created by fitting smooth curves through the shape curves.

Open the tutorial model

1. From the Rhino **File** menu, click [Open](#).

2. Browse to the **Tutorial Models** folder that you downloaded with the *User's Guide*.
3. Click **Loft.3dm** and then click the **Open** button.



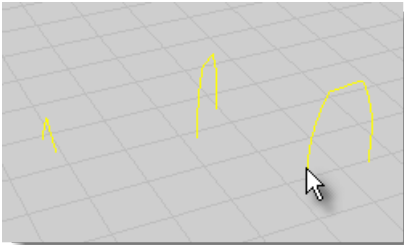
Review the Help topic

- ▶ Review the **Help** topic for the **Loft** command.

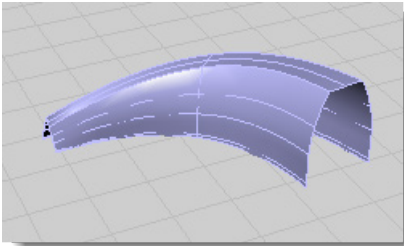


Create a lofted surface

1. From the **Surface** menu, click **Loft**.
2. **Select** the three curves and press Enter.



3. In the **Loft Options** dialog box, click **OK**.



4. Try some of the **Style** options and then click **Preview** to see the various loft styles.

Revolve a curve

Revolving a curve creates a surface by revolving a profile curve about an axis. This is sometimes called **lathing**.

Open the tutorial model

1. From the Rhino **File** menu, click **Open**.
2. Browse to the **Tutorial Models** folder that you downloaded with the *User's Guide*.
3. Click **Revolve.3dm** and then click the **Open** button.



Set up modeling aids

1. Turn on **End** object snap to help you locate the endpoints of the axis line.
2. In the **status bar**, click **Osnap**.
3. In the **Osnap** dialog box, click **End**.

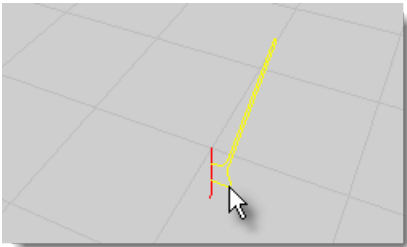
Review the Help topic

- ▶ Review the **Help** topic for the **Revolve** command.

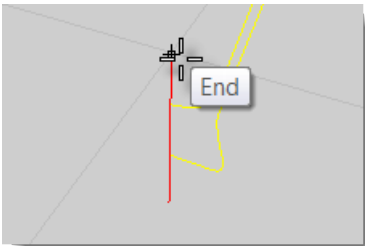


Create a revolved surface

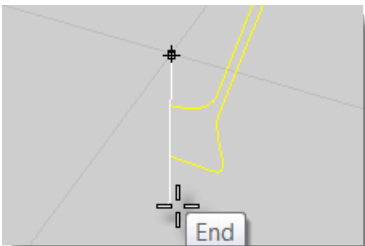
1. From the **Surface** menu, click **Revolve**.
2. **Select the profile curve** and press **Enter**.



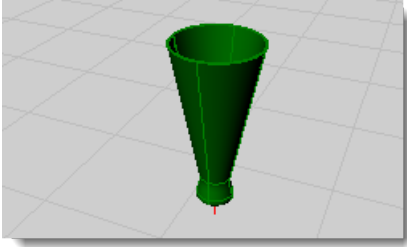
3. At the **Start of revolve axis** prompt, snap to one end of the axis line.



4. At the **End of revolve axis** prompt, snap to the other end of the axis line.



5. At the **Start angle...** prompt, select the **FullCircle** option.

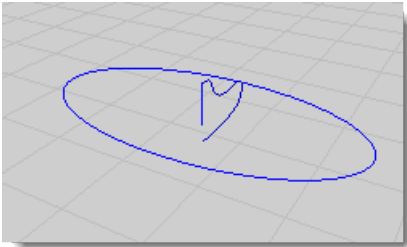


Revolve a curve with a rail

Rail revolve creates a surface by revolving a profile curve around an axis while at the same time following a rail curve. This is basically the same as Sweep Along 2 Rails, except one of the rails is a central point.

Open the tutorial model

1. From the Rhino **File** menu, click **Open**.
2. Browse to the **Tutorial Models** folder that you downloaded with the *User's Guide*.
3. Click **RailRevolve.3dm** and then click the **Open** button.



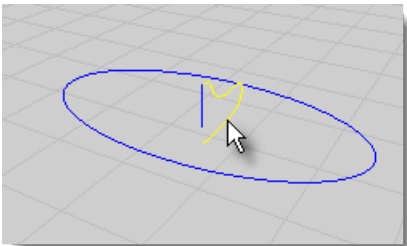
Review the Help topic

- ▶ Review the **Help** topic for the **RailRevolve** command.

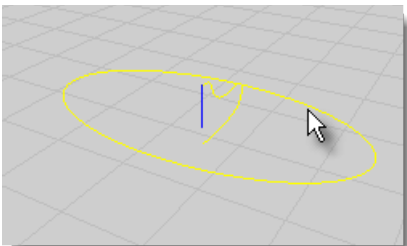


Create a revolved surface with a rail curve

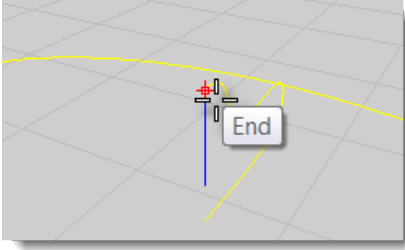
1. From the **Surface** menu, click **Rail Revolve**.
2. **Select** the profile curve (highlighted in yellow below).



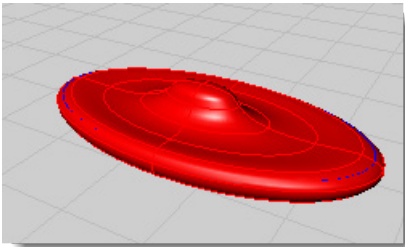
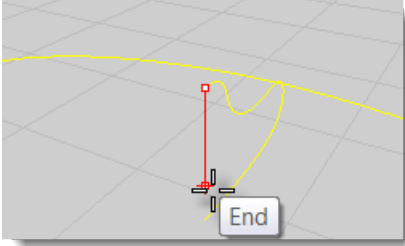
3. At the **Select rail curve...** prompt, select the rail curve the revolve will follow as shown highlighted in yellow in the illustration.



4. At the **Start of RailRevolve axis** prompt, snap to an endpoint of the axis line.



5. At the **End of RailRevolve axis** prompt, snap to the other end of the axis line.

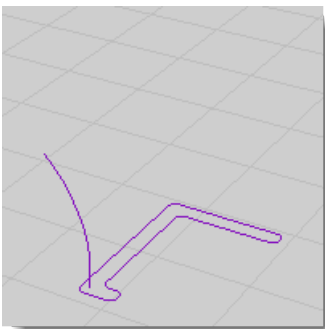


Sweep along a single rail curve

Sweeping creates a surface with cross sections that maintain the initial orientation of the shape curve(s) to the path curve.

Open the tutorial model

1. From the Rhino **File** menu, click **Open**.
2. Browse to the **Tutorial Models** folder that you downloaded with the *User's Guide*.
3. Click **Sweep1.3dm** and then click the **Open** button.



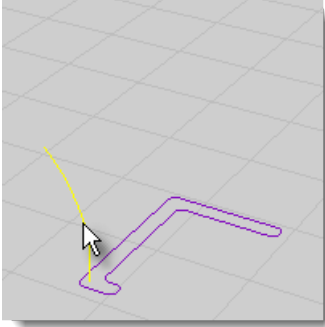
Review the Help topic

- ▶ Review the **Help** topic for the **Sweep1** command.

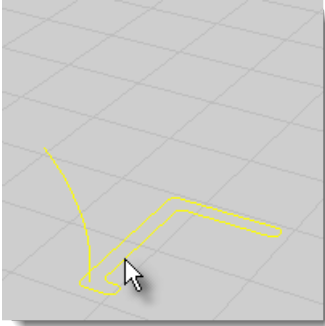


Create a sweep surface

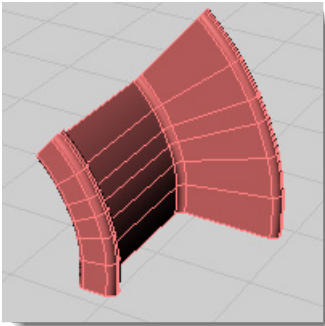
1. From the **Surface** menu, click **Sweep 1 Rail**.
2. **Select** the rail curve as shown in the illustration.



3. At the **Select cross section curves ...** prompt, select the cross-section curve, and press **Enter**.



4. In the **Sweep 1 Rail Options** dialog box, click **OK**.

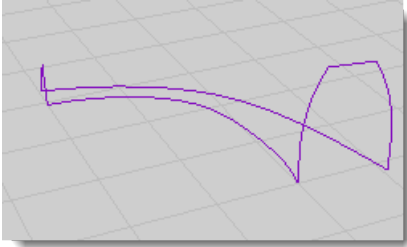


Sweep with two rail curves

Using two rails for a sweep creates a smooth surface through two or more shape curves that follow two curve rails. The rails also affect the overall shape of the surface. Use this command when you want to control the location of the edges of the surface.

Open the tutorial model

1. From the Rhino **File** menu, click **Open**.
2. Browse to the **Tutorial Models** folder that you downloaded with the *User's Guide*.
3. Click **Sweep2.3dm** and then click the **Open** button.



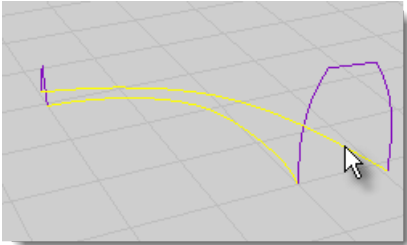
Review the Help topic

- ▶ Review the [Help](#) topic for the [Sweep2](#) command.

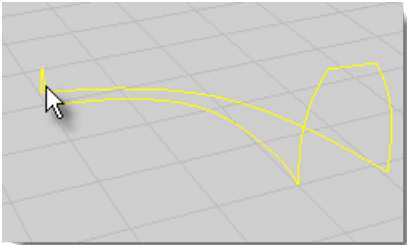


Create a sweep surface with two rail curves

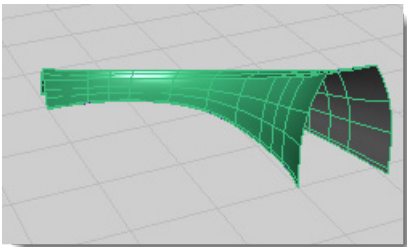
1. From the **Surface** menu, click **Sweep 2 Rail**.
2. **Select** the first rail curve.
3. At the **Select second rail...** prompt, select the second rail curve.



4. At the **Select cross section curves** prompt, select the two cross-section curves (highlighted in yellow below), and press **Enter**.



5. In the **Sweep 2 Rails Options** dialog box, click **OK**.



Chapter 7: Edit Curves and Surfaces

The editing operations in this section break objects apart, cut holes in them, and put them back together. Some of these commands connect curves to curves or surfaces to surfaces or polysurfaces and break a composite curve or polysurface into its components.

The commands: **Join**, **Explode**, **Trim**, and **Split** apply to curves, surfaces, and polysurfaces.

The **Rebuild**, **ChangeDegree**, and **Smooth** commands alter the shape of a curve or surface by changing its underlying control point structure.

In addition, objects have properties that are assigned to them such as color, layer, rendering material, and other attributes depending on the object. The **Properties** command manages these properties.

Join

The **Join** command connects curves or surfaces together into one object. For example, a polycurve can consist of straight-line segments, arcs, polylines, and free-form curves. The **Join** command also connects adjacent surfaces into a polysurface.

Explode

The **Explode** command removes the connection between joined curves or surfaces. For polysurfaces, this is useful if you want to edit each individual surface with control points.

Trim and Split

The **Trim** and **Split** commands are similar. The difference is when you trim an object, you select the parts to remove and they are deleted. When you split an object, all parts are left.

The **Split** command will split a surface with a curve, surface, polysurface, or its own isoparametric curves.

The **Untrim** command removes a surface's trimming curve, with an option to keep the curve so you can re-use it.

Control-point editing

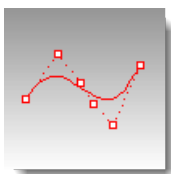
You can make subtle changes in the shape of a curve or surface by moving the location of its control points. Rhino offers many tools for editing control points. Some commands such as **Rebuild**, **Fair**, and **Smooth** offer some automated solutions for redistributing control points over a curve or surface. Other commands, such as control point dragging and nudging, **HBar**, and **MoveUVN**, let you manually control the location of individual or groups of control points.

Control point visibility

To edit curves and surfaces by manipulating control points, use the **PointsOn (F10)** command to turn the control points on.

When you are finished with control-point editing, use the **PointsOff** command or press **Esc** to turn them off.

Control points of polysurfaces cannot be turned on for editing. Editing the control points of polysurfaces could separate the edges of the joined surfaces creating "leaks" in the polysurface.



Change control point locations

When you move control points, the curve or surface changes, and Rhino smoothly redraws it. The curve or surface is not drawn through the control points rather it is attracted to the new positions of the control point. This allows the object to be smoothly deformed. When control points are on, Rhino's transform commands can manipulate the points. You can also rebuild surfaces to add control points and redistribute them.

Add, delete, or redistribute control points

Adding control points to a curve gives you more control over the shape of the curve. Manipulating control points also lets you remove kinks, make curves uniform, and add or subtract detail. The **Delete** key erases curve control points. This changes the shape of the curve.

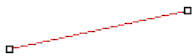
Curve and surface degree

A polynomial is a function like $y = 3x^3 - 2x + 1$. The "degree" of the polynomial is the largest power of the variable. For example, the degree of $3x^3 - 2x + 1$ is 3; the degree of $-x^5 + x^2$ is 5, and so on. NURBS functions are rational polynomials and the degree of the NURBS is the degree of the polynomial. From a NURBS modeling point of view, the (degree -1) is the maximum number of "bends" you can get in each span.

For example:

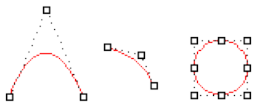
A degree-1 curve must have at least two control points.

A line has degree 1. It has zero "bends."



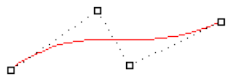
A degree-2 curve must have at least three control points.

A parabola, hyperbola, arc, and circle (conic section curves) have degree 2. They have one "bend."



A degree-3 curve must have at least four control points.

A cubic Bézier has degree 3. If you arrange its control points in a zig-zag shape, you can get two "bends."



Chapter 8: Transforms

Transforms change the location, rotation, number and shape of whole objects by moving, mirroring, arraying, rotating, scaling, shearing, twisting, bending, and tapering. The transform commands do not break the objects into pieces or cut holes in them.

Move

Use the **Move** command when you want to move an object a certain distance or if you want to use object snaps to place an object accurately. The quickest way is to click the object and drag it.

To move selected objects small distances, press and hold the **Alt** key and press an arrow key to activate the **Nudge** feature.

Move objects by dragging

You can drag objects to move them. Select an object and drag it. You can drag objects in any viewport. In this model, grid snap is set to one-half of the grid spacing. Using grid snap, you should be able to line objects up with each other.

Before you start, set the grid snap so the objects stay lined up with each other. In this model snap is set to one grid line.

Turn on grid snap

- ▶ In the status bar, click **Grid Snap** to turn on the grid snap.

The status bar is at the bottom of the Rhino window. You will find the **Grid Snap** pane in a row with other modeling aids: **Ortho**, **Planar**, and **Osnap**.

Grid snap may already be on in your system. Be careful that you do not turn it off instead of on.

This is an important step. Grid snap only lets you move in certain preset intervals. In this model, grid snap is set to one grid line. Grid snap helps you line up your objects.

Open the tutorial model

1. From the Rhino **File** menu, click **Open**.
2. Browse to the **Tutorial Models** folder that you downloaded with the *User's Guide*.
3. Click **Drag Objects.3dm** and then click the **Open** button.

Practice dragging objects

1. In the **Perspective** viewport, select the cone.
2. Drag the cone in the **Perspective** viewport until it lines up with the cylinder.
The cone moves on the base that is represented by the grid. This base is called a *construction plane*. A construction plane is like a tabletop where you move your objects. Each viewport has its own construction plane. When you start Rhino, the **Perspective** viewports construction plane is the same as the **Top** viewports.
3. In the **Front** viewport, drag the cone to the top of the cylinder.
Watch what happens in the **Perspective** viewport.
There are many times when you have to watch what is happening in other viewports to accurately place your objects.

Move objects vertically

So far you have probably moved and copied objects in the x,y-direction and then using a different viewport to move in the z-direction. You can also press the **Ctrl** key to move in the z-direction. This is called elevator mode. **Elevator mode** is like **Ortho**, except the movement is vertical to the active construction plane.

To practice using the **Ctrl** key to move vertically, you are going to move the cones to the tops of the cylinders in one step. Using elevator mode to move objects vertically lets you work more in the **Perspective** viewport.

Open the tutorial model

1. From the Rhino **File** menu, click **Open**.
2. Browse to the **Tutorial Models** folder that you downloaded with the *User's Guide*.
3. Click **Elevator Mode.3dm** and then click the **Open** button.

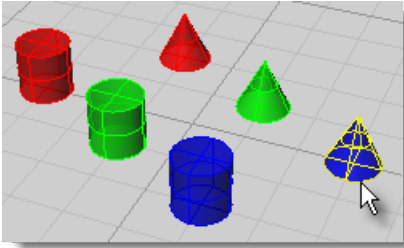
Review the Help topic

- ▶ Review the **Help** topic for the **Move** command.

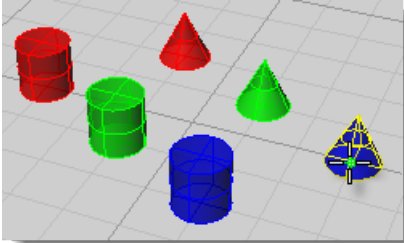


Move the cone

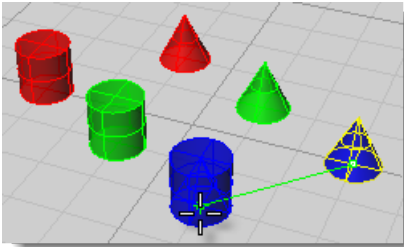
1. From the **Transform** menu, click **Move**.
2. Turn **Grid Snap on** and **Ortho off**.
3. In the **Perspective** viewport, select the cone.



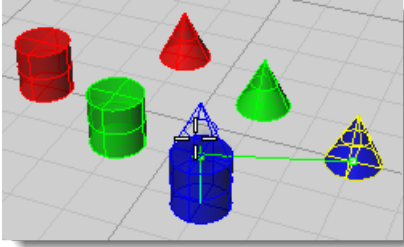
4. At the **Point to move from...** prompt, click the point at the center of the cone.



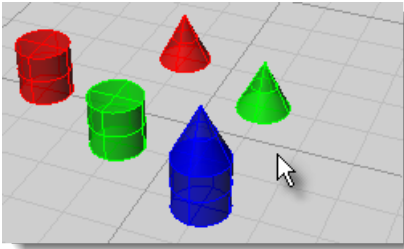
5. At the **Point to move to** prompt, drag the cone to the center of the cylinder and click.



6. Hold the **Ctrl** key, click and drag the cone up to the top of the cylinder in the **Perspective** viewport.



7. Release the left mouse button, and then release the **Ctrl** key and click to finish the move.



Copy

The **Copy** command makes copies of objects.

Some transform commands like **Rotate**, **Rotate3D**, and **Scale** have a **Copy** option. This lets you create a copy of the object as you rotate or scale it.

To copy objects by dragging, drag the objects and hold the **Alt** key .

To create more objects, copy the shapes.

Open the tutorial model

1. From the Rhino **File** menu, click **Open**.
2. Browse to the **Tutorial Models** folder that you downloaded with the *User's Guide*.
3. Click **Copy Objects.3dm** and then click the **Open** button.

Most Rhino commands are arranged in the menus. In this exercise, practice starting a command from the menu.

Review the Help topic

- ▶ Review the **Help** topic for the **Copy** command.

Practice copying objects

1. From the **Transform** menu, click **Copy**.
2. **Select** the cone.
3. At the **Point to copy from** prompt, click somewhere in the **Top** viewport.
It usually helps to click a spot that relates to the object, such as the middle.
4. At the **Point to copy to** prompt, click where you want the first copy.
Zoom in or out if you like.
To zoom, hold the **Ctrl** key and drag with the right mouse button or use the wheel on your mouse.
5. At the next **Point to copy to** prompts, click other places to make some copies of the box.
6. When you have enough copies, press **Enter**.

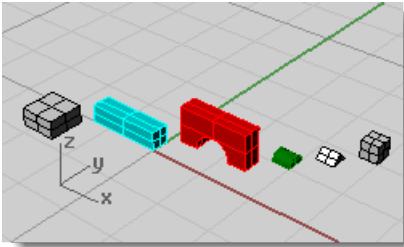
Rotate

The **Rotate** command rotates an object in relation to the construction plane.

In the next exercise, you will learn a new basic Rhino command: **Rotate**. The **Rotate** command rotates an object around a center point.

Open the tutorial model

1. From the Rhino **File** menu, click **Open**.
2. Browse to the **Tutorial Models** folder that you downloaded with the *User's Guide*.
3. Click **Rotate-Scale.3dm** and then click the **Open** button.



3. Practice rotating objects. First, rotate the long box in the **Top** viewport, and then rotate the half-cylinder in the **Right** viewport.

Review the Help topic

- ▶ Review the [Help](#) topic for the **Rotate** command.

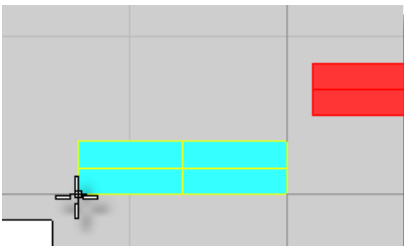


Rotate an object

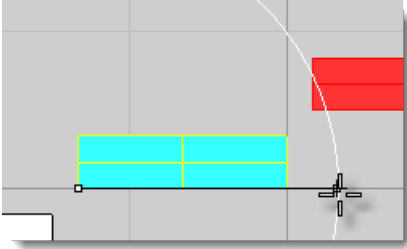
1. From the **Transform** menu, click **Rotate**.
2. On the status bar, turn **Grid Snap on**.
3. In the **Top** viewport, select the long box as shown in the illustration below.



4. At the **Center of rotation...** prompt, click the lower left corner of the box.



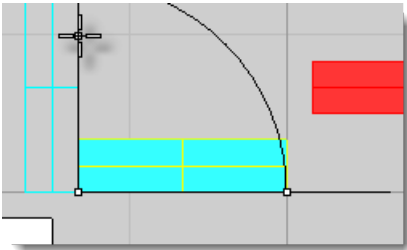
5. At the **Angle or first reference point...** prompt, drag the cursor to the right and click. Grid Snap will help you keep the tracking line straight.



Or, at the command prompt, type **90**.

If you know the angle you want, you can type it instead of dragging with the mouse.

- At the **Second reference point...** prompt, drag the cursor to rotate the box as shown in the illustration below, and click.



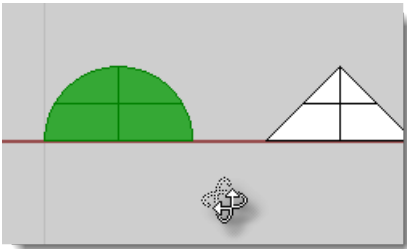
- From the **Edit** menu, click **Undo** to return the box to its original position.

Rotate the half cylinder

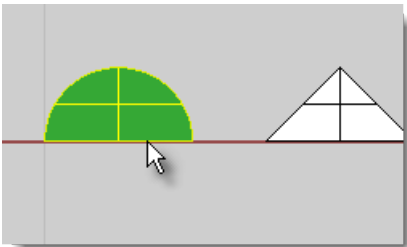
In the **Top** and **Front** viewports, you cannot really see the half cylinder because from your viewpoint, it looks nearly like a box. You will notice, however, that it has a different pattern of lines in its wireframe display. You can see it best in the **Right** viewport.

To rotate the half cylinder

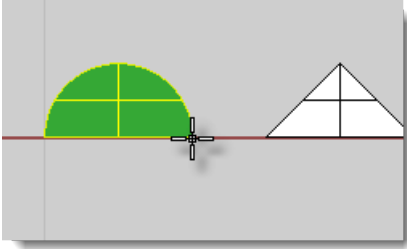
- In the **Right** viewport, zoom in so you can identify the half cylinder.



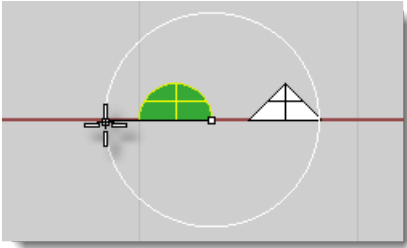
- In the **Right** viewport, select the half cylinder as shown in the illustration below.



- From the **Transform** menu, click **Rotate**.
Or press Enter, the **Spacebar**, or click the right mouse button to repeat the command.
- At the **Center of rotation...** prompt, click the lower right corner of the half cylinder.

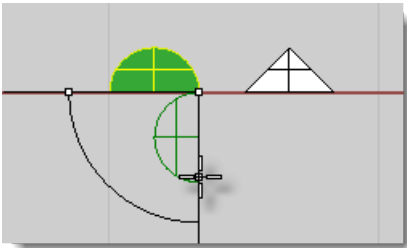


- At the **Angle or first reference point...** prompt, drag the cursor to the left and click. Grid Snap will help you keep the tracking line straight.



Or at the command prompt, type **90**.

- At the **Second reference point...** prompt, drag the cursor to rotate the half-cylinder as shown in the illustration below, and click.



- From the **Edit** menu, click **Undo** to return the half cylinder to its original position.

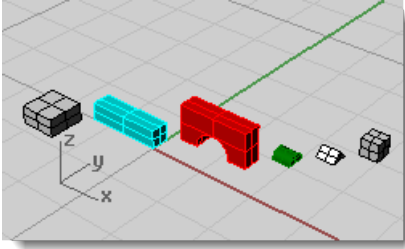
Scale

Scale commands give you control over the direction of the scale. You can resize objects uniformly in one, two, or three directions, or scale an object with a different scale factor in each direction.

You can change the size of objects in all three dimensions, but for now you are going to learn how to scale objects in only one direction. This is good for making objects longer or shorter. It is not good for objects that have circular shapes that you want to maintain. For practice, you will scale the short prism shape and the long box.

Open the tutorial model

- From the Rhino **File** menu, click **Open**.
- Browse to the **Tutorial Models** folder that you downloaded with the *User's Guide*.
- Click **Rotate-Scale.3dm** and then click the **Open** button.

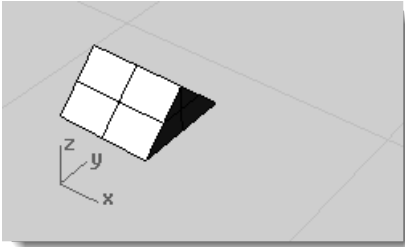


Review the Help topic

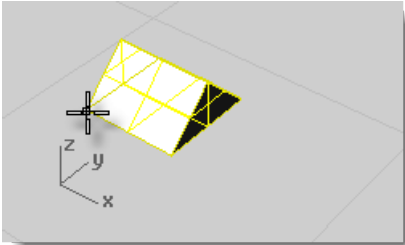
- ▶ Review the [Help](#) topic for the [Scale1D](#) command.

Scale the prism

1. From the **Transform** menu, click **Scale**, and then click **Scale 1-D**.
2. On the status bar, confirm that **Grid Snap** is **on**.
3. In the **Perspective** viewport, zoom in on the small prism.



4. Select the prism shape.
5. At the **Origin point...** prompt, click the corner of the prism as shown in the illustration below.

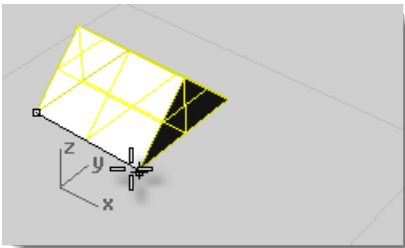


The origin point is the base point from which the object will be scaled. It is like an anchor point. The object will grow or shrink around this point.

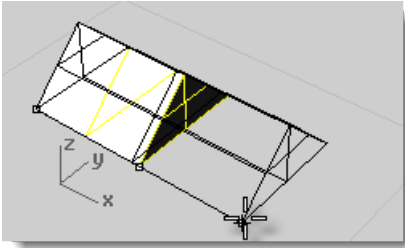
To scale an object, you must first show an original size and then show a new size. You are going to drag the cursor and click to show the original scale, and then drag the cursor and click again to show the new scale.

6. At the **Scale factor or first reference point...** prompt, click the corner of the prism as shown in the illustration below.

This establishes the first reference point.



7. At the **Second reference point...** prompt, drag the cursor.
The object grows in the direction you drag the cursor.
8. Click to set the second reference point.
Use Grid Snap to control the reference point placement.



9. From the **Edit** menu, click **Undo** to return the prism to its original size.

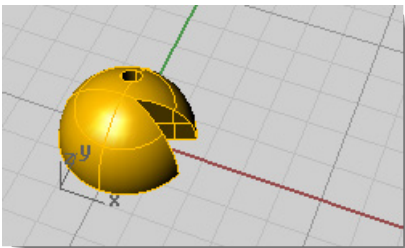
Mirror

The **Mirror** command reverses the orientation of the object across a defined line. By default, a copy is made.

You already know how to move, copy, scale, and rotate objects. In this exercise, you are going to practice another basic editing command: Mirror. The Mirror command makes a mirror image copy of the object. Objects are mirrored around a plane that is perpendicular to the active construction plane. You specify the planes direction with the mirror line.

Open the tutorial model

1. From the Rhino **File** menu, click **Open**.
2. Browse to the **Tutorial Models** folder that you downloaded with the *User's Guide*.
3. Click **Mirror Objects.3dm** and then click the **Open** button.

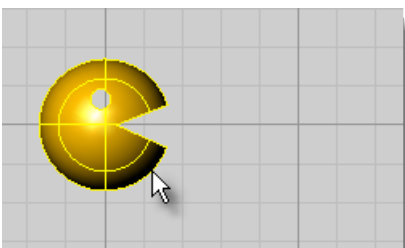


Review the Help topic

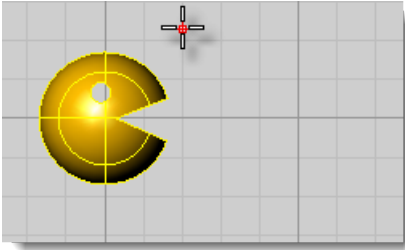
- ▶ Review the **Help** topic for the **Mirror** command.

Mirror an object

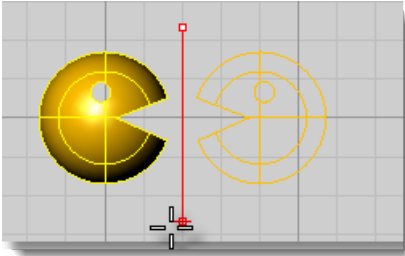
1. From the **Transform** menu, click **Mirror**.
2. On the status bar, turn **Grid Snap** and Ortho **on**.
3. **Select** the object.



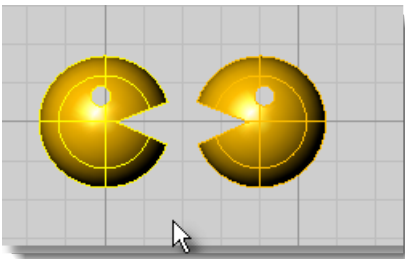
- At the **Start of mirror plane...** prompt, in the **Top** or **Front** viewport, click to the right of the face as shown in the illustration below.



- At the **End of mirror plane...** prompt, drag the line toward the bottom of the screen...



- ... and click to end the mirror plane.

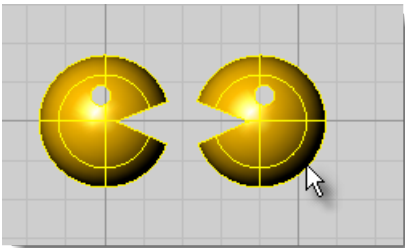


Mirror the objects in a different direction

This time when you mirror the objects, the heads will be spherical instead of half-spheres.

To mirror the objects

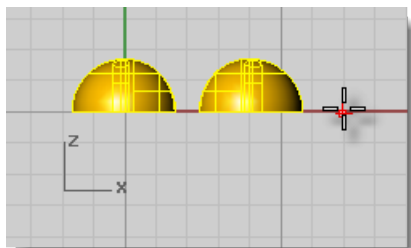
- Select** both objects.
Use a crossing or window.
- From the **Transform** menu, click **Mirror**.



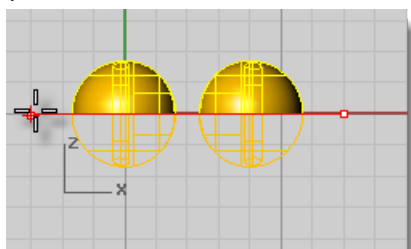
- At the **Start of mirror plane...** prompt, in the **Front** viewport, click on the red construction plane axis line.

Note: You must use the **Front** viewport.

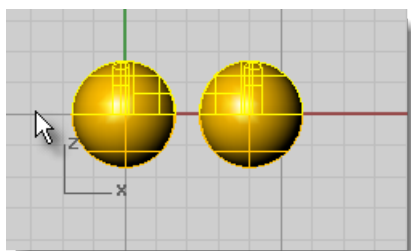
- Look at the status bar coordinates to be sure that the z-coordinate is 0.000.



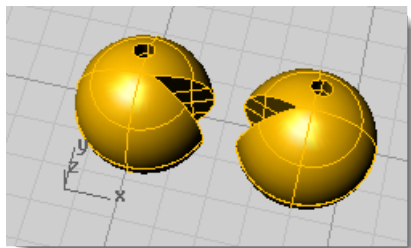
5. At the **End of mirror plane...** prompt, drag the line to the right and click to end the mirror plane.



The mirror makes the faces into two trimmed spheres.



6. Rotate the view in the **Perspective** viewport.



Orient

The **Orient** commands combine move or copy, scale, and rotate operations to help you position and size objects in one command.

See the Rhino **Help** topics for more information.

Array

The **Array** commands copy objects into evenly spaced rows and columns or around a circle.

See the Rhino **Help** topics for more information.

Chapter 9: Curve and Surface Analysis

Since Rhino is a mathematically accurate NURBS modeler, tools that provide accurate information about the objects are provided.

Measure distance, angle, and radius

Some analysis commands provide information about location, distance, angle between lines, and radius of a curve. For example:

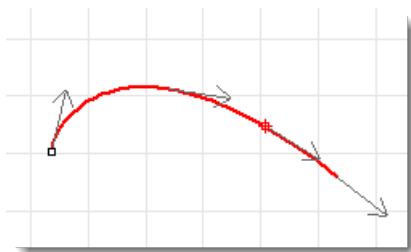
- **Distance** displays the distance between two points.
- **Angle** displays the angle between two lines.
- **Radius** displays the radius of a curve at any point along it.
- **Length** displays the length of a curve.

Curve and surface direction

Curves and surfaces have a *direction*. Many commands that use direction information display direction arrows and give you the opportunity to change (*flip*) the direction.

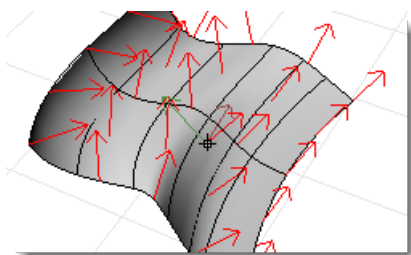
The **Dir** command displays the direction of a curve or surface and lets you change the direction.

The illustration shows the curve direction arrows. If the direction has not been changed, it reflects the direction the curve was originally drawn. The arrows point from the start of the curve toward the end of the curve.



The **Dir** command also displays surface u-, v-, and normal direction. Surface normals are represented by arrows perpendicular to the surface, and the u- and v-directions are indicated by arrows pointing along the surface. Closed surfaces always have the surface normals pointing to the exterior.

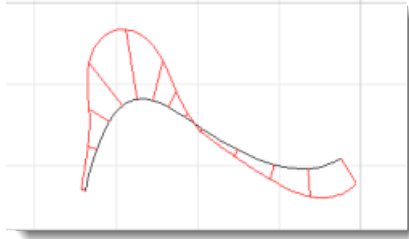
The **Dir** command can change the u-, v-, and normal-directions of a surface. This direction can be important if you are applying textures to the surface.



Curvature

Curve analysis tools let you turn on a graph showing the direction perpendicular to the curve at a point and the amount of curvature, display a curvature circle, test the continuity between two curves and the intervals of overlap between the two curves.

The **CurvatureGraph** command displays a curvature graph on curves and surfaces. The lines on the graph represent a direction perpendicular to the curve at that point. The length of the line indicates the curvature.



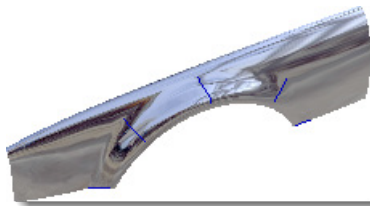
Visual surface analysis

Visual surface analysis commands let you examine surfaces to determine smoothness as determined by its curvature, tangency, or other surface properties. These commands use NURBS surface evaluation and rendering techniques to help you visually analyze surface smoothness with false color or reflection maps so you can see the curvature and breaks in the surface.

Environment map

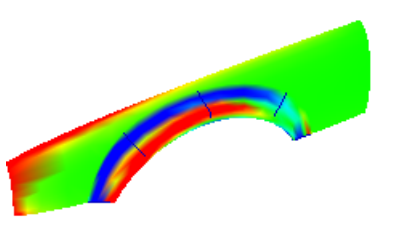
The **EMap** command displays a bitmap on the object so it looks like a scene is being reflected by a highly polished metal. Tool helps you find surface defects and validate your design intent.

The fluorescent tube environment map simulates tube lights shining on a reflective metal surface.



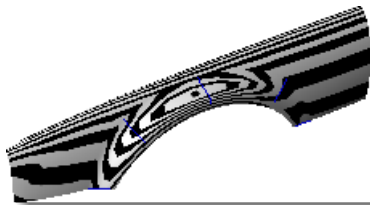
Curvature analysis

The **CurvatureAnalysis** command analyzes surface curvature using false-color mapping. It analyzes Gaussian curvature, mean curvature, minimum radius of curvature, and maximum radius of curvature.



Zebra analysis

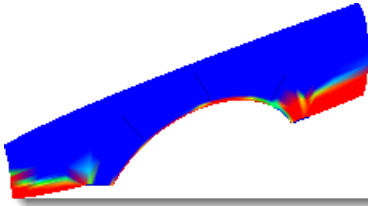
The **Zebra** command displays surfaces with reflected stripes. This is a way to visually check for surface defects and for tangency and curvature continuity conditions between surfaces.



Draft angle analysis

The **DraftAngleAnalysis** command displays by false-color mapping the draft angle relative to the construction plane that is active when you start the command.

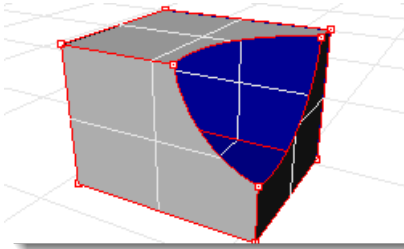
The pull direction for the **DraftAngleAnalysis** command is the z-axis of the construction plane.



Edge evaluation

Geometry problems such as Boolean or join failures can be caused by edges on surfaces that have become broken or edges between surfaces that have been moved through point editing so they create holes. An *edge* is a separate object that is part of the surface's boundary representation.

The **ShowEdges** command highlights all the edges of the surface.



Find open edges

A polysurface may look closed, but the **Properties** command may tell you that it is open. Some operations and export features require closed polysurfaces, and a model using closed polysurfaces is generally higher quality than one with small cracks and slivers.

Rhino provides a tool for finding the unjoined or "naked" edges. When a surface is not joined to another surface, it has naked edges. Use **Properties** command to examine the object details. A polysurface that has naked edges lists as an *open polysurface*. Use the **ShowEdges** command to display the unjoined edges.

Other edge tools let you split an edge, merge edges that meet end-to-end, or force surfaces with naked edges to join. You can rebuild edges based on internal tolerances. Other edge tools include:

- **SplitEdge** splits an edge at a point.
- **MergeEdge** merges edges that meet end to end.
- **JoinEdge** forces unjoined (naked) edges to join nearby surfaces.
- **RebuildEdges** redistributes edge control points based on internal tolerances.

Diagnostics

Diagnostic tools report on an object's internal data structure and select objects that may need repair. The output from the **List**, **Check**, **SelBadObjects**, and **Audit3dmFile** commands is normally most useful to a Rhino programmer to diagnose problems with surfaces that are causing errors.

Chapter 10: Organizing the Model

Rhino offers aids to organizing your work: layers, groups, blocks, and worksessions. Each method offers a different approach to model organization. Using layers lets you assign a layer designation to objects. Groups associate objects so they can be selected as one. Blocks let you store and update an association of objects. Worksessions let you work on a part of a project while using other models in the project as references.

Layers

Layers are a way of grouping objects and applying certain characteristics to all objects that have that layer assignment. There are two “mental models” you can use when you think of layers—layers can be thought of either as a “storage location” for the objects or as a way to assign a set of characteristics or properties to objects.

Layer states include a layer name, the color used to display the objects, and the on/off and locked/unlocked status of all the objects on a layer. Objects on layers that are off are not visible in the model. Objects on locked layers cannot be selected but can be snapped to. Objects are always created on the current layer. This layer assignment can be changed later.

To accomplish the most common tasks related to layers, click the Layer pane in the status bar to display the popup layer list. You can set the current layer; change the on/off, locked/unlocked state; and the layer color. In addition, right-click the layer name to create a new layer, rename a layer, delete the selected layer, select objects on the selected layer, change objects to the selected layer, and copy objects to the selected layer.

Accomplish more detailed layer management with the **Layers** panel. Right-click the **Layer pane** to open the **Layers** panel. The **Layers** panel sets the current layer, locks and unlocks layers, turns layers on and off, changes the layer color and sets the layer render material. You can create new layers, delete layers, move layers up or down in the layer list, filter the layer list, set the current layer to match an object in the model, change objects to a selected layer, select all layers, and invert the selection.

The **SelLayer** command selects all objects on a layer.

Groups

A *group* is a collection of objects that select as one for moving, copying, rotating, or other transforms and applying properties such as object color. Grouping objects assigns a group name to each object that is displayed as a part of its properties. Objects with the same group name belong to the same group.

- **Group** groups objects for selection. A group can contain one or more sub-groups.
- **Ungroup** destroys the group.
- **SetGroupName** changes the name assigned by default. Naming different groups to the same name combines those groups into one.
- **AddToGroup** and **RemoveFromGroup** add and remove objects from groups.
- **SelGroup** selects groups by name.

Blocks

A block is another way of associating objects together to form a single object. The **Block** command creates a block definition in the current model. The **Insert** command places *instances* of this block definition in your model. You can scale, copy, rotate, array, and otherwise transform block instances in the model. If you edit the block definition, all instances of the block are changed to this new definition. Blocks can streamline modeling, reduce model size, and promote standardization of parts and details.

Multiple instances of block can be located, scaled, and rotated into a model with the **Insert** command. Block definitions are created with the **Block** or **Insert** command. Materials and other object properties on block instances are determined by the component objects.

Exploding a block instance places the block geometry using the instance location, scale, and rotation. To redefine a block, use the **BlockEdit** command, to edit the geometry.

The **BlockManager** command displays a dialog box that lists all the block definitions in the model. Use the **Block Manager** dialog box to view block properties, export a block definition to a file, delete a block definition and all its instances, update a block definition from a file, find out what blocks are nested in other blocks, and count the number of block instances in the model.

Worksessions

The **Worksession** command lets many users work on a large project by managing many files. Each user can edit a different file in the project and at the same time see the related portions of the project. By refreshing as needed, each user can see the current version of the related files in the projects. Only one user can have a file open for editing, but many users can see it.

Rhino worksessions let you “attach” external files to your current work environment. Attached geometry cannot be edited (move, scale), but it can be used for input to creation commands (copy, extrude).

Chapter 11: Annotation

Rhino provides the ability to add notation to your model in the form of dimensions, leaders, and text blocks. These appear as objects in the model. A different form of notation, the annotation dots and arrowheads, always display facing towards the view plane.

In addition, you can add notes to the model. Notes do not appear in the model, but display in a separate window.

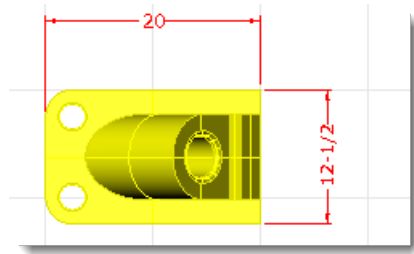
Dimensions

You can dimension objects in your model, with your choice of font, units display, decimal precision, text and arrow size, and text alignment. After dimensions are placed, you can select all dimensions, edit dimension text, turn control points on to move dimension elements, and delete dimensions. You can place horizontal, vertical, aligned, rotated, radial, diameter, and angle, text blocks, leaders, and create a 2-D hidden line drawing.

Dimensions are not associative. Changing your geometry will not update the dimension unless the dimension was drawn with history enabled. Changing the dimension will not update your geometry.

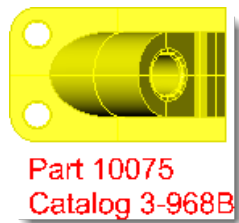
The **Dim** command places horizontal and vertical dimensions depending on the direction you pick the points.

Dimensions are created using the current dimension style. Create new dimension styles to control text size and font, and other dimension properties. Use the settings in the **Document Properties** window to create new styles and set the properties of existing styles.



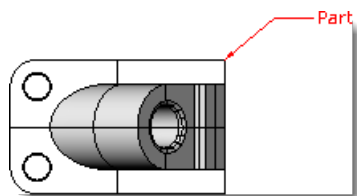
Text

The **Text** command places annotation text in your model.



Leaders

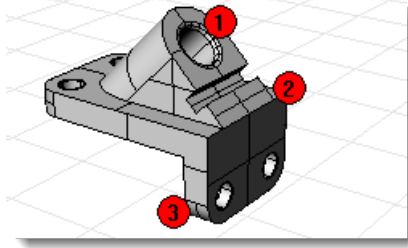
The **Leader** command draws an arrow leader.



Dots

The **Dot** command places a text dot.

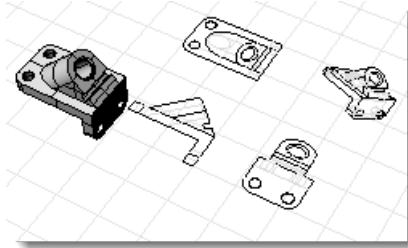
Dots are always parallel to the view. There are no controls for the dot size. Dots are displayed in the layer color. Dot size is constant on the screen. As you zoom in and out, the dot displays the same size.



Hidden line removal

The **Make2D** command creates curves from the selected objects as silhouettes relative to the active view. The silhouette curves are projected flat and then placed on the world x,y-plane.

The command options create the 2-D drawing from the current view, current construction plane, create a four-view layout using US or European projection angles, set layers for the hidden lines, and display tangent edges.



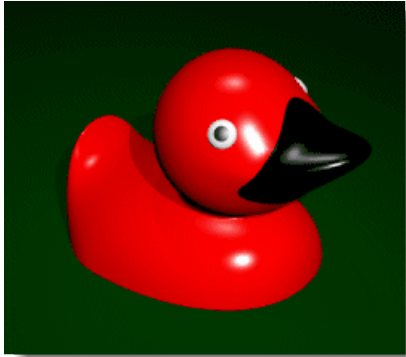
Notes

The **Notes** command provides a means of storing text information in your model file. You can type information directly into the **Notes** text box. If you leave the **Notes** box displayed when you close the model file, it will display the next time the file is opened.

Chapter 12: Render

In addition to shaded previews, Rhino provides full-color rendering with lights, transparency, shadows, textures, and bump mapping. If you want to create photo-realistic renderings, use a full-featured rendering program, such as Flamingo.

Objects will render white until you add render color, highlight, texture, transparency, and bumps. These attributes are controlled through the **Properties** panel, **Material** page.



The process needed to render scenes consists of four basic steps:

- Add lighting
- Assign materials
- Set up the environment and ground plane
- Render

Although the steps do not have to be done in this order, using this method seems to make setting up a scene more efficient. To improve quality, repeat these steps until the image looks right to you.

Lights

In every Rhino rendering there are light sources that Rhino uses to calculate how the objects are to be illuminated. The **Spotlight**, **DirectionalLight**, **LinearLight**, **PointLight**, **RectangularLight**, and **Sun** commands let you put lights into your model. If you do not add any light sources to your scene, the default light is used. The default light is a directional light with parallel rays that acts as though you have a lamp shining over your left shoulder.

Materials

Materials specify the color, finish, transparency, texture, and bump for use by the built-in Rhino renderer.

Assign materials to layers

1. In the **Layers** panel, select one or more layer names, and click in the **Material** column.
2. In the **Layer Material** dialog box, set up material properties.

Assign materials to objects

1. Select an object.
2. On the **Edit** menu, click **Object Properties**.
3. In the **Properties** panel, **Material** page, set up material properties.

Environment

Environments describe the space around the model. This space is reflected in objects and can provide overall lighting.

Set up the rendering environment

The rendering properties include environment settings like background color and image.

1. Open the **Environment Editor** panel.
2. Set up environment properties.

Ground Plane

The ground plane provides an infinite horizontal platform for the image that stretches to the horizon in all directions positioned at a defined elevation. A ground plane renders much faster than using a surface as a background. Any material can be assigned to the ground plane.

Turn on a ground plane

1. Open the **Ground Plane** panel.
2. Set up ground plane properties.

Render and save

1. On the **Render** menu, click **Render**.
2. In the **Render Window**, on the **File** menu, click **Save As**.

Rhinoceros 5

User's Guide

Section 2: Tutorials



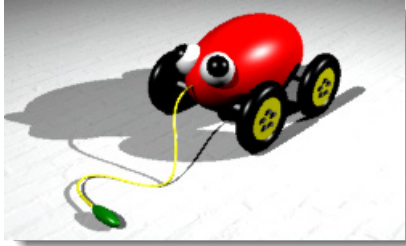
© Robert McNeel & Associates. 2012-08-16.

Chapter 13: Pull Toy Tutorial - Solids and Transforms

This tutorial demonstrates using solid primitives and simple transforms.

You will learn how to:

- Enter coordinates to place points exactly.
- Draw a free-form curve and polygon.
- Create a pipe along a curve.
- Use a polar array to copy objects in a circular pattern.
- Extrude a curve to create a surface.
- Use planar mode.



Enter coordinates

When you pick a point with the mouse, the point lies on the *construction plane* of the active viewport unless you use a modeling aid such as object snap or *elevator mode*. When Rhino prompts for a point, you can enter x-, y-, and z-coordinates instead of picking a point. Each viewport has its own construction plane on which its x- and y-coordinates lie. The z-coordinate for the active viewport is perpendicular to the x-y plane.

The grid is a visual representation of the construction plane. The intersection of the dark red and green lines shows the location of the origin point ($x=0$, $y=0$, $z=0$) of the coordinate system.

Draw the pull toy body

This exercise uses x-, y-, and z-coordinates to accurately place points. When you are to type coordinates, type them just as they are shown in the manual. The format is **x,y,z**. For example, type **1,1,4**. You must type the commas. This sets the point at $x=1$, $y=1$, and $z=4$ in the active viewport.

Whenever you type points, look in all viewports at where the point is placed so you can start getting an idea of how coordinate entry works.

Note: Pay close attention to the viewport required in each instruction.



Start the model

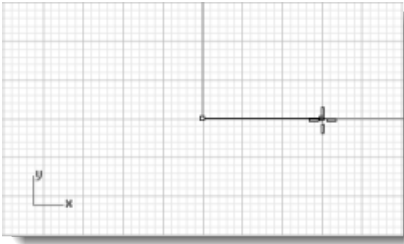
1. Begin a **New** model.
2. In the **Template File** dialog box, select **Small Objects - Centimeters.3dm**, and click **Open**.



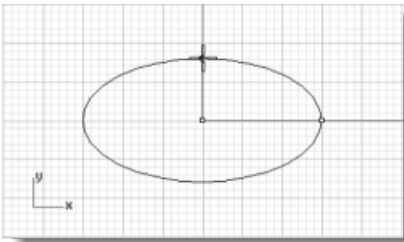
Draw an ellipsoid

1. Turn on **Ortho**.
2. On the **Solid** menu, click **Ellipsoid > From Center**.
3. With the **Top** viewport active, at the **Ellipsoid center...** prompt, type **0,0,11**, and press **Enter**.
This places the center point of the ellipsoid at $x=0$, $y=0$, and $z=11$. Look at the point in the perspective viewport.
4. At the **End of first axis...** prompt, type **15**, and press **Enter**.

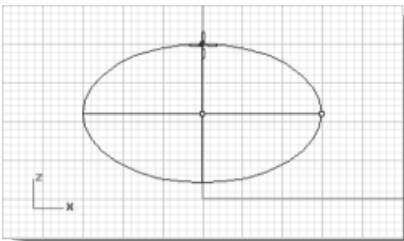
5. Move the cursor to the right to show the direction and click.



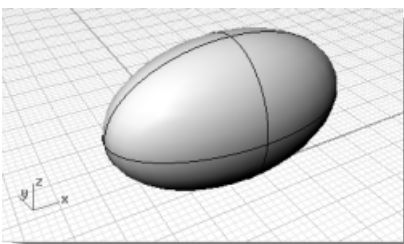
6. At the **End of second axis** prompt, type **8**, and press **Enter**.
 7. Move the cursor up to show the direction and click.
 This sets the width of the ellipsoid.



8. At the **End of third axis** prompt, type **9**, and press **Enter**.
 You now have an egg shape that has different dimensions in all three directions.



9. Rotate the perspective viewport so you are looking along the x-axis as illustrated.
 Turn on **Shaded** display mode in the **Perspective** viewport.



Draw the axles and wheel hubs

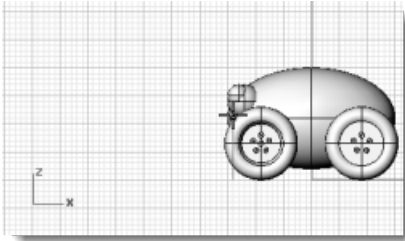
The axles and wheel hubs are cylinders. The axles are long, thin cylinders, and the wheel hubs are short, fat cylinders. You are going to make one axle and one complete wheel. You will then mirror the complete wheel to the other side. You can then either mirror or copy the complete axle and wheel set to the front of the toy.



Create the axle

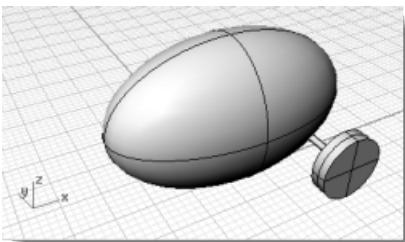
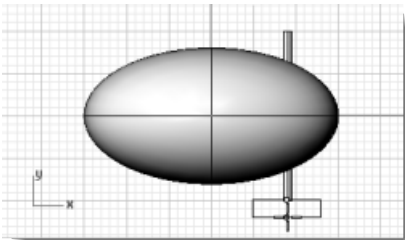
1. On the **Solid** menu, click **Cylinder**.

2. With the **Front** viewport active, at the **Base of cylinder...** prompt, type **9,6.5,10**, and press **Enter**.
3. At the **Radius...** prompt, type **.5**, and press **Enter**.
4. At the **End of cylinder** prompt, type **-20**, and press **Enter**.



Create a wheel hub

1. On the **Solid** menu, click **Cylinder**.
2. With the **Front** viewport active, at the **Base of cylinder...** prompt, type **9,6.5,10**, and press **Enter**.
3. At the **Radius...** prompt, type **4**, and press **Enter**.
4. At the **End of cylinder** prompt, type **2**, and press **Enter**.



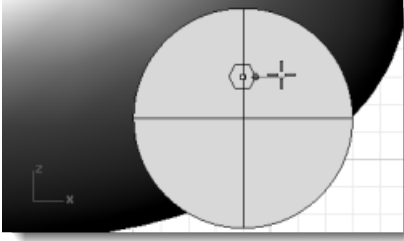
Draw the lug nuts

You will make the lug nuts by extruding a hexagonal polygon curve.



Create a hexagon

1. On the **Curve** menu, click **Polygon > Center, Radius**.
2. At the **Center of inscribed polygon (NumSides=4...)** prompt, type **6**, and press **Enter**.
3. In the **Front** viewport, at the **Center of inscribed polygon...** prompt, type **9,8,12**, and press **Enter**.
This will place the polygon right on the surface of the wheel hub.
4. At the **Corner of polygon...** prompt, type **.5**, and press **Enter**.
5. In the **Front** viewport drag the cursor as illustrated, and click to position the hexagon.



Make a solid from the polygon

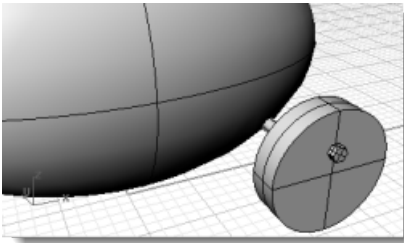
1. **Select** the hexagon you just created.
2. On the **Solid** menu, click **Extrude Planar Curve > Straight**.
3. At the **Extrusion distance (Direction BothSides=No Solid=Yes DeleteInput=Yes ToBoundary SplitAtTangents=No SetBasePoint)** prompt, notice the command-line options.

Many commands have options. You will learn how to change and use them as you learn to use the commands. Take a moment and look at the options available for the **ExtrudeCrv** command.

Press **F1** to look at the Help topic for this command. The Help topic explains the options.

4. At the **Extrusion Distance...** prompt, type **-.5**, and press **Enter**.

Notice the negative number. If you type a positive number at this point, the nuts will be buried in the wheel hub. You want them to stick out.



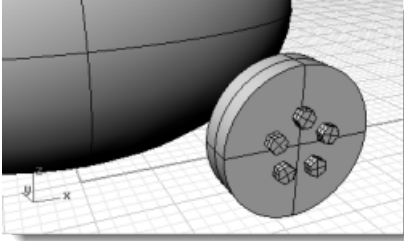
Array the lug nuts

To create the lug nuts on the first wheel, you are going to use a polar (circular) array. An array is a set of copies of an object. You control how the copies are made. A polar array copies the objects around a central point. The objects are rotated as they are copied.



Array the nuts

1. **Select** the lug nut.
2. On the **Transform** menu, click **Array > Polar**.
The hexagon curve is still there, so be sure you select the extruded lug nut. (The selection menu will list it as a *closed extrusion*.)
3. With the **Front** viewport active, at the **Center of polar array** prompt, use the **Cen** object snap to snap to the center of the hub.
4. At the **Number of elements...** prompt, type **5**, and press **Enter**.
5. At the **Angle to fill <360>** prompt, press **Enter**.



Draw the tires

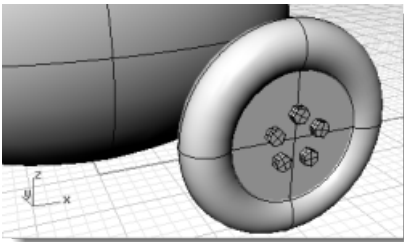
The tires are a solid form called a torus, which looks like a donut. When you are drawing a torus, the first radius is the radius of a circle around which the “tube” is drawn. The second radius is the radius of the tube itself.

To draw the tires, you will draw the center of the torus tube a bit larger than the diameter of the wheel hub. The tube itself is slightly larger than the hub. This makes it dip into the hub.



Create a torus for the tires

1. On the **Solid** menu, click **Torus**.
2. In the **Front** viewport, at the **Center of torus...** prompt, type **9,6.5,11**, and press **Enter**.
This places the center of the torus at the same point as the center of the wheel hub.
3. At the **Radius...** prompt, type **5**, and press **Enter**.
This makes the center of the torus tube one unit larger than the wheel hub.
4. At the **Second radius...** prompt, type **1.5**, and press **Enter**.
This makes the hole .5 units smaller than the wheel hub.



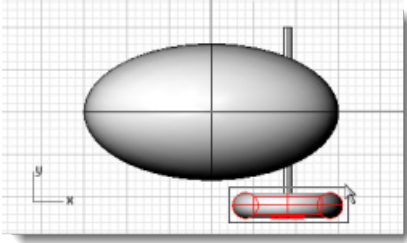
Mirror the wheels

Now that you have a whole wheel created, you can use the **Mirror** command to create the other three.

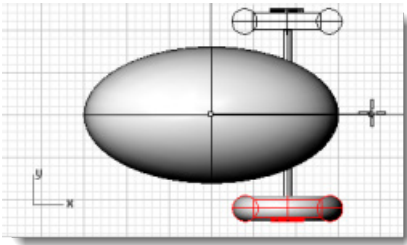


Mirror the wheel to the other side

1. In the **Top** viewport, use a window to select the wheel as illustrated.
2. On the **Transform** menu, click **Mirror**.
3. At the **Start of mirror plane...** prompt, type **0,0,0**, and press **Enter**.

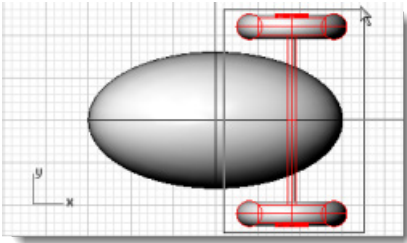


4. At the **End of mirror plane...** prompt, with **Ortho** on, drag to the right in the **Top** viewport as illustrated and click.

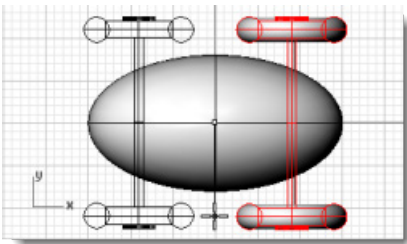


Mirror the front wheels and axle

1. In the **Top** viewport, use a window to select the wheels and axle as illustrated.
2. On the **Transform** menu, click **Mirror**.
3. At the **Start of mirror plane...** prompt, type **0,0,0**, and press **Enter**.



4. At the **End of mirror plane...** prompt, with **Ortho** on, drag down in the **Top** viewport as illustrated and click.



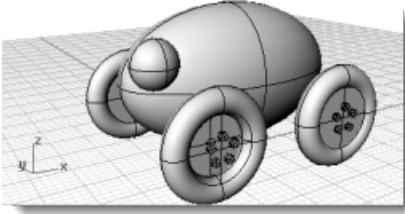
Draw the eyes

You are going to draw a sphere for an eye and a smaller sphere for the pupil.

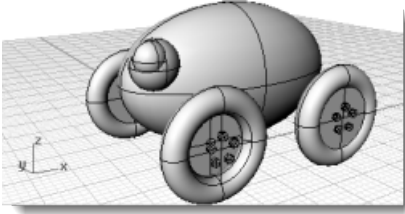


Create an eye using a sphere

1. On the **Solid** menu, click **Sphere > Center, Radius**.
2. At the **Center of sphere...** prompt, in the **Top** viewport, type **-12,-3,14**, and press **Enter**.
3. At the **Radius...** prompt, type **3** and press **Enter**.

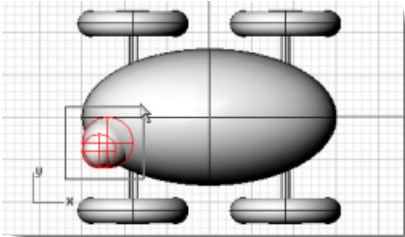


4. Repeat the **Sphere** command.
5. At the **Center of sphere...** prompt, in the **Top** viewport, type **-13,-4,15**, and press **Enter**.
6. At the **Radius...** prompt, type **2** and press **Enter**.
7. Change the color of the pupil to black.

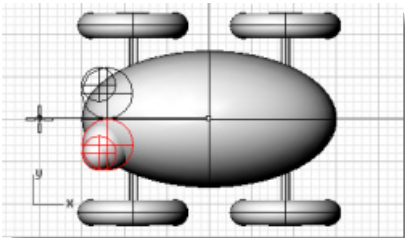


Mirror the eye

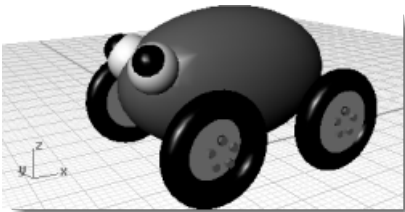
1. In the **Top** viewport, use a window to select the eye as illustrated.
2. On the **Transform** menu, click **Mirror**.



3. At the **Start of mirror plane...** prompt, type **0,0,0**, and press **Enter**.
4. At the **End of mirror plane...** prompt, with **Ortho** on, drag to the left in the **Top** viewport as illustrated and click.



5. Right-click the **Perspective** viewport title.
6. On the **Viewport title** menu, click **Rendered**.



Make the pull cord

To make the cord, you are going to draw a freehand curve using elevator and planar mode. When the curve is complete, use the **Pipe** command to make it a thick solid.

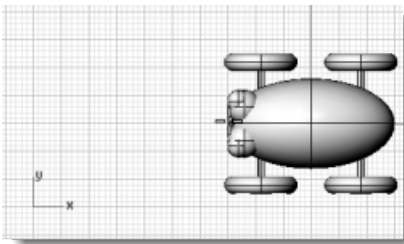
Set up the view

1. **Zoom** out in all the viewports; you are going to need some space to work.
2. On the status bar, turn **Planar** mode on, and turn **Ortho** off.
3. In the **Osnap** dialog box, click **Disable** to turn off all object snaps.

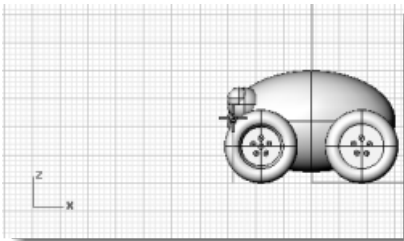


Create the pull cord at the front of the toy

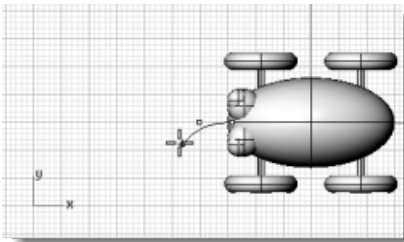
1. On the **Curve** menu, click **Free-form > Control Points**.
2. At the **Start of curve...** prompt, in the **Top** viewport, hold **Ctrl** to activate elevator mode and click near the front end of the ellipsoid.



3. Move the cursor to the **Front** viewport, drag the marker near the end of the ellipsoid, and click.

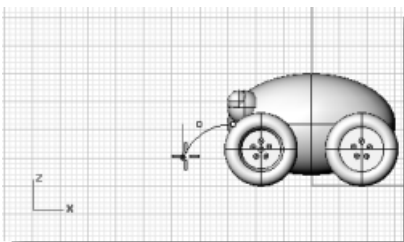


4. At the **Next point...** prompt, click to the left of the ellipsoid in the **Top** viewport.

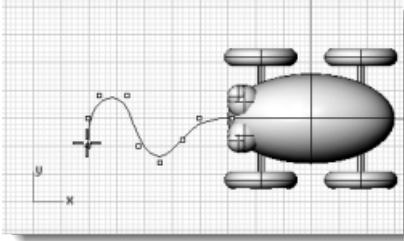


Planar mode keeps successive points at the same construction plane elevation. Planar mode can be overridden with elevator mode or object snaps. Watch the curve in the **Top** and **Front** viewports.

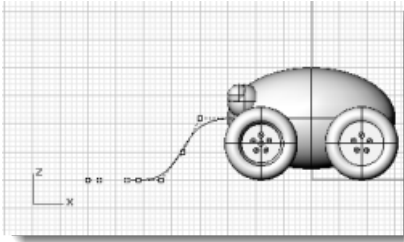
5. At the **Next point...** prompt, use elevator mode to add another point in the **Top** viewport.



6. At the **Next point...** prompts, turn off **Planar** mode and click several more points in the **Top** viewport to create a curved line.

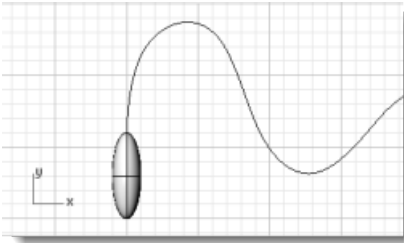


Notice that the points are projected to the **Top** construction plane.



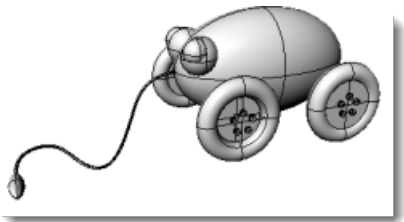
Make the cord handle

1. Draw an **Ellipsoid** to represent a handle at the end of the curve.
2. In the **Osnap** dialog box, clear the **Disable** checkbox and use the **End** object snap to snap the ellipsoid to the end of the curve.



Thicken the curve with a pipe

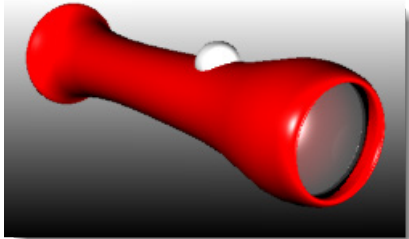
1. **Select** the curve you just made at the front of the pull toy.
 2. On the **Solid** menu, click **Pipe**.
 3. At the **Start radius...** prompt, type **.2**, and press **Enter**.
 4. At the **End radius...** prompt, press **Enter**.
 5. At the **Point for next radius** prompt, press **Enter**.
- The pipe will be the same diameter for the full length of the curve.



Chapter 14: Flashlight Tutorial - Revolve Curves

Drawing objects using solid primitives, as you have done in the previous exercises, limits the shapes you can create. Creating surfaces from curves and joining the surfaces together allows you much greater freedom.

This tutorial introduces the concept of drawing curves and one method of creating surfaces from those curves. This exercise creates a revolved surface from a profile curve. Revolving curves is a good method for creating tubular shapes like vases, wineglasses, and chair legs.



You will learn how to:

- Draw free-form curves based on an existing object.
- Edit control points.
- Revolve surfaces around an axis.
- Assign properties and render.

You are going to use a basic flashlight as a guide for drawing the curves you will need for the new model. Using the basic flashlight gives you a frame of reference for deciding about the size and shape of the object. If you did not work through this tutorial or save your model, a completed model is provided.



To get started

1. On the Rhino **File** menu, click **Open**.
2. Browse to the **Tutorial Models** folder that you downloaded with the *User's Guide*.
3. Open the model file **Flashlight.3dm**.

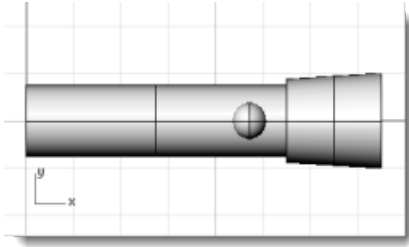
Set up the model

You are going to trace around the old flashlight. To make this easier, you will lock the objects. When objects are locked, you can see them and snap to them, but you cannot select them. This keeps the objects from interfering when you want to select things close by. You can still use object snaps to snap to locked objects. You will then create some curves and revolve them to make the new flashlight.



Lock the flashlight objects

1. **Select** all the objects.
Press **Ctrl + A** to select all the objects in the model.
2. On the **Edit** menu, click **Visibility > Lock**.



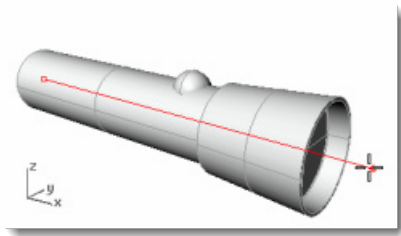
Draw a centerline

Draw a construction centerline through the center of the old flashlight.



Draw the construction centerline

1. On the **Curve** menu click **Line > Single Line**.
2. At the **Start of line...** prompt, use the **Center** object snap to place the start of the line at the center of the flashlight base.
3. At the **End of line...** prompt, turn **Ortho** on, and draw the line through the exact center of the old flashlight.



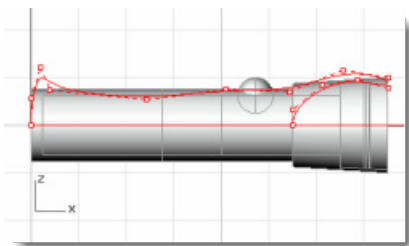
Draw the body profile curve

You are going to draw a *profile curve* that you will use to revolve to create the flashlight body. A profile curve defines a cross-section of one half of the part.



Draw the body curve

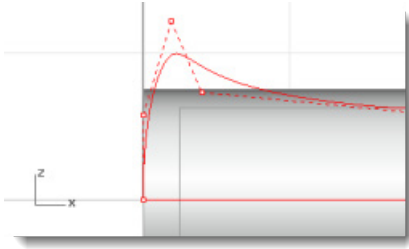
1. On the status bar, click the **Layer** pane and make the layer **Free Form Body** current.
2. On the **Curve** menu, click **Free-Form > Control Points**.
3. At the **Start of curve...** prompt, in the **Front** viewport, start drawing a curve around the flashlight body as illustrated.



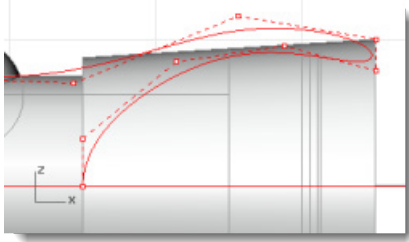
Use the **End** object snap to start the curve at the end of the construction center line.

Use the **Near** object snap to end the curve on the construction center line.

Starting and ending the curve exactly on the line is important so that later when you revolve the curve to create a solid, there will be no gaps or overlapping parts.



When drawing the curve, use **Ortho** to control the first two points on the curve. If the first two points and the last two points are placed in a straight line, the curve will start and end tangent to that line.



4. When you have placed the last control point, press **Enter** to finish drawing the curve.
To place the last two points in a straight line with each other, use **Grid Snap**, **Ortho**, or **Perp** object snap.

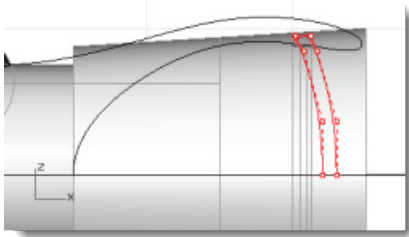
Draw the lens profile curve

Make another profile curve for the lens.



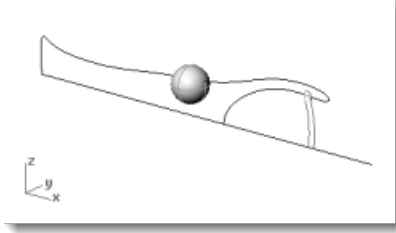
Create the lens

1. On the **Curve** menu, click **Free-Form > Control Points**.
2. At the **Start of curve...** prompt, in the **Front** viewport, place the first control point of the lens profile.
Use the **Near** object snap to start and end the curve on the construction centerline.
Place control points in the upper part of the lens curve so it crosses the body profile curve.



Get the old flashlight out of your way

1. On the **Edit** menu, click **Visibility > Unlock**.
2. Select all the objects except the two profile curves you just drew and the switch sphere.
3. On the **Edit** menu, click **Visibility > Hide**.



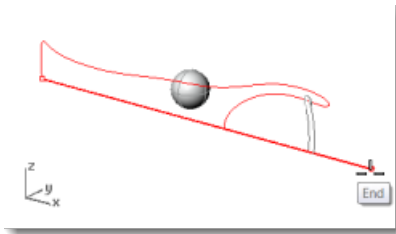
Build the flashlight body

To make the body, you will revolve the profile curve 360 degrees. You will use the endpoint of the curve and ortho to establish the rotation axis.

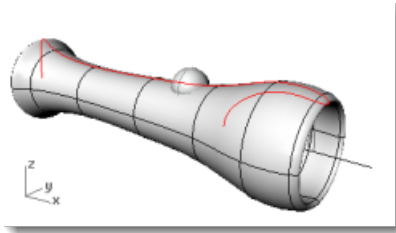


Create the flashlight body

1. On the **Surface** menu, click **Revolve**.
2. At the **Select curve to revolve** prompt, select the body profile curve.
3. At the **Start of revolve axis** prompt, snap to one endpoint of the body curve.
4. At the **End of revolve axis** prompt, turn **Ortho** on, and specify the revolve axis line as illustrated.



5. In the **Start angle...** prompt, click the **FullCircle** option.



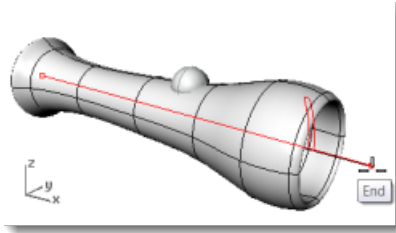
Create the lens

Now revolve the lens profile curve in the same way as the body.

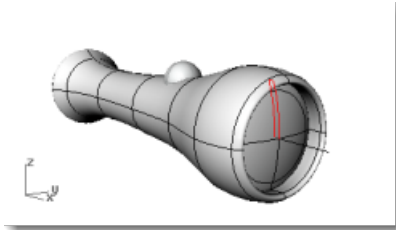


Revolve the lens profile curve

1. On the **Surface** menu, click **Revolve**.
2. At the **Select curve to revolve** prompt, select the lens profile curve.
3. At the **Start of revolve axis** prompt, use **End** object snap to locate the endpoint of one of the curve profiles.
4. At the **End of revolve axis** prompt, turn **Ortho** on, and draw the revolve axis line as illustrated.



5. In the **Start angle...** prompt, click the **FullCircle** option.



Chapter 15: Headphone Tutorial - Sweep, Loft, and Extrude

This tutorial demonstrates creating surfaces from profile curves using lofts, sweeps, and extrudes.

You will learn how to:

- Create a surface from a planar curve.
- Loft, revolve, sweep, and extrude surfaces.
- Cap planar holes to create a solid.
- Create solid pipes.
- Mirror objects.
- Use layers.
- Use object snaps.



To open the headphone model

1. On the Rhino **File** menu, click **Open**.
2. Browse to the **Tutorial Models** folder that you downloaded with the *User's Guide*.
3. Open the model file **Headphone.3dm**.

Create the speaker shell

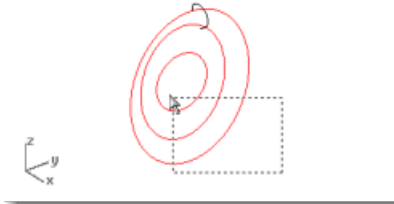
The speaker shell is created using a lofted surface, a one-rail sweep, a solid extrusion of a planar curve, and a surface fillet. The resulting geometry is joined into one solid.



Loft curves to create a surface

One way to create a surface is to use existing curves as a guide. When lofting through curves, the curves are used as a guide for creating a smooth surface.

1. Turn on **Shaded** mode in the **Perspective** viewport.
2. **Select** the three circular curves, with a crossing selection as illustrated.

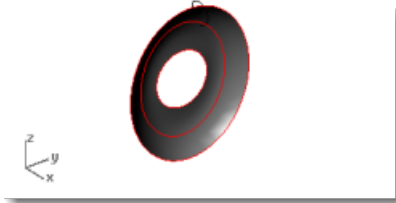


3. On the **Surface** menu, click **Loft**.
4. At the **Adjust curve seams...** prompt, note the display of the curve direction arrows at the seam points, and press **Enter**

In this model, they are nicely lined up for you, so you do not need to adjust them.



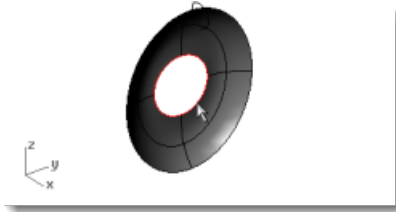
5. In the **Loft Options** dialog box, click **OK** to create the loft.



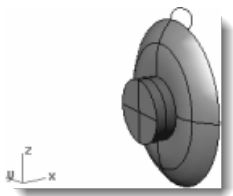
Make a solid cylinder by extruding a circular curve

Extrude the curve in the center to make a magnet housing.

1. **Select** the curve at the center of the lofted surface.
2. On the **Solid** menu, click **Extrude Planar Curve > Straight**.



3. At the **Extrusion Distance...** prompt, type **-2** and press **Enter**.
This makes a solid cylinder for the magnet housing that is two units thick and extends in the negative direction from the original curve.



Reset the view

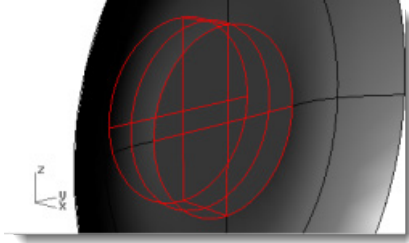
1. **Select** the cylinder.
2. On the **View** menu, click **Zoom > Zoom Selected**.



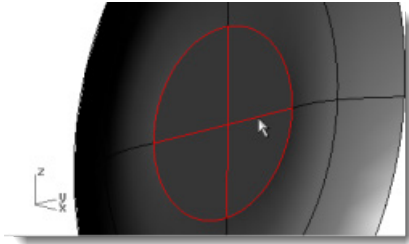
Extract the bottom surface

The cylinder you just created is a closed polysurface (solid) consisting of three joined surfaces—the side, top, and bottom. To remove the bottom, extract the face.

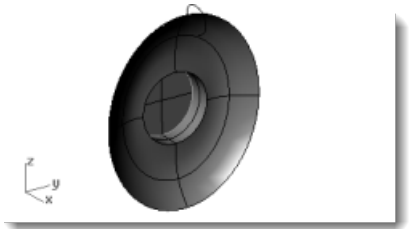
1. From the **Solid** menu, click **Extract Surface**.



2. At the **Select surfaces to extract...** prompt, select the surface as illustrated and press **Enter**.

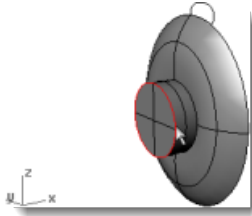


3. Press the **Delete** key.

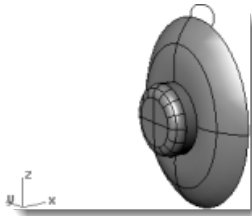


 **Fillet the edge of the cylinder surface**

1. On the **Solid** menu, click **Fillet Edge > Fillet Edge**.
The current radius setting should be **1**.
2. At the **Select edges to fillet...** prompt, select the edge at the top of the cylinder press **Enter**.



3. At the **Select fillet handle to edit** prompt, press **Enter**.



 **Join the surfaces**

Surfaces that share an edge can be joined into a polysurface. You will join all the surfaces. Since the faces are sometimes hard to see, use two viewports to select them all.

1. **Select** the surface and the polysurface.
2. From the **Edit** menu, click **Join**.

To join surfaces, you must select surfaces that are adjacent to each other and the edges must match.



Create the padding and cover

Sweep a curve around the edge of the speaker cone to create the padding around the edge of the speaker.



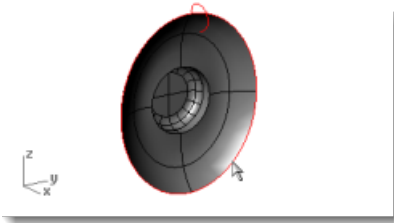
Reset the view

- ▶ On the **View** menu, click **Zoom > Zoom Extents All**.

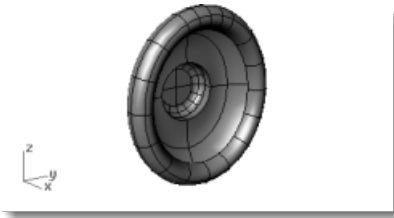


Sweep a curve along one rail

1. **Select** the curves as illustrated.
2. From the **Surface** menu, click **Sweep 1 Rail**.



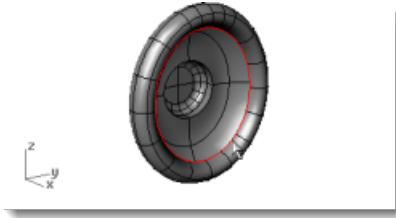
3. In the **Sweep 1 Rail Options** dialog box, click **OK**.



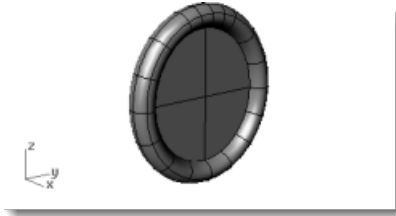
Make a surface from planar curves

Fill the area at the base of the padding with a planar surface created from the edge of the sweep.

1. On the **Surface** menu, click **From Planar Curves**.
2. **Select** the curve on the edge of the speaker cone as illustrated.



A planar surface is created at the base of the padding.



Create the mounting bracket

The next part is the bracket that holds the speaker to the headband. Since the speaker unit is complete, you can turn its layer off and make the Bracket layer current.



Reset the layers

1. On the status bar, click the **Layer** pane.
2. Make **Bracket** the current layer and turn on **Bracket Shape Curves**. Turn all other layers off.



Reset the view

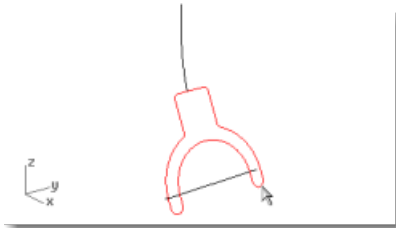
- ▶ From the **View** menu, click **Zoom > Zoom Extents All** to zoom in on the bracket shape curves in all viewports.



Extrude a curve into a solid

Use a planar curve to create a solid shape.

1. **Select** the closed curve.
2. On the **Solid** menu, click **Extrude Planar Curve > Straight**.



3. At the **Extrusion distance...** prompt, type **-1** and press **Enter**.



Fillet the edges

Round the sharp edges with a fillet.

1. On the **Solid** menu, click **Fillet Edge > Fillet Edge**.
2. At the **Select edges to fillet...** prompt, type **.2** and press **Enter**.
3. At the **Select edges to fillet...** prompt, select both edges and press **Enter**.



4. At the **Select fillet handle to edit** prompt, press **Enter**.



Create a tubular surfaces from the shape curves

1. **Select** the curve at the top of the bracket.
2. On the **Solid** menu, click **Pipe**.
3. At the **Starting radius...** prompt, type **.3** and press **Enter**.
Before typing the radius, set the command-line options to **Cap=Flat** and **Thick=No**.
4. At the **End radius...** prompt, press **Enter**.
5. At the **Point for next radius** prompt, press **Enter**.



For the second tube

1. **Select** the curve at the bottom of the bracket.

2. On the **Solid** menu, click **Pipe**.



3. At the **Starting radius...** prompt, type **.2**, and press **Enter**.
4. At the **End radius...** prompt, press **Enter**.
5. At the **Point for next radius** prompt, press **Enter**.



Create the headband

The headband consists of a series of ellipses swept along a path.



Reset the layers

1. On the status bar, click the **Layer** pane.
2. Make **Headband** the current layer and turn on **Headband Shape Curves**. Turn all other layers off.



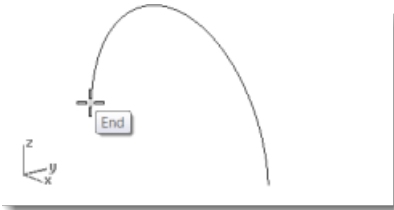
Reset the view

- ▶ From the **View** menu, click **Zoom > Zoom Extents All** to zoom in on the bracket shape curves in all viewports.

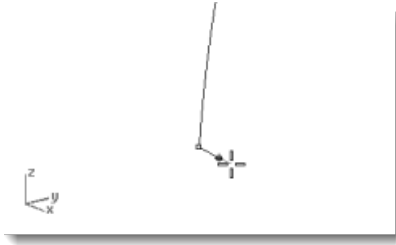


Create an ellipse perpendicular to a curve

1. Turn **Ortho** on.
2. On the **Curve** menu, click **Ellipse > From Center**.
3. At the **Ellipse center...** prompt, click **AroundCurve**.



4. At the **Ellipse center** prompt, snap to an endpoint of the headband curve. Use the **End** object snap.
5. At the **End of first axis** prompt, type **0.5**, and press **Enter**.
6. At the **End of first axis** prompt, drag the cursor in the x-direction and click.

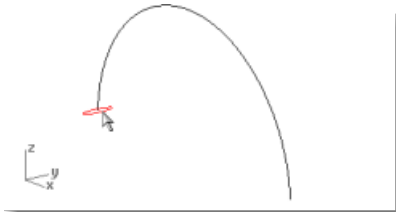


7. At the **End of second axis** prompt, type **2**, and press **Enter**.
8. At the **End of second axis** prompt, drag the cursor in the y-direction and click.

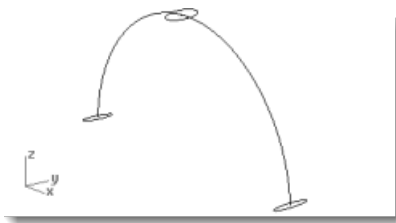


Array a curve along a path

1. **Select** the ellipse.
2. On the **Transform** menu, click **Array > Along Curve**.
3. At the **Select path curve** prompt, select headband curve.



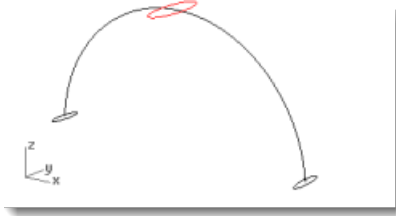
4. In the **Array Along Curve Options** dialog box, under **Method**, set the **Number of items** to **3**.
5. Under **Orientation**, click **Freeform**, and click **OK**.



Scale the ellipse

Scale the center ellipse to make it larger.

1. **Select** the center ellipse.
2. On the **Transform** menu, click **Scale > Scale1-D**.
Scale1D stretches an object in one direction.
3. At the **Origin point...** prompt, in the **Perspective** viewport, snap to the center of selected ellipse.
4. At the **Scale factor or first reference point...** prompt, type **2**, and press **Enter**.



5. At the **Second reference point...** prompt, drag the cursor in the y-direction and click.



Reset the view

- ▶ On the **View** menu, click **Zoom > Zoom Extents All**.



1 Sweep along one rail

1. **Select** the curves.
2. From the **Surface** menu, click **Sweep 1 Rail**.



3. At the **Adjust curve seams...** prompt, examine the direction and seam points of the curves to make sure they are not twisted, and press **Enter**.
4. In the **Sweep 1 Rail Options** dialog box, click **OK**.



Round the headband ends

Use the same ellipse that formed the first cross-section curve for the headband to create a rounded end for the headband. Start by splitting the ellipse in half.



Reset the view

1. On the **View** menu, click **Zoom > Window**.
2. In the **Perspective** viewport, zoom in on the left end of the headband you just created.

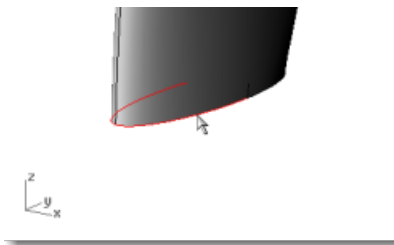


Split the ellipse in half

1. **Select** the ellipse.
2. From the **Edit** menu, click **Split**.

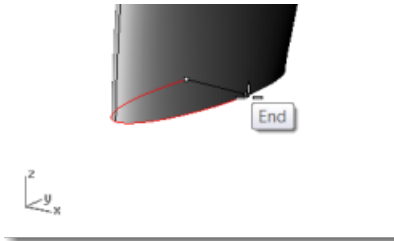


3. At the **Select cutting objects...** prompt, type **P**, and press **Enter**.
4. Turn on **Quad** object snap.
5. At the **Point to split curve** prompts, snap to the two quadrants at the narrow axis of the ellipse.
6. At the **Point to split curve** prompt, press **Enter**.
The ellipse is split into two halves.

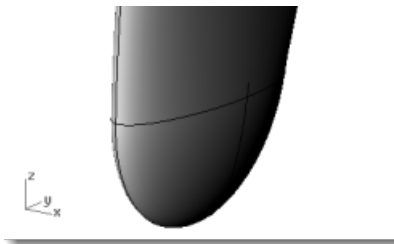


Create a surface of revolution

1. **Select** left half of the ellipse.
2. On the **Surface** menu, click **Revolve**.



3. At the **Start of revolve axis** prompt, snap to the end of the ellipse half.
4. At the **End of revolve axis** prompt, snap to the other end of the ellipse half.
5. At the **Start angle...** prompt, type **0**, press **Enter**.
6. At the **Revolution angle...** prompt, type **180**, press **Enter**.
A rounded surface is created at the end of the headband.

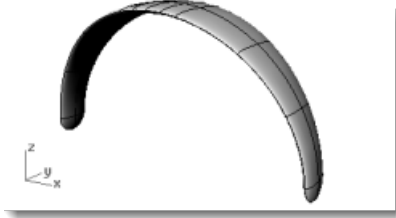


7. Repeat these steps for the other side of the headband.



Join the surfaces

1. **Select** the surfaces.
2. On the **Edit** menu, click **Join**.
Three surfaces joined into one polysurface.



Create the speaker wire

Use a separate layer to create the speaker wire.



Reset the layers

1. On the status bar, click the **Layer** pane.
2. Make **Wire Shape Curves** the current layer and turn on **Wire**.
Turn all other layers off.



Reset the view

- ▶ On the **View** menu, click **Zoom > Zoom Extents All**.

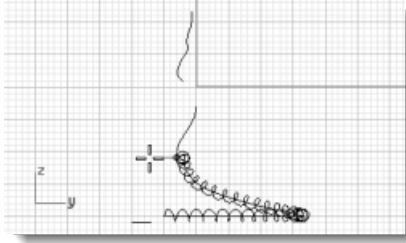


Make the helix

1. On the **Curve** menu, click **Helix**.
2. At the **Start of axis...** prompt, click **AroundCurve**.
3. At the **Select curve** prompt, select the long free-form curve.



4. At the **Radius and start point...** prompt, type **1** and press **Enter**.
This sets the radius for the helix.
5. At the **Radius and start point...** prompt, set **Turns=30**, and **NumPointsPerTurn=8**.
6. At the **Radius and start point...** prompt, in the **Right** viewport drag the cursor to the left and click.



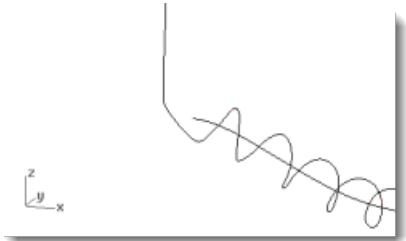
Reset the view

1. On the **View** menu, click **Zoom > Window**.
2. In the **Perspective** viewport, zoom in on the left end of the helix you just created.



Match and join the helix to the end curves

1. From the **Curve** menu, click **Curve Edit Tools > Match**.



2. At the **Select open curve to change - pick near end** prompt, select near the left end of the helix.
3. At the **Select open curve to match - pick near end...** prompt, select near the lower end of the vertical curve.
4. In the **Match Curve** dialog box, under **Continuity**, click **Tangency**, under **Preserve other end**, click **Position**, and click **Join**.
5. Repeat steps 3 through 6 for the other end of the helix.



Create the speaker wire

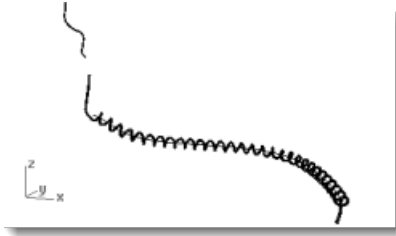
1. **Select** the extended helical curve.
2. On the **Solid** menu, click **Pipe**.
3. At the **Starting radius...** prompt, type **.2** and press **Enter**.
4. At the **End radius...** prompt, press **Enter**.
5. At the **Point for next radius** prompt, press **Enter**.



Create the second wire

1. **Select** the curve at the top left.
2. On the **Solid** menu, click **Pipe**.

3. At the **Starting radius...** prompt, type **0.1** and press **Enter**.
4. At the **End radius...** prompt, press **Enter**.
5. At the **Point for next radius** prompt, press **Enter**.



Mirror the headphone parts

Mirror the parts to create the parts for the other side of the headphones.



Reset the layers and view

1. On the status bar, click the **Layer** pane.
2. Turn on all layers.



Reset the view

- ▶ On the **View** menu, click **Zoom > Zoom Extents All**.

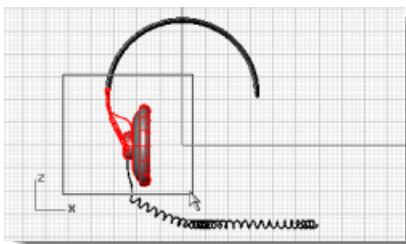
Delete all the shape curves

1. Press **Esc** to deselect everything.
2. On the **Edit** menu, click **Select Objects > Curves**.
3. Press the **Delete** key.

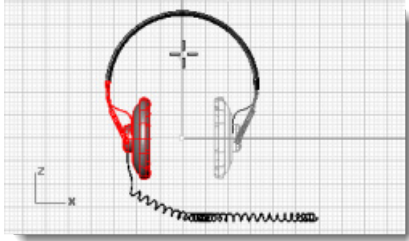


Mirror the left half of the headphones

1. In the **Front** viewport, window select the objects as illustrated. (**Select** the speaker, bracket, small wire, and rotated ellipse.)



2. On the **Transform** menu, click **Mirror**.
The **Mirror** command depends on which viewport is active. It uses the construction plane in the active viewport to define the mirror plane. The mirror plane is perpendicular to the construction plane. Two points define the line in this plane about which the selected objects are mirrored.
3. At the **Start of mirror plane** prompt, type **0,0**.
This is the first point of the mirror line.



4. At the **End of mirror plane** prompt, turn on **Ortho**, and drag the mirror line straight up and pick.



Learn more

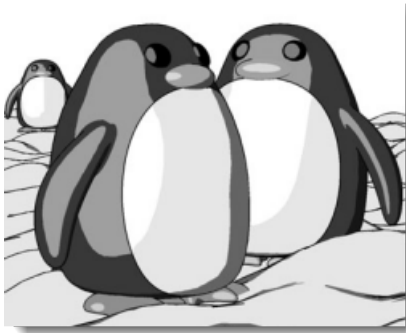
For a video tutorial showing a more sophisticated modeling method for a headphone set using Rhino's Gumball feature, see: <http://vimeo.com/30140235>.

Chapter 16: Penguin Tutorial - Point Editing and Blending

This tutorial demonstrates point-editing techniques including moving and scaling control points and adding knots to surfaces to increase control. In addition, you will use blends to create smooth transitions between surfaces.

You will learn how to:

- Rebuild surfaces to add additional control points.
- Insert knots in a surface to add control points in a specific location.
- Edit surface control points to define a shape.
- Scale control points to change the object shape.
- Use object snaps projected to the construction plane.
- Orient an object on a surface.
- Create smooth blends between surfaces.



Rendered with **Penguin** renderer by Jari Saarinen.

The body

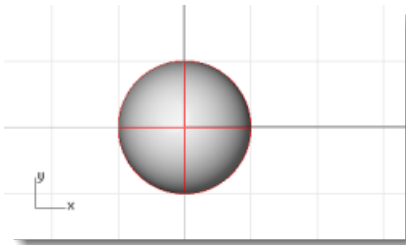
If you like, open the example model, **Penguin.3dm**, and try to match the shapes as you are building the model. Experiment with your own shapes, too.

The body and head are created from one sphere. The shape is formed by moving the control points in the sphere to create the head.



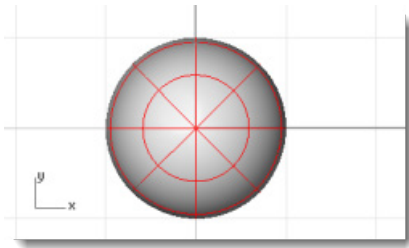
Draw a sphere

- ▶ In the **Top** viewport, use the **Sphere** command to draw a sphere with a radius of **10** units.



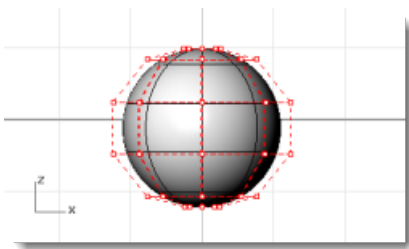
Rebuild the sphere

- ▶ Use the **Rebuild** command to add more control points to the sphere.
In the **Rebuild Surface** dialog box, set the **Point count** in the **U** and **V** directions to **8** and the **Degree** in the **U** and **V** directions to **3**.
Check **Delete input**.
Click **OK**.



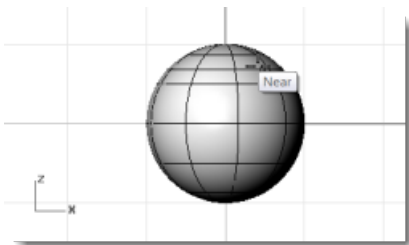
Turn control points on

- ▶ Use the **PointsOn (F10)** command to turn on the sphere's control points. Look in all the viewports at the structure of the control points.
The next step will change this structure so the influence of moving the control points does not extend over the whole sphere.

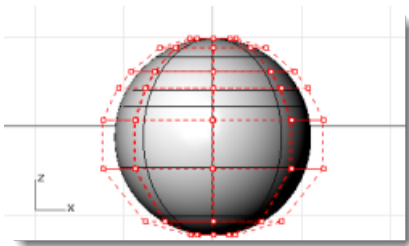


Insert knots

- ▶ Use the **InsertKnot** command to insert two knots in the sphere in the area where you want the neck.
Insert the knots in the u-direction only as illustrated.



Examine the control point structure after inserting the knot.



Reposition control points to create the indentation for the neck and to reform the body shape.

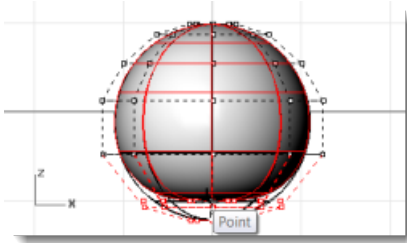


Flatten the bottom

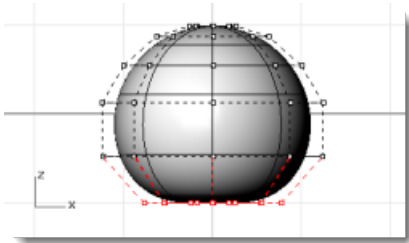
- ▶ In the **Front** viewport, select all the control points in the lowest rows of the sphere and use the **SetPt** command to match them to the bottom pole point in the world z-direction only.

In the **Set Points** dialog box, check **Set Z**, clear the **Set X** and **Set Y** checkboxes, and click **World**.

Drag the selected control points up.

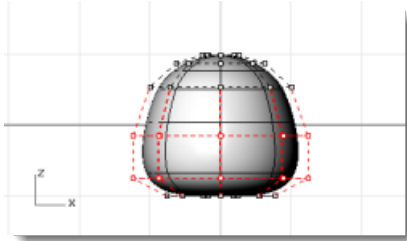


This will align all of the selected control points to the same z-value (up in **Front** viewport), flattening the surface.

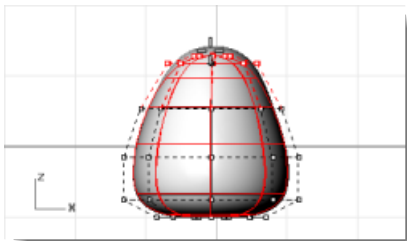


Drag points

- ▶ **Select** rows of control points with a window and drag them up or down in the **Front** viewport.

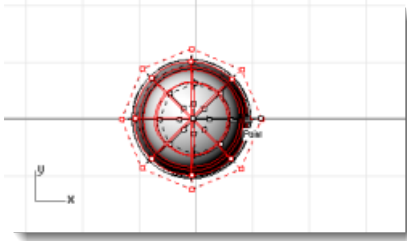


Use **WireFrame** display mode if you find it easier to select control points in wireframe views.



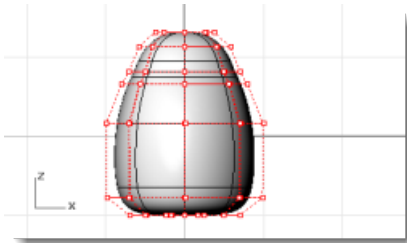
Scale points

1. **Select** rows of control points with a window in the **Front** viewport. In the **Top** viewport, use the **Scale2D** command to move them closer or farther away from the central point. To pick the base point for the **Scale2D** command use the **Point** object snap with **Project** turned on. This will scale the points parallel to the construction plane. Watch the **Front** viewport to see the changes in the body shape as you move the control points closer to and farther from the center.

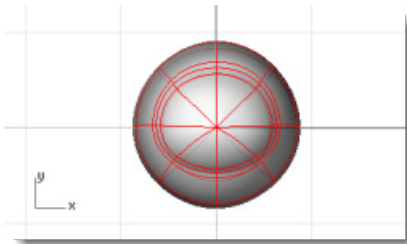


Experiment with the **Project** setting in the **Osnap** toolbar to see how it works. You will be able to see the tracking line projected to the construction plane in the viewports.

Match the example model or use your own shape.



2. Drag individual groups of control points to make the body slightly flatter in the front near the neck as illustrated.



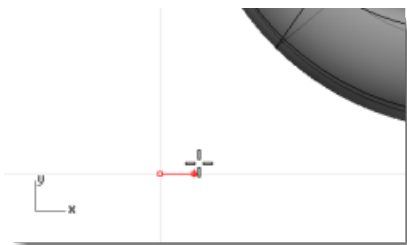
The eyes

The eye is an ellipsoid shape that is oriented onto the surface.



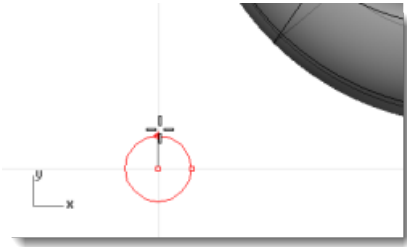
Create the eye

1. In the **Top** viewport, start the **Ellipsoid** command.
Place the center point anywhere.
2. At the **End of first axis** prompt, type **1.1** to constrain the distance from the center point to the end of the axis to 1.1 units.
Drag the cursor to the right and pick.

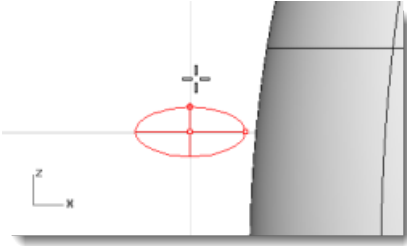


3. At the **End of second axis** prompt, type **1.1** to constrain the distance.
Using these constraints has created a circular ellipsoid when seen from the top.

Drag the cursor up or down in the **Top** viewport and pick.

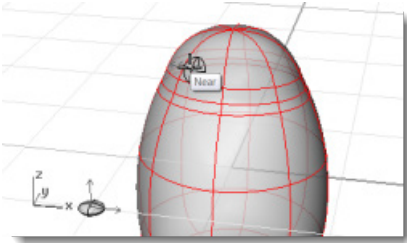


4. At the **End of third axis** prompt, type **.5**, press **Enter**.



Orient the eye on the surface

1. **Select** the eye ellipsoid in the **Top** or **Perspective** viewport.
2. Start the **OrientOnSrf** command.
3. At the **Base point (OnSurface)** prompt, in the **Top** viewport, pick the center of the ellipsoid.
4. At the **Reference point for scaling and rotation** prompt, pick any point to the right or left of the eye ellipsoid.
The exact location is not important.
5. At the **Surface to orient on** prompt, select the penguin body/head.
6. In the **Orient on Surface** dialog box, click **OK**.

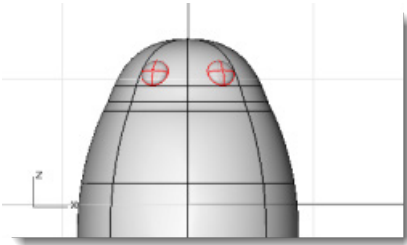


7. At the **Point on surface to orient to...** prompt, move the cursor onto the head to where you want to place the eye and click.



Mirror the eye

- ▶ Use the **Mirror** command in the **Front** viewport to create the second eye.



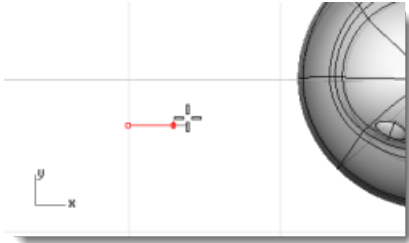
The beak

The beak is another ellipsoid that you can edit to change the shape.

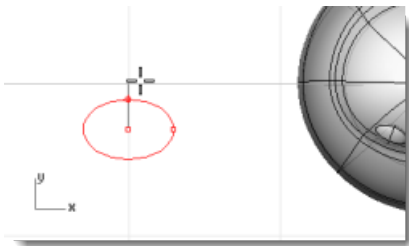


Create the basic beak shape

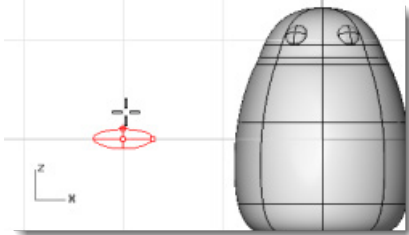
1. In the **Top** viewport, start the **Ellipsoid** command.
Place the center point anywhere.
2. At the **End of first axis** prompt, type **3** to constrain the distance from the center point to the end of the axis to three units.
Drag the cursor to the right and pick.



3. At the **End of second axis** prompt, type **2** to constrain the distance.
Using these constraints creates a circular ellipsoid when seen from the top.
Drag the cursor up or down in the **Top** viewport and pick.

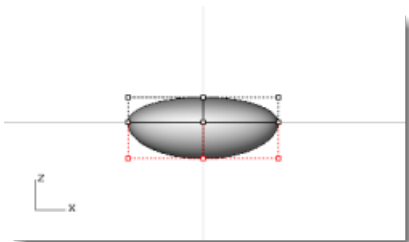


4. At the **End of third axis** prompt, type **1**, press **Enter**.

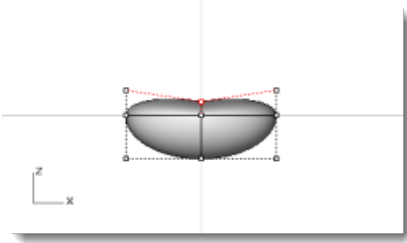


Shape the beak

1. Turn on the **control points (F10)**.
In the **Front** viewport, select the lower row of points and drag them down.

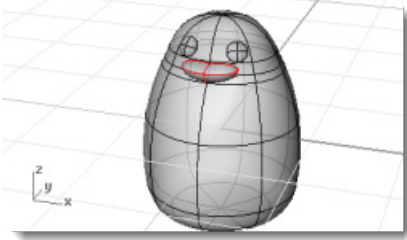


2. **Select** the row of points in the top center and drag them down to shape the beak.
Try using the **Nudge** keys (**Alt** + **Arrow** direction keys) to nudge the selected points.



Move the beak

- ▶ **Move** the beak into position.



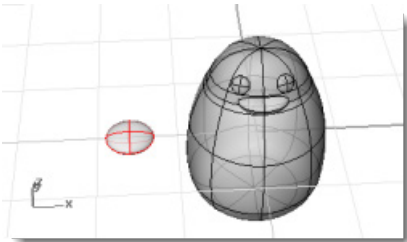
The feet

The feet are created using another ellipsoid. Knots are added to help create the webbed toes.



Draw the beginning ellipsoid

1. In the **Front** viewport, start the **Ellipsoid** command.
Place the center point anywhere.
2. At the **End of first axis** prompt, type **1** to constrain the distance from the center point to the end of the axis to one unit.
Drag the cursor up and pick.
3. At the **End of second axis** prompt, type **3** to constrain the distance.
In the **Top** viewport, drag the cursor up and pick.
4. At the **End of third axis** prompt, type **3**, press **Enter**.



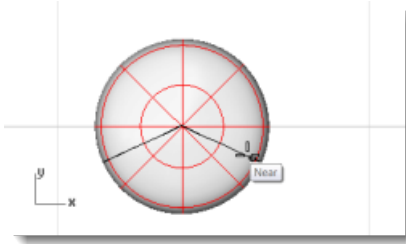
Rebuild the ellipsoid

- ▶ Use the **Rebuild** command to add more control points to the ellipsoid.
In the **Rebuild Surface** dialog box, set the **Point count** in the **U** and **V** directions to **8** and the **Degree** in the **U** and **V** directions to **3**.
Check **Delete input**.



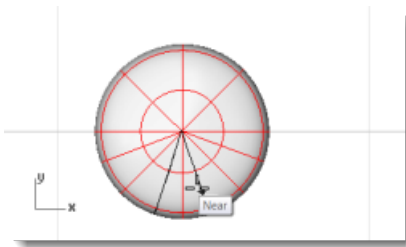
Insert knots to create the webbed feet

- ▶ Use the **InsertKnot** command to insert four knots in the ellipsoid as illustrated.



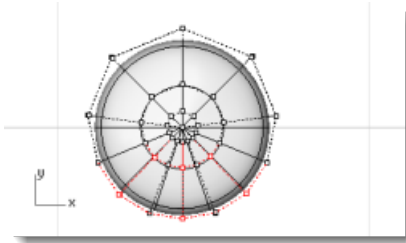
Set the **Symmetrical=On**.

Insert the knots in the **V-direction**.

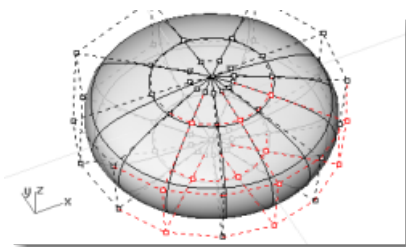


Scale the points from the center

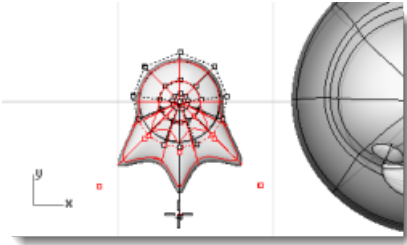
1. **Select** control points as illustrated.



Use window and crossing selections to select the control points on both the top and bottom of the ellipsoid.



2. Use the **Scale2D** command to scale the control points out from the center of the foot.
Use the **Point** object snap to set the base point of the scale to the center point of the ellipsoid.
Drag the points to make the whole foot about twice the size of the original ellipsoid.



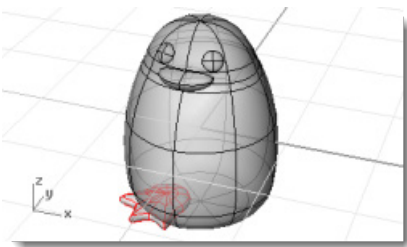
Move the foot into position

- ▶ Use the **Move** command to move the foot under the penguin body.



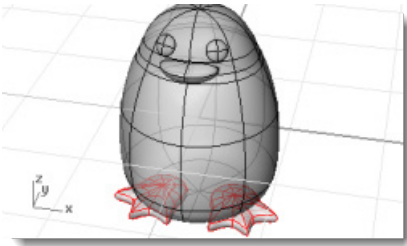
Rotate the foot out

- ▶ Use the **Rotate** command to rotate the foot out slightly.



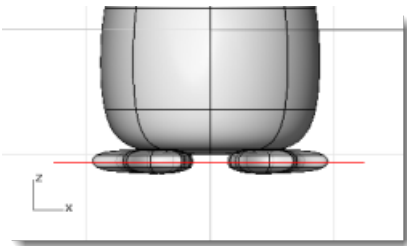
Mirror the foot

- ▶ Use the **Mirror** command to create the second foot.

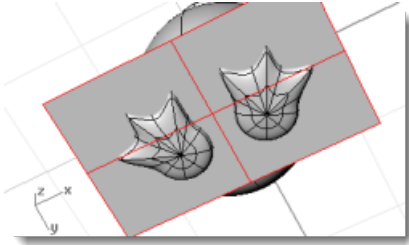


Create a cutting plane

1. **Select** the feet.
2. In the **Front** viewport, use the **CutPlane** command to make a planar surface that passes through the feet as illustrated.

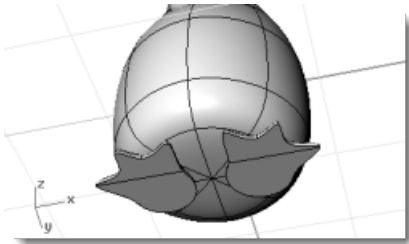


The **CutPlane** command makes a plane that passes through the selected surfaces along the line you draw.



Boolean the feet and the plane

1. **Select** the plane and the feet.
2. Start the **Boolean2Objects** command.
3. Click through the preview options until the result is the flat feet as illustrated, and press **Enter**.



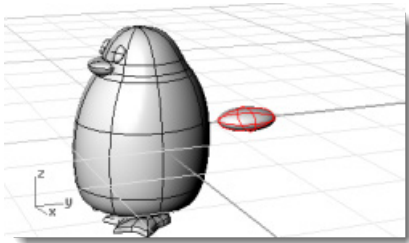
The tail

The tail is another ellipsoid. It is joined to the body with a smooth blend surface.



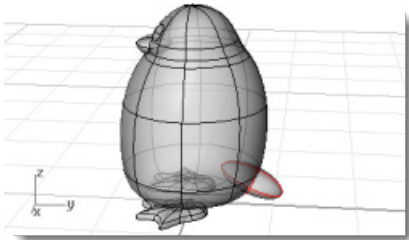
Create the tail shape

- ▶ Draw an **Ellipsoid** that is **4** units long, **3** units wide (**Top** viewport), and **1.5** units tall (**Front** viewport).



Position the tail

- ▶ **Move** and **Rotate** the tail in position.



Union the tail and body

- ▶ Use the **BooleanUnion** command to trim and join the tail and the body shapes.
The transition between the tail and body is rather abrupt; so replace this with a smooth blend surface.

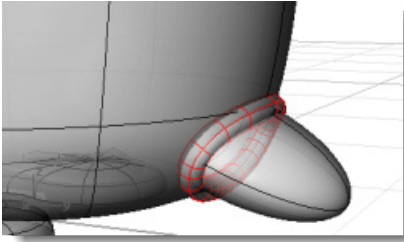
To do this, you must first create a gap between the two parts for the blend surface to fill.



Pipe the intersection

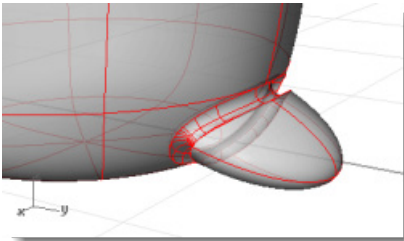
- ▶ Use the **Pipe** command to create a circular surface around the edge between the body and tail.
At the **Select curve to create pipe around** prompt, select the edge between the tail and the body.

At the **Radius for closed pipe** prompt, type **.4**.



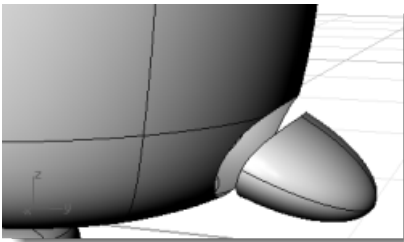
Difference the body and tail with the pipe

1. Use the **BooleanDifference** command to trim both the body and the tail surfaces inside the pipe.
Use the **DeleteInput** option to delete the original surfaces.
2. At the **Select first set of surfaces...** prompt, select the body/tail and press **Enter**.
3. At the **Select second set of surfaces...** prompt, select the pipe surface and press **Enter**.



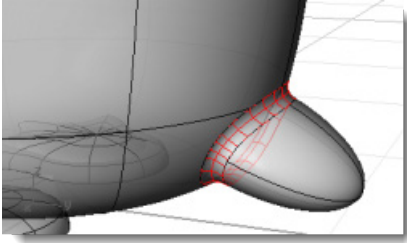
Explode the parts

1. Use the **Explode** command to separate the parts.
2. **Delete** the part of the pipe remaining between the body and tail.



Blend between the tail and body

- ▶ Use the **BlendSrf** command to create a smooth surface between the tail and the body.

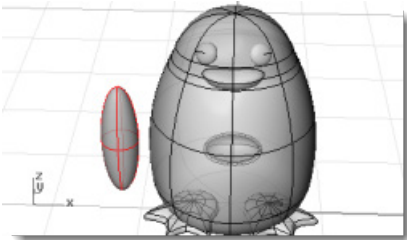


The wings



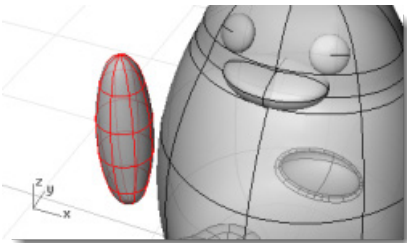
Create the base wing shape

- ▶ Draw an **Ellipsoid** that is **2** units long, **2** units wide (**Top** viewport), and **6.5** units tall (**Front** viewport).

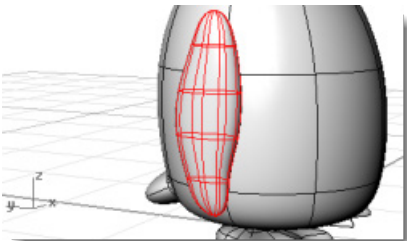


Rebuild the wing

1. Use the **Rebuild** command to add more control points to the ellipsoid.
In the **Rebuild Surface** dialog box, set the **Point count** in the **U** and **V** directions to **8** and the **Degree** in the **U** and **V** directions to **3**.
Check **Delete input**.



2. Drag control points to create the shape.



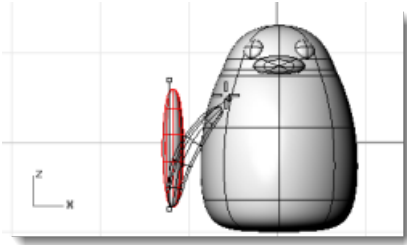
Bend the wing to the body

1. Use the **Bend** command in the **Front** viewport to bend the top of the wing shape toward the body.

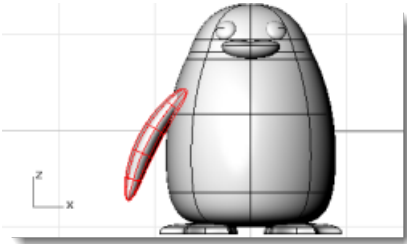
At the **Start of spine** prompt, in the **Front** viewport, pick near the bottom of the wing.

At the **End of spine** prompt, pick near the top of the wing.

At the **Point to bend through...** prompt, drag the top of the wing toward the body.

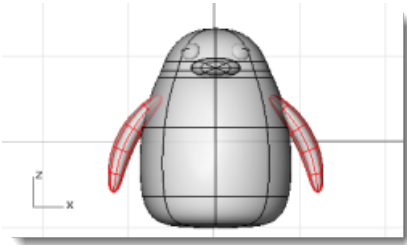


2. If further positioning is needed, use the **Rotate** and **Move** commands to place the wing.



Mirror to the other side

- ▶ Use the **Mirror** command to create the opposite wing.



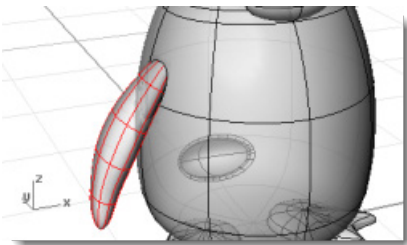
Union the wing and the body

- ▶ To trim the wing hole and the wing, select both wings and the body and use the **BooleanUnion** command.



Explode the parts

- ▶ Use the **Explode** command to separate the parts.



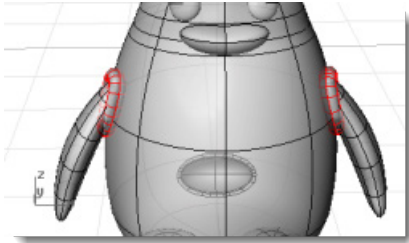


Pipe the intersection

- ▶ Use the **Pipe** command to create a circular surface around the edge between the body and each wing.

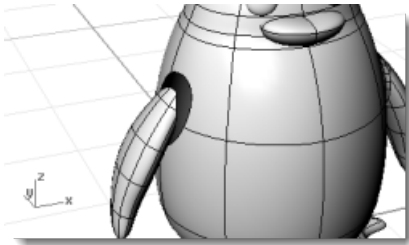
At the **Select curve to create pipe around** prompt, select the edge of the hole in the body or the edge of the wing surface.

At the **Radius for closed pipe** prompt, use a radius of about **.6**.



Trim the body and wing

1. Use the **Trim** command to trim the body and wing surfaces inside the pipe surfaces.
2. **Delete** the pipe surfaces.



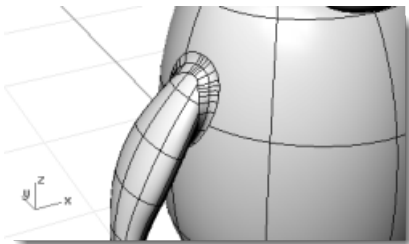
Blend between the body and wing

- ▶ Use the **BlendSrf** command to create a smooth surface between each wing and the body.



Join the body and wing

- ▶ **Join** the blends and wings to the body



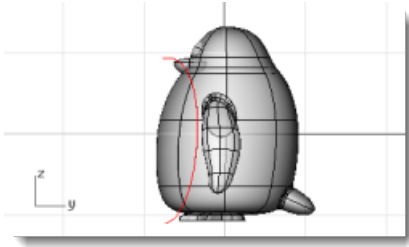
Finishing touches

To finish the penguin, split the front part of the body so a different material can be applied to it.



Draw a trim curve

- ▶ In the **Right** viewport, draw a **Curve** from the beak down to the bottom as illustrated.



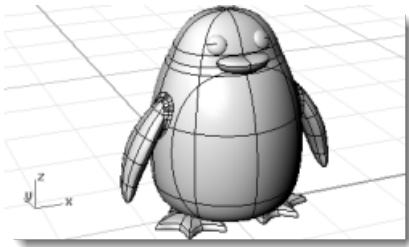
Split the body with the curve

- ▶ Use the **Split** command to split the body surface with the curve.



Join the body parts

- ▶ Use the **Join** command to join the body (except the front), the tail, and the wings.



Render

Rendering creates a realistic picture of your model with colors you assign. These render colors are different from the layer colors you might be using, which control the display in wireframe and shaded modes.



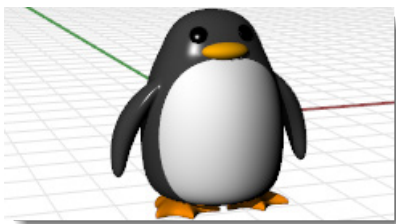
Set up the view

- ▶ Use the **Rendered** display mode to set the viewport rendered mode.



Assign materials

1. **Select** the body.
2. Start the **Properties** command.
3. In the **Properties** window, click the **Material** button.
4. Set **Assign material by**, to **Object**.
5. Under **Basic Settings**, click the **Color** swatch.
6. In the **Select Color** dialog box, select a color for the body.
7. Set the **Gloss finish** to about **40**.
8. **Select** the other parts and apply materials in the same way.



Chapter 17: Boat Tutorial - Loft and Sweep

This tutorial demonstrates classic boat hull lofting techniques using typical plan and profile curves. The classic hull shape is based a design from an old *Boat Builder's Handbook* magazine. Many designs similar to this are available over the Internet.

You will learn how to:

- Create 3-D curves from a 2-D lines drawing.
- Rebuild and simplify the curves.
- Use analytical techniques to ensure fairness.
- Loft surfaces from the curves.



Rhino is used by marine designers in many segments of the industry. For more tutorials and information about marine design, see the Rhino Web site at www.rhino3d.com.

Marine Terms Used in this Tutorial

Sheer

The fore-and-aft curvature from the bow to the stern of a ship's deck as shown in side elevation.

Chine

The intersection of the bottom and the sides of a flat or v-bottomed boat.

Transom

The planking forming the stern of a square-ended boat.

Fair

The meaning of "fair" is much debated in the marine industry. No one can define it, but they know when they see it. Although fairing a surface is traditionally associated with hull surfaces, all visible surfaces on any object can benefit from this process. In Rhino, the first cue for fairness in a surface is the spacing of the surface display isocurves.

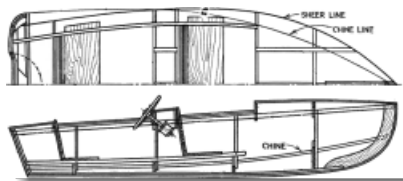
There are other characteristics of fair curves and surfaces. Although a curve or surface may be fair without exhibiting all of the characteristics, they tend to have these characteristics. If you keep these in mind while modeling, you will end up with a better final product.

The guidelines for creating a fair surface include:

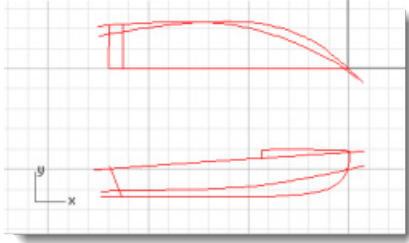
- Use the fewest possible control points to get the curve shape.
- Use the fewest possible curves to get the surface shape.

Lay out the hull curves

The hull lines were created by tracing the original plans using a background bitmap. The first step is to check the lines for fairness before creating surfaces from them.



The designer's lines are illustrated. The sheer and chine have been extended at the forward and aft ends to accommodate the lofting process.



Start the Model

1. From the Rhino **File** menu, click **Open**.
2. Browse to the **Tutorial Models** folder that you downloaded with the *User's Guide*.
3. Open the model file **Victory.3dm**.

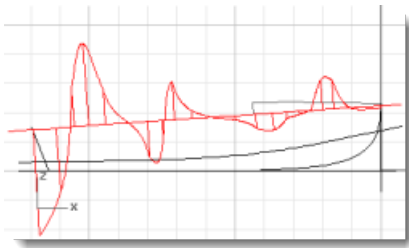
The lines are laid out on the **Plan** layer and the **Profile** layer.

Check the curves for fairness

Select each of the designer's curve pairs in plan and profile and use the **CurvatureGraph** command to determine if the curves are "fair." In this case, the file has the original curves that were traced from the background bitmap. They are not "fair." In other words, the curves do not smoothly transition from one end of the sheer to the other. If any curve is not fair, adjust points to make it fair. Start with the sheer (the curve at the top of the hull shape). It has the biggest impact on the appearance of the vessel.

1. **Select** the curves you want to check.
2. Use the **CurvatureGraph** command to display its curvature graph.

The illustration shows the curvature graph applied to the two-dimensional sheer in profile.



The curvature graph should be continuous and exhibit the characteristics desired for the curve. When the curve is concave downward, the graph will be above the curve. Conversely, concave upward curves will have their graphs below them. The point of inflection (where the curve is neither concave upward nor downward) is indicated where the graph crosses the curve.

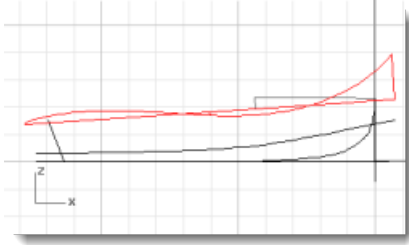
Rebuild the curves

Before doing any point editing to make the curves fair, rebuild the curves to remove excess control points.

Select each curve and use the **Rebuild** command to reduce the number of points and set the degree. Do not use more points than you absolutely need.

Use the **CurvatureGraph** command to check the curves again for fairness. If the curvature graph is still not satisfactory, move the control points until you have a smooth graph. Proceed with the rest of the curves in the model to be certain they are fair before beginning to surface the model.

1. **Select** the sheer curve.
2. Start the **Rebuild** command.
3. In the **Rebuild Curve** dialog box, change the **Point count** to **6** and the **Degree** to **5**.



Create the 3-D curves

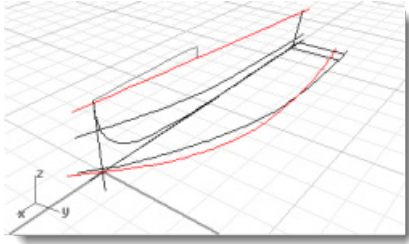
So far, you have been working with two-dimensional curves. In order to loft the surfaces, these planar curves will be used to create three-dimensional curves and the planar curves can be discarded.

With the **3D Lines** layer current, select the profile and plan view representations of each curve. Use the **Crv2View** command to create the three-dimensional curve that combines the x-, y-, and z-coordinates of the two-dimensional curves. The two-dimensional curves must be planar for this command to work.

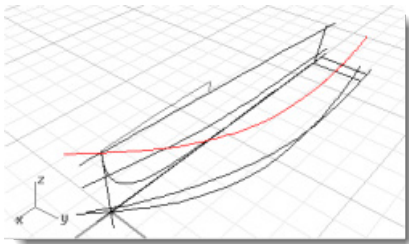


Create the three-dimensional curves

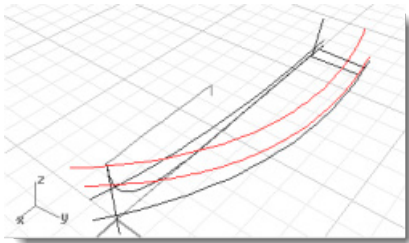
1. Set the **3D Lines** layer current.
2. **Select** the plan and profile representations of the shear curve.



3. Start the **Crv2View** command.
The three-dimensional representation of that curve will be created.



4. When you are satisfied that the proper curve was created, delete or **Hide** the two-dimensional representations.
5. Repeat the **Crv2View** command for the chine curve.



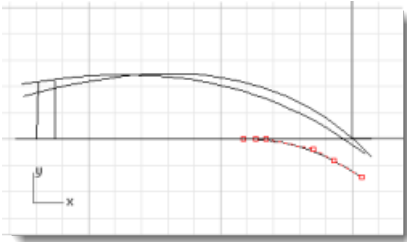
Revise the curves

For the loft process to work on the bottom panel, it cannot come to a point. The lofted shape must be rectangular. This is why the curves are extended beyond the centerline. The curves can be lofted into a rectangular surface that can then be trimmed back. The curves in the Victory model are already extended for you except for the bottom centerline curve.



Create the extended curve

1. Start the **Curve** command.
2. Use the **Near** object snap to place the first three control points along the centerline. Draw the curve so it aligns nicely with the chine and sheer curves in the plan view as illustrated.



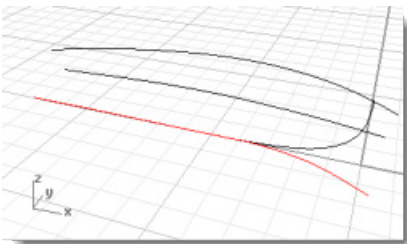
Split the centerline

- ▶ **Split** the centerline curve with the curve extension.



Join the centerline and extension

- ▶ **Join** the curve extension to the aft part of the split centerline. This creates a new bottom curve to use for the surface loft.



Loft the hull surfaces

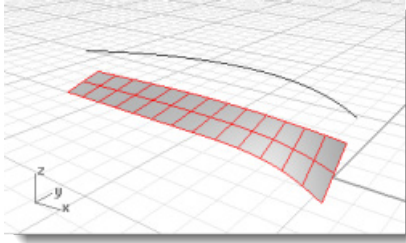
Now that you have created a set of edge curves for the side and bottom, create lofted surfaces from these curves. Start by lofting the bottom surface. Once you have finished it, use its upper edge as the curve from which to loft the side panel.

To loft the bottom panel, select the two edges (chine and centerline) and use the **Loft** command. In this case, be sure to select the new centerline you created in the previous step.

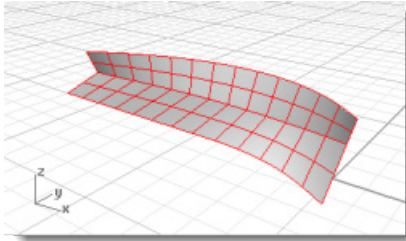


Loft the side and bottom and the chine and centerline

1. **Select** the chine and 2-D centerline.
2. Start the **Loft** command.
3. In the **Loft Options** dialog box, under **Cross-section curve options**, select **Rebuild with...**, and set the control point count to **15**, click **OK**.



4. Repeat the **Loft** for the side panel, selecting the surface edge and the sheer curve. Loft with the same settings in the **Loft Options** dialog box.



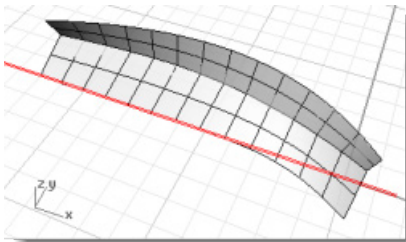
Trim the bow and bottom

When you have successfully created both the side and bottom surfaces, construct a buttock one-half inch off the centerline and trim both surfaces to this buttock. To do this, in the **Top** viewport, draw a line longer than the hull and one-half inch to the right of centerline.



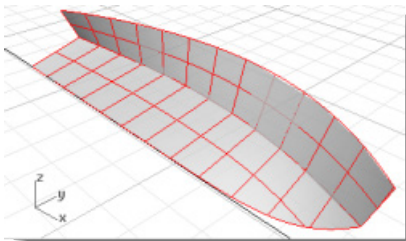
Draw the trim line

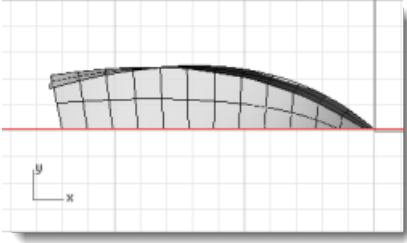
1. In the **Top** viewport, draw a **Line** along the x-axis that is longer than the hull.
2. In the **Top** viewport, offset the line 1/2 inch toward to hull surfaces.



Trim the side and bottom to the trim line

- ▶ Use the **Trim** command to trim the side and bottom as illustrated.





Build the transom

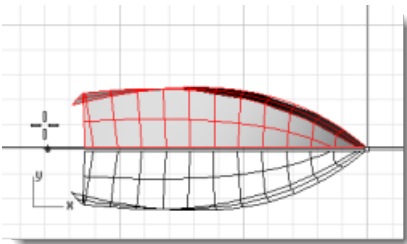
Like all surfaces in this tutorial, the transom will be built with a surface larger than the finished surface and then trimmed to the hull.

To make sure there is enough surface area to trim, **Extend** the transom centerline by a foot or two both above the sheer and below the centerline. **Trim** the hull surfaces with the transom centerline.



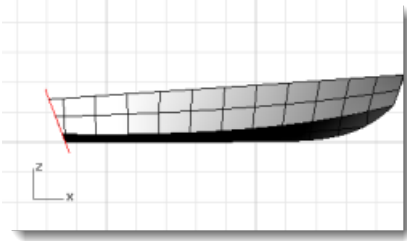
Extend the centerline

1. Start the **Extend** command.
2. At the **Select boundary objects or enter extension length. Press Enter for dynamic extend** prompt, press **Enter**.
3. **Select curve to extend...** prompt, in the **Front** viewport, select near the top of the transom centerline.
4. At the **End of extension** prompt, select a point above the current top of the transom centerline.
5. At the next **Select curve to extend...** prompt, select near the bottom of the transom centerline.
6. At the **End of extension** prompt, select a point below the current bottom of the transom centerline, press **Enter**.



Trim the hull surfaces

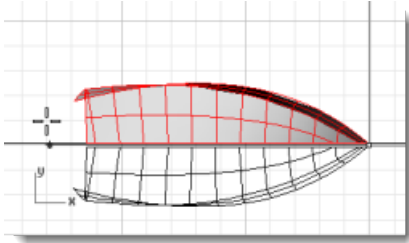
1. **Select** the transom centerline.
2. Start the **Trim** command.
3. In the **Front** viewport, at the **Select object to trim...** prompt, select the hull side and bottom surfaces aft of the transom centerline. Set **UseApparentIntersections=Yes**.



Mirror the hull surfaces

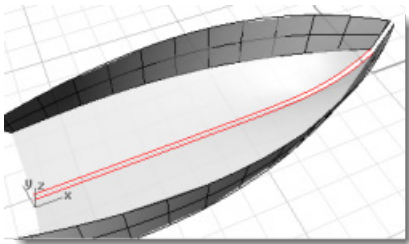
In the **Right** or **Top** viewport, **Mirror** the two hull surfaces about the centerline. Use the **EdgeSrf** command to create surfaces between the two hull halves.

1. **Select** the two hull surfaces.
2. Start the **Mirror** command.
3. In the **Top** viewport, at the **Start of mirror plane...** prompt, type **0**, press **Enter**.
4. At the **End of mirror plane** prompt, with **Ortho** on, drag the mirror plane along the x-axis, and click.

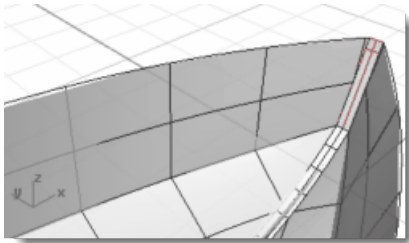


Create the keel surface

1. Start the **EdgeSrf** command.
2. At the **Select 2, 3, or 4 curves** prompt, select the two inner edges of the hull bottom along the keel.



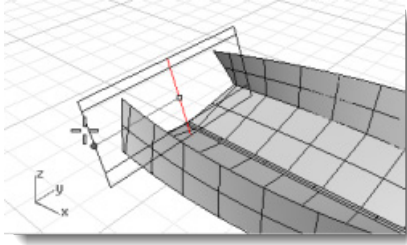
3. Repeat the **EdgeSrf** command.
4. At the **Select 2, 3, or 4 curves** prompt, select the two inner edges of the hull sides along the keel.



Extrude the surface

To create the transom surface, **Extrude** the transom centerline.

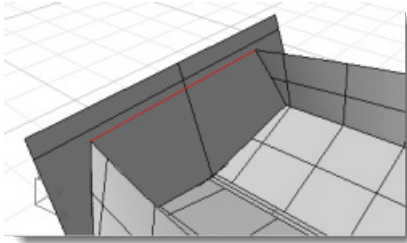
1. In the **Front** viewport, select the extended transom centerline.
2. Start the **ExtrudeCrv** command.
3. At the **Extrusion distance** prompt, set the command-line options **BothSides=Yes** and **Mode=Straight**.
4. In the **Perspective** viewport, drag the extension beyond the hull surface.



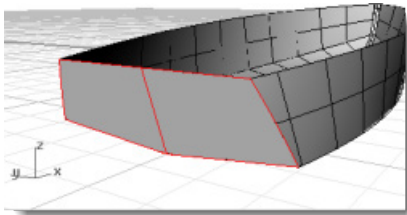
Trim the transom

Trim the transom surface with the hull and a line from the hull edges.

1. Draw a line between the two hull edges.
2. Start the **Trim** command.

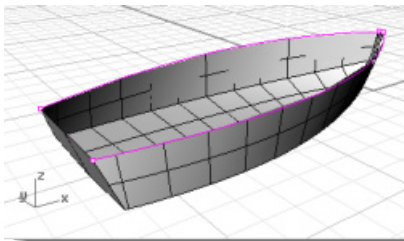


3. At the **Select cutting objects** prompt, select all of the hull surfaces, including the keel surface and the line at the top of the hull, press **Enter**.
4. At the **Select object to trim...** prompt, select the transom surface outside of the hull lines, press **Enter**.



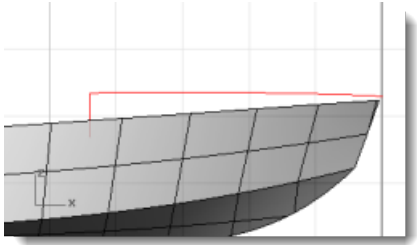
The transom is now complete. **Join** the all surfaces. Use the **ShowEdges** command to check that the join was successful. Display the *naked* edges. Naked edges are surface edges that are not joined to other surfaces. In this case, the only naked edges should be the ones you expect around the outside of the hull surfaces – not those between the surfaces.

When you have your surfaces built and joined, and have no unjoined edges, look at the surface with the curvature analysis tools.



Add the deck

The last step is to create the deck surface. In the profile lines, two curves describe the silhouette of the deck curve. You will use this curve to create the deck.

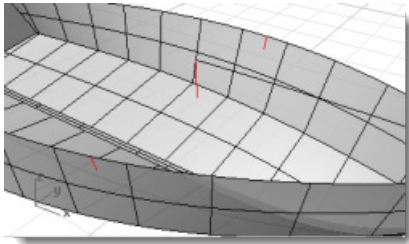


Use the **Project** command to project the vertical line to the side of the hull. This line will act as a marker for the end of the curve. In the **Front** viewport, draw a curve from the end of the deck centerline curve to the end of the projected curve on one side of the hull. Use **Planar** mode to keep the curve planar. Place the first three control points using **Ortho** to keep them lined up at the center.



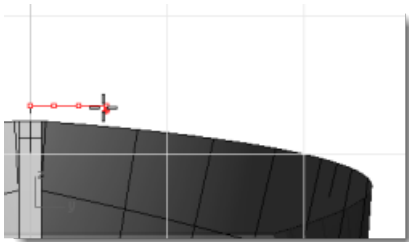
Project the vertical deck edge to the hull

1. **Select** the hull and the vertical line.
2. In the **Front** viewport, use the **Project** command to project the curve to the hull.
The curve will project to both sides of the hull, so you can draw your cross-section curve on either side.

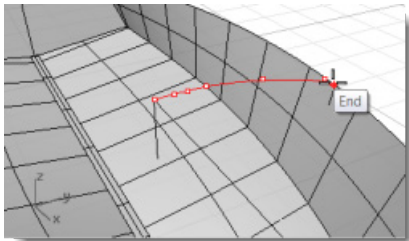


Draw the cross-section curve

1. Click the **Planar** pane in the status bar to turn on **Planar** mode.
2. In the **Front** viewport, use the **Curve** command to draw a control point curve from the top end of the deck centerline curve to the top of the curve projected to the hull.
Use **Ortho** to place the first three control points in a straight line.



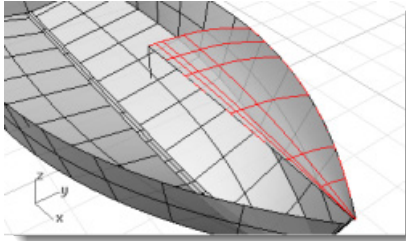
Use the **End** object snap to place the last point at the top of the projected curve on the hull.





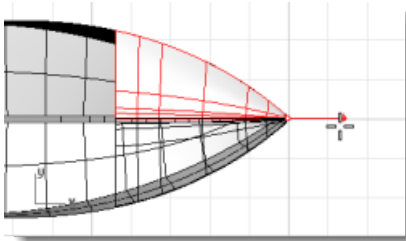
Create the deck surface

1. Use the **Sweep2** command to create the deck surface.
2. At the **Select rail curves** prompts, select the centerline curve and the hull edge.
3. At the **Select cross section curves...** prompt, select the cross-section curve you created from the deck centerline curve to the projected curve on the hull, press **Enter**.



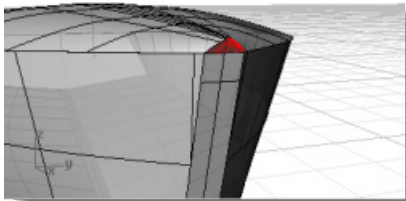
Mirror the deck

1. Use the **Mirror** command to copy the deck surface to the other side.
At the **Start of mirror plane...** prompt, in the **Top** viewport, type **0**, press **Enter**.
2. At the **End of mirror plane...** prompt, in the **Top** viewport, drag the mirror plane with **Ortho** on.



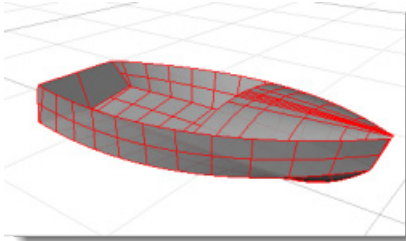
Fill in the missing surface

- ▶ Use the **EdgeSrf** command to create the small triangular surface at the tip of the bow.



Join the parts

- ▶ **Select** all of the surfaces and use the **Join** command.



Chapter 18: Dragonfly Tutorial - Trace Images

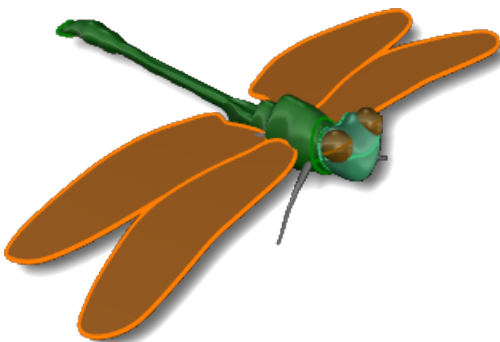
This tutorial demonstrates how to get started modeling an object using photographs as reference material.

You will learn how to:

- Trace an image to create profile curves.
- Create cross-section curves for lofting the profiles.
- Edit control points to change a surface shape.



Starting images.



Finished model.

Note The top and side views are actually of different specimens of this dragonfly. In the side view, the wings are folded up. We will be using the side view image only to draw the side view curves of the body.

Draw the body

Since the dragonfly is symmetrical in the top view, and the model is not going to be a scientific reproduction, trace one side of the dragonfly and mirror the curve to the other side. For the side view, draw two curves since the profile is not symmetrical. Then we will loft cross section curves to make the body. The head will be made separately.

The tail and body will all be made in one piece. The tail is actually several segments that flex. If you were making an animation or a scientific model, you probably would want to divide the dragonfly into smaller surfaces.



Start the model

1. Begin a **New** model.
2. In the **Template File** dialog box, select **Small Objects - Millimeters.3dm**, and click **Open**.



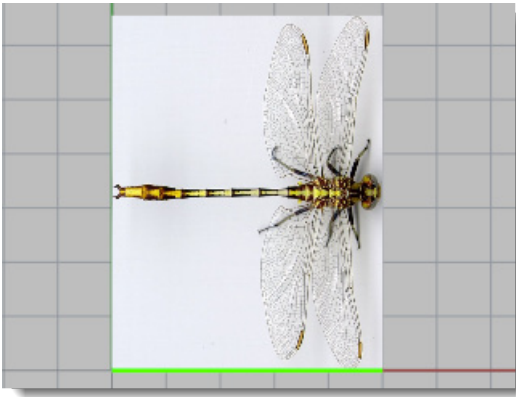
Draw a reference line

- ▶ In the Top view use the **Line** command to draw a reference line **50 millimeters** long starting at **0,0,0**.



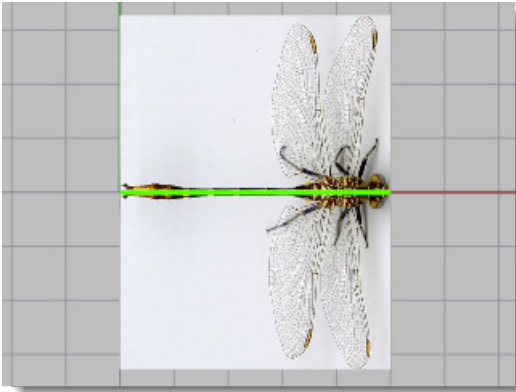
Place the top view image

1. Start **PictureFrame** command.
In the **Tutorial Models** folder that you downloaded with the *User's Guide*, you will find the images you need for this exercise.
2. Find the image file **DragonFly Top.jpg**, and place the image in the **Top** viewport.
Use the reference line to set the length of the picture frame image.



Move the image into place

- ▶ Use object snaps to **Move** the image from the midpoint of the left side (**Mid**) to the construction plane origin at **0,0,0**.



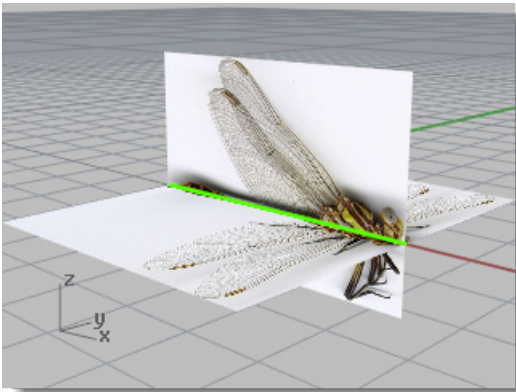
Place the side view image

1. Start **PictureFrame** command.
In the **Tutorial Models** folder that you downloaded with the *User's Guide*, you will find the images you need for this exercise.
2. Find the image file **DragonFly Side.jpg**, and place the image in the **Front** viewport.

Use the reference line to set the length of the picture frame image.



3. Using **Ortho**, **drag** the image down in the **Front** viewport until the reference line matches the center of the dragonfly body.



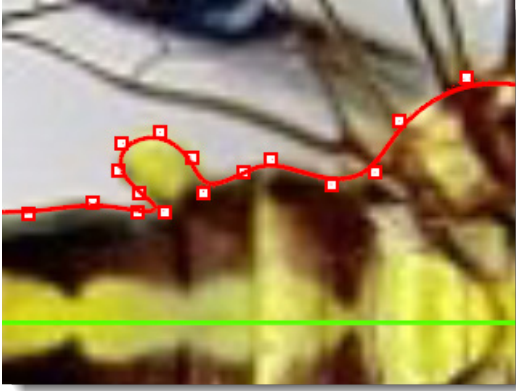
Prepare the view

- ▶ **Hide** the side view picture frame.



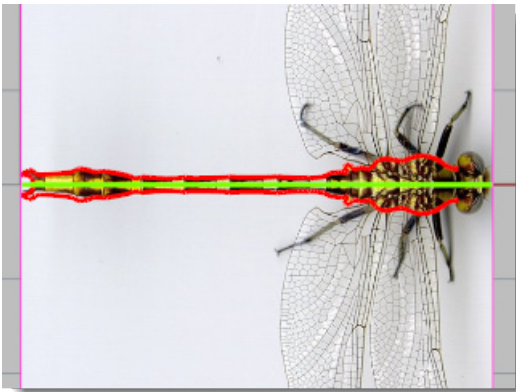
Draw the outline curve

- ▶ In the **Top** viewport, use the **Curve** command to draw an outline of the top half of the dragonfly body. Use as many control points as you think are necessary for the detail. Draw only up to the neck. You will be creating the head another way.



 **Mirror the curve**

- ▶ In the **Top** viewport, use the **Mirror** command to copy the curve around the reference line. The photograph shows that the dragonfly is not symmetrical about its center line. However, since your dragonfly will be stylized, it does not matter in this case. You can choose the level of accuracy you need.

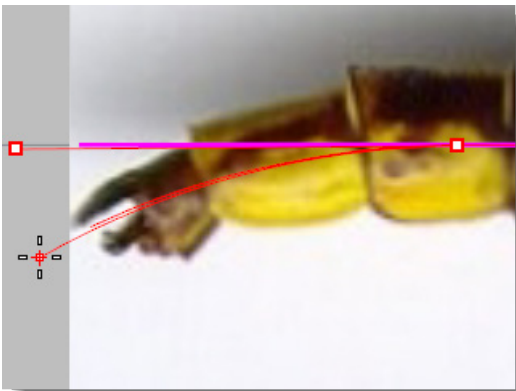


 **Show the side view image**

- ▶ Use the **Show** command to show the side-view picture frame object.

 **Bend the curve**

- ▶ In the **Front** viewport, use the **Bend** command to bend the curves down at the tail to match the bend in the body curve in that view.

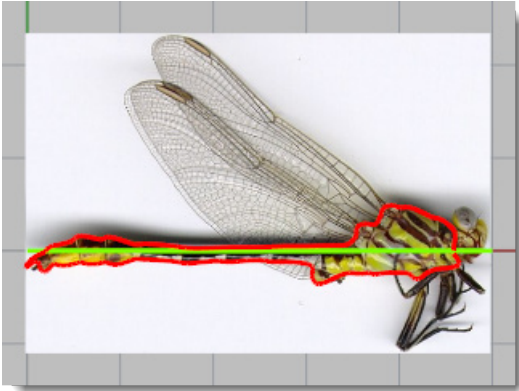




Trace side-view body

- ▶ In the **Front** viewport, use the **Curve** command trace the body outline using two curves, one above the reference line and one below the reference line.

Maximize the viewport and zoom in. Pick as many points as you need to create the curves. Use more points when rounding a corner and fewer points for a straight section.



Prepare the view

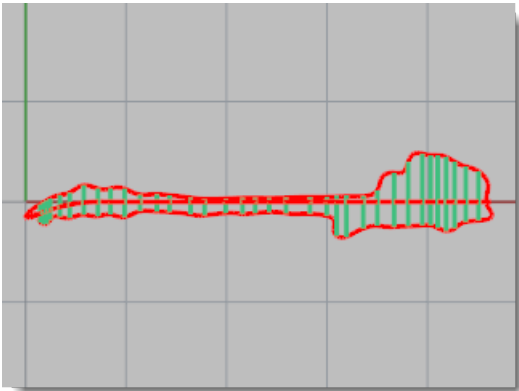
- ▶ **Hide** the picture frame objects and the reference line.



Create the body surface

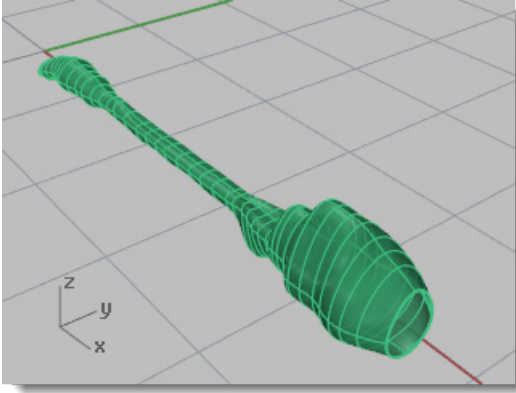
- ▶ Use the **CSec** command to create cross-section profile curves through the top, bottom, and side curves.

Draw as many cross-section curves as you need to maintain the detail. You will be able to see whether you have enough curves when you loft the surface in the next step. If you do not have enough curves to maintain the shape in an area, add more and retry the surface loft.



Loft the body

1. **Select** all the cross-section curves you just created.
2. Use the **Loft** command to create a surface through the cross-section profiles.



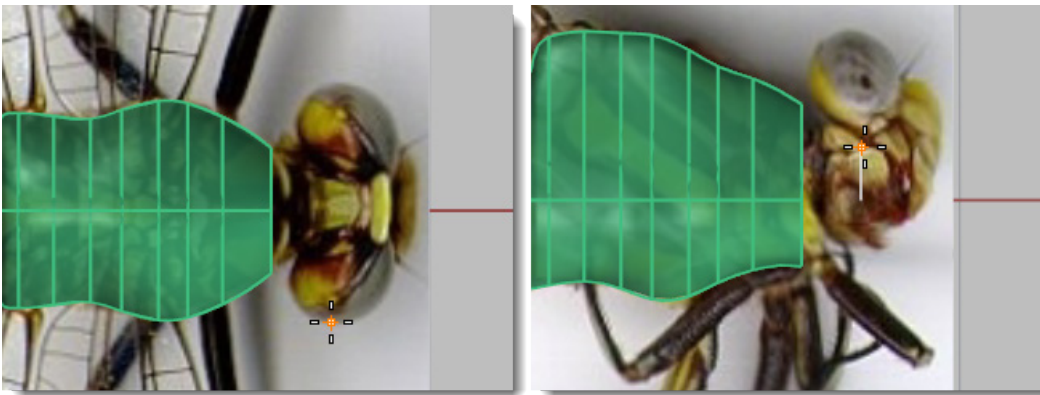
Draw the head

Draw the head with an ellipsoid and move the control points around to deform the head. The eyes are also ellipsoids. The neck is a surface blend.

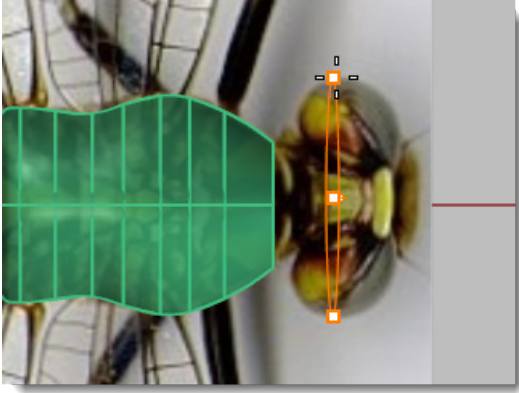


Draw the head

1. Use the **Ellipsoid** command with the **Diameter** option to start the ellipsoid in the **Front** viewport.
2. Use **Elevator Mode** to position the first point.
At the **Start of first axis** prompt, press and hold the **Ctrl** key and in the Top viewport click near the side of the head.
3. In the **Front** viewport, move the cursor up to the center of the head in the side view and click.

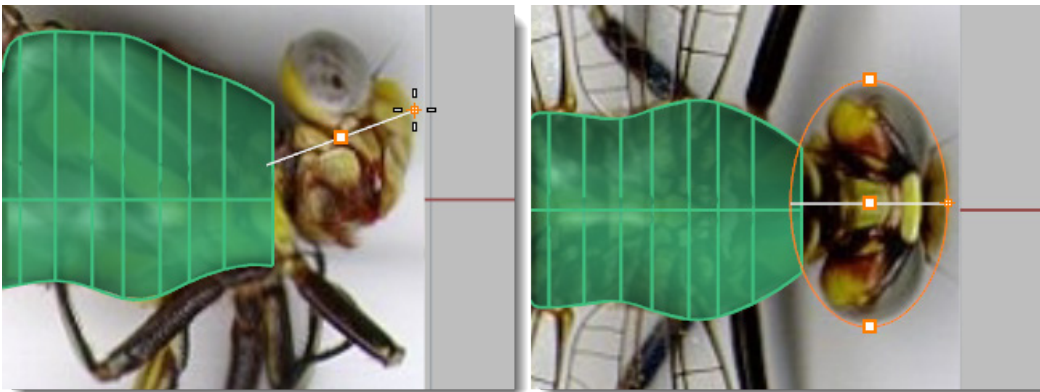


2. At the **End of first axis** prompt, turn on **Ortho** in the **Top** viewport, click at the other side of the head.

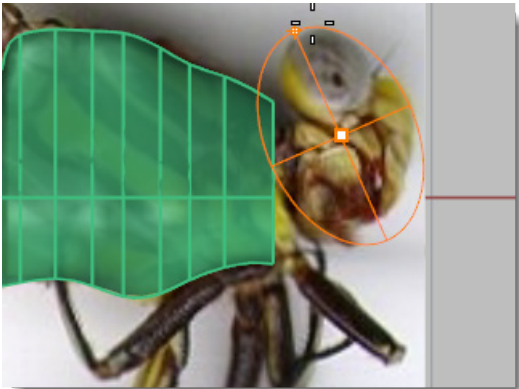


3. At the **End of second axis** prompt, draw pick a point in the **Front** viewport to establish the head size from front to back.

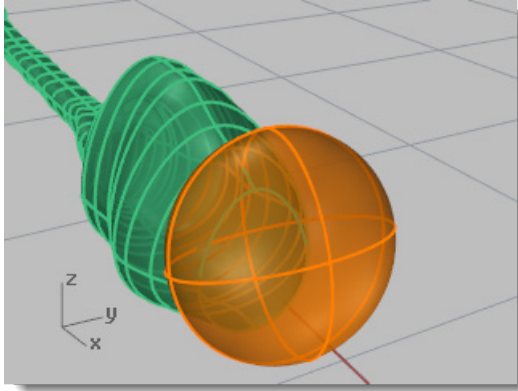
Watch the preview in the **Top** viewport to check the overall size.



4. At the **End of third axis** prompt, pick a point in the **Front** viewport at the top of the head.

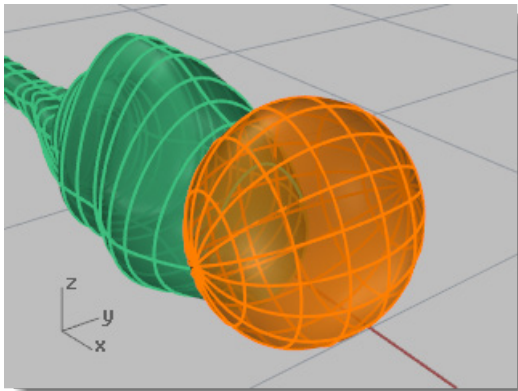


Note: Drawing the ellipsoid in this order and using these viewports is important to get the poles of the ellipsoid in the right place for the next step.



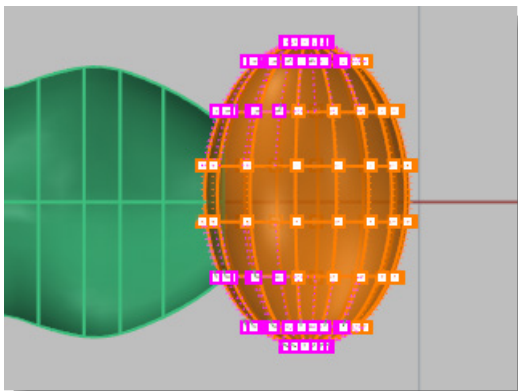
Rebuild the ellipsoid

- ▶ Use the **Rebuild** command to add more control points to the ellipsoid. Set the point count to **16** in the u-direction and **10** in the v-direction.

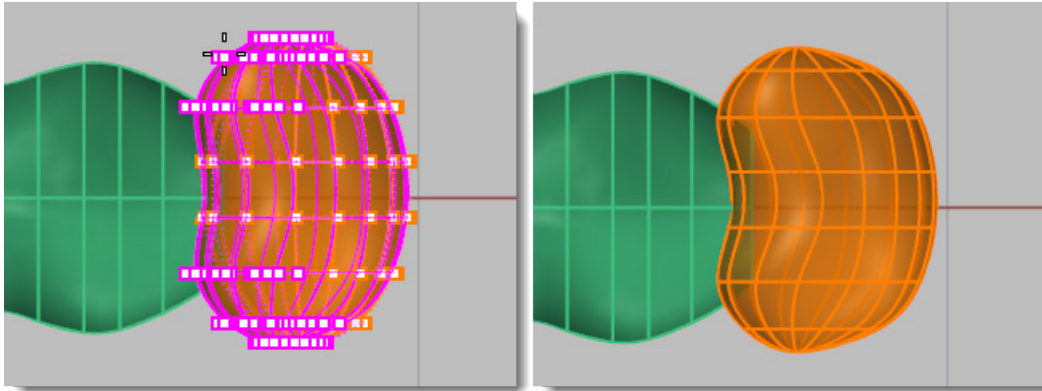


Drag control points to shape the head

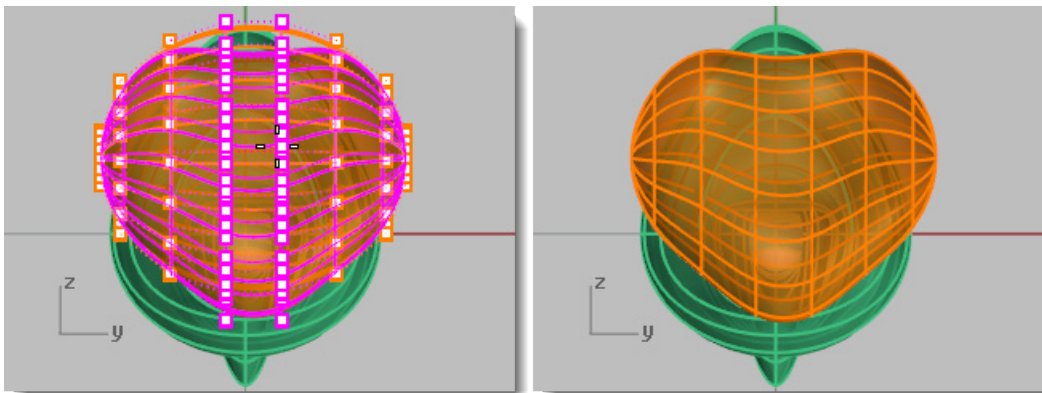
1. Use the **PointsOn** command to turn on control points for the ellipsoid.



2. In the **Top** viewport, select and drag points on both sides of the ellipsoid toward the back to deform the head.



3. In the **Right** viewport, drag the middle two rows of points down.

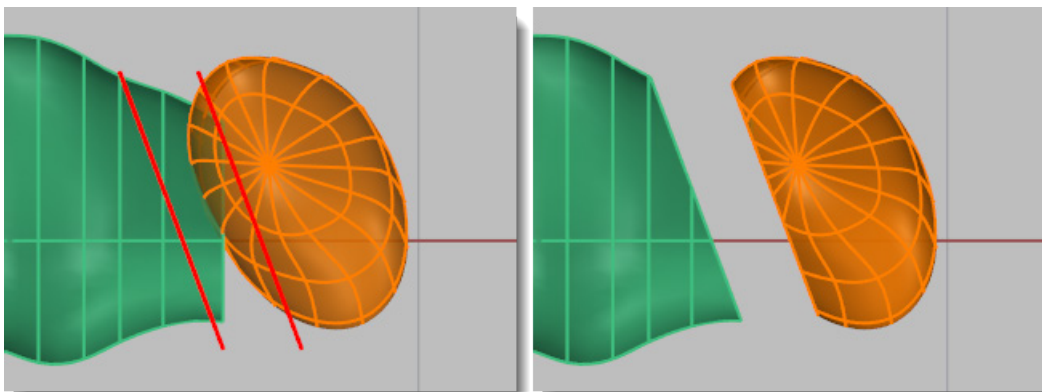


Blend the head and body

The neck is a blend surface between the head shape and the body. First, you are going to trim the head shape to make an opening.

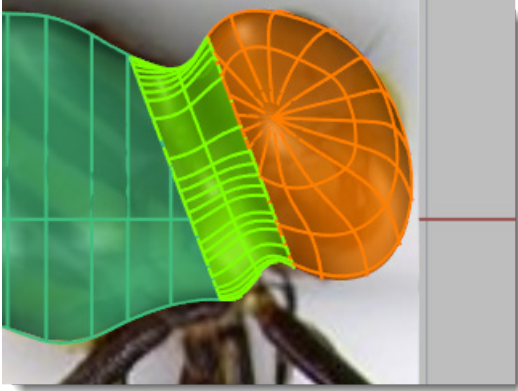
Trim the neck

- ▶ In the **Front** viewport, draw lines as illustrated and use the **Trim** the head and body shapes with the lines.



Blend the neck and body

- ▶ Use the **BlendSrf** command to make a blend surface between the head and body. Be sure the seams are aligned and the direction arrows point the same way.



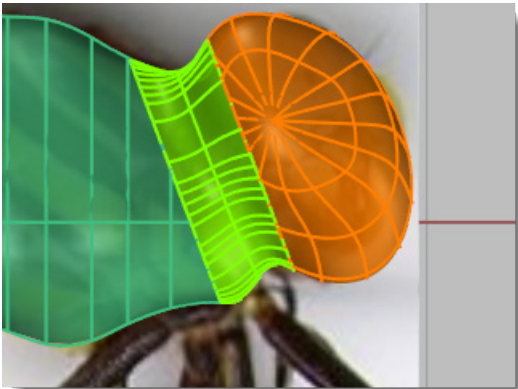
Draw the eyes

The eyes are simple ellipsoids.



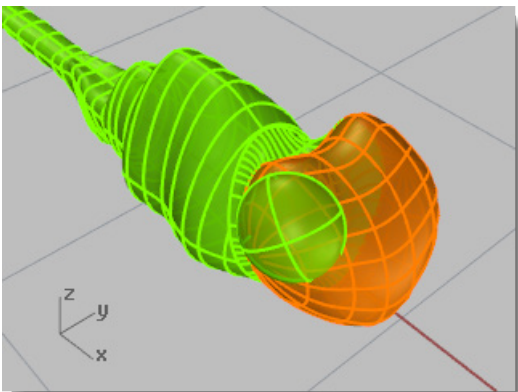
Draw the base ellipsoid

- ▶ Use the **Ellipsoid** command to draw the eye. Base the size and position on the images.



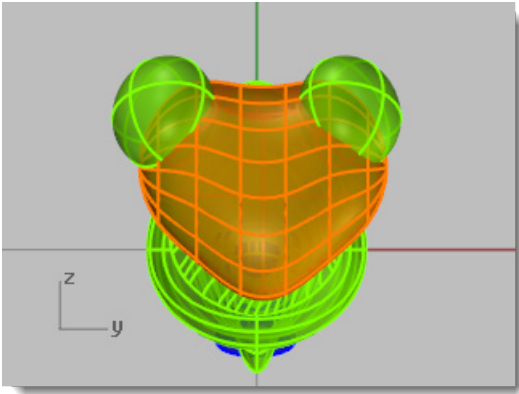
Position the eye

- ▶ Use the **Move** and **Rotate** commands or the **Orient** command to adjust the position of the eye.



Mirror the other side.

- ▶ Use the **Mirror** command to copy the eye to the other side.



Shape the tail

The end of the tail has a rounded shape cut out of it. Use a Boolean to make this shape.



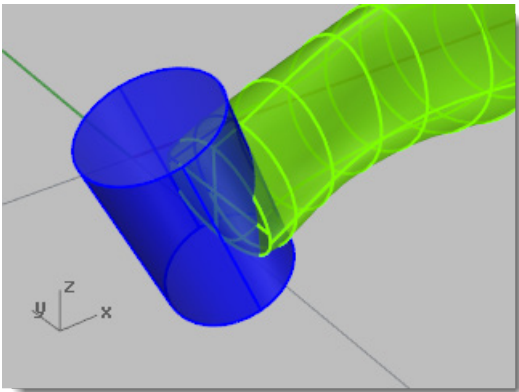
Cap the body

1. If necessary, extend the tail section by turning on the control points and dragging them to match the bitmap.
2. Use the **Cap** command to make the body into a solid.



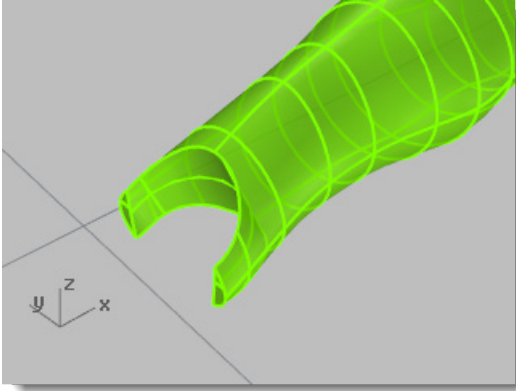
Draw a cutting cylinder

- ▶ Use the **Cylinder** command to draw a solid cylinder so it cuts through the tail as illustrated.



Boolean the tail

- ▶ Use the **BooleanDifference** command to cut the end out of the tail.



Trace the wings

The wings are solids created from closed curves. The legs are created by tracing a polyline down the center of a leg and using a pipe surface to make a series of tubes around the polyline.



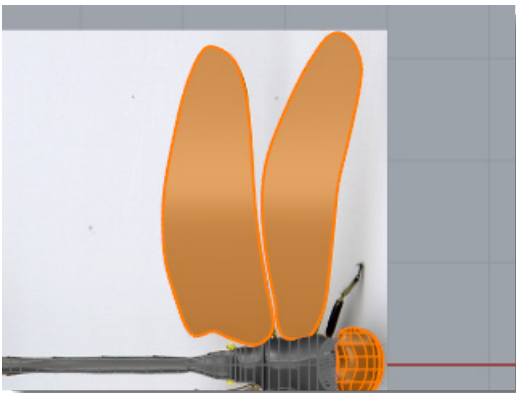
Draw the outline curve

- ▶ In the **Top** viewport, use the **Curve** command to trace the wings on one side of the dragonfly.



Extrude the curve to make a solid

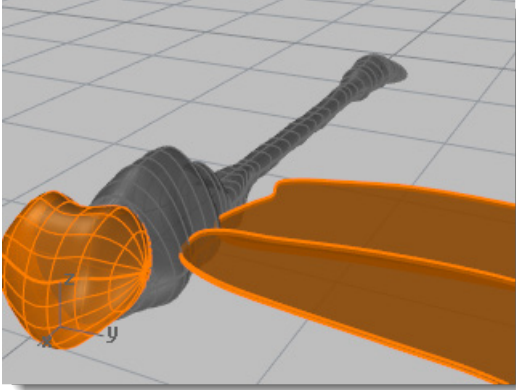
- ▶ Make the curves into thin solids with the **ExtrudeCrv** command. Set the command-line option **Solid=Yes** to **Yes**.





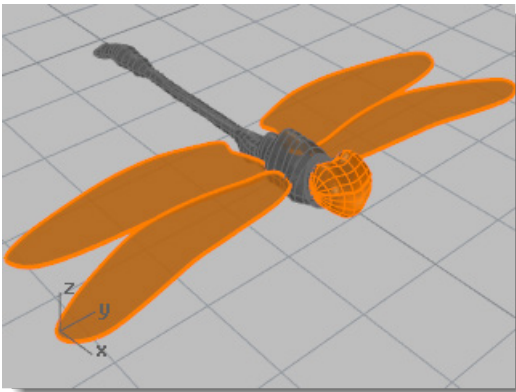
Move the wings into position

- ▶ Position the wings on the back with the **Move** command.
Consult the side view image of the dragonfly. The front wing is slightly higher than the back wing.



Mirror the wings to the other side

- ▶ Use the **Mirror** command to copy the wings to the other side.



Draw the legs



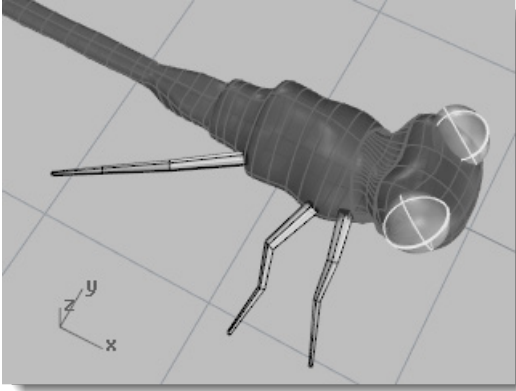
Draw the base polyline

1. In the **Top** viewport, use the **Polyline** command to trace down the center of the legs.
2. Edit the control points to position the legs in the **Top** and **Front** viewports.
You will have to use your imagination a little for this since the two pictures do not show the legs of the same insect.



Pipe the legs

- ▶ Use the **Pipe** command to draw the legs around the polylines.

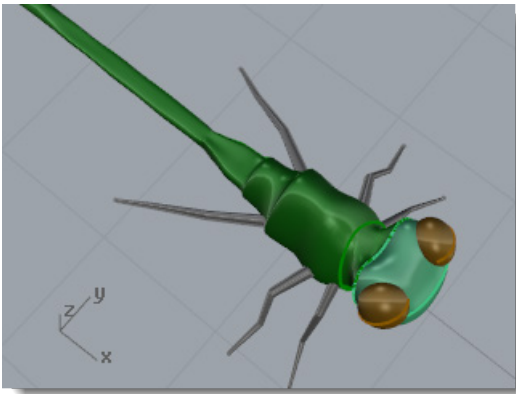


Refer to the background picture to determine the starting and ending diameter of the pipe.



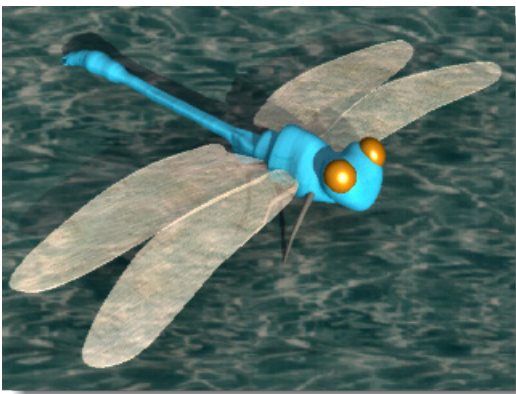
Mirror the legs

- ▶ Use the **Mirror** command to copy the legs to the other side, or draw different legs for the other side.



Finishing touches

- ▶ Add **materials** and **textures** and **render**.



Chapter 19: Wrap Text Tutorial - Flow

This tutorial demonstrates wrapping text solids and other objects on a cylinder. These objects could be used to trim holes in the cylinder.

You will learn how to:

- Create text as solid objects.
- Wrap the objects to a surface.



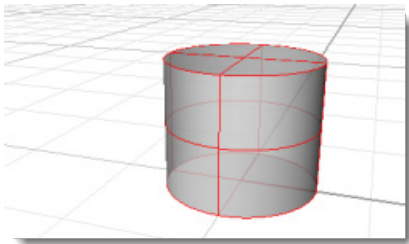
Make a surface

For this example, create a simple cylinder. Once you have learned the basic technique, you will be able to use other types of surfaces. Remember that trimmed surfaces maintain their basic rectangular shape. This underlying shape will affect the placement of the text.



Create a cylinder

- ▶ In the **Top** viewport, use the **Cylinder** command with the **Vertical** option to create a solid cylinder.



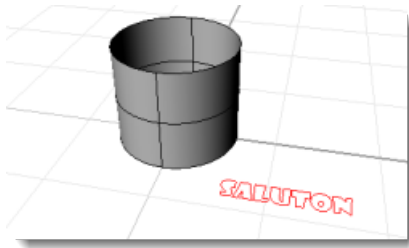
Create the objects to wrap

These solid objects will be wrapped on the cylinder surface.



Create the text

1. Use the **TextObject** command to create your text using **Solids**.
Choose a font that is fairly large and blocky rather than one with many details.
Set the **Height** at about **1.5** units.
Set the **Solid thickness** to **.1** units.
2. Place the text on the construction plane near the cylinder. The location is not important.



Control the object placement

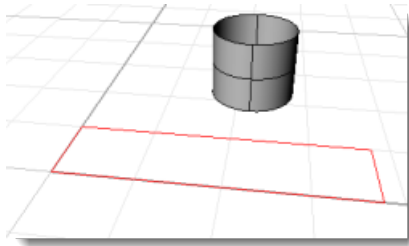
The **CreateUVCrv** command generates the planar border curves of a surface that can be used as a guide to orient your text. Use the border rectangle to lay your text out before re-applying it to the cylinder. The rectangle then is used as a reference to guide the placement of the other objects.



Create the UV curve

1. Use the **CreateUVCrv** command on the cylinder side to create curves that represent the border of the surface on the construction plane.

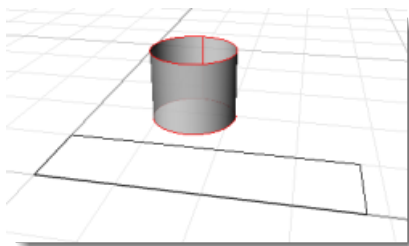
In this case a rectangle is created starting at 0,0 on the **Top** construction plane.



2. Select the cylinder and use the **Properties** command to turn off the isocurve display on the cylinder.

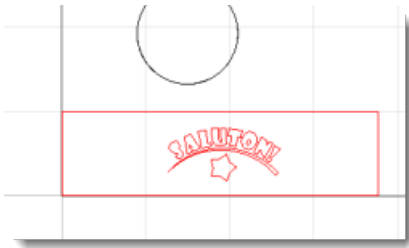
This will show you where the seam of the surface is located. The seam location is important because the rectangle edges match the top and bottom of the cylinder and the seam. Knowing where the seam is will help you visualize how the text will be laid out on the cylinder.

In our example the cylinder is rotated so the seam is toward the back in the view.



Position the text objects

- ▶ **Move**, **Rotate**, and **Scale** the text objects to arrange them inside the rectangle. Add any other decoration curves you want to use.



Create a reference surface

- ▶ Use the **PlanarSrf** command to make the rectangle into a surface.
You will use this surface later as a reference object for the **FlowAlongSrf** command.



Extrude the decoration curves

1. If you have created other curves, select these.
2. Use the **ExtrudeCrv** command to thicken the decorations to match the letters.
3. At the **Extrusion Distance...** prompt, set **Cap=Yes**.
4. At the **Extrusion distance...** prompt, type **.1**.

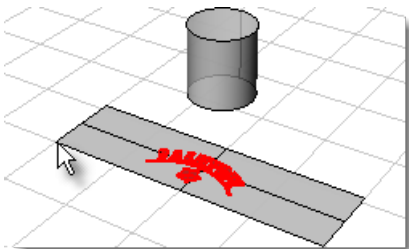


Wrap the objects

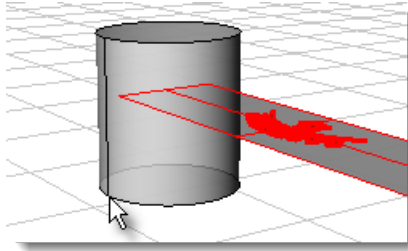


Wrap the lettering on the cylinder

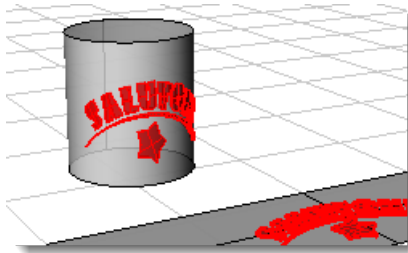
1. Select the lettering and the decoration.
2. Start the **FlowAlongSrf** command.
3. At the **Base surface...** prompt, set **Rigid=No**.
4. Click the **rectangular plane** near the "lower-left" corner as illustrated.



5. At the **Target surface...** prompt, click the cylinder near the lower edge of the seam as illustrated.



The text solids wrap around the cylinder.

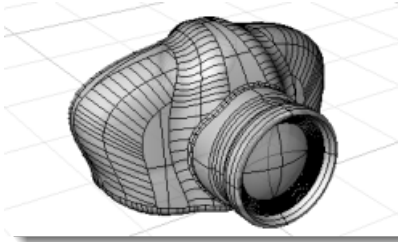


Chapter 20: Camera Tutorial - Blends and Trims

At first glance, this camera model looks complex. But after a little analysis, you'll see that it's made of a three basic blocks, glued together liberally with blends. The three basic blocks are the body, the viewfinder, and the lens housing.

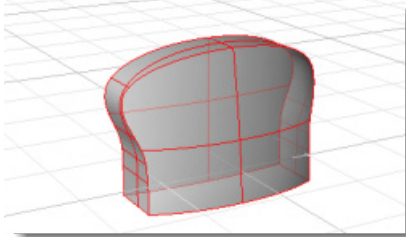


The primary tool used to make this model is the **BlendSrf** command. It creates smooth, curvature-continuous blends between two or more surfaces. This demonstration shows several ways of creating surfaces (and more importantly the gaps between surfaces) that are amenable to blending.



The creation of the model can be broken into seven steps:

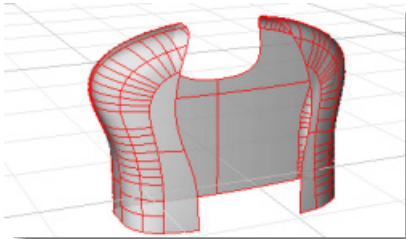
1. Create basic body shape.



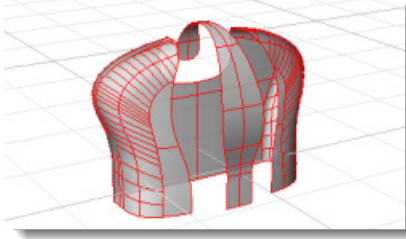
2. Blend the front and back edges.



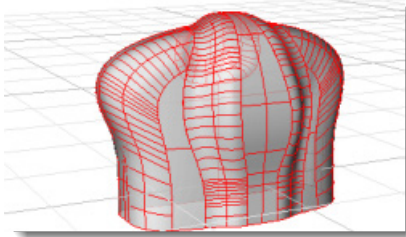
3. Trim a hole in the body for the viewfinder.



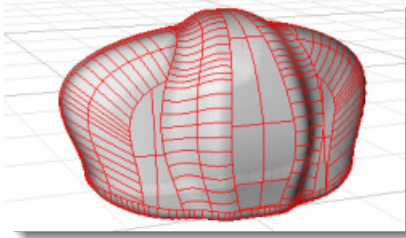
4. Create the viewfinder surface.



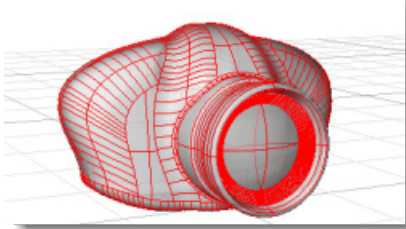
5. Blend between the body and the viewfinder.



6. Boolean bottom surface and blend bottom edge.



7. Create lens and blend to body.



Start the Model

1. From the Rhino **File** menu, click **Open**.
2. Browse to the **Tutorial Models** folder that you downloaded with the *User's Guide*.
2. Open the model file **Camera.3dm**.

The model is organized into layers that follow the steps. You can open this model and follow along with the instructions by turning the layers on and off.

Almost every stage involves the creation of a surface that will later be blended to create the smooth, organic model shown above.

Create basic body shape

The basic body shape is made of three trimmed surfaces. All three are made with extrude commands. The first step in creating these surfaces is to create the curves that define them.

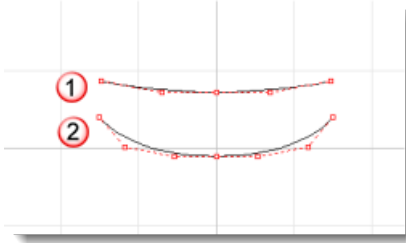
Both the front and back surfaces have a slight curve. The back surface curves in one direction, and is most easily made using the **ExtrudeCrv** command to extrude a curve. The front surface is curved in two directions, so it is made by extruding one curve along another.



Draw the profile curves

1. In the **Top** viewport, use the **Curve** command to draw curves **1** and **2**.

Use the minimum number of control points necessary to create the shape. Keeping control points to a minimum keeps file sizes smaller, makes surfaces smoother, and makes future modeling tasks faster and easier. Notice that the control points are symmetrical. This ensures that the curve is symmetrical. In addition, the middle three control points are lined up parallel to the x-axis. This makes a nice smooth, flat curve that is exactly tangent to the x-axis.



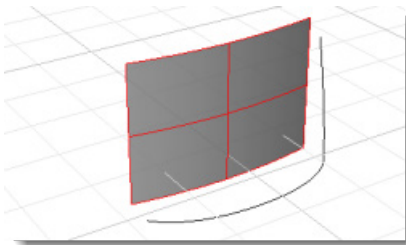
2. Draw curve **3** in the **Right** viewport.

Start it at the endpoint of curve **2** and use **Planar** mode to keep the curve lined up.

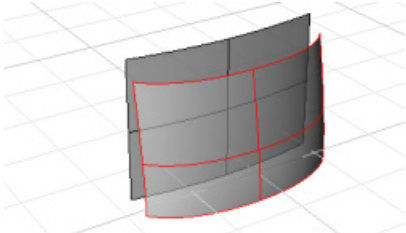


Extrude the front and back surfaces

1. To create the back surface, use the **ExtrudeCrv** command to extrude curve **1** in the z-direction. Estimate the height. Make sure it is taller than curve **3**. The height is not important, since the top will be trimmed off with the side surface.



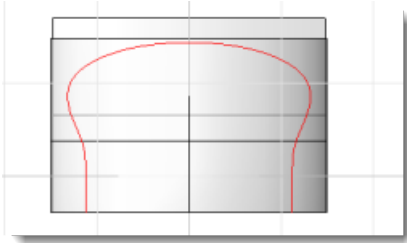
2. To make the front surface, use the **ExtrudeCrv** command to extrude curve **2** along curve **3**.





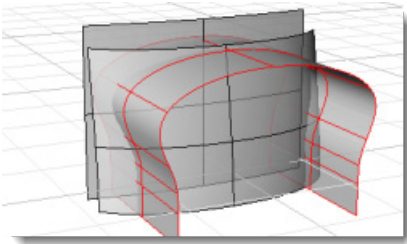
Create profile curve for side surface

- ▶ With the **Curve** command, create a profile curve for the side surface.
Create this curve in the **Front** viewport. Be careful to make the curve start and end exactly at the edge of the surface. If the curve stops short, or extends beyond the bottom of the existing surfaces, the trim used in the next step will fail.



Extrude the side surface

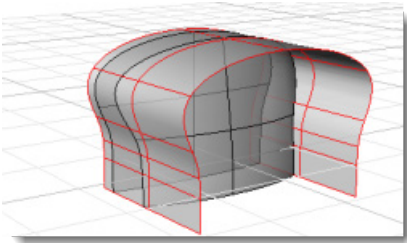
- ▶ With the **ExtrudeCrv** command, extrude the profile curve toward the back.
Make sure it fully intersects both the front and back surfaces. An incomplete intersection will make the trims in the next step fail.



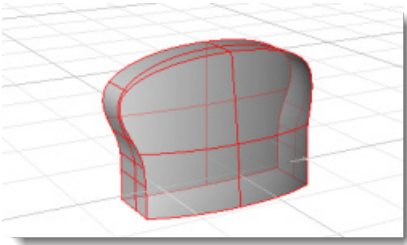
Trim the surfaces

Trim the three surfaces.

1. Use the **Trim** command to trim the front and back surfaces with the extruded side surface.



2. Use the **Trim** command to trim the side surface with the front and back surfaces.



Blend the front and back edges

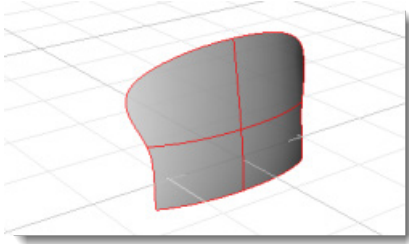
A blend surface matches smoothly (the blend surface has curvature continuity along the edges shared with the other surfaces) between two or more surface edges. Blend surfaces were used for both the front and back edges of the camera body. Two techniques for trimming the surfaces to open up a gap for blending are shown in this step.

The most straightforward and flexible method to create a gap for the first blend is to trim each surface with curves. This method allows the blend surface to vary in width at different points along the blend.



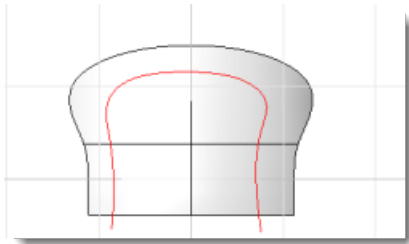
Hide the unneeded surfaces

- ▶ **Hide** the back and side surfaces.



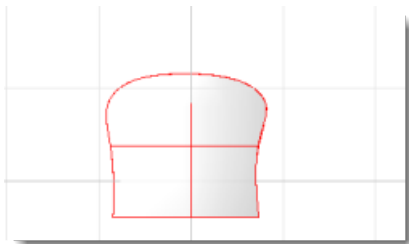
Draw the profile curve

- ▶ In the **Front** viewport, use the **Curve** command to draw a profile curve.



Trim the surface

- ▶ Use the **Trim** command to trim the surface with the profile curve.



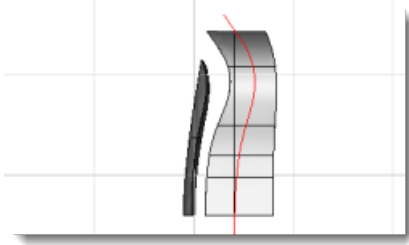
Show the surfaces

- ▶ Use **ShowSelected** command to unhide the side surface.



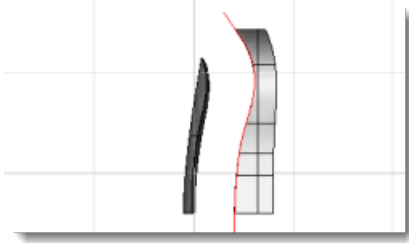
Draw a profile curve

- ▶ In the **Right** viewport, use the **Curve** command to draw a profile curve. This curve is planar and lies on the construction plane in the **Right** viewport.



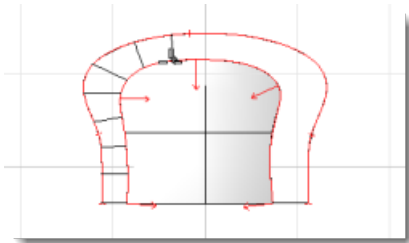
Trim the surface

- ▶ With the **Trim** command, trim the side surface with the profile curve.



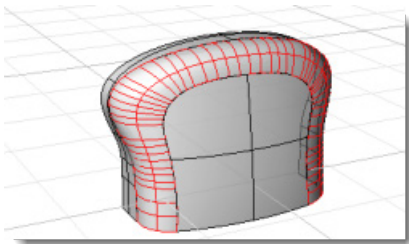
Blend between front and side surfaces

1. Use the **BlendSrf** command to create a surface between the front and side surfaces.
If you simply allow the surface to be created from the defaults, the corners will be rather square and the way the surface travels around the bends is not very smooth.
2. To improve this, delete the original blend and make a new blend.



The **BlendSrf** command lets you control the cross-sections of the blend. In this case set the bulge height to around .7 and then place cross section curves along the opening to control how the surface will move around the curves.

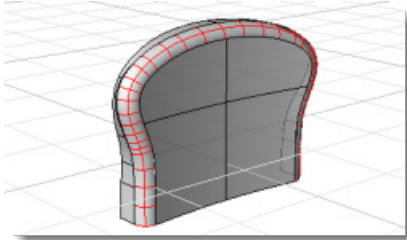
When Rhino creates a blend, it creates a series of sections between the two surfaces to blend. These sections flow smoothly from one surface to the other. The number of sections you need depends on the complexity of the blend—use more cross-sections for more complex blends.



Fillet the surfaces

Another approach for creating a gap to blend is to create a fillet surface. A fillet surface has a constant radius. In the process of creating the fillet surface, both surfaces that are being filleted are trimmed with the new surface. You can then delete the fillet and replace it with a blend. You can also create a variable width blend surface with this method, but it is not as flexible as the previous technique.

1. Use the **FilletSrf** command to create a rolling ball fillet surface between the back surface and the side surface. Use the **Trim=Yes option** and a radius of **0.7**.

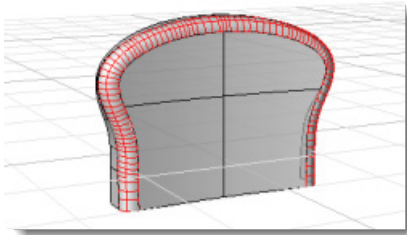


- >2. Delete the fillet surface.



Blend the surfaces

- ▶ Use the **BlendSrf** command to create a blend surface in place of the fillet surface. Place extra cross-sections around the corners.

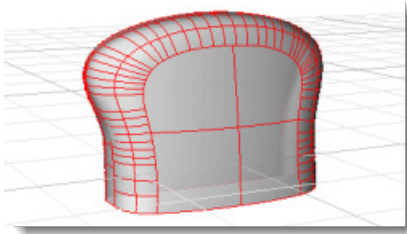


The results may look almost identical in these pictures – and they are, almost – but when the model is shaded and rotated, the blend surface matches the back and side surfaces more smoothly because of the curvature continuity of the blend. Fillets are only tangent to the surfaces, blends are curvature continuous. Try it yourself to see the difference.



Join the surfaces

- ▶ Use the **Join** command to join all the surfaces together into a polysurface.



Trim the body for the viewfinder

The viewfinder bulges out of the body. This bulge houses the viewfinder window and the necessary optical components that let you look through the camera. As with the rest of the camera, the viewfinder should blend in smoothly with the rest of the body.

The blend is made with the same approach used to create the front surface blend: trim both parts and create a blend surface between them. Since the camera body is actually a polysurface, and the blend fills a more complex hole, more steps are required to create the blend.

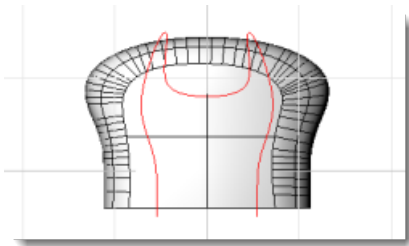
The viewfinder will wrap around the top of the body. This means that the hole in the body needs to wrap around the top, too.



Draw a profile curve

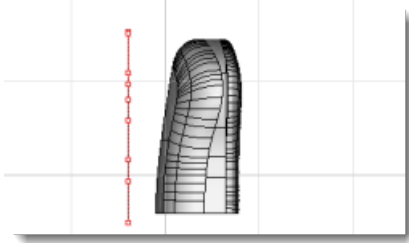
- ▶ With the **Curve** command, draw a rough approximation of the hole the **Front** viewport. Draw the curve symmetrical about the y-axis.

Ensure the symmetry by drawing half the curve, mirroring it about the y-axis, and joining the two sides. The last two control points (the ones at the end where the two halves meet) are lined up horizontally to ensure the curve doesn't have a kink when mirrored.

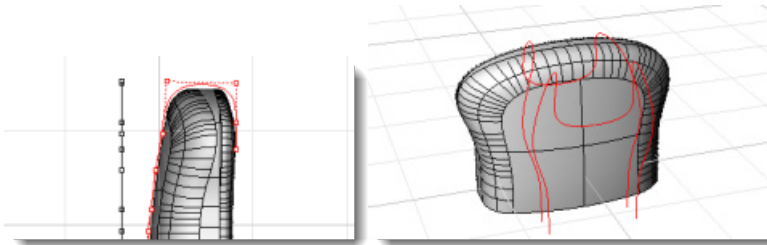


Edit the curve

1. Press **F10** to turn on control points and move them to get the curve to wrap around the surface.

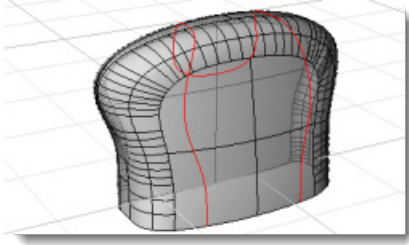


2. Drag the control points in the **Right** viewport. Turn on **Ortho** so that the control points only drag parallel to the world y-axis.



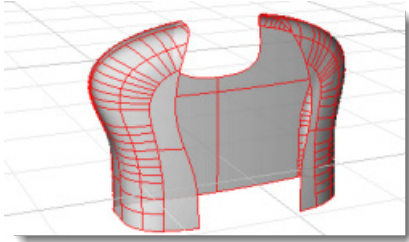
Pull the trimming profile curve onto the body surfaces

1. Use the **Pull** command to pull the curve to each surface separately, resulting in a series of curves.
2. Delete extra curves until you have a series of curves around the surfaces that match the original curve as illustrated.



Split the body parts with the curve

1. Use the **Split** command to split each surface with the pullback curves.
2. Delete the unnecessary geometry as illustrated.



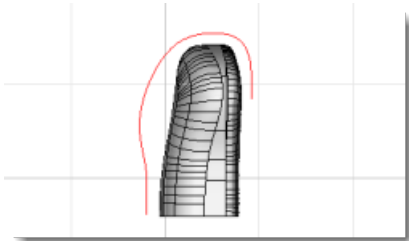
Create the viewfinder

The next step in creating the viewfinder is to create the main shape of the viewfinder surface. It is a simple extruded surface trimmed to complement the hole in the body.



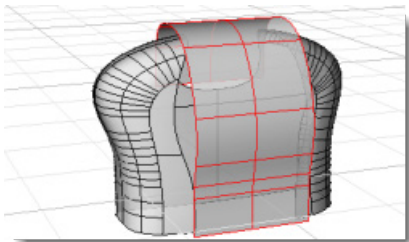
Create the profile curve

- ▶ In the **Right** viewport, use the **Curve** command to draw a profile curve for the viewfinder surface.



Extrude the profile curve

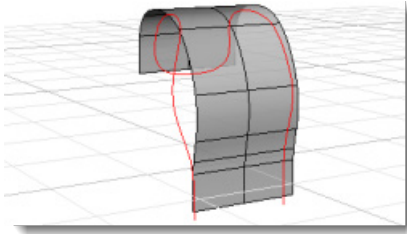
- ▶ Use the **ExtrudeCrv** command to extrude the curve in both directions from the center profile.



Scale the curve

To create the viewfinder trimming curve, start with the curve that was pulled back to the camera body.

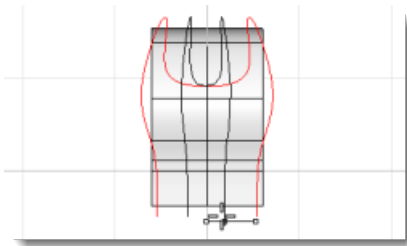
1. Use the **Scale1D** command to scale the curve several times to get the curve roughly the right shape.



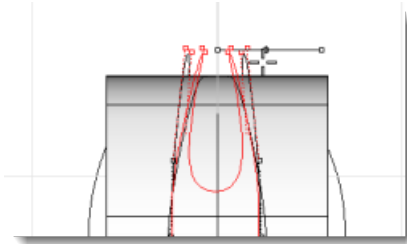
Scale the curve vertically in the **Right** viewport.



Scale the curve horizontally in the **Front** viewport.

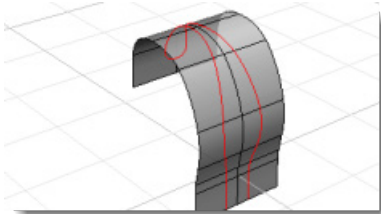


2. The final shape is achieved with control point editing.



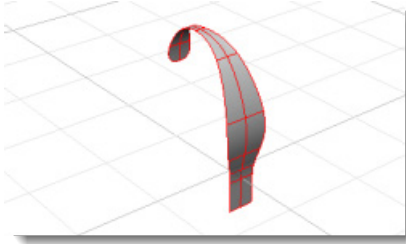
Pull the curve to the surface

- ▶ With the **Pull** command, pull the resulting curve back to the surface.



Trim the surface

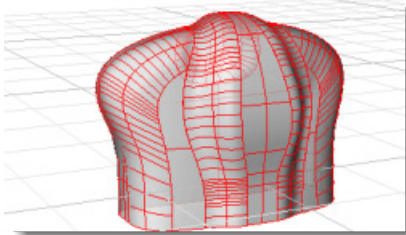
- ▶ Use the **Trim** command to trim the surface with the curve.



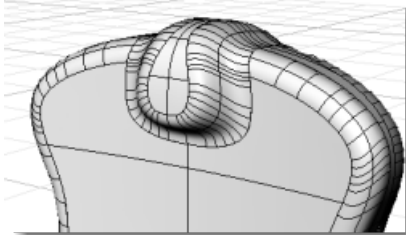
Blend between the body and viewfinder

Blending between the body and the viewfinder surface is more difficult than the blends between the front, back and sides because it follows such a complex path.

1. Start the **BlendSrf** command.
2. Select all the edges on the body surface (in order), then select all the edges for the viewfinder.



3. Add enough cross sections so that the transitions around the tight turns at the back are smooth.

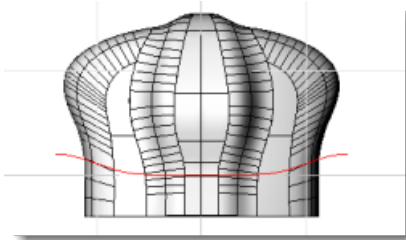


Create bottom of camera

At this point, the bottom of the camera is open. To close it, draw a curve that represents the bottom, extrude it, and use Boolean intersection to trim and join the surfaces.

Draw the profile curve

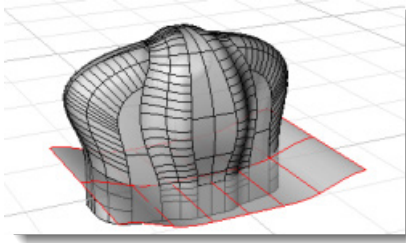
- ▶ In the **Front** viewport, draw a profile curve with the **Curve** command.





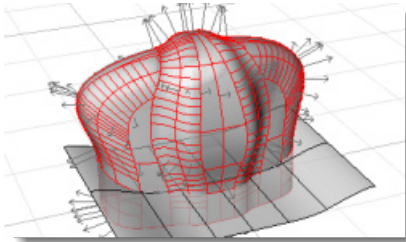
Extrude the profile curve

- ▶ With the **ExtrudeCrv** command, extrude this profile curve beyond the front and back of the camera.

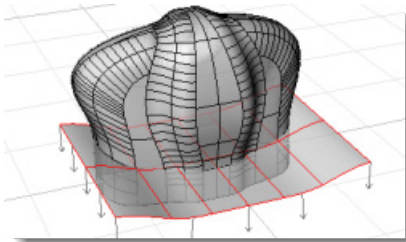


Check the surface direction

- ▶ With the **Dir** command, check the direction of the surfaces to ensure the body points out, and the bottom points down.

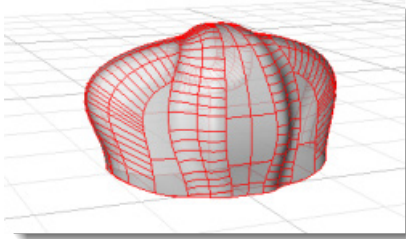


Use the **Flip** option to correct the direction if necessary.



Boolean the parts

- ▶ Use the **BooleanIntersection** command to trim and join the two surfaces in one step.



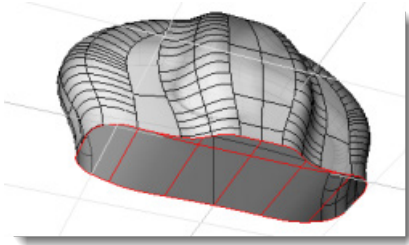
Create bottom edge blend

We have used trimming and filleting to create a gap between surfaces for blending. A third technique for creating a gap for a blend is to create a pipe around the edge, split the surfaces with the pipe, and blend between them. This usually gives a slightly different result than the fillet technique.



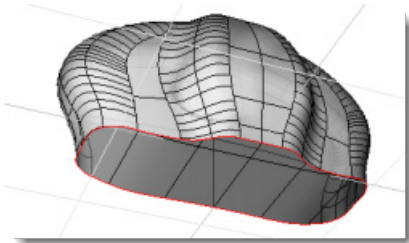
Extract the bottom surface

- ▶ To create the pipe, with the **ExtractSrf** command, extract the bottom surface from the polysurface.



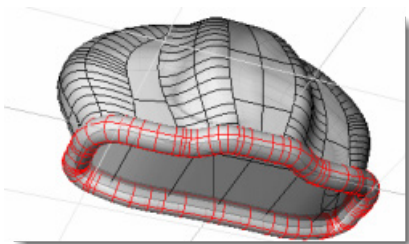
Duplicate the surface border

- ▶ Use the **DupBorder** command to create a single closed border curve. This creates a curve that can be used to create the pipe.



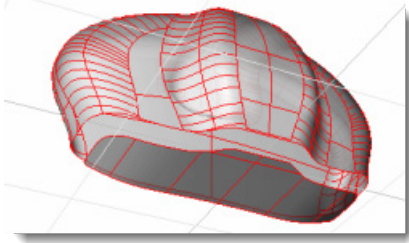
Pipe the curve

- ▶ With the **Pipe** command, create a pipe surface around the duplicated border. Use a radius of **0.5**.



Split the surfaces

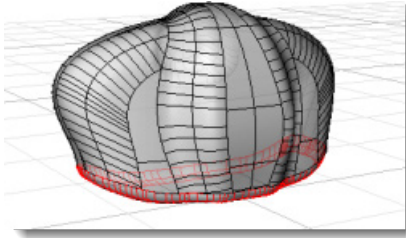
1. With the **Split** command, trim the body and the bottom with the pipe.
2. Delete the unnecessary pieces, including the pipe.





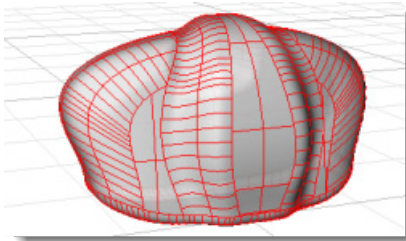
Blend surface on bottom surface

- ▶ With the **BlendSrf** command, create a blend to fill the gap.



Join the parts

- ▶ With the **Join** command, join the parts together.



Create the lens

The last step is to create the lens and to blend the surface between them.



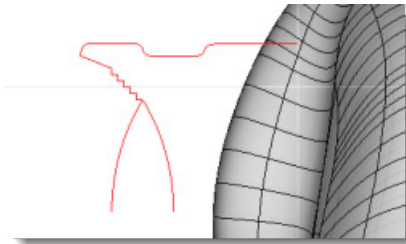
Create the lens profile curve

- ▶ With the **Polyline** command, draw the upper half of lens profile curve.



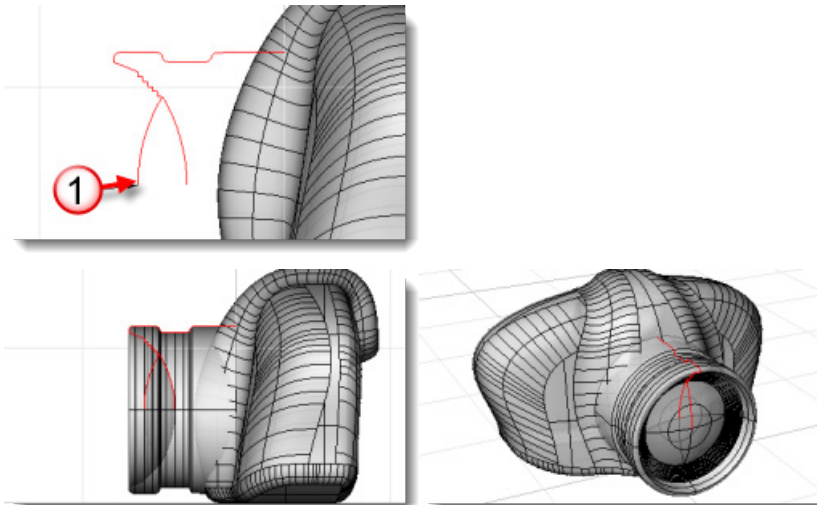
Round some corners

- ▶ With the **Fillet** command, fillet the polycurve in a few places to round of some of the sharp corners.



Revolve the curve

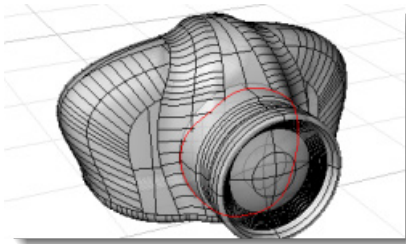
- ▶ Use the **Revolve** command to create a revolved surface from the profile curves. Snap to end **1** as illustrated for the beginning of the revolve axis. Use **Ortho** to make the revolve axis parallel to the world y-axis.



Create an intersection curve

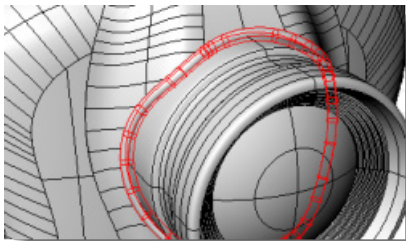
Blend between the body and the lens the same way the bottom edge was blended.

- ▶ With the **Intersect** command, create the intersection curve between the body and the lens surface.



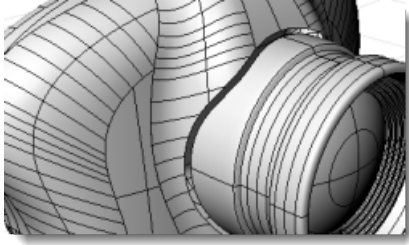
Pipe the intersection curve

- ▶ With the **Pipe** command, create a pipe surface around the intersection curve with a radius of **0.15**.



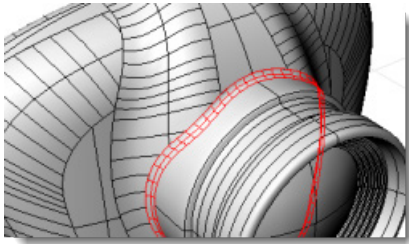
Split the body and lens

1. **Split** the lens and body with the pipe.
2. **Delete** the pipe and the extra surfaces.



Blend between the body and lens

- ▶ With the **BlendSrf** command, fill the gap with a blend surface between body and lens.



Chapter 21: Mechanical Part Tutorial - Layouts

This example shows how to use Rhino to model a simple mechanical part.

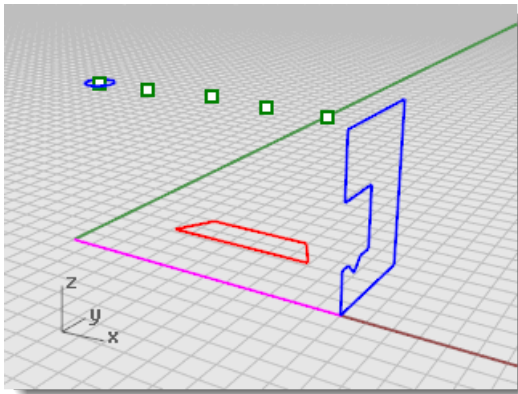
You will learn to

- Create extrusion objects.
- Boolean difference shapes.
- Drill holes.
- Create a 2-D line drawing from the 3-D shapes.
- Dimension the 2-D line drawing and modify dimension text.
- Create a layout with a title block for printing.



Open the tutorial model

1. From the Rhino **File** menu, click **Open**.
2. Browse to the **Tutorial Models** folder that you downloaded with the *User's Guide*.
3. Click **Toolblock.3dm** and then click the **Open** button.



Create solid shapes

Start by creating two basic solid shapes from the profile curves on layers Profile-01 and Profile-02.



Set up the layers

- ▶ In the **Layers** panel, confirm that the **Profile-01** layer is current.



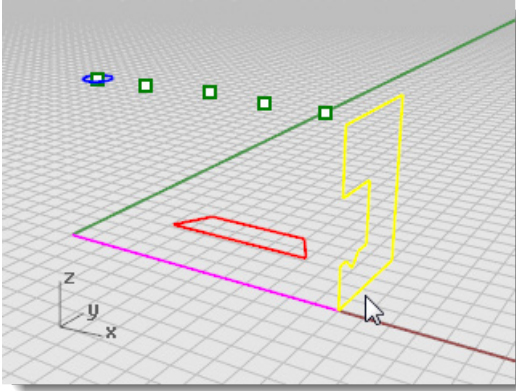
Turn on object snaps

- ▶ Turn on the **End** object snap.

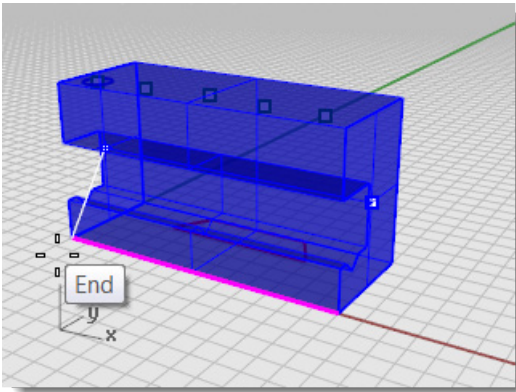


Extrude the profile curve

1. On the **Solid** menu, click **Extrude Planar Curve > Straight**.
2. At the **Select curves to Extrude** prompt, select the blue profile curve, and press **Enter**.



3. At the **Extrusion distance** prompt, set the command-line **Solid** and the **DeleteInput** options to **Yes**.
4. Click the end of the magenta construction line.



The extruded shape is a solid because it forms a closed volume in space.



Hide the solid

- ▶ **Select** the solid, on the **Edit** menu click **Visibility > Hide**.



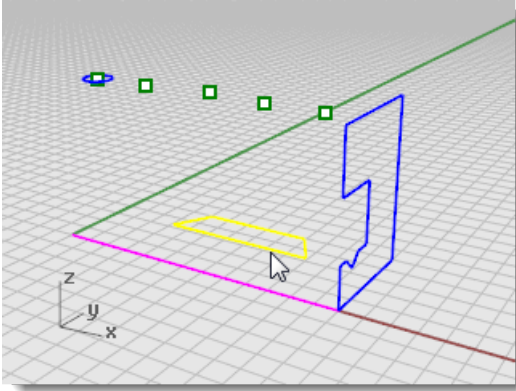
Set up the layers

- ▶ In the **Layers** panel, make layer **Profile-02** current.

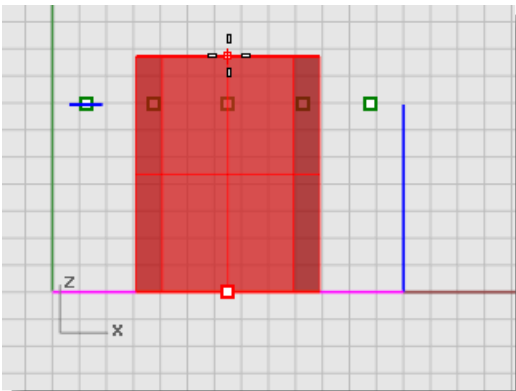


Extrude the profile curve

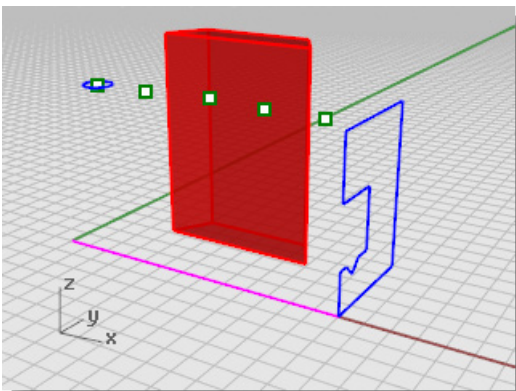
1. **Select** the red profile curve.



2. On the **Solid** menu, click **Extrude Planar Curve > Straight**.
3. At the **Select curves to Extrude** prompt, set the command-line **Solid** and the **DeleteInput** options to **Yes**.
4. **Select** the red profile curve, and press **Enter**.
5. At the **Extrusion distance** prompt, in the **Front** viewport, drag the extrusion above the height of the blue curve and click.

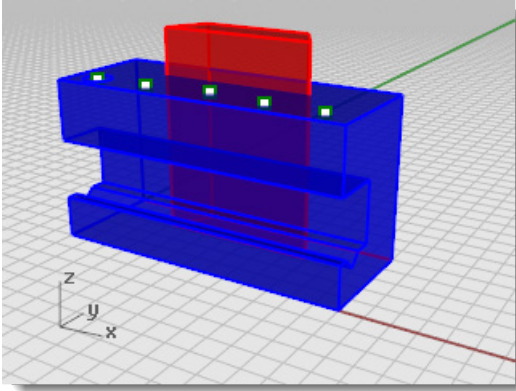


The solid appears on the current red layer Profile-01.



Show the solid

- ▶ On the **Edit** menu click **Visibility > Show**.



Set up the layers

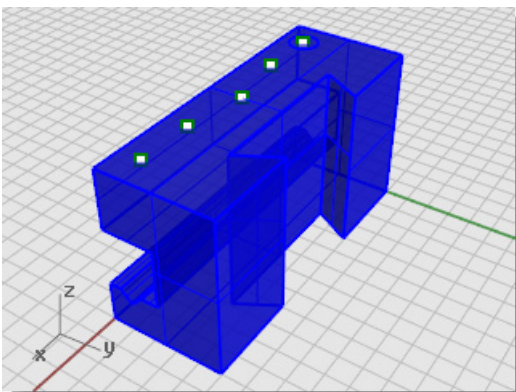
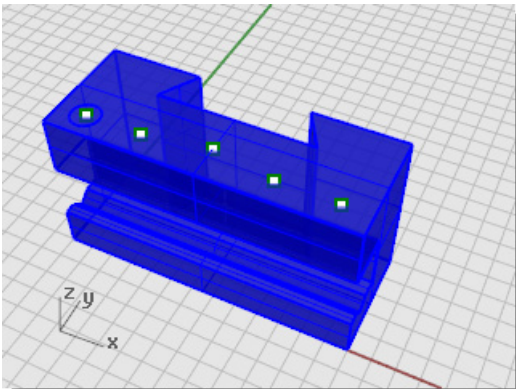
- ▶ In the **Layers** panel, make layer **Profile-02** current.



Boolean the two solids

1. On the **Solid** menu, click **Difference**.
2. At the **Select surfaces or polysurfaces to subtract from** prompt, select the blue solid, and press **Enter**.
3. At the **Select surfaces or polysurfaces to subtract with** prompt, set the command-line **DeleteInput** option to **Yes**.
4. **Select** the red solid, and press **Enter**.

The result will be a new solid or polysurface. A polysurface is a collection of surfaces that can be closed or open. A solid is a collection of surfaces that is closed.



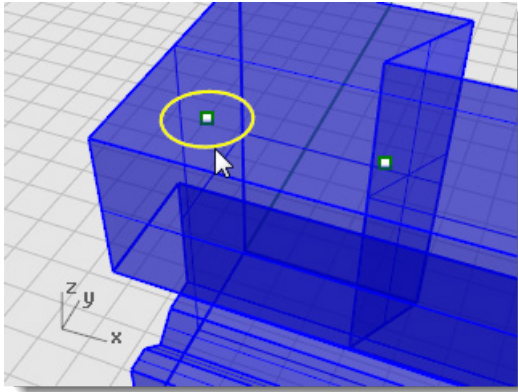
Drill the holes

A construction circle is already in place for creating the first hole.

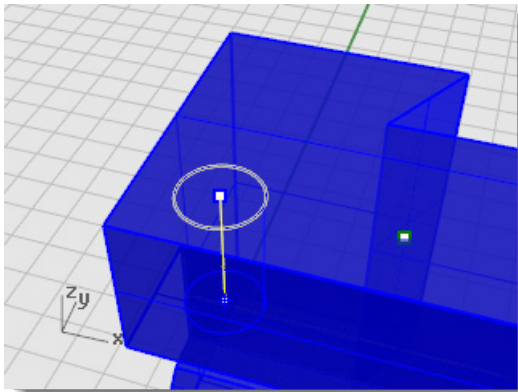


Make holes in the solid

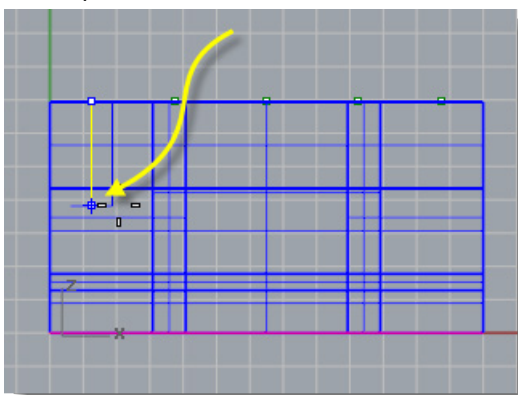
1. **Select** the green circle as shown.

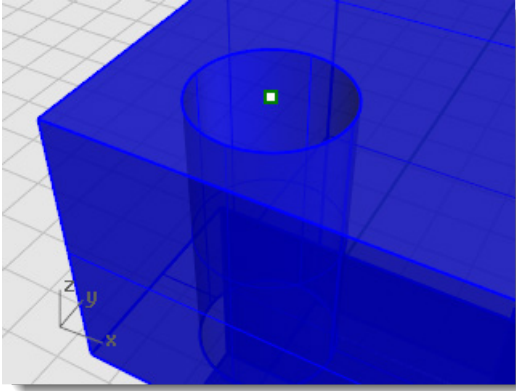


2. On the **Solid** menu click **Solid Edit Tools > Holes > Make Hole**.
3. At the **Select a surface or polysurface** prompt, select the blue polysurface.
4. At the **Cut depth point** prompt, drag the hole through the upper portion of the object.



5. Pick a point in **Front** view.

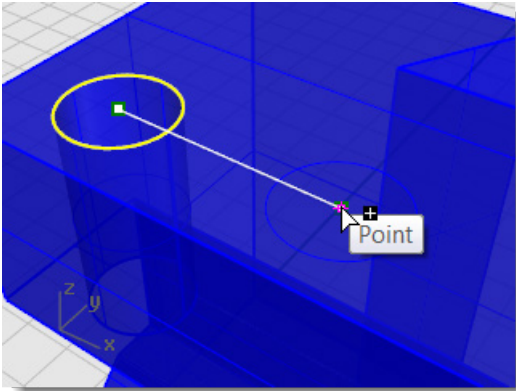




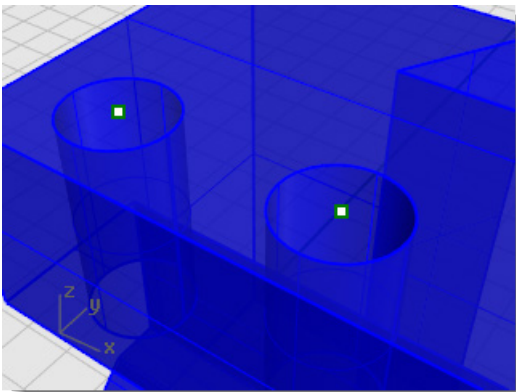
Copy the holes

Copy the three remaining holes that are aligned with the previous hole with this command.

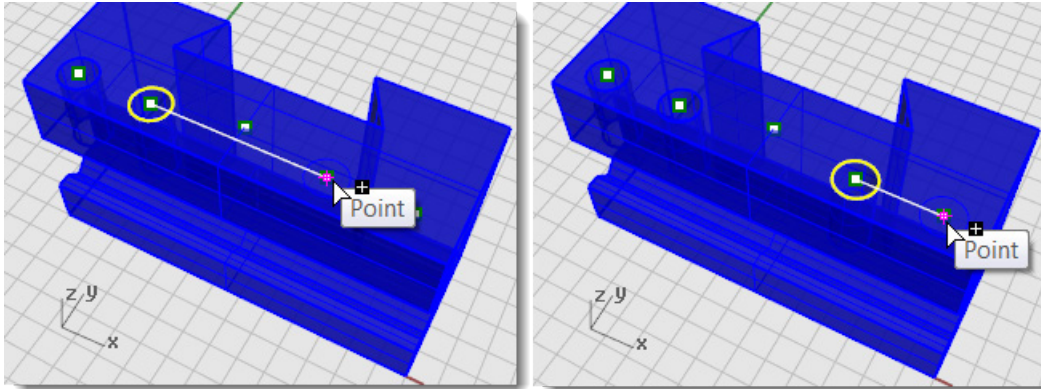
1. In the **Osnap** toolbar, turn on the **Point** object snap.
2. On the **Solid** menu, click **Solid Edit Tools > Holes > Copy Holes**.
Note: **Copy Holes** is actually the **MoveHole** command with the command-line **Copy** option set to **Yes**.
3. At the **Select holes in one planar surface** prompt, select the first hole, and press **Enter**.



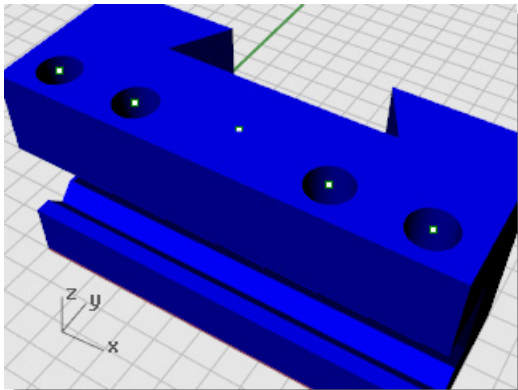
4. At the **Point to copy from** prompt, pick the point object in the center of the first circle.
5. At the **Point to copy to (Copy=Yes):** prompt, pick the point that makes the center of the next hole.



7. Repeat this for the two holes that are on the other side of the part.



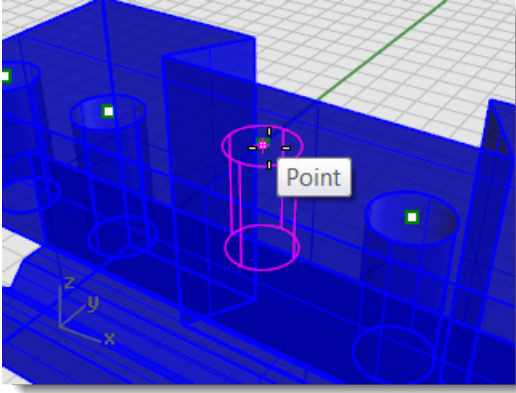
Note: Do not use the point in the center of the part.



Create the round hole

The center hole is different in that it does not pass entirely through the upper part of the blue solid. There is no reference circle to start from.

1. On the **Solid** menu, click **Solid Edit Tools > Holes > Round Hole**.
2. At the **Select target surface** prompt, select the top surface of the blue solid.
3. At the **Center point** prompt, set the command-line options as follows:
 - Depth=0.5**
 - Diameter=0.312**
 - DrillPointAngle=180**
 - Through=No**
 - Direction=CPlaneNormal**
4. Click the point object in the middle of the blue solid to finish creating the hole.



Test the solid

The resulting polysurface is a closed solid. A solid defines a closed volume in space. The **Properties** command will report if this part is a closed solid. The **Properties** command will give you information about the open/closed status of the object.

1. **Select** the part.
2. On the **Edit** menu, click **Object Properties (F3)**.
3. In the **Properties** panel, click the **Details** button.

In the **Object Description** window, you will find the listing to confirm that the object is valid and closed.

```
Geometry:
  Valid polysurface.
  Closed solid polysurface with 23 surfaces.
```

Make a 2-D drawing

The **Make2D** command generates 2-D lines from the 3-D solid.



Create a 2-D line drawing

1. **Select** the part.
2. On the **Dimension** menu, click **Make 2D**.
3. In the **2-D Drawing Options** dialog box, under **Drawing layout** click **4 view (USA)**.
Under **Options**, check the **Show tangent edges** and **Show hidden lines** boxes.
4. Click **OK**.

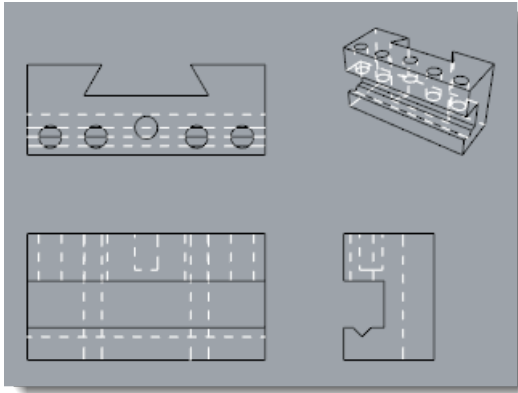
Dimension the 2-D drawing

Using the 2-D drawing, add dimensions for the part.



Set up the layers

- ▶ In the **Layers** panel, make **Dimensions** layer current.
1. In the **Layers** panel, turn off all layers except **Dimensions** and the **Make2D** layers.
 2. In the **Linetype** column for the **Make2D > hidden > lines** layer, click **Continuous**.
 3. In the **Select Linetype** window, select **Dashed**.



Set up the viewport

- ▶ Double-click the viewport title to maximize the **Top** viewport.

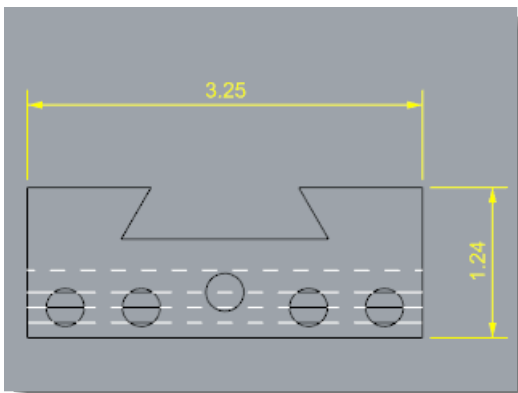
Set up the object snaps

- ▶ In the **Osnap** toolbar, turn on the **End** object snap; turn off the **Point** object snap



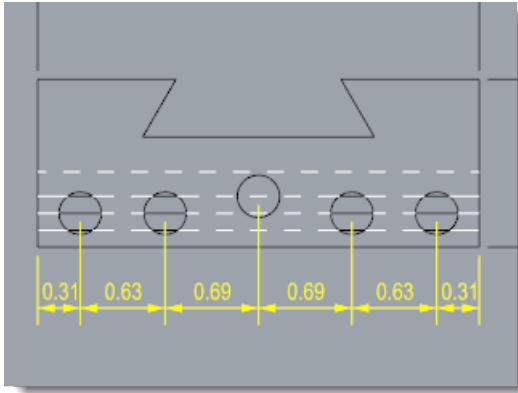
Dimension the part

3. On the **Dimension** menu, click **Linear Dimension**.
4. At the **First dimension point** prompt, pick the upper left end put of the part.
5. At the **Second dimension point** prompt, pick the upper right corner of the part.
6. At the **Dimension location** prompt, pick a location for the dimension line.
7. Repeat to generate a vertical dimension on the right side of the part.



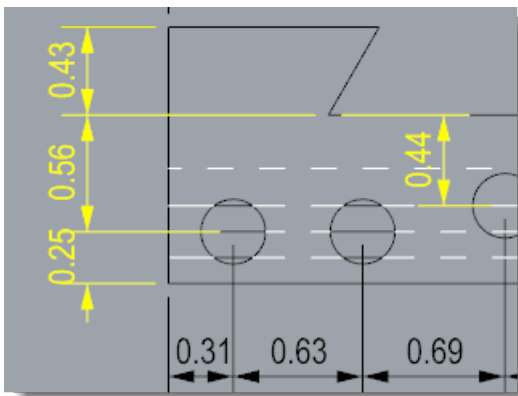
Chain dimension the holes horizontally

1. On the **Dimension** click **Linear Dimension**.
2. At the **First dimension point** set the command-line **Continue** option to **Yes**. This will generate a chain of dimensions.
3. At the **First dimension point** prompt, pick the lower left corner of the part.
4. At the **Second dimension point**, prompt turn on the **Cen** object snap and pick the center of the first circle.
5. At the **Dimension location** prompt, pick below the part.
6. At the next **Dimension location** prompts, continue picking the centers of the circles.
7. Finish by picking the lower right corner of the part, press **Enter**.



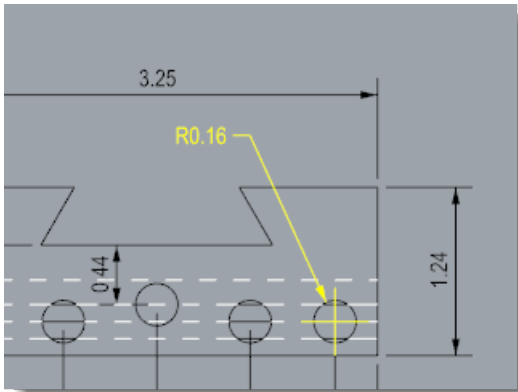
Chain dimension the holes vertically

- ▶ Repeat the chain dimensions to create vertical dimensions.

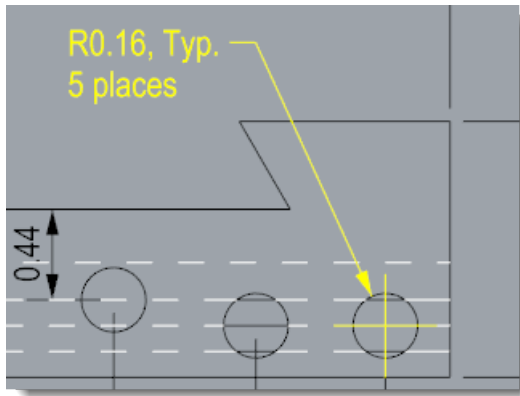


Add radial dimensions

1. On the **DIMENSION** menu, click **Radial Dimension**.
2. At the **Select curve for radius dimension** prompt, select the hole on the far right.
3. At the **Dimension location** prompt, pick above the part.



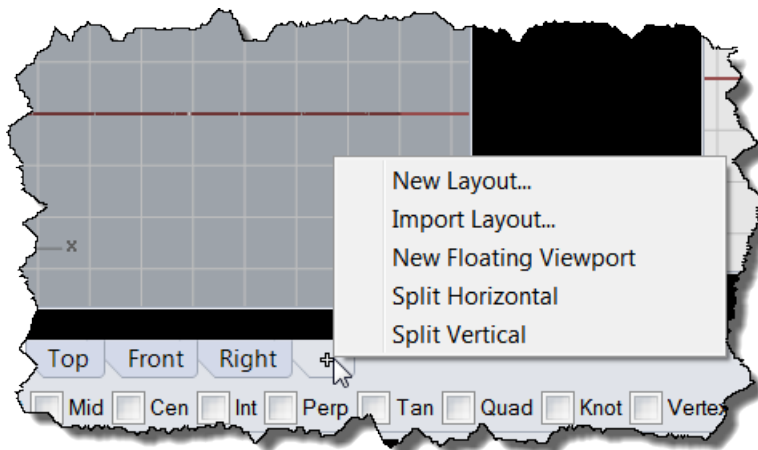
4. Double click the radial dimension text, and in the **text edit** box, add the text **Typ. 5 Places**.



Add a layout viewport

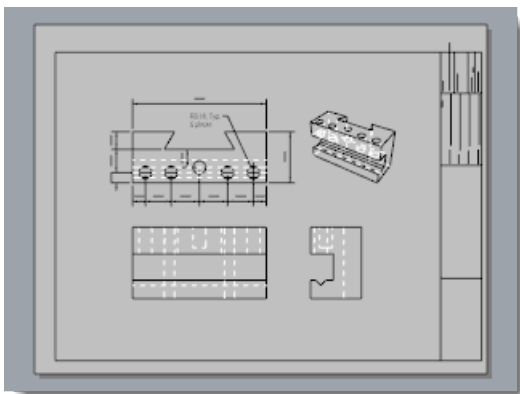
Use a layout to arrange the 2-D lines for printer.

1. On the **Viewport tabs**, click the **Add new tab (+)** icon.
2. On the menu, click **New Layout**.



2. In the **New Layout** window, select the following:
 - Landscape**
 - Width=11 inches**
 - Height=8.5 inches**
 - Initial Detail Count=1**

Page 1 will appear in the viewport tabs.



Insert a title block

In this section you will add a title block and resize the detail to fit.



Set up the layer

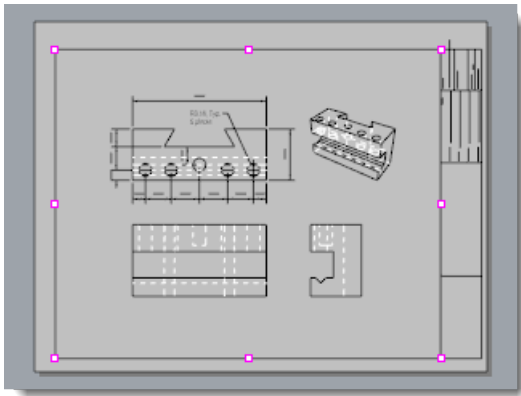
1. In the **Layers** panel, make the **Title Block** layer current.
2. Confirm that you are in layout space.



Insert the title block

1. On **Edit** menu, click **Block > Insert Block Instance**.
2. From the name list, select **Ansi_A_TB**.
3. Under **Insertion point**, clear the **Prompt** checkbox, and click **OK**.
4. **Select** the detail.

Note: Click on the detail edge to select it. Make sure the detail is not active.
5. On the **Edit** menu click **Control Points > Control Points On (F10)**.



6. Drag the control point in the lower left corner to resize the detail to fit in the title block.
7. Press the **Esc** key to turn off the control points.



Lock the detail view

1. **Select** the detail.
2. On the **Edit** menu, click **Object Properties (F3)**.
3. In the **Properties** panel on the **Detail** page, under **Scale**, set the **inches on page** and **inches in model** to **1.0**.
4. Check the **Locked** box

This prevents future panning and zooming so the details stays the correct size.



Add print widths

1. In the status bar, right-click the **Layer pane** to open the **Layer** panel.
2. Click **Default** in the **PrintWidth** column, and set the values to the following:

Dimension = .35.

Title Block = .5.

Make2D > visible > lines = .70.

The layout view displays a preview of the print widths as they will appear on the printed page.

