

Rhinoceros[®]

NURBS modeling for Windows

Version 1.0
Introduction Guide

IntroductionToRhino.doc

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1 Introduction

Rhinoceros is a 3-D NURBS modeling program for Windows. With Rhino you can model anything from a heart valve to a ship hull and from a mouse to a monster. Rhino provides a flexible, accurate, and fast working environment. You can model and render objects that you could previously create only using software and hardware many times more expensive.

Rhino is easy to learn and use. With Rhino, you can create free-form curves, surfaces, and solids. You are free to create the model any way you want.

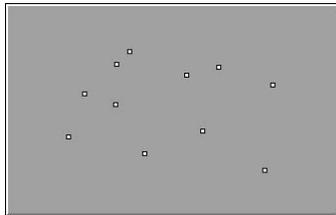
NURBS

Non-uniform rational B-splines (NURBS) geometry is a mathematical representation that can accurately define any shape from a simple line, circle, arc, or box to the most complex 3-D free-form organic surface or solid.

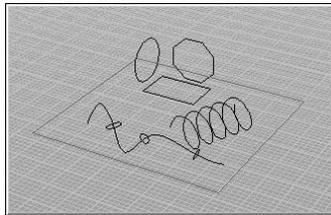
Because of their flexibility and accuracy, NURBS models can be used in any process from illustration and animation to manufacturing.

Rhino's Geometry Types

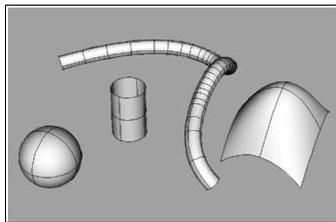
There are five fundamental geometric objects in Rhino: points, NURBS curves and surfaces, and polygon meshes.



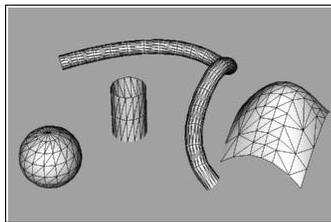
Points.



NURBS curves.



NURBS surfaces.



Polygon meshes.

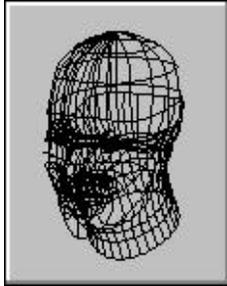
Polygon mesh objects are used by some programs for rendering and animation, stereolithography, VRML, and finite element analysis to approximate a smooth surface. Rhino can create polygon mesh objects that approximate the NURBS objects to export to those programs.

For additional information on Rhino features go to page 66.

2 Your First Look

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. When you have a view you like, you can render the view. By rendering you can add color, texture, shadows, and lights.



Wireframe.



Shaded preview.

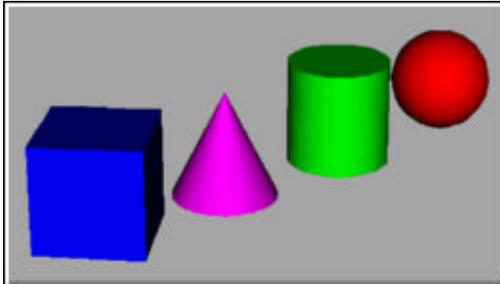


Rendered.

If you have experience using a 3-D modeling program, the following exercises may seem simple, but they will help you understand Rhino's interface and navigation tools.

If you have never worked with 3-D computer modeling before, *Getting Started with 3D, A Designer's Guide to 3D Graphics and Illustration*, by Janet Ashford and John Odam is one of several excellent books that will help you learn some of the terminology and basic principles of 3-D design on computers.

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



The first exercise model.

Rhino Commands

Rhino uses *commands* to accomplish various tasks. As you work through the exercises, you will learn what commands do, when and how to use them and how to decide which command to use to accomplish your task.

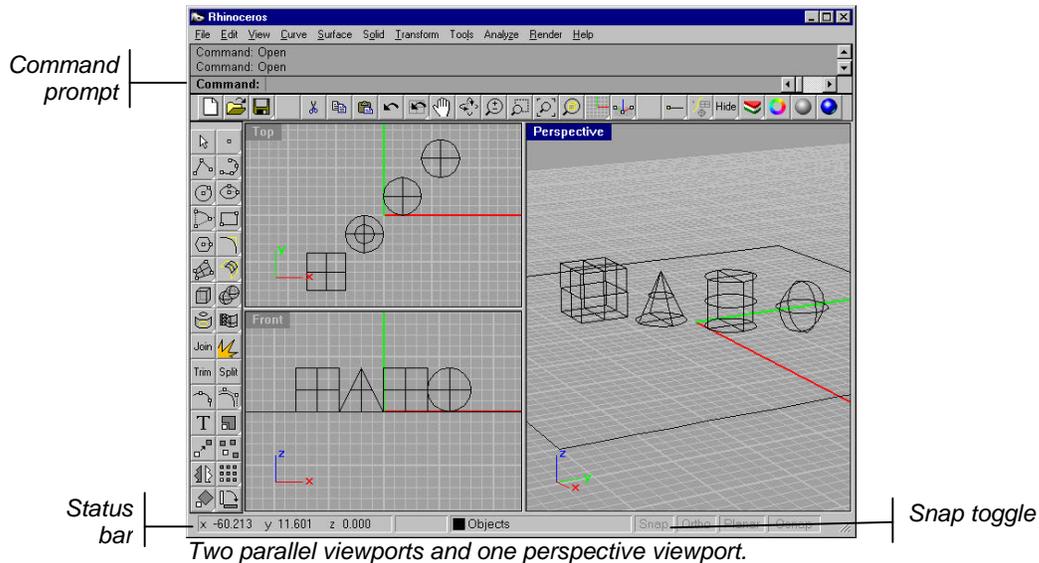
Start Using Rhino

In the first exercise you will learn how to:

- Shade the model
- Rotate, pan, and zoom in wireframe and shaded preview
- Drag objects to move them

To look at your first Rhino model:

- 1 Start **Rhino**.
- 2 From the **File** menu, click **Open**.
- 3 In the Open dialog box, in the **Tutorials** folder, select **First Model.3dm**.



Two parallel viewports and one perspective viewport.

This model contains five objects: a cube, a cone, a cylinder, a sphere, and a rectangular plane.

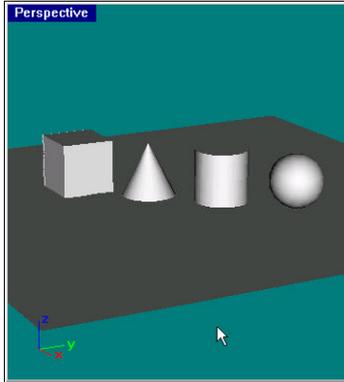
You won't be able to select or move the plane. You will learn about how to do this later. It is just there to provide you with a floor underneath the objects.

- 4 Click the mouse in the **Perspective** viewport to make it active.

An active viewport is the viewport where all your commands and actions take place. The active viewport's title highlights so you can easily see which viewport is active.

- 5 From the **Render** menu, click **Shade**.

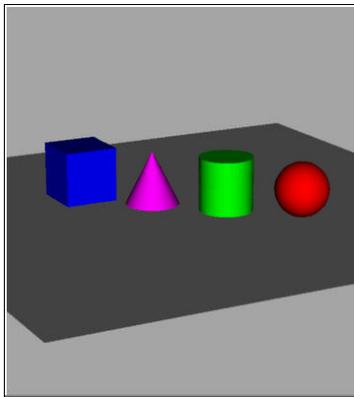
The objects appear shaded. Shade lets you preview the shapes with a surface instead of a mass of curves. Previewing with no color also lets you focus on the geometric shape of the object instead of color and texture. The model is shaded in the viewport. You can still manipulate the view in shaded mode. In addition, Rhino provides rendering with color and texture.



Shaded preview.

- 6 From the **Render** menu, click **Render**.

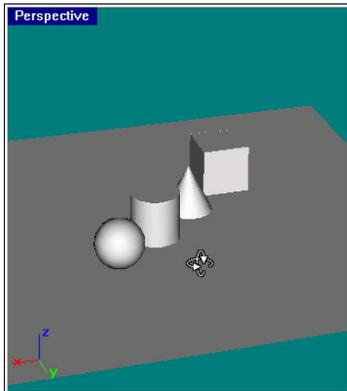
When you render the model, you can see the color and texture that has been applied to the objects. Rendering opens a separate display window. You cannot manipulate the view in the render display window. In addition to assigning color and texture to the objects, you can set lights and a background. Rendering is slower than the shaded preview.



Render.

- 7 Close the render Display Window
- 8 From the **Render** menu, click **Shade**.

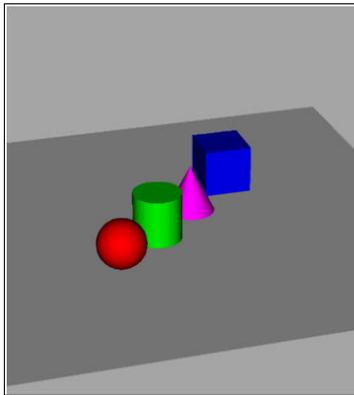
- 9 In the **Perspective** view, click and drag with your right mouse button held down to rotate the view.



Rotate the view in shaded mode.

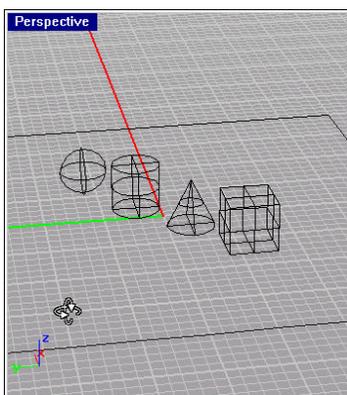
The plane helps you stay oriented. If the objects disappear, you are looking at the bottom of the plane.

- 10 From the **Render** menu, click **Render**.



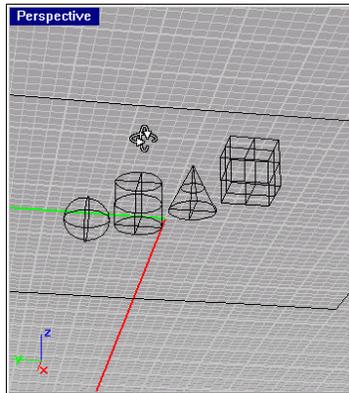
Render the rotated view.

- 11 Close the render Display Window.
- 12 In the large **Perspective** viewport on the right, click and drag with your right mouse button to rotate the view around.



Rotating the wireframe view.

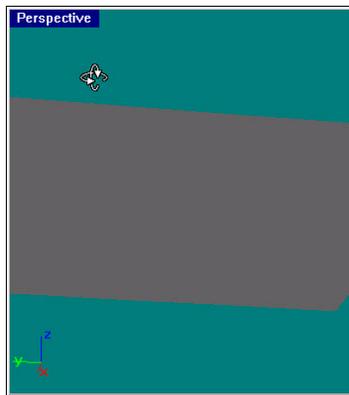
- 13 Rotate your view by dragging from the bottom of the view toward the top.



Looking at the objects from the bottom in wireframe mode.

You are now under the objects looking up.

- 14 From the **Render** menu, click **Shade**.



Looking at the objects from the bottom in shaded mode.

In shaded mode, the plane helps you see when your viewpoint is below the objects.

To get back to your original view:

- ◆ Press the **Home** key to undo your view changes.

Try on Your Own

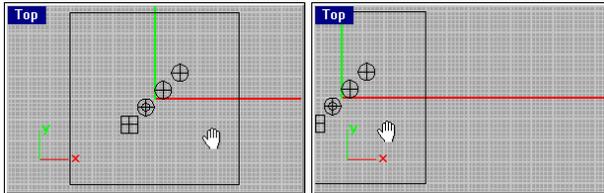
- Click in other viewports and shade them.
- Render the other viewports.
- What happens when you drag with your right mouse in other viewports?
- Dragging the right mouse in the viewports that do not have a perspective view pans instead of rotating. You will learn more about navigating around in Rhino as we go along.
- You can change the way Rhino rotates the view. See View Options in Rhino Help.

Navigating Around the Model

You have used the right mouse button to rotate in the Perspective viewport. You can drag with the right mouse button to pan. Dragging the right mouse button to manipulate the view does not interrupt any commands you are using.

To pan in a viewport:

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



Panning with **Shift** and the right mouse button.

Zooming In and Out

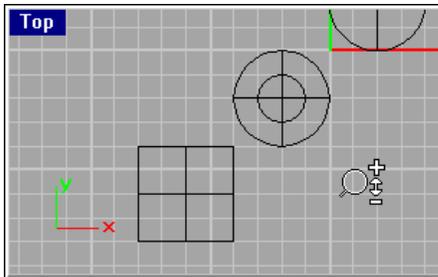
Sometimes you want to get closer to your objects or move back so you can see more. This is called **zooming**. As with many things in Rhino, there are several ways to do this. The easiest way is to hold down the **Ctrl** key and drag up and down in a viewport with the right mouse button. If you have a mouse with a scroll button, turn the scroll button to zoom in and out.

To zoom in and out:

- ◆ In the **Top** viewport, hold the **Ctrl** key, click and hold the right mouse button, and drag the mouse up and down.

Drag up to zoom in.

Drag down to zoom out.



Zooming with **Ctrl** and the right mouse button.

What if ...

Instead of panning or rotating, something funny happened.

If you just right-click in viewport, the last command starts again. You must hold the right mouse button down while panning or rotating.

I panned in another viewport, and it shaded too. What's going on?

When you are in shaded mode and make a change in any viewport, the viewport also displays in shaded mode. Press **Esc** or click in a viewport to exit shaded mode.

Zooming Extents

The **Zoom Extents** command zooms your viewports so the objects fill the viewport as much as possible. You can use this command to make everything visible.

To zoom extents in a viewport:

- ◆ From the **View** menu, click **Zoom**, and then click **Extents**.
If you get lost, it's often handy to zoom extents in all your viewports at once, so there is a command to do just that.

To zoom extents in all viewports:

- ◆ From the **View** menu, click **Zoom**, and then click **Extents All**

Drag Objects

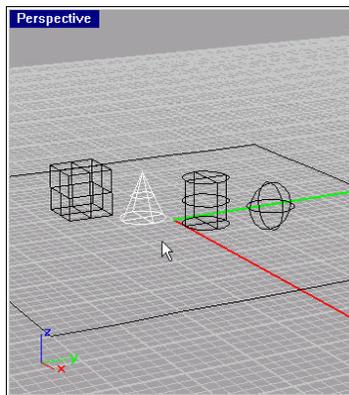
Now drag the objects around. You can drag in any viewport. In this model snap is set to one-half of a grid line. Using this snap, you should be able to line objects up with each other.

To drag objects:

- 1 In the status bar, click **Snap** to turn on the grid snap.
Grid snap may already be on in your system. Be careful that you don't turn it off instead of on. If grid snap is on, the word "Snap" will be black in the status bar. If it is off, the word "Snap" will be gray.

Note This is an important step. Grid snap only lets you move in certain intervals. In this model, grid snap is set to one half of a grid line. Grid snap helps you line up your objects as if you were building with LEGO® blocks.

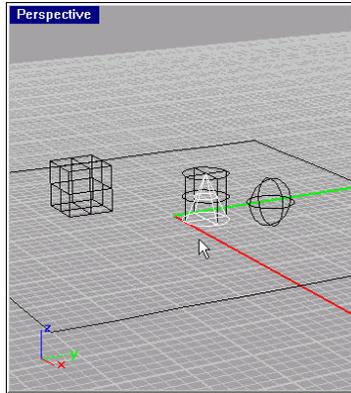
- 2 Click the cone.
The cone turns yellow to show it is selected.



The selected cone highlights.

- 3 Drag the cone in the **Perspective** viewport until it lines up with the cylinder. It will be inside the cylinder.

The cone moves on the base that is represented by the grid. This base is called a *construction plane*. Each viewport has its own construction plane. When you start Rhino, the Perspective viewport construction plane is the same as the Top viewport. You will learn more about using construction planes later.

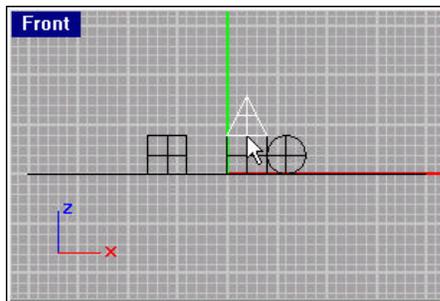


Drag the cone.

- 4 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.



Drag the cone in the Front viewport.

- 5 Click in the **Perspective** viewport.
- 6 From the **Render** menu, click **Shade**.
- 7 From the **Render** menu, click **Render**.

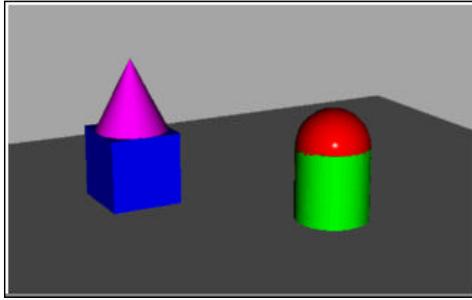
Try on Your Own

Drag the objects around. Use the Front viewport to drag the objects vertically and the Top or Perspective viewport to drag them horizontally.

Try these:

- Drag the cone to the top of the box.

- Drag the sphere so its equator is at the top of the cylinder.



Drag the cone and sphere.

- Try some other movements on your own.
- Use **Shade** and **Render** to look at your work.
- From the **Transform** menu, click **Move** and try this command. It is a little different from simply dragging. The **Move** command lets you use some placement techniques you will learn later.
- Select a command from the menu. Press **F1** to read the Help topic for that command.

Copy Objects

To create more objects, copy the shapes.

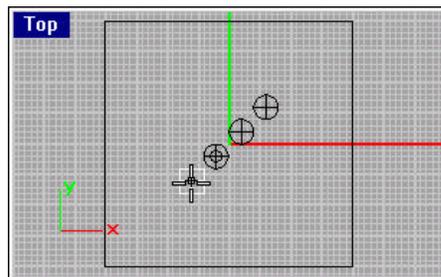
To start with a fresh model:

- 1 From the **File** menu, click **Open**.
- 2 Don't save the changes.
- 3 In the Open dialog box, in the **Tutorials** folder, select **First Model.3dm**.

To copy objects:

- 1 Click the box to select it.
- 2 From the **Transform** menu, click **Copy**.
- 3 At the **Point to copy from (Vertical InPlace)** prompt, click somewhere in the **Top** viewport.

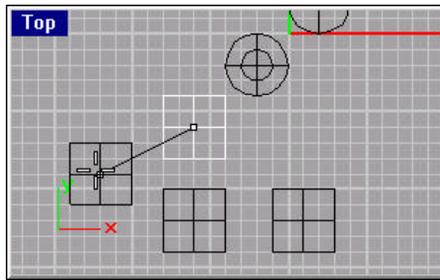
It usually helps to click a spot that relates to the object like the middle.



Select and copy the box.

- 4 At the **Point to copy to** prompt, click where you want the first copy.
Zoom in closer if you like.

To zoom, hold **Ctrl** and drag with the right mouse button.



Zoom in closer.

- 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**.

Review

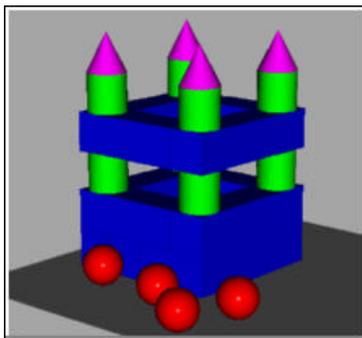
In this chapter you learned some basic Rhino commands and techniques. You learned how to:

- Rotate the view in a perspective viewport
- Zoom and pan
- Drag objects to move them
- Copy objects using the Copy command
- Shade and render

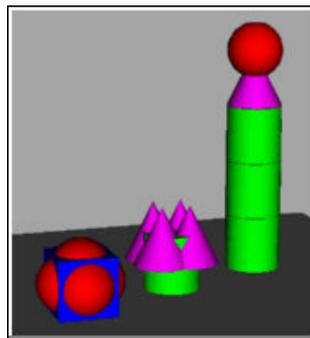
Try on Your Own

Make copies of more objects and drag them around. See if you can build something.

- Watch the prompt for instructions.
- Move your mouse from one viewport to another in the middle of a command to see what happens.
- Shade your viewport often to see what your shapes look like.
- Select a command from the menu. Press F1 to read the Help topic for that command.



Build a castle.



Hint: there are six overlapping spheres in the box.

Repeat the Last Command

Many tasks in Rhino are repetitive. You might want to move or copy several objects, for example. It's easy to repeat the command you just used. Any time you press **Enter** when no command is active, the previous command repeats.

In addition to pressing either **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.

You can press the **Esc** key to cancel a command any time.

Undo a Mistake

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

To undo commands:

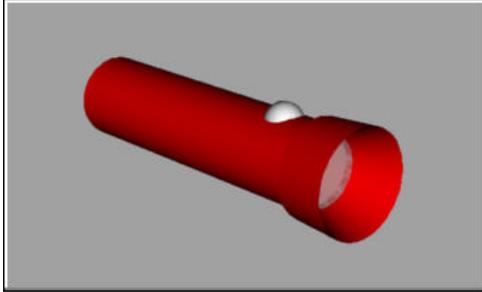
- ◆ From the **Edit** menu, click **Undo**.
The prompt displays the action you are undoing.
You can undo a series of commands.
You can also Redo the Undo if you go too far.

To redo a command:

- ◆ From the **Edit** menu, click **Redo**.

3 Create a Flashlight

You will learn how to use Rhino solids to create a simple model of a flashlight. While building the model you will create the basic shape with a cylinder, and a truncated cone. A sphere is added to create the switch. To complete the model you will add solids together and subtract other solids from your basic shapes.



The rendered flashlight with color.

You will be entering numbers to set sizes for the shapes. This lets you draw to a certain scale. The units in Rhino can stand for anything you want them to be: inches, feet, centimeters. For the moment, they will be simply *units*. When the model is finished you will have three distinct parts—the flashlight body, the lens, and the switch.

This exercise shows how to:

- Create a cylinder
- Type distances at the prompt
- Create a truncated cone
- Create a sphere
- Add two solids together
- Subtract one solid from another
- Assign colors to an object for rendering
- Render the model

Start the Flashlight Model

Each command is available from either the menu or the toolbars. In this exercise, we will use the menu commands. You are, of course free to find the command in the toolbars if you prefer to use them. There is no “right” way. Some people just prefer one method to another.

To start the model:

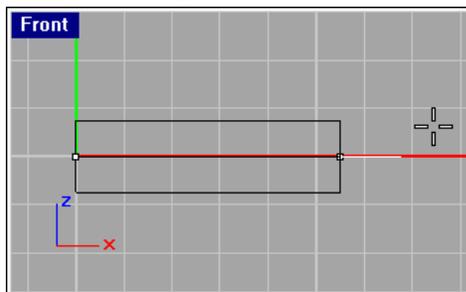
- 1 From the **File** menu, click **New**.
- 2 In the Template File dialog box, select **4 Viewports.3dm** and click **Open**.

Draw the Body

The flashlight body is created using cylinders and truncated cones. You will also create a cylinder and a truncated cone that will be used to hollow out the inside of the flashlight body.

To draw the body:

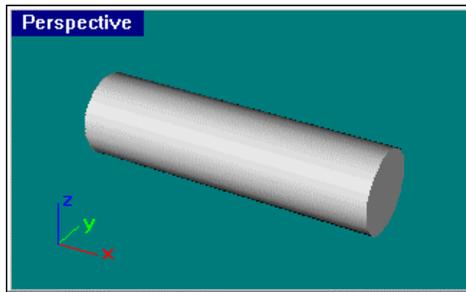
- 1 From the **Solid** menu, click **Cylinder**.
- 2 Move the cursor to the **Right** viewport.
In this case, you don't have to click in the viewport, just move the cursor there to make it active. The viewport title highlights.
- 3 At the **Base of cylinder (Vertical)** prompt, type **0,0**.
This sets the base point for the cylinder at the origin (0,0,0 point) of the coordinate system for the Right viewport.
Coordinates define points in space. You will learn more about using coordinates later.
- 4 At the **Radius (Diameter)** prompt, type **.75** and press **Enter**.
- 5 At the **End of cylinder** prompt, type **5.5** and press **Enter**.
This turns on the distance constraint that restricts the distance from the last point to 5.5 units.
- 6 Turn **Ortho** on.
Click **Ortho** in the status bar or hold down **Shift**.
- 7 At the next **End of cylinder** prompt, in the **Front** viewport, click so the cylinder points to the right.



With ortho on, drag the cylinder to the right in the Front viewport.

- 8 From the **View** menu click **Zoom**, then click **Extents All**.
Or, right-click the **Zoom Extents** button.
- 9 Click in the **Perspective** viewport.
When you are not in the middle of a command or selecting objects, you have to click in a viewport to make it the active viewport.

- 10 From the **Render** menu, click **Shade**.



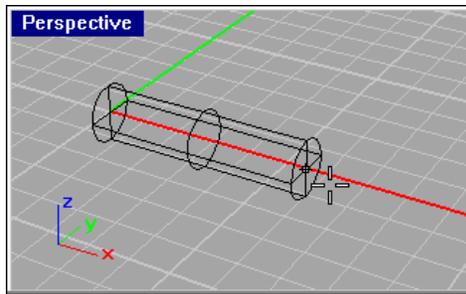
Shaded view of the first cylinder.

To draw the flashlight reflector shield:

- 1 From the **Solid** menu, click **Truncated Cone**.
- 2 At the **Base point of truncated cone (Vertical)** prompt, snap to the center of the right end of the cylinder.

To turn on the Center object snap, on the status bar, click **Osnap**, then check **Cen**.

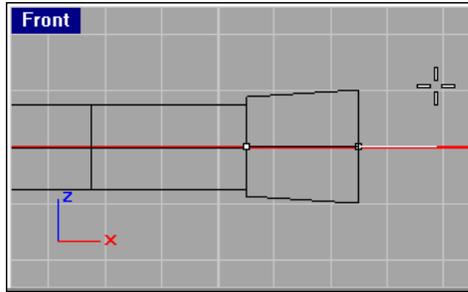
It's easiest to use the **Perspective** viewport for this.



Snap to the center of the cylinder.

- 3 At the **Radius (Diameter)** prompt, type **.875** and press **Enter**.
A truncated cone has two radius values, the base and top.
- 4 At the next **Radius (Diameter)** prompt, type **1** and press **Enter**.
- 5 At the **End of cone** prompt, type **2** and press **Enter**.
This sets the length of the cone.
- 6 At the next **End of cone** prompt, pick in the **Front** viewport to the right of the cylinder.

You may have to zoom out a little to get some space in the Front viewport. Hold down **Ctrl** and drag with the right mouse button. You can do this right inside the command. The end of the cone won't be entered until you click with the left mouse button.



Use the Front viewport to drag the truncated cone to the right.

Being able to zoom and pan inside a command makes moving around in Rhino easy.

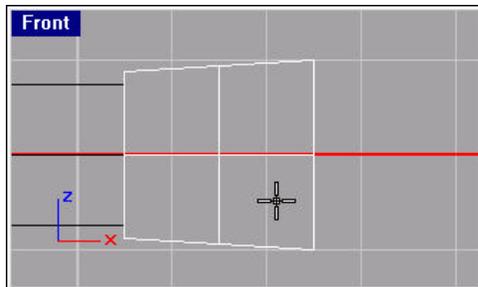
- 7 Zoom extents in all viewports.
- 8 **Shade** the **Perspective** viewport.

Cut the Reflector Shield

You will now copy the truncated cone to create another truncated cone that will be used to cut the inside shape of the reflector shield.

To create the cutting cone:

- 1 Select the truncated cone.
- 2 From the **Transform** menu, click **Copy**.
- 3 At the **Point to copy from (Vertical InPlace)** prompt, in the **Front** viewport pick any point.

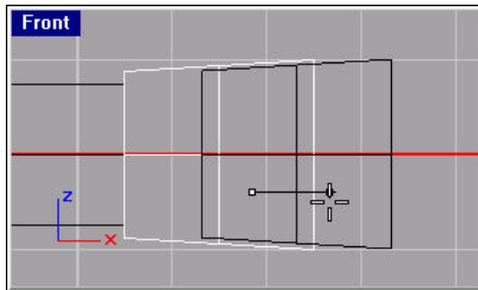


Pick a base point for the copy. It can be anywhere.

- 4 At the **Point to copy to** prompt, turn on **Ortho** and drag the cone to the right until the sides of the copy are inside the first cone.
- 5 Pick a point to place the copy, and press **Enter**.

We need to move the truncated cone to the right so it does not cut into the sides of the flashlight.

It may help at this point to turn off the Center object snap.



Move the copy out beyond the first truncated cone.

Ortho will keep it moving in a straight line.

To add the two parts of the flashlight body together:

- 1 From the **Solid** menu, click **Union**.
- 2 At the **Select surface or polysurface** prompt, select the cylinder.
You can't select the object first when using these commands. Starting the command cancels your selection. You can only select one object.
- 3 At the **Select surface or polysurface to union** prompt, select the first truncated cone.
Not much appeared to happen, but the cylinder and the truncated cone are now one piece.

To cut the inside of the reflector shield:

- 1 From the **Solid** menu, click **Difference**.
- 2 At the **Select surface or polysurface** prompt, select the flashlight body.
- 3 At the **Select surface or polysurface to subtract** prompt, select the new truncated cone.
- 4 **Shade** the **Perspective** viewport.
- 5 Rotate the view a bit so you can see inside the flashlight.

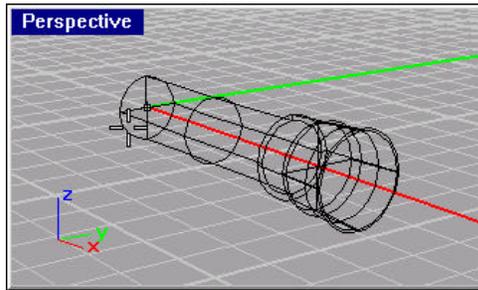
Cut Out the Inside of the Flashlight Body

You will now draw a cylinder that you will use to “hollow out” the inside of the flashlight body.

To draw the cutting cylinder:

- 1 From the **Solid** menu, click **Cylinder**.
You will create a cylinder that will be subtracted from the inside of the flashlight.

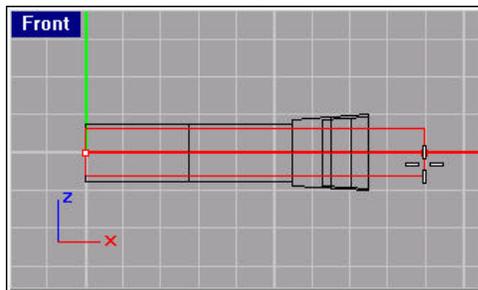
- 2 At the **Base of cylinder (Vertical)** prompt, pick a point in the **Perspective** viewport that is at the center of the cylinder at the flashlight base.



Start the cylinder at the center of the base.

- 3 At the **Radius (Diameter)** prompt, type **.625** and press **Enter**.
- 4 At the **End of cylinder** prompt, pick a point in the **Front** viewport to the right of the truncated cone.

This sets both the length of the cylinder and the rotation angle at the same time.



Extend the cylinder beyond the right end.

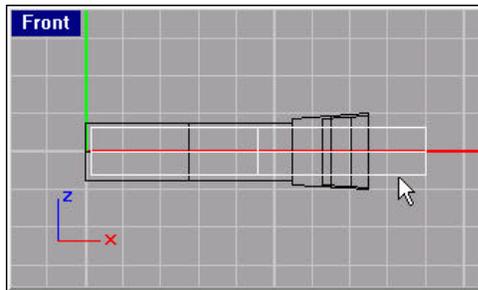
To move the cylinder away from the base:

We need to move the cylinder away from the base a little so it won't cut through the flashlight's bottom.

You can move objects just by selecting and dragging them.

- 1 Select the cylinder you just created.
- 2 Turn **Ortho** on, if it is not already.
- 3 Turn off the Center object snap.
Sometimes having object snaps on interferes with your ability to click and drag freely.
- 4 In the **Front** viewport, click near the right end of the cylinder and drag it a little bit to the right.

Ortho will keep it moving in a straight line.



Move the cylinder away from the base.

- 5 Zoom extents in all viewports.
- 6 **Shade** the **Perspective** viewport.

To cut out the interior of the flashlight:

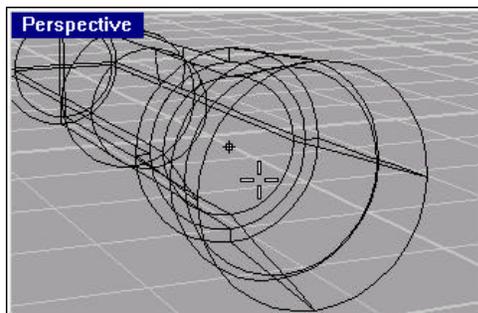
- 1 From the **Solid** menu, click **Difference**.
- 2 At the **Select surface or polysurface** prompt, select the flashlight body.
- 3 At the **Select surface or polysurface to subtract** prompt, select the new cylinder.
- 4 **Shade** the **Perspective** viewport.

Draw the Lens

The lens is a small cylinder inside the flashlight body.

To draw the lens:

- 1 From the **Solid** menu, click **Cylinder**.
- 2 At the **Base of cylinder (Vertical)** prompt, snap to the center of the inside edge of the truncated cylinder.

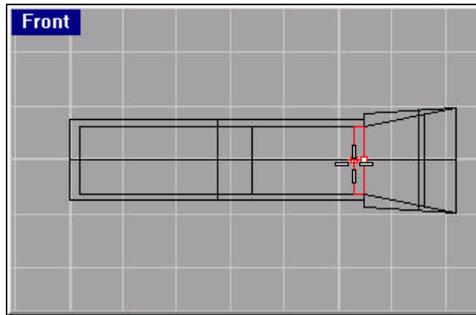


Start the lens cylinder at the center.

- 3 At the **Radius <0.625> (Diameter)** prompt, press **Enter**.
The radius is already set to .625 because you entered it for the last cylinder radius. You can simply press **Enter** to use this radius again.
- 4 At the **Choose a point** prompt, in the **Front** viewport pick a point slightly to the left of the center point.

It helps at this point to turn off the Center object snap. Otherwise, it's hard to drag the cylinder a very small amount.

You are trying to make a thin piece of glass.



The lens is a very shallow cylinder.

The glass won't show much in your model, but later, when you render it, you can make the flashlight more realistic by making this cylinder transparent.

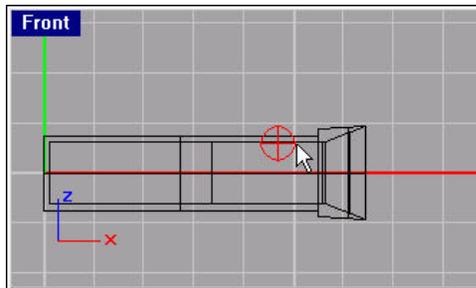
Draw the Switch

The switch is just a sphere drawn so part of it sticks up above the flashlight body.

To draw the switch button:

- 1 From the **Solid** menu, click **Sphere**, and then click **Center, Radius**.
- 2 At the **Center point** prompt, in the **Front** viewport pick a point near the top of the cylinder.
- 3 At the **Radius (Diameter)** prompt, pick a radius.

You can type **.4** and press **Enter**, or just draw a size that looks good to you.



Draw the switch.

- 4 **Shade** the **Perspective** viewport.

Render the Flashlight with Color

You can use Rhino to add color, texture, and lights to your model to make simple presentations. In this exercise you will add only color to the flashlight parts and use Rhino's render to create a more realistic picture.

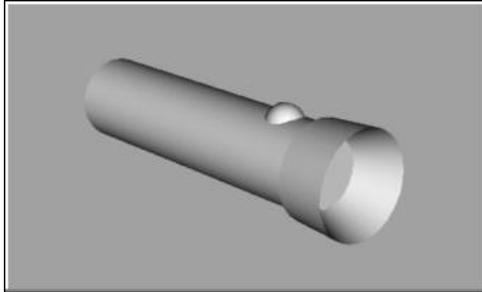
Render the Flashlight with No Color

To see what our starting point is, render the flashlight model with no color or lights assigned.

To render the model:

- 1 Click in the **Perspective** viewport.
- 2 From the **Render** menu, click **Render**.

A separate window appears with the model rendered.



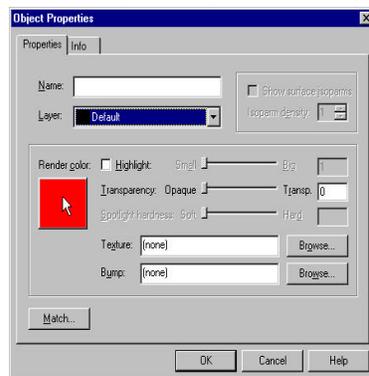
The fully rendered model.

Add Color to the Flashlight Parts

Render color and texture are object properties that are not visible in wireframe or shaded mode.

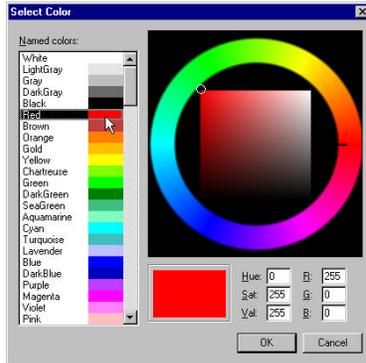
To set the color for the flashlight body:

- 1 Select the flashlight body.
- 2 From the **Edit** menu, click **Object Properties**.
- 3 In the Object Properties dialog box, under **Render color**, click the color swatch.



The Object Properties dialog box.

- 4 In the Select Color dialog box, from the **Named Colors** list, select **Red**, and click **OK**.

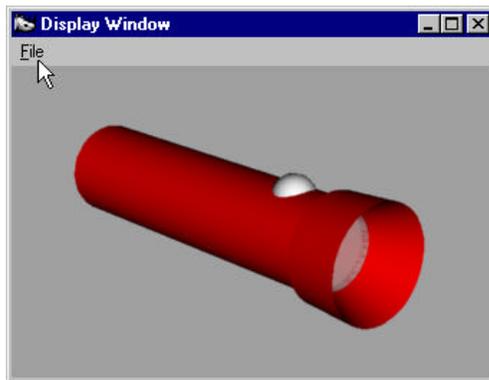


The Select Color dialog box.

- 5 In the Object Properties dialog box, check **Highlight**.
- 6 Move the **Highlight** slider to near the middle of the Small to Big range.
- 7 Click **OK**.

To set the lens to be transparent:

- 1 Select the lens.
- 2 From the **Edit** menu, click **Object Properties**.
- 3 In the Object Properties dialog box, move the **Transparency** slider to the right. Don't move it all the way to the right or it will disappear completely. You want it to be a little opaque so you can still see it.
- 4 From the **Render** menu, click **Render**.



The rendered flashlight with color.

Note If you want to save your flashlight image, from the **File** menu in the **Display Window**, click **Save As**.

Try on Your Own

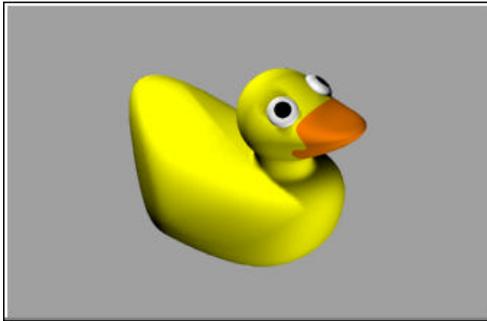
Try using some of the following techniques:

- Draw some solids that overlap each other. Try the **Add**, **Subtract**, and **Intersection** commands on them to cut and join them.
- Change the colors and the highlight, and render in other viewports.

4 Create a Rubber Ducky

When building models in Rhino, you should first determine which methods should be used for each part of the project. There are two basic ways to model in Rhino—free-form and accurate. Some models require more attention to exact dimensions because they might have to be manufactured or parts may have to fit together. Sometimes it is the shape of the object, not the accuracy that is important. These techniques can be merged together to create accurate, free-form shapes.

This exercise focuses only on the free-form, squishy aspect. The exact size and placement of the objects is not critical. The overall form is the main objective.



The rubber ducky.

You will:

- Create simple surfaces
- Rebuild a surface
- Edit surface control points
- Draw and project curves
- Split surfaces
- Blend between two surfaces
- Light and render the model

When you model the rubber ducky, you will use similar modeling techniques for the head and the body. In this exercise you will create spheres that will be deformed to make the shapes.

To start the model:

- ◆ Start a new model.

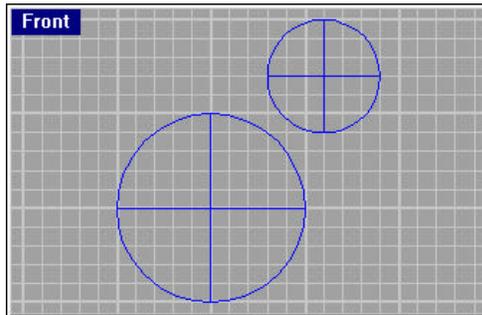
You can use layers to separate your parts, but for this model, it isn't necessary.

Create the Body and Head Shapes

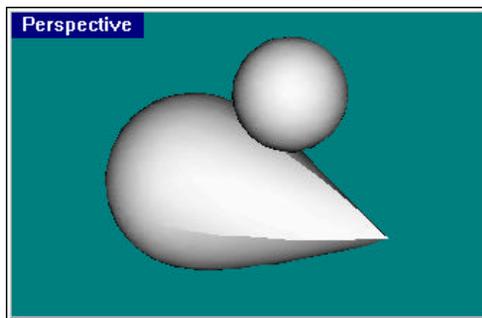
The body and head of the ducky are created by modifying two spheres. The size and placement of the spheres doesn't need to be exact.

To create the basic shapes:

- 1 From the **Solid** menu, click **Sphere**, then click **Center, Radius**.
- 2 At the **Center of sphere** prompt, in the **Front** viewport, pick a point.
- 3 At the **Radius (Diameter)** prompt, pick another point in the same viewport to create a sphere as shown.
- 4 Repeat this procedure for the second sphere.

*Draw two spheres.***To move a control point on the sphere:**

- 1 Select the large sphere.
- 2 From the **Edit** menu, click **Point Editing**, then click **Control Points On**.
- 3 Select a control point and drag it.
- 4 Click in the **Perspective** view.
- 5 **Shade** the **Perspective** viewport.

*Moving control points creates sharp edges.*

Notice that you have sharp edges in the sphere. The spheres you just created do not lend themselves well to deformation. To make them smoothly deformable, you have to add control points and thereby the ability to deform smoothly. Rhino has a command to do this.

- 6 Click in a viewport to exit shaded mode.
- 7 Press **Esc** to turn off control points.
- 8 From the **Edit** menu, click **Undo**.

To make the spheres deformable:

- 1 Select both spheres.
- 2 From the **Surface** menu, click **Edit Tools**, then click **Rebuild**.

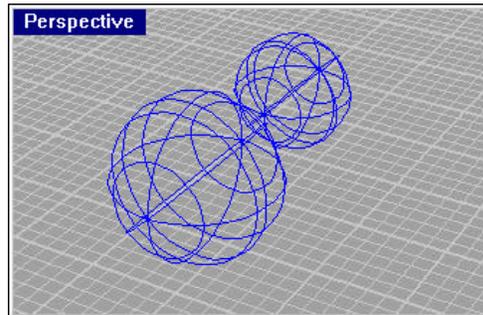
- 3 In the **Rebuild Surface** dialog box, change the **Point Count** to **8** for both **U** and **V**.

Change the **Degree** to **3** for both **U** and **V**.

Check **Delete input**.

Clear **Current layer**.

Click **OK**.

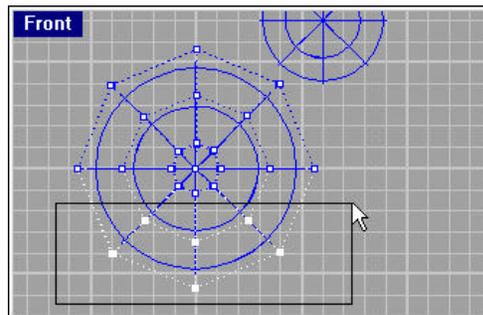


Rebuilding the surface adds control over deformation.

The spheres are now deformable. Having more control points allows more control over smaller parts of the surface. A degree-three surface will have a smoother shape when deformed.

To modify the body shape:

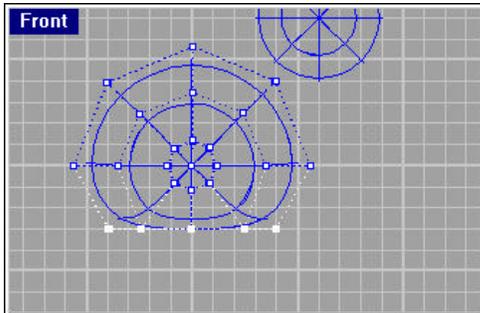
- 1 Select the large sphere.
- 2 From the **Edit** menu, click **Point Editing**, then click **Control Points On**.
- 3 In the **Front** viewport use a window to select the control points near the bottom of the sphere.



Select the control points as shown.

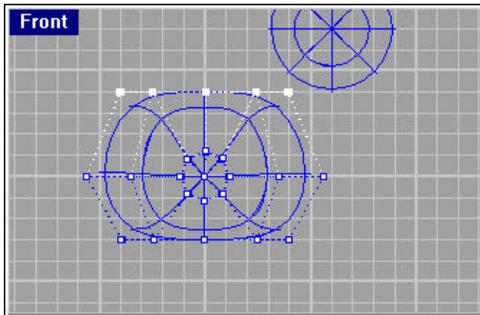
- 4 From the **Transform** menu, select **Set Points**.

- 5 In the Set Points dialog box, check **Set Z**, and click **World Coordinates**.
- 6 In the **Front** viewport, 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.



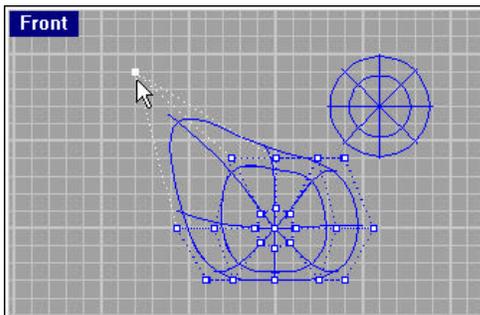
Use Set Points to move the control points up in the Front viewport.

- 7 Adjust the control points at the top of the body using the same technique.



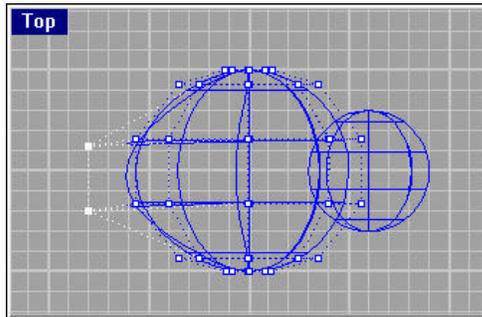
Use Set Points to move the control points down in the Front viewport.

- 8 Use a window to select the control points at the upper left edge of the body, and drag them up to form the tail.



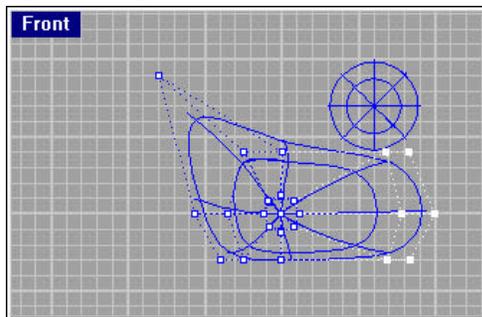
Drag two control points in the Front viewport.

Notice in the Top viewport that two control points are selected, though in the Front viewport, it looks like only one is selected. This is because the second control point is directly behind the one you can see in the Front viewport.



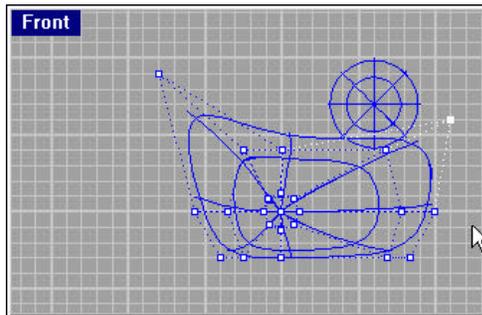
In the Top viewport, you can see both points.

- 9 Window select the control points at the front of the body, and drag them to the right to bulge out the chest.



Drag out the chest.

- 10 Window select the control points at the upper right, and drag them up and to the right.



Adjust the chest up.

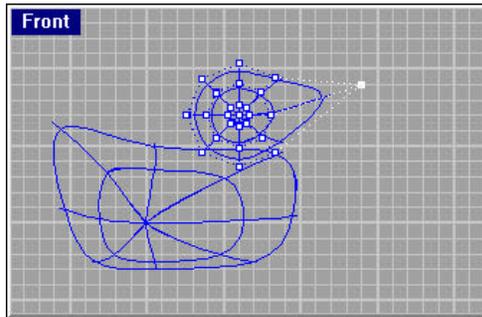
Adjust the control points further until you get the shape you want.

- 11 **Shade** the **Perspective** viewport.
- 12 Press **Esc** to exit shaded mode.
- 13 Press **Esc** to turn off control points.

To create the head:

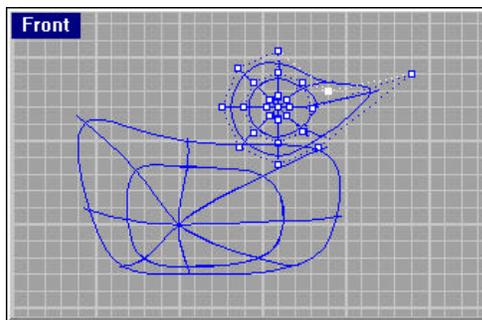
- 1 Select the small sphere.
- 2 From the **Edit** menu, click **Point Editing**, then click **Control Points On**.

- 3 Select the control points on the right side and drag them to begin forming the bill.



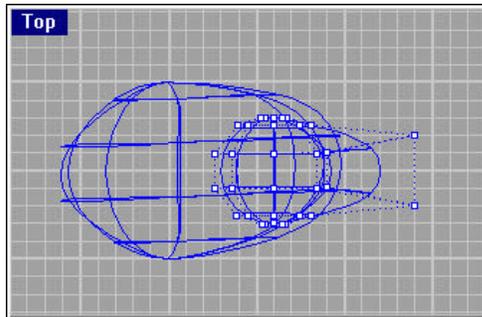
Drag control points to form the bill.

- 4 Select the control points at the top right side and drag down to continue forming the bill.



Drag the point down to form a forehead.

- 5 Selecting control points in the **Top** viewport and drag to make the bill wider.



- 6 Press **Esc** to turn control points off.
- 7 **Shade** the **Perspective** viewport.

Separate the Bill From the Head

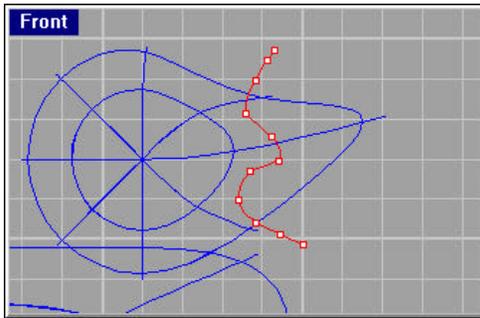
For the final rendering, the bill must be a different color from the body, so they must be separate surfaces. You can split a single surface into multiple surfaces many ways. The following technique is just one.

To make the head and bill two separate surfaces, split it with a curve. It is easiest to draw a curve in one view and project it to the surface to create a 3D curve.

To create the curve for the bill:

- 1 From the **Curve** menu, click **Free-form**, and then click **Interpolate Points**.

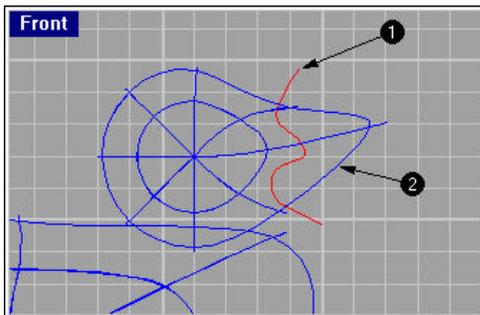
- In the **Front** viewport create a curve that looks like the red curve below.



Draw a curve in the Front viewport.

To project the curve onto the surface of the head:

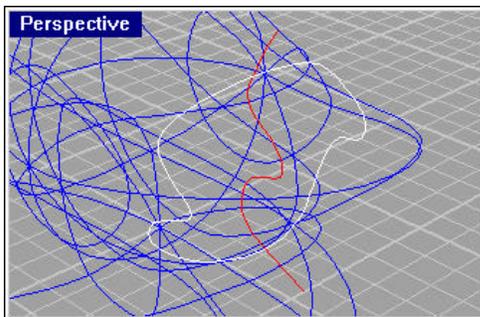
- From the **Curve** menu, click **From Objects**, then click **Project**.
- At the **Select curve(s) to project** prompt, in the **Front** view, select the curve **1** and press **Enter**.



Project the curve onto the head surface.

- At the **Select object(s) to project onto** prompt, select the head **2**, and press **Enter**.

The curve is projected onto the head.



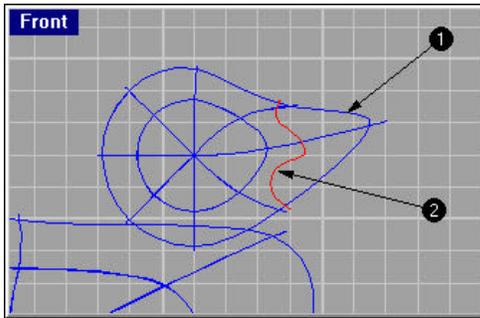
The projected curve wraps around the head.

- Delete the original curve.

To split a surface with a curve:

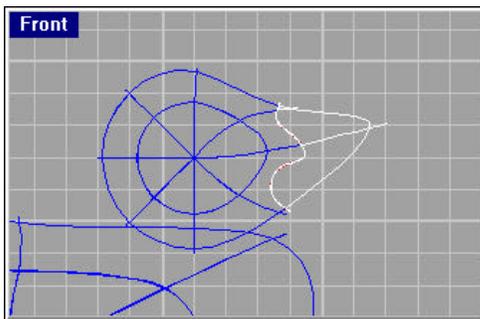
- From the **Edit** menu, click **Split**.

- At the **Select object to split** prompt, select the head **1**.



Split the head with the projected curve to create two parts.

- At the **Select cutting objects** prompt, select the new projected curve (**2**) and press **Enter**.



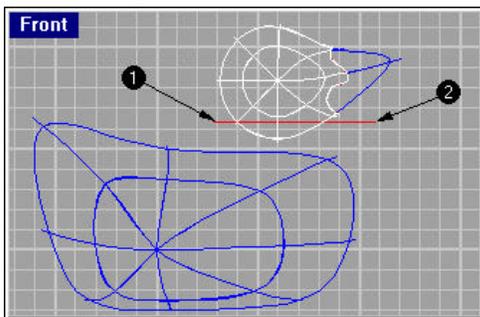
The bill separates from the head.

Create the Ducky's Neck

The ducky needs a neck. We are first going to make an edge on the surface of the head and a corresponding edge on the surface of the body so we can create a blended surface between the two edges.

To create a cutting plane:

- Select the head.
- From the **Surface** menu, click **Rectangle**, then click **Cutting Plane**.
- At **Start of cut plane** prompt, in the **Front** view, pick the first point of a line (**1**) to define a plane.

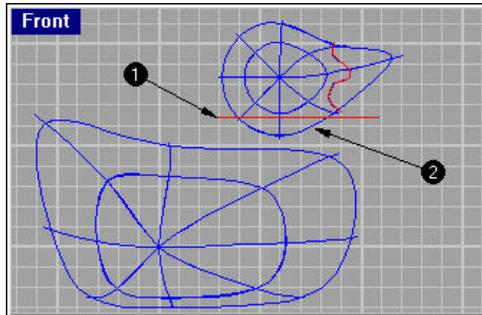


*Create a cutting plane at the bottom of the head.
(We cheated a bit and moved the head up.)*

- 4 Toggle **Ortho** on.
- 5 At the **End of cut plane** prompt, select the end point of a line (2) that cuts through the lower part of the head, and press **Enter**.

To trim the head with the cutting plane:

- 1 From the **Edit** menu, click **Trim**.
- 2 At the **Select cutting edges (Options)** prompt, select the cutting plane you created (1).

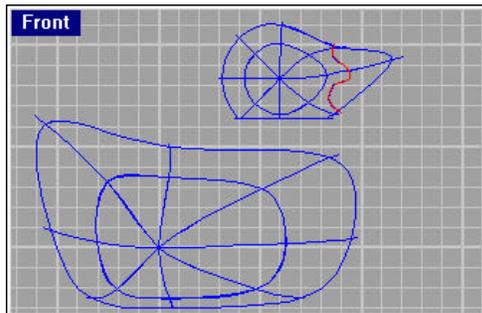


Trim off the bottom of the head.

- 3 At the **Select cutting edges (Options)** prompt, press **Enter**.
- 4 At the **Select object to trim (Options)** prompt, select the bottom edge of the head (2).
- 5 At the **Select object to trim (Options)** prompt, press **Enter** to end the command.

The bottom of the head will be trimmed.

- 6 Delete the cutting plane.

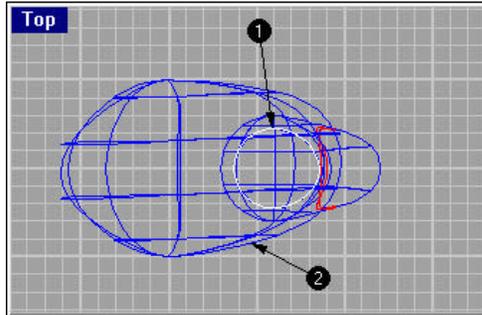


The trimmed head.

To cut a hole in the body that matches the opening in the bottom of the head:

- 1 From the **Curve** menu, click **From Objects**, then click **Project**.

- 2 At the **Select curve(s) to project** prompt, in the **Perspective** viewport, select the edge curve at the bottom of the head (1).



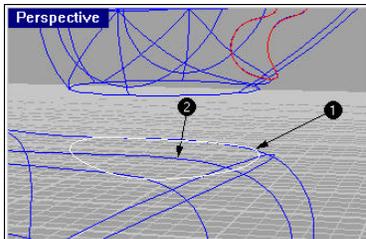
- 3 At the **Select curve(s) to project** prompt, press **Enter**.
 4 At the **Select Object to project onto** prompt, in the **Top** viewport, select the body (2) and press **Enter**.

It is important to project this in the **Top** or **Perspective** viewport. The curve is projected perpendicular to the active construction plane.

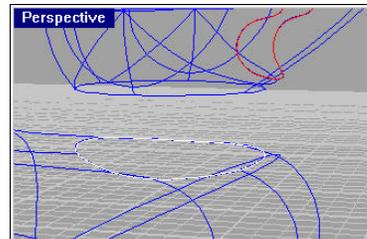
The curve is projected to the top and the bottom of the body.

- 5 From the **Edit** menu, click **Trim**.
 6 At the **Select cutting edges (Options)** prompt, select the curve at the top of the body and press **Enter**.
 7 At the **Select object to trim (Options)** prompt, select the body on the inside of the curve and press **Enter**.

A hole will be created in the body. If **Trim** doesn't work try **Split**.



It may be difficult to find the isoparm to pick inside the curve.

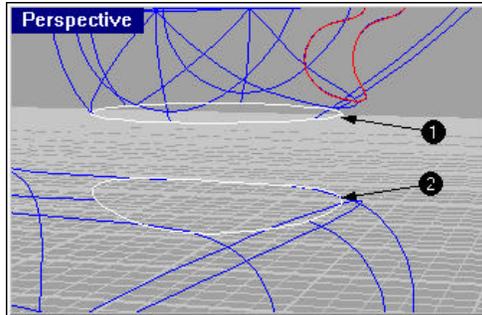


The trimmed body.

- 8 Delete the projected curves.
 Use a window in the **Front** viewport. Occasionally the curve gets broken into pieces.

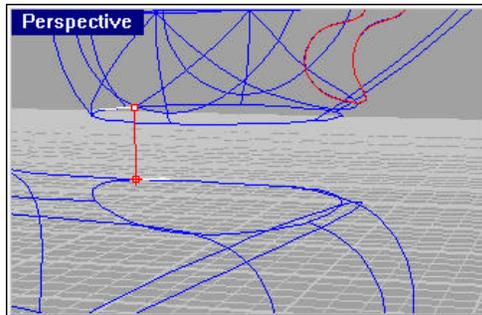
To create the blend surface between the head and body:

- 1 From the **Surface** menu, click **Blend**.
- 2 At the **Select edge to blend - pick near one end (PlanarSections Degree=Quintic)** prompt, select the curve at the bottom of the head (1).



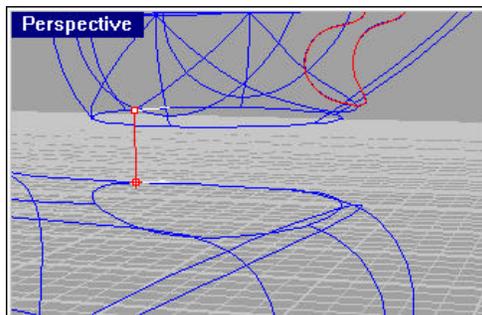
- 3 At the **Select edge to blend - pick near one end (PlanarSections Degree=Quintic)** prompt, select the curve on the body (2).
- 4 At the **Select seam point to adjust, press Enter when done (FlipDirection Automatic Natural)** prompt, check to see that the seam point lines up.

If it doesn't, select one of the seam points and drag it until it lines up with the other one.



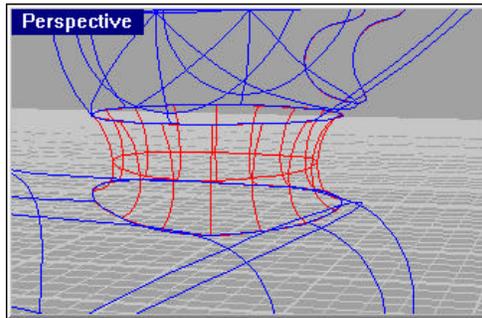
The direction on one of these curves needs to be flipped.

- 5 To flip a curve direction, at the **Location of seam point (FlipDirection)** prompt, pick the seam point on the curve you want to flip, then type **F** and press **Enter**.



The direction arrows now point the same way.

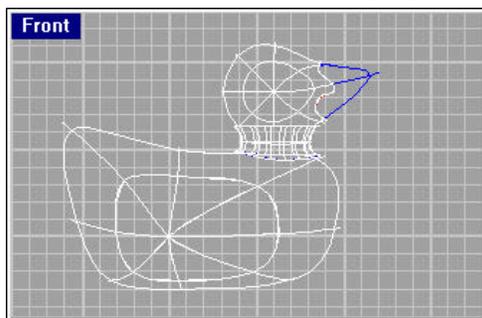
- 6 At the **Location of seam point (FlipDirection)** prompt, press **Enter**.
A surface is blended between the body and the head.



The neck blend surface.

To join the parts:

- 1 From the **Edit** menu, click **Join**.
- 2 At the **Select object for join** prompt, select the **body**.
- 3 At the **Select next surface or polysurface to join** prompt, select the **neck**.
- 4 At the **Select next surface or polysurface to join** prompt, select the **head**.
- 5 At the **Select next surface or polysurface to join** prompt, press **Enter**.



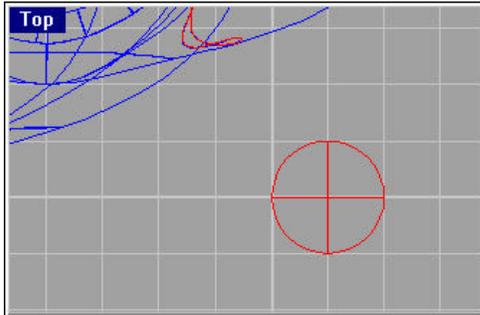
The joined head, neck and body.

- 6 **Shade** the **Perspective** viewport.

To make an eye:

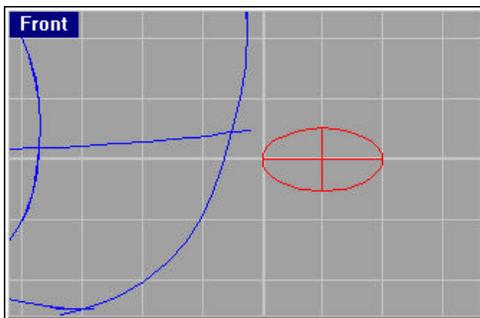
- 1 From the **Solid** menu, click **Ellipsoid**.
- 2 Toggle **Ortho** and **Snap** on to help.
- 3 At the **Center of ellipsoid** prompt, pick a point in the **Top** viewport.
- 4 At the **End of first axis** prompt, pick a point.

- 5 At the **End of second axis** prompt, pick a point that gives approximately the same radius as the previous pick.



Create a flattened ellipsoid.

- 6 At the **End of third axis** prompt, pick a point in the **Front** viewport that creates a flat ellipsoid.



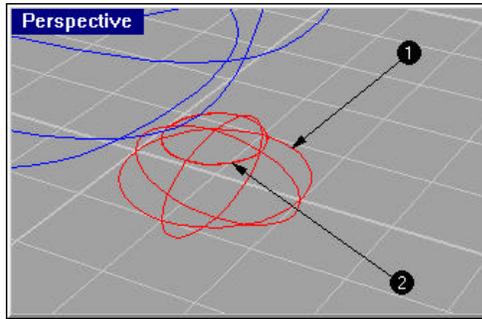
Drag the height in the Front viewport.

To make the pupil for the eye:

To be able to assign a different color for the pupil of the eye, the ellipsoid surface has to be split into two parts.

- 1 From the **Curve** menu, click **Circle**, then click **Center, Radius**.
- 2 At the **Center of circle (Vertical AroundCurve)** prompt, pick the center of the ellipsoid.
- 3 At the **Radius <1> (Diameter)** prompt, pick a point in the **Top** viewport that is slightly smaller than ellipsoid.
- 4 From the **Curve** menu, click **From Objects**, then click **Project**.
- 5 At the **Select curve(s) to project** prompt, in the **Top** viewport, select the circle and press **Enter**.

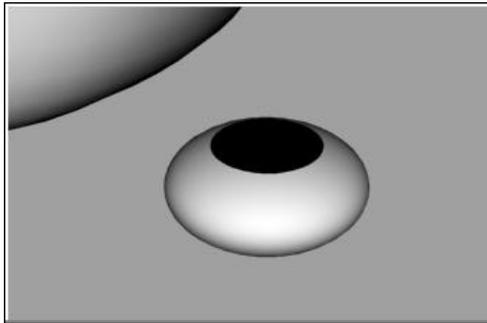
- 6 At the **Select object(s) to project onto** prompt, in the **Top** view, select the ellipsoid and press **Enter**.
- 7 Delete the original circle and the one on the bottom of the ellipsoid.
- 8 From the **Edit** menu, click **Split**.
- 9 At the **Select object to split** prompt, select the ellipsoid.
- 10 At the **Select cutting objects** prompt, select the circle on the top of the ellipsoid.
- 11 At the **Select cutting objects** prompt, press **Enter**.



Split the ellipsoid with the projected circle.

To assign a render color to the pupil:

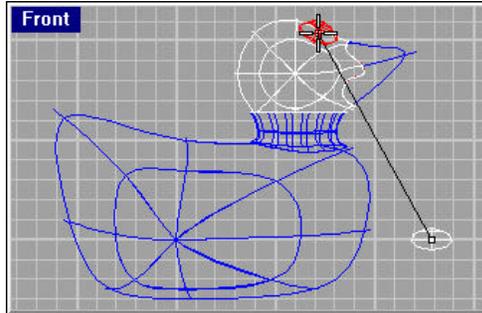
- 1 Select the surface at the top of the ellipsoid. If it is broken into two halves, join the pieces together. Make sure the top of the ellipsoid is selected.
- 2 From the **Edit** menu, click **Object Properties**
- 3 In the **Object Properties** dialog, under **Render Color**, click the color control patch and select a render color for the pupil of the eye, like black.
- 4 **Render** the **Perspective** viewport.



To move the eyes to the surface of the head:

- 1 From the **Transform** menu, click **Orient**, then click **On Surface**.
- 2 At the **Select base surface** prompt, select the head.
- 3 At the **Select objects to orient** prompt, window select the whole eye and press **Enter**.
- 4 At the **Base point on surface (WorldCoordinates)** prompt, type **W** and press **Enter**.

- 5 At the **Base point in world coordinates (OnSurface)** prompt, snap to the center of the ellipsoid.
- 6 At the **Offset point** prompt, move the cursor onto the head to place the eye and press **Enter**.



Orient the eye to the head.

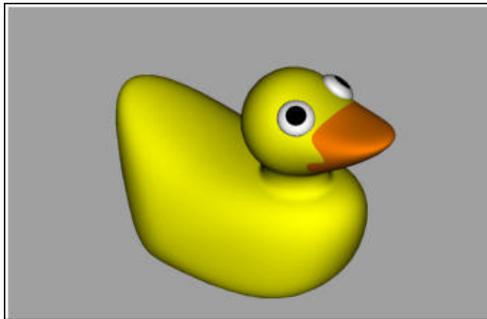
- 7 **Shade** the **Perspective** viewport.
- 8 **Mirror** the eye to the other side of the head.
- 9 **Delete** the extra eye on the floor.
- 10 **Render** the **Perspective** viewport.

Render a Picture of the Ducky

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 mode.

To render the ducky:

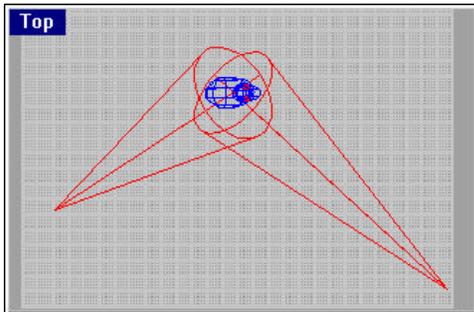
- 1 Select the beak.
- 2 From the **Edit** menu, click **Object Properties**.
- 3 In the **Object Properties** dialog, set a color for the beak, like orange.
- 4 Select the body.
- 5 From the **Edit** menu, click **Object Properties**.
- 6 In the **Object Properties** dialog, set a color for the body, like yellow.
- 7 From the **Render** menu, click **Render**.



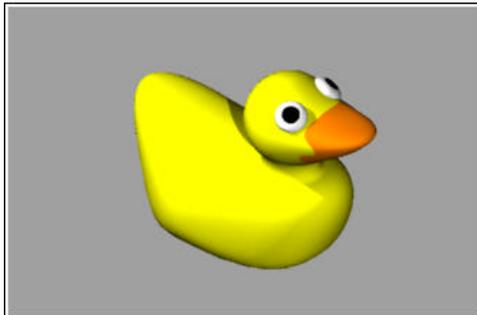
Add colors to the body and bill.

To place lights:

- 1 From the **Render** menu, click **Create Spotlight**.
- 2 At the **Base of cone (Vertical)** prompt, select a point in the middle of the model.
- 3 At the **Radius < > (Diameter)** prompt, drag the radius until it is approximately twice as large as the model.
- 4 At the **End of cone** prompt, use elevator mode and two viewports to place the end of the cone so the light cone shines on the ducky from the upper left side.
- 5 From the **Render** menu, click **Render**.
- 6 Try placing two lights.



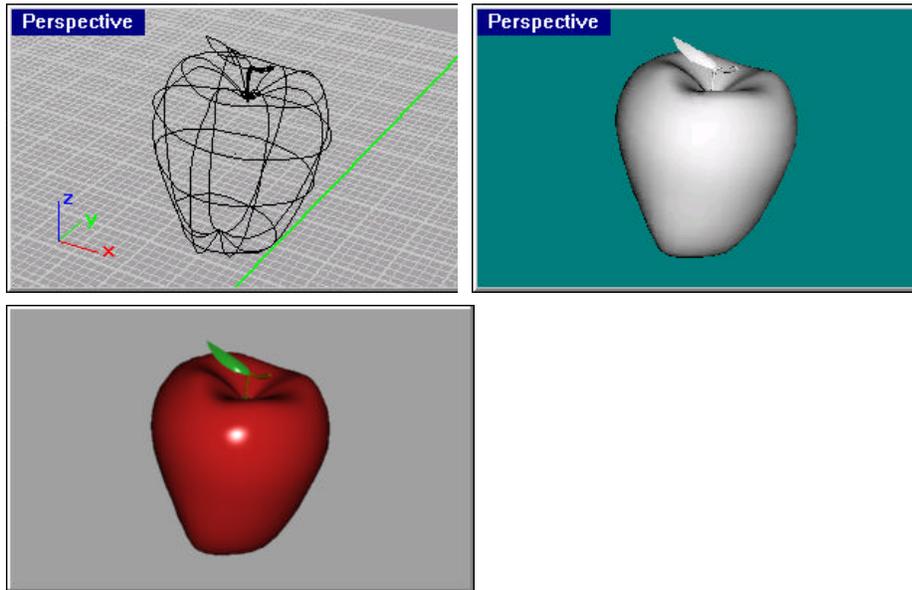
- 7 From the **Render** menu, click **Render**.

**Try on Your Own**

Try using some of the following techniques:

- Draw a sphere.
- Try editing the control points to create an apple, an orange or a potato.
- Try changing the colors and the highlight.

- Render one of the orthographic viewports to see the highlight better.



Look at the gallery images

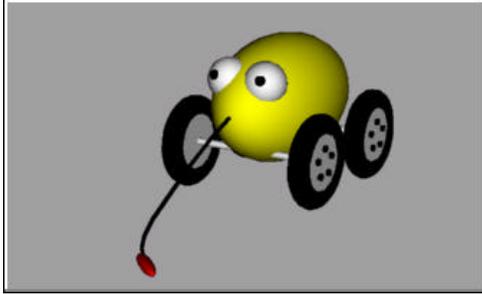
To see what else can be modeled in Rhino, check out the Rhino gallery at <http://www.rhino3d.com>.

Build your own models

Try modeling anything you have on your desk or around your house. Come up with your own designs to model. Try a few different versions of each idea.

5 Create a Pull Toy

In the following exercise, you will create a wheeled pull toy.



The completed pull toy.

You will learn how to:

- Enter coordinates to place points exactly
- Draw a curve, polygon, ellipsoid, torus
- Create a pipe along a curve
- Use a polar array to copy objects in a circular pattern
- Extrude a curve to create a surface

Enter Coordinates

In the exercises you have done so far, when you picked a point with the mouse, the point lay on the *construction plane* of the active viewport unless you used a modeling aid such as object snap or *elevator mode*. 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.

When Rhino prompts for a point, you can enter x-, y-, and z-coordinates instead of picking a point.

The grid is a visual representation of the construction plane. The intersection of the red and green lines show 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.

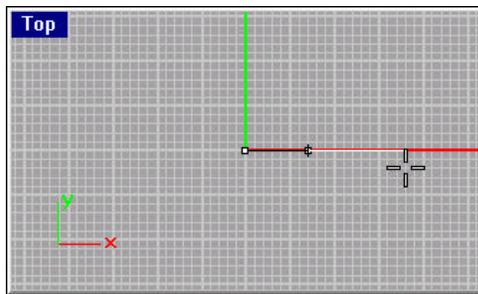
Note Pay close attention to the viewport required in each instruction.

To start the model:

- ◆ Begin a new model.
Use the viewport layout you are comfortable with.

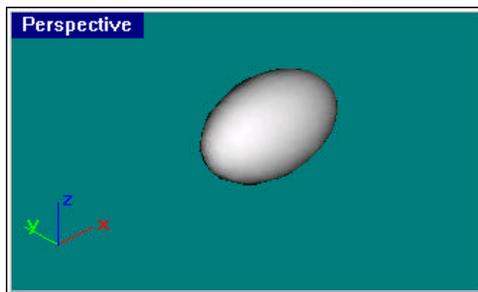
To create an ellipsoid:

- 1 Turn on **Ortho**.
- 2 From the **Solid** menu, click **Ellipsoid**.
- 3 In the **Top** viewport, at the **Center point** prompt, type **0,0,6** and press **Enter**.
This places the center point of the ellipsoid at $x=0$, $y=0$, and $z=6$. Look at the point in the perspective viewport.
- 4 At the **End of first axis** prompt, type **7** and press **Enter**.
- 5 Drag the cursor to the right and click to show the direction.



Drag the cursor to the right to show the first axis direction.

- 6 At the **End of second axis** prompt, type **4** and press **Enter**.
This sets the width of the ellipsoid.
- 7 At the **End of third axis** prompt, type **4.5** and press **Enter**.
You now have an egg shape that has different dimensions in all three directions.
- 8 Rotate the perspective viewport so you are looking along the x-axis as shown.
- 9 **Shade** the **Perspective** viewport.



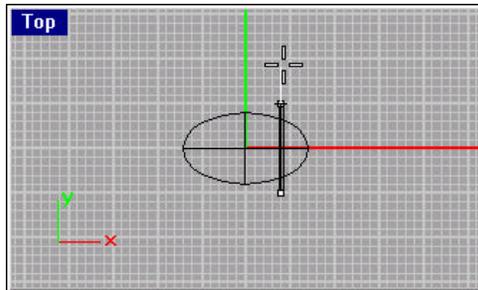
An ellipsoid is an egg shape.

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.

To create the axle:

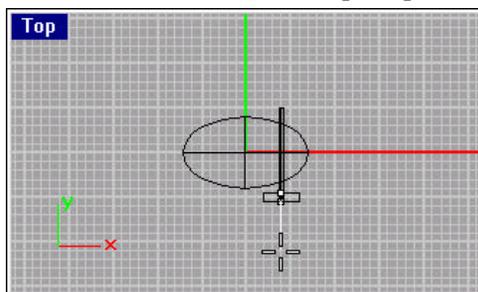
- 1 From the **Solid** menu, click **Cylinder**.
- 2 In the **Front** viewport, at the **Base of cylinder (Vertical)** prompt, type **4,3,5** and press **Enter**.
Look at where this point is. It is four units in the x-direction, three units in the y-direction, and five units in the z-direction of the active viewport.
- 3 At the **Radius (Diameter)** prompt, type **.25** and press **Enter**.
- 4 At the **End of cylinder** prompt, type **10** and press **Enter**.
- 5 In the **Top** viewport drag the cylinder as shown to place the axle.



Drag up in the Top viewport to place the axle.

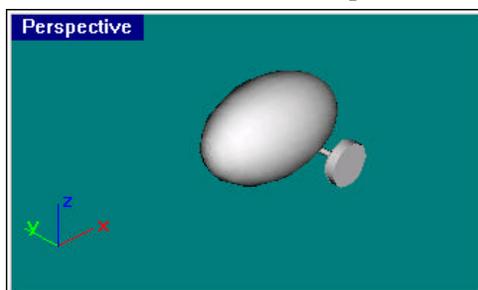
To create a wheel hub:

- 1 From the **Solid** menu, click **Cylinder**.
- 2 In the **Front** viewport, at the **Base of cylinder (Vertical)** prompt, type **4,3,4.5** and press **Enter**.
- 3 At the **Radius (Diameter)** prompt, type **2** and press **Enter**.
- 4 At the **End of cylinder** prompt, type **1** and press **Enter**.
- 5 At the next **End of cylinder** prompt, click in the top viewport to place the hub.



Drag down in the Top viewport to place the wheel.

- 6 **Shade** the **Perspective** viewport.



One hub made.

Draw the Lug Nuts

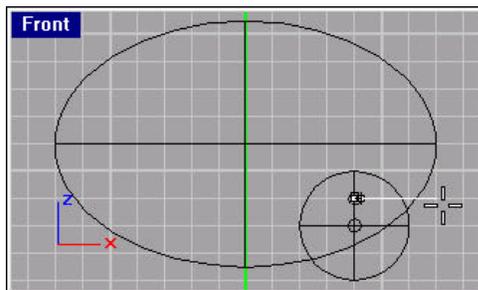
You will make the lug nuts by extruding a hexagonal polygon curve.

To create the hexagon:

- 1 From the **Curve** menu, click **Polygon**, and then click **Center, Radius**.
- 2 At the Center of polygon (NumSides=4 Circumscribed) prompt, type **6** and press **Enter**.
- 3 In the **Front** viewport, at the Center of polygon (NumSides=6 Circumscribed) prompt, type **4,4,5.5** and press **Enter**.

This will place the polygon right on the surface of the wheel hub.

- 4 At the **Radius** prompt, type **.25** and press **Enter**.



We have zoomed in the Front viewport.

To make a solid from the polygon:

- 1 Select the hexagon you just created.
- 2 From the **Solid** menu, click **Extrude Planar Curve**.
- 3 At the **Extrusion distance (Direction Cap=yes Bothsides Tapered)** prompt, notice the *command options*.

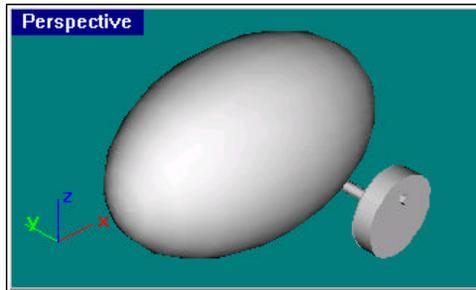
Many commands have options. You will learn how to change and use them as you learn to use the commands. Take a moment and look at the options available for Extrude Planar Curve.

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

- 4 At the **Extrusion distance (Direction Cap=yes Bothsides Tapered)** prompt, type **-.25** 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.

5 Shade the **Perspective** viewport.



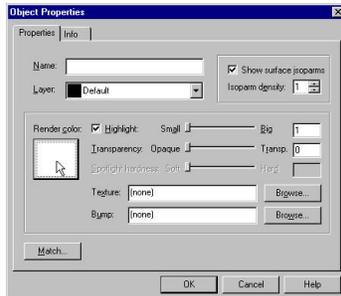
The lug nut should protrude.

Assign Colors

Now that you have the basic parts built, you are going to assign colors to them before we start copying them. If we wait until we have all the parts, you will have to select 20 lug nuts separately. If we assign colors now, the color property will be copied when we copy the parts.

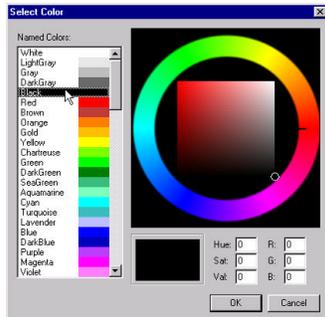
To assign color to the parts:

- 1 Select the lug nut.
- 2 From the **Edit** menu, click **Object Properties**.
- 3 In the Object Properties dialog box, under **Render color**, click the color swatch as shown.



Click the color swatch to set object color.

- 4 In the Select Color dialog box, under **Named Colors**, click **Black** and click **OK**.

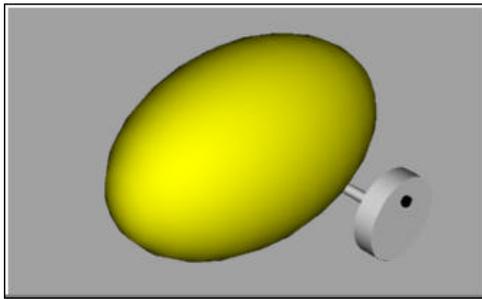


The Select Color dialog box

- 5 In the Object Properties dialog box, click **OK**.
- 6 Follow steps 1 through 6 to set a color for the toy body.

You will be assigning colors to objects as we go along.

7 Render the Perspective viewport.



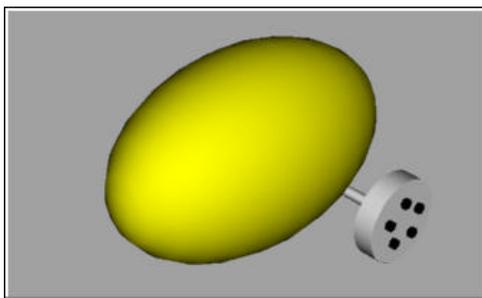
The wheel and axle look gray because they are assigned the default white color and are not directly lit.

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.

To array the nuts:

- 1** Select the lug nut.
- 2** From the **Transform** menu, click **Array**, and then click **Polar**.
The hexagon is still there, so be sure you select the extruded lug nut. (The Select One Object dialog will list it as a *polysurface*.)
- 3** At the **Center point** prompt, snap to the center of the hub.
- 4** At the **Number of elements < 1 >** prompt, type **5** and press **Enter**.
- 5** At the **Angle to fill <360>** prompt, press **Enter**.
- 6** **Render the Perspective viewport.**



Polar array copies objects around a center point.

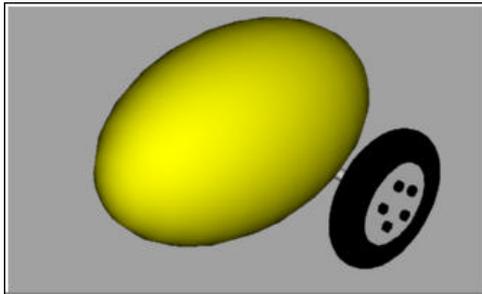
Draw the Tires

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.

To create a torus for the tires:

- 1 From the **Solid** menu, click **Torus**.
- 2 In the **Front** viewport, at the **Center point (Vertical AroundCurve)** prompt, type **4,3,5** 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 (Diameter)** prompt, type **2.5** and press **Enter**.
This makes the center of the torus 0.5 units larger than the wheel hub.
- 4 At the **Second radius (Diameter)** prompt, type **.75** and press **Enter**.
This makes the hole .25 units smaller than the wheel hub.
- 5 Set the **Color** of the tire to **Black**.
- 6 **Render** the **Perspective** viewport.



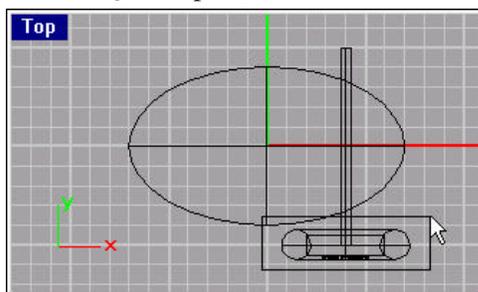
Change the color of the tire to black.

Mirror the Wheels

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

To mirror the wheel to the other side:

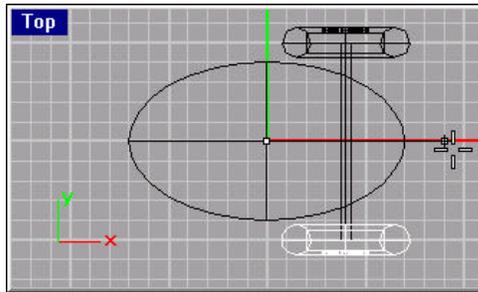
- 1 In the **Top** viewport, use a window to select the wheel as shown.



Use a window to select all the wheel parts.

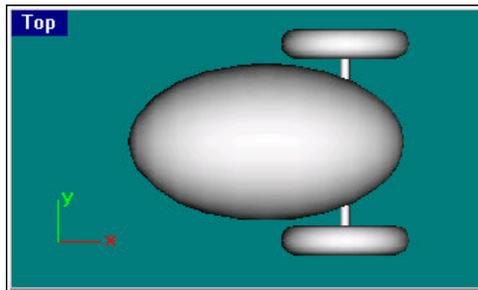
- 2 From the **Transform** menu, click **Mirror**.
- 3 At the **Start of mirror plane** prompt, type **0,0,0**.

- 4 In the **Top** viewport, with **Ortho** on, drag to the right as shown and click.



Mirror the wheel in the Top viewport.

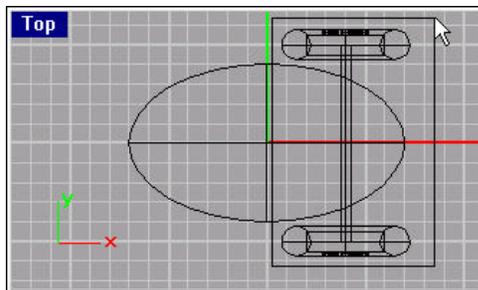
- 5 **Shade** the **Top** viewport.



The completed back wheels.

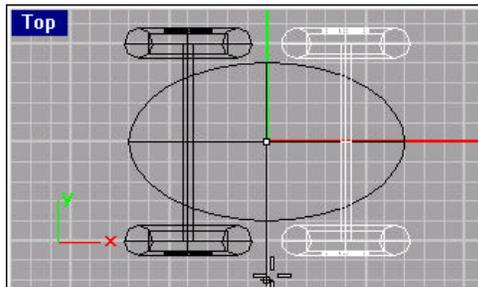
To mirror the front wheels and axle:

- 1 In the **Top** viewport, use a window to select the back wheels and axle as shown.



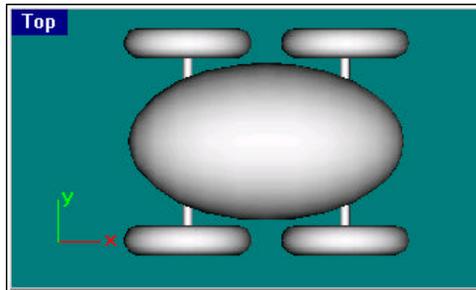
Use a window to select the wheels and axle.

- 2 From the **Transform** menu, click **Mirror**.
 3 At the **Start of mirror plane** prompt, type **0,0,0**.
 4 In the **Top** viewport, with **Ortho** on, drag down as shown and click.



Mirror the wheels in the Top viewport.

5 Shade the Top viewport.



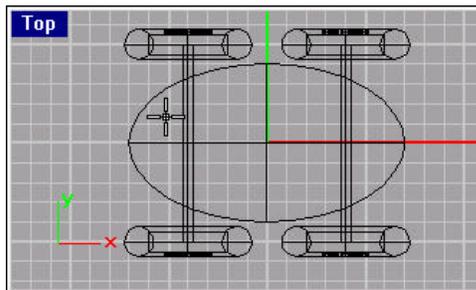
The completed front wheels.

Draw the Eyes

While drawing the eyes, you are going to practice using elevator mode. You are going to start the center of the eye spheres in the top viewport and use elevator mode to move them up.

To create an eye:

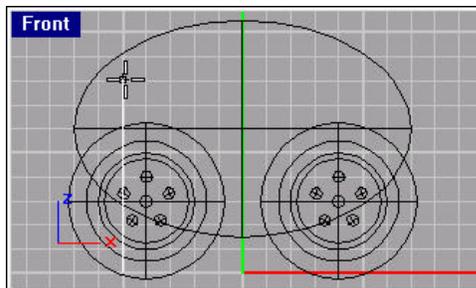
- 1 From the **Solid** menu, click **Sphere**, and then click **Center, Radius**.
- 2 At the **Center point** prompt, in the **Top** viewport hold **Ctrl** and click near the front edge of the ellipsoid.



Position the sphere in the Top viewport.

This will start elevator mode.

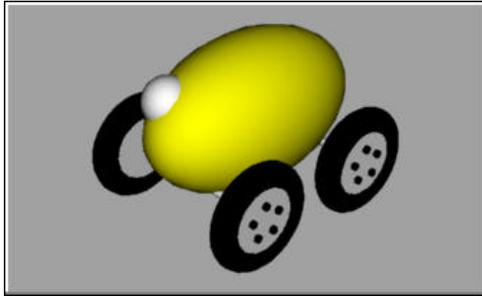
- 3 Move the cursor to the **Front** viewport, drag the sphere near the top edge of the ellipsoid, and click.



Drag it up in the Top viewport.

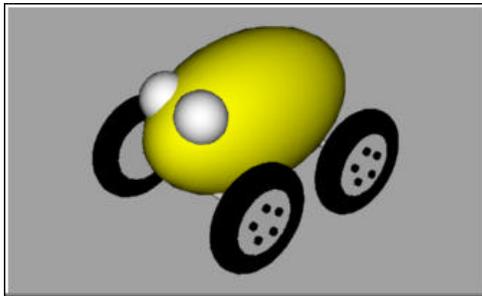
- 4 At the **Radius (Diameter)** prompt, type **1.5** and press **Enter**.

- 5 Render the **Perspective** viewport.



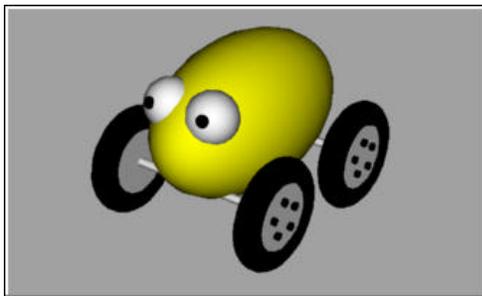
You now have one eye.

- 6 In the **Top** viewport, **Mirror** the sphere to the other side of the ellipsoid. Use the same procedure you used to mirror the wheel hub.
- 7 Render the **Perspective** viewport.



Mirror the eye.

- 8 Use the same technique to make two more small black pupils. Make one **Sphere**, move it if you have to. Change its color to black and mirror it.



Add pupils.

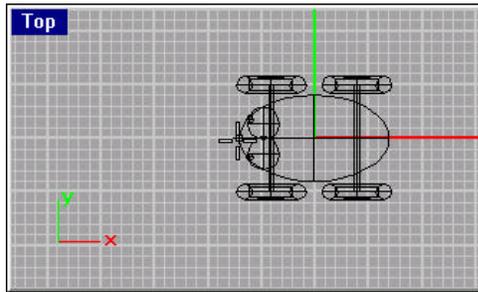
Make the Pull Cord

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

To create the pull cord at the front of the toy:

- 1 Zoom out in all the viewports, you are going to need some space to work.
- 2 From the **Curve** menu, click **Free-form**, and then click **Control Points**.

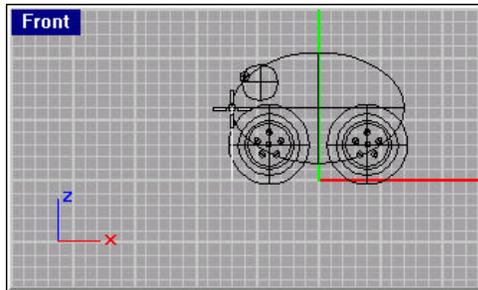
- 3 At the **Start of curve** prompt, in the **Top** viewport, hold **Ctrl** and click near the front end of the ellipsoid.



Start a curve.

- 4 Move the cursor to the **Front** viewport, drag the curve near the end of the ellipsoid, and click.

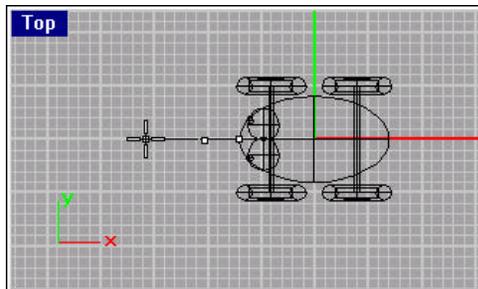
This is similar to the vertical ortho constraint, but lets you draw off the construction plane.



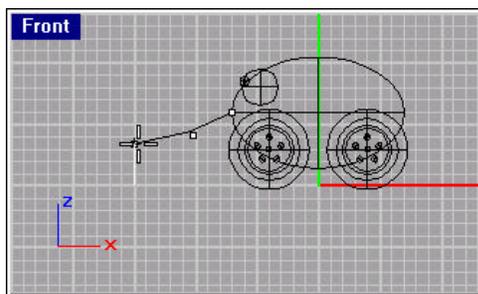
Use elevator mode to move point up in Front viewport.

- 5 At the **Next point, press Enter when done (Undo)** prompt, click to the left of the ellipsoid in the **Top** viewport.

Use elevator mode to move it up. Watch the curve in the Top and Front viewports.

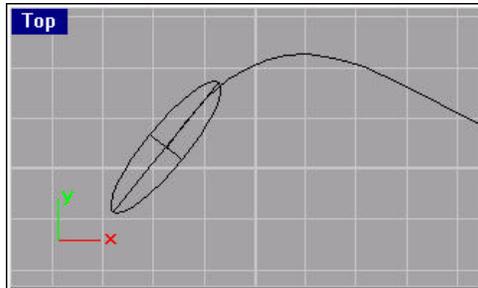


Add more points using elevator mode.



Drag up in the Front viewport.

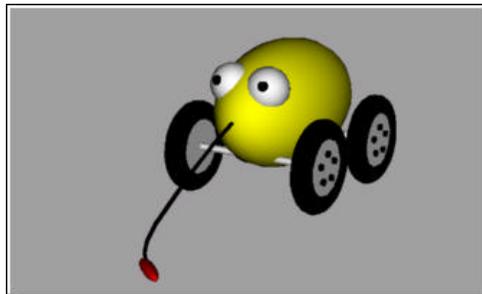
- 6 At **Next point, press Enter when done (Close Undo)** prompts, click several more points to create a curved line. Experiment with elevator mode. You want a free-form curve.
- 7 Draw an **Ellipsoid** to represent a handle at the end of the curve.



Draw a small ellipsoid for the handle.

Make the cord fat:

- 1 Select the curve you just made from the front of the pull toy.
- 2 From the **Solid** menu, click **Pipe**.
- 3 At the **Starting radius <1> (Diameter Cap=yes Thick=No)** prompt, type **.125** and press **Enter**.
- 4 At the **End radius <.125> (Diameter)** prompt, press **Enter**.
The pipe will be the same diameter for the full length of the curve.
- 5 Change the cord color to black and the handle color to any color you like.
- 6 **Render** the **Perspective** viewport.



The finished pull toy.

Try On Your Own

Using what you know, try drawing some things of your own. You should be able to use the basic Rhino commands.

6 Create Surfaces

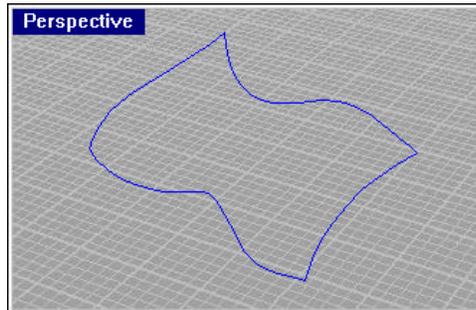
You can draw surfaces directly by placing corner points or by drawing a rectangular plane. However, most surfaces you create in Rhino will be based on a curve or another surface.

Rhino contains a variety of surface constructions, including free-form surfaces fit to points, tapered offset surface constructions, rolling-ball fillets, and blend surfaces.

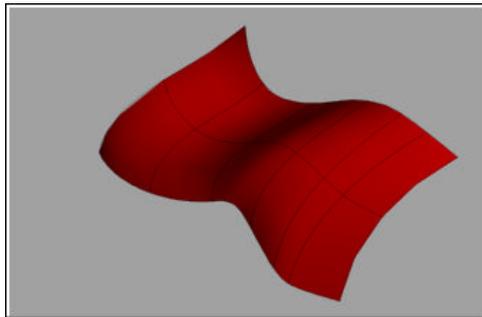
Note When using any of these creation methods, if the curves you are using have kinks, the result will be a polysurface instead of a surface. You cannot turn on control points for editing a polysurface.

Create a Surface From Edge Curves

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



The curves.



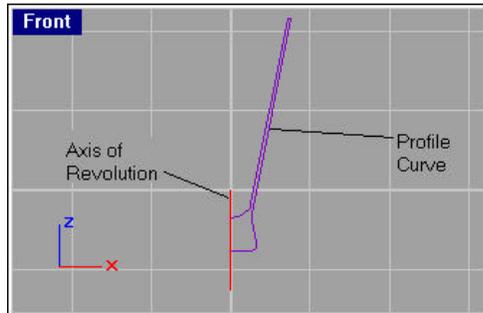
The surface created from the curves.

To try this command:

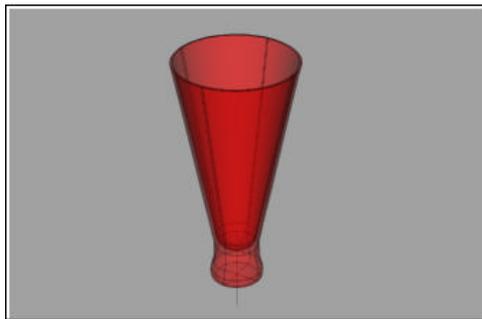
- 1 Open **Edge Curves** in the **Tutorials** folder.
- 2 From the **Surface** menu, click **Edge Curves**.
- 3 At the **Select 2, 3, or 4** curves prompt, select the curves.

Revolve a Curve Around an Axis

Revolving a curve creates a surface by revolving a shape curve about an axis. The resulting object can be either an open surface or a closed solid depending on whether or not the shape curve is open or closed and how far you revolve it. In this example, both endpoints of the shape curve intersect the axis of revolution, so a solid is created. This is sometimes called *lathing*.



The profile curve and axis.



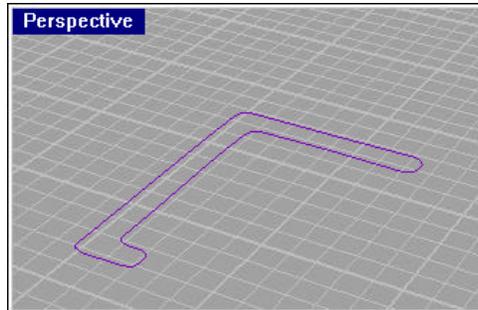
The revolved closed polysurface.

To try this command:

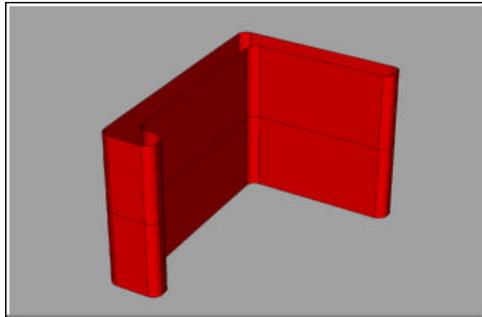
- 1 In the **Tutorials** folder, open **Revolve.3dm**.
- 2 From the **Surface** menu, click **Revolve**.
- 3 At the **Select curve to revolve** prompt, select the profile curve.
- 4 At the **Start of revolve axis** prompt, snap to one end of the axis of revolution line.
- 5 At the **End of revolve axis** prompt, snap to the other end of the axis of revolution line.
- 6 In the Rotate Options dialog box, click **OK**.

Extrude a Curve Straight

Extrude creates a surface from a curve. If the curve is planar, the extrusion is perpendicular to the plane of the curve. If the curve is not planar, the extrusion direction depends on the active construction plane.



Planar curve.



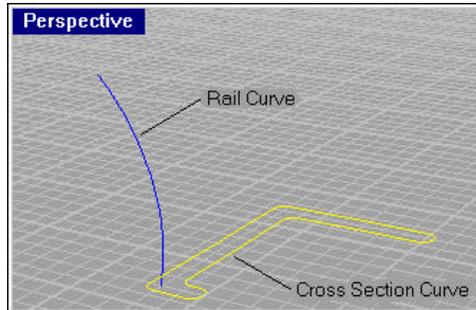
Extruded surface.

To try this command:

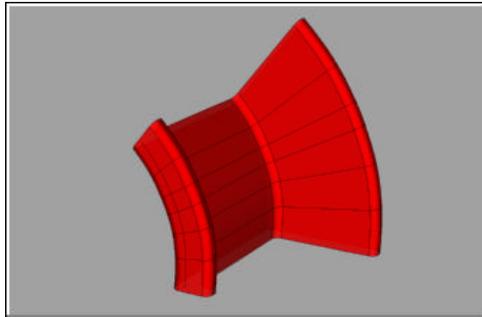
- 1 In the **Tutorials** folder, open **Extrude.3dm**.
- 2 From the **Surface** menu, click **Extrude**, and then click **Straight**.
- 3 At the **Select curves to extrude** prompt, select the purple curve, and press **Enter**.
- 4 At the **Extrusion distance (Direction Cap=No BothSides Tapered)** prompt, drag a distance and click.

Sweep a Curve 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. Note the difference between the surface created with this method and the previous Extrude example that used the same path curve.



The rail curve and the cross section curve.



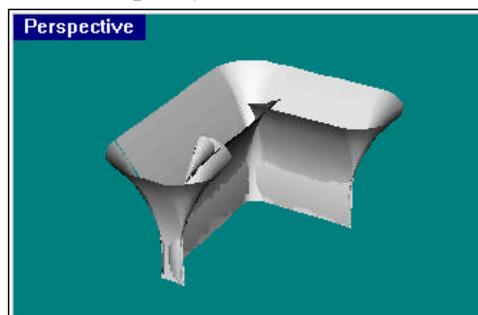
The swept surface.

To try this command:

- 1 In the **Tutorials** folder, open **1Rail.3dm**.
- 2 Select both curves.
- 3 From the **Surface** menu, click **Sweep 1 Rail**.
- 4 In the Sweep 1 Rail Options dialog box, click **OK**.

The Sweep 1 Rail may not be able to decide which curve is the rail and which is the cross section curve.

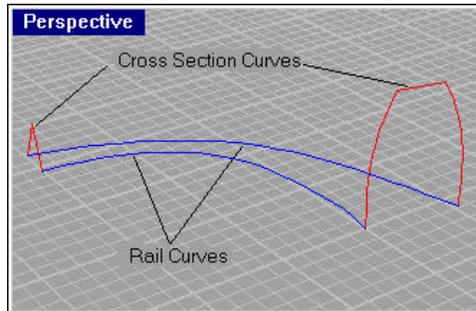
Your sweep may look like this:



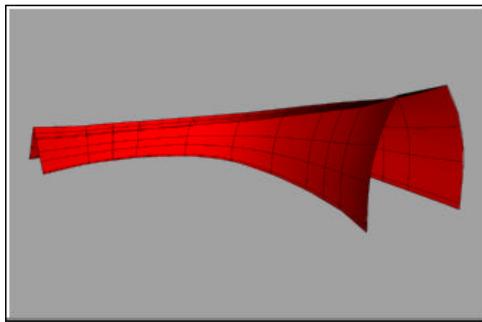
If this happens, start the Sweep 1 Rail command again and follow the prompts to select the rail curve and the cross section curve separately.

Sweep a Curve with Two Rails

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.



Cross section and rail curves.



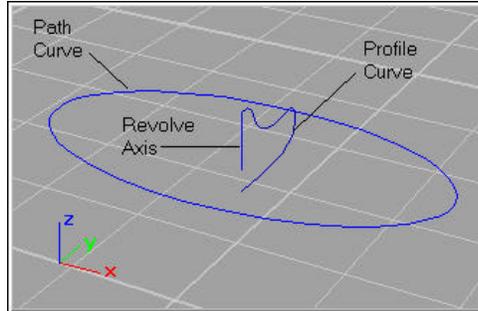
The swept surface.

To try this command:

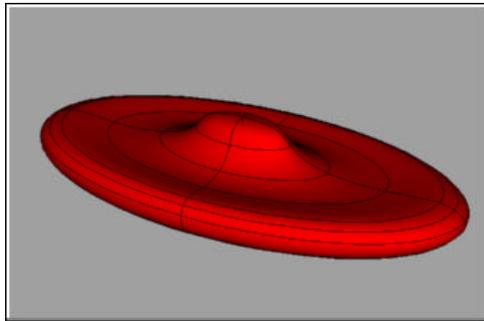
- 1 In the **Tutorials** folder, open **2Rail.3dm**.
- 2 From the **Surface** menu, click **Sweep 2 Rails**.
- 3 At the **Select 2 rail curves** prompt, select the two rail curves.
- 4 At the **Select cross-section curves (Point)** prompt, select the two cross-section curves, and press **Enter**.
- 5 In the Sweep 2 Rails Options dialog box, click **OK**.
Or, click **Shaded Preview** and then click **Preview**.

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 a Sweep Along 2 Rails, except one of the rails is a central point.



The profile curve, path curve, and revolve axis.



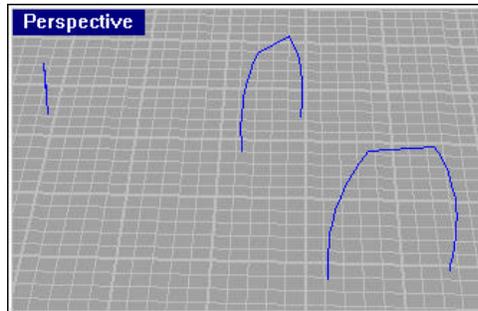
The revolved closed polysurface.

To try this command:

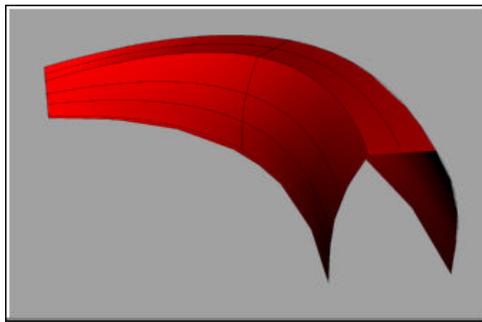
- 1 In the **Tutorials** folder, open **RailRev.3dm**.
- 2 From the **Surface** menu, click **Rail Revolve**.
- 3 At the **Select profile curve** prompt, select the profile curve.
- 4 At the **Select path curve** prompt, select the path curve the revolve will follow.
- 5 At the **Start of revolve axis** prompt, snap to an endpoint of the revolve axis line.
- 6 At the **End of revolve axis** prompt, snap to the other end of the revolve axis line.

Loft a Surface Through Curves

Lofting creates a smooth surface that blends between selected shape curves. This surface looks similar to the example in the next section, “Sweep a Curve with Two Rails,” but is created without rail curves. Instead, the edges of the surface are created by fitting smooth curves through the shape curves.



Curves to loft.



The lofted curve.

To try this command:

- 1 In the **Tutorials** folder, open **Loft.3dm**.
- 2 Select the three curves.
- 3 From the **Surface** menu, click **Loft**.
- 4 In the Loft Options dialog box, click **OK**.
Or, try some of the **Style** options, click **Shaded Preview**, and then click **Preview** to see the various loft styles.

When to Use Loft and When to Use Sweep

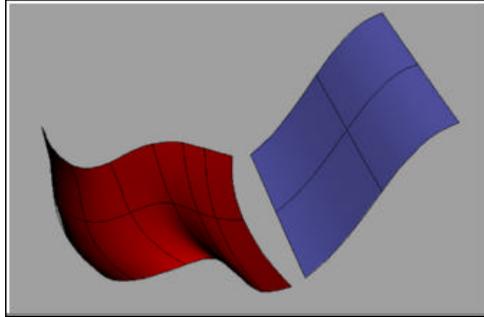
A surface created with **Loft** normally has no creases. Unless the cross-cross section curves have kinks, the surface is smooth. The Straight sections style of loft creates shapes with creases at shape curves and straight lines connecting the shapes. This is sometimes called a *ruled* surface.

If you find yourself adding lots of extra shape curves to make the shape you want, you might want to use **Sweep 1 Rail** instead. The rail curve directs the surface along it.

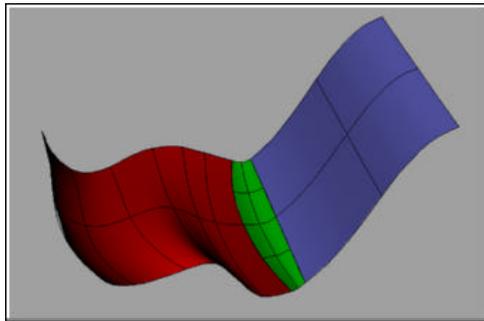
If you are having trouble getting the edges of a loft or one-rail sweep to go where you want, use a **Sweep 2 Rails**. The two-rail sweep lets you select the surface's edges.

Blend a Surface Between Two Surfaces

You can create a blend between two surfaces that will smoothly meet at the surface edges.



Two surfaces.



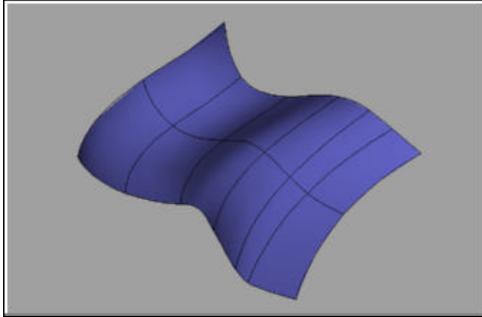
A surface blend between the two surfaces.

To try this command:

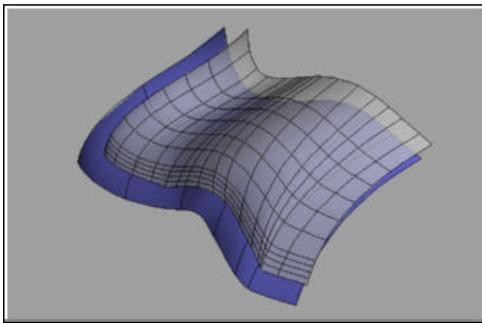
- 1 In the **Tutorials** folder, open **BlendSrf.3dm**.
- 2 From the **Surface** menu, click **Blend**.
- 3 At the **Select edge to blend - pick near one end (PlanarSections Degree=Quintic)** prompt, select an edge to blend.
- 4 At the **Select edge to blend - pick near one end (PlanarSections Degree=Quintic)** prompt, select the edge on the other surface.

Offset a Surface

You can offset a surface a specified distance to create a new surface.



A surface.



The new surface is offset two units.

To try this command:

- 1 In the **Tutorials** folder, open **OffsetSrf.3dm**.
- 2 Select the surface.
- 3 From the **Surface** menu, click **Offset**.
- 4 At the **Offset distance** prompt, type **2**.

The surface is offset in the direction of the surface normal.

To view the direction of the surface normal, select the surface, and then from the **Analyze** menu, click **Direction**.

Other Ways to Create Surfaces

Rhino includes many other tools for creating surfaces. In addition to a blend, you can create fillets or chamfers between two surfaces. You will learn some of these by going through the exercises. Others are designed for special tasks. See the topics *Draw Surfaces* in Help.

7 Features

Features

Rhino can create, edit, analyze, and translate NURBS curves, surfaces, and solids in Windows. There are no limits on complexity, degree, or size.

Features include:

User interface: extremely fast 3-D graphics, unlimited viewports, working perspective views, coordinate read-out, named views, named construction planes, customizable icons and toolbars, extensive on-line help, electronic updates, newsgroup support.

Construction aids: unlimited undo and redo, exact numeric input, object snaps, grid snap, ortho, planar, construction planes, layers, background bitmaps, object hide/show, object lock/unlock.

Create curves: point, line, polyline, free-form curve, circle, arc, ellipse, rectangle, polygon, helix, spiral, conic, TrueType text, point interpolation, control points (vertices), sketch.

Create curves from other objects: extend, fillet, chamfer, offset, blend, from 2 views, cross section profiles, intersection, contour, section, border, silhouette, extract isoparm, projection, pullback, sketch, wireframe, detach trim, create 2-D drawings, flatten developable surfaces.

Edit curves: control points, edit points, handlebars, smooth, fair, change degree, add/remove knots, add kinks, rebuild, refit, match, simplify, change weight, make periodic, adjust end bulge, adjust seam.

Create surfaces: from 3 or 4 points, from 3 or 4 curves, from planar curves, rectangle, extrude, ribbon, rule, loft, sweep along a path, sweep along two rail curves, revolve, rail revolve, blend, patch, drape, point grid, heightfield, fillet, chamfer, offset, TrueType text.

Edit surfaces: control points, handlebars, change degree, add/remove knots, match, extend, merge, join, untrim, rebuild, shrink, make periodic, Boolean (union, difference, intersection).

Create solids: box, sphere, cylinder, tube, pipe, cone, truncated cone, ellipsoid, torus, extrude planar curve, extrude surface, cap planar holes, join surfaces, TrueType text.

Edit solids: fillet edges, extract surface, Booleans (union, difference, intersection).

Create polygon meshes: from NURBS surface, from closed polyline, mesh face, plane, box, cylinder, cone, and sphere.

Edit polygon meshes: explode, join, weld, unify normals, apply to surface.

Edit tools: cut, copy, paste, delete, delete duplicates, move, rotate, mirror, scale, stretch, align, array, join, trim, split, explode, extend, fillet, chamfer, offset, twist, bend, taper, shear, orient, flow along curve, smooth, project, object properties.

Analysis: point, length, distance, angle, radius, bounding box, normal direction, area, area centroid, area moments, volume, volume centroid, volume moments, curvature graph, geometric continuity, deviation, naked edges, nearest point.

Rendering: flat shade, smooth shade, render with textures, spotlights and shadows, and customizable resolution.

File formats supported: DWG, DXF, 3DS, LWO, STL, OBJ, AI, RIB, POV, UDO, VRML, BMP, TGA, JPG, IGES (Alias, Ashlar Vellum, AutoShip, Breault, CAMSoft, Catia, Cosmos, FastSurf, Intergrity Ware, MasterCAM, ME30, Mechanical Desktop, Microstation, NuGraf, Pro/E, SDRC I-DEAS, Softimage, Solid Edge, Solidworks, SurfCAM, TeKSoft, Unigraphics).

File management: Notes, templates, merge files, and export selected objects.

I/O plug-ins: 3D Studio MAX, Softimage, and programmer's I/O tool kit with source code.

3-D digitizing support: MicroScribe 3D and Faro Space Arm.

Rhino is a Companion to Other Design Programs

Rhino is designed for designing and creating 3-D models. Although it has some rendering capabilities that can be quite useful, this is not Rhino's primary function. In addition, Rhino you cannot generate 2-D drawings with annotation and dimensioning; you must import your model into a CAD program to accomplish that task.

Rhino is a companion for other modeling and rendering software. Rhino gives you full support for trimmed NURBS surfaces, a robust IGES translator, and a streamlined interface that lets you model quickly. Rhino's free-form modeling complements:

- Parametric feature-based modelers
- Rendering and animation software
- Computer-aided manufacturing (CAM) software
- Finite element analysis (FEA) software

Technical Support and Additional Documentation

One of the most important learning methods for Rhino is to try the commands on the menu and to read the description of the command in the Help.

Rhino Help

Rhino commands are documented in Rhino Help. If you have a question about a specific command, look first in Help. Many topics such as importing and exporting Rhino models to other software, using a digitizing arm, tracing bitmaps to create curves, and using Rhino analysis tools are only documented in Help.

To access Help:

- ◆ From the Rhino **Help** menu, click **Help Topics** or **Command List**.
Or, press **F1**.

The Rhino Web Site

The Rhino Web site contains a gallery of images, short examples, white papers on technical issues, links to Rhino related web sites, and news about Rhino. Connect to the Rhino Web site at the following location: *www.rhino3d.com*

To connect to the Rhino Web site directly from Rhino:

- ◆ From the Rhino **Help** menu, click **Rhino Web Site**.

The Rhino Newsgroup

Ask questions about using Rhino and get answers from other users and Rhino technical support at the following newsgroup location:
news://news.rhino3d.com/rhino

To connect to the newsgroup directly from Rhino:

- ◆ From the Rhino **Help** menu, click **Rhino Technical Support**.

Technical Support

Technical support is also available via e-mail.

E-mail: *rhino@mcneel.com*

Ordering information

The Rhino Web site contains ordering information. You can order from a dealer in your local area. Connect to the Rhino Web site at the following location:
www.rhino3d.com/sales