Rhinoceros®

NURBS modeling for Windows

Version 1.0
Training Manual
Level 1

rhinolevel 1.doc

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Part One: Introduction

1 Before You Start

This course guide accompanies the Level 1 training sessions in Rhino Release 1. Level 1 shows you how to produce 3-D models in Rhino.

In class, you will receive information at an accelerated pace. For best results, practice at a Rhino workstation between class sessions, and consult your Rhino reference manual for additional information.

Course Objectives

In Level 1, you learn how to:

- Customize your modeling environment
- Create basic graphic objects—lines, circles, arcs, curves, solids, and surfaces
- Draw with precision using coordinate input and object snaps
- Modify objects with edit commands
- Display any portion of the model
- Import and merge models
- Output models to different file formats
- Render models

Notes:

Part Two:
Getting Started With Rhino

2 Rhino Basics

The Rhino for Windows Interface

Before learning individual tools, we will get acquainted with the Rhino interface. The following exercises examine the interface elements used in Rhino, the Rhino window, viewports, menus, toolbars, and dialog boxes. There are many ways to access the commands in Rhino—the keyboard, menus, and toolbars. We will focus on the toolbars in this class.

To open Rhino:

- Double-click the Rhino Icon from the Windows desktop.

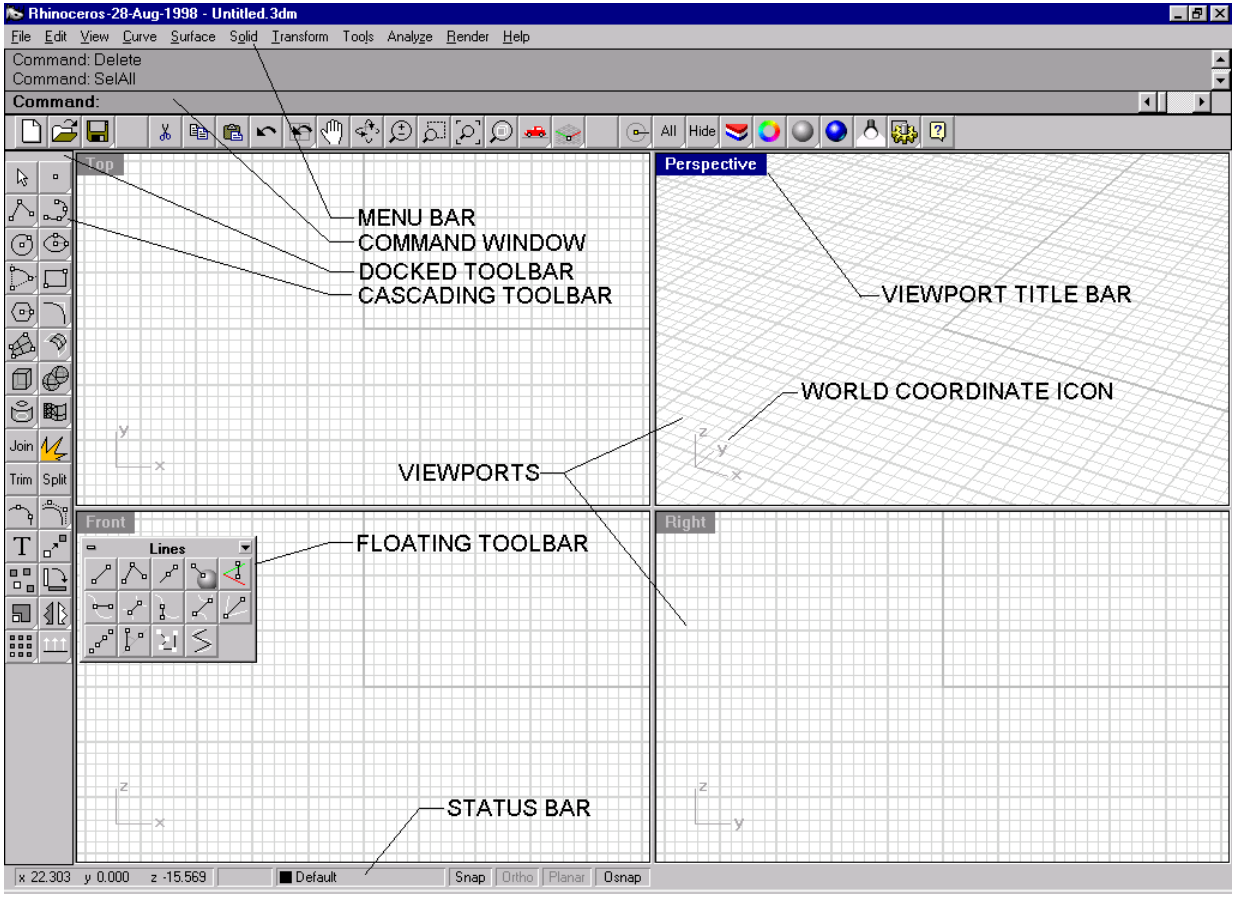
The Rhino Screen

Rhino divides its window into six areas that supply information or prompt you for input.

Screen Area	Description
Menu Bar	Access commands, options, and help.
Toolbars	Access shortcuts to commands and options.
Graphics area	Displays the open model. Several viewports can be displayed. The default viewport layout displays four viewports (Top, Front, Right, Perspective).

Notes:

Screen Area	Description
Viewports	Displays different views of the model within the graphics area.
Command line	Lists commands you enter and information produced.
Status bar	Displays the coordinates of the pointer, the status of the model, options, and toggles.



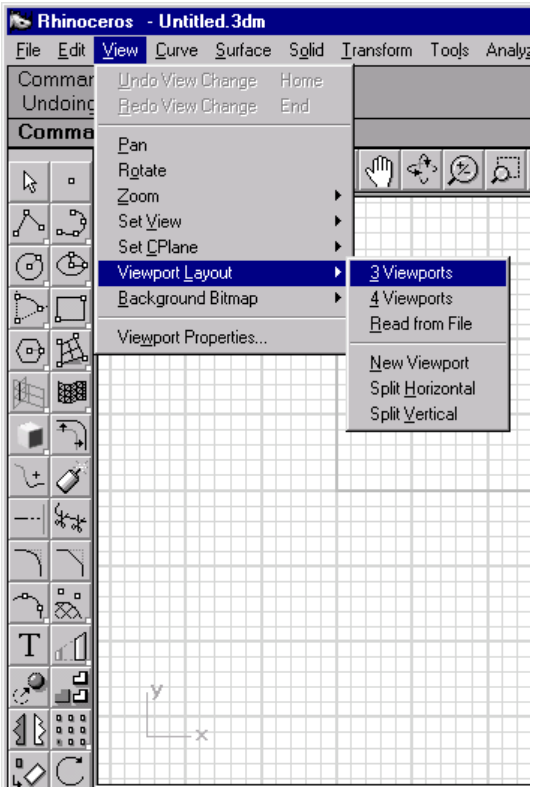
Rhino screen

Notes:

Watch the command line to find out what's happening.

Menus

Most of the Rhino commands can be found in the menus.



Rhino View menu

Toolbars

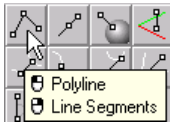
Rhino toolbars contain buttons that provide shortcuts to commands. You can float a toolbar anywhere on the screen, or dock it at the edge of the graphics area

Rhino starts up with the Main 1 Toolbar docked above the graphics area and the Main 2 toolbar docked on the left.

Notes:

Tooltips

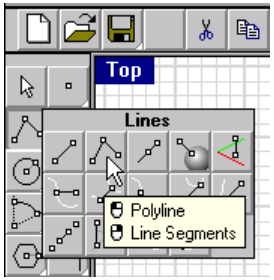
Tooltips tell what each button does. Move your pointer over a button without clicking it. A small yellow tag with the name of the command appears. In Rhino, many buttons can execute two commands. The tooltip will indicate which buttons have dual functions with a vertical line between the two commands.



Flyouts

Buttons on a toolbar may include other command options in a flyout. Usually the flyout contains variations on the base command. After you select a button on the flyout the flyout disappears.

Buttons with flyouts are marked with a small white triangle in the lower right corner. Open the flyout toolbar by holding down the left button for a second or by pushing both buttons at the same time.



The Lines toolbar is linked to the Main 2 Toolbar.

If you hold the left mouse down the flyout toolbar will open.

Graphics Area

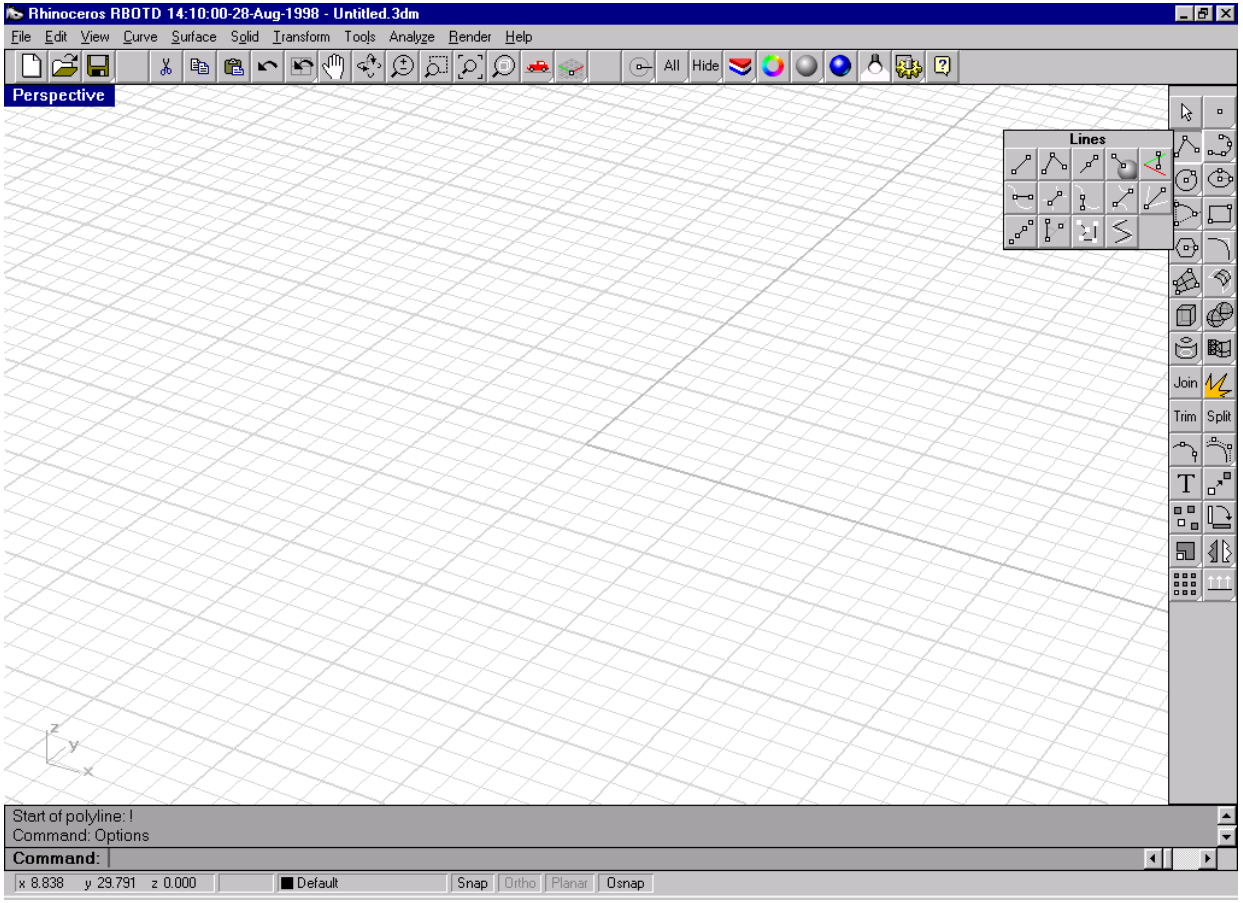
The Rhino graphics area can be customized to suit your preferences. The position of viewports can be arranged in different configurations.

Notes:

Viewports

Viewports are windows in the display area that show you views of your model. You can move and resize viewports by dragging the title bar or borders. Each viewport has its own construction plane that the cursor moves on, and a projection mode. You can create new viewports, rename viewports, and use predefined viewport configurations.

You can toggle between a small viewport and one that fills the graphics area by double-clicking the viewport title bar.



Rearranged Rhino screen. Command line at the bottom, single viewport, toolbars placed in different locations.

Notes:

Command Window

The command window displays commands and command prompts. It can be positioned at the top or the bottom of the screen. The command window shows three lines by default.

The Mouse

In a Rhino viewport, the left mouse button selects objects, and the right mouse button enters. Use the left mouse button to select objects in the model, commands or options on the menus, and buttons in the toolbars. Use the right mouse button to complete a command, to move between stages of commands, and to repeat the previous command. The right mouse button is used to initiate some commands from toolbar buttons.

Drag with the right mouse button to **PAN** and **ROTATE** in viewports. Hold down the CTRL key and drag with the right mouse button to zoom in and out in a viewport. You must press and hold the right mouse button down to activate this feature.

Entering Commands

Use the command line to type commands, type command options, type coordinates, type distances, angles, or radii, type shortcuts, and view command prompts.

To enter information typed at the command line, press ENTER, SPACEBAR, or right mouse button.

Shortcuts are customizable key combinations. You can program the function keys and CTRL key combinations to perform Rhino commands.

Note In Rhino, ENTER and SPACEBAR perform the same function.

Entering options

To enter command options, type the first letters of the option. The interior capitalization is meaningless for entering the option. For example, for the AroundCurve option, type A not AC. You only need to type enough of the option word to differentiate it from other options that begin with the same letter.

Notes:

Tab to complete command name

Type the first few letters of a command name and press TAB. If the letters you type correspond to a unique command name, the command will be completed on the command line. If there is more than one command that starts with the letters you typed, Rhino completes as much as possible.

Repeating commands

You can repeat the last command by right-clicking in a viewport, or by pressing ENTER or SPACEBAR. You can repeat previous commands by right clicking in the command line window and selecting from a list.

Canceling commands

Cancel a command by pressing ESC, or by entering a new command from a button or a menu.

Help

Rhino Help can be accessed at any time by pressing the F1 key. In addition to finding information about each command, Rhino help has conceptual information as well as many examples and graphics to help you complete your model. When you get stalled for any reason, the first place you should look is the help file. You can also access help for a specific command by starting the command and then press the F1 key.

View the command line history

The command line history window lists the last 500 command lines from the current Rhino session. The command history can be viewed at any time by pressing the F2 key.

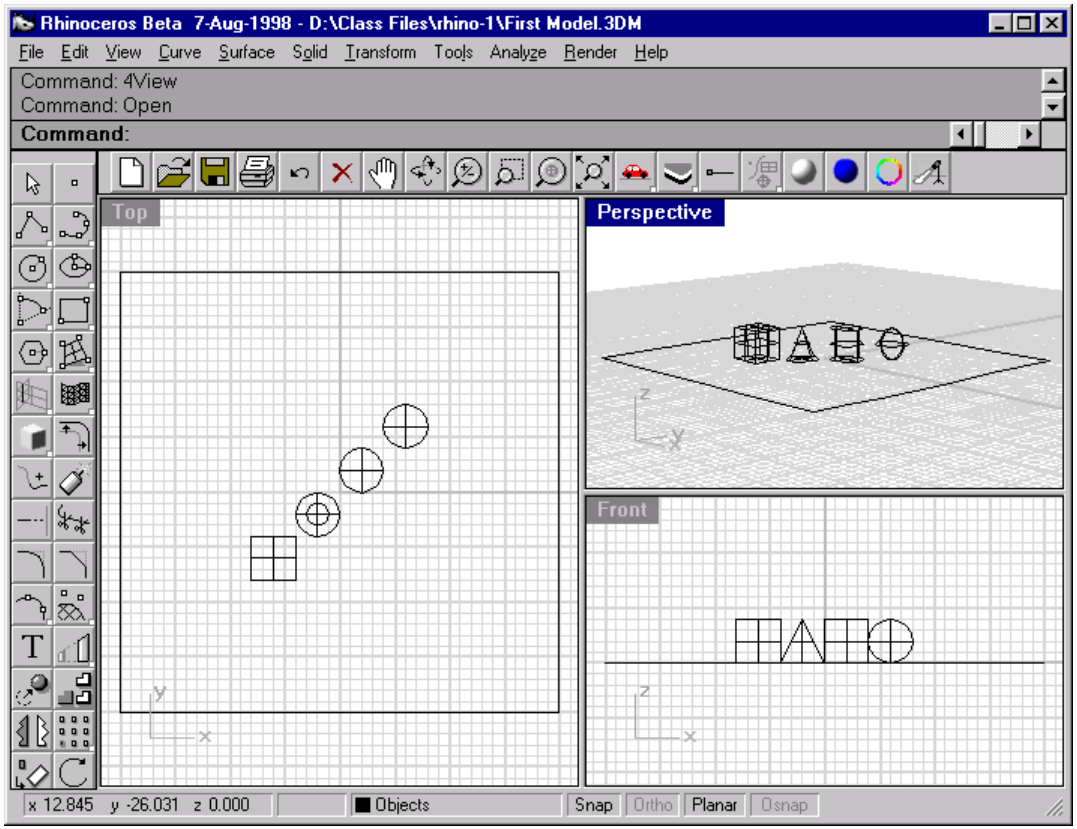
Notes:

Exercise 1—Rhino Basics

To look at your first Rhino model:

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

2 In the **Open** dialog box, select **First Model.3dm**.



Two parallel viewports and one perspective viewport.

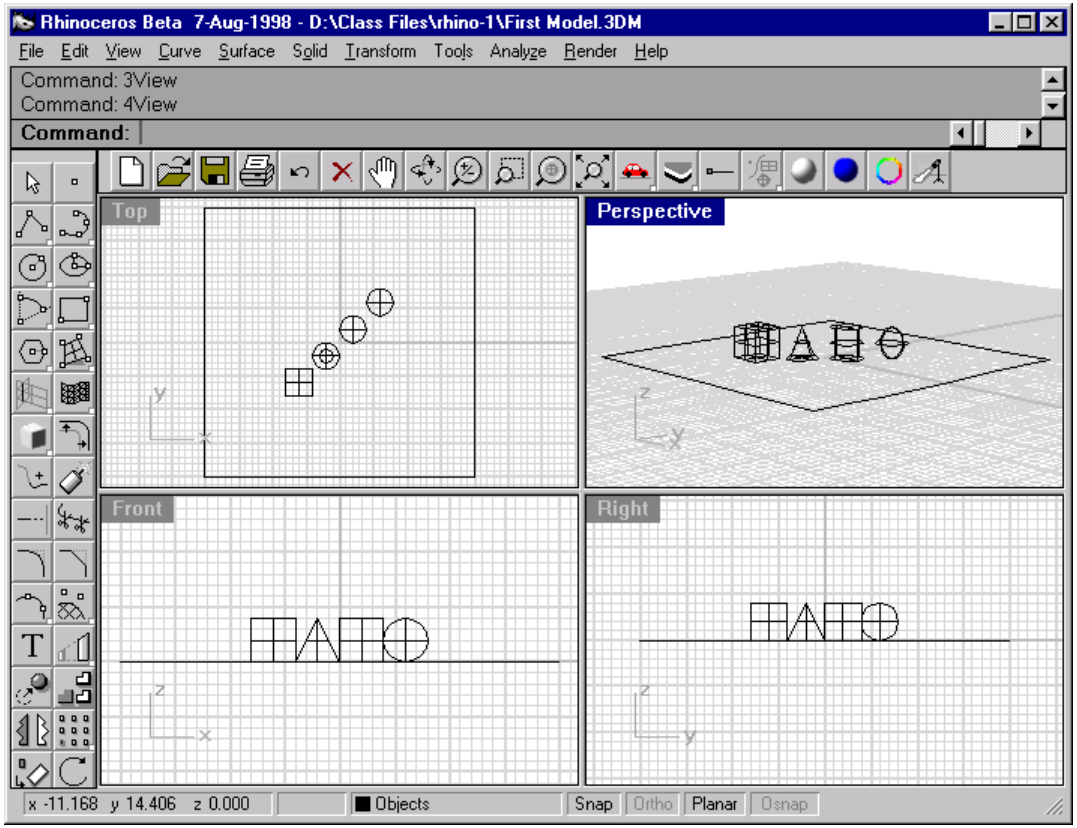
This model contains five objects: a cube, a cone, a cylinder, a sphere, and a rectangular plane.

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.

Notes:

Open
Look for this button.

3 From the **View** menu, click **Viewport Layout**, then click **4 Viewports.**



Three parallel viewports and one perspective viewport

4 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 your cursor move in certain intervals. In this model, by default 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.

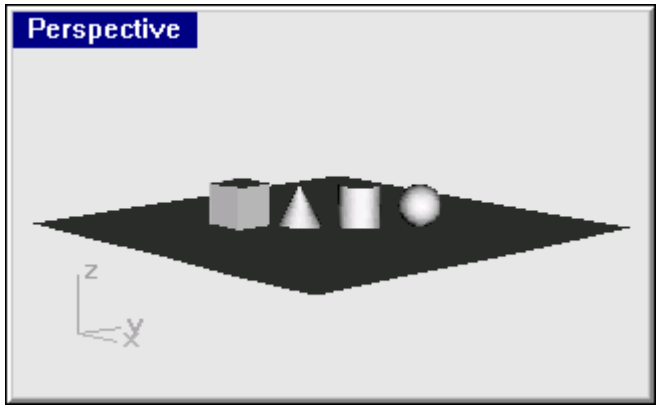
Notes:

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

The title bar highlights when it is active. An active viewport is the viewport where all your commands and actions take place.

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

The objects appear shaded. Shade lets you preview the shapes with no color. The model is shaded right in the viewport, and in shaded mode you can manipulate the view.



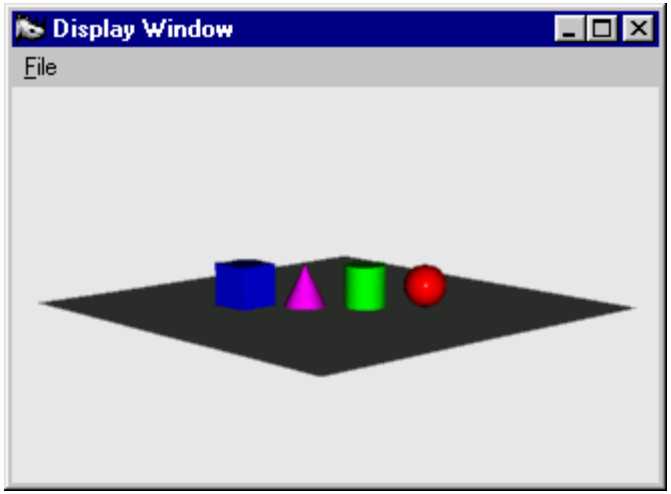
Shaded preview.

Notes:

Shade
Look for this button.

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

Rendering the model opens a separate display window. The model displays in colors previously assigned to the objects. You can also set spotlights and a background. You will learn about doing this later. You cannot manipulate the view in the render display window but the image can be saved to a file.



Render.

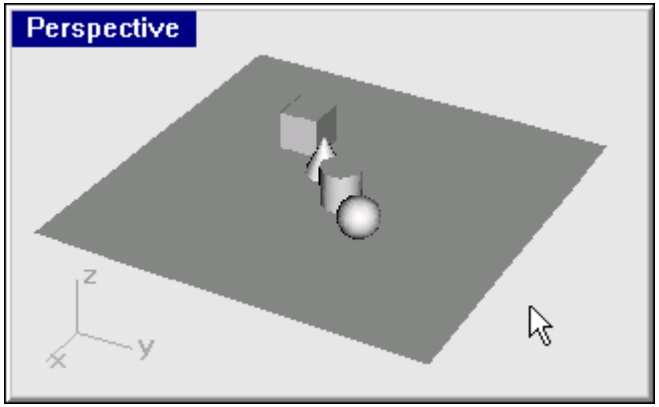
8 Close the render Display Window.

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

Notes:

Render
Look for this button.

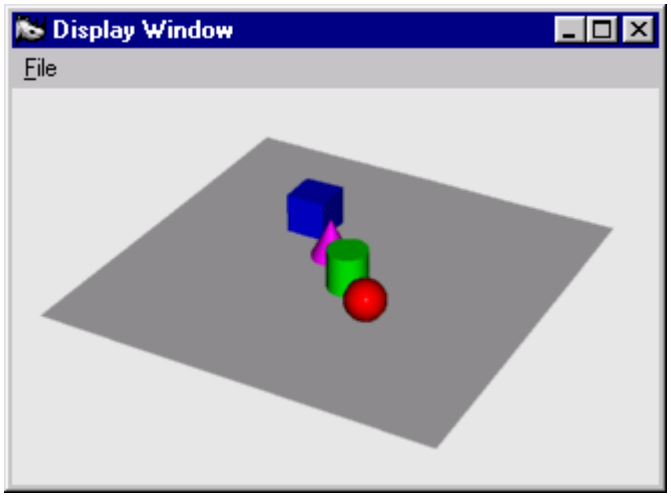
10 From the **Render** menu, click **Shade**.



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.

11 From the **Render** menu, click **Render**.

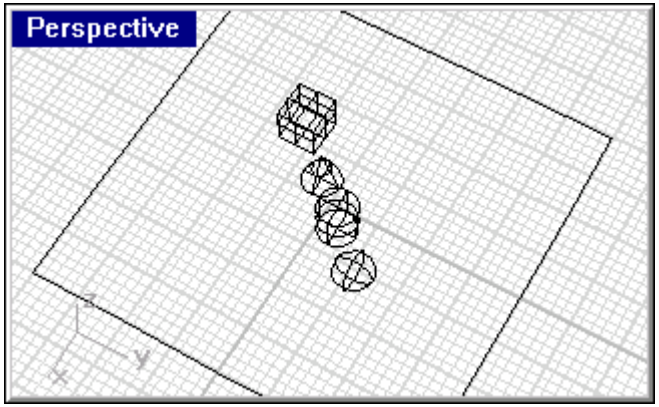


Render the rotated view.

12 Close the render Display Window.

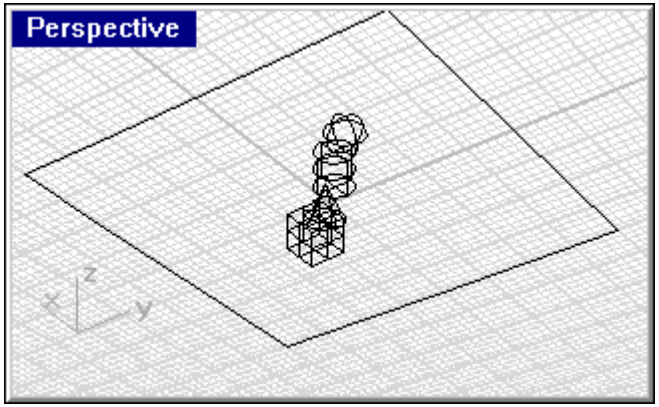
Notes:

13 In the **Perspective** viewport on the right, drag with your right mouse button to rotate the view around.



Rotating the wireframe view.

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



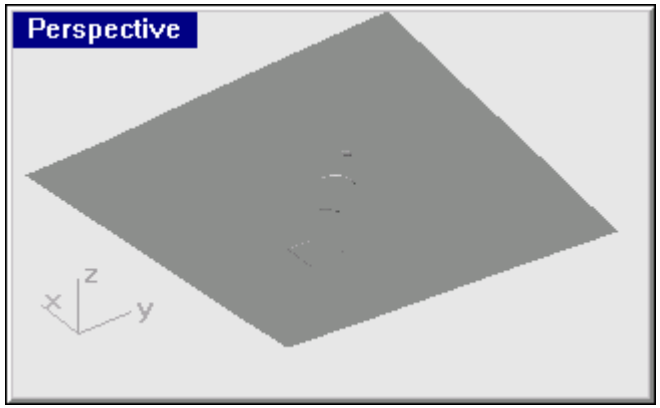
Looking at the objects from the bottom in wireframe mode.

You are now under the objects looking up.

Notes:

15 From the **Render** menu, click **Shade**.

The objects are obscured by the plane.



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.

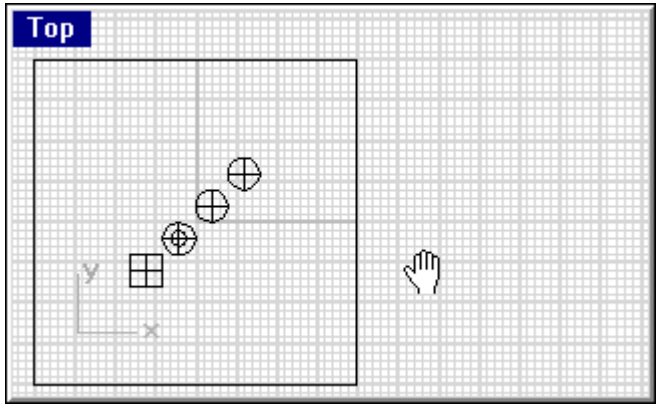
Navigating Around the Model

You have used the right mouse button to rotate in the Perspective viewport. You can hold SHIFT and drag with the right mouse button to pan. Dragging the right mouse button to move around does not interrupt any commands in progress.

Notes:

To pan in a viewport:

- In the **Top** viewport, hold SHIFT and 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.

Notes:

What if …

Instead of panning or rotating, something funny happened.

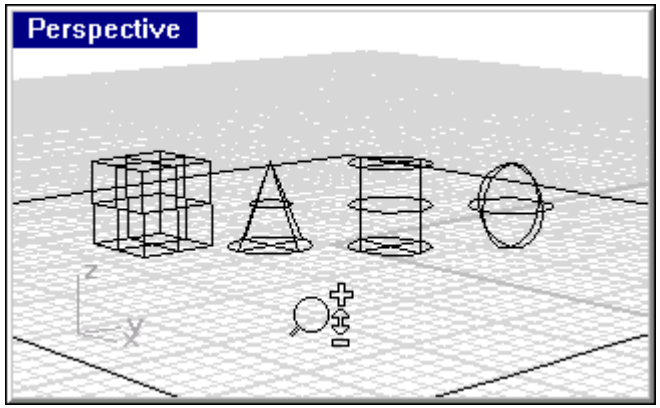
If you just right-click quickly once in viewport, the last command starts again. You must hold the right mouse button down while panning or rotating.

I panned in another viewport, 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.

Drag up to zoom in.

Drag down to zoom out.



Zooming with CTRL and the right mouse button.

Zooming Extents

The **Zoom Extents** command zooms a viewport so the objects fill up 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**.

Notes:

ZoomExtents
Look for this button.
Left Click this button.

Zooming Extents in the perspective viewport is sometimes inaccurate.

ZoomExtentsAll
Look for this button.
Right Click this button.

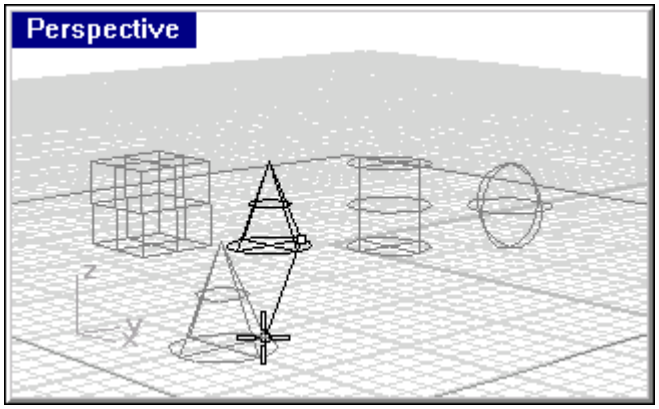
Move Objects

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 move objects:

1 Click the cone and drag it.

The cone turns yellow to show it is selected.



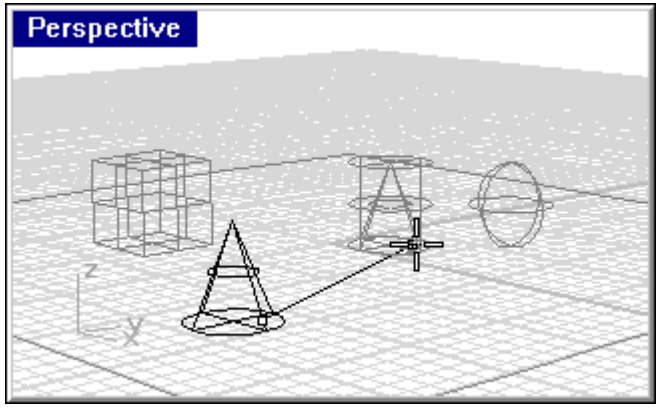
The selected cone highlights.

Notes:

2 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 has the same construction plane as the Top viewport. You will learn more about using construction planes later.



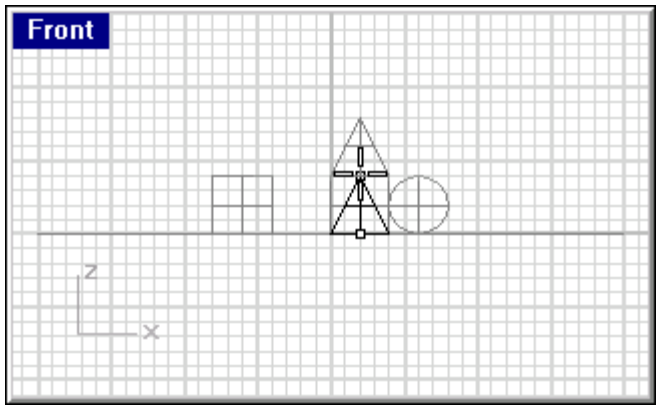
Drag the cone to move it.

Notes:

3 In the **Front** viewport, drag the cone to the top of the cylinder.

Watch what happens in the Perspective viewport.

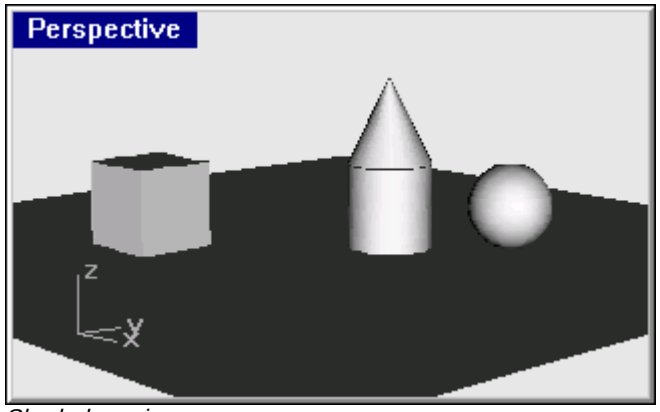
There are many times when you have to watch what is happening in other viewports to accurately place your objects.



Move the cone in the Front viewport.

4 Click in the **Perspective** viewport.

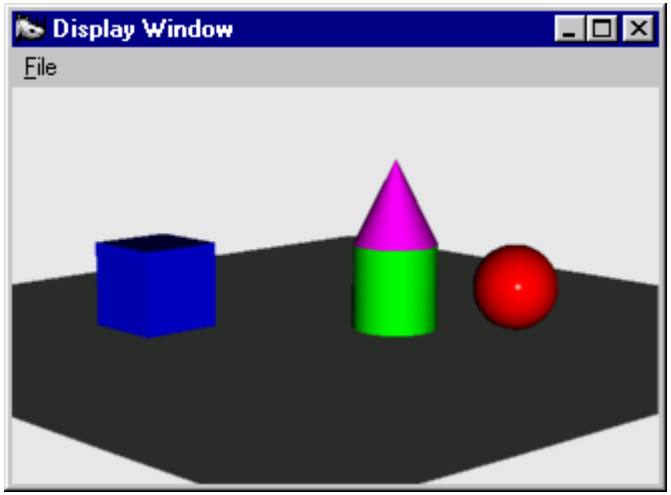
5 From the **Render** menu, click **Shade**.



Shaded preview.

Notes:

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



Render.

Try on Your Own

- Move the objects around. Use the Front viewport to move the objects vertically and the Top or Perspective viewport to move them horizontally.

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, select **First Model.3dm**.

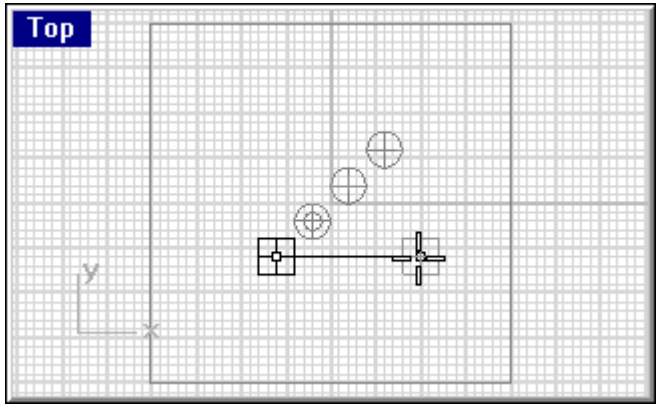
You will find this model in the **Rhino-1** folder.

Notes:

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.

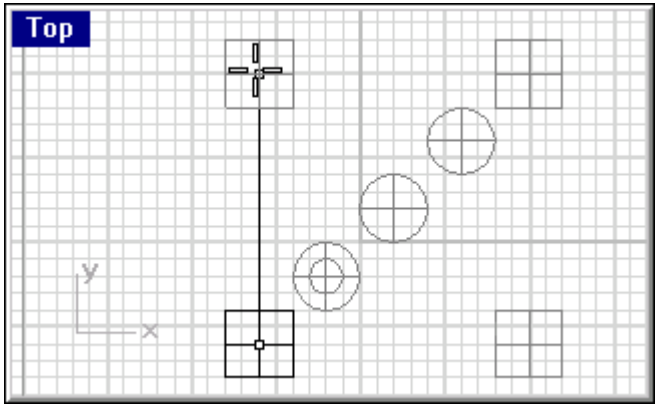
   Zoom in closer if you like.

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

Notes:

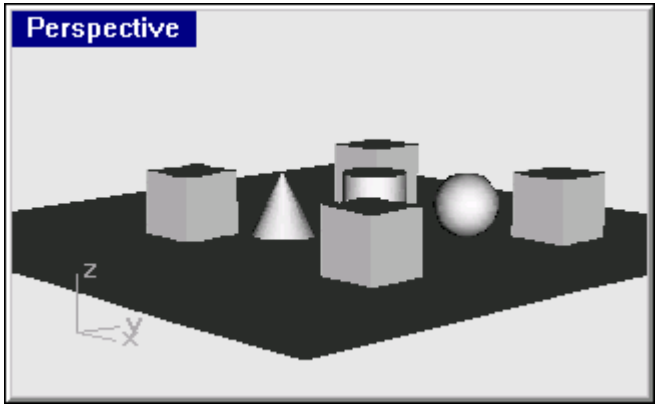
Copy
Look for this button.

5 At the next **Point to copy to** prompts, click other places to make some copies of the box.



Make three copies

6 When you have enough copies, press ENTER.



Shade the new boxes.

Try on Your Own

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

Notes:

3 Creating Two-dimensional Objects

Drawing Lines

The **Line**, **Lines**, and **Polyline** commands draw straight lines. **Line** draws a single line segment. **Lines** draws continuous line segments, end to end. **Polyline** draws a line with multiple vertices (a single linear object with multiple segments).

Exercise 2—Drawing Lines

1. From the **File** menu, click **New.**
2. In the template **File** dialog, click **4viewports.3dm**.
3. From the **File** menu, click **Save.**
4. In the **Save** dialog, type **Lines**, then click **Save**.

To draw line segments:

1. From the **Curve** menu, click **Line**, then click **Line Segments** to begin the Line command.
2. At the **Start of first line** prompt, pick a point in a viewport.
3. At the **End of line segment (Undo)** prompt pick another point in a viewport. A line segment appears between the two points.
4. At the **End of line segment, press ENTER when done (Undo)** prompt, pick another point .

Line Segments
Look for this button.
Right click for Line Segments

Notes:

5 At the **End of line segment, press ENTER when done (Close Undo)** prompt, continue pick points.

Additional segments appear. Each segment meets, but isn't joined to it, the previous segment.

Option	Description
Close	Closes the shape by drawing a segment from the last point picked to the first point picked. This ends the command.
Undo	Deletes the last point picked.

6 At the **End of line segment, press ENTER when done (Close Undo)** prompt, press ENTER to end the command.

To draw a polyline:

1 From the **Curve** menu, click **Line**, then click **Polyline** to begin the Polyline command.

2 At the **Start of polyline** prompt, pick a point .

3 At the next **Next point of polyline (Undo)** prompt pick another point .

4 At the **Next point of polyline, press ENTER when done (Undo)** prompt, pick another point .

5 At the **Next point of polyline, press ENTER when done (Close Undo)** prompt, continue to pick points.

Polyline
Look for this button.
Left click for Polyline.

To draw a single line segment:

1 From the **Curve** menu, click **Line**, then click **Single Line** to begin the Line command.

2 At the **Start of line (BothSides)** prompt, pick a point .

3 At the **End of line (BothSides)** prompt, pick another point.

4 The Single Line command ends after one segment is drawn.

Single Line
Look for this button.

To use the Close option:

1 From the **Curve** menu, click **Line**, then click **Line Segments** to begin the Line command.

2 At the **Start of first line** prompt, pick a point .

3 At the **End of line segment (Undo)** prompt pick another point .

4 At the **End of line segment, press ENTER when done (Undo)** prompt, pick another point .

Notes:

You can press the right button to enter.

5 At the **End of line segment, press ENTER when done (Close Undo)** prompt, type **C** and press ENTER. The line will close back to the original start point.

To use the Undo option:

1 From the **Curve** menu, click **Line**, then click **Polyline** to begin the Polyline command.

2 At the **Start of polyline** prompt, pick a point .

3 At the **Next point of polyline (Undo)** prompt pick another point .

4 At the **Next point of polyline, press ENTER when done (Undo)** prompt, pick another point .

5 At the **Next point of polyline, press ENTER when done (Close Undo)** prompt, type **U** and press ENTER. One segment of the polyline is removed.

6 At the **Next point of polyline, press ENTER when done (Undo)** prompt, pick another point .

7 At the **Next point of polyline, press ENTER when done (Undo)** prompt, press ENTER to end the command.

To use the Bothsides option:

1 From the **Curve** menu, click **Line**, then click **Single Line** to begin the Line command.

2 At the **Start of line (BothSides)** prompt, type **B** and press ENTER.

3 At the **Start of line** prompt, pick a point.

4 At the **End of line** prompt, pick another point. A segment is drawn with equal length on both sides of the start point.

Drawing Freeform Curves

The **Curve from Interpolated Points** and **Curve** commands draw freeform curves. Interpolated Points draws a curve through the points you pick. Curve uses control points to create a curve.

Exercise 3—Drawing Interpolated Curves

1 From the **Curve** menu, click **Freeform**, then click **Interpolate Points.**

2 At the **Start of curve (Tangent Knots=SqrtChord)** prompt, pick a point .

3 At the **Next point of curve (Tangent Knots=SqrtChord Undo)** prompt, pick another point .

4 At the next several **Next point of curve (Tangent Knots=SqrtChord Undo)** prompts, pick more points.

Notes:

Interpolated Points
Look for this button.

5 Or, type an option letter, and press ENTER.

Option	Description
Close	Closes the shape by drawing a segment from the last point picked to the first point picked. This ends the command.
Tangent	After choosing a point on another curve, the next segment will be tangent to the point you picked and end the command.
Undo	Deletes the last point picked.

Exercise 4—Drawing Curves from Control Points

1 From the **Curve** menu, click **Freeform**, then click **Control Points.**

2 At the **Start of curve** prompt, pick a point .

3 At the **Next point (Undo)** prompt, pick another point .

4 At the next several **Next point, press ENTER when done (Undo)** prompts, pick more points.

5 Or, type an option letter, and press ENTER.

Curve
Look for this button.

Notes:

Notice the points you pick are off the curve as control points.

Modeling Aids

Mode commands are modeling aids that you can toggle on or off by pressing shortcut keys, a function key, typing a single letter command, or picking a button.

Mode	Command	Function Key	Description
Ortho	O	F8 or SHIFT	Ortho on restricts cursor movement to the points at a specified angle from the last point created. The default angle is 90 degrees. Pressing and holding the shift key down toggles Ortho. If Ortho is set to on, the shift key toggles Ortho off. If Ortho is off, the shift key toggles Ortho on.
Grid	Grid	F7	Hides or shows a reference grid in the current viewport of the graphics screen at the construction plane. This toggle does not work if you are in the middle of a command.

Mode	Command	Function Key	Description
Snap	S	F9 or	Forces the pointer to snap on grid intersections.
Planar	P		This is a modeling aid similar to ortho. This helps you model planar objects even if the modeling command was normally able to draw non-planar objects by forcing input to be on a plane parallel to the construction plane that passes through the last point that you picked.

Modeling aids can be toggled on and off by clicking their mode buttons on the status bar:

Exercise 5—Drawing lines and curves using mode functions

1. Toggle Snap on and draw some lines.

   The cursor snaps to each grid intersection.

2. Toggle Snap off, toggle Ortho on and draw some lines and curves.

   You can only input points that are 90 degrees from the last point. By using grid snap and ortho toggles you can draw with precision. We will discuss other ways to get precision in a later session.

Model Setup

In Rhino you can create full-size models using precise measurements. You might need to change the modeling environment depending on the type of model you are creating; the default options may not always work.

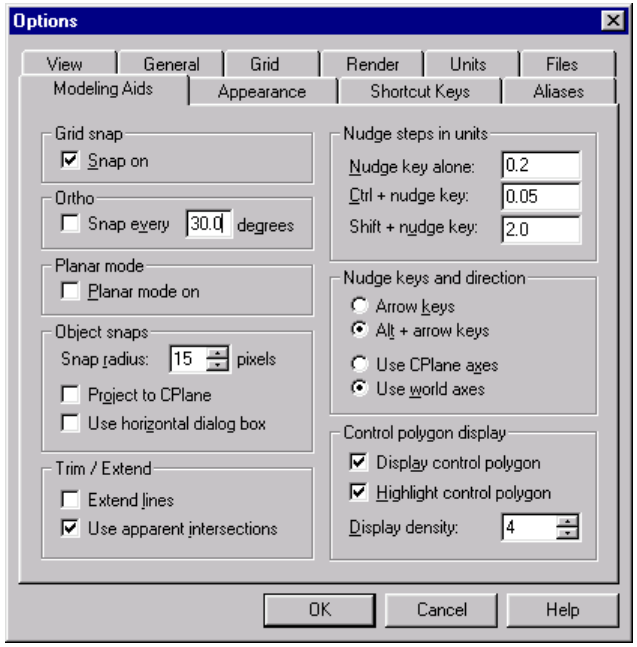
Notes:

To change the options:

1 From the **Tools** menu, click **Options**. The **Rhino Options** dialog box appears.

2 Click on the **Modeling Aids** tab.

The **Modeling Aids** tab allows you to control **ortho**, **object snap**, **grid snap**, and other mode options.

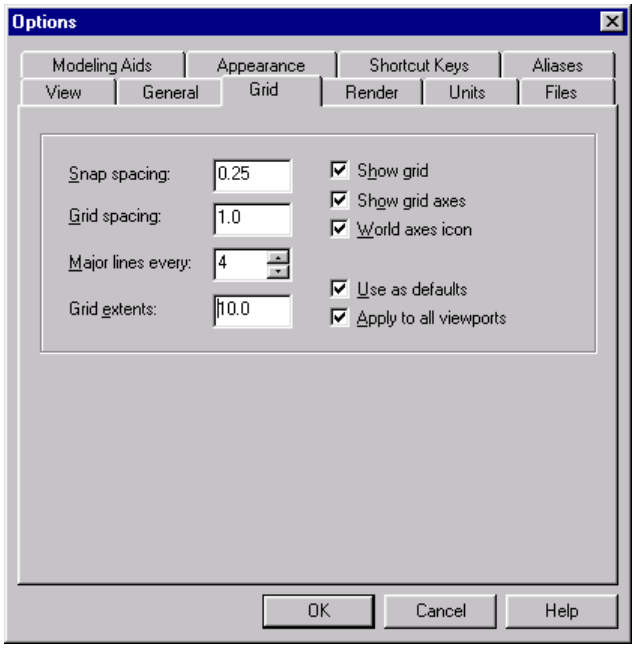


3 Change the **Ortho** options to **Constrain every 30** degrees.

Notes:

4 Click on the **Grid** tab.

You can change the appearance of the modeling environment by changing the grid elements. The grid size, the frequency of the thick lines, and the number of grid elements can be changed. The **Grid** dialog allows you to configure grid settings.



5 Change the **Snap Spacing** setting to **.25**.

6 Change the **Thick line every** setting to **4**.

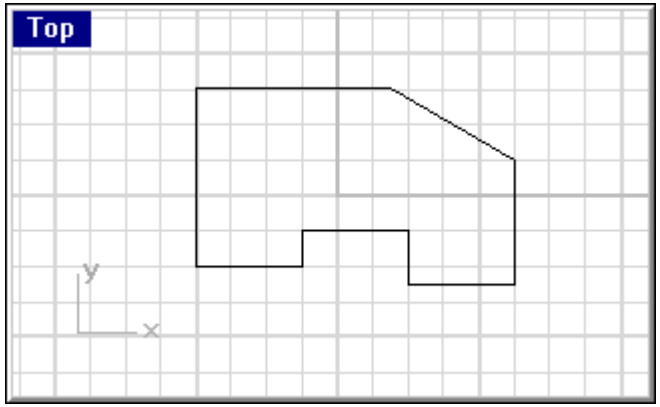
7 Change the **Grid Extents** setting to **10**.

8 Click **OK**.

9 Draw some more **lines** and **curves** with **Snap** and **Ortho** on. Notice that the cursor now snaps between the grid intersections and that ortho snaps at every 30 degrees.

Notes:

10 Try to draw the object below with snap and ortho turned on.



Saving your work

Save your work periodically during a session to keep it from getting accidentally deleted.

To save your model:

1 From the File menu, click Save.

Or, click one of the other options. You will have an opportunity to save your work.

Command	Description
Save	Saves your model and keeps it open.
Save As	Saves your model to a specified file name, location, and format.
New	Offers to save your model before opening a new one.
Open	Offers to save your model before opening a different one.
Exit	Offers to save your model before ending your Rhino session.

Notes:

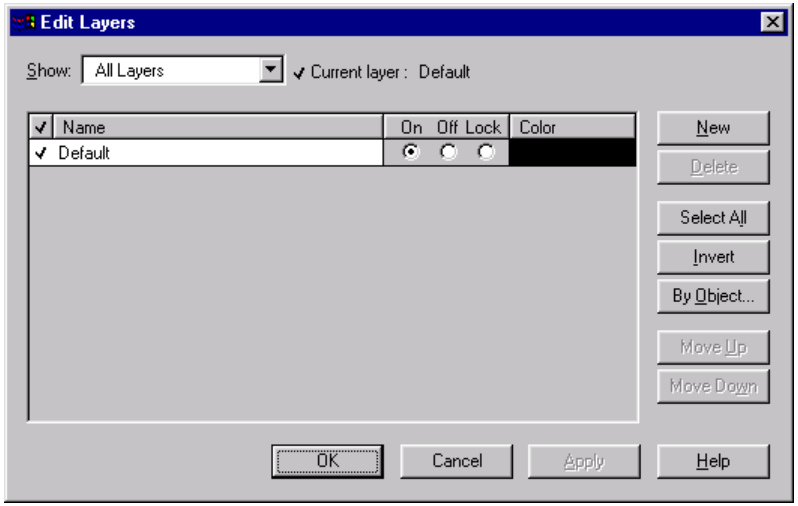
It is good practice to save your model in stages under different names, using the Save As command. This will allow you to go back to an earlier version of your model for modifications if necessary.

Layers

Rhino layers work like CAD layering systems. By creating objects on different layers, you can edit and view related portions of a model separately or as a composite. You can create as many layers as you like.

You can display all layers simultaneously or turn any of them off. You can lock layers so they are displayed but cannot be selected. Each layer has a color. You can assign a unique name to each layer (for example, BASE, BODY, TOP) to organize the model or you can use preset layer names (Default, Layer 01, Layer 02, Layer 03).

The Edit Layers dialog box manages layers. Use it to set up layers for your model.

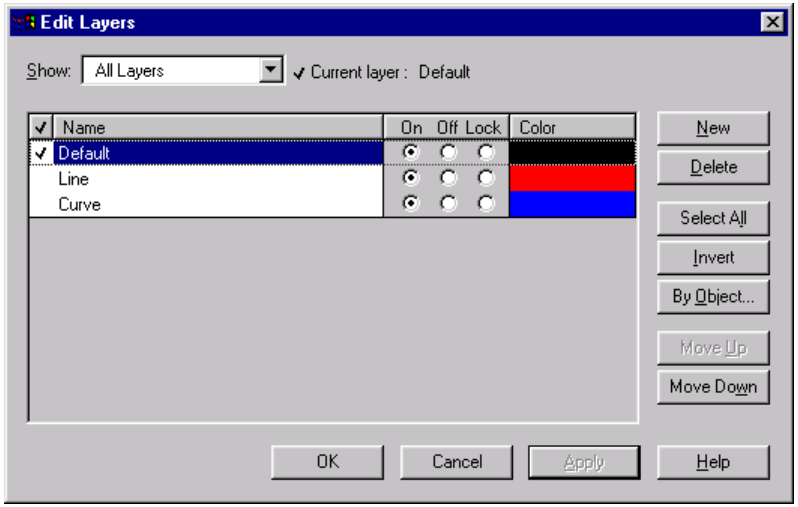


Notes:

Exercise 6—Layers

To create a new layer::

1 From the **Edit** menu, click **Layers**, then click **Edit.**



2 The layer Default is created automatically when you start a new model.

3 In the **Edit Layers** dialog box, click **New**.

4 The new layer **Layer 01** appears in the list, type **Line** and press ENTER.

5 Click **New**.

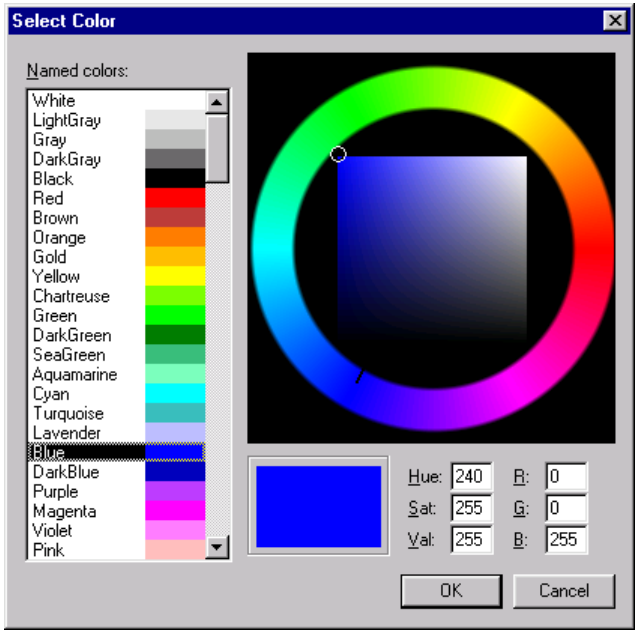
6 The new layer **Layer 02** appears in the list, type **Curve** and press ENTER.

Notes:

Edit Layers
Look for this button.

To assign a color to a layer::

1 Click the **Color** bar on the **Line** row in the list.



2 In the **Select A Color** dialog box, click **red** from the list.

The sample square turns red.

3 Click **OK**.

4 In the **Edit Layers** dialog box, the new color appears in the color bar on the **Line** row of the layer list.

5 Repeat steps 1–3 to make the **Curve** layer **Blue**.

6 Click **OK** to close the dialog box.

To make a layer current:

1 Click in the **Layer** pane of the **Status Bar**.

2 Click **Line**.

Notes:

Hue *is controlled by moving the line around the circular portion of the color wheel.*

Hue is the color that is referred to as a scale ranging from red through yellow, blue and green and then circularly back to red.

Saturation *and* ***Value*** *are controlled by moving the circle around in the square portion in the middle of the color wheel.*

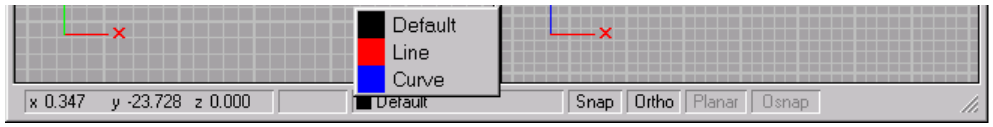
Saturation is the vividness of hue.

Value is the relative lightness or darkness of a color.

3 Draw some **lines**.

4 The lines appear on the **Line** layer and they are red.

5 Change layers by clicking the **Layer** area of the status bar. Click **Curve**.



6 Draw some curves.

7 Draw more lines and curves on each layer.

To lock a layer:

1 From the **Edit** menu, click **Layers**, then click **Edit.**

2 In the **Edit Layers** dialog box, click the radio button under **Lock** in the row for **Line**.

Locking a layer turns it into a reference only layer. You can see and snap to objects on locked layers. You cannot select any objects on locked layers. You cannot make a locked layer current without unlocking it.

3 Click **OK**.

To turn a layer off:

1 From the **Edit** menu, click **Layers**, then click **Edit.**

2 In the **Edit Layers** dialog box, click the radio button under **Off** in the row for Curve.

Turning a layer off makes all objects on it invisible.

3 Click **OK**.

Exercise 7—Selecting objects

To select an object:

♦ Move your pointer arrow over the object and left-click.

Notes:

Notes:

To select more than one object:

1. Move your pointer arrow over the first object and left-click.
2. While holding the SHIFT key down, move your pointer over another object and left-click.

To select more than one object using a window:

1. Move your pointer arrow into an open area to the lower left of the objects you want to select. Hold your left button down and drag diagonally to the right until you have several objects inside the window. Release your mouse button.

   All objects completely inside the window will be selected.
2. You can add to your selection set by holding the SHIFT key down while making another selection.
3. Or, type **Selwindow** and press enter to start a **Window** selection.

To select more than one object using a crossing window:

1. Move your pointer arrow into an open area to lower right of the objects you want to select. Hold your left button down and drag diagonally to the left until you have several objects inside or touching the window. Release your mouse button.

   All objects inside or touching the window will be selected.
2. You can add to your selection set by holding the SHIFT key down while making another selection.
3. Or, type **Selcrossing** and press enter to start a **Crossing Window** selection.

To hide an object:

1. Select an object.
2. From the **Edit** menu, click **Visibility**, then click **Hide.**

   The object becomes invisible.

Hide
Look for this button.
Left click for hide.

To show hidden objects:

- From the **Edit** menu, click **Visibility**, then click **Show.**
  The **Show** command shows all hidden objects globally.

Show
Look for this button.

To Lock an object:

1. Select an object.
2. From the **Edit** menu, click **Visibility**, then click **Lock.**
   The object becomes shaded gray. You can see the Locked object, you can snap to it, but you cannot select it.

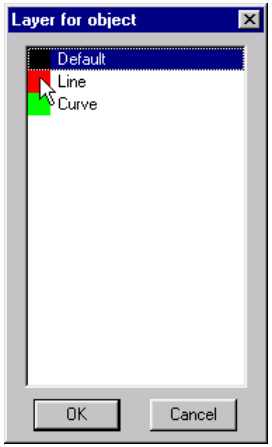
To Unlock locked objects:

- From the **Edit** menu, click **Visibility**, then click **Unlock.**
  The **Unlock** command redisplays all Locked objects globally.

To move an object from one layer to another:

1. Select an object.
2. From the **Edit** menu, click **Layers**, then click **Change Object Layer.**

   The **Layer for object** dialog appears.



3. Select the layer you want the object to move to, then press **OK**.

Notes:

Lock
Look for this button.

Unlock
Look for this button.
Right click for unlock.

Change Layer
Look for this button.

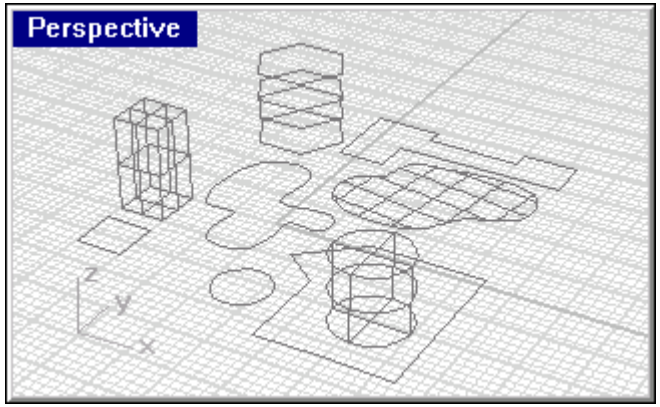
Deleting objects

Delete removes selected objects from the model. Use **Delete** to practice selection.

Exercise 8—Practice Using Delete and Selection options

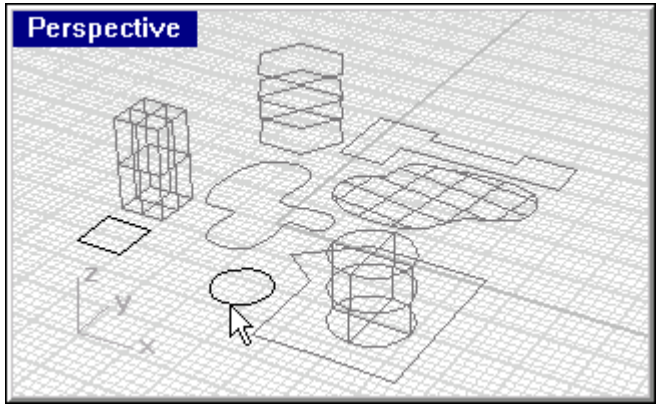
To open the practice model:

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



2 In the Open dialog box click **Delete.3dm** and click Open or double-click **delete.3dm** to open the model.

3 Select the square and the circle.



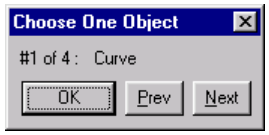
Notes:

4 From the **Edit** menu, click **Delete** or press the DELETE key.

The objects disappear.

To start the first practice:

1 Select one of the lines on the hexagon in the top view. Because there are several lines superimposed on each other, the choose between multiple objects dialog appears.

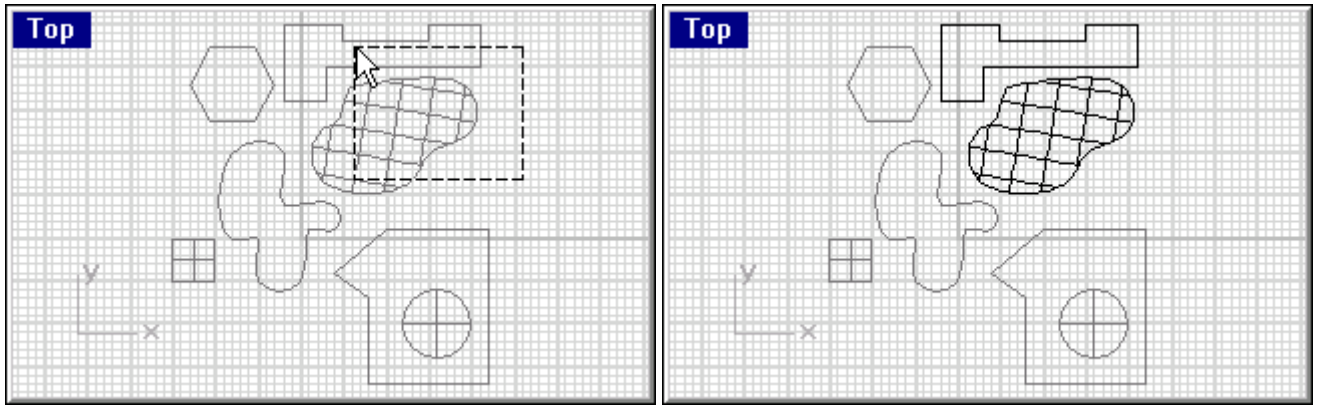


2 Click Next.

3 It will highlight the next item that is in the same location. You can see the change in the perspective or side view. When the hexagon on the top is highlighted, click **OK**.

4 From the **Edit** menu, click **Delete.**

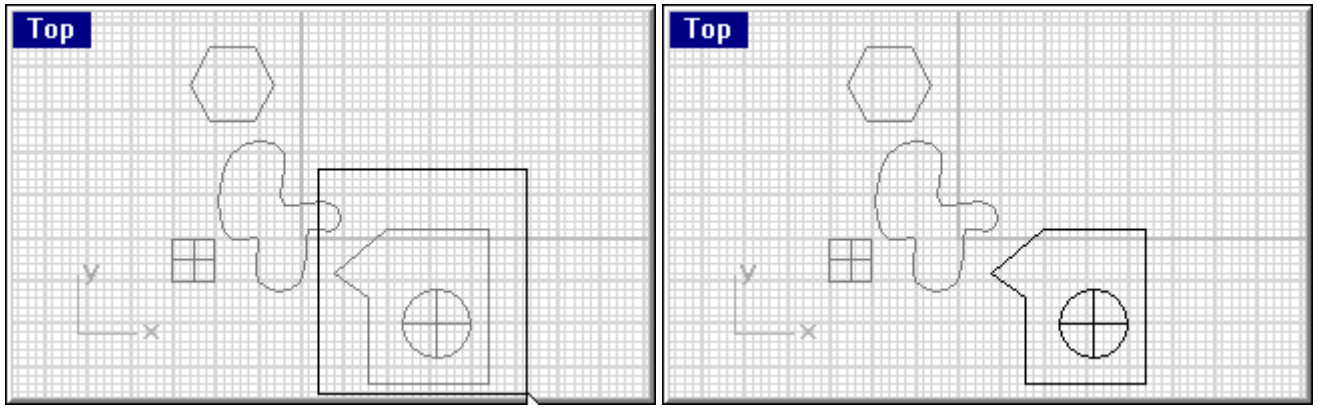
5 Make a crossing window to select the surface and the polyline in the top right part of the drawing. Both objects are selected.



6 From the **Edit** menu, click **Delete.**

Notes:

7 Make a window to select the polyline and the cylinder in the lower right part of the drawing. Only those objects that are completely inside the window are selected.



8 From the **Edit** menu, click **Delete.**

9 Continue deleting objects in the drawing. Practice using different selection methods to select and deselect objects. Use crossing and window. The SHIFT key while selecting will let you add to your selection set. The CTRL key while selecting will let you to remove objects from your selection set.

To undo deletions:

♦ From the **Edit** menu, click **Undo.**
Each time you click it will take you back one command sequence.

To redo:

♦ From the **Edit** menu, click **Redo.**
Each time you click it will redo the previous **Undo**.

Notes:

Undo
Look for this button.
Left click for undo.

Redo
Look for this button.
Right click for redo.

4 Precision Modeling

Exercise 9—Setting Up a Model

To start the new model :

1. From the **File** menu, click **New.**
2. Click **4viewports.3dm**, then click **Open**.
3. From the **File** menu, click **Save.** Name the model **BOXES**.

Modeling With Coordinates

So far you have been drawing imprecise lines. Now we will look at drawing lines at specific places. To do this you will use ***coordinates***.

Whenever you draw a curve, place a point, or create a solid primitive, Rhino asks you for a series of points. You can tell that Rhino is asking for point input two ways: the command prompt has a prompt like **Start of first line, Start of polyline, Start of curve, or Next point** and the cursor turns into a marker.

Marker

You can enter a point two ways: by picking a point in a viewport with the mouse, or by typing coordinates in the command line.

Rhino uses a fixed Cartesian coordinate system called the World Coordinate System (***WCS***), based on three axes (the ***x-, y-,*** and ***z-axes***) that define locations in two- and three-dimensional space.

Notes:

Each viewport has a construction plane that defines coordinates for that viewport. We will work in the Top (and Perspective) viewports where the two coordinate systems are the same.

Absolute Coordinates

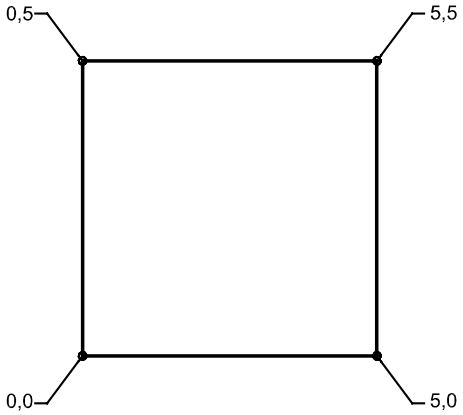
The first form of coordinates you will use are called ***absolute*** coordinates. Absolute coordinates are exact locations on the x-, y-, and z-axes.

Use the BOXES model to learn how to draw with absolute coordinates.

Exercise 10—Entering Absolute Coordinates

To draw lines using absolute coordinates:

1. Maximize the **Top** Viewport by double-clicking the title bar.
2. From the **Curve** menu, click **Line**, then click **Polyline.**
3. At the **Start of polyline** prompt, type **0,0**, and press ENTER.
4. At the **Next point of polyline (Undo)** prompt, type **5,0**, and press ENTER.
5. At the **Next point of polyline, press ENTER when done (Undo)** prompt, type **5,5**, and press ENTER.
6. At the **Next point of polyline, press ENTER when done (Close Undo)** prompt, type **0,5**, and press ENTER.
7. At the **Next point of polyline, press ENTER when done (Close Undo)** prompt, type **C**, and press ENTER, to close the line.



Notes:

Polyline
Look for this button.
Left click for Polyline.

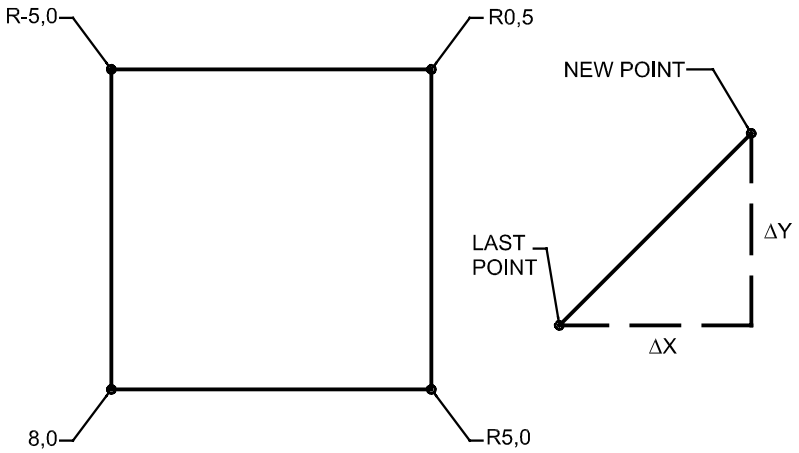
Relative Coordinates

Absolute coordinates can be slow and cumbersome, but they do work. Most of the time, ***relative*** coordinates are easier to use. Every time you select a point, Rhino saves that point as the ***last point***. Relative coordinates are based on this last point, instead of on the origin (0,0,0) of the CPlane.

Precede the x,y,z coordinates with a single "**R**" to enter relative coordinates.

Exercise 11—Entering Relative Coordinates

1. From the **Curve** menu, click **Line**, then click **Polyline.**
2. At the **Start of polyline** prompt, type **8,0**, and press ENTER.
3. At the **Next point of polyline (Undo)** prompt, type **R5,0**, and press ENTER.
4. At the **Next point of polyline, press ENTER when done (Undo)** prompt, type **R0,5**, and press ENTER.
5. At the **Next point of polyline, press ENTER when done (Close Undo)** prompt, type **R-5,0** and press ENTER.
6. At the **Next point of polyline, press ENTER when done (Close Undo)** prompt, type **C**, and press ENTER, to close the line.



Notes:

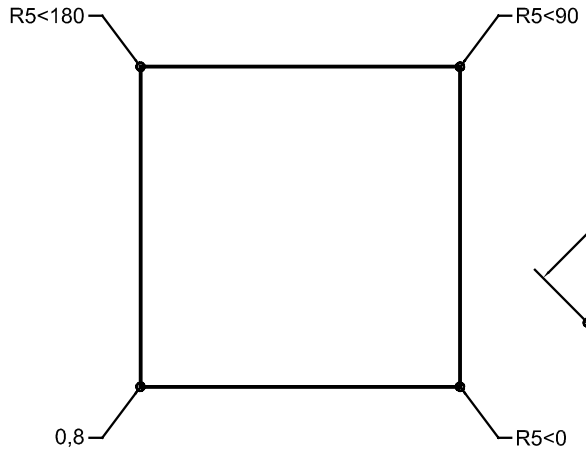
Polyline
Look for this button.
Left click for Polyline.

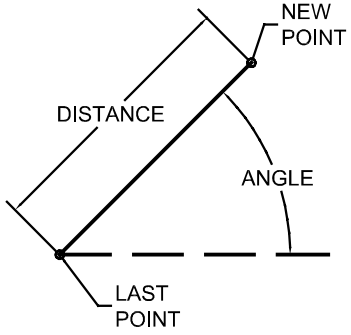
Polar Coordinates

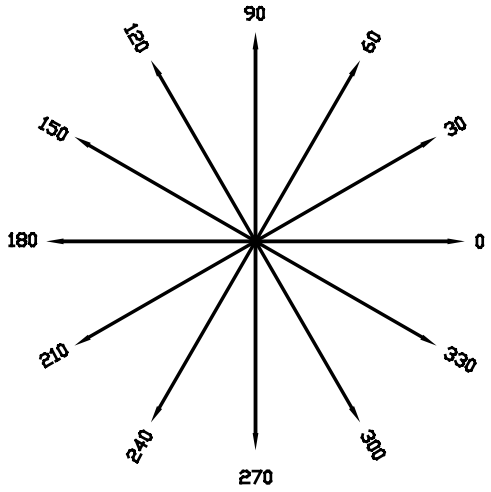
Polar coordinates specify a point that is a distance and direction away from 0,0 in the current construction plane. For example, if you want a point four units away from the construction plane origin, at a 45° angle clockwise from the construction plane x-axis, type 4<45, and press ENTER. Relative polar coordinates are preceded by the "r", absolute polar coordinates are not. Instead of using x-, y-, and z-coordinates, enter relative polar coordinates like this: ***rdistance<angle***.

Exercise 12—Entering Polar Coordinates

1. From the **Curve** menu, click **Line**, then click **Polyline.**
2. At the **Start of polyline** prompt, type **0,8**, and press ENTER.
3. At the **Next point of polyline (Undo)** prompt, type **R5<0**, and press ENTER.
4. At the **Next point of polyline, press ENTER when done (Undo)** prompt, type **R5<90**, and press ENTER.
5. At the **Next point of polyline, press ENTER when done (Close Undo)** prompt, type **R5<180**, and press ENTER.
6. At the **Next point of polyline, press ENTER when done (Close Undo)** prompt, type **C**, and press ENTER, to close the line.







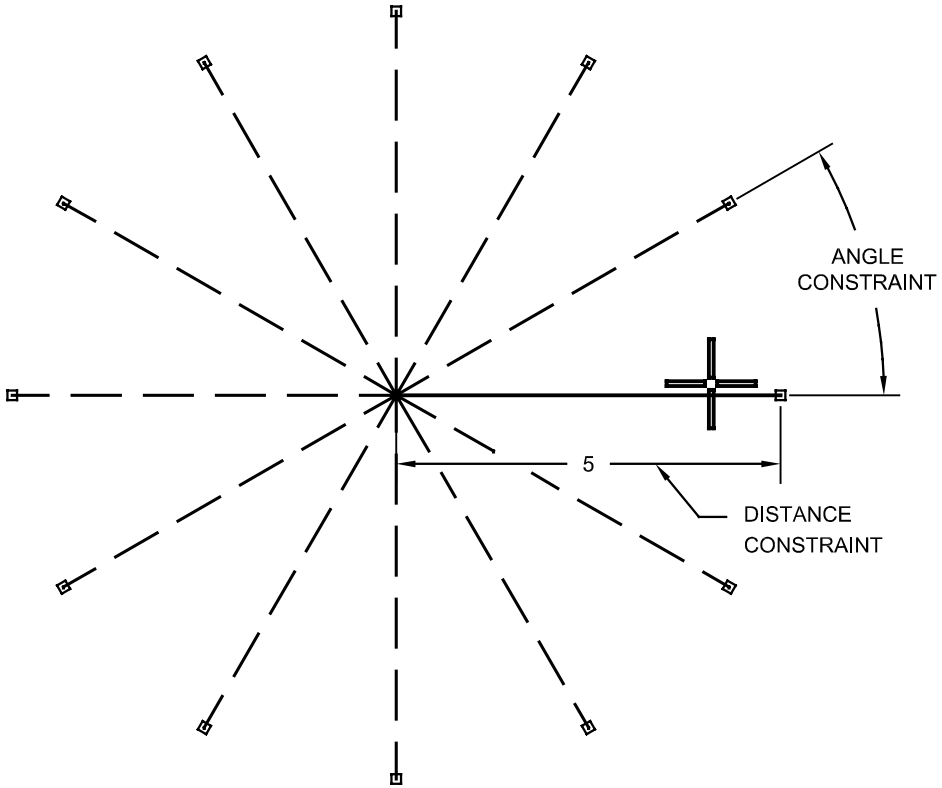
Notes:

Polyline
Look for this button.
Left click for Polyline.

Distance and Angle Constraint Entry

Using distance constraint entry, you can specify a point by typing a distance and pressing ENTER. As you move your cursor in any direction, the length of line will be constrained. This is a good way to specify a line length quickly.

Using angle constraint entry, you can specify an angle by typing < followed by a value and pressing ENTER. The next point is constrained to lines at the angle you specified.



Distance constraining value 5, angle constraint value 30

Notes:

Using the shift key to toggle Ortho on and off:

When Ortho is off you can hold the SHIFT key down to toggle it on. This method is an efficient way to draw perpendicular lines. In the following example, you draw a line 5 units long using distance constraints.

Exercise 13—Distance Constraint Entry

1. From the **Curve** menu, click **Line**, then click **Polyline.**
2. At the **Start of polyline** prompt, type **8,8**, and press ENTER.
3. At the **Next point of polyline (Undo)** prompt, type **5**, and press ENTER.
   Hold the SHIFT key down and pick a point 90 degrees to the right.
4. At the **Next point of polyline, press ENTER when done (Undo)** prompt, type **5**, and press ENTER.
   Hold the SHIFT key down and pick a point 90 degrees up.
5. At the **Next point of polyline, press ENTER when done (Close Undo)** prompt, type **5**, and press ENTER.
   Hold the shift key down and pick a point 90 degrees to the left.
6. At the **Next point of polyline, press ENTER when done (Close Undo)** prompt, type **C**, and press ENTER, to close the line.

Polyline
Look for this button.
Left click for Polyline.

Exercise 14—Distance and Angle Constraint Entry

1. From the **Curve** menu, click **Line**, then click **Polyline.**
2. At the **Start of polyline** prompt, type **16,5**, and press ENTER.
3. At the **Next point of polyline (Undo)** prompt, type **5**, and press ENTER, then type <45 and press ENTER.
   As you drag your cursor around it snaps to a distance of 5 and an angle of 45 degrees.
   Pick a point 45 degrees down and to the right.
4. At the **Next point of polyline, press ENTER when done (Undo)** prompt, type **5**, and press ENTER, then type <45 and press ENTER.
   Pick a point 45 degrees up and to the right.
5. At the **Next point of polyline, press ENTER when done (Close Undo)** prompt, type **5**, and press ENTER, then type <45 and press ENTER.
   Pick a point 45 degrees up and to the left.
6. At the **Next point of polyline, press ENTER when done (Close Undo)** prompt, type **C**, and press ENTER, to close the line.

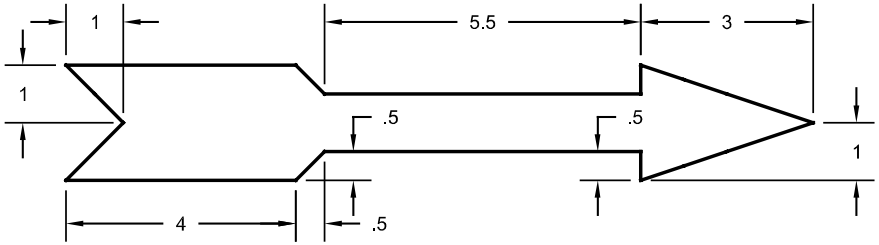
Polyline
Look for this button.
Left click for Polyline.

Notes:

7 **Save** your model. We will use this model for another exercise.

Exercise 15—Practice

1 Start a new model. Save as **Arrow**.



2 Draw the arrow with a polyline, using a combination of **absolute** coordinates (x,y), **relative** coordinates (**R**x,y), **polar** coordinates (**R**distance<angle), and **distance constraint**. Begin your model at **0,0**.

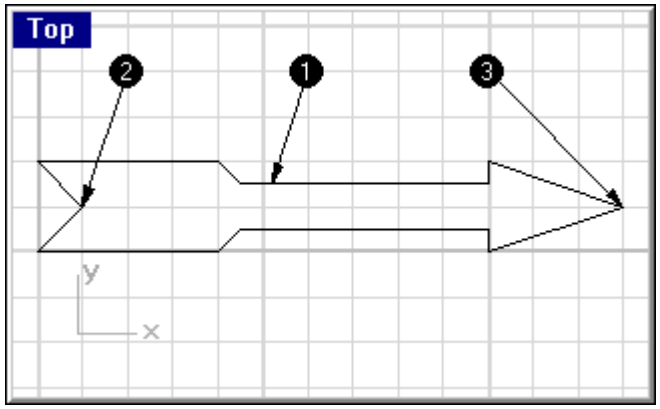
3 **Save** your model.

To make it 3D:

1 Select the Polyline.

2 From the **Surface** menu and click **Revolve**.

3 Toggle **Snap** on.



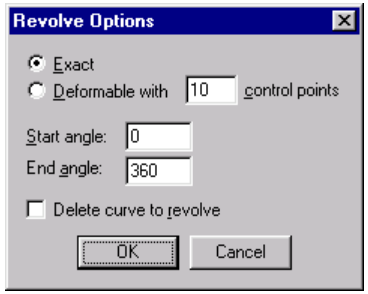
Notes:

Revolve
Look for this button.

4 At the **Start of revolve axis** prompt, choose the end of the arrow along the center line.

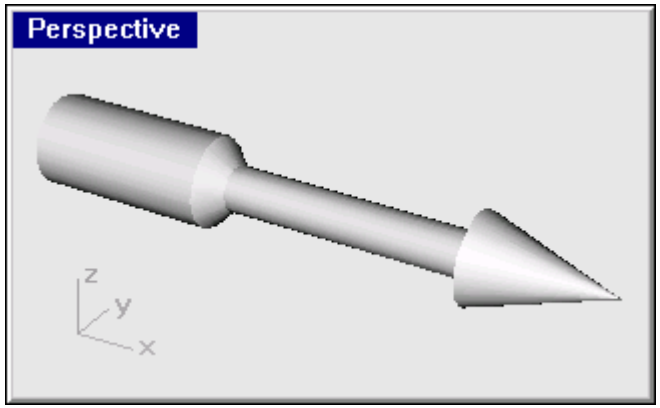
5 At the **End of revolve axis** prompt, choose the other end of the arrow along the center line.

6 The **Revolve Options** dialog box will appear. Click **OK**. Your arrow is now a three-dimensional model.



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

8 From the **Render** menu, click **Shade**.

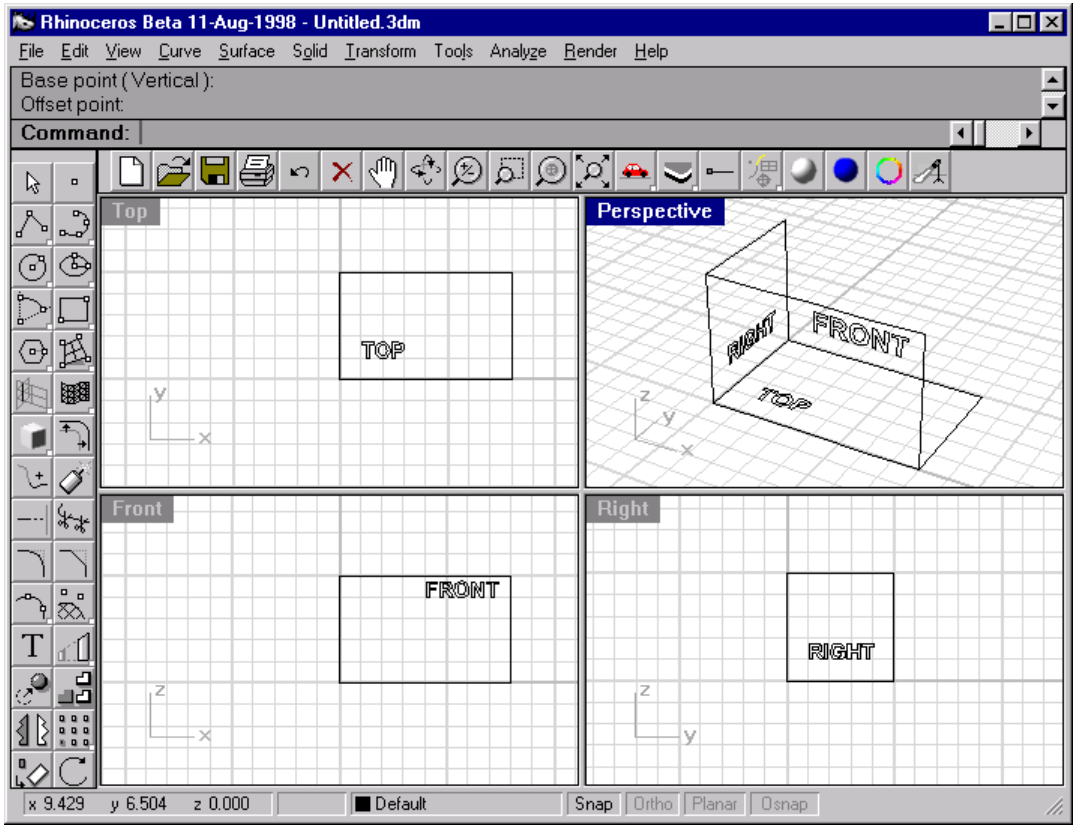


Notes:

Shade
Look for this button.

Viewports

Viewports are windows in the Rhino graphics area that show you a view of your model. You can move and resize viewports by dragging the title bar or borders. The cursor moves along a construction plane, which is defined for each viewport. You can create new viewports, rename viewports, and use predefined viewport configurations. To activate a viewport click anywhere in the viewport and the title highlights. If you are in a command sequence you simply have to move your cursor into a viewport to activate it.



Notes:

Construction planes

The construction plane is the guide used for modeling Rhino objects. Points you pick are always on the construction plane unless you use coordinate input, elevator mode, or object snaps.

Each construction plane has its own axes, a grid, and an orientation relative to the world coordinate system.

Default construction planes are provided with the default viewports.

- The Top construction plane x- and y-axes align with the world x- and y-axes.
- The Right construction plane x- and y-axes align with the world y- and z-axes.
- The Front construction plane x- and y-axes align with the world x- and z-axes.
- The Perspective view uses the Top construction plane.

Each construction plane has a grid. The grid is a plane of perpendicular lines lying on the construction plane. On default grids, every fifth line is slightly thicker. The red line represents the construction plane x-axis. The green line represents the construction plane y-axis. The red and green lines meet at the construction plane origin.

The red and green axes on the grid are the x- and y-axes of that viewport's construction plane. The construction planes happen to be lined up with world axes when you first start Rhino, so it is confusing in the front and side viewports where the y-axis of the construction plane happens to be pointing up in the world z- direction. The icon in the left hand corner always shows world coordinates, which are different from the construction plane axes.

Exercise 16—Modeling in different viewports

1. Start a new model. Save as **Viewports**.

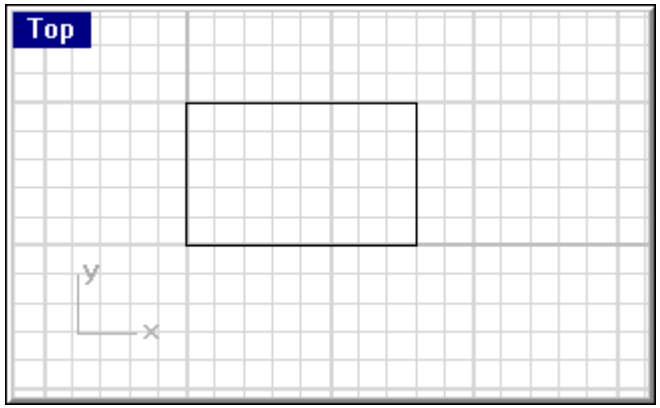
   In this exercise you will use snap and ortho to draw in different viewports. When modeling in Rhino, the active viewport and the construction plane are the same.
2. Toggle **Ortho** and **Snap** on.
3. From the **Curve** menu, click **Line**, then click **Polyline.**
4. Move your cursor into the **Top** viewport.
5. At the **Start of polyline** prompt, type **0,0**, and press ENTER.
6. At the **Next point of polyline** prompt, pick **8** units to the right.
7. At the **Next point of polyline** prompt, pick **5** units up.

Notes:

Polyline
Look for this button.
Left click for Polyline.

8 At the **Next point of polyline** prompt, pick **8** units to the left.

9 At the **Next point of polyline** prompt, type **C**, and press ENTER to close the line.



To draw in the front viewport:

1 From the **Curve** menu, click **Line**, then click **Polyline.**

2 Move your cursor into the **Front** viewport.

3 At the **Start of polyline** prompt, type **0,0** and press ENTER.

4 At the **Next point of polyline** prompt, pick **8** units to the right.

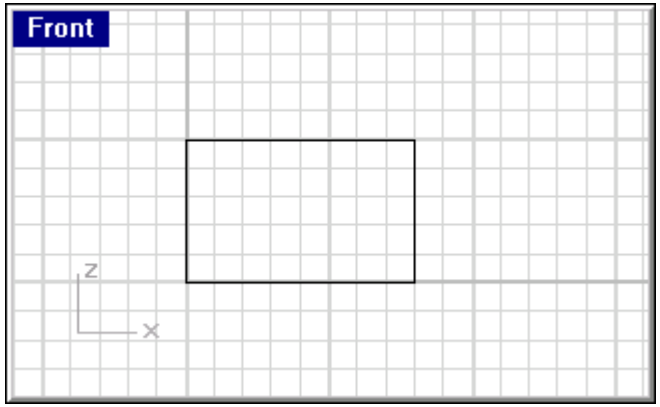
5 At the **Next point of polyline** prompt, pick **5** units up.

6 At the **Next point of polyline** prompt, pick **8** units to the left.

Notes:

Polyline
Look for this button.
Left click for Polyline.

7 At the **Next point of polyline** prompt, type **C**, and press ENTER to close the line.



To draw in the right viewport

1 From the **Curve** menu, click **Line**, then click **Polyline.**

2 Move your cursor into the **Right** Viewport.

3 At the **Start of polyline** prompt, type **0,0**, and press ENTER.

4 At the **Next point of polyline** prompt, pick **5** units to the right.

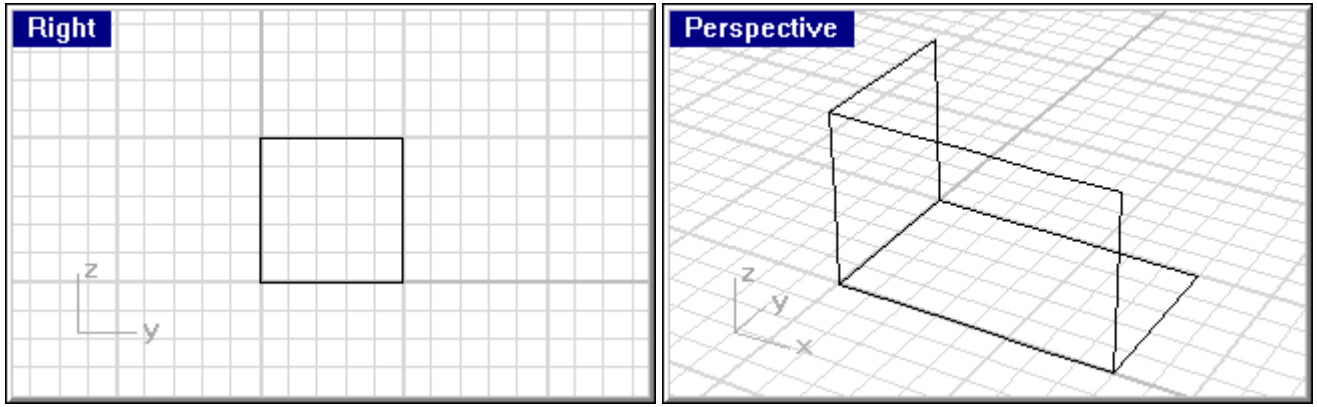
5 At the **Next point of polyline** prompt, pick **5** units up.

6 At the **Next point of polyline** prompt, pick **5** units to the left.

Notes:

Polyline
Look for this button.
Left click for Polyline.

7 At the **Next point of polyline** prompt, type **C**, and press ENTER to close the line.



Look in the perspective view. Each rectangle was drawn on a different construction plane. As you input coordinate data the points will be placed on the construction plane of the active viewport.

Modeling in 3d Space

Rhino makes it easy to draw in 3d space. As demonstrated in the previous exercise you can draw on a different construction plane by simply moving your cursor into a different viewport. Another useful tool for modeling in 3d space is elevator mode.

Elevator Mode

Elevator mode lets you pick points that are off the construction plane. Elevator mode requires two point picks to completely define the point. The first specifies the base point. The second specifies how far the final point is above or below the base point.

After the base point is specified, the marker is constrained to a tracking line perpendicular to the CPlane that passes through the base point.

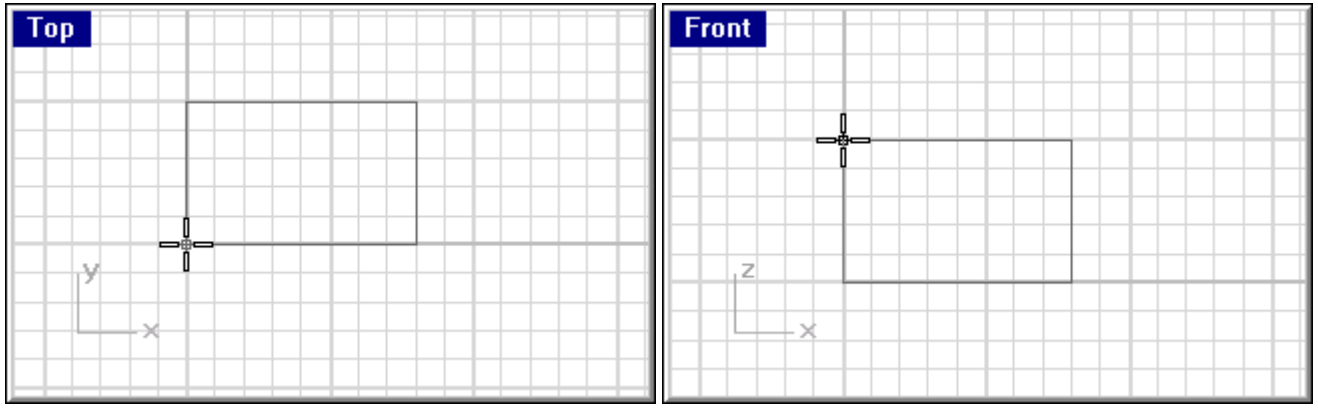
Pick a second point to specify the z-coordinate of the desired point. You can pick a point with the mouse, or type a single number to specify the height above the Cplane. Positive numbers are above the construction plane, negative numbers are below.

To activate elevator mode, hold the CTRL key while clicking with the left mouse button.

Notes:

To draw rectangle using elevator mode:

1. From the **Curve** menu, click **Line**, then click **Polyline.**
2. Move your cursor into the **Top** Viewport.
3. At the **Start of polyline** prompt, pick the **0,0** point while holding the CTRL key down.
   A tracking line constrains the marker through the base point perpendicular to the CPlane of the current view.
4. Move the cursor to the **Front** viewport to see the marker track along the tracking line.
   Pick a point at the top left of the rectangle. This is a secondary pick that elevates the first point off the construction plane.



5. At the **Next point of polyline** prompt, drag your cursor back to the top viewport and pick **8** units to the right.
6. At the **Next point of polyline** prompt, pick **5** units up.

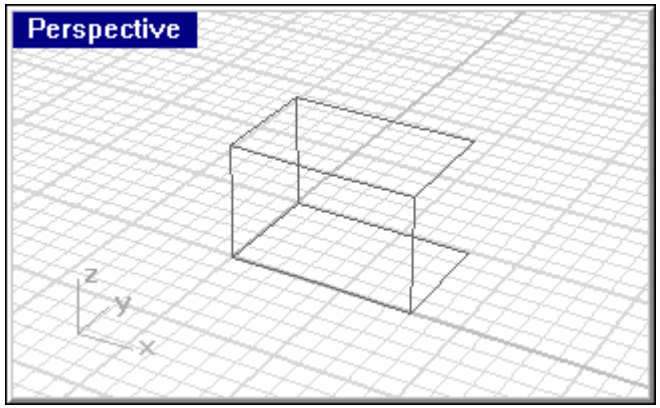
Notes:

Polyline
Look for this button.
Left click for Polyline.

7 At the **Next point of polyline** prompt, pick **8** units to the left.

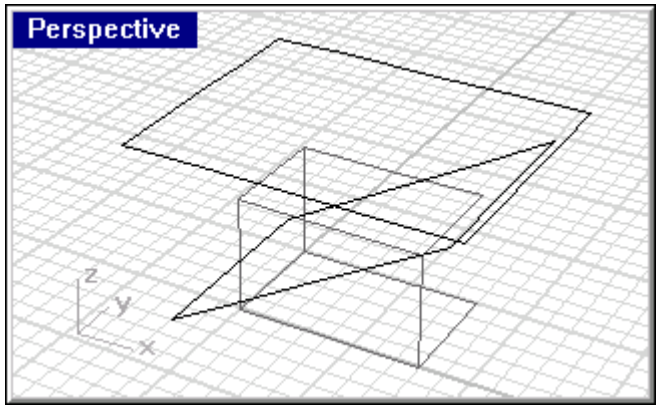
8 At the **Next point of polyline** prompt, type **C**, and press ENTER to close the line.

Notice that the polyline stays at the same elevation from the construction plane as you pick your next point. This is because **Ortho** is turned on.



9 Experiment using elevator mode with **Ortho** off.

10 Experiment using elevator mode with **Planar** mode on.

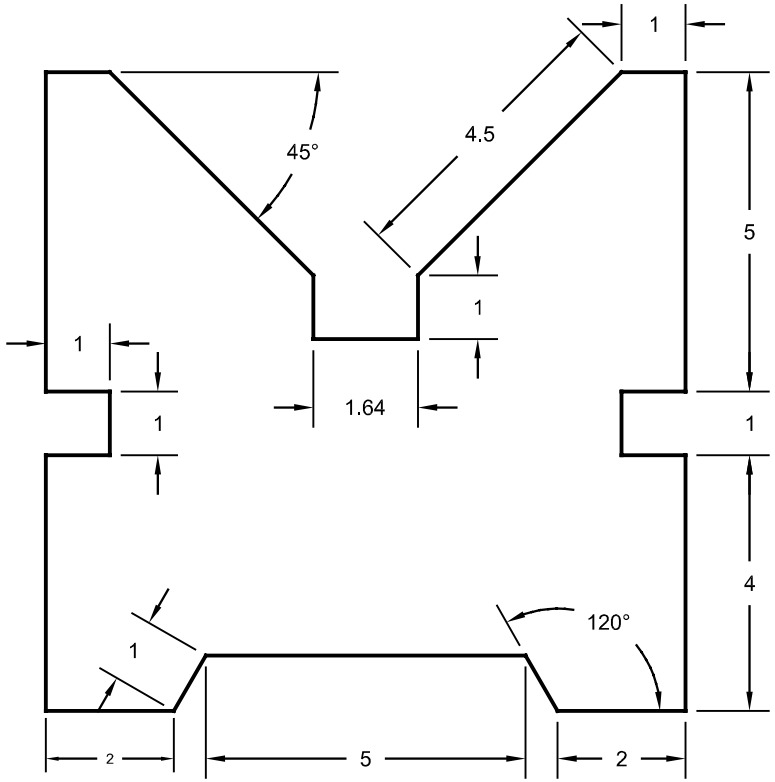


Notes:

Notes:

Exercise 17—Practice using distance and angle constraints

1. Start a new model. Save as **V-Block**.
2. Double Click on the title bar in the **Front** viewport. Create the following model in the front construction plane.
3. Draw the object below using a combination of absolute coordinates (x,y), relative coordinates (**r**x,y), and polar coordinates (**r**distance<angle). Start your model at **0,0** in the Front Viewport.
   Try to create the model using a single polyline.



4. **Save** your model.
5. Double Click on the title bar of the **Front** viewport to restore your views.

6 Select the object.

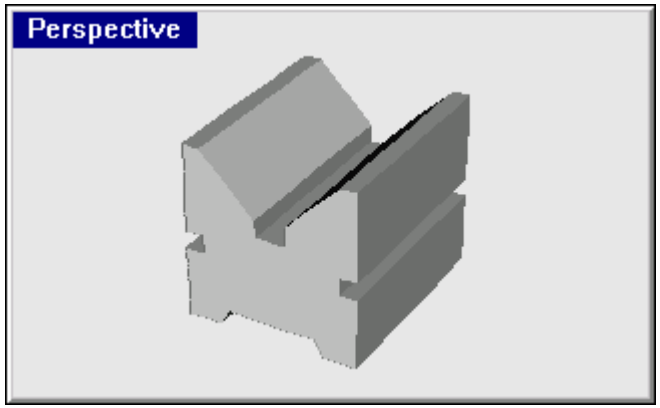
7 From the **Solid** menu, click **Extrude Planar Curve**.

8 At the **Extrusion distance <1> (Direction Cap=yes Bothsides Tapered)** prompt, type **12** and press ENTER.

You can view the model as a three-dimensional object in the perspective viewport.

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

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



11 **Save** your model.

Notes:

Shade
Look for this button.

Object Snaps

Object Snaps are tools for selecting specific points on existing objects. Use them for precision modeling and to get accurate data. Object Snaps are often referred to as ***osnaps***.

To open the **Osnap** toolbar, click **Osnap** in the status bar. This toolbar controls persistent object snaps. Use persistent objects snaps to maintain an object snap through choosing several points without having to reactivate the object snap.

When an object snap is active, moving the cursor near a specified point on an object causes the marker to jump to that point.

Check a box to turn on the object snap. You can place the toolbar anywhere on your desktop.

Command	Button	Description
End		**End** snaps to the end of a curve.
Near		Near snaps to the nearest point on an existing curve.
Point		Point snaps to a control point.
Mid		Midpoint snaps to the midpoint of a curve.
Cen		Center snaps to the center point of a curve. This works best with circles and arcs. With other curves, you will snap to the center points of circles based on the curvature at any given point.
Int		Intersection snaps to the intersection of two curves.
Perp		Perpendicular To snaps to the point on a curve that is perpendicular to the last selected point. Works only when you're selecting a series of points.
Tan		Tangent To snaps to the point on a curve that is tangent to the last selected point. This only works when you're selecting a series of points.
Quad		Quad snaps to the quadrant point of a circle, arc, or ellipse.

Notes:

Command	Button	Description
Lock		Temporarily turns off running object snaps.
Project		Projects the snap point to the construction plane.

In Rhino, reliable modeling and easy editing depends on objects actually meeting at specified points. Objects snaps gives you precision you cannot get using the “eyeball” method.

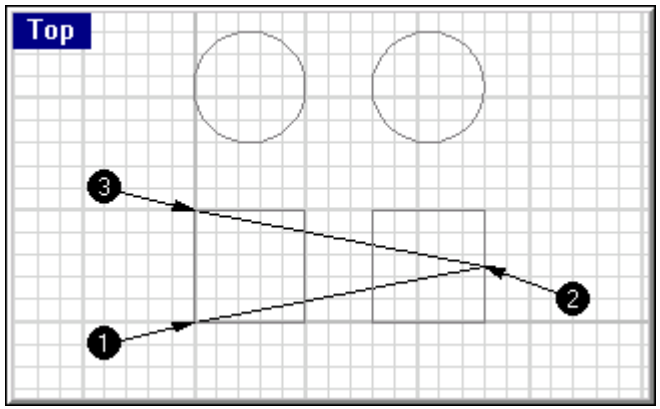
Exercise 18—Using Object Snaps

1. Open the **Osnap** model.
2. Toggle **Snap** and **Ortho** off.

Using End and Midpoint:

1. Click **Osnap** on the status bar. You can leave the **Osnap** toolbar displayed.
2. Check **End** and **Mid**. You can check and uncheck individual osnaps to make modeling with precision easy.
3. From the **Curve** menu, click **Line**, then click **Polyline.**
4. At the **Start of polyline** prompt, move your cursor close to the end of the line at the lower left corner of the first box and pick when the marker snaps to the end of the line. Pick to draw the polyline.

   The line starts exactly at that corner.



5. At the **Next point of polyline** prompt, snap to the Midpoint of the right vertical edge of the second box.

Notes:

Polyline
Look for this button.
Left click for Polyline.

Osnap
End and Mid on

The marker snaps to the midpoint of the line that the cursor touches, making the new line cross to the exact midpoint of that side. Pick to draw the polyline.

6 At the **Next point of polyline** prompt, snap to the End at the upper left corner of the first box.

The marker snaps to the End of the line. Pick to draw the polyline.

7 Press ENTER to end the command.

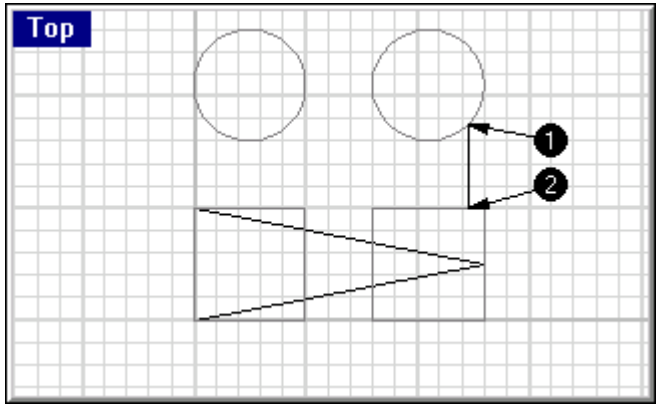
Using Near to and Perpendicular to:

1 In the **Osnap** toolbar check **Near** and **Perp**; uncheck **End** and **Mid**.

2 From the **Curve** menu, click **Line**, then click **Polyline.**

3 At the **Start of polyline** prompt, pick on the lower edge the circle at the top right.

The marker snaps to the point on the circle nearest to where the cursor was positioned.



4 At the **Next point of polyline** prompt, pick the top horizontal edge of the second box.

The marker snaps to a point perpendicular to the previous point. Pick to draw the polyline.

5 Press ENTER to end the command.

Using Intersection and Tangent to:

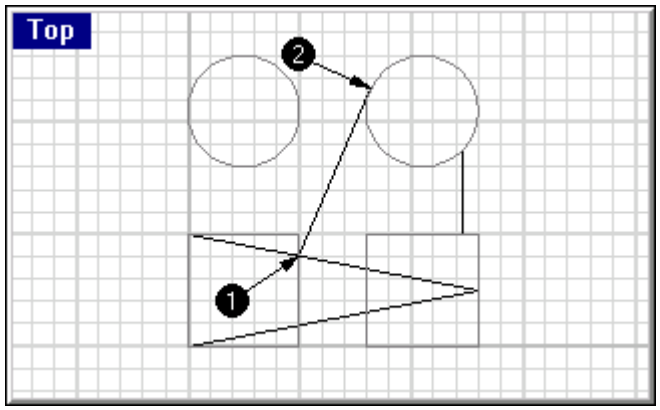
1 In the **Osnap** toolbar check **Int** and **Tan**; uncheck **Near** and **Perp**.

2 From the **Curve** menu, click **Line**, then click **Polyline.**

Notes:

3 At the **Start of polyline** prompt, pick on the intersection where the diagonal line crosses the vertical line on the first box.

The marker snaps to the intersection between the two lines.



4 At the **Next point of polyline** prompt, pick the top, left edge of the circle on the right.

The marker snaps to a point tangent to the circle.

5 Press ENTER to end the command.

Using Center of:

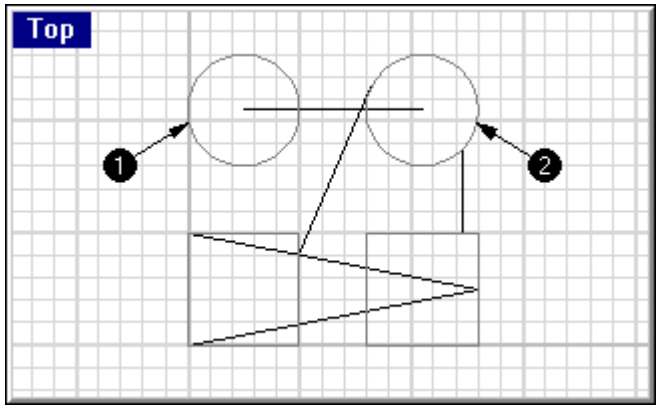
1 In the **Osnap** toolbar check **Cen**; uncheck **Int** and **Tan**.

2 From the **Curve** menu, click **Line**, then click **Polyline.**

Notes:

3 At the **Start of polyline** prompt, pick on the edge of a circle.

The marker snaps to the center of the circle.



4 At the **Next point of polyline** prompt, pick on the edge of the other circle.

The marker snaps to the center of the circle.

5 Press ENTER to end the command.

Using Quadrant of:

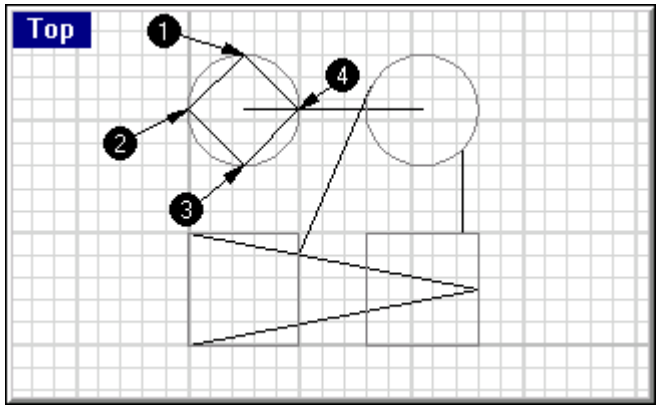
1 In the **Osnap** toolbar Check **Quad**; uncheck **Cen**.

2 From the **Curve** menu, click **Line**, then click **Polyline.**

Notes:

3 At the **Start of polyline** prompt, pick a point on the top edge of the first circle.

The marker snaps to the quadrant point of the circle.



4 At the **Next point of polyline** prompt, pick the left edge of the circle.

The marker snaps to the quadrant point of the circle.

5 At the **Next point of polyline** prompt, pick the bottom edge of the circle.

6 At the **Next point of polyline** prompt, pick the right edge of the circle.

7 At the **Next point of polyline** prompt, type **C** and press ENTER.

Analysis Commands

Rhino provides analysis tools for finding lengths, angles, areas, distances, volumes and centroids of solids. Additional commands let you analyze curve curvature, continuity between curves, and find unjoined edges.

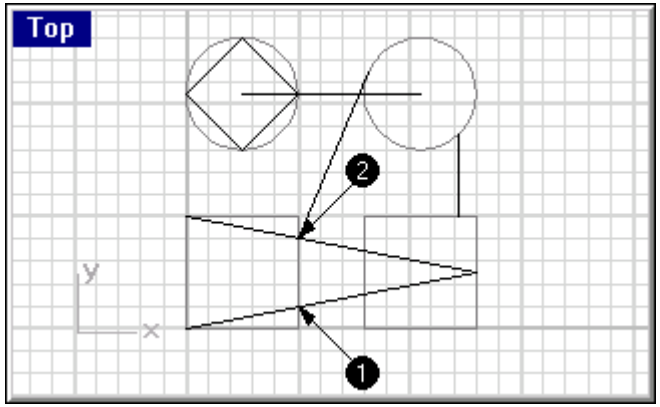
Command	Description
Distance	Displays the distance between two points.
Length	Displays the length of an object.
Angle	Displays the angle between two lines.
Radius	Measures the radius of curvature of a curve, circle, or arc at the point where you pick the curve and displays it on the command line.

Notes:

Command	Description
Point	The Cartesian coordinates of the point, in both world and CPlane coordinates are displayed on the command line in x,y,z format.

To find the distance between two points:

1. From the **Analyze** menu, click **Distance**.
2. At the **First point for distance** prompt, pick the intersection where a diagonal line intersects a vertical line.
3. At the **Second point for distance** prompt, pick the intersection where the other diagonal line intersects the same vertical line.



4. Press F2 to display the information.

   Construction coordinates:
   angles: xy = 90 elevation = 0
   deltas: dx = 0 dy = 3.07692 dz = 0

   World coordinates:
   angles: xy = 90 elevation = 0
   deltas: dx = 0 dy = 3.07692 dz = 0

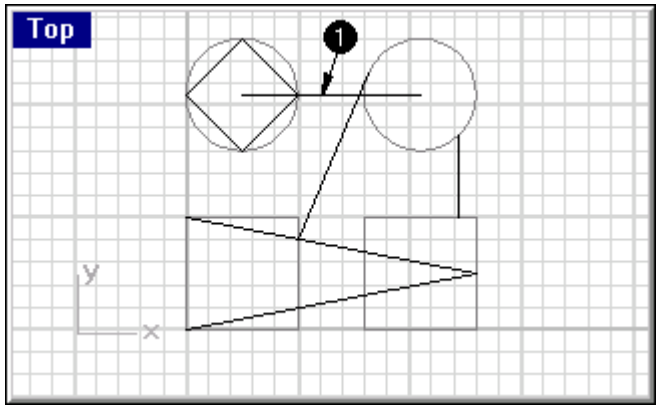
   Distance = 3.07692 units

Notes:

Distance
Look for this button.

To find the length of a line:

1. From the **Analyze** menu, click **Length**.
2. At the **Select Curve to measure** prompt, select the line between the centers of the circles.



The length of the object displays on the command line.

Length: 8.0000 (+/- 0.0008)

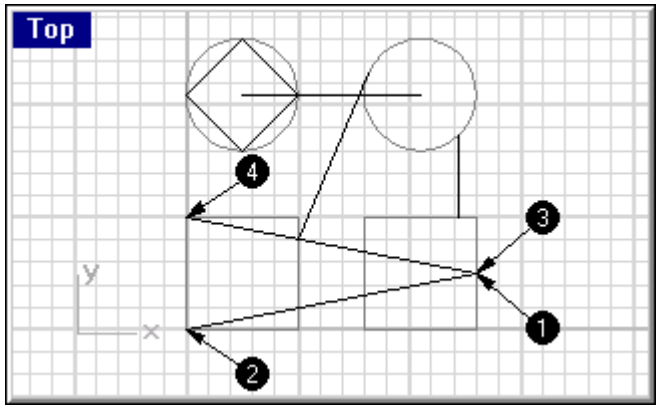
To measure the Angle between two lines:

1. From the **Analyze** menu, click **Angle**.
2. At the **Start of line 1** prompt, select a point that defines the start of an angle line.
3. At the **End of line 1** prompt, select a point that defines the end of an angle line.
4. At the **Start of line 2** prompt, select a point that defines the beginning of the second angle line.
5. At the **End of line 2** prompt, select a point that defines the end of the second angle line.

Notes:

Length
Look for this button.

Angle
Look for this button.



The angle will display on the command line in the following format:

Angle is 21.7711 degrees

To measure the radius of a circle:

1. From the **Analyze** menu, click **Radius**.
2. At the **Select curve for radius measurement** prompt, select one of the circles. This will also measure a radius on a curve.

   The radius will display on the command line in the following format:

   Radius at pick point is 2.5

To measure a point:

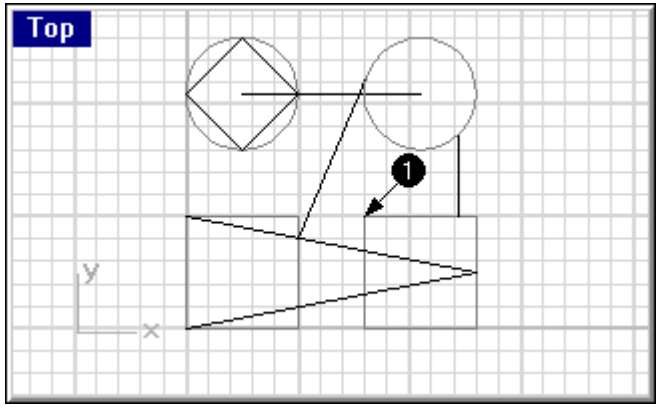
1. From the **Analyze** menu, click **Point**.

Notes:

Radius
Look for this button.

Point
Look for this button.

2 At the **Point to measure** prompt, snap to the end point of one of the squares.



The x,y,z point is displayed for the world coordinate plane and the current construction plane.

Point in world coordinates = 8,5,0 cplane coordinates = 8,5,0

Drawing Circles

You can create circles using a center point and radius, center point and a diameter, two points on the diameter, three points on the circumference, tangent points to two coplanar curves and a radius.

Button	Command	Description
	Circle	Creates a circle from a center point and a radius.
	Circle3Pt	Creates a circle from three points on the circumference.
	CircleD	Creates a circle from two points on the circumference that are the ends of the diameter.
	CircleTTR	Creates a circle from a radius and two objects that the circle is tangent to.
	CircleTTT	Creates a circle that is tangent to three other objects.

Notes:

Button	Command	Description
	Circle V	Creates a circle that is perpendicular to the construction plane

Option	Description
Diameter	Type **D**, and press ENTER to change to diameter input. You will have to type R to change back to radius input.
Vertical	Type **V**, and press ENTER to create a circle that is perpendicular to the construction plane.
AroundCurve	Type **A**, and press ENTER to draw a circle whose axis is perpendicular to a curve.

Exercise 19—Drawing Circles

To draw a center, radius circle:

1. From the **Curve** menu, click **Circle**, then click **Center, Radius.**
2. At the **Center of circle (Vertical AroundCurve)** prompt, type **20,10** and press ENTER.
3. At the **Radius <1> (Diameter)** prompt, type **3** and press ENTER. A circle is created.

Circle
Look for this button.

To draw a center, diameter circle:

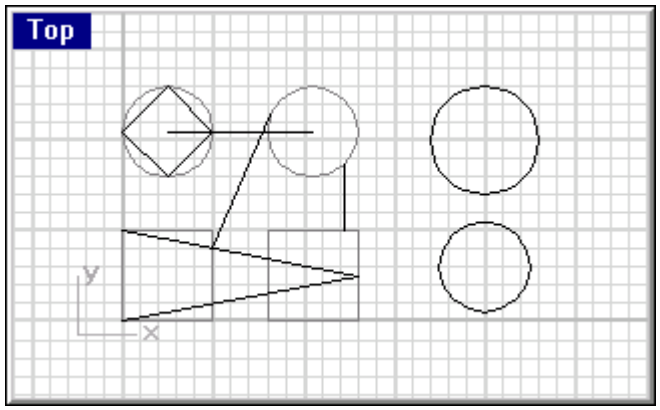
1. From the **Curve** menu, click **Circle**, then click **Center, Radius.**
2. At the **Center of circle (Vertical AroundCurve)** prompt, type **20,3** and press ENTER.
3. At the **Radius <3> (Diameter)** prompt, type **D** and press ENTER.

Circle
Look for this button.

Notes:

4 At the **Diameter <6.00> (Radius)** prompt, type **5** and press ENTER.

A circle is created that is based on a center point and a diameter. The diameter option will be the default until it is changed.



To draw a 3-point circle:

1 From the **Curve** menu, click **Circle**, then click **3 Points.**

2 At the **First point on circle** prompt, snap to an End on one of the squares.

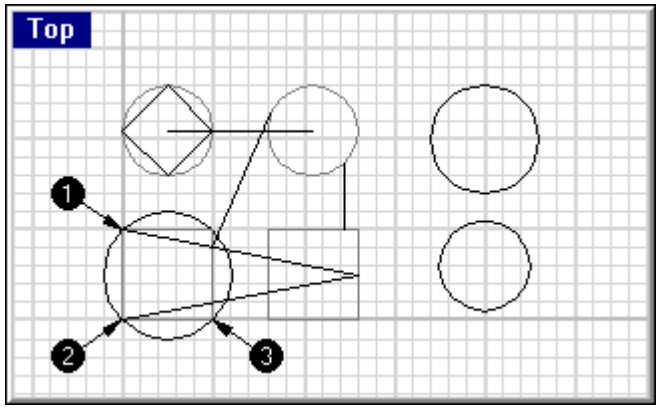
3 At the **Second point on circle** prompt, snap to another End on the same square.

Notes:

Circle 3Pt
Look for this button.

4 At the **Third point on circle** prompt, snap to a third End on the same square.

A circle is created with a circumference that intersects the three ends you picked.



To draw circle diameter:

1 From the **Curve** menu, click **Circle**, then click **Diameter.**

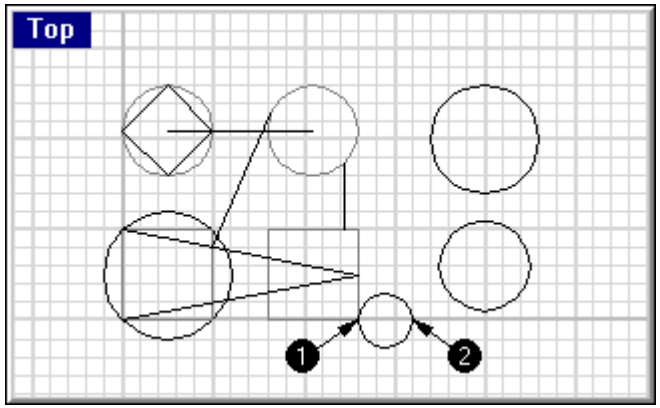
2 At the **Start of diameter (Vertical)** prompt, pick the first point on the circle diameter by snapping to the End of a square.

Notes:

Circle Diameter
Look for this button.

3 At the **End of diameter** prompt, type **3** and press ENTER, then turn Ortho on and pick to the right.

A circle is created with the two points you picked as the diameter, and the diameter was constrained to a value of 3.



To draw a tangent, tangent, radius circle:

1 From the **Curve** menu, click **Circle**, then click **Tangent, Tangent, Radius.**

2 At the **Select first curve for CircleTTR** prompt, select the circle you just created near the top edge.

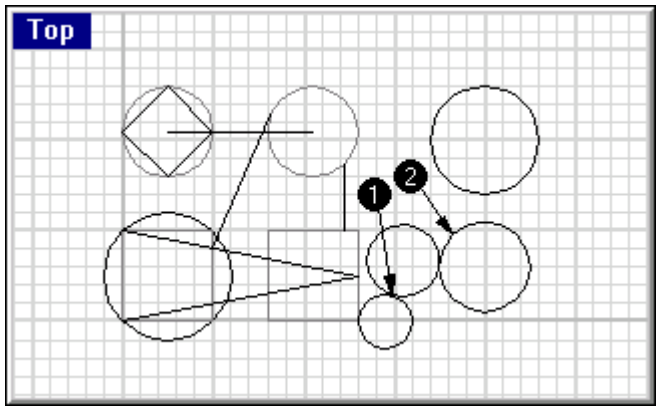
3 At the **Select second curve for CircleTTR** prompt, select the circle to the right of the circle you picked near the top edge.

Notes:

Circle Tangent, Tangent, Radius
Look for this button.

4 At the **Radius <1>** prompt, type **2** and press ENTER. Or, press ENTER to accept the default value.

A circle is created that is tangent to the two circles you selected with a radius of 2. Sometimes this command creates multiple circles. Delete the extras.



To draw a tangent, tangent, tangent circle:

1 From the **Curve** menu, click **Circle**, then click **Tangent to 3 Curves.**

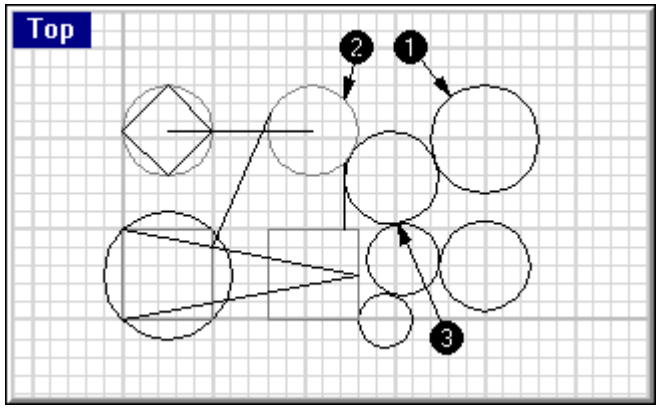
2 At the **Select First curve:** prompt, select the first object that you want the circle to be tangent to.

3 At the **Select Second curve:** prompt, select the second object that you want the circle to be tangent to.

Notes:

Circle Tangent to 3 Curves
Look for this button.

4 At the **Select Third curve** prompt, select the third object that you want the circle to be tangent to. Sometimes this command creates multiple circles. Delete the extras.



To draw a circle vertical to the construction plane:

1 From the **View** menu, click **Viewport Layout**, then click **4 Viewports.**

2 From the **Curve** menu, click **Circle**, then click **Diameter.**

3 At the **Start of diameter (Vertical)** prompt, type V and press enter.

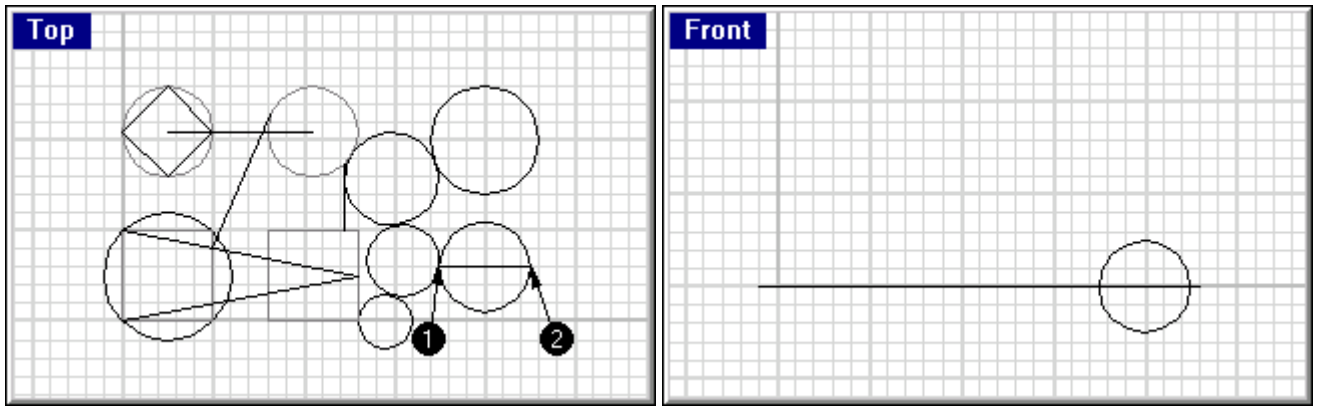
4 At the **Start of diameter** prompt, snap to the Quadrant of one of your circles in the top viewport.

Notes:

Circle Diameter
Look for this button.

5 At the **End of diameter** prompt, snap to the other Quadrant of the same circle.

The circle is drawn perpendicular to the construction plane. You will be able to see it in the one of the other viewports.



To draw a circle around a curve:

1 From the **Curve** menu, click **Circle**, then click **Center, Radius.**

2 At the **Center of circle (Vertical AroundCurve):** type **A** and press ENTER.

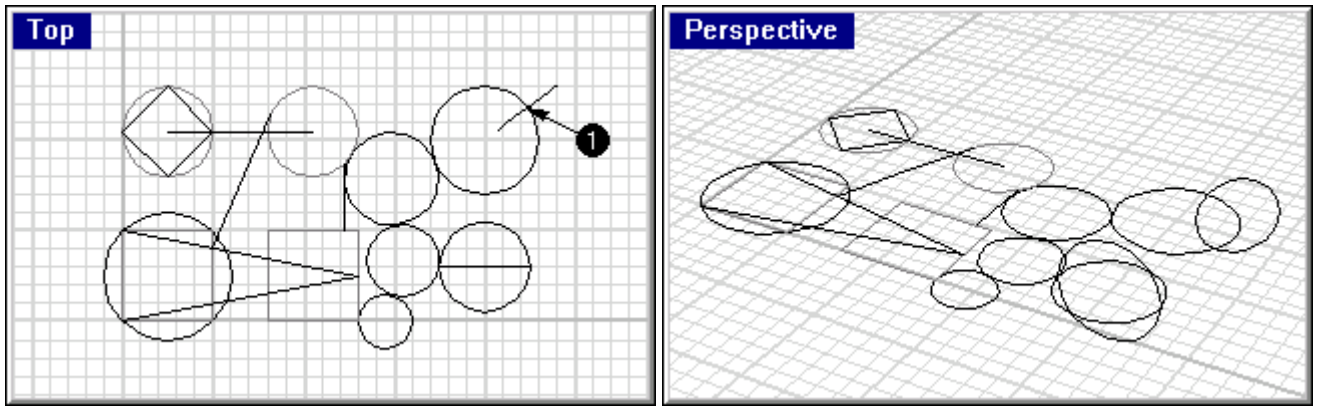
3 At the **Center of circle:** prompt, choose a point on a circle.

Notes:

Circle
Look for this button.

4 At the **Radius:** prompt, type **3** and press ENTER.

The circle is drawn perpendicular to the curve at the point you picked. You will be able to see it in the perspective viewport.

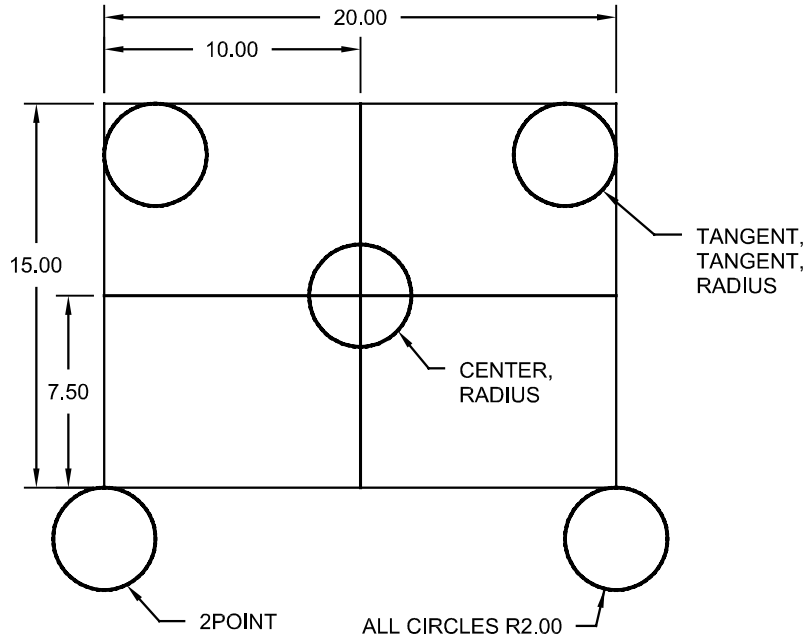


Notes:

Notes:

Exercise 20—Practice

1. Start a new model. Save as **Circles**.
2. From the **Edit** menu, click **Layers**, then click **Edit.**
3. Click **New** three times.
4. Rename the new layers **Box, Line, Circle**.
5. Change the **color** of the **Box** layer to **green**, **Line** to **cyan**, and **Circle** to **red**.
6. Draw the **lines** and **circles** on the appropriate layers. Use **Line Segments** for the Box and **Single Line** for the center lines. You will use several of the **circle** options and **object snaps** to complete this model .



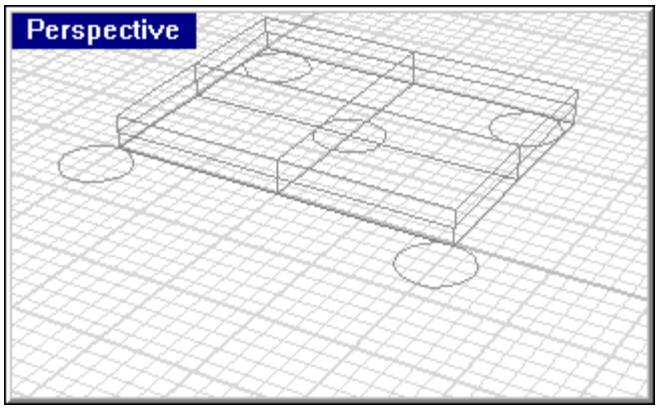
To make it 3D:

1. Select the lines that form the rectangle.

2 From the **Solid** menu, click **Extrude Planar Curve.**

3 At the **Extrusion distance <1> (Direction Cap=yes Bothsides Tapered)** prompt, type **2** and press ENTER.

The rectangle generates a box.

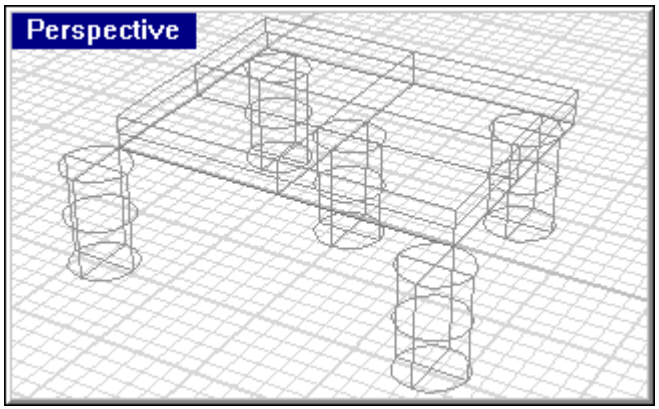


4 Select the circles.

5 From the **Solid** menu, click **Extrude Planar Curve.**

6 At the **Extrusion distance <2> (Direction Cap=yes Bothsides Tapered)** prompt, type **-6** and press ENTER.

The circles generate cylinders.

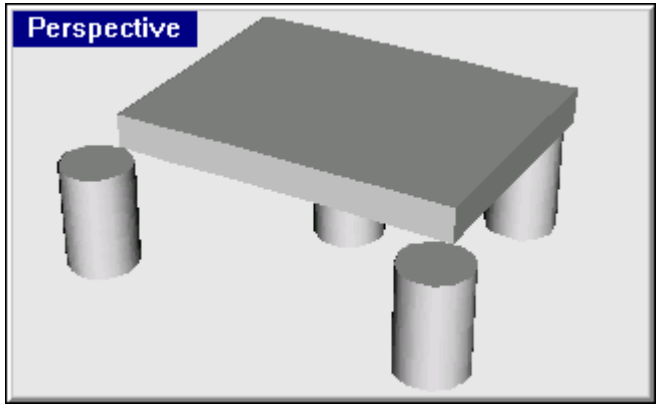


7 Click in the **perspective** viewport.

Notes:

Extrude Planar Curve
Look for this button.

8 From the **Render** menu, click **Shade.**

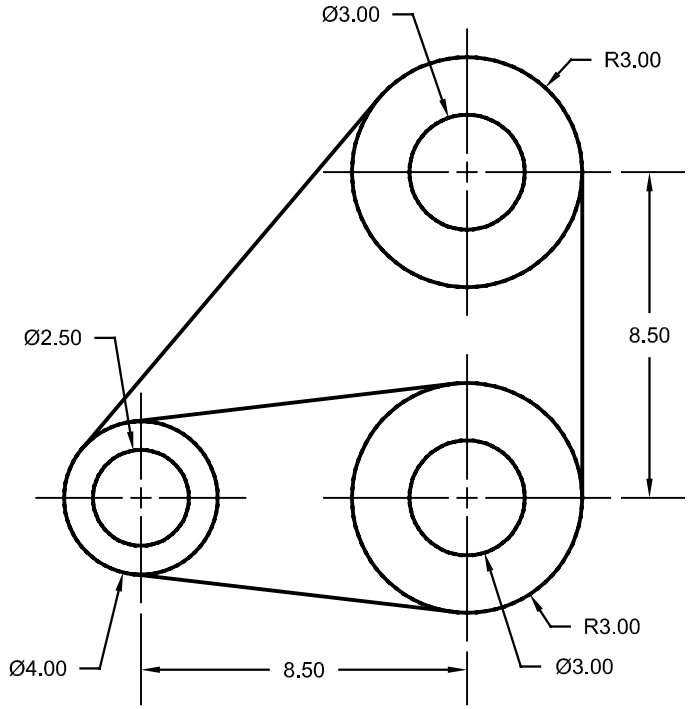


Notes:

Shade
Look for this button.

Exercise 21—Using Circle related Object Snaps

1. Start a new model. Save as **Link**.
2. Complete the model as shown.
3. Draw the three large circles first.
4. Draw the small holes next. Use object snaps to snap to the center of the large circles.



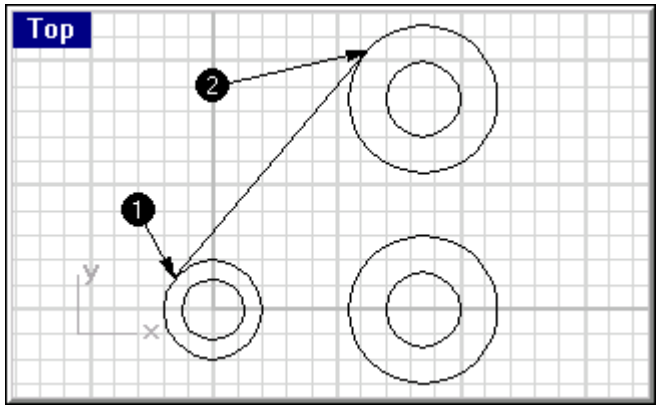
To draw the lines:

1. From the **Curve** menu, click **Line**, then click **Tangent to 2 Curves.**
2. At the **Select first curve near tangent point** prompt, pick on the edges of the circles where you want the tangent lines to attach.

Notes:

Line Tangent to 2 Curves
Look for this button.

3 At the **Select second curve near tangent point** prompt, pick on the edges of the circles where you want the tangent lines to attach.



4 Continue to use this command to complete the model.

5 **Save** your model.

Drawing Arcs

Arc draws a circular arc through three non-collinear points. You can create arcs from a center point, start point, and angle, two ends and a point along the arc, the ends and a direction, a start point, circumference point, and an end point, tangent points two coplanar curves and a radius.

You can continue an existing curve with an arc to an existing curve to a point or by an angle.

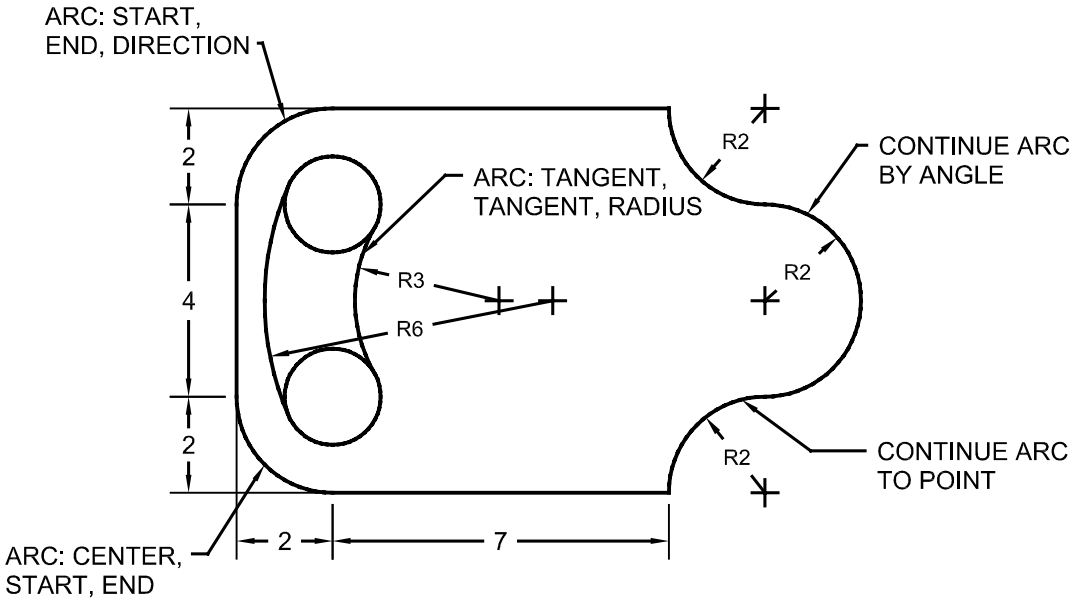
Button	Command	Description
	Arc	Creates an arc from a center point, a starting point and an end or an angle.
	Arc3Pt	Creates an arc from three points.
	ArcDir	Creates an arc from start point, end point, and direction.

Notes:

Button	Command	Description
	ArcTTR	Creates an arc from tangents and radius.
	ExtendByArc	Continues an arc by angle.
	ExtendByArcToPt	Continues an arc to a point.

Exercise 22—Practice With Arcs

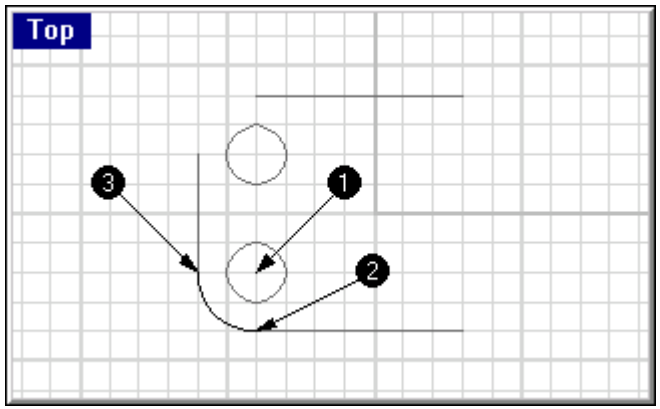
♦ Open the **Arc1** model.



Notes:

To draw a center, start, end or angle arc:

1. From the **Curve** menu, click **Arc**, then click **Center, Start, Angle.**
2. At the the **Center of arc** prompt, snap to the center of the circle at the lower left.
3. At the **Start or arc** prompt, snap to the end of the line.
4. At the **End point or angle** prompt, snap to the end of the other line, or drag your cursor in the direction you want the arc to go then type **90** and press ENTER.



To draw a start, end, direction arc:

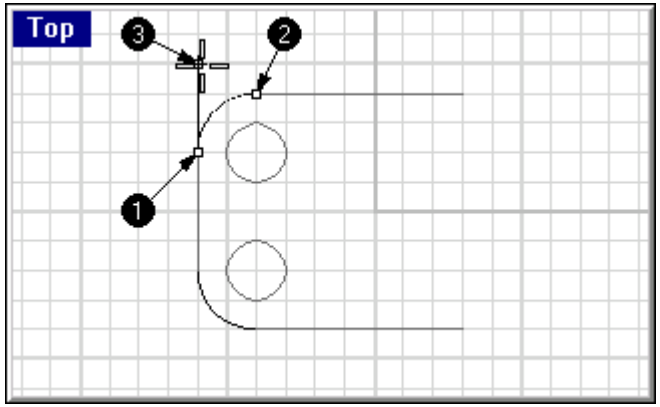
1. From the **Curve** menu, click **Arc**, then click **Start, End, Direction.**
2. At the **Start of arc** prompt, pick a start point.
3. At the **End of arc** prompt, pick an end point.

Notes:

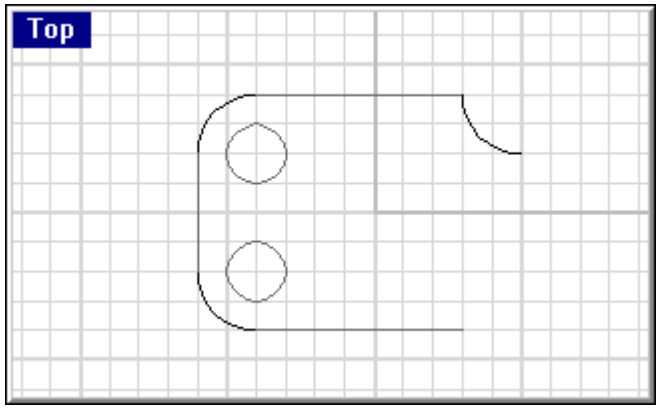
Arc
Look for this button.

Direction Arc
Look for this button.

4 At the **Direction** prompt, toggle **ortho** on and drag straight up for the tangent at the start point and pick.



5 Make another Direction Arc at the upper right.



To add more arc segments:

1 From the **Curve** menu, click **Extend**, then click **By Arc.**

2 At the **Select curve to extend** prompt, pick the end point of the arc you just created.

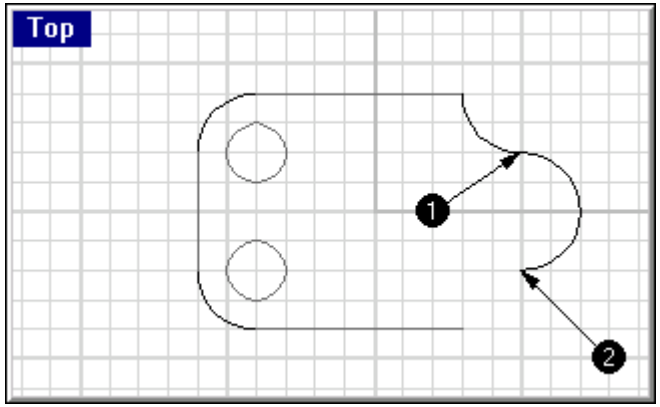
3 At the **Radius of arc (MatchRadius)** prompt, type **2** and press ENTER.

4 At the **End point or angle** prompt, toggle **ortho** on and pick a point straight down.

Notes:

Extend By Arc
Look for this button.

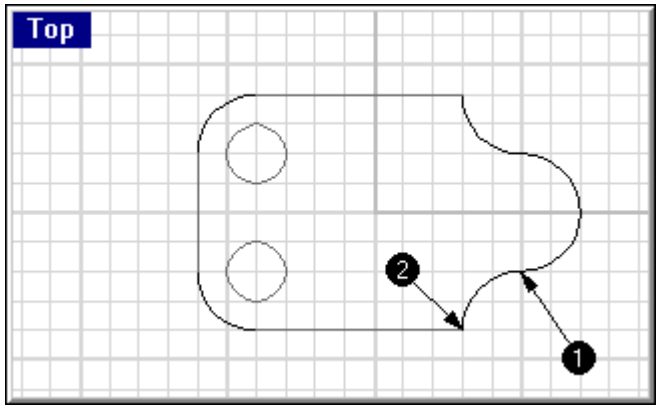
The arc will be tangent to the curve you chose.



5 From the **Curve** menu, click **Extend**, then click **By Arc to Point.**

6 At the **Select curve to extend** prompt, pick the end of the last arc.

7 At the **End of arc** prompt, pick the end of the line.



To draw a tangent, tangent, radius arc:

1 From the **Curve** menu, click **Arc**, then click **Tangent, Tangent, Radius.**

2 At the **Select first curve for ArcTTR** prompt, pick an arc, line, circle, or curve.

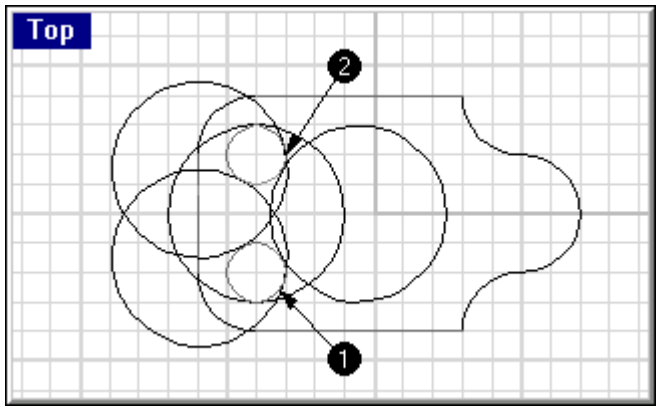
Notes:

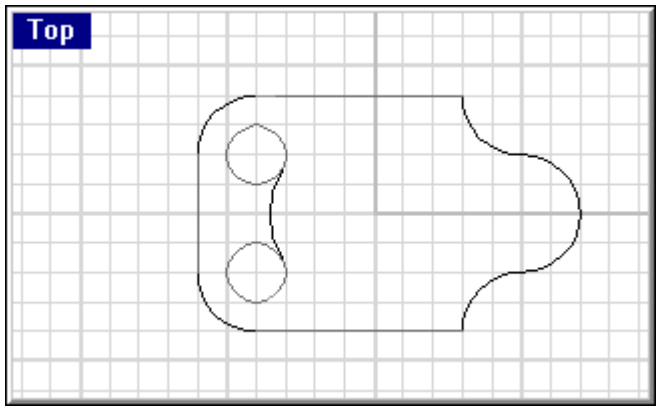
Extend By Arc To Pt
Look for this button.

Arc Tangent, Tangent, Radius
Look for this button.

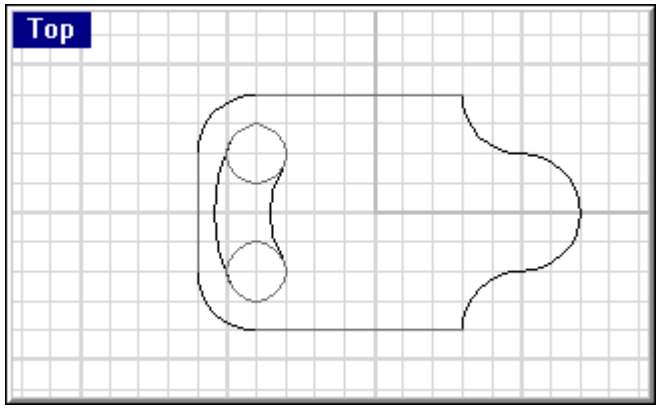
3 At the **Select second curve for ArcTTR** prompt, pick another arc, line, circle, or curve.

4 At the **Radius<1>** prompt, type **3** and press ENTER. All of the arcs that are tangent are created. Delete the arcs that you don't need.





5 Repeat this for the other tangent arc with a radius of 6.

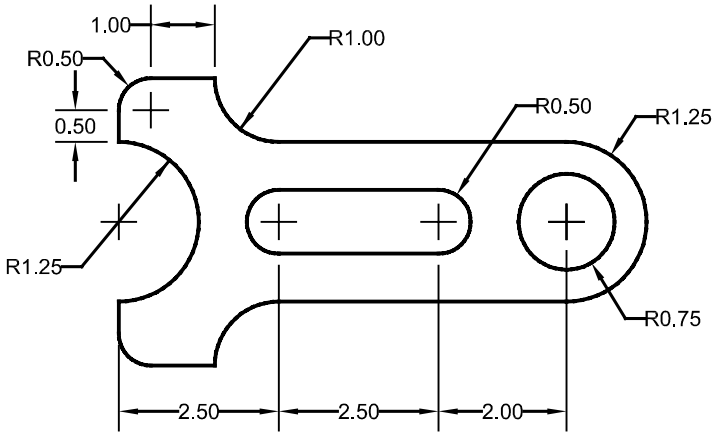


6 **Save** your model.

Notes:

Exercise 23—Practice With Arcs

1. Start a new model. Save as **Arc2**.
2. Create this model using object snaps and the Line, Circle, and Arc commands.



To make it 3D:

1. Select the all of the lines and arcs that form the object.
2. From the **Solid** menu, click **Extrude Planar Curve.**

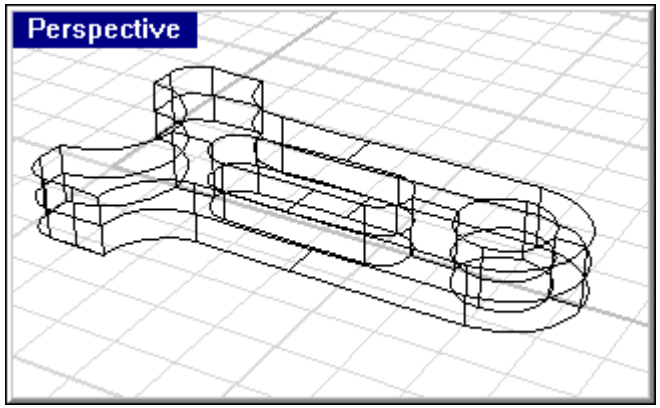
Notes:

Draw the center lines first: use their intersection point to draw your arcs and circles.

Extrude Planar Curve
Look for this button.

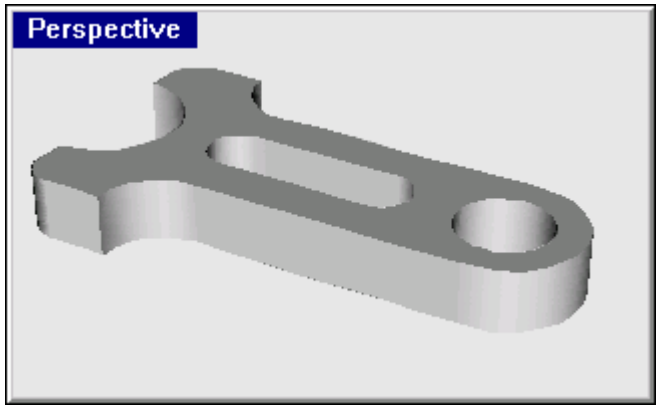
3 At the **Extrusion distance <1> (Direction Cap=yes Bothsides Tapered)** prompt, type **1** and press ENTER.

The object now has thickness.



4 Click in the **perspective** viewport.

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



Notes:

Drawing Ellipses and Polygons

You can draw ellipses from the center or by the ends. You can draw Polygons from a center point or an edge. You can draw a rectangle from diagonal corners or by choosing three points.

Button	Command	Description
	Ellipse	Creates an ellipse by specifying the center point, the end of one axis, and a height.
	EllipseD	Creates an ellipse by specifying the ends of one axis, and a height.
	Polygon	Creates a polygon from its center and radius.
	PolygonEdge	Creates a polygon by specifying the ends of one edge.
	Rectangle	Creates a rectangle using opposite corners.
	RectangleCen	Creates a rectangle from adjacent corners and a point on the opposite edge.

Ellipse Options

Option	Description
Vertical	Type V, and press ENTER, to create an ellipse that is perpendicular to the construction plane.
AroundCurve	Type A, and press ENTER to draw an ellipse whose axes is perpendicular to a curve.

Polygon Options

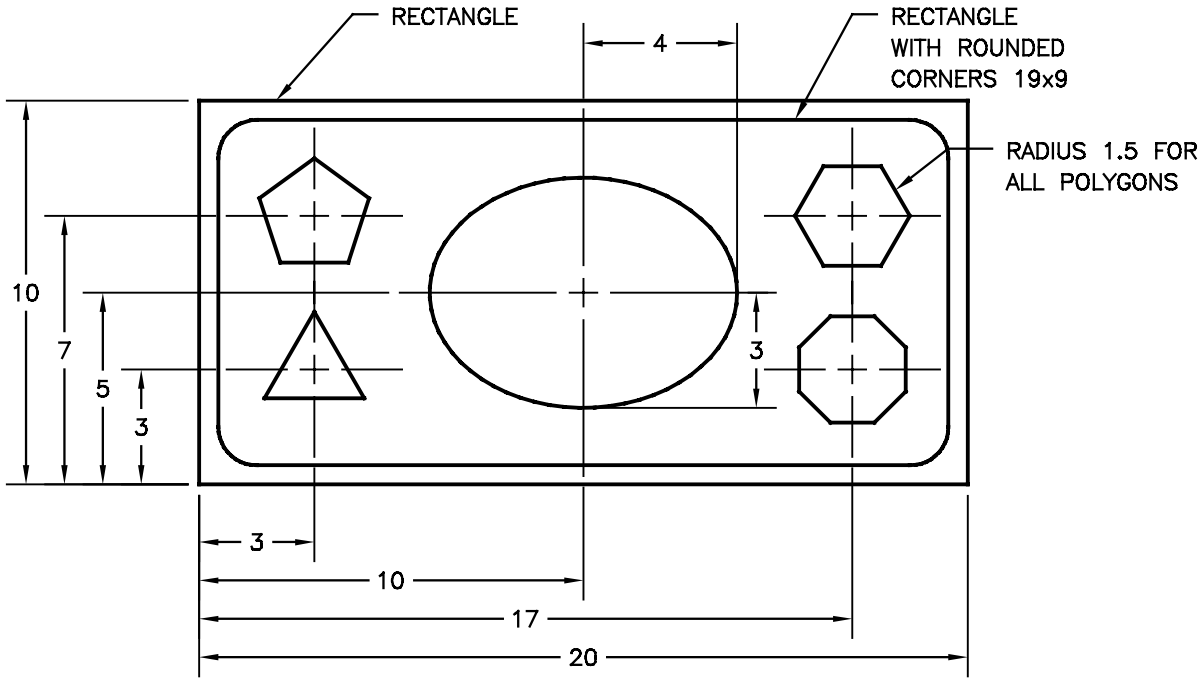
Option	Description
Circumscribed	Type C, and press ENTER, to create a polygon that is circumscribed about a radius. The default is to create a polygon that is inscribed in a specified radius.

Notes:

Notes:

Exercise 24—Practice With Ellipses and Polygons

Start a new model. Save as **Ellipse**.



To draw a rectangle from diagonal corners:

1. From the **Curve** menu, click **Rectangle**, then click **Corner to Corner.**
2. At the **First corner of rectangle (Rounded)** prompt, type **-10,-5**, and press ENTER.
3. At the **Other corner or length** prompt, type **20** and press ENTER.
4. At the **Width** prompt, type **10** and press ENTER.

Rectangle
Look for this button.

Draw a rectangle from a center point with a length and width, and rounded corners:

1. From the **Curve** menu, click **Rectangle**, then click **Center, Corner.**
2. At the **Center of rectangle (Rounded)** prompt, type **R**, and press ENTER to give the rectangle rounded corners**.**
3. At the **Center of rectangle** prompt, type **0,0** and press ENTER.
4. At the **Corner or length** prompt, type **19** and press ENTER.
5. At the **Width** prompt, type **9** and press ENTER.
5. At the **Choose a point in the corner to define roundness (**Corner=Arc**)** prompt type **1**,and press ENTER.

   If you are creating a rounded rectangle, choose a point in the corner to set its curvature. Or, type C, and press ENTER to toggle from circular rounded corners to conic rounded corners.

To draw an ellipse from its center and axis ends:

1. From the **Curve** menu, click **Ellipse**, then click **From Center.**
2. At the **Center of ellipse (Vertical AroundCurve)** prompt, type **0,0** and press ENTER.
3. At the **End of First axis** prompt, type **4**, and press ENTER
4. At the **End of First axis** prompt, toggle **ortho** on and pick to the right.
5. At the **End of Second axis** prompt, type **3**, and press ENTER.

To draw a polygon from its center and radius:

1. From the **Curve** menu, click **Polygon**, then click **Center, Radius.**
2. At the **Center of polygon (NumSides=4 Circumscribed)** prompt, type 3 and press enter to change the number of sides for the polygon.
3. At the **Center of polygon (NumSides=3 Circumscribed)** prompt, type -7,-2 and press enter to locate the center point for the polygon.
4. At the **Radius** prompt, type 1.5 and press enter.
5. At the **Radius** prompt, pick a point to orient the polygon.
6. Continue drawing the rest of the polygons. Use the same radius for each one.

Notes:

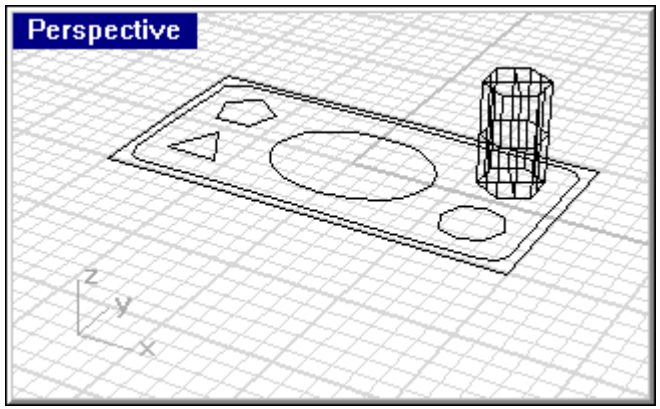
Centered Rectangle
Look for this button.

Ellipse
Look for this button.

Polygon
Look for this button.

To make it 3D:

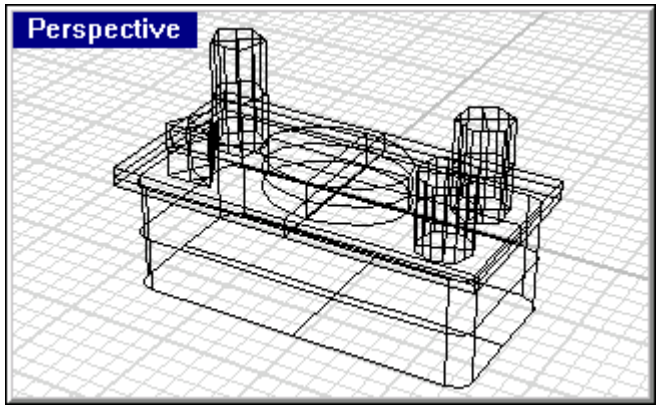
1. Select the pentagon.
2. From the **Solid** menu, click **Extrude Planar Curve.**
3. At the **Extrusion distance <1> (Direction Cap=yes Bothsides Tapered)** prompt, type **6** and press ENTER.



Notes:

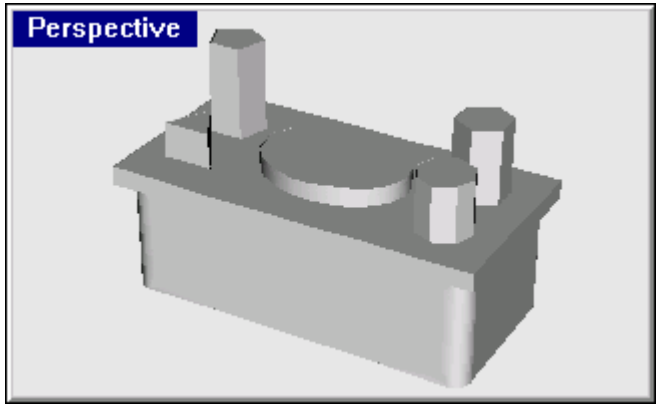
Extrude Planar Curve
Look for this button.

4 Continue to extrude all of the polygons, the rectangles, and the ellipse as follows:
hexagon, Extrusion distance <**5**>
octagon, Extrusion distance <**4**>
triangle, Extrusion distance <**3**>
rectangle, Extrusion distance <**1**>
ellipse, Extrusion distance <**2**>
rounded rectangle, Extrusion distance <**-6**>.



5 Click in the **perspective** viewport.

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



Notes:

Modeling Freeform Curves

The use of freeform curves allows more flexibility to create complex shapes. If you need to control precision, you can make construction lines that define your overall parameters.

Button	Command	Description
	Curve	Curve by control points creates a curve from specified control points. Control points do not lie on the curve, but they determine its curvature.
	Interpolated curve	Interpolated curve creates a curve going through specified interpolate points. These points lie on the curve and determine its curvature.
	Sketch curve	Sketch uses a brush-stroke effect to create a curve. You simply hold your left mouse button down and move your cursor to create a sketch curve.
	Helix	Creates a helix. User can specify a radius, the number of turns, the length and direction of the axis.
	Spiral	Creates a spiral. User can specify two radii, the number of turns, the length and direction of the axis.
	Conic	Creates a conic curve, which is part of an ellipse, parabola or hyperbola.

Option	Description
Undo	Type U, and press ENTER to remove one point at a time.
Close	Type C, and press ENTER to close the curve to its start point.

Exercise 25—Practice With Curves

1. Open the **Curve** model.

   In this exercise we will learn how to make a curve by control points, an interpolated curve, and a sketch curve to compare the differences between the three methods.

2. In the Osnap dialog, check **End** Osnap, uncheck all others. If you click with your right mouse on End it will uncheck all but the one you click on.

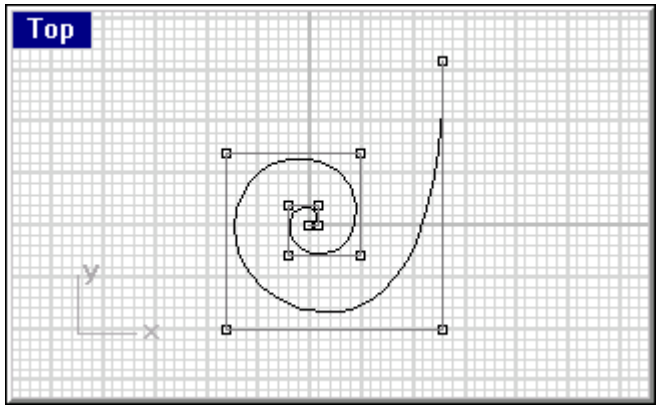
3. Toggle Ortho and Snap off.

Notes:

To draw a Curve by Control Points:

1. From the **Curve** menu, click **Freeform**, then click **Control Points.**
2. At the **Start of curve** prompt, snap to the end point of the polyline at **0,0**.
3. At the **Next point (Undo)** prompt, snap to the next End.
4. At the **Next point, press Enter when done (Undo)** prompt, continue to snap to the Ends on the polyline until you get to the end.
5. At the **Next point, press Enter when done (Close Undo)** prompt, press ENTER.

   A freeform curve is drawn. The control points were not on the curve except at the two ends. Control points control the curvature of the curve but usually do not fall on the curve.



To draw an Interpolated Curve:

1. From the **Curve** menu, click **Freeform**, then click **Interpolate Points.**
2. At the **Start of curve (Tangent Knots=SqrtChord)** prompt, snap to the End of the polyline at **0,0.**
3. At the **Next point on curve (Tangent Knots=SqrtChord Undo)** prompt, snap to the next End.
4. At the **Next point on curve , press Enter when done (Tangent Knots=SqrtChord Undo)** prompt, continue to snap to the next End until you get to the end.

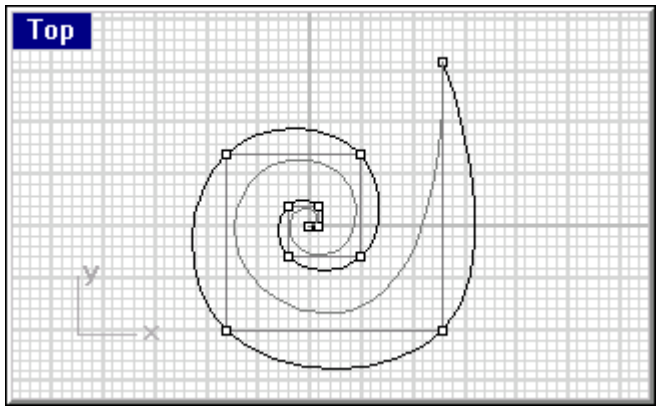
Notes:

Curve
Look for this button.

Interpolated Curve
Look for this button.

5 At the **Next point on curve , press Enter when done (Tangent Knots=SqrtChord Undo)** prompt, press ENTER.

A freeform curve is created from specified interpolate points. These points lie on the curve and determine its curvature.



To draw a sketch curve:

1 From the **Curve** menu, click **Freeform**, then click **Sketch.**

2 At the **Click and drag to sketch (Closed),** hold your left mouse button down and drag.

Points will appear as you are dragging, when you release your mouse the curve will replace the points.

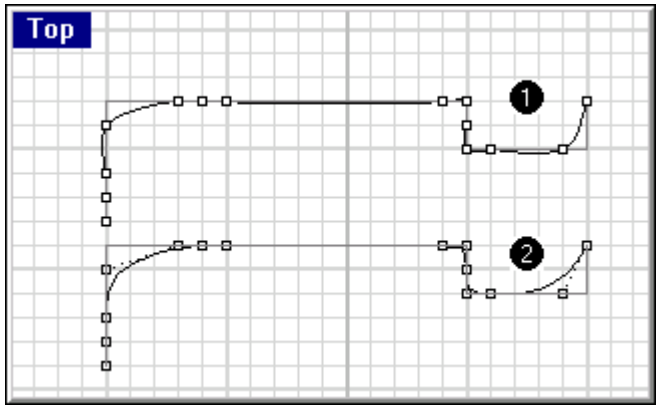
To draw a curve with tangency and straight segments.

1 Toggle Snap on.

Notes:

Curve
Look for this button.

2 Try to follow the construction lines to generate two curves like the illustration below. Use **Curve by Control Points** (2) for one of the curves and **Interpolated Curve** (1) for the other curve.



Notice that the Interpolated Curve (1) has bulges at some points and doesn't follow the construction line very well.

The Curve by Control Points (2) is much smoother. Notice that if you put three points in a row ninety degrees to each other it creates a straight segment. Three points equidistant and in each direction and ninety degrees from each other creates a smooth corner curve.

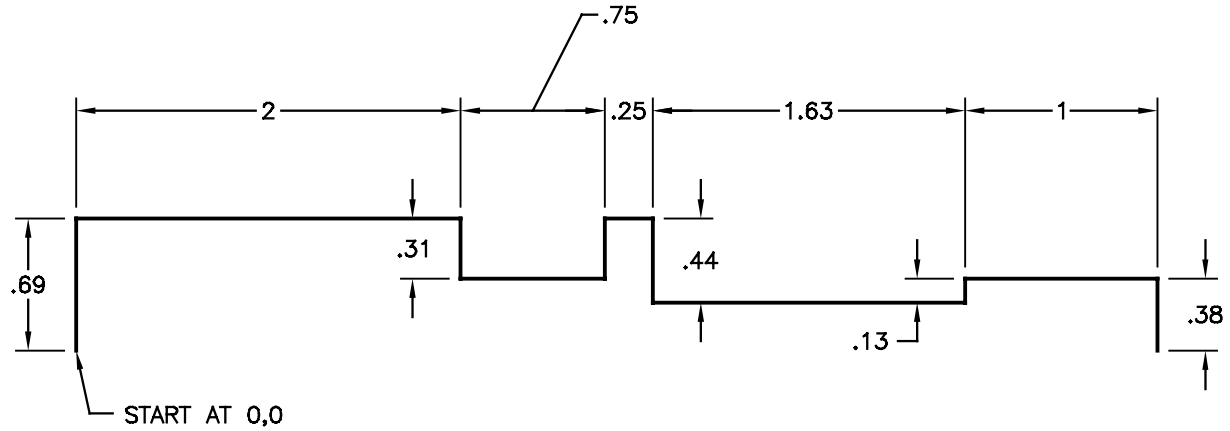
Notes:

Exercise 26—Practice With Curves

1. Start a new model. Save as **Screwdriver**.
2. In this exercise we will learn how to make a Curve by Control Points to create the following part.
3. Create **Construction**, **Curve** layers. Make them different colors.

To create the construction lines:

1. Change to the **Construction** layer.



2. From the **Curve** menu, click **Line**, then click **Polyline.**
3. At the **Start of polyline** prompt, type **0,0**, and press ENTER.
4. At the **Next point (undo)** prompt, type .69, and press ENTER. With ORTHO on pick a point above the first point.
5. Continue to use distant constraints and ortho mode to draw the construction line.

To create the Curve by Control Points:

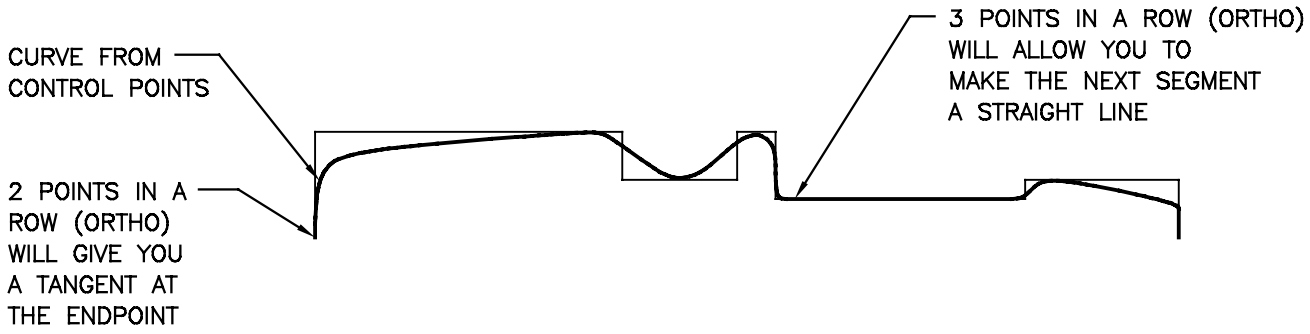
1. Change to the **Curve** layer.
2. From the **Curve** menu, click **Freeform**, then click **Control Points.**

Notes:

Polyline
Look for this button.

Curve
Look for this button.

3 At the **Start of curve** prompt, snap to the end point of the polyline at **0,0**.



4 At the **Next point (Undo)** prompt pick another point on the same segment of the construction line.

5 At the **Next point - enter to finish (Undo)** prompt pick a point in the direction of the next segment of the construction line (horizontal).

6 Continue picking points to create the curve. Use the construction lines to help you create the curve.

7 When you get to the shank of the screwdriver, you can get the curve to continue as a straight line if you put three points in a row (ortho) at the beginning. The next segment will be a straight line.

8 **Save** your model.

To make it 3D:

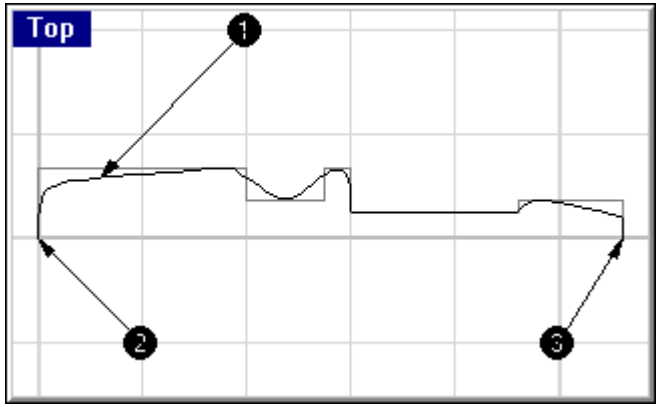
1 Toggle **Snap** and **Ortho** on.

2 From the **Surface** menu and click **Revolve**.

Notes:

Revolve
Look for this button.

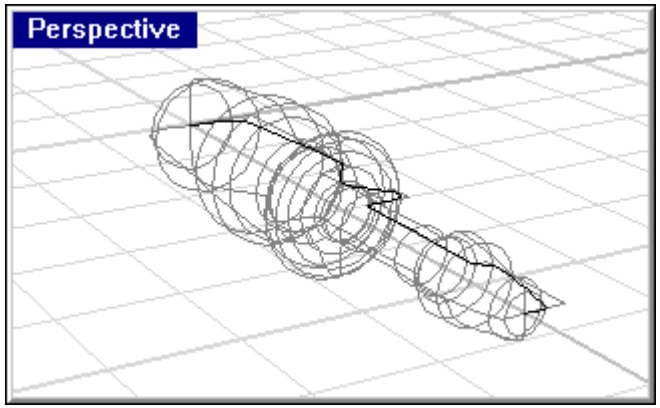
3 At the **Select curve to revolve** prompt, select the curve.



4 At the **Start of revolve axis** prompt, choose the end of the curve along the center line.

5 At the **End of revolve axis** prompt, choose the other end of the curve along the center line.

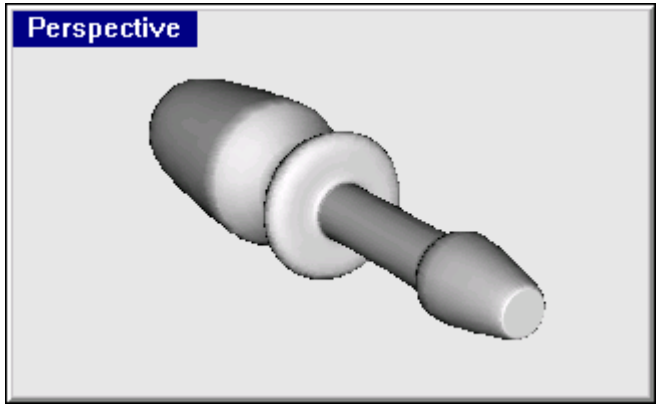
6 The **Revolve Options** dialog box will appear. Click **OK**. Your model is now a three-dimensional model.



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

Notes:

8 From the **Render** menu, click **Shade**.



Exercise 27— Drawing special curve forms: helix, spiral.

Start a new model. Save As **Helix.**

In this exercise we will learn how to make a helix, and a spiral.

To make a helix:

1 Toggle **Snap** and **Ortho** on.

2 From the **Curve** menu, click **Helix.**

3 At the **Start of axis (Vertical AroundCurve)** prompt, place your cursor in the **Front** viewport, type **0,0** and press ENTER.

4 At the **End of axis** prompt, type **10** and press ENTER.

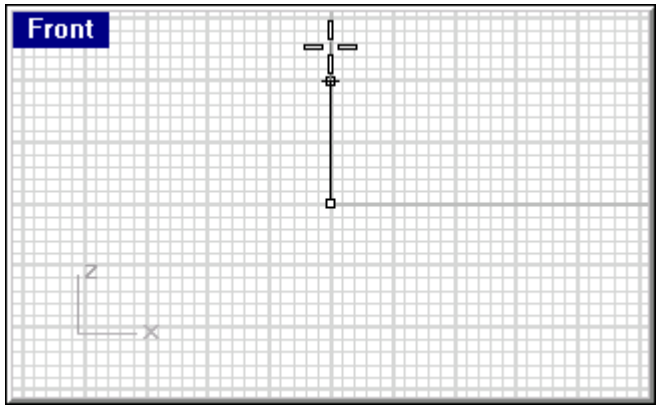
Notes:

Shade
Look for this button.

Helix
Look for this button.

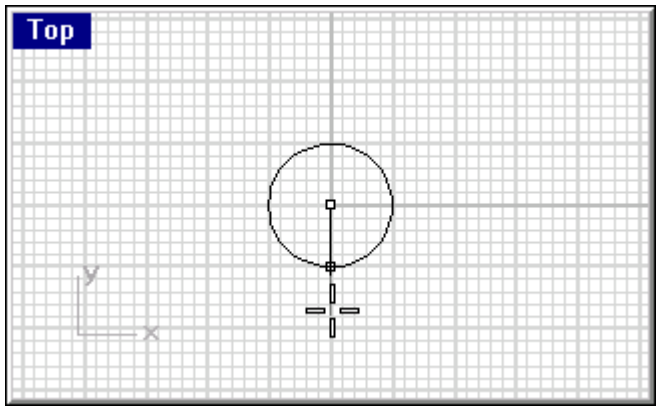
5 At the **End of axis** prompt, pick directly above your first point.

This makes a vertical axis of length 10.



6 At the **Radius and start point** prompt, type **5** and press ENTER.

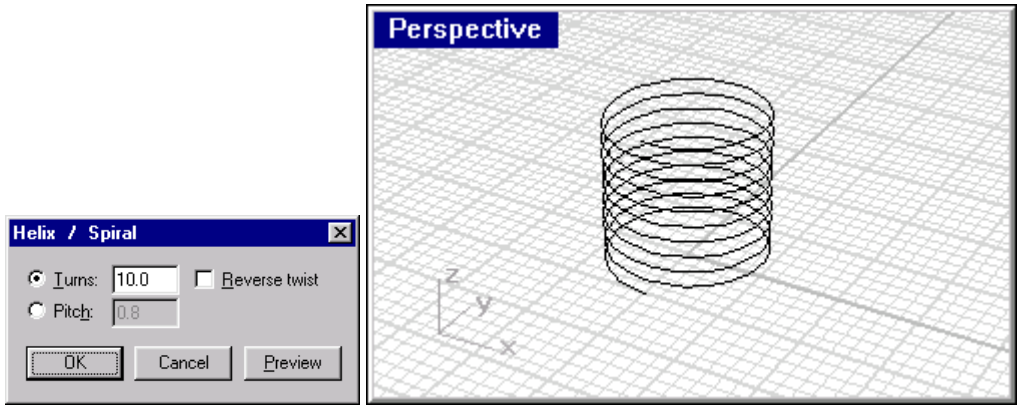
7 At the **Start of helix** prompt, move your cursor into the **Top** viewport and pick directly below your last point. This establishes where the helix will begin.



Notes:

8 In the **Helix / Spiral** dialog, click **OK**.

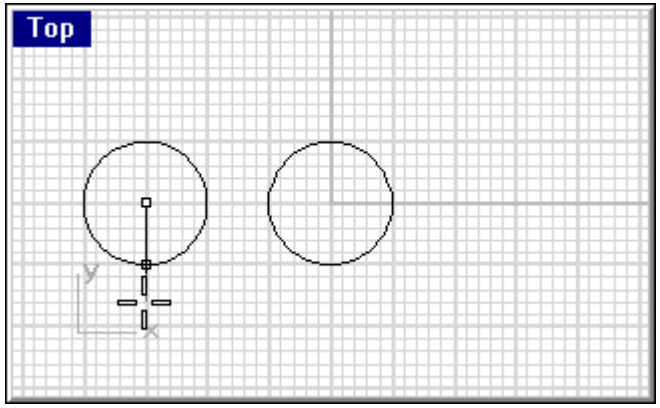
A helix is created



To create a spiral:

1 From the **Curve** menu, click **Spiral.**

2 At the **Start of axis (Flat Vertical AroundCurve)** prompt, drag your cursor into the **Front** viewport , type **-15,0** and press ENTER.



3 At the **End of axis** prompt, type **10** and press ENTER.

Notes:

Spiral
Look for this button.

4 At the **End of axis** prompt, pick directly above your last point.

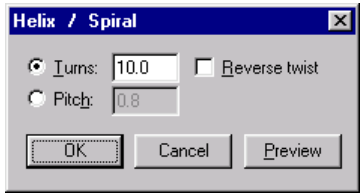
5 At the **Radius** and start point prompt, type **5** and press ENTER.

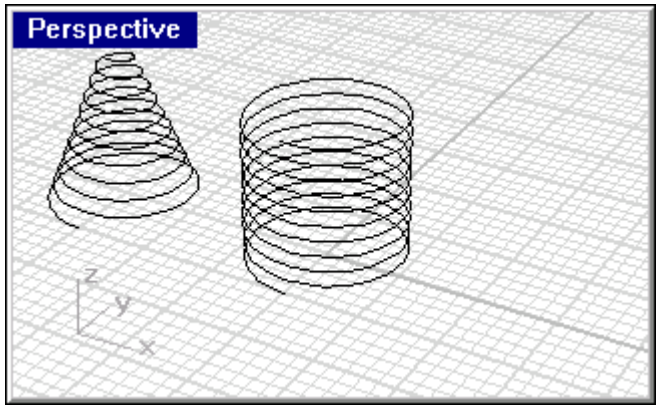
6 At the **Start of spiral** prompt, drag your cursor into the **Top** viewport and pick directly below your last point.

7 At the **End radius** prompt, type **1** and press ENTER.

8 In the **Helix / Spiral** dialog, click **OK**.

A spiral is created





Notes:

5 The Display

Changing the view of your model

When you add detail to your models, you'll need to see different parts of your model with different magnifications. You can use the view commands, the mouse, and the keyboard to change the view in a viewport.

Each view corresponds to the view through a camera lens. The invisible target of the camera is located in the middle of the viewport.

Viewports

With Rhino you can open an unlimited number of viewports. Each viewport has its own projection, view, construction plane, and grid. If a command is active, a viewport becomes active when you move the mouse over it. If a command is not active, you must click in the viewport to activate it.

Parallel vs. Perspective Projection

Unlike other modelers, Rhino lets you work in both parallel and perspective views. To toggle a viewport between parallel and perspective, right click on the title bar of the viewport, click Properties, then click parallel or perspective and OK.

Panning and Zooming

The simplest way to change the view is to hold down the SHIFT key and drag the mouse with right mouse button held down. This pans the view. To zoom in and out, hold down the CTRL key and drag up and down.

Notes:

You can also use the keyboard to move around:

Key	Action	+ CTRL
LEFT ARROW	Rotate left	Pan left
RIGHT ARROW	Rotate right	Pan right
UP ARROW	Rotate up	Pan up
DOWN ARROW	Rotate down	Pan down
PAGE UP	Zoom in	
PAGE DOWN	Zoom out	

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

Resetting your view

If you get lost, four view commands can help you get back to a starting place.

To undo and redo view changes:

- Click in a viewport, then press your HOME or END key on your keyboard to undo and redo view changes.

To set your view so you are looking straight down on the construction plane:

- From the **View** menu, click **Set View**, and then click **Plan**.

To bring all your objects into view:

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

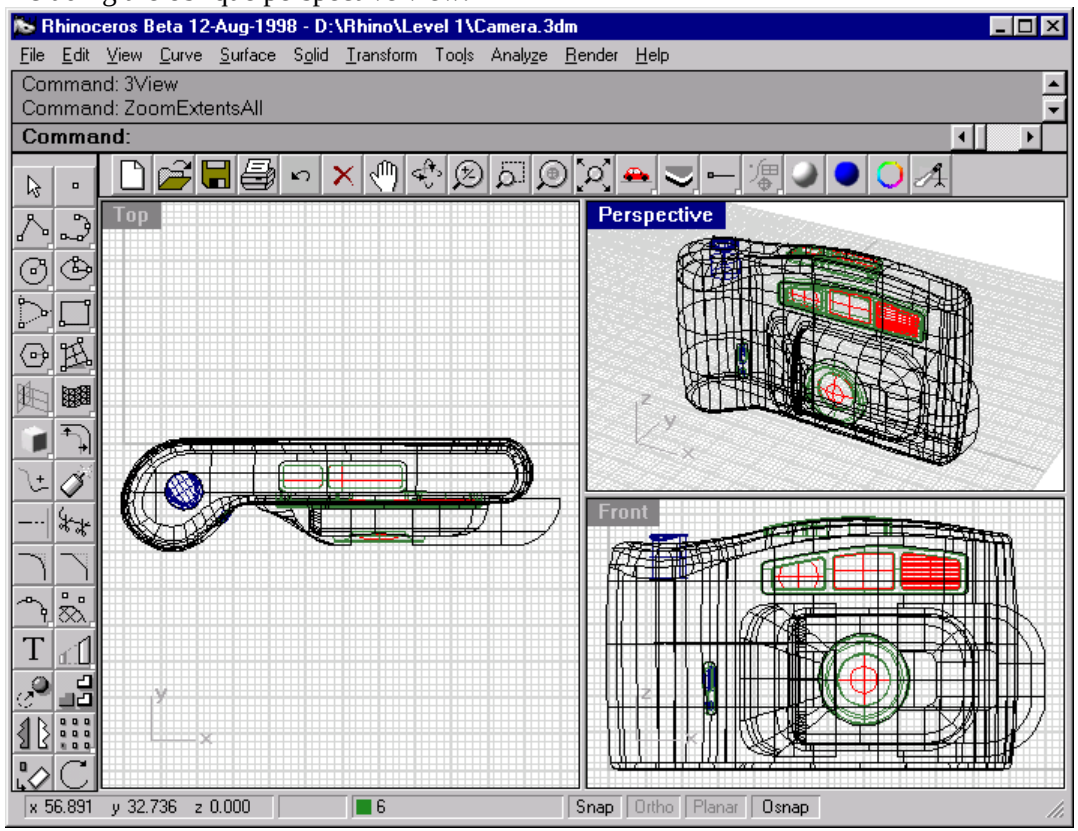
To bring all your objects into view in all viewports:

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

Notes:

Exercise 28—Zooming

- Open the **Camera** model. We will use this to practice changing views. We will create views from six directions including the oblique perspective view.



To change the viewport layout:

1. From the **View** menu, click **Viewport Layout**, then click **4 Viewports.**
2. From the **View** menu, click **Zoom**, then click **Extents All.**

Notes:

4 Viewports
Look for this button.

To change the number of viewports:

1. Make the **Top** Viewport active.
2. From the **View** menu, click **Viewport Layout**, then click **Split Vertical.**
3. From the **View** menu, click **Set View**, then click **Bottom.**
4. From the **View** menu, click **Zoom**, then click **Extents All.**
5. Make the **Front** Viewport active.
6. From the **View** menu, click **Viewport Layout**, then click **Split Vertical.**
7. From the **View** menu, click **Set View**, then click **Left.**
8. From the **View** menu, click **Zoom**, then click **Extents All.**
9. Make the **Right** Viewport active.
10. From the **View** menu, click **Viewport Layout**, then click **Split Vertical.**
11. Click in the lower right viewport.
12. From the **View** menu, click **Set View**, then click **Back.**
13. From the **View** menu, click **Zoom**, then click **Extents All.**

Notes:

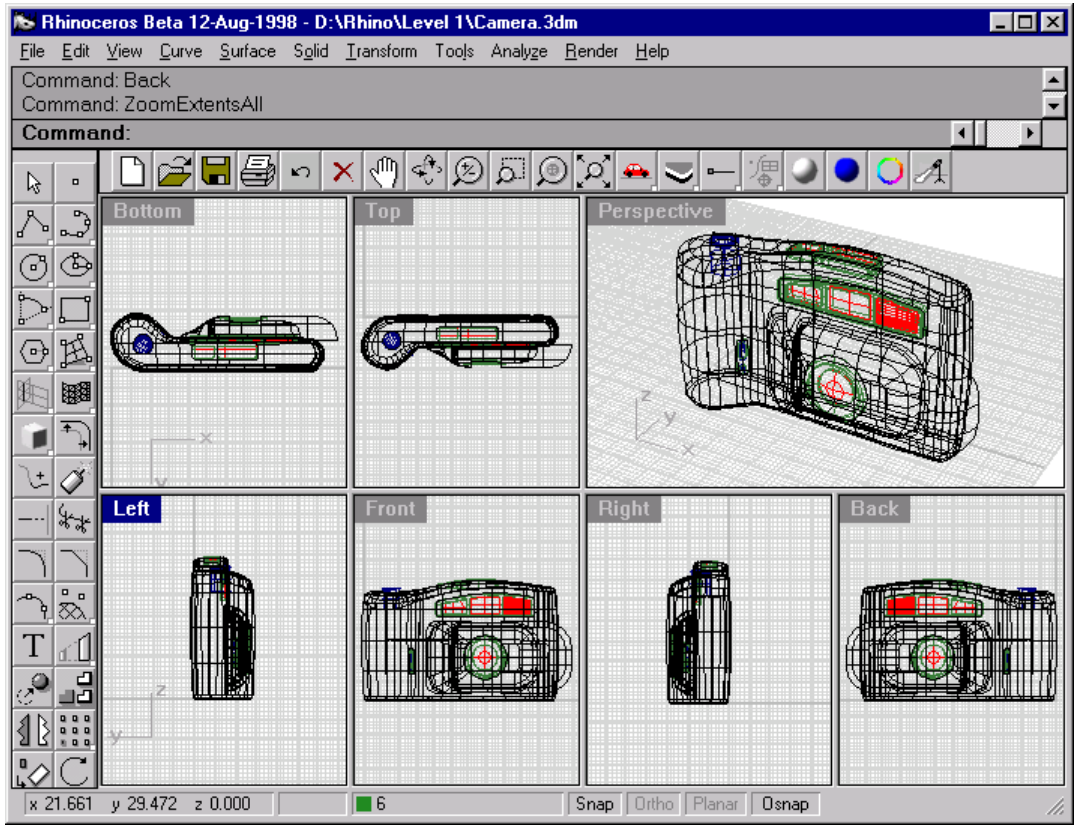
Zoom Extents All
Look for this button.

Bottom View
Look for this button.

Left View
Look for this button.

Back View
Look for this button.

14 Adjust the zoom in the **Right** and the **Left** viewport.



To change the shape of viewports:

1 Move your cursor to the edge of a viewport until you see the resizing cursor, hold the left mouse button down and drag the bar. If two viewports share the edge, both resize.

2 Move your cursor to the corner of a viewport until you see the resizing cursor, hold the left mouse and drag the intersection in any direction. If several viewports touch at that corner, all resize.

Notes:

To pan in a viewport:

- Hold down the SHIFT key and drag with the right mouse button to pan in any viewport.

To zoom in a viewport:

- Hold down the CTRL key and drag up or down with the right mouse button. Dragging down zooms out, dragging up zooms in.

To rotate in a viewport:

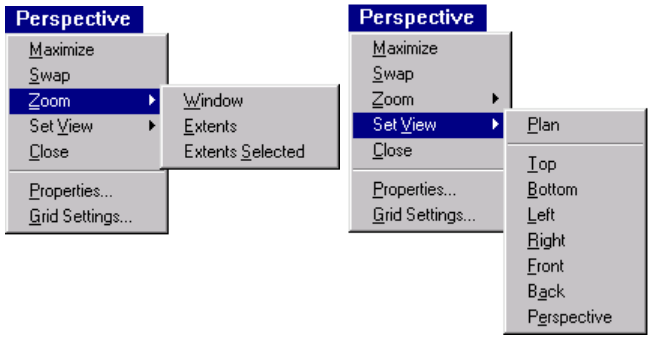
1. Drag with right mouse button in a perspective viewport to rotate the view.
2. To rotate the view in a parallel viewport use the arrow keys. See “Using keys to change your view” on page 114.

To maximize and restore a viewport:

1. Double click on the title of the viewport to maximize it.
2. Double click on the title of the maximized viewport to restore it to its smaller size.

Right mouse options:

- Right mouse click on the title of a viewport.
  A menu appears that allows you to maximize and restore a viewport, zoom a window or extents, change to the top, front or right view, and close a viewport.



Notes:

6 Editing Objects

Once you create objects, you can move and edit them to produce complex and detailed variations.

Fillet

Fillet connects two lines, arcs, circles or curves extending or trimming them to intersect or to join with a circular arc.

Option	Description
Radius	Sets the fillet radius. A radius of 0 extends or trims the curves to a corner, but doesn't create a fillet.
Join	**Yes** joins the fillet to the curves. **No** doesn't join the fillet to the curves.

Exercise 29—Fillet

To start the practice model:

- Open the **Fillet** model.

To fillet lines at a zero radius:

1. From the **Curve** menu, click **Fillet.**
2. At the **Select first curve to fillet (Radius=0 Join=No)** prompt, type **0** and press ENTER to change the radius. This radius will remain the default until it is changed.

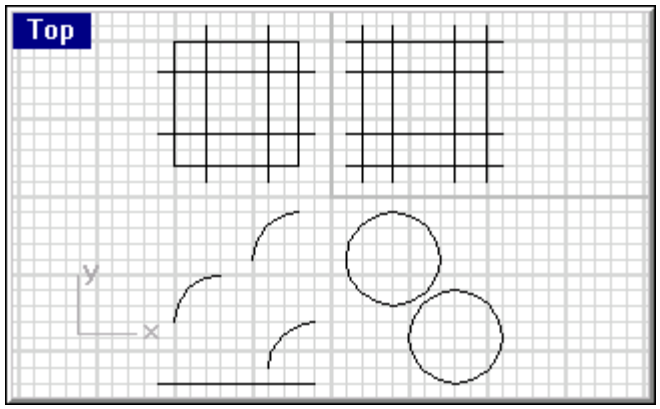
Notes:

Fillet
Look for this button.

3 At the **Select first curve to fillet (Radius=0 Join=No)** prompt, select an outer vertical line.

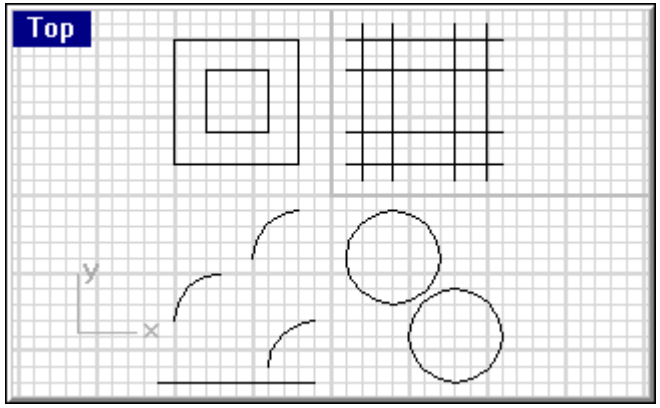
4 At the **Select second curve to fillet (Radius=0 Join=No)** prompt, select an adjacent horizontal line.

The ends of the lines are trimmed to a corner.



5 Press ENTER to repeat the command.

6 **Fillet** the other corners, as shown.



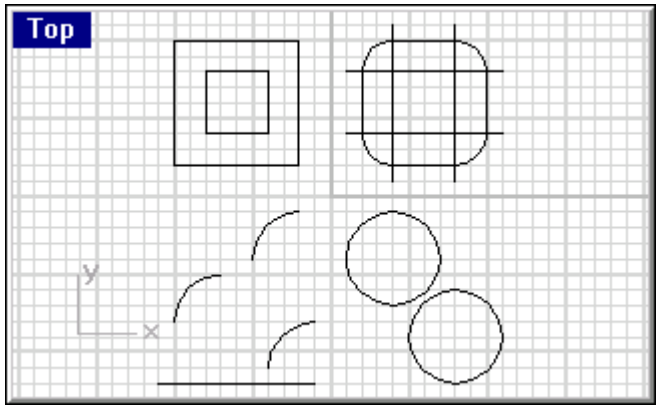
Notes:

Remember to pick on the part of the line that you want to keep.

To fillet lines using an arc:

1. From the **Curve** menu, click **Fillet.**
2. At the **Select first curve to fillet (Radius=0 Join=No)** prompt, type **2** and press ENTER to change the radius. This radius will remain the default until it is changed.
3. At the **Select first curve to fillet (Radius=2 Join=No)** prompt, select an outer vertical line.
4. At the **Select second curve to fillet (Radius=2 Join=No)** prompt, select an adjacent horizontal line.

   The ends of the lines are trimmed to a radius.
5. Press ENTER to repeat the command.
6. **Fillet** the other corners, as shown.

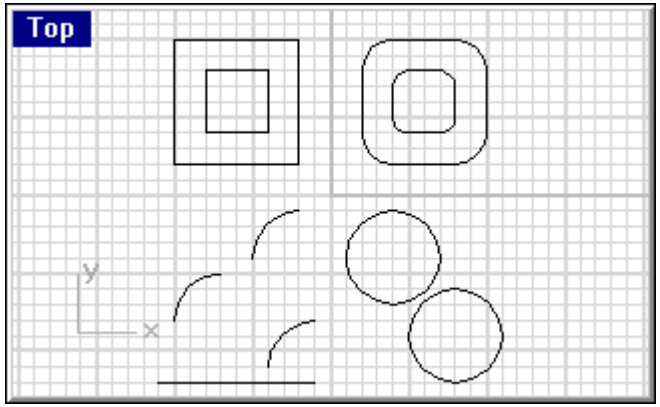


7. Press ENTER to repeat the command.
8. At the **Select first curve to fillet (Radius=2 Join=No)** prompt, type **1** and press ENTER. This radius will be used for the smaller object.
9. At the **Select first curve to fillet (Radius=1 Join=No)** prompt, select an outer vertical line.
10. At the **Select second curve to fillet (Radius=1 Join=No)** prompt, select an adjacent horizontal line.

Notes:

Fillet
Look for this button.

11 Fillet the other corners, as shown.



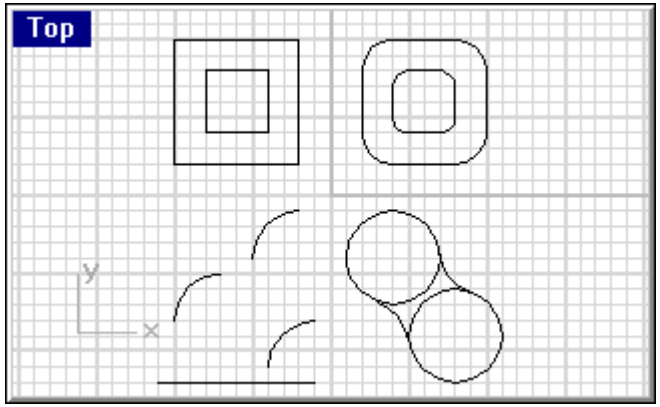
To fillet circles:

1. From the **Curve** menu, click **Fillet.**
2. At the **Select first curve to fillet (Radius=1 Join=No)** prompt, type **3** and press ENTER. This radius will be used for the smaller object.
3. At the **Select first curve to fillet (Radius=3 Join=No)** prompt, select right edge of the circle.
4. At the **Select second curve to fillet (Radius=3 Join=No)** prompt, select right edged of another circle.

Notes:

Fillet
Look for this button.

5 Repeat the command for the other side of the circles.



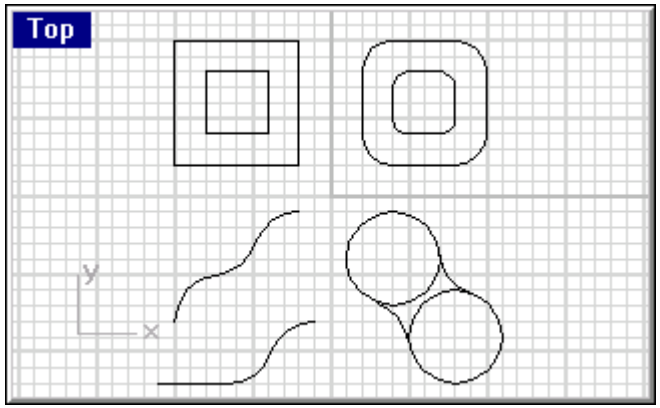
To fillet and join objects:

1 From the **Curve** menu, click **Fillet.**

2 At the **Select first curve to fillet (Radius=3 Join=No)** prompt, type **J** and press ENTER. This radius will be used for the smaller object.

3 At the **Select first curve to fillet (Radius=3 Join=Yes)** prompt, select one of the arcs in the middle of the screen.

4 At the **Select second curve to fillet (Radius=3 Join=Yes)** prompt, select the other arc in the middle of the screen.



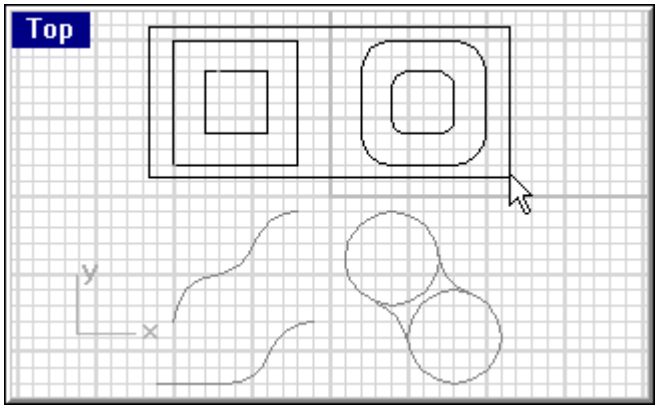
Notes:

Fillet
Look for this button.

5 Repeat this procedure for the arc and the line at the bottom left.

To join the filleted objects:

1 Select the objects at the top with a window.



2 From the **Edit** menu, click **Join.**

The objects are joined together. Pre-selected curves join only if they touch..

To make 3D with a lofted surface:

1 Change to the **Perspective** viewport

2 Change to the **Surfaces** layer.

3 Select the two squares in the upper left part of the **Top** viewport.

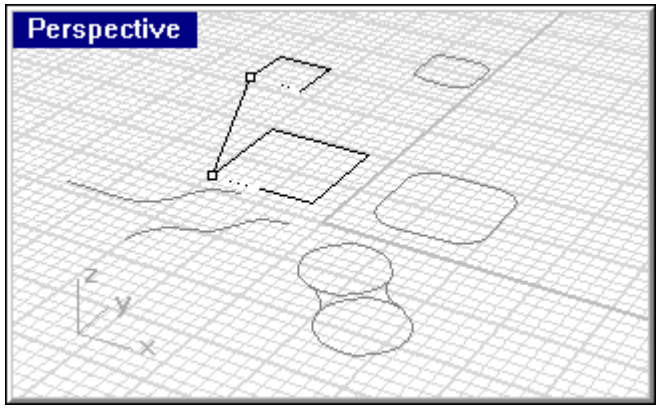
Notes:

Join
Look for this button.

The lines in this model are at two different elevations. We are going to connect the lines at different elevations with a surface.

4 From the **Surface** menu, click **Loft.**

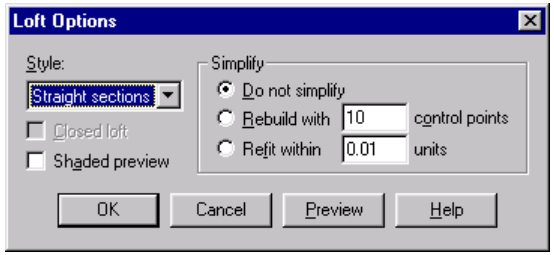
The two squares show a seam direction arrow. They point the same direction.

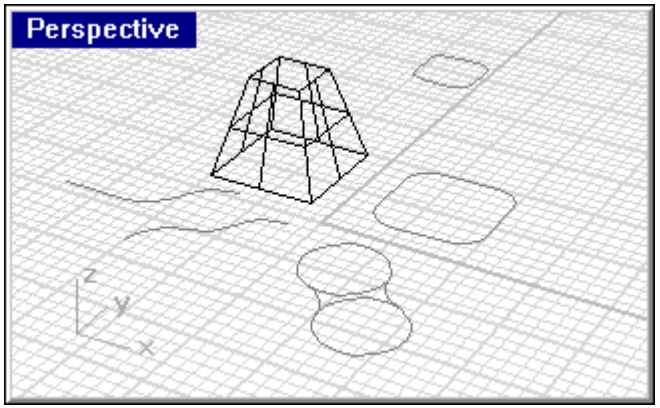


5 At the **Select seam point to adjust, press ENTER when done (FlipDirection Automatic Natural)** prompt, press ENTER.

6 In the **Loft Options** dialog, change the **Style** to **Straight Sections**, then click **OK**.

A surface is generated between the two squares.



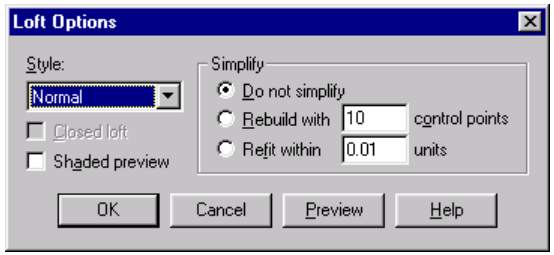


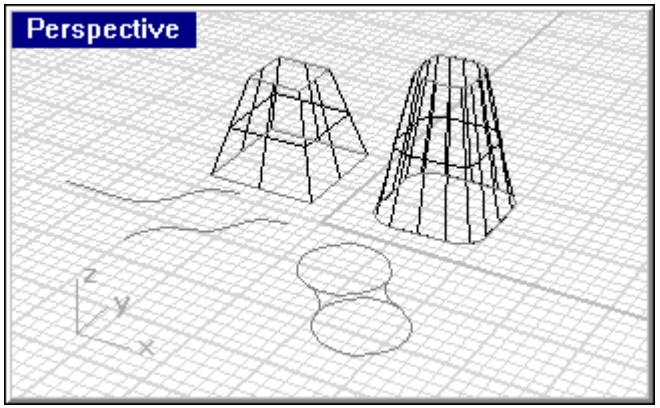
Notes:

Loft
Look for this button.

7 Repeat the procedure for the rounded squares until the **Loft Options** dialog appears.

8 In the **Loft Options** dialog, change the **Style** to **Normal**, then click **OK**.

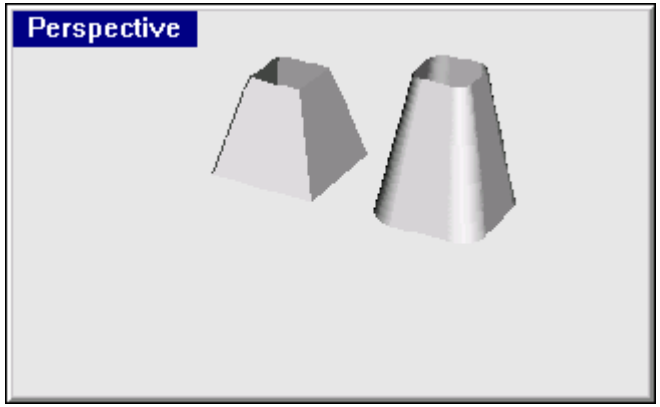




9 **Save** your model.

10 Click in the **Perspective** viewport.

11 From the **Render** menu, click **Shade**.



Notes:

Shade
Look for this button.

Chamfer

Chamfer connects two curves by extending or trimming them to intersect or to join with a beveled line. Chamfer works on convergent or intersecting lines.

Option	Description
Distances	The first chamfer distance specifies the distance from the chamfer end on the first curve to the point where the two curves would intersect. The second chamfer distance specifies the distance of the chamfer end on the second curve to the intersection point. A chamfer distance of 0 trims or extends that curve to the intersection point. If the chamfer distance is non-zero, a chamfer line is created if needed to extend the other curve to the intersection. If you enter 0 for both distances, the curves are trimmed or extended to their intersection, but no chamfer is created.

Exercise 30—Chamfer

To start the practice model:

- Open the **Chamfer** model.

To chamfer lines:

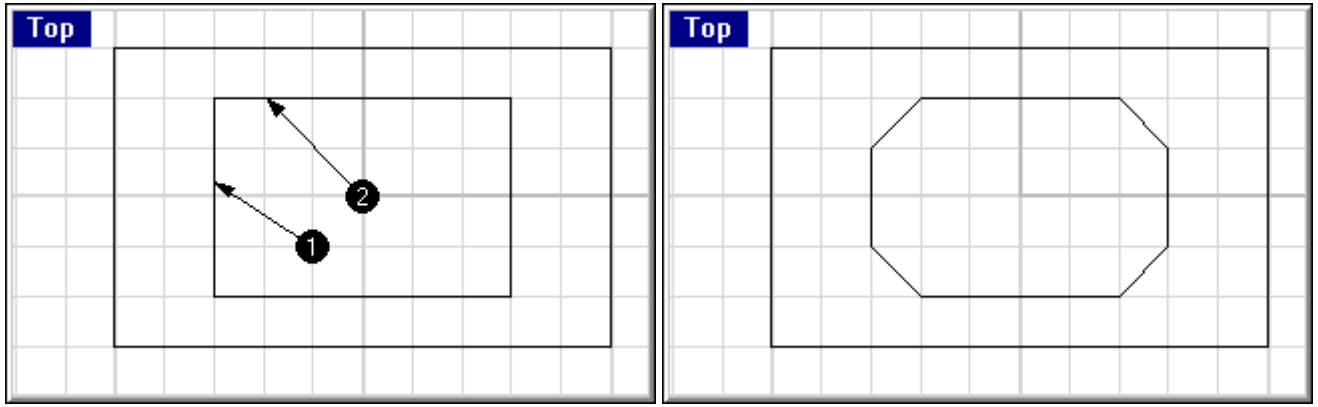
1. From the **Curve** menu, click **Chamfer.**
2. To set a distance, at the **Select first curve to chamfer (Distances=0,0)** prompt, type **1**, and press ENTER.
3. At the **Second chamfer distance<1>** prompt, press ENTER.

   The default second chamfer distance always matches the first chamfer distance. This creates chamfers that begin and end the same distance from the point where the two lines would intersect.
4. At the **Select First Curve to chamfer (Distances=1,1)** prompt, choose one of the inner vertical lines.

Notes:

Chamfer
Look for this button.

5 At the **Select Second Curve to chamfer (Distances=1,1)** prompt, choose an adjacent horizontal line.



6 Continue creating chamfers on all of the corners as shown.

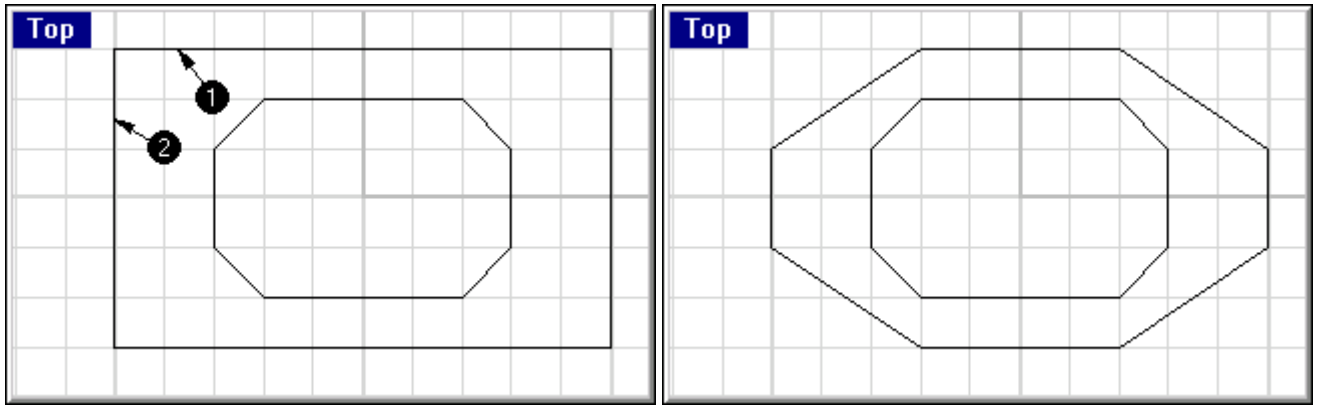
7 Press ENTER to repeat the command.

8 At the **Select First Curve to chamfer (Distances=1,1)** prompt, type **3**, and press ENTER.

9 At the **Second chamfer distance<3>** prompt, type **2**, and press ENTER.

10 At the **Select First Curve to chamfer (Distances=3,2)** prompt, choose one of the outer horizontal lines.

11 At the **Select Second Curve to chamfer (Distances=3,2)** prompt, choose an adjacent vertical line.



Notes:

12 Continue creating chamfers on all of the corners as shown.

To join the chamfered objects:

1 From the **Edit** menu, click **Select**, then click **All**, then click **Curves.**

2 From the **Edit** menu, click **Join.**

The curves have been joined together.

To make it 3D:

1 Change to the **Surfaces** layer.

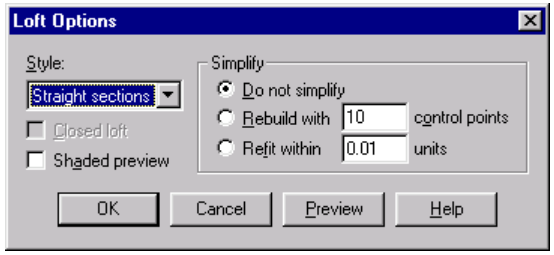
2 From the **Edit** menu, click **Select**, then click **All**, then click **Curves.**

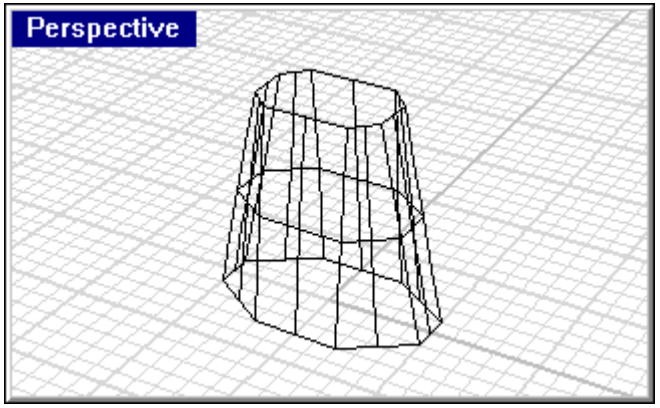
3 From the **Surface** menu, click **Loft.**

4 At the **Select seam point to adjust, press ENTER when done (FlipDirection Automatic Natural)** prompt, press ENTER.

5 In the **Loft Options** dialog, change the **Style** to **Straight Sections**, then click **OK**.

A surface is generated between the two squares.





6 **Save** your model.

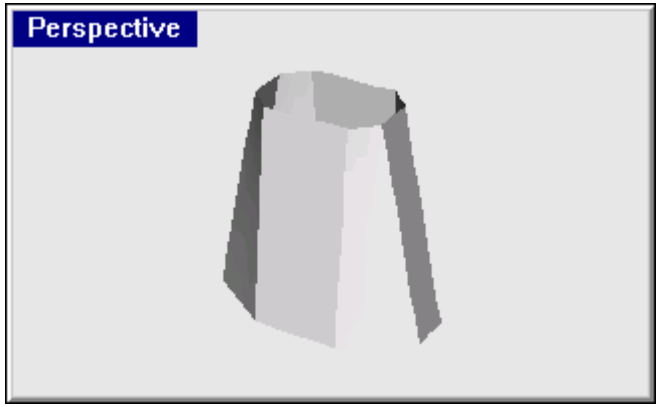
7 Click in the **Perspective** viewport.

Notes:

Join
Look for this button.

Loft
Look for this button.

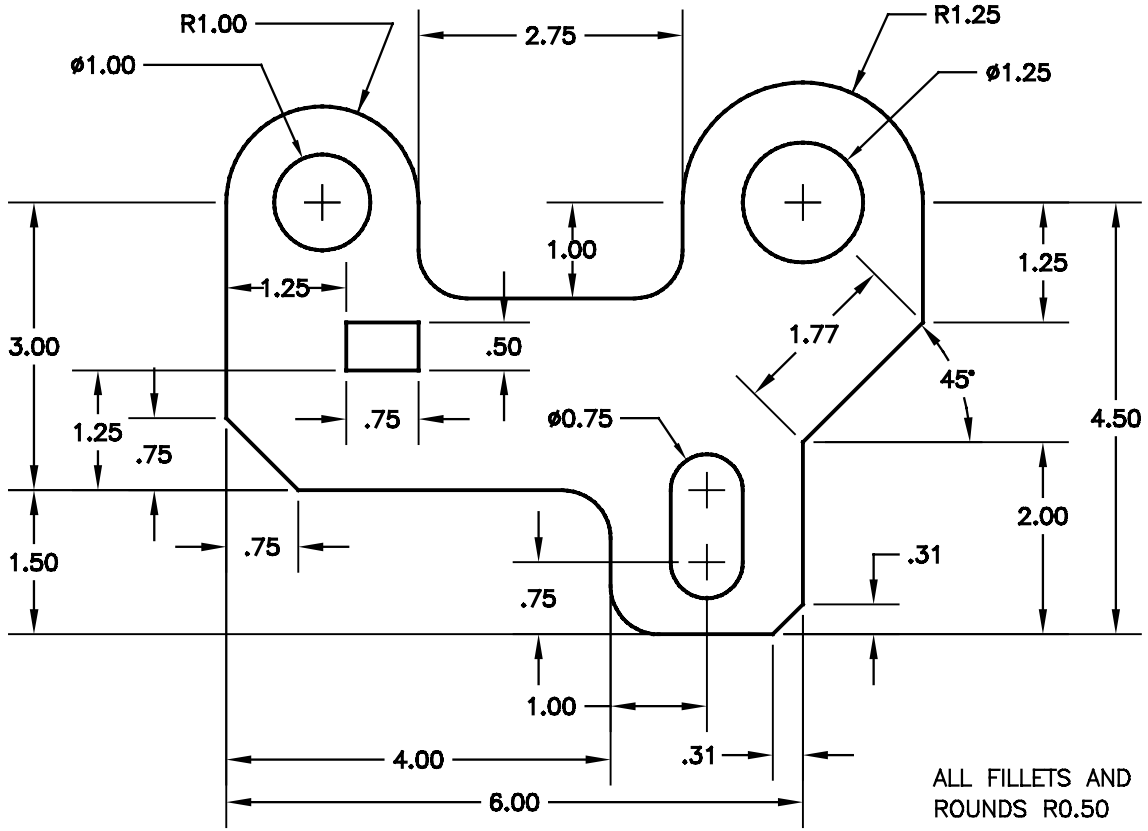
8 From the **Render** menu, click **Shade**.



Notes:

Shade
Look for this button.

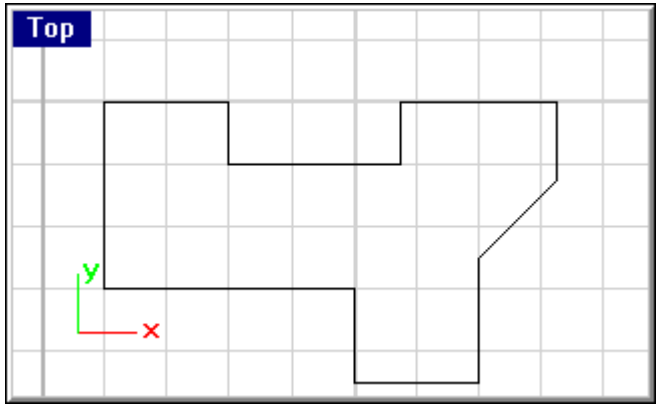
Exercise 31—Practice with Fillet and Chamfer

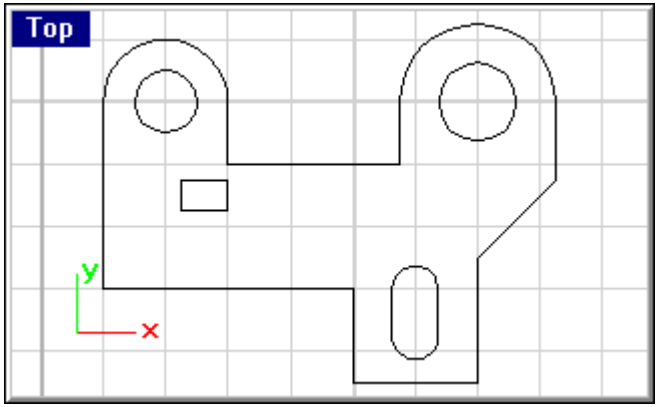


1 Start a new model. Save as **Filletex**.

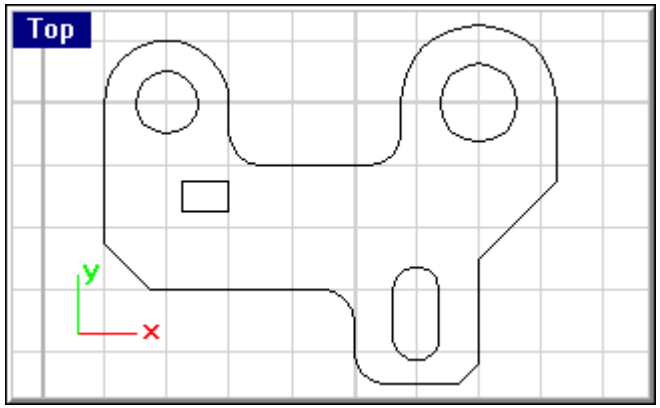
Notes:

2 Draw the outline of the part with Line Segments, Circles, and Arcs.





3 Use **Fillet** and **Chamfer** to edit the drawing as shown.



To make it 3D:

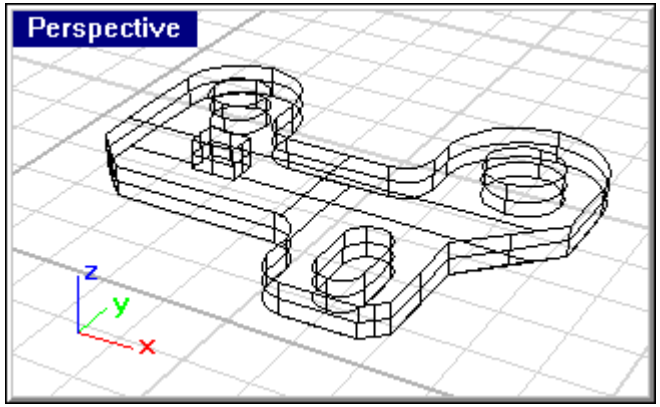
1 From the **Edit** menu, click **Select**, then click **All**, then click **Curves.**

2 From the **Solid** menu, click **Extrude Planar Curve.**

Notes:

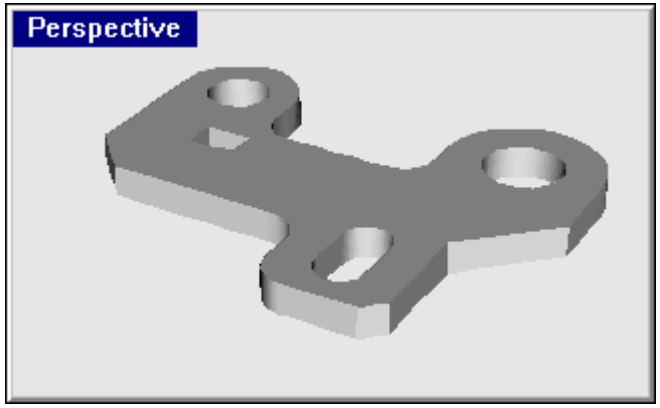
Extrude Planar Curve
Look for this button.

3 At the **Extrusion distance (Direction Cap=Yes BothSides Tapered)** prompt, type **.5** and press ENTER.



4 Click in the **Perspective** viewport.

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



Notes:

Shade
Look for this button.

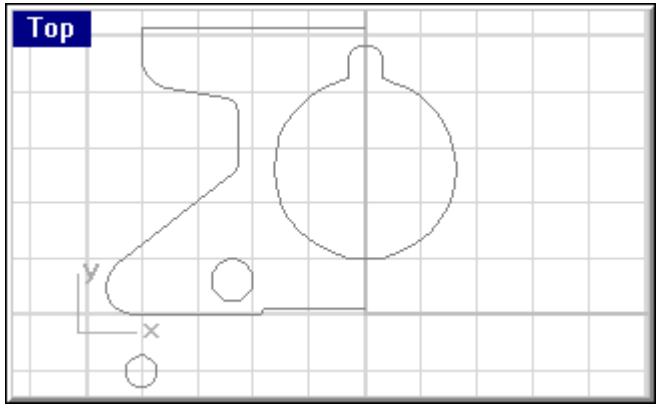
Move

Use Move to move objects without changing orientation or size.

Option	Description
Vertical	Moves the selected objects vertical to the current construction plane.

Exercise 32—Move

1 Open the **Move** model.



2 Turn off both **Ortho** and **Snap** so you can move objects freely.

3 Turn **Cen** object snap on.

To move using object snaps for placement:

1 Select the small circle at the bottom left side of the top view.

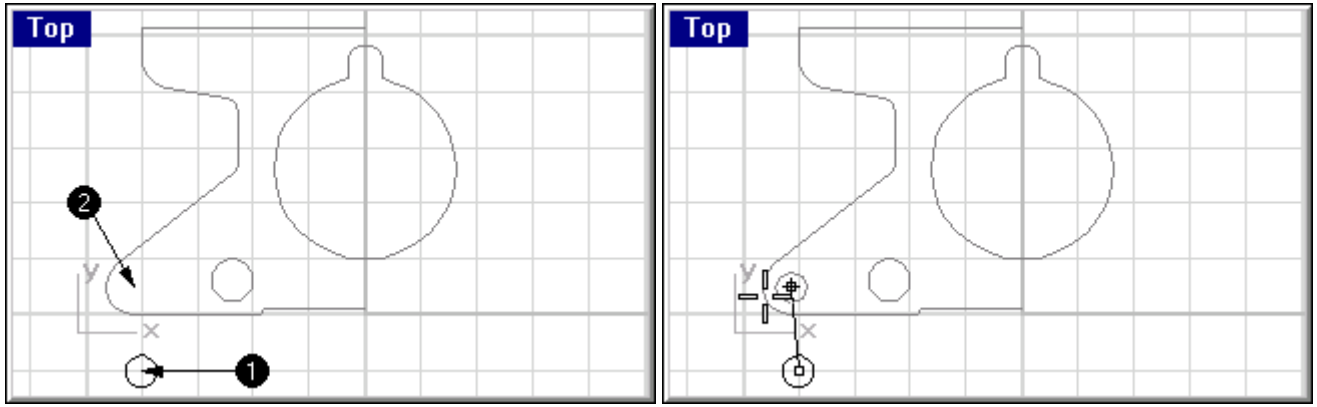
2 From the **Transform** menu, click **Move.**

3 At the **Point to move from (Vertical)** prompt, snap to the center of the small circle.

Notes:

Move
Look for this button.

4 At the **Point to move to** prompt, snap to the center of the arc at the lower left of the object.



To move using absolute coordinates:

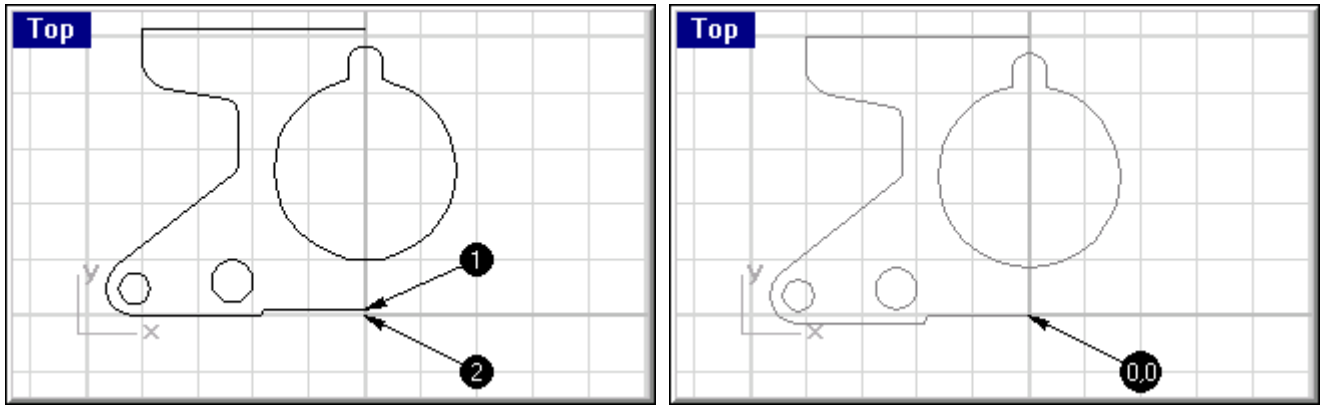
1 From the **Edit** menu, click **Select**, then click **All**, then click **Curves.**

2 From the **Transform** menu, click **Move.**

3 At the **Point to move from (Vertical)** prompt, snap to the end of the line at the lower part of the object.

4 At the **Point to move to** prompt, type **0,0** and press ENTER.

The end of the line is exactly at point 0,0 in the top view.

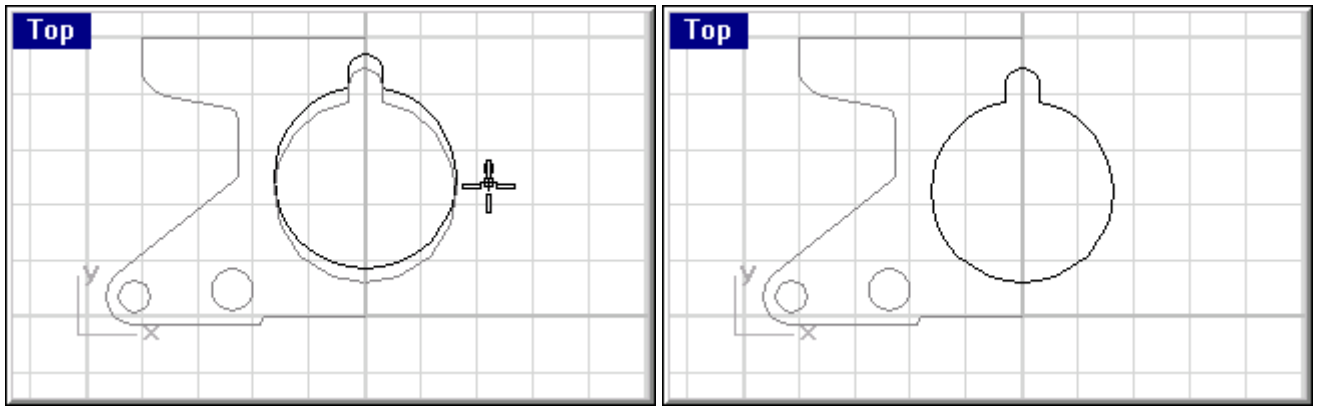


Notes:

Move
Look for this button.

To move using relative coordinates:

1. Select the large slotted circle in the middle of the object. We will move the slotted circle relative to the part.
2. From the **Transform** menu, click **Move.**
3. At the **Point to move from (Vertical)** prompt, pick any point in the top view.
   It is usually better if you pick near the object you are going to move.
4. At the **Point to move to** prompt, type **r0,-.25** and press ENTER.
   The circle is moved down .25 units.



Copy

Copy duplicates selected objects and places them in a new location. The command repeats to create more than one copy in the same command sequence.

Option	Description
Vertical	Copies the selected objects vertical to the current construction plane.
InPlace	Duplicates the object at the current location.

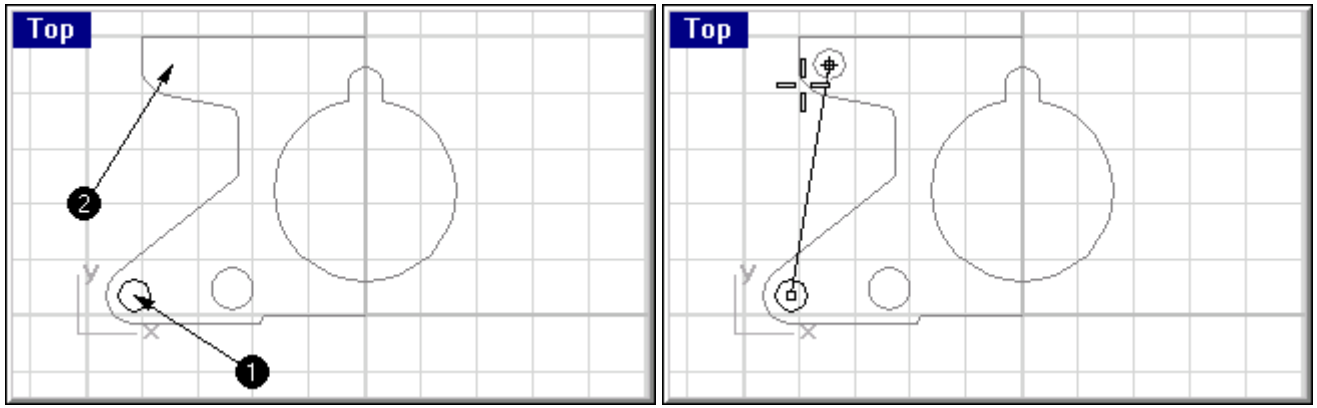
Notes:

Move
Look for this button.

Exercise 33—Copy

To copy using object snaps for placement:

1. Select the small circle at the lower left of the object.
2. From the **Transform** menu, click **Copy.**
3. At the **Point to copy from (Vertical InPlace)** prompt, snap to the center of the small circle.
4. At the **Point to copy to** prompt, snap to the center of the arc at the upper left of the object.



5. Press ENTER to end the command.

To make multiple copies:

1. Select the small circle at the lower left of the object.
2. From the **Transform** menu, click **Copy.**
3. At the **Point to copy from (Vertical InPlace)** prompt, snap to the center of the small circle.

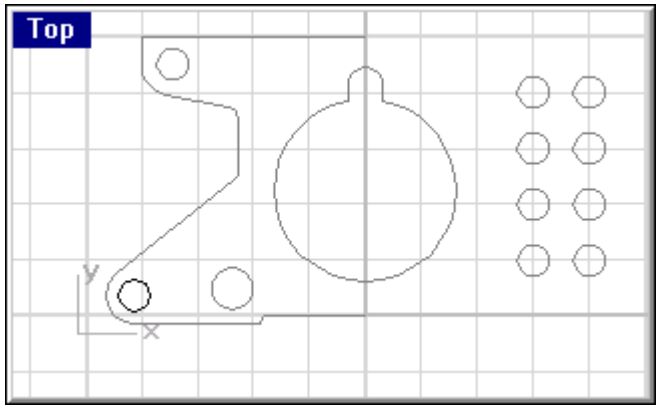
Notes:

Copy
Look for this button.

Copy
Look for this button.

4 At the **Point to copy to** prompt, begin picking points on the screen.

Each time you pick a circle will be copied to that location.



5 Press ENTER to end the command.

Undo and Redo

If you make a mistake or don't like the results of a command, use **Undo**. If you decide that you did like those results, after undoing them, use **Redo** command. Redo restores the last thing undone.

Rhino Button	Mouse Button	Command	Description
	Left-click or CTRL+Z	Undo	Cancels the effects of the last command.
	Right-click or CTRL+Y	Redo	Restores the effects of undo.

The number of undos kept in memory can be set in Rhino **Options** under the **General** tab,

If a command has an **Undo** option, type **U** to enter it. At the command prompt, type **Undo**.

You cannot use Undo after you exit a modeling session or open a different model.

Notes:

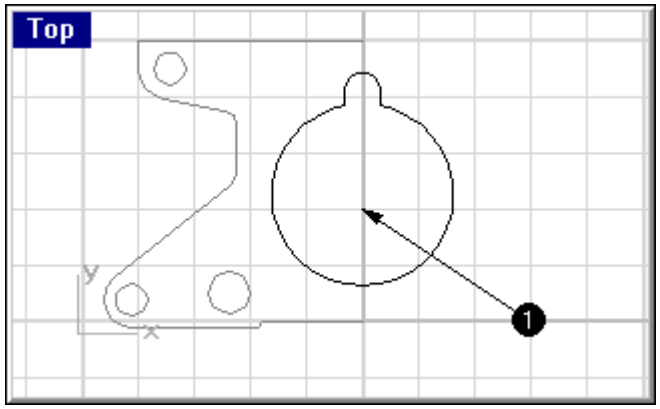
Rotate

Use **Rotate** to move objects in a circular motion around a base point. For precise rotation, enter a number of degrees to rotate. **Positive** numbers rotate **counterclockwise**, **negative** numbers rotate **clockwise.**

Option	Description
Copy	Duplicates the object as it is rotated

Exercise 34—Rotate

1. Select the large slotted circle in the middle of the object.
2. From the **Transform** menu, click **Rotate.**
3. At the **Center of rotation (Copy)** prompt, snap to the center of the slotted circle.

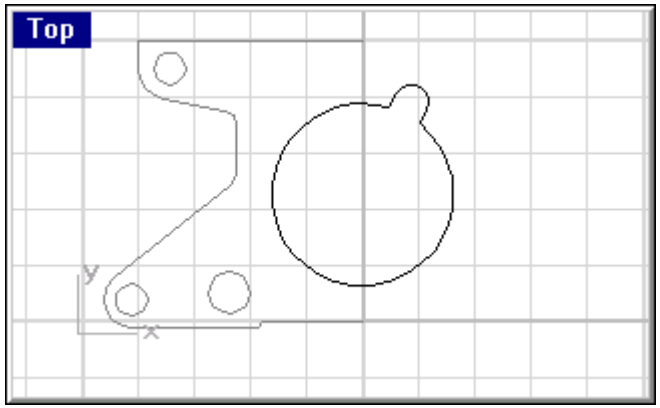


Notes:

Rotate
Look for this button.

4 At the **Angle or first reference point (Copy)** prompt, type **-28** and press ENTER.

The slotted circle rotated clockwise 28 degrees.



Mirror

Mirror creates a copy of the objects flipped over a specified axis on the construction plane.

Exercise 35—Mirror

1 From the **Edit** menu, click **Select**, then click **Invert.**

This selects everything but the object that was previously selected. All of the curves except the slotted circle should be highlighted.

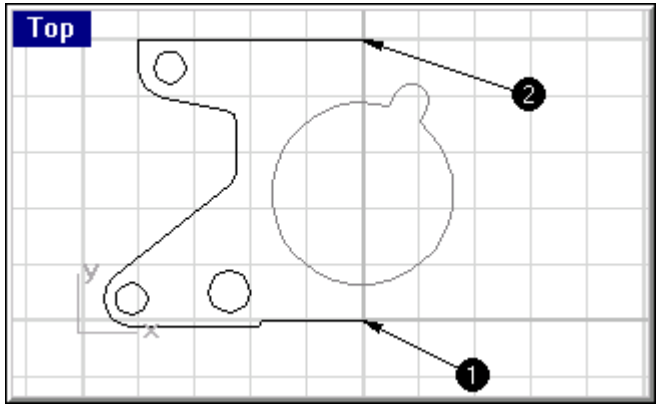
2 From the **Transform** menu, click **Mirror.**

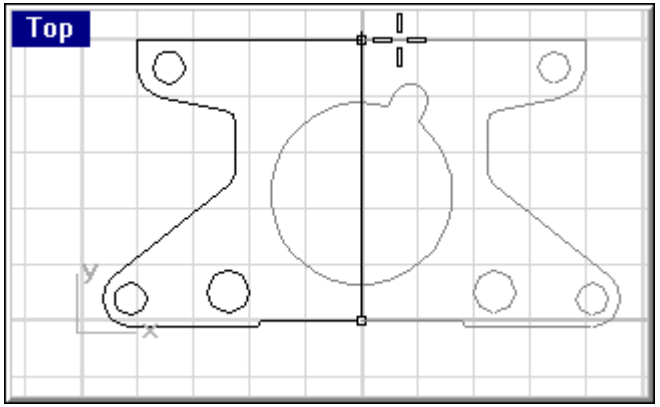
3 At the **Start of mirror plane** prompt, type 0,0 or snap to the End of the line at the lower right of the part.

Notes:

Mirror
Look for this button.

4 At the **End of mirror plane** prompt, toggle **Ortho** on and pick directly above the previous point.





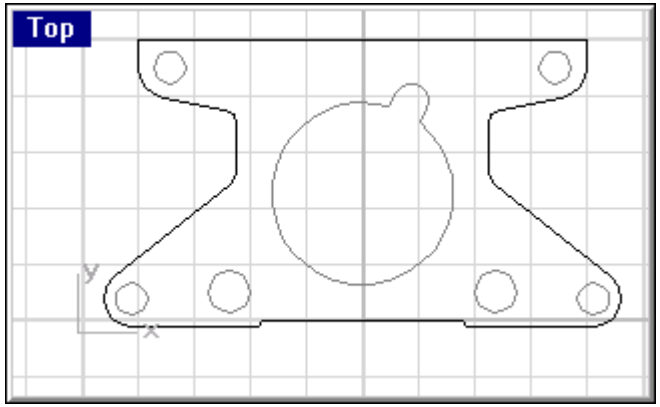
Join

Join unites curves that meet at a common end, making a single curve. Join can unite curves that don't touch, if you select them after the command has started. As you pick non-touching curves a dialog asks if you want to fill the gap.

Exercise 36—Join

1 From the **Edit** menu, click **Select**, then click **All**, then click **Curves.**

2 From the **Edit** menu, click **Join.**



Notes:

Join
Look for this button.

Scale

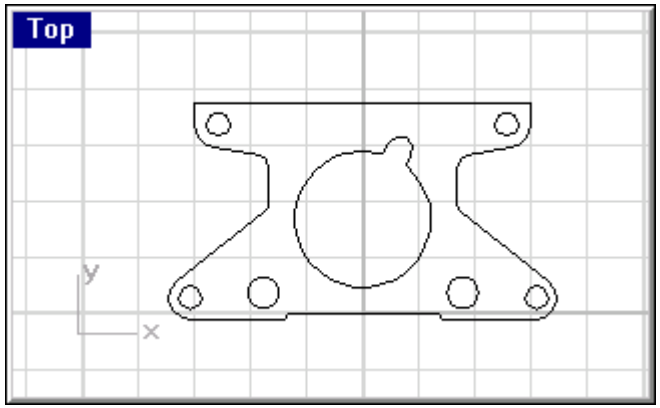
Scale changes the size of existing objects without changing their shape. This command will scale three-dimensional objects along all three axes. There are also 2-dimensional, one-dimensional, and non-uniform scale command.

Option	Description
Copy	Preserves the object as it creates a new scaled one.
Scale factor	Inputs a value for scale factor. Less than one reduces the size, more than one increases the size.
Reference point	The first reference point shows the current scale factor. The second reference point shows the new scale factor.

Exercise 37—Scale

1. From the **Edit** menu, click **Select**, then click **All**, then click **Curves.**
2. From the **Transform** menu, click **Scale**, then click **Scale 2-D.**
3. At the **Origin point (Copy)** prompt, type **0,0** and press ENTER.
4. At the **Scale factor or first reference point (Copy)** prompt, type **.75** and press ENTER.

   The whole part has been scaled to seventy-five percent of its original size.



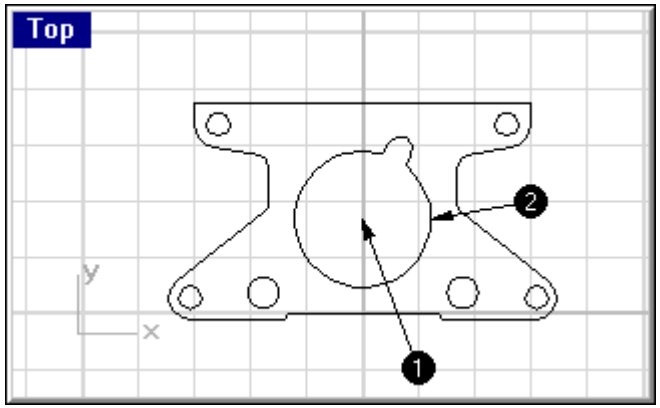
Notes:

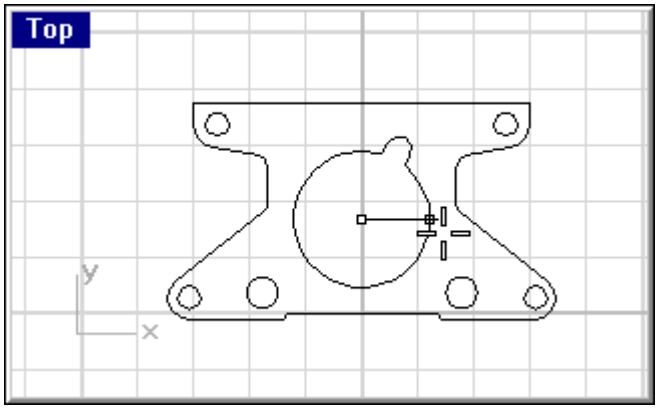
Scale
Look for this button.

To scale using the reference point option:

1. From the **Edit** menu, click **Select**, then click **All**, then click **Curves.**
2. From the **Transform** menu, click **Scale**, then click **Scale 2-D.**
3. At the **Origin point (Copy)** prompt, snap to the center of the slotted circle.
4. At the **Scale factor or first reference point (Copy)** prompt, snap to the quadrant of the slotted circle.

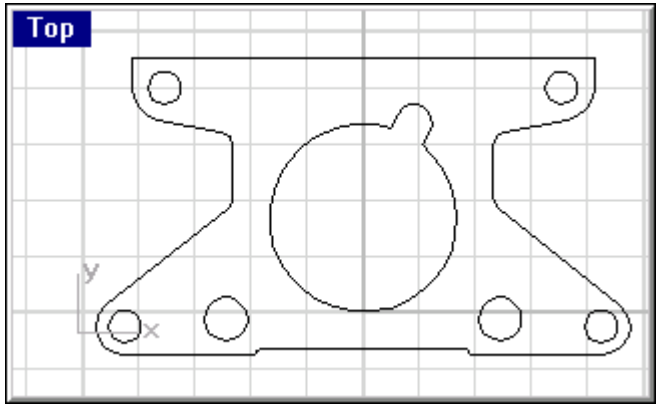
   The radius of the slotted circle is the reference for the scale factor.





5. At the **Second reference point (Copy)** prompt, type **1.375** and press ENTER.

   The slotted circle now has a radius of 1.375. Everything else was scaled proportionally.



Notes:

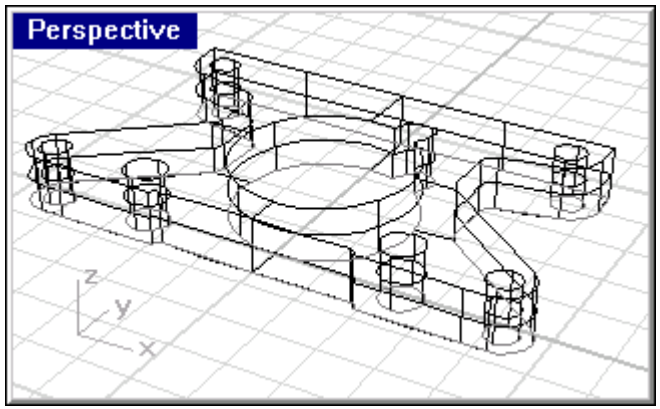
Scale
Look for this button.

To make it 3D:

1 From the **Edit** menu, click **Select**, then click **All**, then click **Curves.**

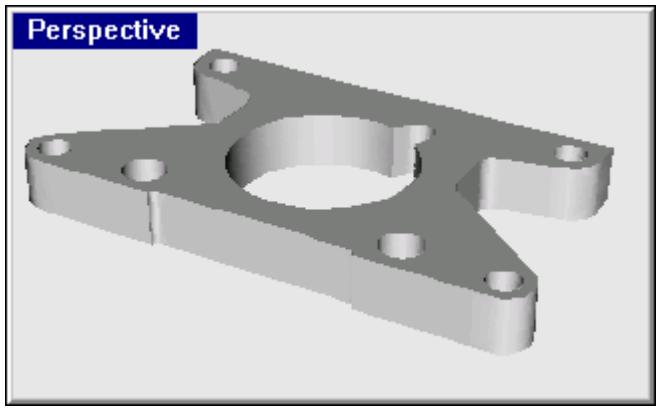
2 From the **Solid** menu, click **Extrude Planar Curve.**

3 At the **Extrusion distance (Direction Cap=Yes BothSides Tapered)** prompt, type **1** and press ENTER.



4 Click in the **Perspective** viewport.

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



Notes:

Extrude Planar Curve
Look for this button.

Shade
Look for this button.

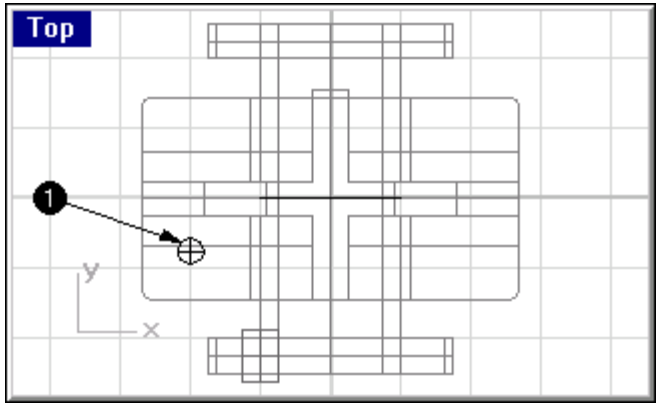
Array

Use the Array commands to make multiple copies of selected objects. Array Rectangular makes rows and columns of objects. Array Polar copies objects in a circle around a center point

Exercise 38—Array

To create a rectangular array:

1. Open the **T-Flange** model.
2. Select the cylinder that is visible in the **Top** viewport.



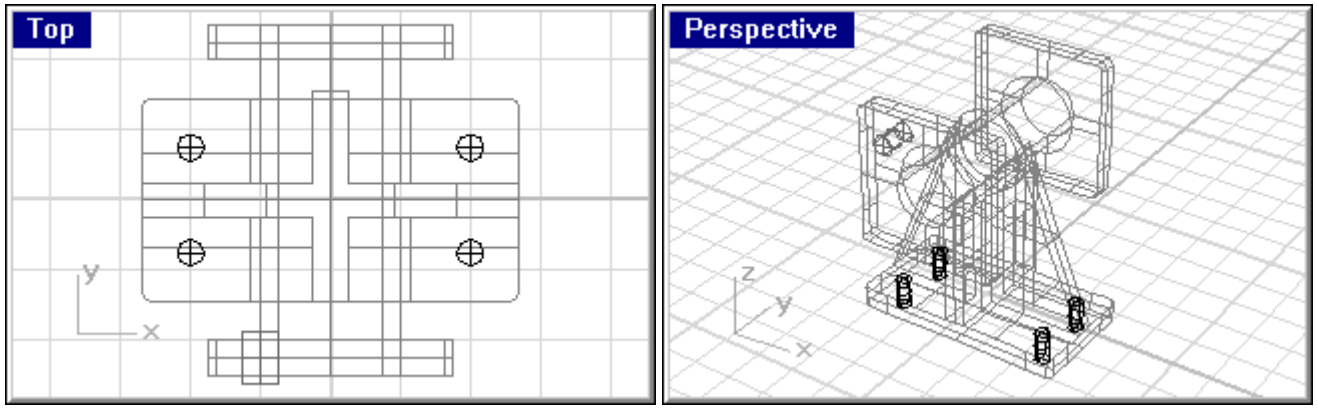
3. From the **Transform** menu, click **Array**, then click **Rectangular.**
4. At the **Number in x direction <1>** prompt, type **2** and press ENTER.
5. At the **Number in y direction <1>** prompt, type **2** and press ENTER.
6. At the **Number in z direction <1>** prompt, type **1** and press ENTER.
7. At the **Unit cell or x spacing** prompt, type **4** and press ENTER.

Notes:

Array Rectangular
Look for this button.

8 At the **Y spacing** prompt, type **1.5** and press ENTER.

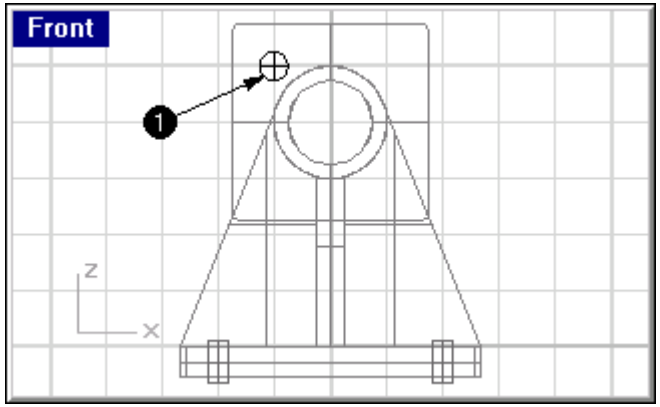
The cylinders are arrayed on the base of the flange.



To create a 3D array:

1 Select the cylinder that is visible in the **Front** viewport.

When entering distances in the Front viewport the construction plane Y direction is up and the Z direction is toward you. The icon that is visible is the World Coordinate Icon.



Notes:

2 From the **Transform** menu, click **Array**, then click **Rectangular.**

3 At the **Number in x direction <2>** prompt, type **2** and press ENTER.

4 At the **Number in y direction <2>** prompt, type **2** and press ENTER.

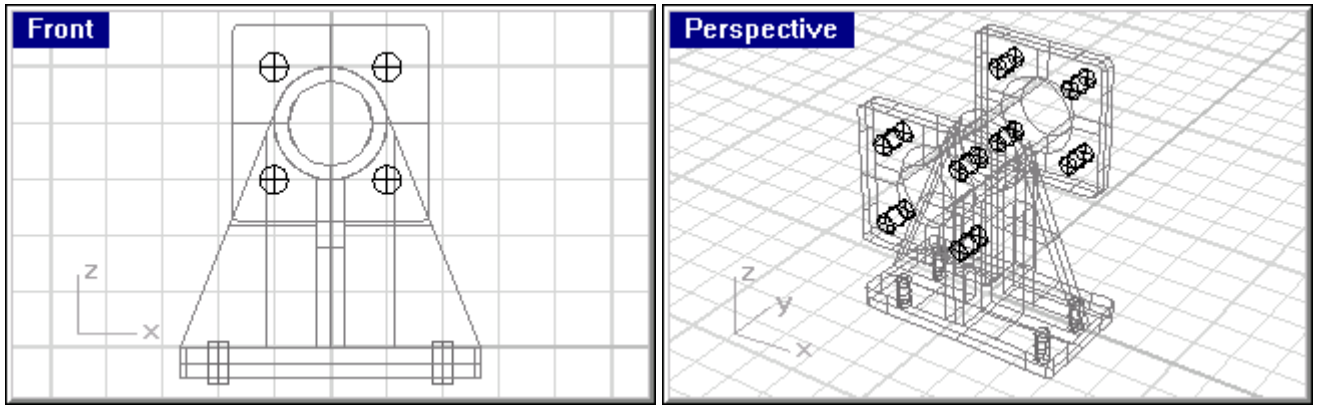
5 At the **Number in z direction <1>** prompt, type **2** and press ENTER.

6 At the **Unit cell or x spacing** prompt, type **2** and press ENTER.

7 At the **Y spacing** prompt, type **-2** and press ENTER.

8 At the **Z spacing** prompt, type **-4.5** and press ENTER.

The cylinders are arrayed on both flanges.



To remove the holes from the flange:

1 **Save** your model.

2 From the **Solid** menu, click **Difference.**

3 At the **Select surface or polysurface** prompt, pick the flange.

Notes:

Array Rectangular
Look for this button.

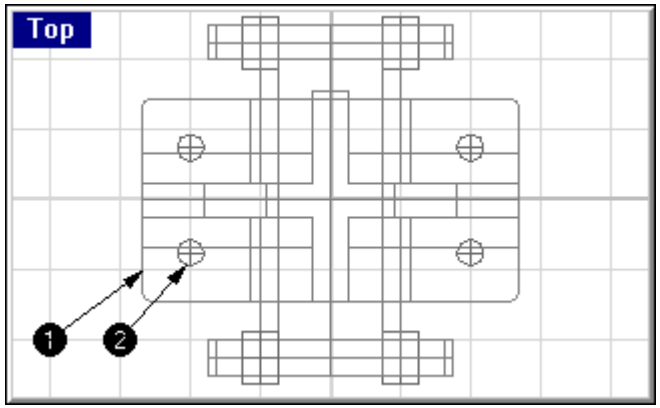
The reason that negative numbers are used for the y and z spacing is because the hole is arrayed in the negative direction on each axis.

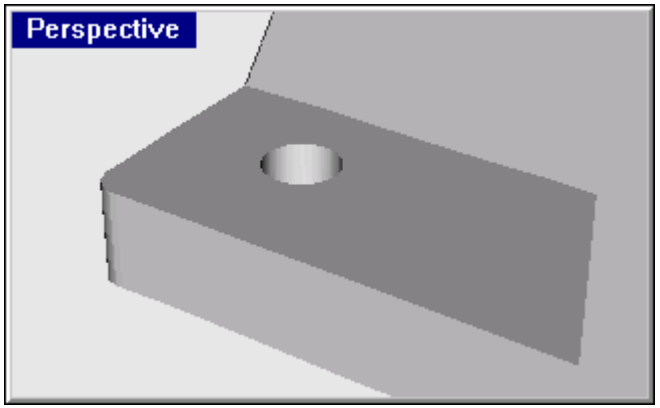
Always save your model before doing any boolean operation.

Boolean Difference
Look for this button.

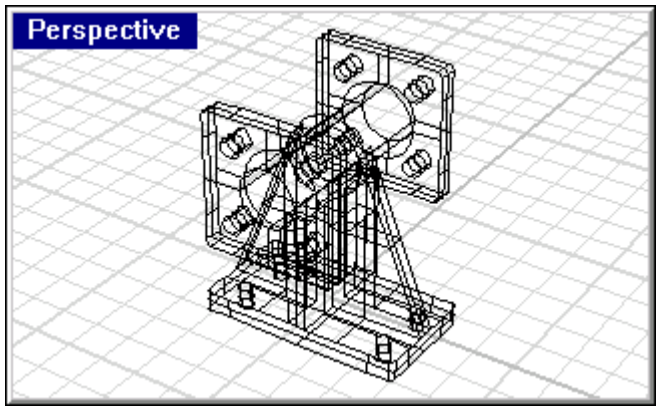
4 At the **Select surface or polysurface to subtract** prompt, pick one of the holes.

The hole is subtracted from the flange.



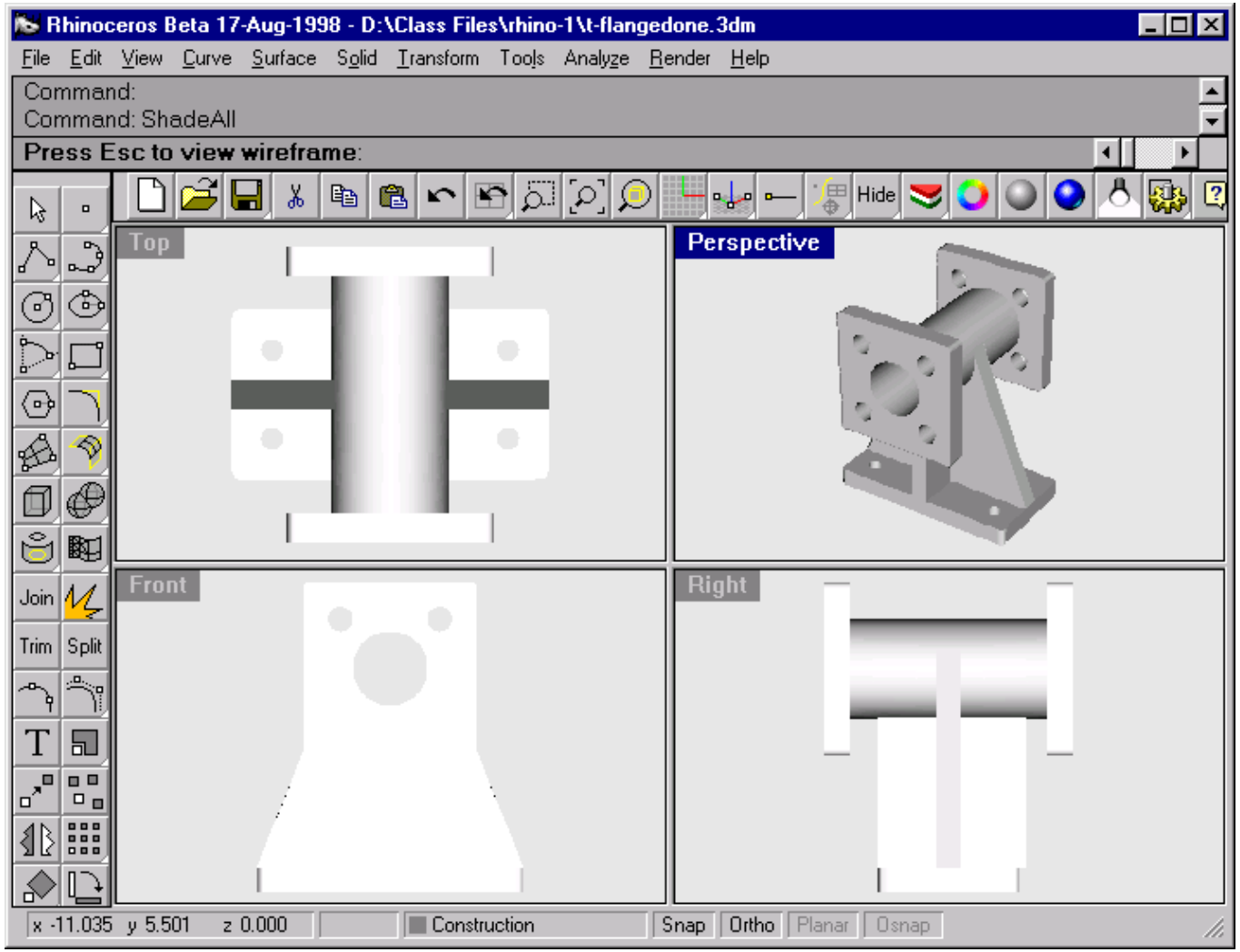


5 Repeat this process for each of the holes.



Notes:

6 From the **Render** menu, click **Shade All Viewports**.

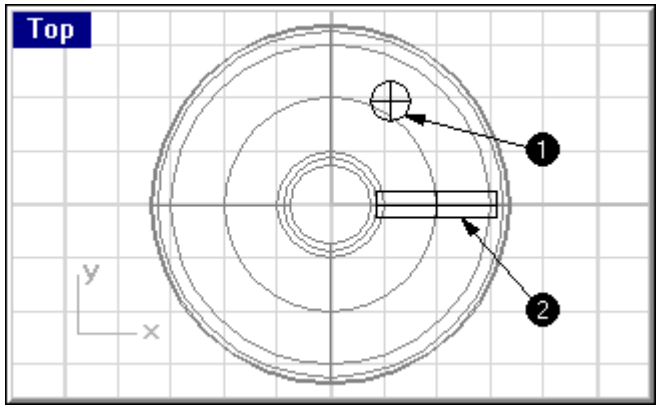


Notes:

Shade All Viewports
Look for this button.
Right click this button.

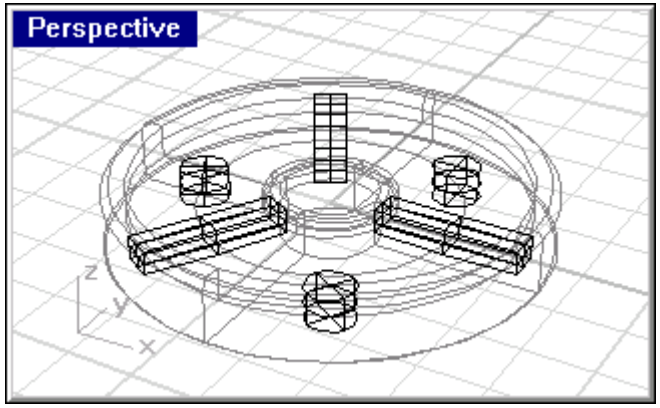
To create a polar array:

1. Open the **Wheel** model.
2. Select the cylinder and the rib in the **Top** viewport.



3. From the **Transform** menu, click **Array**, then click **Polar.**
4. At the **Center of polar array** prompt, snap to the center of the wheel.
5. At the **Number of elements<1>** prompt, type **3** and press ENTER.
6. At the **Angle to fill<360>** prompt, press ENTER.

   The cylinders and ribs are arrayed in a circular pattern around the center of the wheel.



Notes:

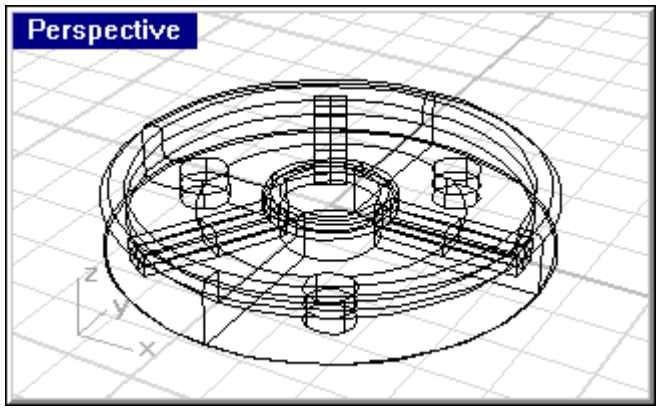
Array Polar
Look for this button.

You must include the original and the copies as the number in the array.

To remove the holes from the wheel:

1. **Save** your model.
2. From the **Solid** menu, click **Difference.**
3. At the **Select surface or polysurface** prompt, pick the wheel.
4. At the **Select surface or polysurface to subtract** prompt, pick one of the holes.

   The hole is subtracted from the flange.
5. Repeat this process for each of the holes.



To add the ribs to the wheel:

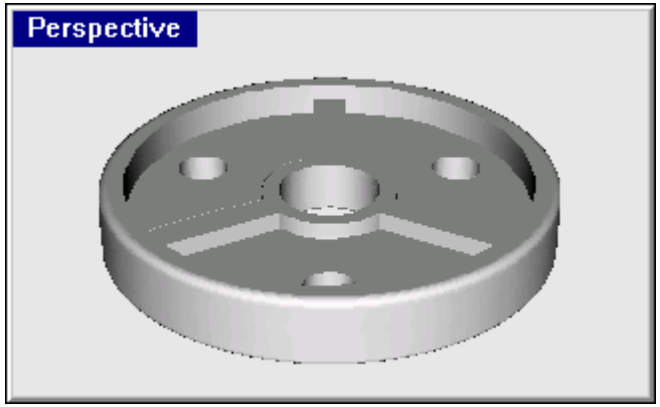
1. From the **Solid** menu, click **Union.**
2. At the **Select surface or polysurface** prompt, pick the wheel.
3. At the **Select surface or polysurface to union** prompt, pick one of the ribs. The rib is added to the wheel.
4. Repeat this process for each of the ribs.

Notes:

Boolean Difference
Look for this button.

Boolean Union
Look for this button.

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



Notes:

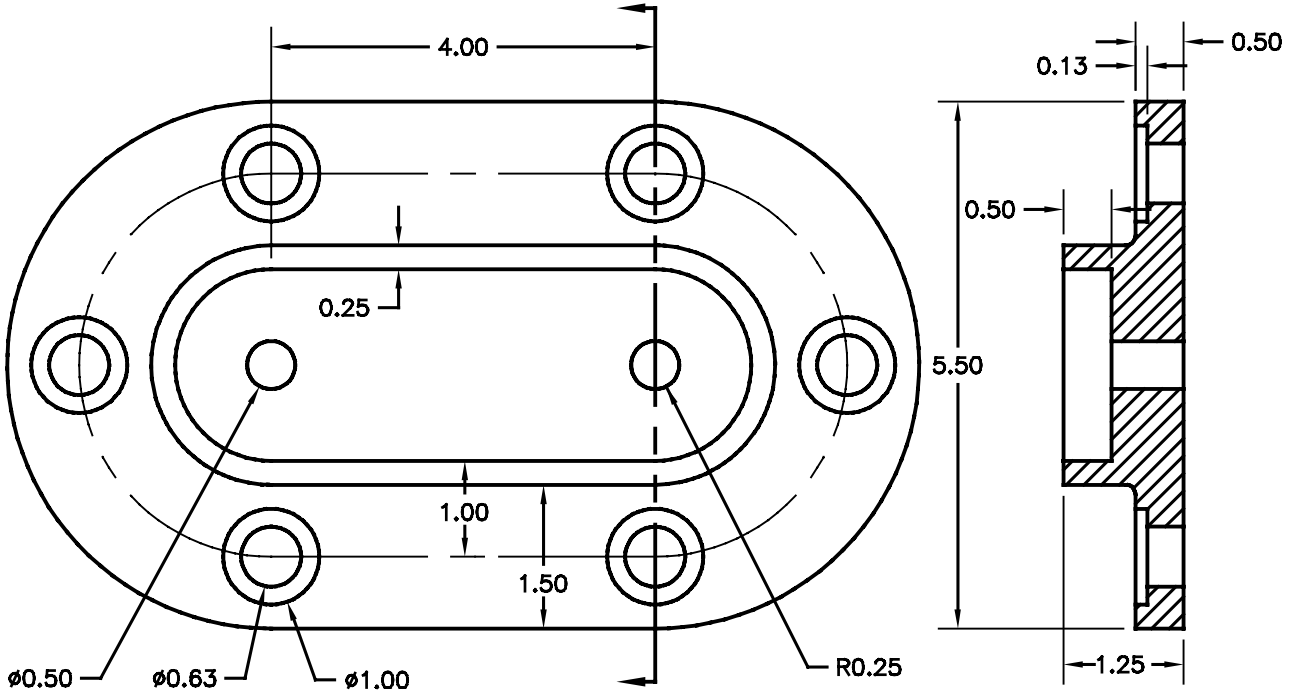
Shade
Look for this button.

Offset

Offset creates an object parallel or concentric to another object. Use Offset to create specialized copies, such as parallel lines, concentric circles, and concentric arcs, through specified points or at pre-set distances.

Exercise 39—Offset

Start a new model. Save as **Shaft Base**.



To create the rounded rectangle:

1. From the **Curve** menu, click **Rectangle**, then click **Center, Corner.**
2. At the **Center of rectangle (Rounded)** prompt, type **R** and press ENTER.
3. At the **Center of rectangle** prompt, move you cursor into the **Top** viewport, type **0,0** and press ENTER.

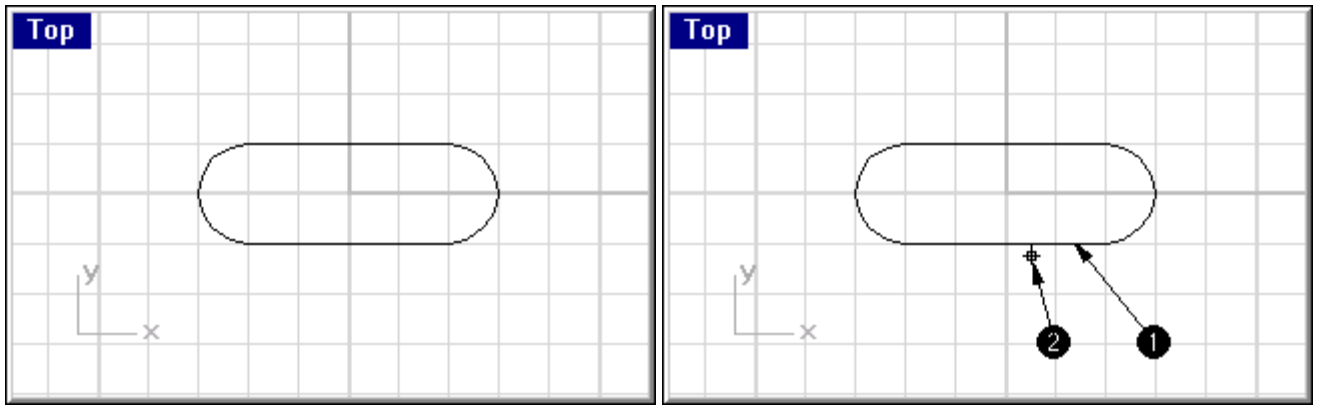
Notes:

Rectangle Centered
Look for this button.

4 At the **Corner or length** prompt, type **6** and press ENTER.

5 At the **Width** prompt, type **2** and press ENTER.

6 At the **Point for rounded corner to pass through (Corner=Arc)** prompt, type **1** and press ENTER.



To create offset rounded rectangles:

1 From the **Curve** menu, click **Offset.**

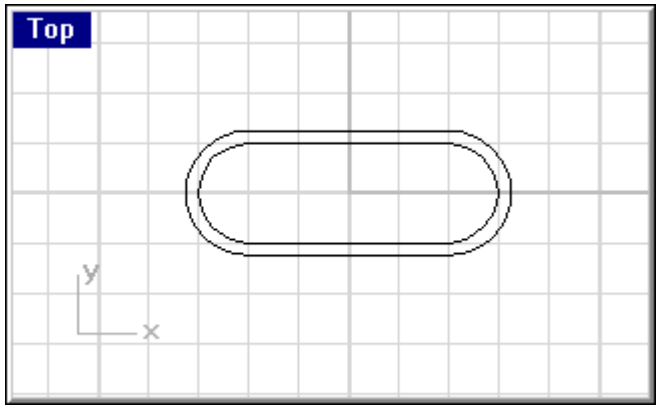
2 At the **Select curve to offset (Distance=1 Corner=sharp ThroughPoint)** prompt, type **.25** and press ENTER.

3 At the **Select curve to offset (Distance=.25 Corner=sharp ThroughPoint)** prompt, pick the rounded rectangle.

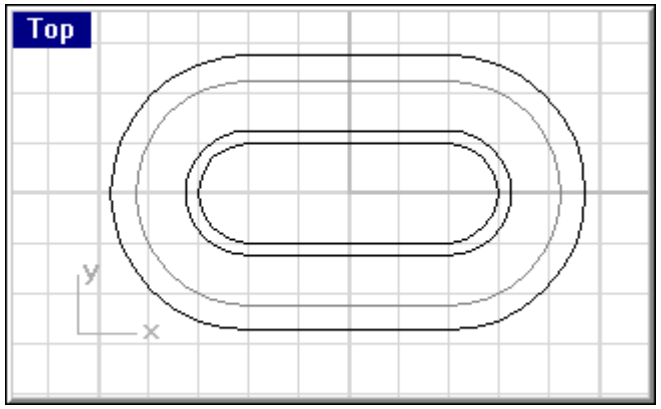
Notes:

Offset
Look for this button.

4 At the **Side to offset (Distance=.25 Corner=sharp ThroughPoint)** prompt, pick outside the rounded rectangle. A larger rounded rectangle will be created.



5 Continue to offset the remainder of the curves. Make sure to change the offset distance for each curve.



To draw the holes:

1 Turn on **End** Osnap.

2 From the **Curve** menu, click **Circle**, then click **Center, Radius.**

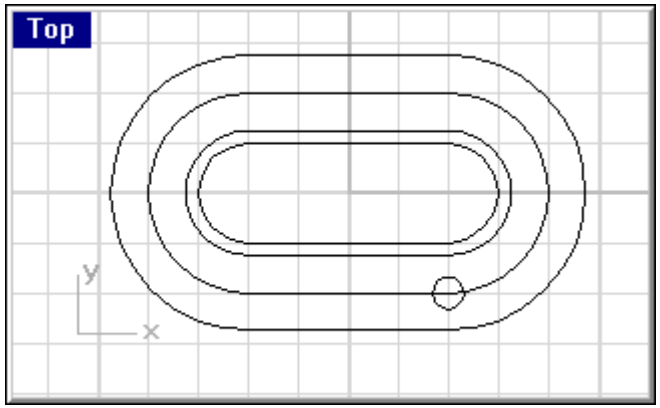
3 At the **Center of circle (Vertical AroundCurve)** prompt, snap to the end point at the bottom right corner of the center line.

Notes:

Circle
Look for this button.

4 At the **Radius <1> (Diameter)** prompt, type **D** and press ENTER.

5 At the **Diameter <2> (Radius)** prompt, type **.63** and press ENTER.

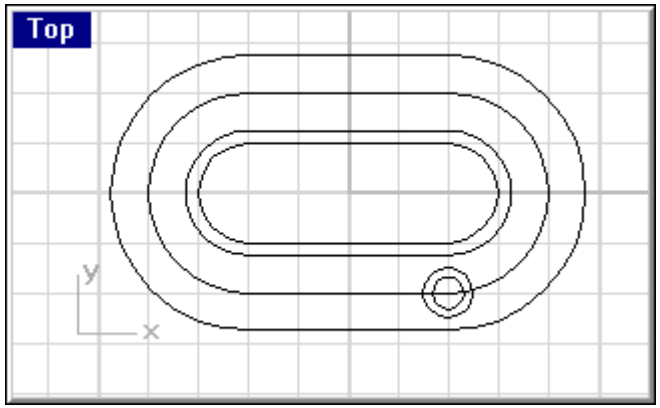


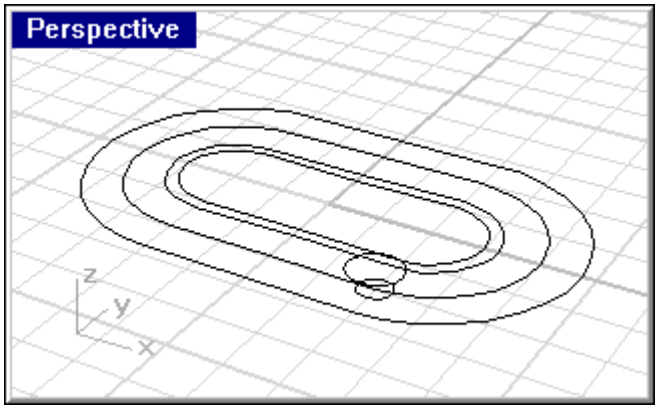
6 From the **Curve** menu, click **Circle**, then click **Center, Radius.**

7 At the **Center of circle (Vertical AroundCurve)** prompt, snap to the same End while holding your CTRL key down to activate elevator mode, then type **.375** and press ENTER.

8 At the **Diameter <0.63> (Radius)** prompt, type **1** and press ENTER.

This moves the center of the circle off the construction plane.



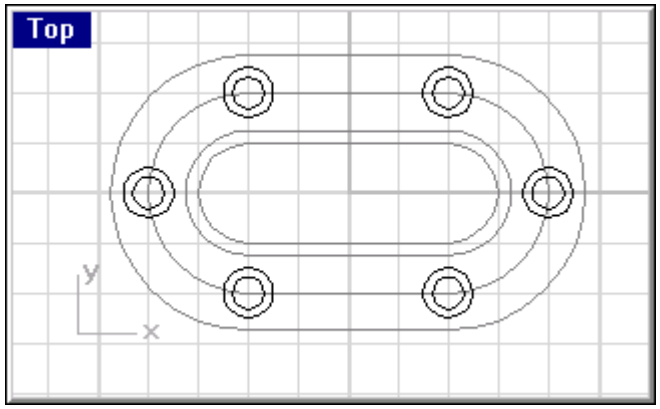


Notes:

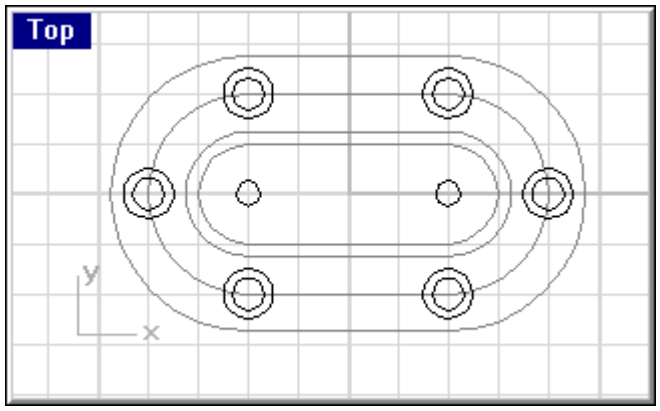
Circle
Look for this button.

To copy the circles:

1. Select the circles you just created.
2. Turn on **End** and **Quad** osnaps.
3. From the **Transform** menu, click **Copy.**
4. At the **Point to copy from (Vertical InPlace)** prompt, pick the end point where the circles are located.
5. At the **Point to copy to** prompt, pick the End point and Quadrant points for the location of the circles as shown in the drawing.



6. Create the smaller holes at both ends of the shaft base.



Notes:

Copy
Look for this button.

To make it 3D:

1. Select the smallest rounded rectangle.
2. From the **Transform** menu, click **Move.**
3. At the **Point to move from (Vertical)** prompt, move your cursor into the **Front** viewport and pick a point.
4. At the **Point to move to** prompt, type **R0,.75** and press ENTER.
5. Select the center line curve and delete it.
6. Select the largest rounded rectangle.
7. From the **Solid** menu, click **Extrude Planar Curve.**
8. At the **Extrusion distance (Direction Cap=Yes BothSides Tapered)** prompt, type **.5** and press ENTER.
9. Select the smallest rounded rectangle.
10. From the **Solid** menu, click **Extrude Planar Curve.**
11. At the **Extrusion distance (Direction Cap=Yes BothSides Tapered)** prompt, type **.75** and press ENTER.
12. Select the largest rounded rectangle.
13. From the **Solid** menu, click **Extrude Planar Curve.**
14. At the **Extrusion distance (Direction Cap=Yes BothSides Tapered)** prompt, type **1.25** and press ENTER.

To add the parts together:

1. **Save** your model.
2. From the **Solid** menu, click **Union.**
3. At the **Select surface or polysurface** prompt, pick the base.

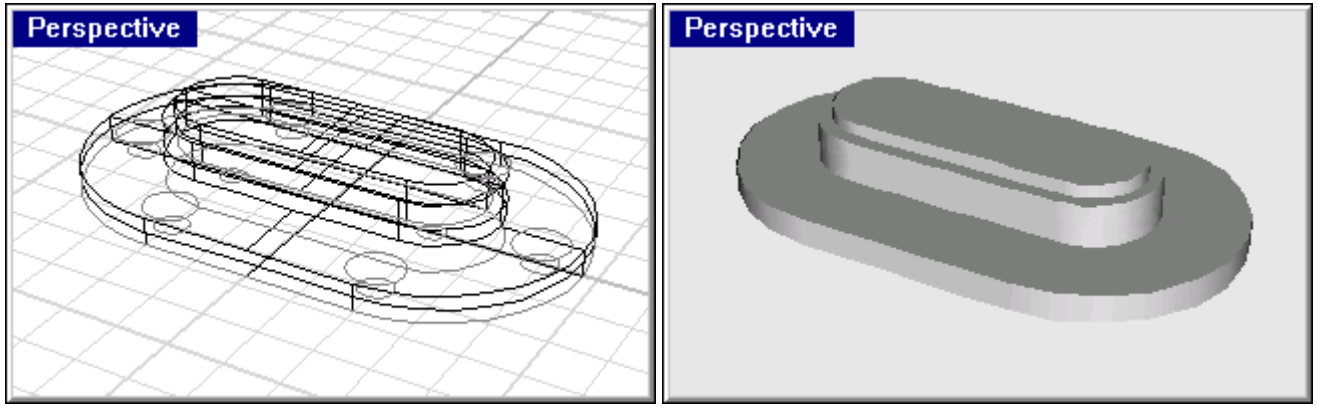
Notes:

Circle
Look for this button.

Extrude Planar Curve
Look for this button.

Boolean Union
Look for this button.

4 At the **Select surface or polysurface to union** prompt, pick the boss.



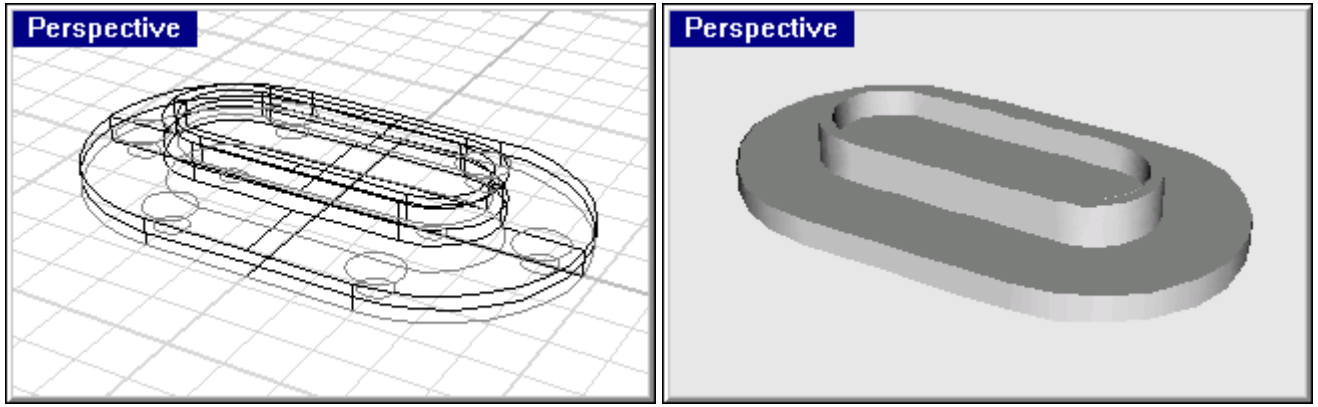
To subtract the inside of the boss from the base:

1 **Save** your model.

2 From the **Solid** menu, click **Difference.**

3 At the **Select surface or polysurface** prompt, pick the base.

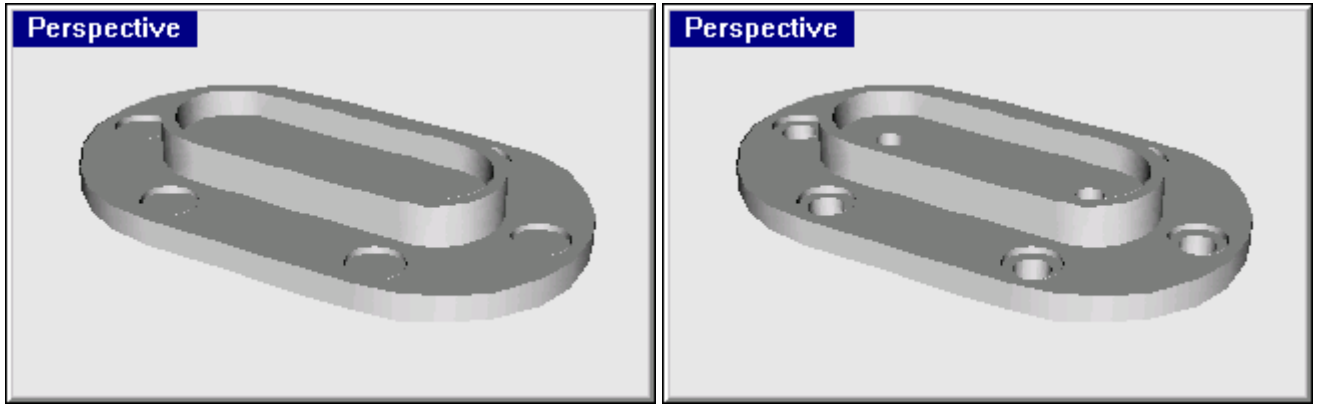
4 At the **Select surface or polysurface to subtract** prompt, pick the inside of the part that sticks up.



Notes:

Boolean Difference
Look for this button.

5 Repeat this process for the holes. First extrude the holes and then subtract them from the base.

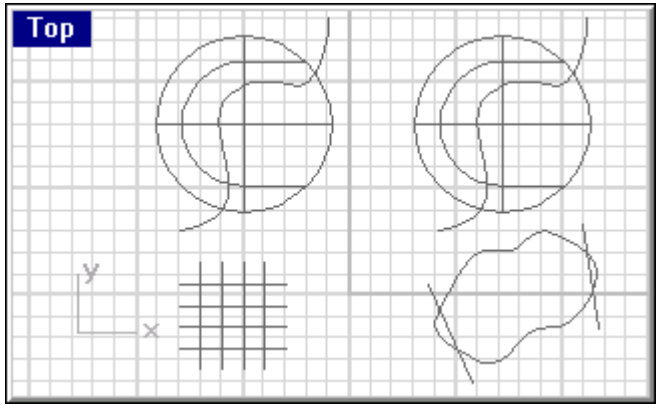


Trim

Trim cuts and deletes portions of an object to make it end precisely at its intersection with another object.

Exercise 40—Trim

1 Open the **Trim-Split** model.

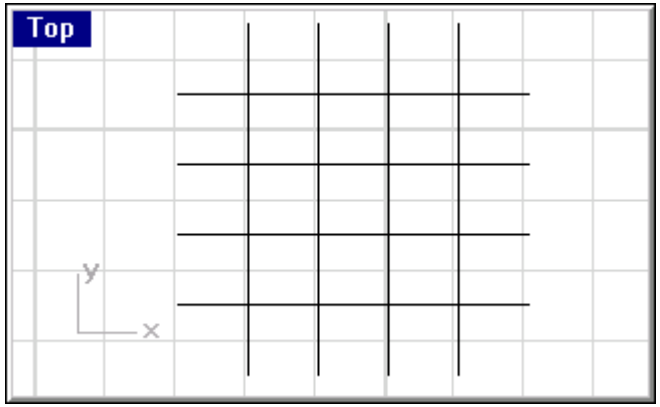


2 From the **View** menu, click **Zoom**, then click **Window.**

Notes:

Zoom Window
Look for this button.

3 Make a window around the grid in the lower left corner of the **Top** viewport.

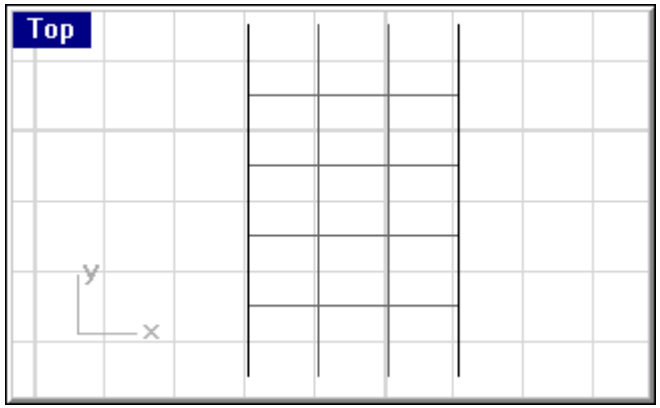


4 From the **Edit** menu, click **Trim.**

5 At the **Select cutting edge (Options)** prompt, select the two outside vertical lines in the grid.

6 At the **Select cutting edge (Options)** prompt, press ENTER.

7 At the **Select object to trim (Options)** prompt, select each of the horizontal lines at their left and right ends. The lines will be trimmed to the cutting edges.



8 At the **Select object to trim (Options)** prompt, press ENTER to end the command**.**

9 From the **View** menu, click **Zoom**, then click **Extents All.**

Notes:

Trim
Look for this button.

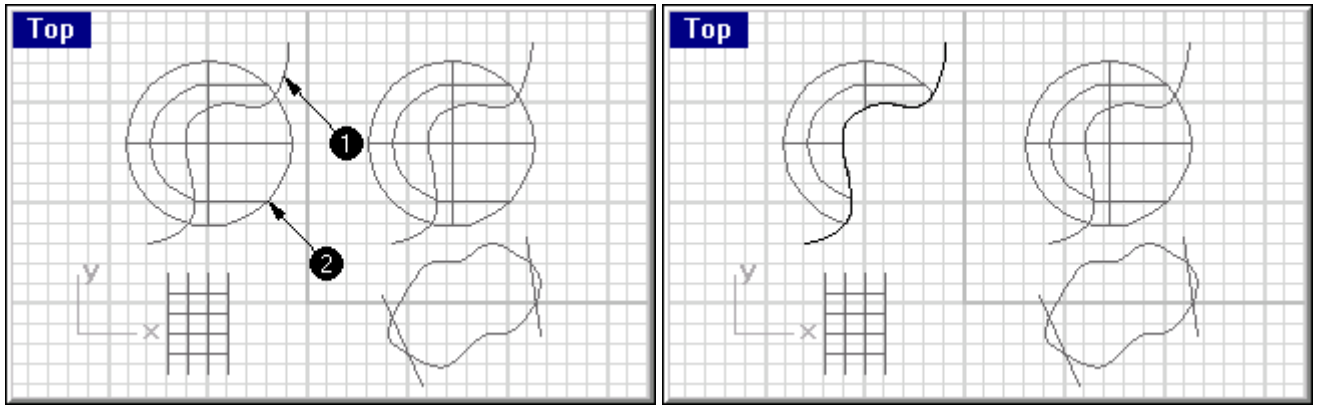
Zoom Extents All
Look for this button.

10 From the **Edit** menu, click **Trim.**

11 At the **Select cutting edge (Options)** prompt, select the surface that intersects the sphere in the **Perspective** viewport.

12 At the **Select cutting edge (Options)** prompt, press ENTER.

13 At the **Select object to trim (Options)** prompt, select the right side of the sphere. The sphere is trimmed at the edge of the surface.



14 At the **Choose object to trim** prompt, press ENTER to end the command**.**

Split

Splits one object with another, or a curve at a point. Split breaks the object where it intersects of the cutting object, but does not delete anything.

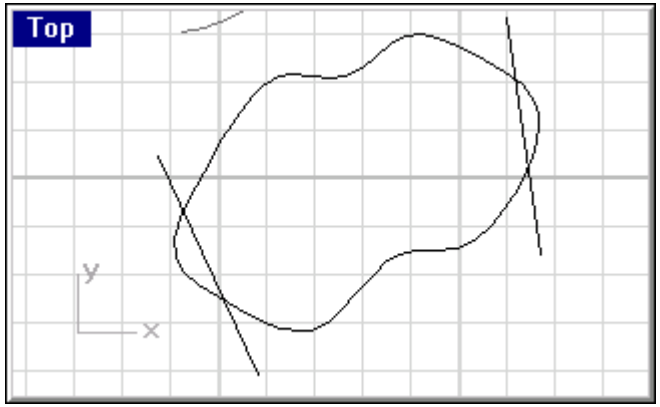
Exercise 41—Split

1 From the **View** menu, click **Zoom**, then click **Window.**

Notes:

Trim
Look for this button.

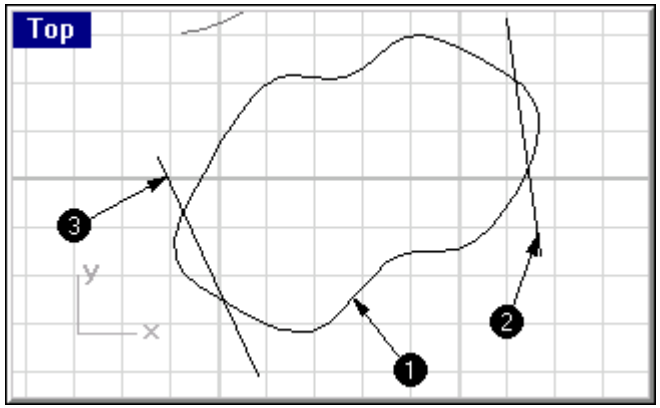
2 Make a window around the closed curve in the lower right corner of the **Top** viewport.



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

4 At the **Select object to split** prompt, choose the closed curve.

5 At the **Select cutting objects (Point)** prompt, choose the lines.

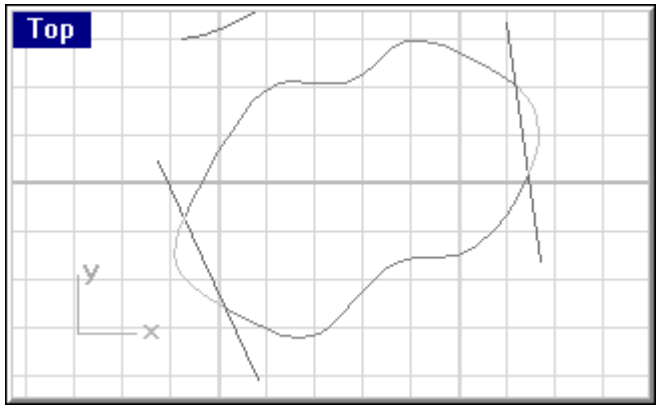


Notes:

Split
Look for this button.

4 At the **Select cutting objects (Point)** prompt, press ENTER.

The curve is separated into four pieces precisely where the lines intersect it.



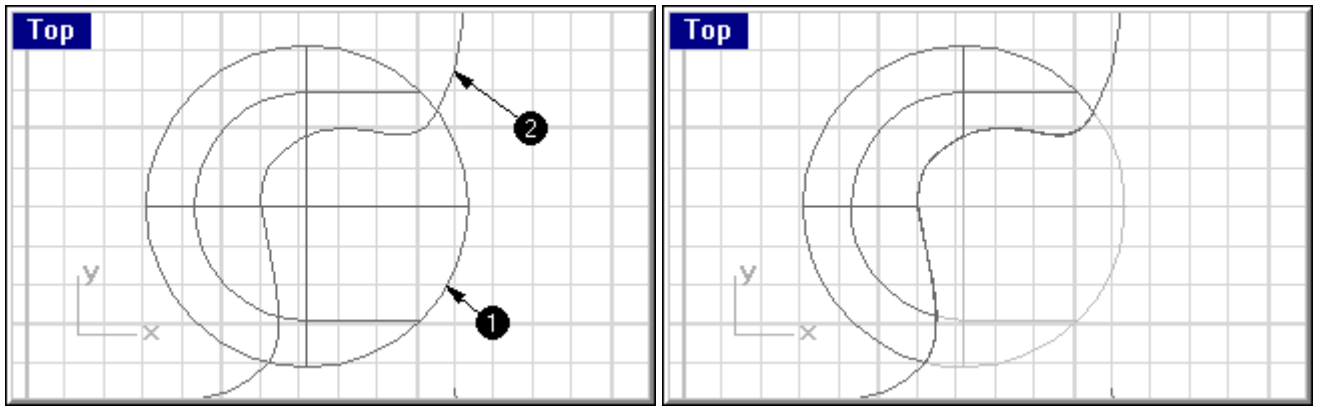
6 From the **View** menu, click **Zoom**, then click **Extents All.**

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

8 At the **Select object to split** prompt, choose the second sphere.

9 At the **Select cutting objects (Point)** prompt, choose the surface that intersects the sphere.

The Sphere is separated into two pieces precisely where the surface intersects it.



Notes:

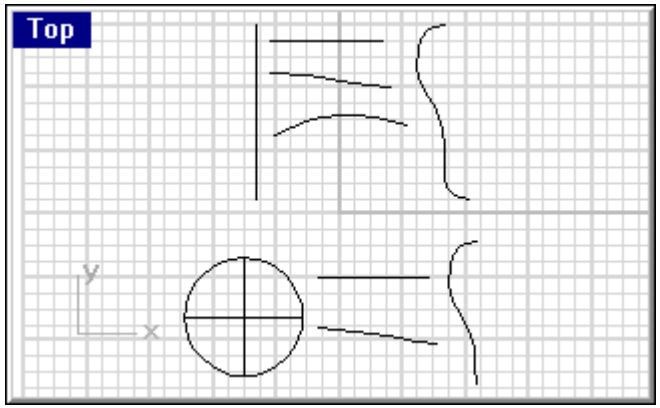
Split
Look for this button.

Extend

Extend lengthens an object to make it end precisely at its intersection with another object or you can lengthen an object when there is no intersection.

Exercise 42—Extend

1. Open the **Extend** model



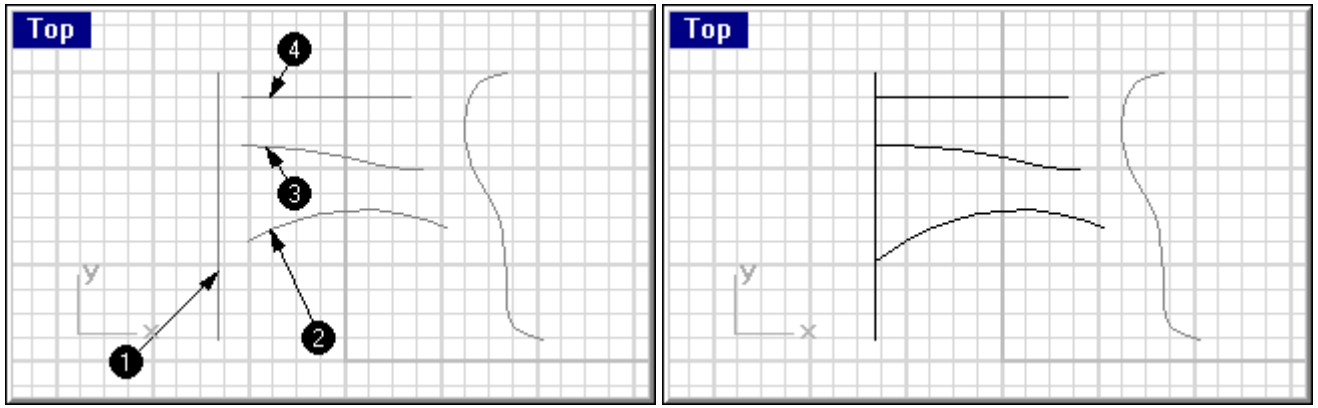
2. From the **Curve** menu, click **Extend**, then click **Extend Curve.**
3. At the **Select boundary objects** prompt, pick the line at the left.
4. At the **Select boundary objects** prompt, press ENTER.

Notes:

Extend
Look for this button.

5 At the **Select object to extend (Type=Line)** prompt, choose the left ends of the three curves.

The line and curves will be extended to touch the boundary edge on the left.



6 At the **Select object to extend (Type=Line)** prompt, press ENTER to end the command.

To extend with some of the other options:

1 From the **Curve** menu, click **Extend**, then click **Extend Curve.**

2 At the **Select boundary objects** prompt, pick the curve at the right.

3 At the **Select boundary objects** prompt, press ENTER.

4 At the **Select object to extend (Type=Line)** prompt, pick the right end of the line.

The line is extended straight.

5 At the **Select object to extend (Type=Line)** prompt, type **A** and press ENTER.

6 At the **Select object to extend (Type=Arc)** prompt, pick the right end of the curve.

The curve continues with a tangent arc to the boundary edge.

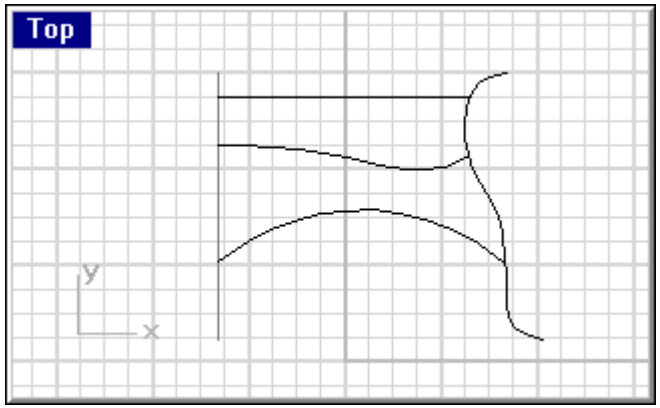
7 At the **Select object to extend (Type=Arc)** prompt, type **S** and press enter.

Notes:

Extend
Look for this button.

8 At the **Select object to extend (Type=Smooth)** prompt, pick the right end of the arc.

The arc continues with a tangent curve.



To extend to a surface:

1 From the **Curve** menu, click **Extend**, then click **Extend Curve.**

2 At the **Select boundary objects** prompt, choose the cylinder on the left.

3 At the **Select boundary edges** prompt, choose the surface on the right.

4 At the **Select boundary edges** prompt, press ENTER.

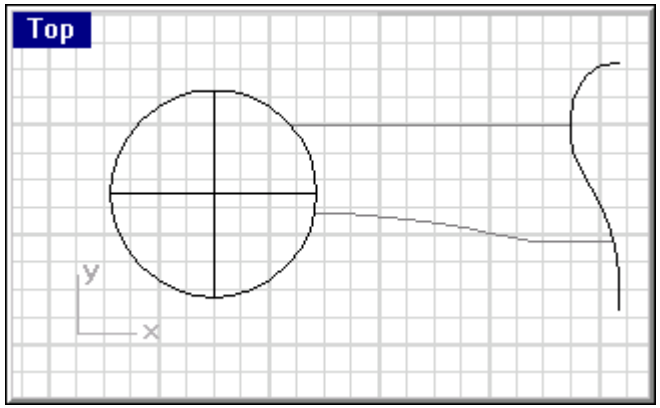
5 At the **Select object to extend (Type=Smooth)** prompt, type **L** and press ENTER.

Notes:

Extend
Look for this button.

6 At the **Select object to extend (Type=Line)** prompt, choose both ends of the line and the curve.

The curves will extend to the surface of the sphere and to the surface.



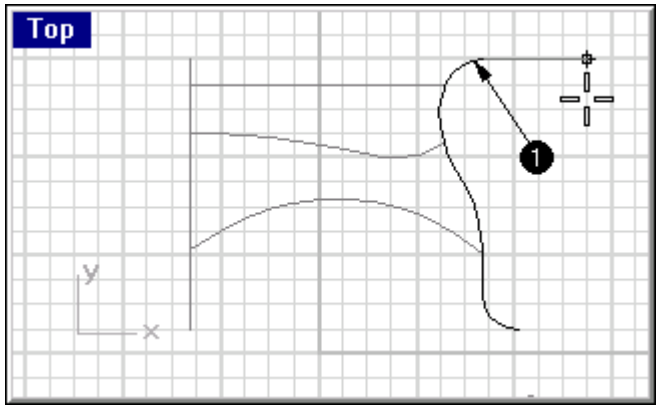
To extend a curve without boundary edges:

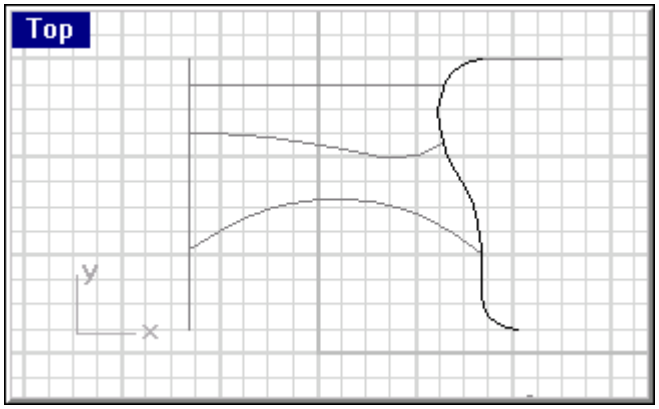
1 From the **Curve** menu, click **Extend**, then click **By Line.**

2 At the **Select curve to extend** prompt, choose near the curve in the upper right side.

3 At the **End of line or length** prompt, pick a point.

The curve is extended as a tangent line to the point you picked.





Notes:

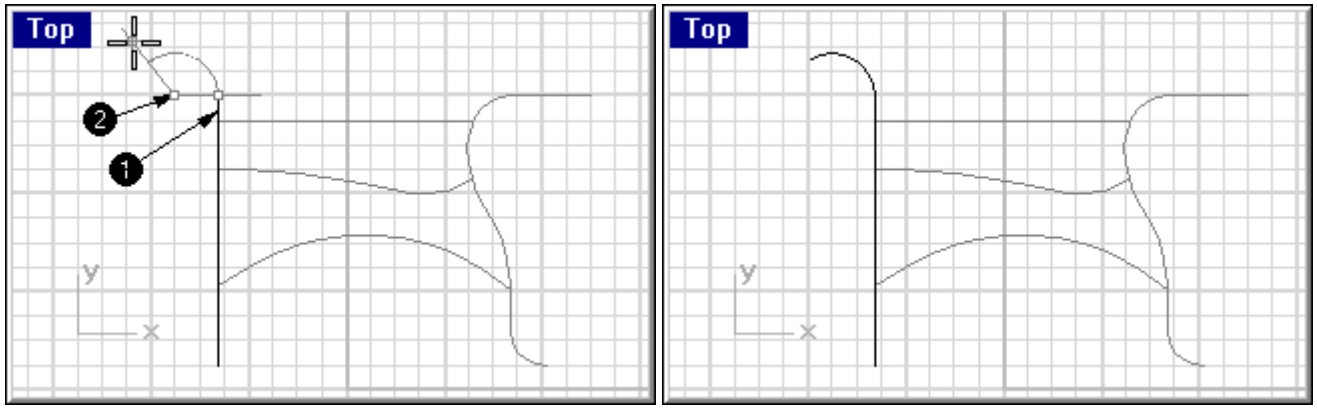
Extend by Line
Look for this button.

4 From the **Curve** menu, click **Extend**, then click **By Arc.**

5 At the **Select curve to extend** prompt, select near the upper end of the line on the left.

6 At the **Radius of arc** prompt, pick a point to set a radius, or type a value for the radius.

7 At the **End point or angle** prompt, pick an end for the arc, or type an included angle for the arc.



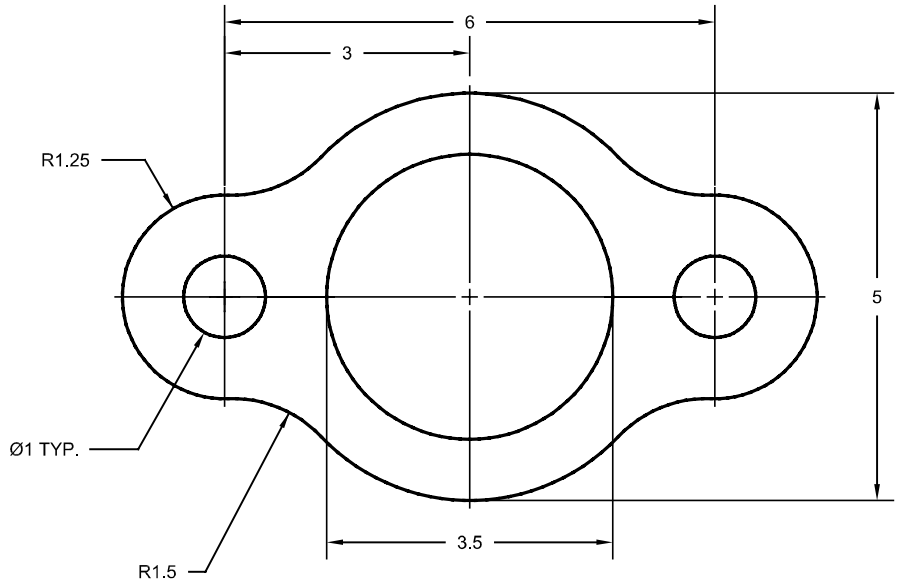
Notes:

Extend by Arc
Look for this button.

Exercise 43—Practice

1 Start a new model. Save as **Gasket1**

2 Use **Circle**, **Arc, Trim**, **Fillet**, and **Join** to create the part shown.



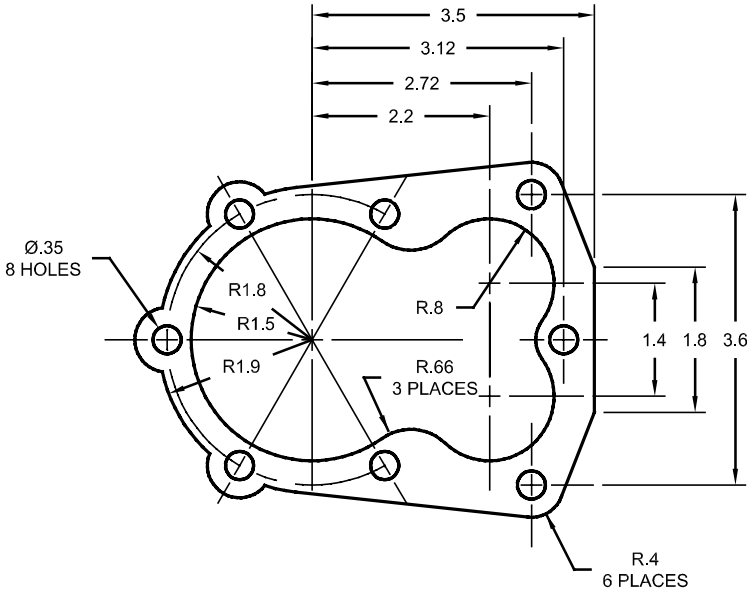
3 Use **Extrude** to create the 3d part. The extrusion thickness is **.125**.

Notes:

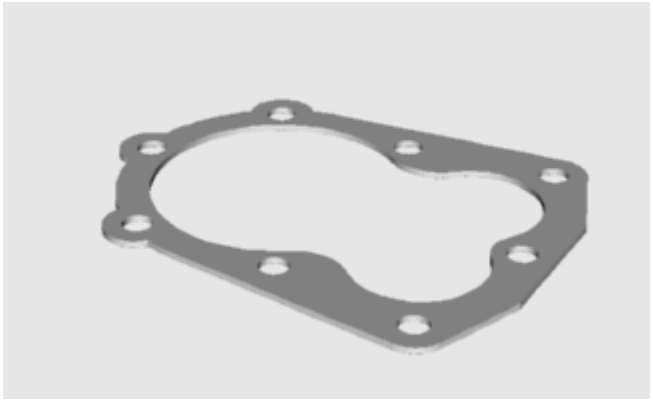
Exercise 44—Practice

1 Start a new model. Save as **Gasket2**.

2 Use **Circle, Arc, Line, Trim, Offset, Fillet, Copy,** and **Mirror** to draw the part shown.



3 Use **Extrude** to create the 3d part. The extrusion thickness is **.125**.

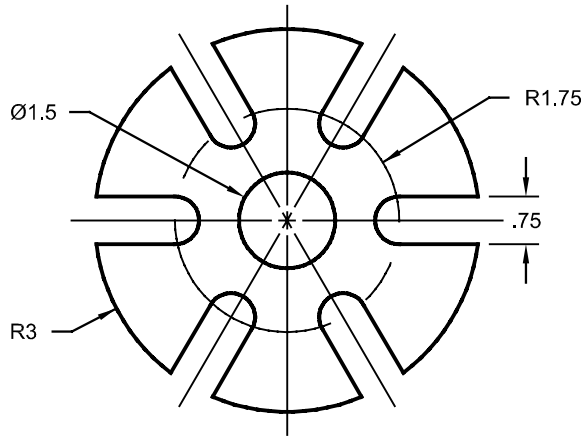


Notes:

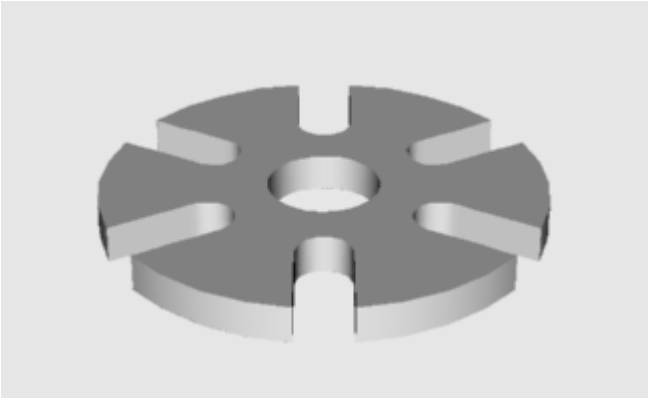
Exercise 45—Practice

1 Start a new model. Save as **Cam**.

2 Use **Circle**, **Arc**, **Line**, **Trim**, **Join**, and **Array** to draw the part shown.



3 Use **Extrude** to create the 3d part. The extrusion thickness is **.5**.

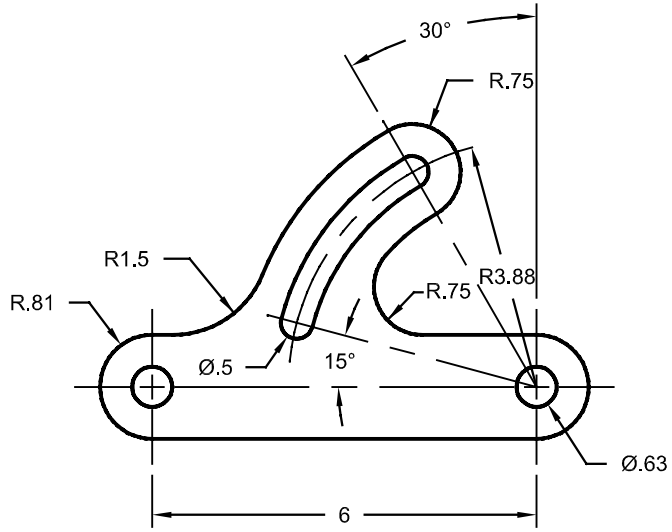


Notes:

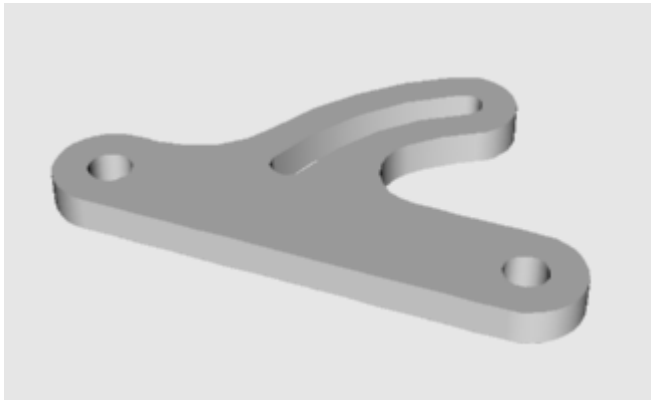
Exercise 46—Practice

1 Start a new model. Save as **Link**.

2 Use **Line**, **Arc**, **Trim**, **Offset**, **Join**, **Fillet** and **Circle** to draw the part shown.



3 Use **Extrude** and **Subtract** to create the 3d part. The extrusion thickness is **.5**.



Notes:

7 Point Editing

Point editing

You can display the control points or the edit points of an object so that you can adjust the shape of an object rather than manipulating the whole object at once. That is called “point editing.”

You can use point editing on meshes, curves, and surfaces, but not on polysurfaces or solids.

NURBS Curves

Rhino's curves are represented internally by Non-Uniform Rational B-Splines (NURBS). The shape of a NURBS curve is determined by three things:

- A list of points called "control points"
- Degree
- A list of numbers called "knots"

If you change any of these things, it changes the shape of the curve.

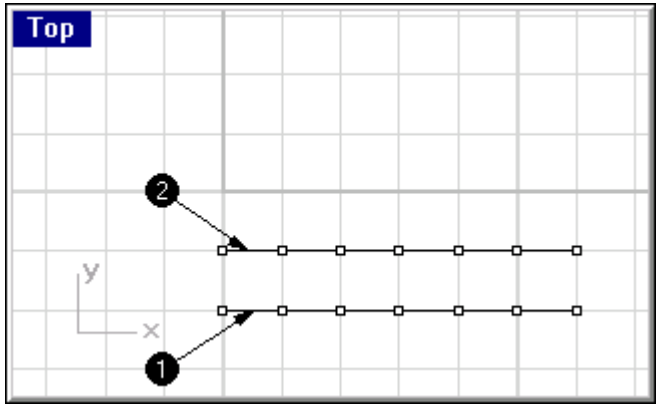
Exercise 47—Control Point Editing

In this exercise we are going to experiment with moving control points. Understanding how curves and lines react when control points are moved is very important to understanding NURBS modeling.

Notes:

1. Start a new model. Save as **Points**.
2. Create **Polyline**, **Curve** and **Surface** layers.
3. Turn **Ortho** and **Snap** on.
4. Change to the **Polyline** layer.
5. In the **Top** viewport draw a **polyline** 6 units long with a vertex at each unit along the line.

   The points disappear when the command terminates. The illustration below shows the line with the points on.
6. Change to the **Curve** layer.
7. In the **Top** viewport draw a **curve by control** points 6 units long. Make sure that you pick at each snap point, so that the curve will have a control point at each unit.

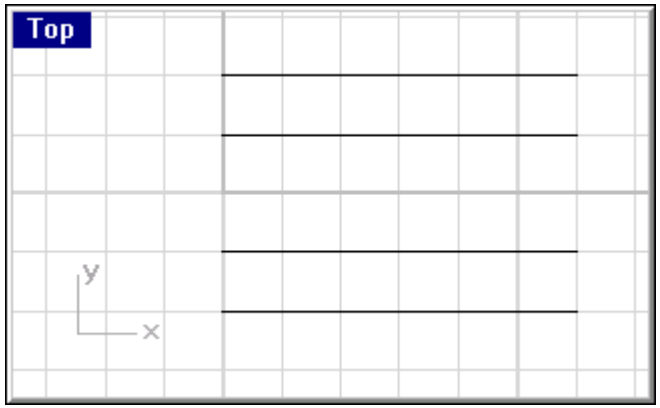


Notes:

Polyline
Look for this button.

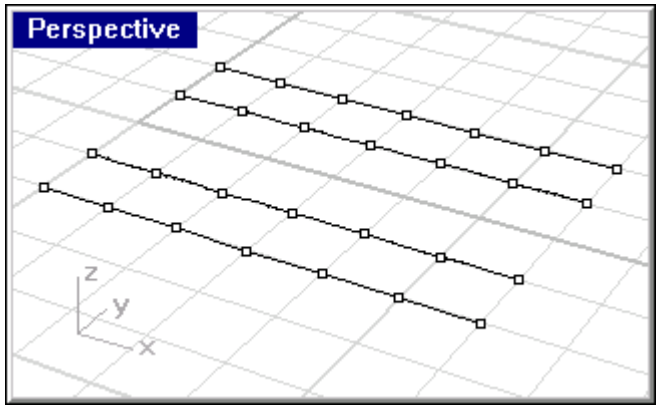
Curve
Look for this button.

8 Use **mirror** to create two more lines, one **curve** and one **polyline**. Mirror the curves starting at 0,0 in the **Top** viewport. Toggle ortho on to pick the second point. Make sure the polylines are on the top and the bottom.



9 From the **Edit** menu, click **Select**, then click **All**, then click **Curves.**

10 From the **Edit** menu, click **Point Editing**, then click **Control Points On.**



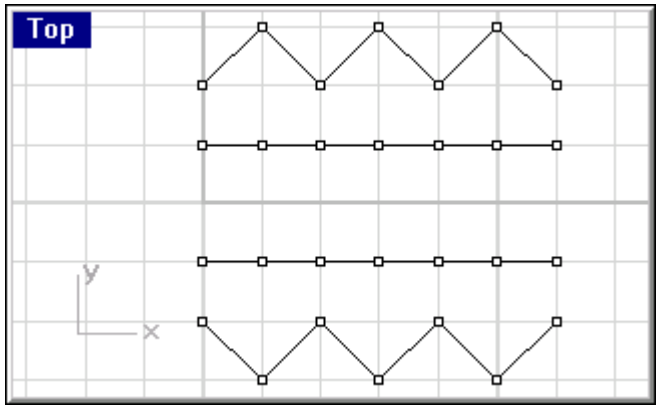
Notes:

Mirror
Look for this button.

Select All Curves
Look for this button.

PtOn
Look for this button.

11 In the **Top** view, click and drag some points on the polylines. Notice that the line has a pointed edge at each control point and that the control points are exactly on the line. When a curve or polyline bends at a point like this it is called a kink. If you create a surface from a line that has a kink it will have a seam at the kink.



12 Push ESC twice to turn off the control points.

13 From the **Edit** menu, click **Layers**, then click **Edit**.

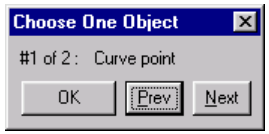
14 **Lock** the **Polyline** layer.

15 Select one of the curves and turn the control points on.

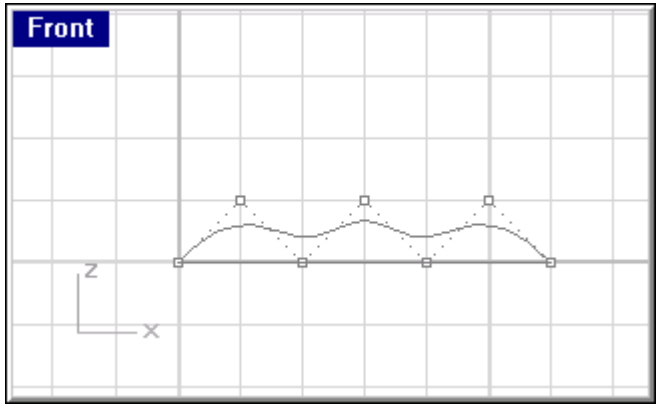
Notes:

Layer
Look for this button.

16 Next, we will elevate the points off the Top Cplane. In the **Front** viewport click and drag some of the control points. If you the **Choose One Object** dialog appears make sure it displays **Curve point** and pick **OK**.

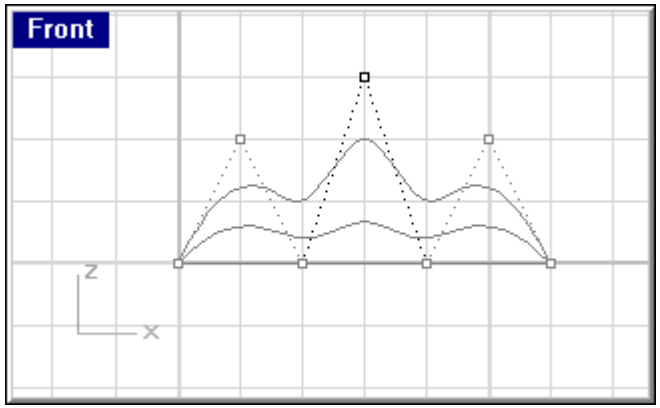


Note that as you move the point, the curve deforms at the point, but stays the same everywhere else.



Notes:

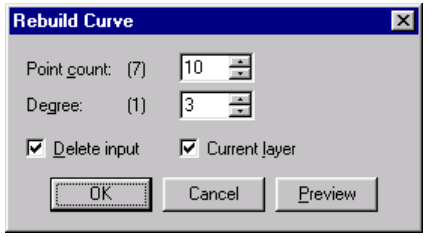
17 Repeat this process for the second curve. Change the location of some of the points, so that it looks different from the previous curve.



To change the polylines into curves without kinks:

1. From the **Edit** menu, click **Layers**, then click **Edit**.
2. Turn on the **Polyline** layer.
3. Select both polylines in the **Top** viewport.
4. From the **Curve** menu, click **Edit Tools**, then click **Rebuild**.

   The Rebuild Curve dialog will appear.



Notes:

Layer
Look for this button.

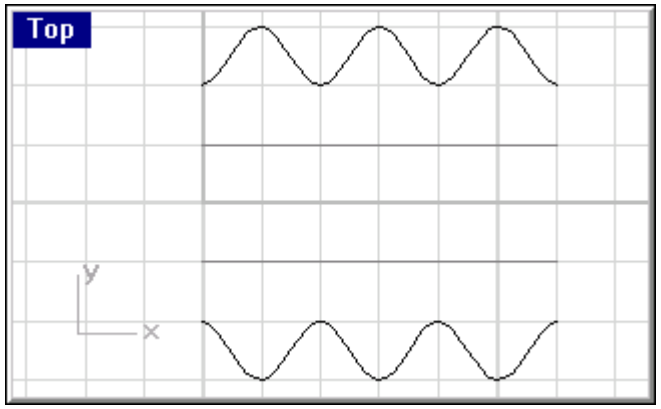
Rebuild
Look for this button.

5 In the **Rebuild Curve** dialog change the point count to **9** and the degree to **3** and press **Preview**.

A degree 3 curve cannot have kinks.

6 Change the point count and degree, and press **Preview**. Notice how the curve will approximate the geometry at different degrees and point counts.

7 When you are satisfied with the preview, click **OK**.



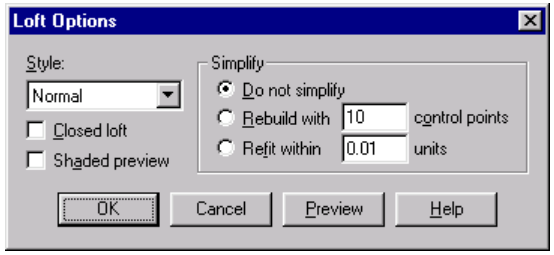
To loft a surface over the curves:

1 Change to the **Surfaces** layer.

2 Select all of the curves.

3 From the **Surface** menu, click **Loft.**

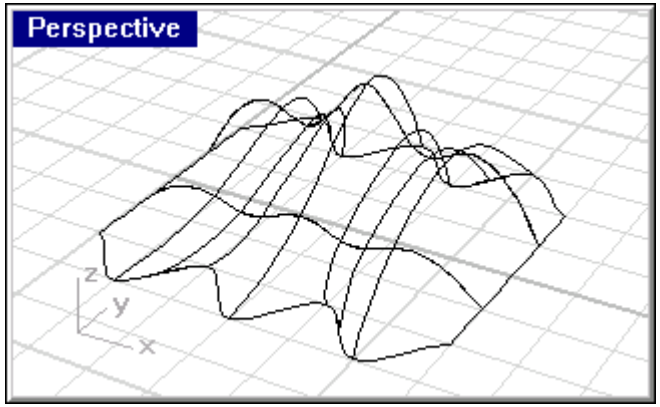
4 In the **Loft Options** dialog, click **OK**.



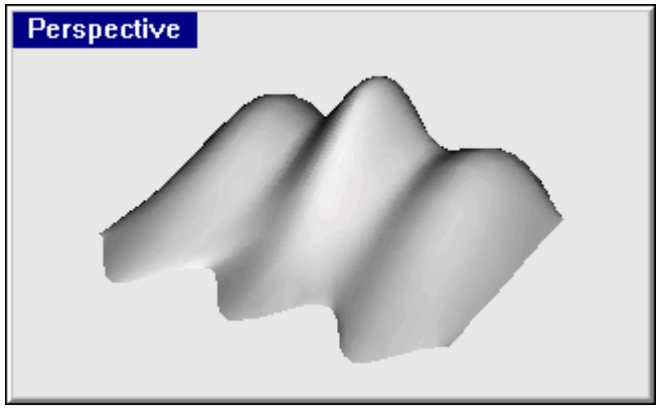
Notes:

Loft
Look for this button.

A surface appears over the curves.



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



6 Turn off the **Curve** and **Polyline** layers.

7 Select the lofted surface.

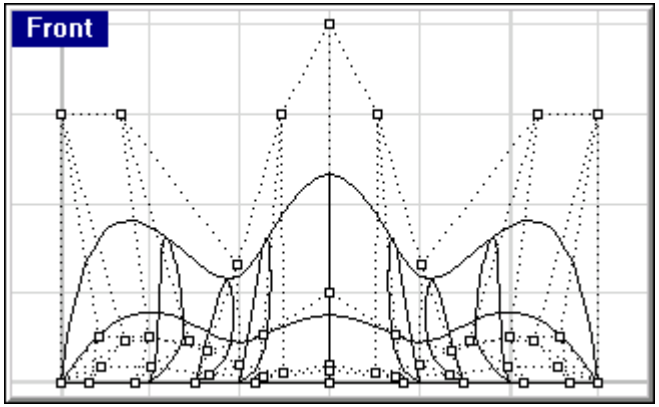
8 From the **Edit** menu, click **Point Editing**, then click **Control Points On.**

Notes:

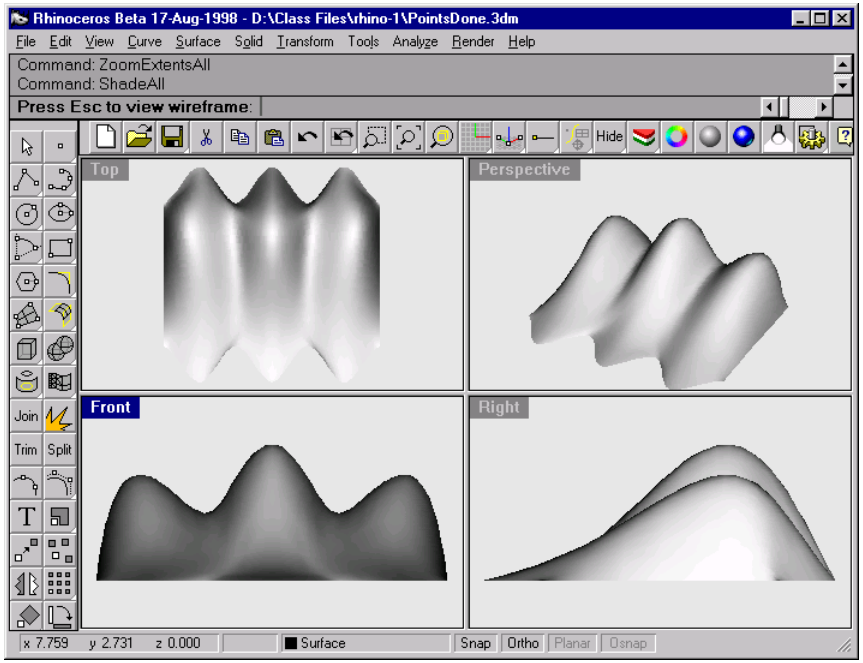
Shade
Look for this button.

Points On
Look for this button.

9 Move some of the control points of the surface to see what effect it has on a surface.



10 From the **Render** menu, click **Shade All Viewports.**



Notes:

Shade All Viewports
Look for this button.
Right click this button.

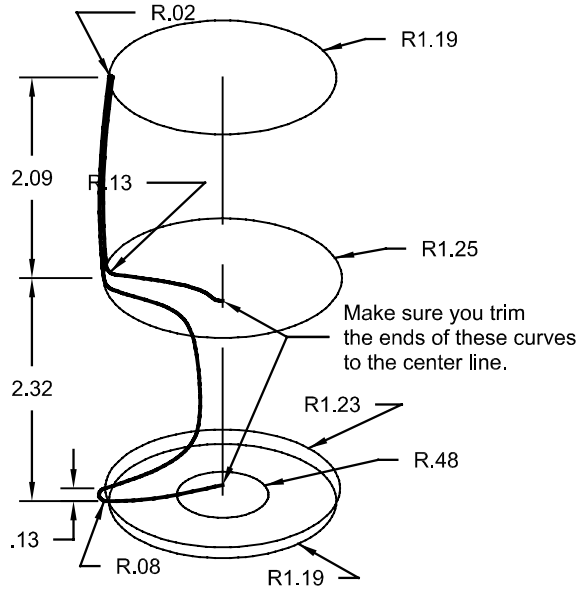
A few facts about control points, edit points and knots

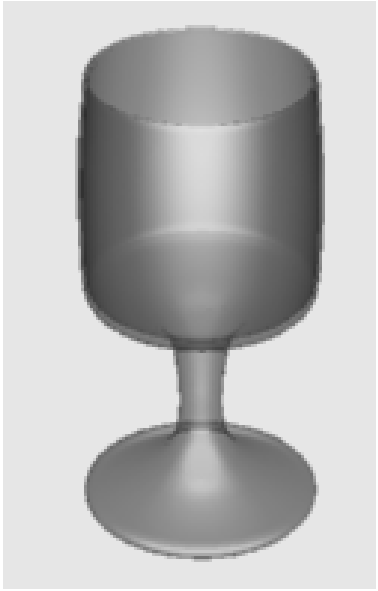
- Knots are parameters (that is, numbers, not points). You can evaluate a NURBS curve or surface at a knot and get a "knot point."
- Control points do not have to be on the curve.
- Edit points and knot points are always on the curve.
- A curve has the same number of control points as edit points.
- Generally, there are more control points than knot points.
- On a polyline, control points = edit points = knot points.
- If a curve is not a polyline, then control points are different from edit points and knot points.
- Rhino lets you edit curves and surfaces by moving control points and edit points.
- Rhino lets you add and remove knots, but you cannot explicitly move knot points.

Notes:

Exercise 48—Practice with Freeform Curves and Control Point editing

1 Start a new model. Save as **Glass**.

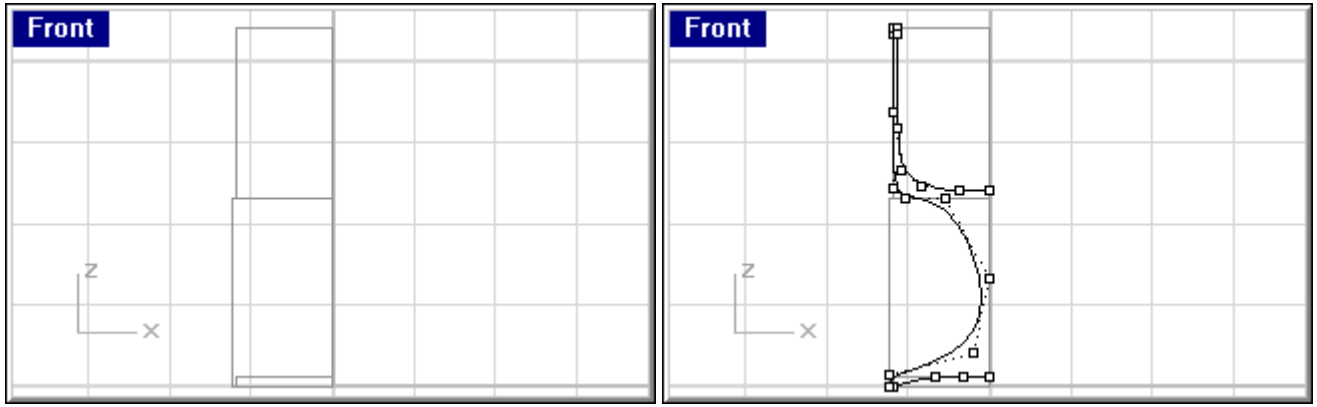




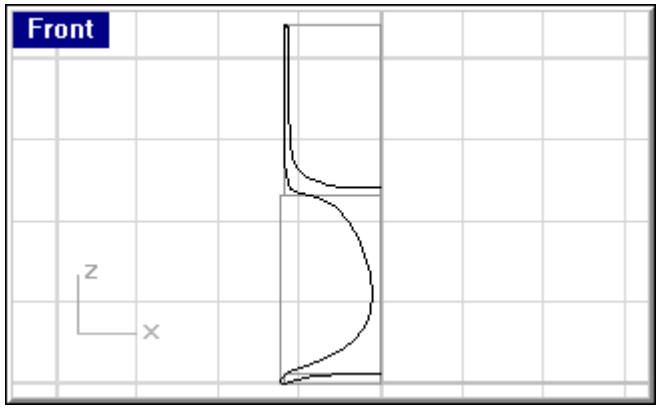
2 Use **Rectangle** and **Curve** to draw the part shown in the **Front** viewport.

Notes:

3 Start by drawing construction lines to help you create an accurate curve.



4 Adjust control points if necessary to achieve the design you want for your glass.



To make it 3D:

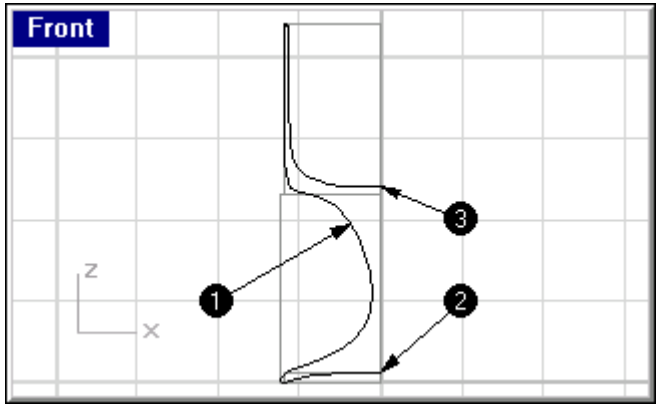
1 From the **Surface** menu, click **Revolve.**

2 At the **Select curve to revolve** prompt, select the curve you created.

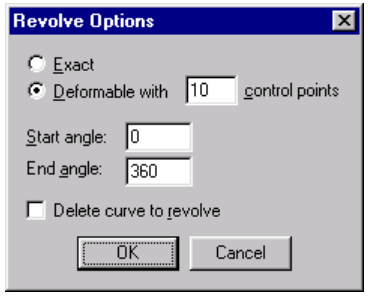
3 At the **Start of revolve axis** prompt, pick one end of the curve.

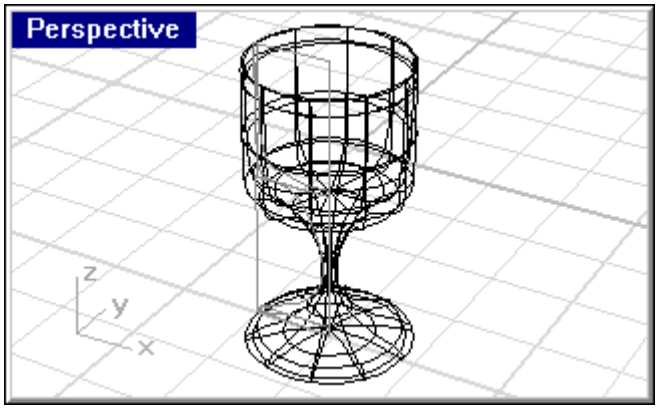
Notes:

4 At the **End of revolve axis** prompt, pick the other end of the curve.



5 In the Revolve Options dialog, click Deformable, then click OK.





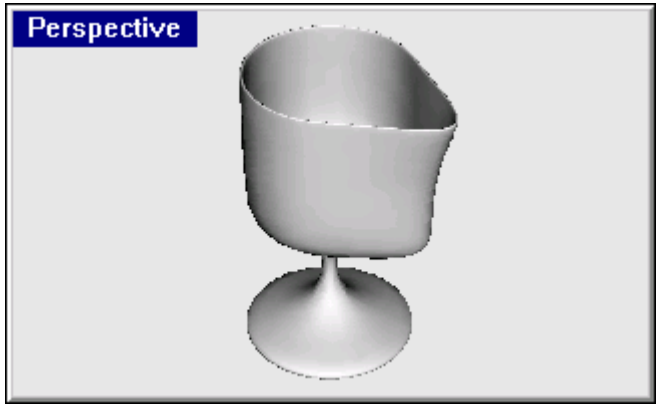
This will make it so that the glass will have control points that can be adjusted.

Notes:

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

7 **Save** your model.

8 Experiment with adjusting the control points to see what happens.



Notes:

Part Three:
3D Modeling and Editing

8 Creating Deformable Shapes

Freeform Modeling

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—freeform 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, freeform shapes. This tutorial focuses only on the freeform, squishy aspect. The exact size and placement of the objects is not critical. The overall form is the main objective.

This exercise shows

- simple surface creation
- surface rebuilding
- control point editing
- curve creation (drawing, projecting)
- splitting surfaces with curves and surfaces
- blending between two surfaces
- lighting and rendering

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.

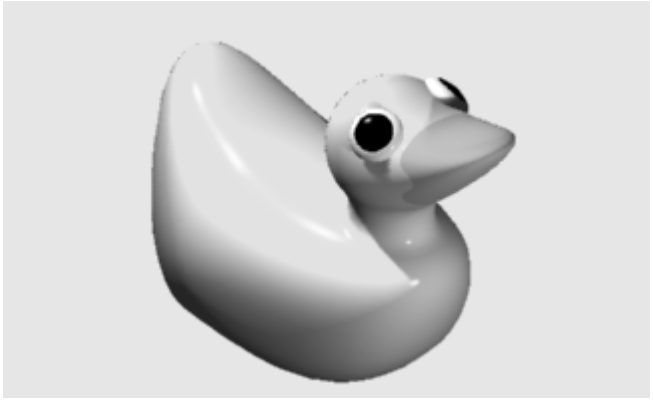
Notes:

If you need to know more about control points and surfaces search, the *Rhino Help* index for **Control points**.

Exercise 49—Creating a rubber ducky.

1. Start a new model. Save as **Ducky**.
2. You can use layers to separate your parts, but for this model, it isn't necessary.

   For more information about layers, search the *Rhino Help* index for **layer**.



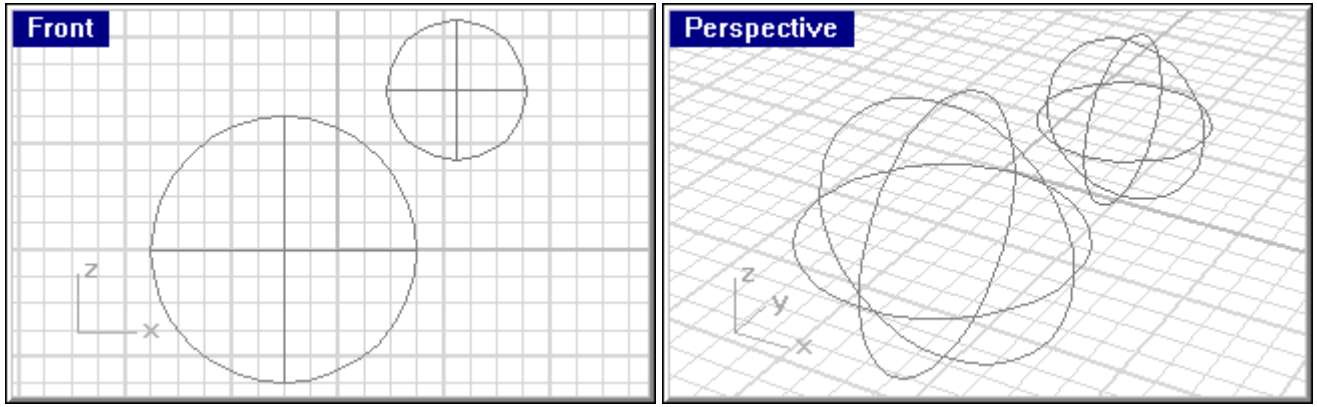
Create the Body and Head Shapes

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.

Notes:

To create the basic shapes:

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



5. Select the large sphere.
6. From the **Edit** menu, click **Point Editing**, then click **Control Points On.**
7. Select a control point and drag it.
8. Click in the **Perspective** view.

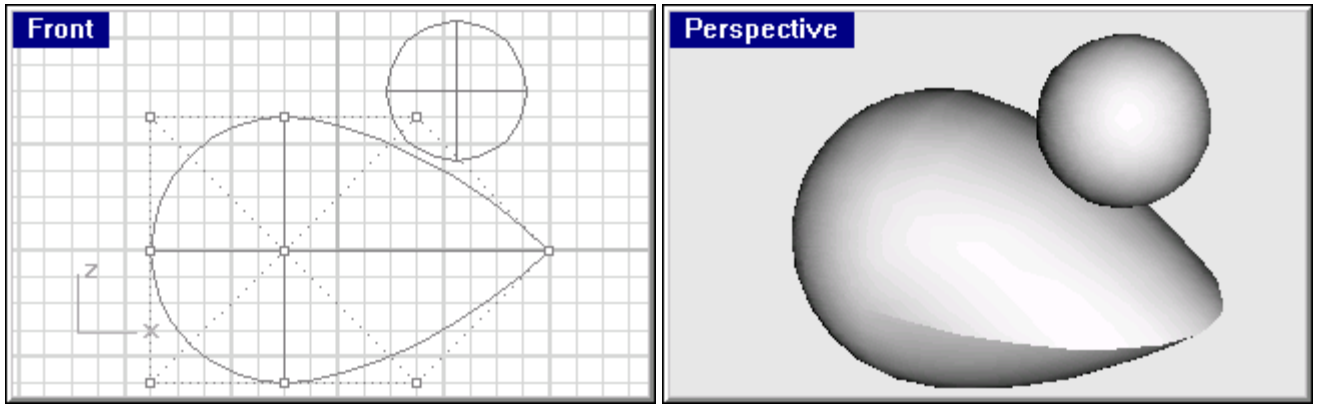
Notes:

Sphere
Look for this button.

Control Points On
Look for this button.

9 From the **Render** menu, click **Shade**.

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



10 Press ENTER or ESC to exit Shade mode. Press ESC again to turn off control points.

11 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.**

Notes:

Shade
Look for this button.

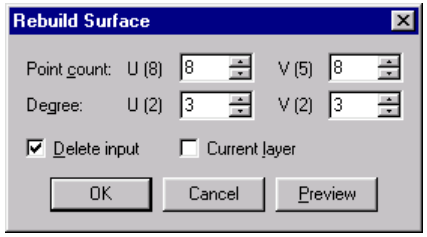
Undo
Look for this button.

Rebuild Surface
Look for this button.

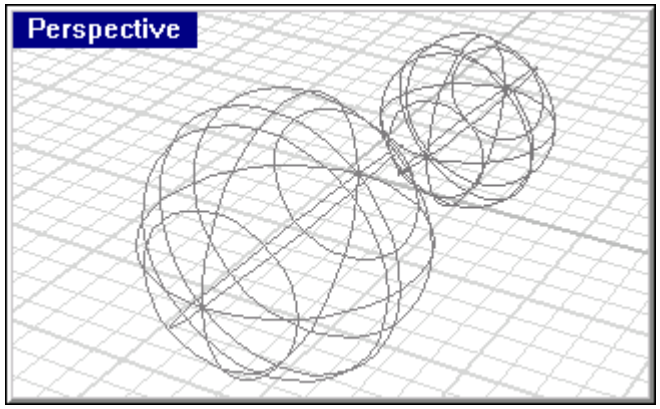
3 In the **Rebuild Surface** dialog, change the **Point Count** to **8** for both U and V.

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

Check **Delete Input**, clear **Current Layer**, and click **OK**.



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.

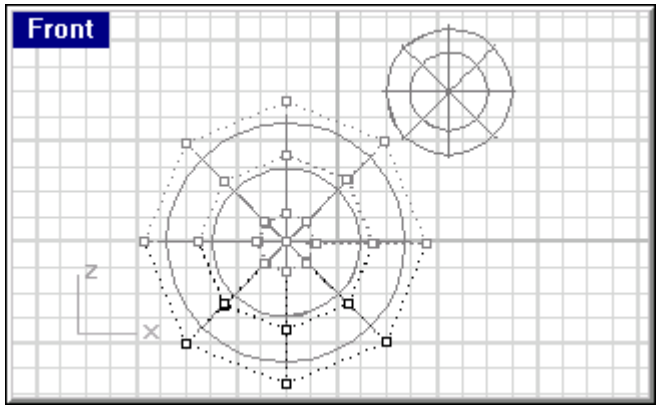
2 From the **Edit** menu, click **Point Editing**, then click **Control Points On.**

Control Points On
Look for this button.

Notes:

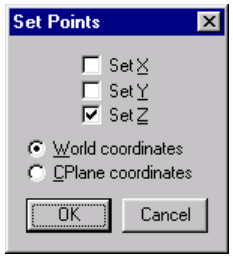
3 In the **Front** view select the control points near the bottom of the sphere.

Window select by dragging a box around the control points.



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

5 Make the Set Points dialog look like this, and click **OK.**

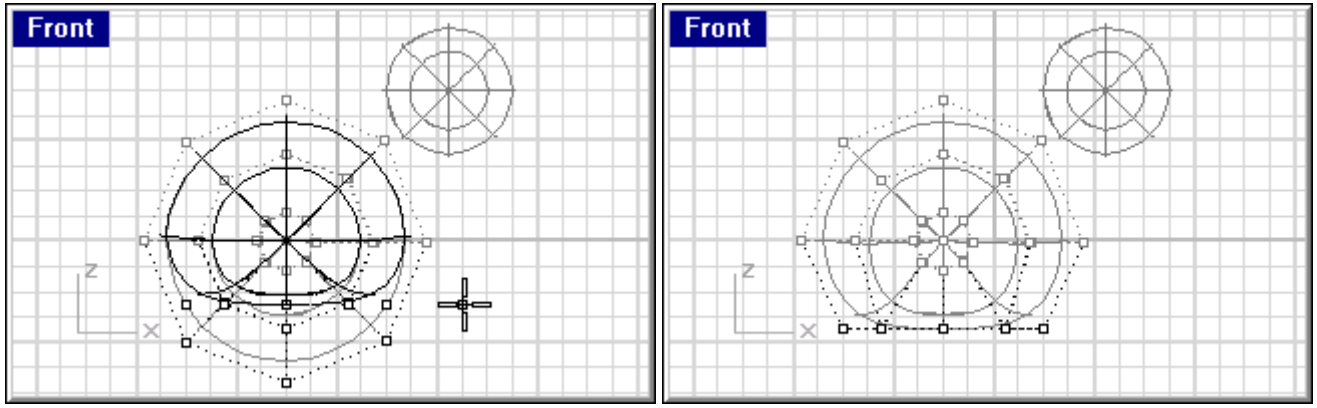


Notes:

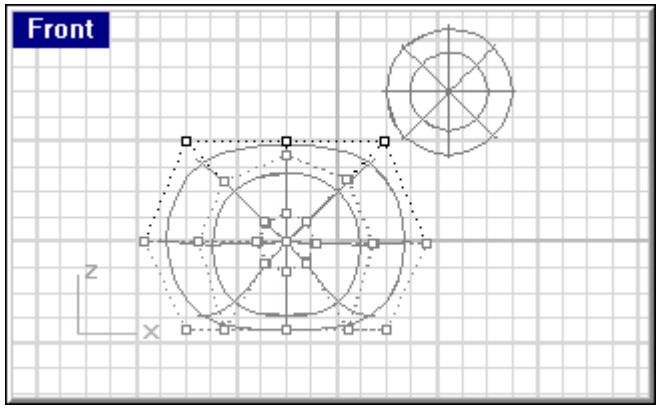
Set Points
Look for this button.

6 Drag the selected control points up.

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



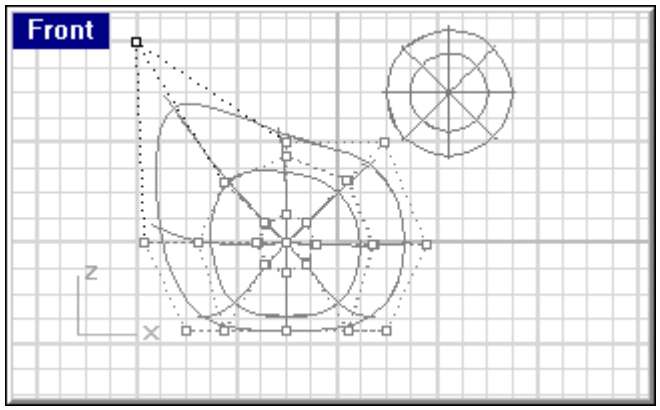
7 Adjust the control points at the top of the body using the same technique.



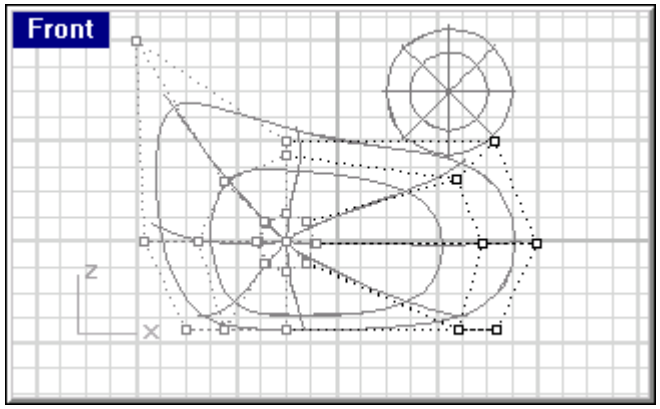
Notes:

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

Notice in the Top view that two control points are selected, though in the Front view, 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 view.

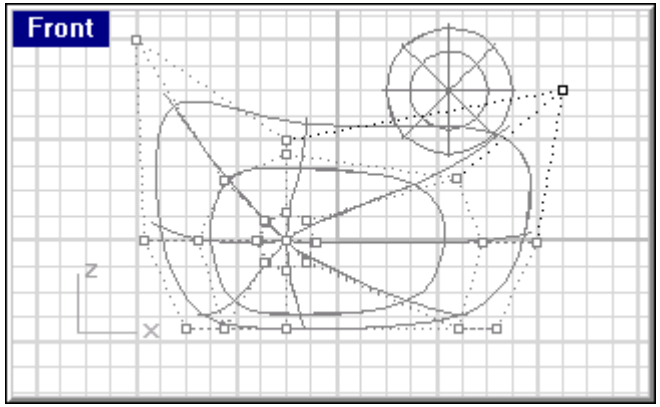


9 Window select the control points at the front of the body, and drag them to the right to bulge out the chest.



Notes:

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



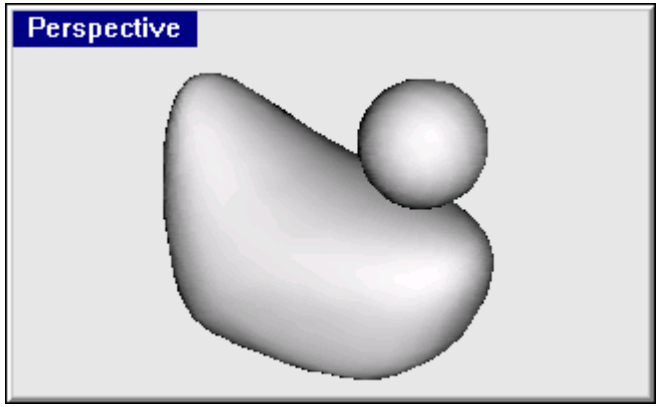
11 You can adjust the control points further until you get the shape you want.

12 **Save** your model.

To shade the model:

1 Click in the **Perspective** viewport.

2 From the **Render** menu, click **Shade.**



Notes:

Shade
Look for this button.

3 Press ENTER or ESC to exit Shade mode.

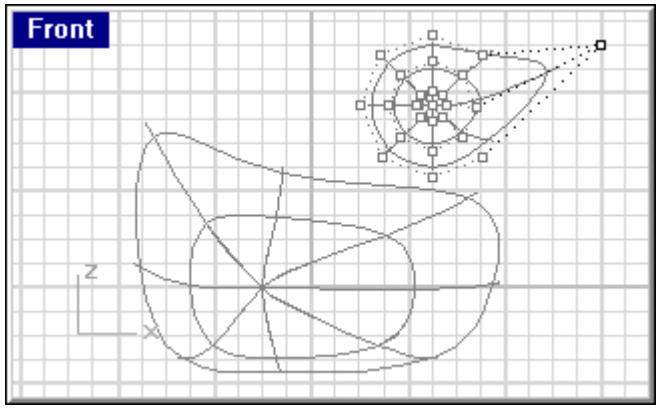
4 Press ESC to turn off control points. The first ESC cancels any commands that are active. The next deselects all selected objects. The next ESC turns off control points.

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.

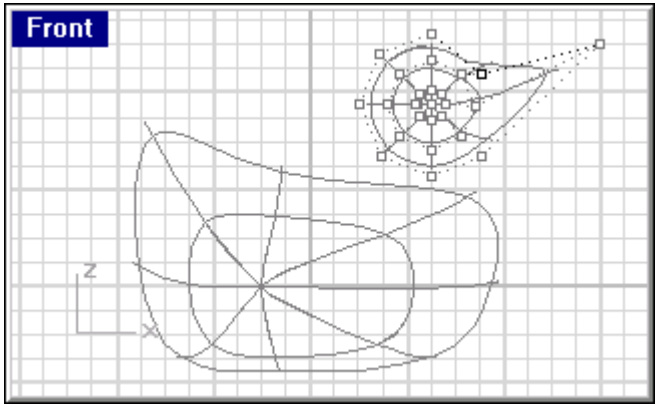


Notes:

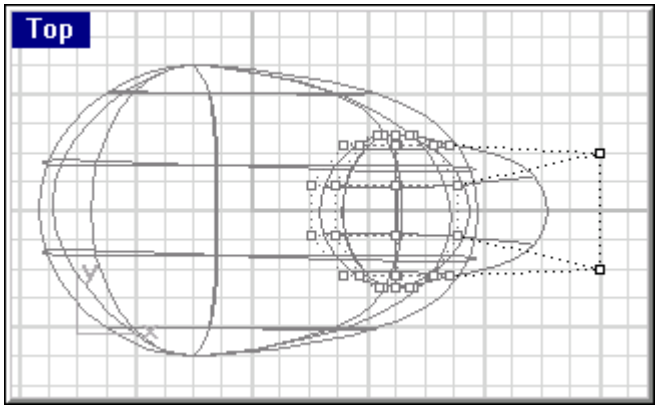
Control Points On
Look for this button.

Remember to use a window to select the control points. There may be multiple control points in the same location in this view.

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

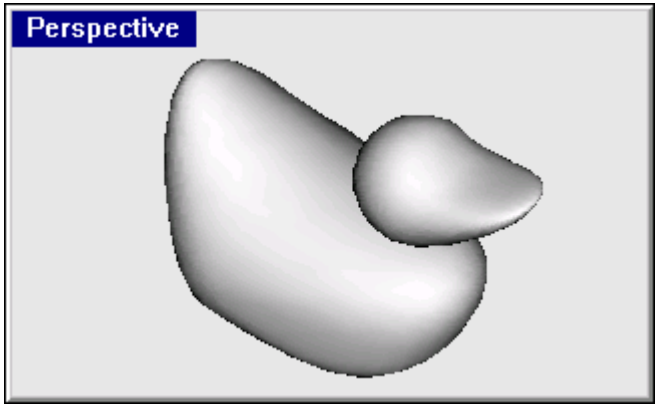


5 Continue selecting control points and deforming the head until you make it look like a bill as shown below.



Notes:

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



7 Press ESC to turn control points off.

Separate the Bill From the Head

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

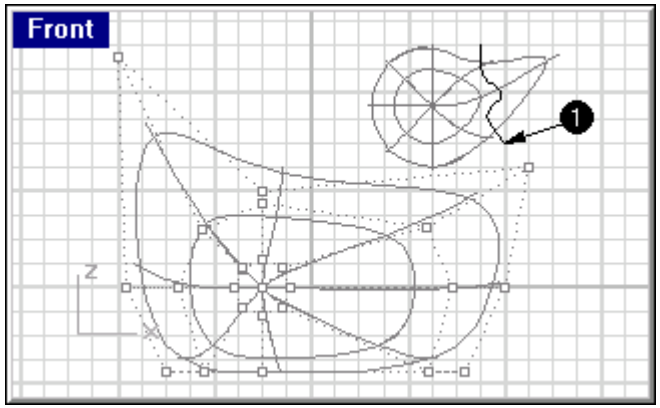
To make the head and bill two separate surfaces, split it with a curve. It is easiest to draw a curve in one view and project it to the surface to create a 3D curve.

Notes:

Shade
Look for this button.

To create the curve for the bill:

- In the **Front** view create a curve that looks like the illustration below.



To Project the curve onto the surface of the head to create the shape for the bill.

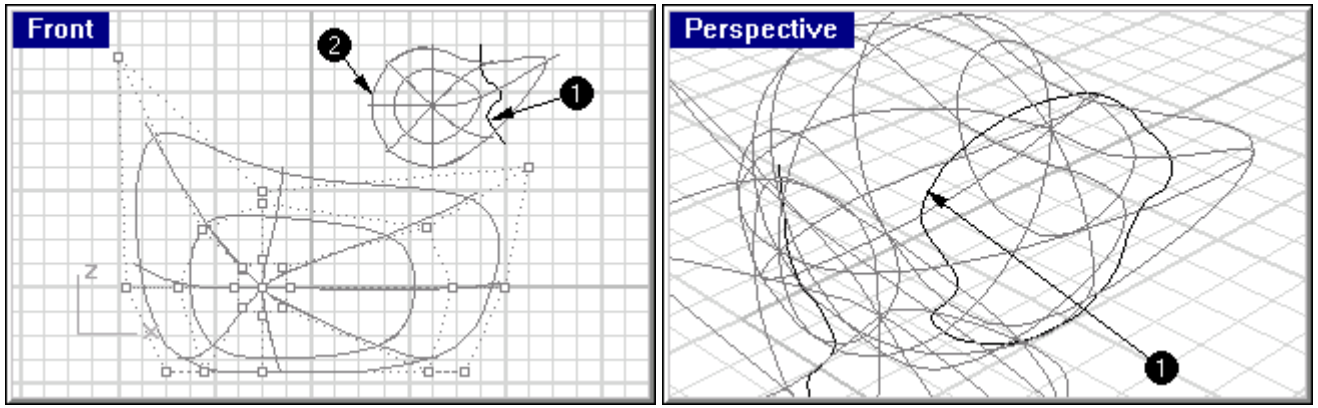
1. From the **Curve** menu, click **From Objects**, then click **Project**
2. At the **Select curves to project** prompt, in the **Front** view, select the curve (1).
3. At the **Select curves to project** prompt, press ENTER.
4. At the **Select surfaces or polysurfaces to project onto** prompt, select the head you created (2).

Notes:

Project
Look for this button.

5 At the **Select surfaces or polysurfaces to project onto** prompt, press ENTER.

The curve is projected onto the head.



6 Delete the original curve.

To split a surface with a curve:

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

2 At the **Select object to split** prompt, select the head (1).

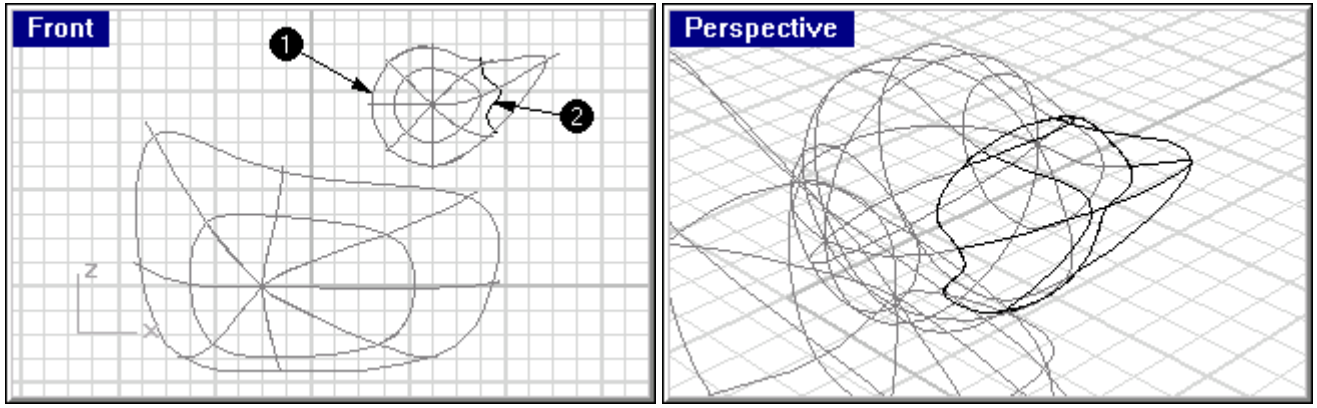
3 At the **Select cutting objects** prompt, select the curve you just created (2).

4 At the **Select cutting objects** prompt, press ENTER.

Notes:

Split
Look for this button.

The bill separates from the head.



5 **Save** your model.

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:

1 Select the head.

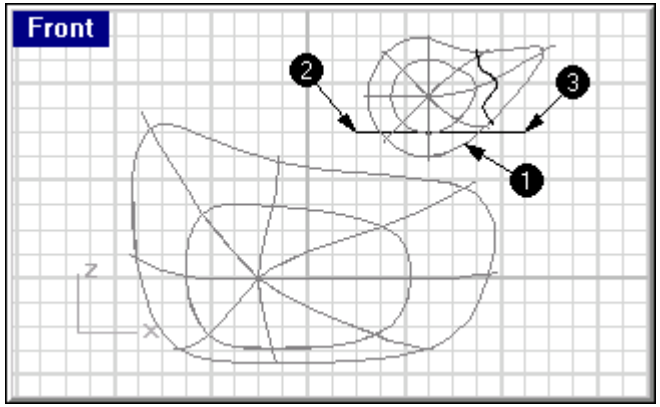
2 From the **Surface** menu, click **Rectangle**, then click **Cutting Plane**.

3 At **Start of cut plane** prompt, in the **Front** view, pick the first point of a line (2) to define a plane.

4 Toggle **Ortho** on.

Notes:

5 At the **End of cut plane** prompt, select the end point of a line (3) that cuts through the lower part of the head.



6 At the **Start of cut plane** prompt, press ENTER to end the command.

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).

3 At the **Select cutting edges (Options)** prompt, press ENTER.

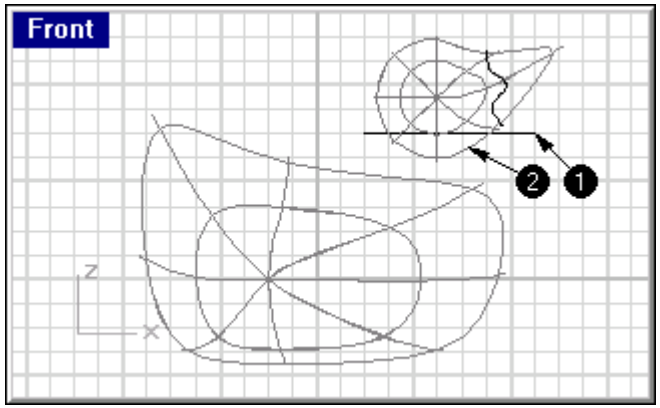
4 At the **Select object to trim (Options)** prompt, select the bottom edge of the head (2).

Notes:

Trim
Look for this button.

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.

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 curves to project** prompt, in the **Top** view, select the curve at the bottom of the head (1).

3 At the **Select curves to project** prompt, press ENTER.

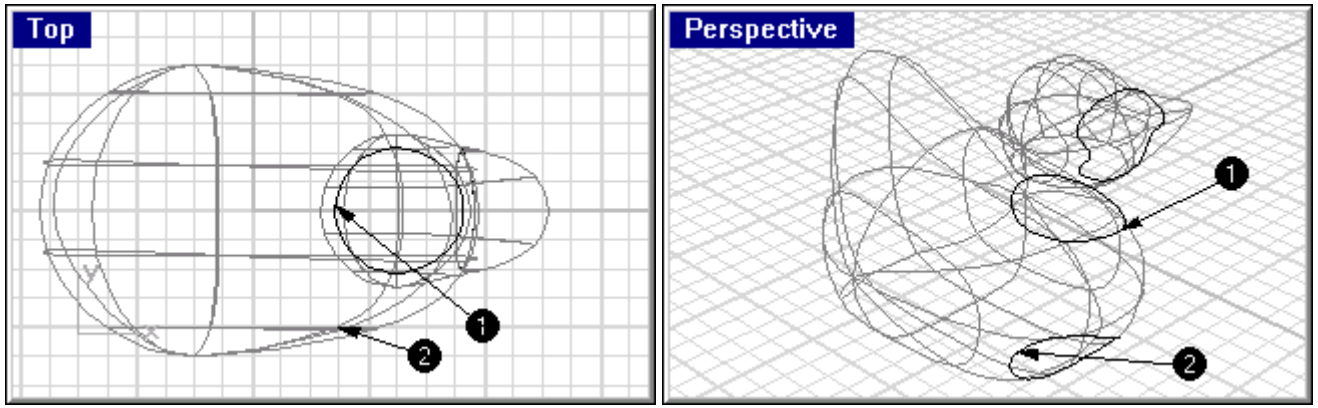
4 At the **Select surfaces or polysurfaces to project onto** prompt, in the **Top** view, select the body (2).

Notes:

Project
Look for this button.

5 At the **Select surfaces or polysurfaces to project onto** prompt, press ENTER.

The curve is projected to the top (1) and the bottom (2) of the body.



6 From the **Edit** menu, click **Trim.**

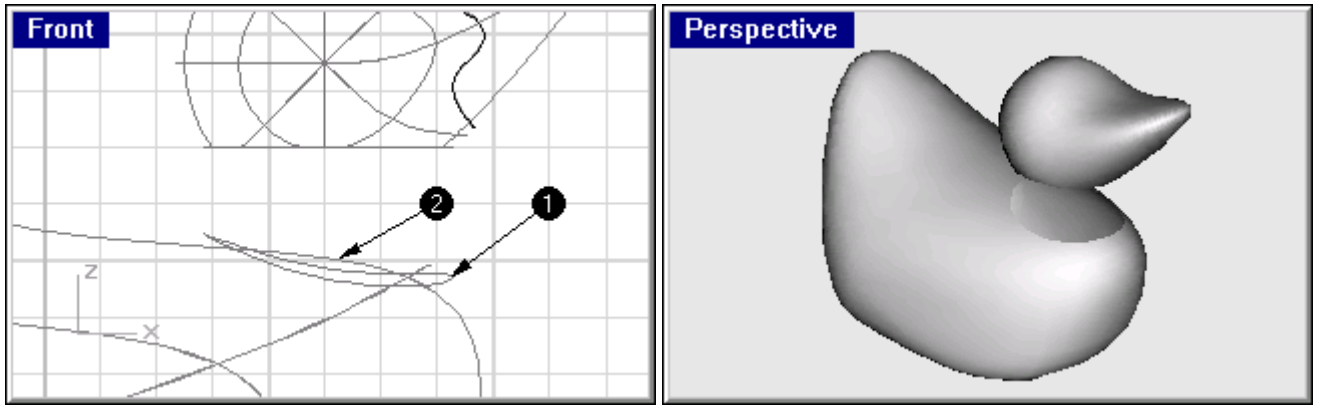
7 At the **Select cutting edges (Options)** prompt, select the curve at the top of the body.

8 At the **Select cutting edges (Options)** prompt, press ENTER.

9 At the **Select object to trim (Options)** prompt, select the body on the inside of the curve.

10 At the **Select object to trim (Options)** prompt, press ENTER.

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



Notes:

11 You can delete the projected curves.

This may take more than one pick. Occasionally the curve gets broken into pieces.

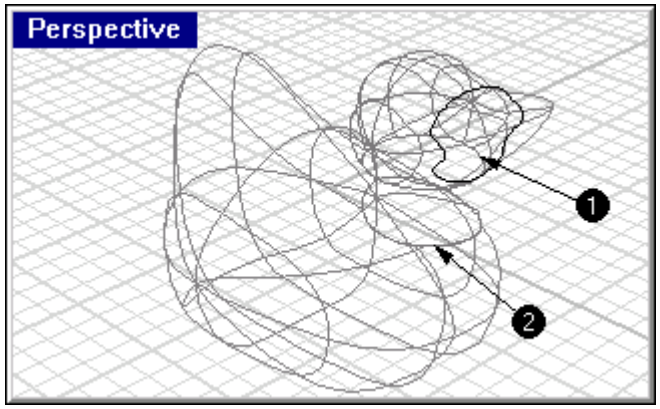
12 **Save** your model.

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.

3 At the **Select edge to blend - pick near one end (PlanarSections Degree=Quintic)** prompt, select the curve on the body.



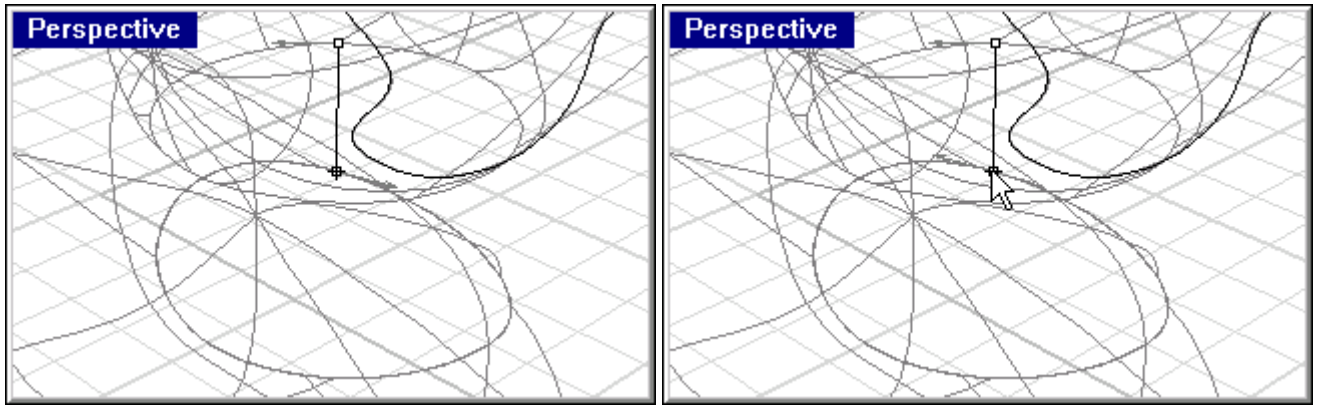
4 At the **Select seam point to adjust, press Enter when done (FlipDirection Automatic Natural)** prompt, check to see that the seam point lines up.

If it doesn't, select one of the seam points and drag it until it lines up with the other one. If the curve direction arrows are not pointing in the same direction flip one of them.

Notes:

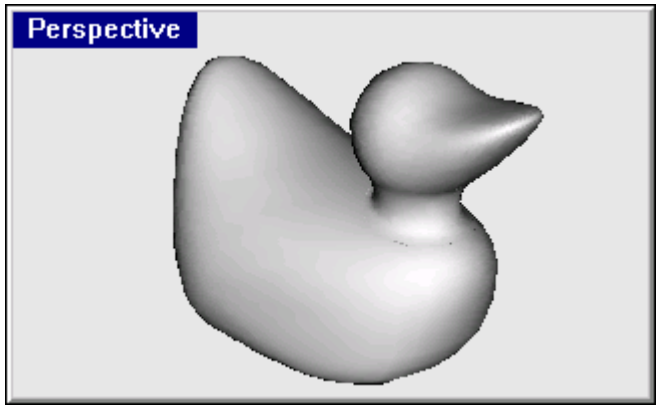
Blend Surface
Look for this button.

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



6 At the **Location of seam point (FlipDirection)** prompt, press ENTER.

A surface is blended between the body and the head.

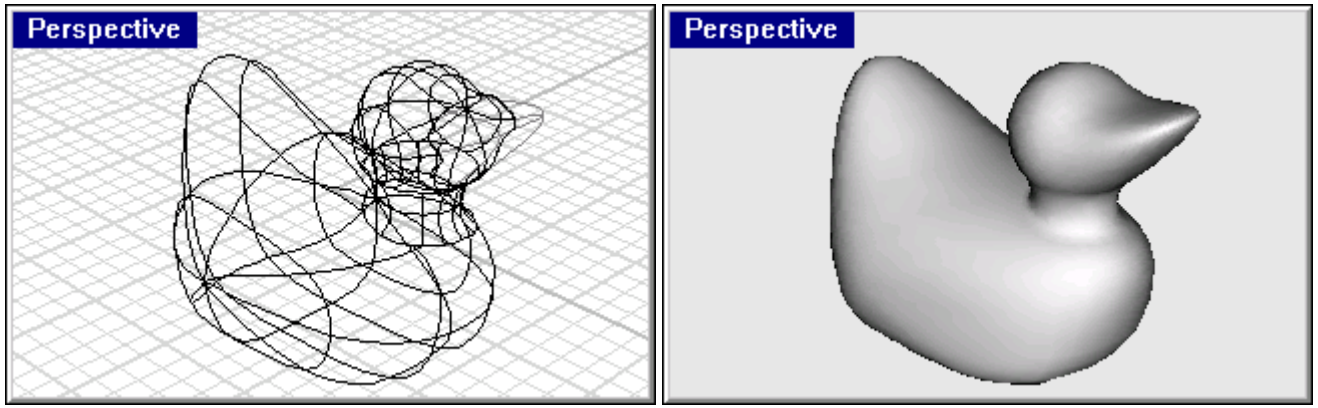


7 **Save** your model.

Notes:

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.



To make an eye:

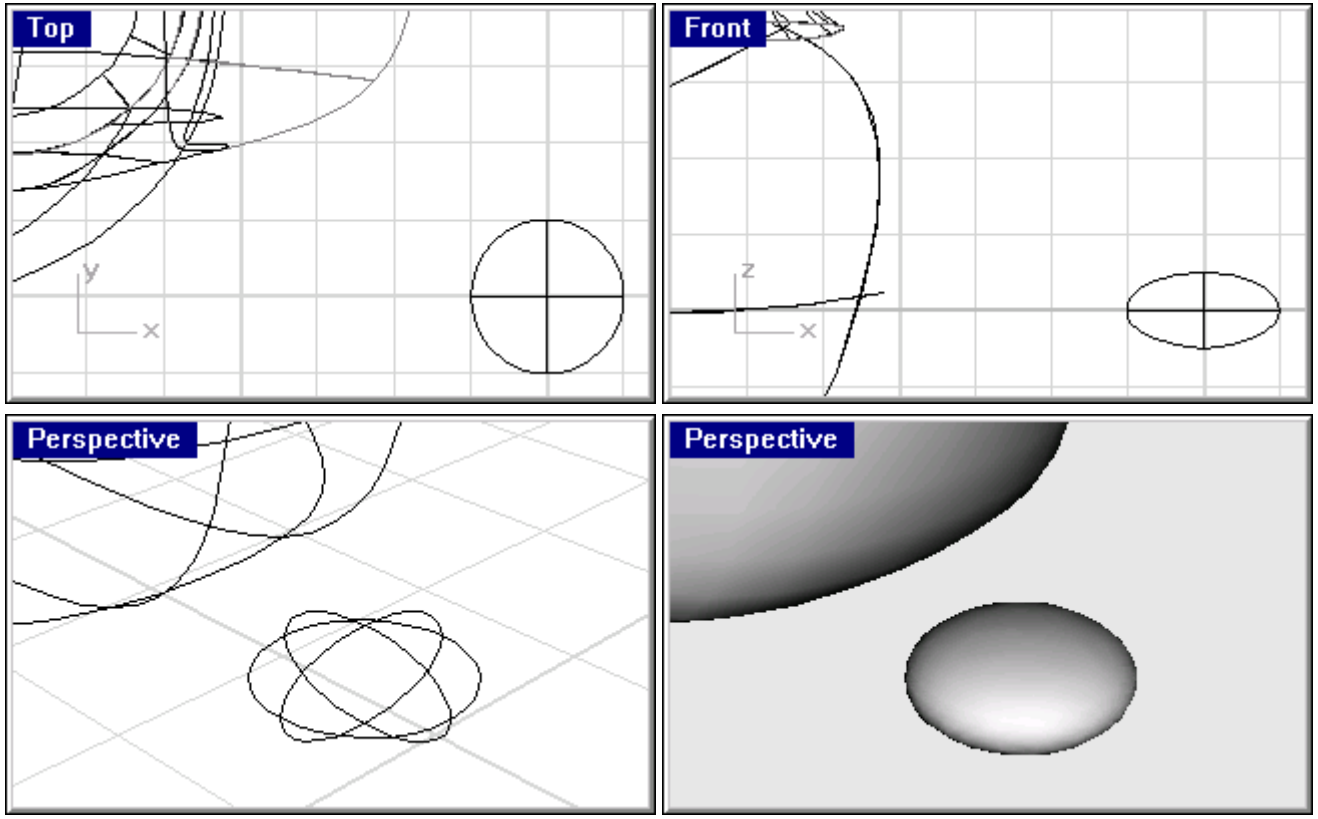
1. From the **Solid** menu, click **Ellipsoid.**
2. Toggle **Ortho** and **Snap** on to help.
3. At the **Center of ellipsoid** prompt, pick a point in the **Top** viewport.
4. At the **End of first axis** prompt, pick a point.
5. At the **End of second axis** prompt, pick a point that is approximately the same radius as the previous pick.

Notes:

Join
Look for this button.

Ellipsoid
Look for this button.

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



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.

Notes:

Circle
Look for this button.

4 From the **Curve** menu, click **From Objects**, then click **Project.**

5 At the **Select curves to project** prompt, in the **Top** view, select the circle.

6 At the **Select curves to project** prompt, press ENTER.

7 At the **Choose surfaces or polysurfaces to project onto** prompt, in the **Top** view, select the ellipsoid.

8 At the **Choose surfaces or polysurfaces to project onto** prompt, press ENTER.

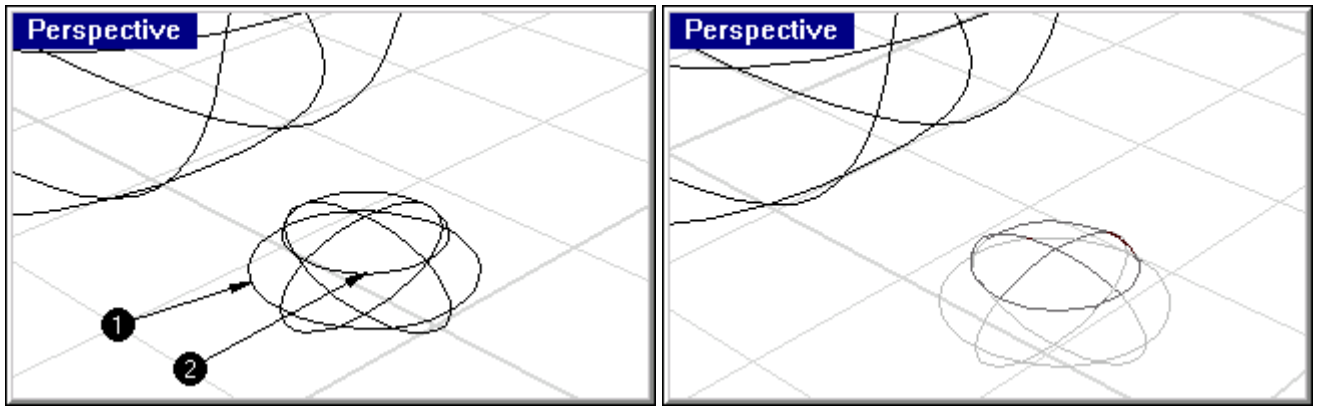
9 Delete the original circle and the one on the bottom of the ellipsoid.

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

11 At the **Select object to split** prompt, select the ellipsoid.

12 At the **Select cutting objects** prompt, select the circle on the top of the ellipsoid.

13 At the **Select cutting objects** prompt, press ENTER.



14 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.

15 From the From the **Edit** menu, click **Object Properties.**

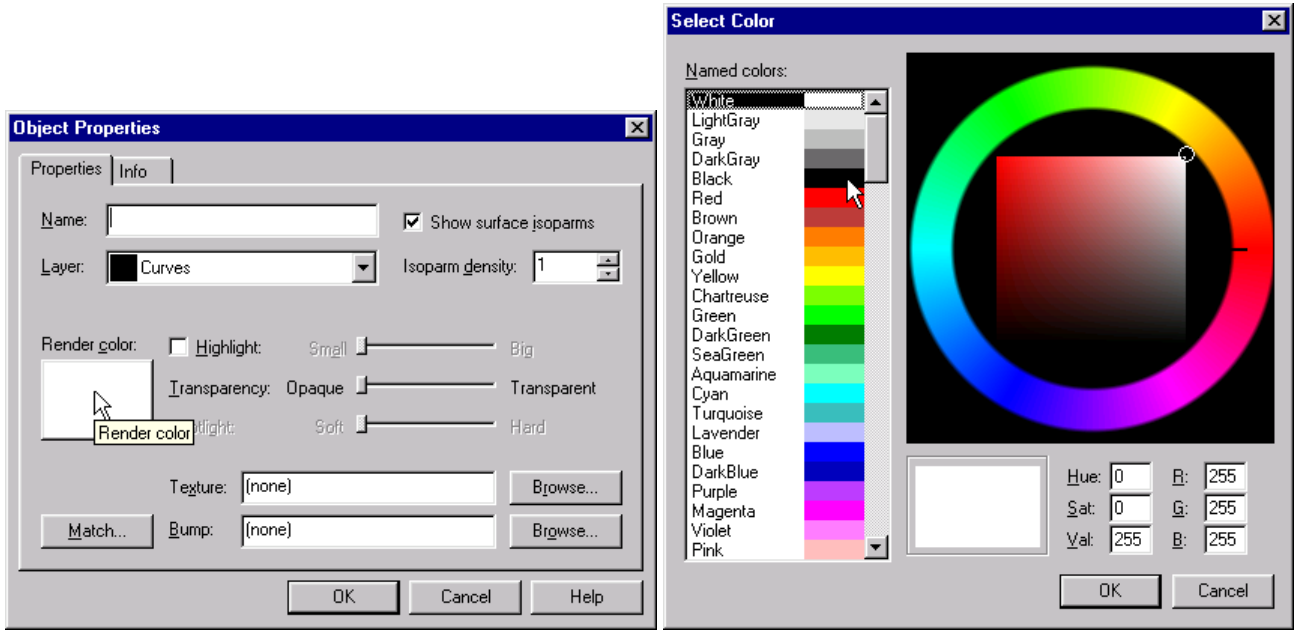
Notes:

Project
Look for this button.

Split
Look for this button.

Properties
Look for this button.

16 In the **Object Properties** dialog, click **Render Color** and select a render color for the pupil of the eye, like black.



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.

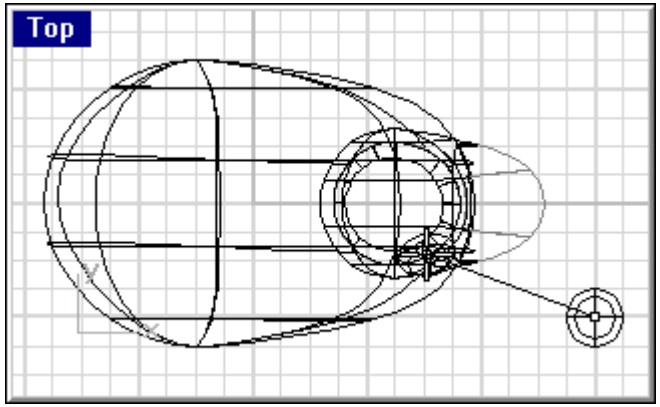
5 At the **Base point in world coordinates (OnSurface)** prompt, pick the center of the ellipsoid.

6 At the **Offset point** prompt, move the cursor onto the head to place the eye.

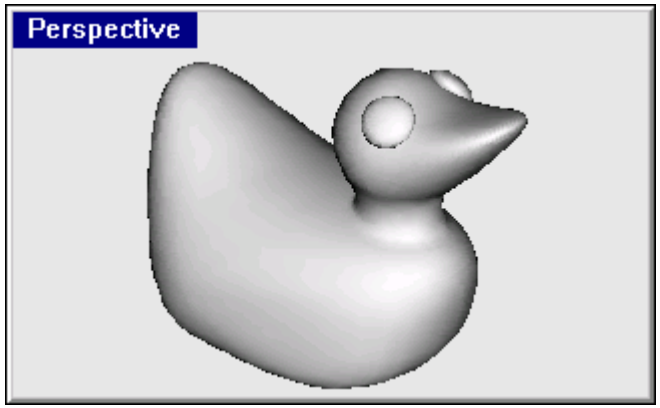
Notes:

Orient On Surface
Look for this button.

7 At the **Offset point** prompt, press ENTER.



8 **Mirror** the eye to the other side of the head.



9 **Save** your model.

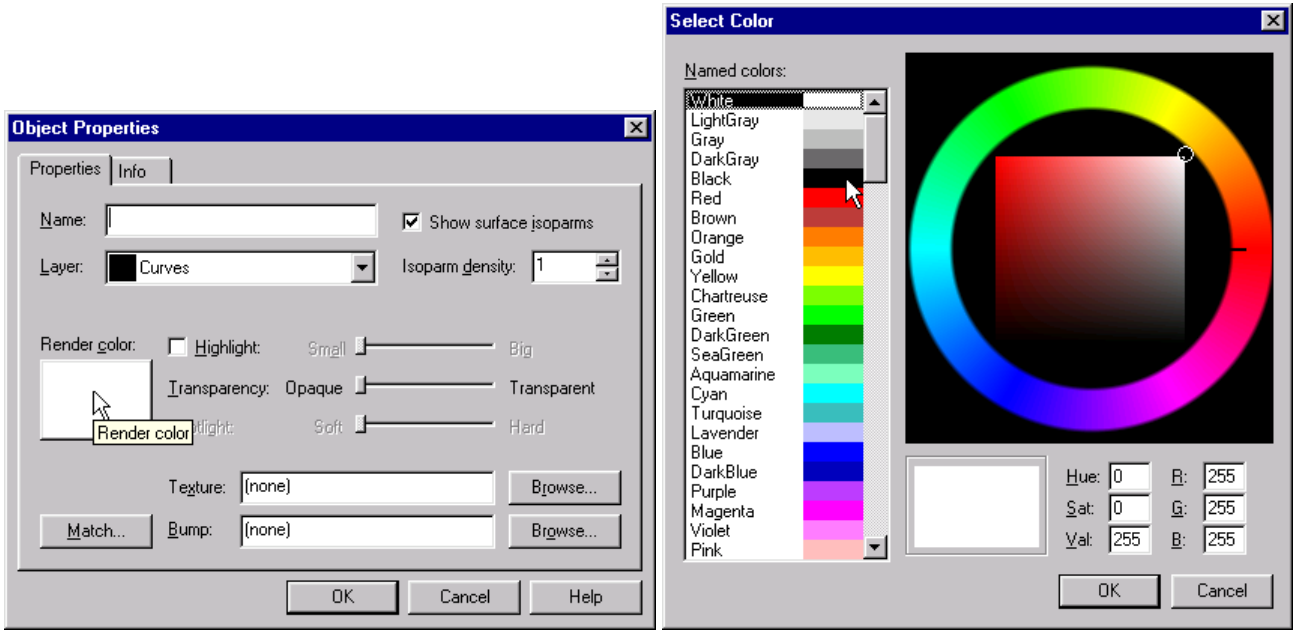
Render a Picture of the Ducky

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.

Notes:

To render the ducky:

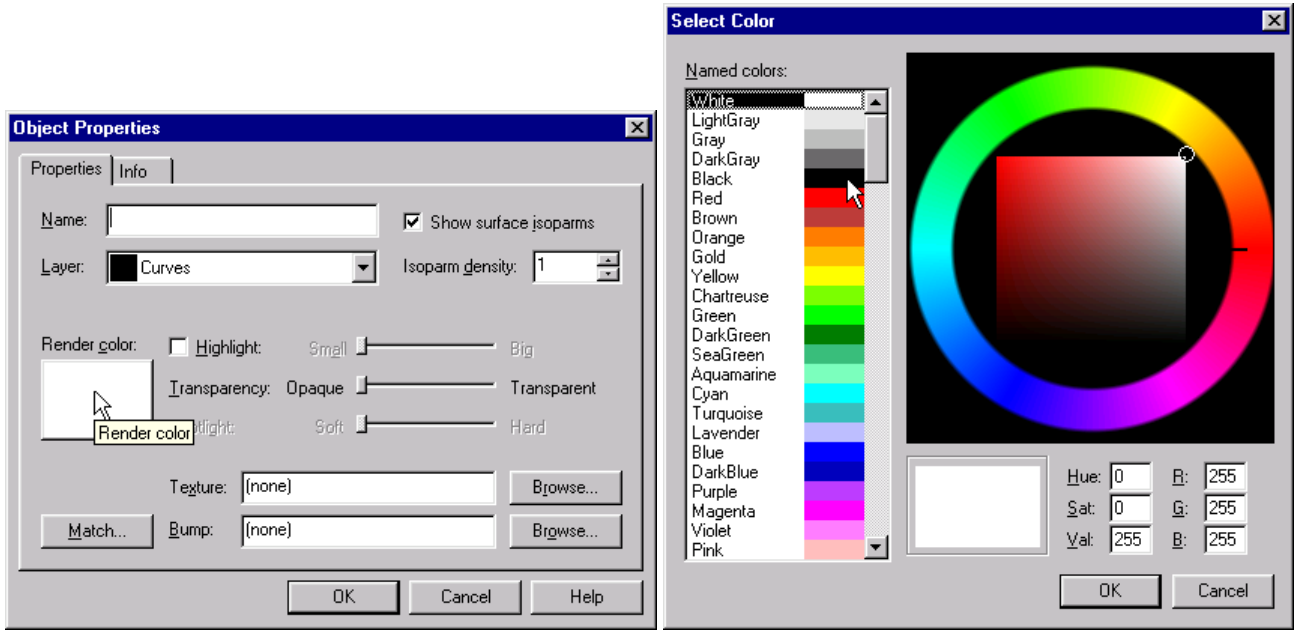
1. Select the beak.
2. From the From the **Edit** menu, click **Object Properties.**
3. In the **Object Properties** dialog, click **Render Color** and select a color for the beak, like orange.



4. Select the body.
5. From the From the **Edit** menu, click **Object Properties.**

Notes:

6 In the **Object Properties** dialog, click **Render Color** and select a color for the body, like yellow.



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

Notes:

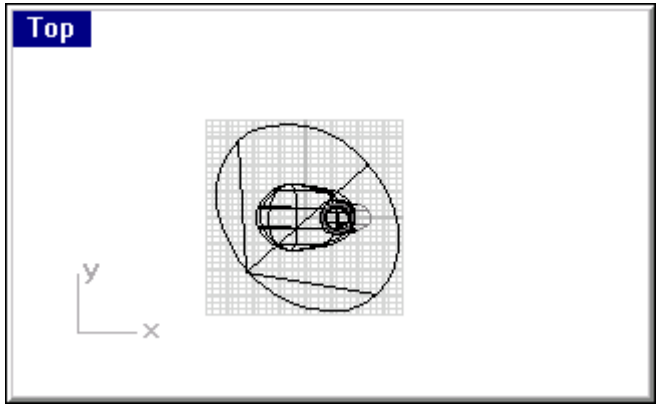
Render
Look for this button.

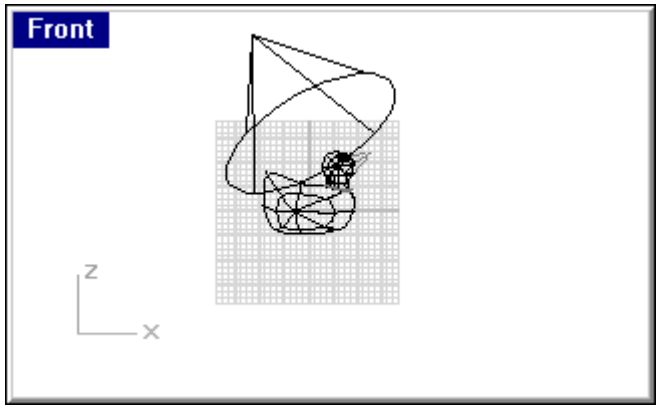
To place lights:

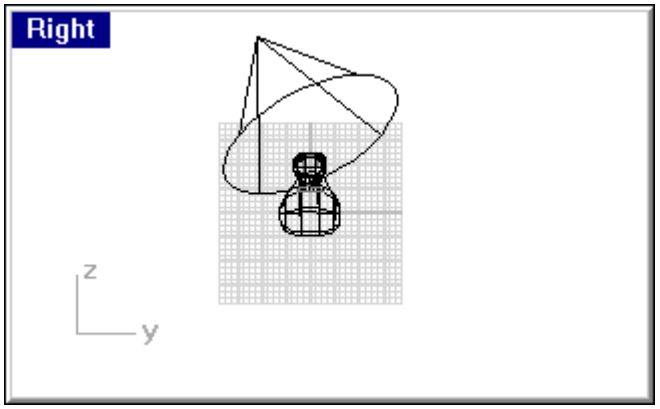
1. From the **Render** menu, click **Create Spotlight.**
2. At the **Base of cone (Vertical)** prompt, select a point in the middle of the model.
3. At the **Radius** prompt, drag the radius until it is approximately twice as large as the model.
4. At the **End of cone** prompt, pick a point in the **Top** view while holding the CTRL key down.

   This technique activates **Elevator Mode**. The point should be slightly to the left and below the model.

   In the **Front** view pick a point slightly above the object.





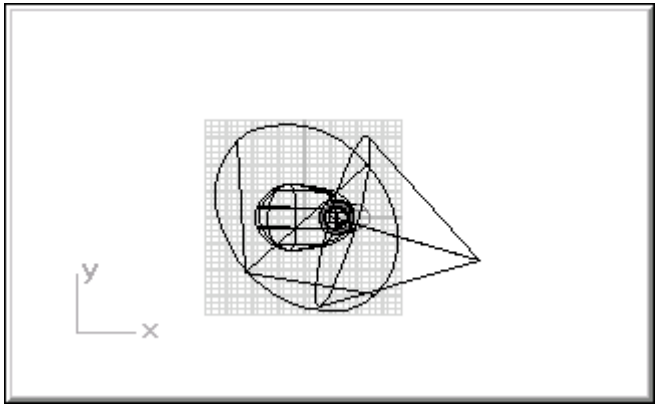


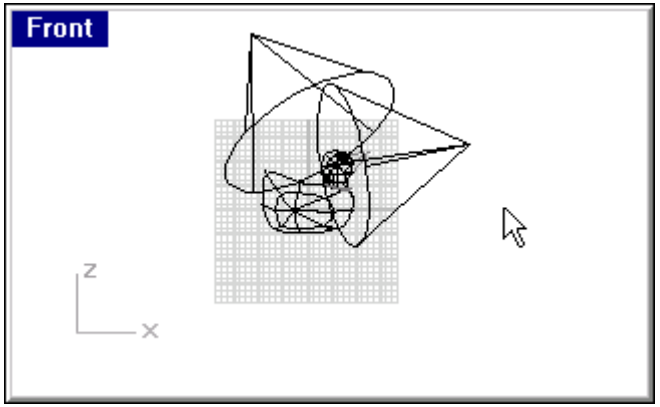
Notes:

Spotlight
Look for this button.

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

6 Try placing two lights.

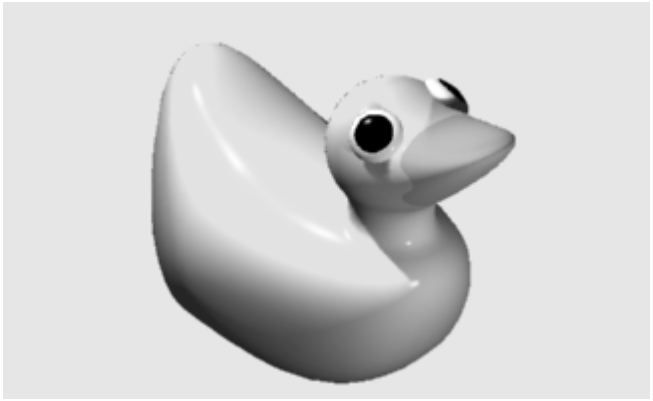




Notes:

Render
Look for this button.

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



8 **Save** your model.

Notes:

Render
Look for this button.

9 Modeling with Solids

Modeling solids

Modeling solids in Rhino is easy. There are several commands that enable you create and edit solid objects.

Button	Command	Description
	Box	Creates a rectangular box from two diagonal corners and a height. Right button option: Creates a box from two adjacent corners, a point on the edge across from them, and a height.
	Sphere	Creates a sphere from a center point and a radius.
	SphereD	Creates a sphere from the two ends of its diameter.
	Sphere3Pt	Creates a sphere from three points on its surface.
	Cylinder	Creates a solid cylinder from a center point, a radius, and a height.
	Tube	Creates a solid tube from a center point, two radii, and a height.

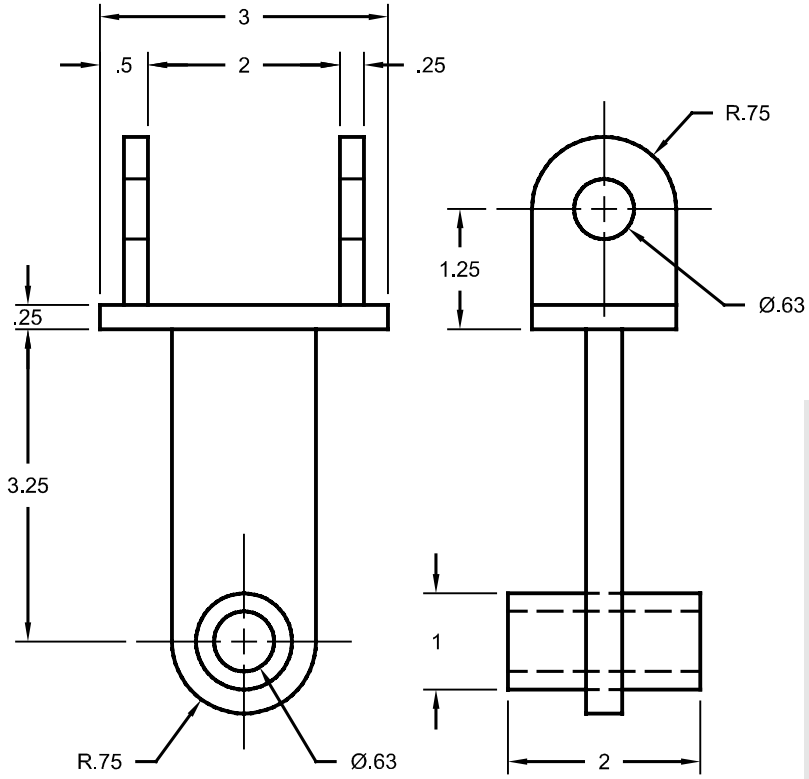
Notes:

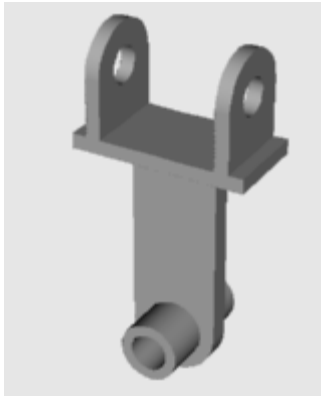
Button	Command	Description
	Cone	Creates a solid cone from a base point, a base radius, and a height.
	TCone	Creates a truncated cone that ends in a circular plane from a base point, two radii, and a height.
	Ellipsoid	Creates an ellipsoid from a center point, and three axis endpoints.
	Torus	Creates a solid torus from a center point, a radius for the center of the tube, and a tube radius.
	Pipe	Creates a pipe around an existing curve. Pipe bundles the Circle command with the OnCurve option of one-rail sweep and caps the ends to create solid pipes. The Thick option lets you specify two radii at each end of the curve to create pipes with holes bored through the middle like tubing.
	Extrude Closed Planar Curve	Creates a solid by extruding a closed planar curve vertically.
	Extrude Surface	Creates a solid by extruding a surface vertically. Extrude planar curve with draft angle
	Cap Planar Holes	Creates simple planar surfaces to close simple planar holes in a surface or partial solid.
	Boolean Union	Boolean operation in Rhino to combine solids.
	Boolean Difference	Boolean operation in Rhino to subtract surfaces and solids from another.
	Boolean Intersection	Boolean operation in Rhino to create the intersection of two surfaces or solids.

Notes:

Exercise 50—Creating solids.

Start a new model. Save as **Swing Bracket**.

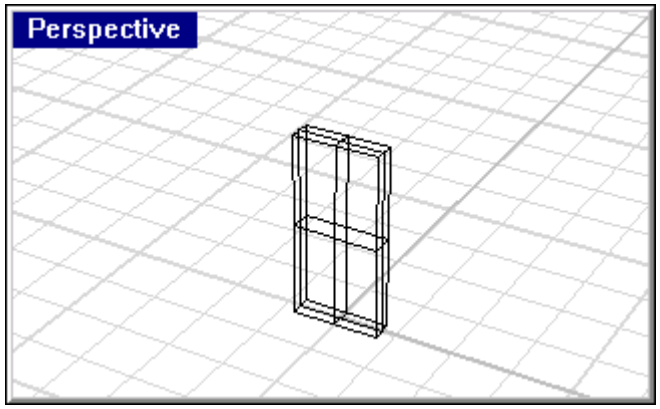




Notes:

To create the boxes:

1. From the **Solid** menu, click **Box**, then click **Corner to Corner, Height.** Make sure your cursor is in the **Front** viewport.
2. At the **First corner of box** prompt, type **-.75,0,-.1875** and press ENTER.
3. At the **Other corner or length** prompt, type **1.5** and press ENTER.
4. At the **Width** prompt, type **3.25** and press ENTER.
5. At the **Height** prompt, type **.375** and press ENTER. The first box is created.

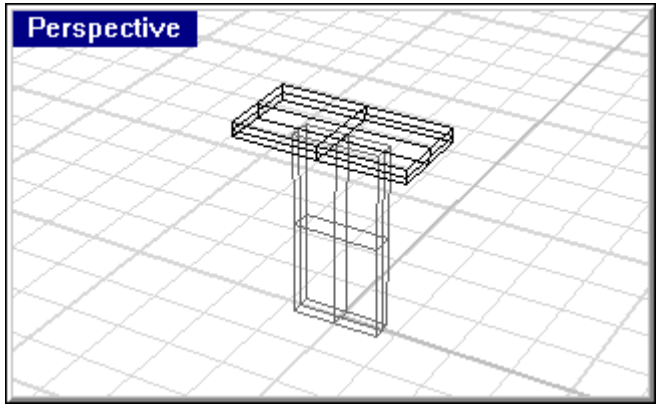


6. From the **Solid** menu, click **Box**, then click **Corner to Corner, Height.** Make sure your cursor is in the **Top** viewport.
7. At the **First corner of box** prompt, type **-1.5,-.75,3.25** and press ENTER.
8. At the **Other corner or length** prompt, type **3** and press ENTER.
9. At the **Width** prompt, type **1.5** and press ENTER.

Notes:

Box
Look for this button.

10 At the **Height** prompt, type **.25** and press ENTER.



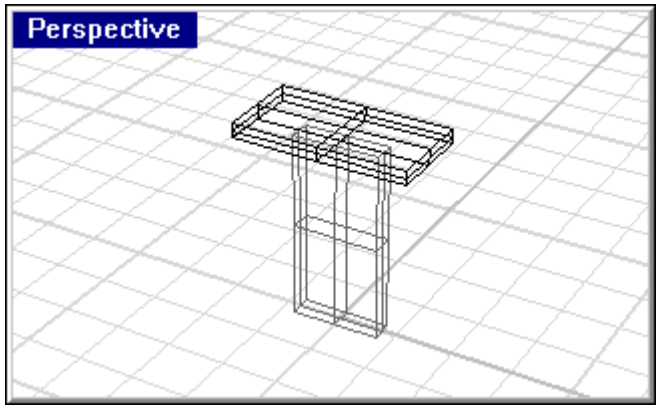
11 From the **Solid** menu, click **Box**, then click **Corner to Corner, Height.** Make sure your cursor is in the **Right** viewport.

12 At the **First corner of box** prompt, type **-.75,3.5,-1.25** and press ENTER.

13 At the **Other corner or length** prompt, type **1.5** and press ENTER.

14 At the **Width** prompt, type **1** and press ENTER.

15 At the **Height** prompt, type **.25** and press ENTER.



16 Select the last box you created.

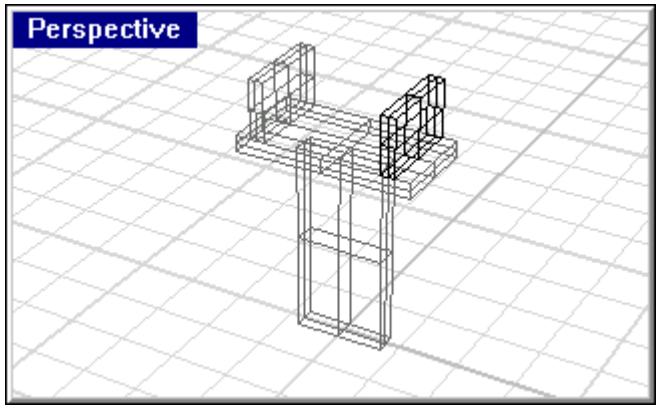
Notes:

17 From the **Transform** menu, click **Mirror.** Make sure your cursor is in the **Front** viewport.

18 At the **Start of mirror plane** prompt, type **0,0** and press ENTER.

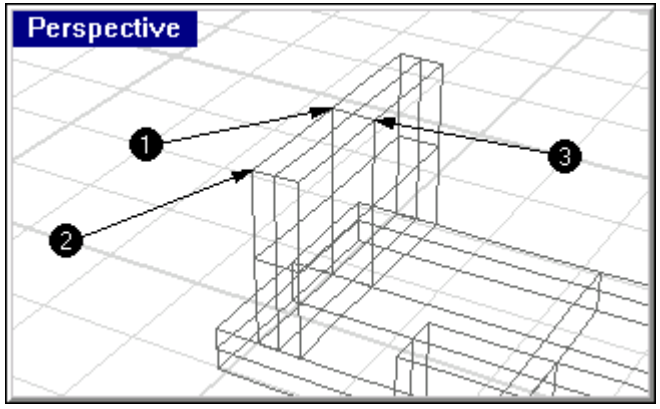
19 At the **End of mirror plane** prompt, toggle **Ortho** on and pick a point directly above **0,0**.

The box is mirrored to the other part.



To create the cylinders:

1 Turn on **End** and **Mid** osnap.



2 From the **Solid** menu, click **Cylinder.**

Notes:

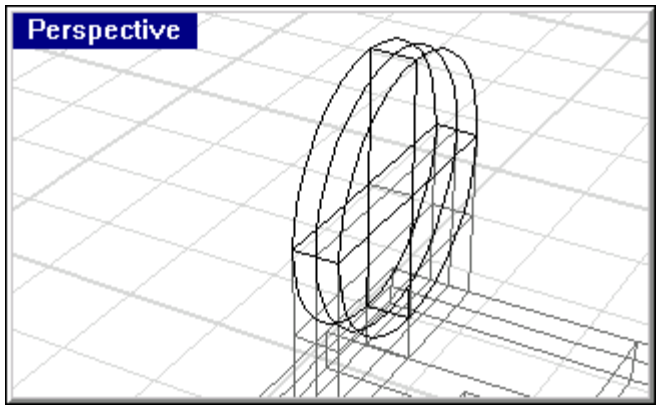
Mirror
Look for this button.

Cylinder
Look for this button.

3 At the **Base of cylinder (Vertical)** prompt, snap to the midpoint at the top of the last box you created.

4 At the **Radius <0.75> (Diameter)** prompt, snap to the endpoint of the left top edge of the same box.

5 At the **End of cylinder** prompt, snap to the other midpoint at the top of the same box.



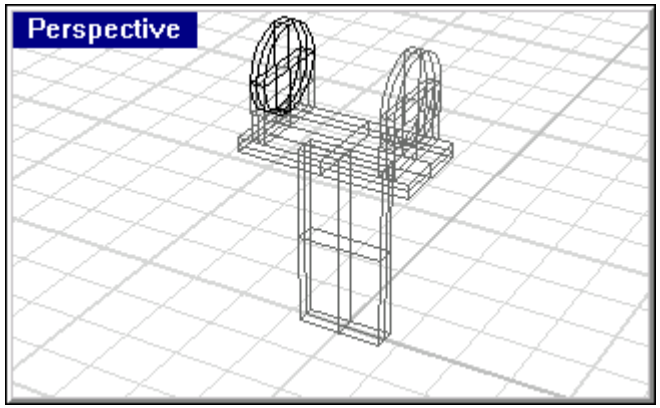
6 Select the cylinder you just created.

7 From the **Transform** menu, click **Copy.**

8 At the **Point to copy from (Vertical InPlace)** prompt, snap to the midpoint at the top left edge of the box.

9 At the **Point to copy to** prompt, snap to the midpoint of the corresponding box at the other end of the part.

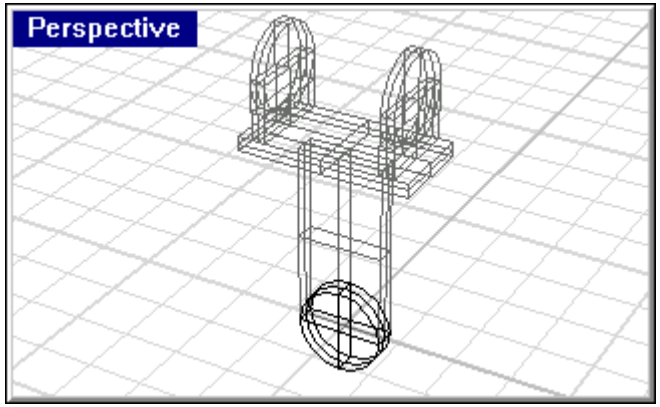
10 At the **Point to copy to** prompt, press ENTER.



Notes:

Copy
Look for this button.

11 Use the same procedure to make the cylinder at the bottom of the part.



To create the tube:

1 From the **Solid** menu, click **Tube.**

2 At the **Base of tube (Vertical)** prompt, move your cursor into the **Front** viewport and type **0,0,1** and press ENTER.

3 At the **Radius <0.75> (Diameter)** prompt, type **D** and press ENTER.

4 At the **Diameter <1.5> (Radius)** prompt, type **.63** and press ENTER.

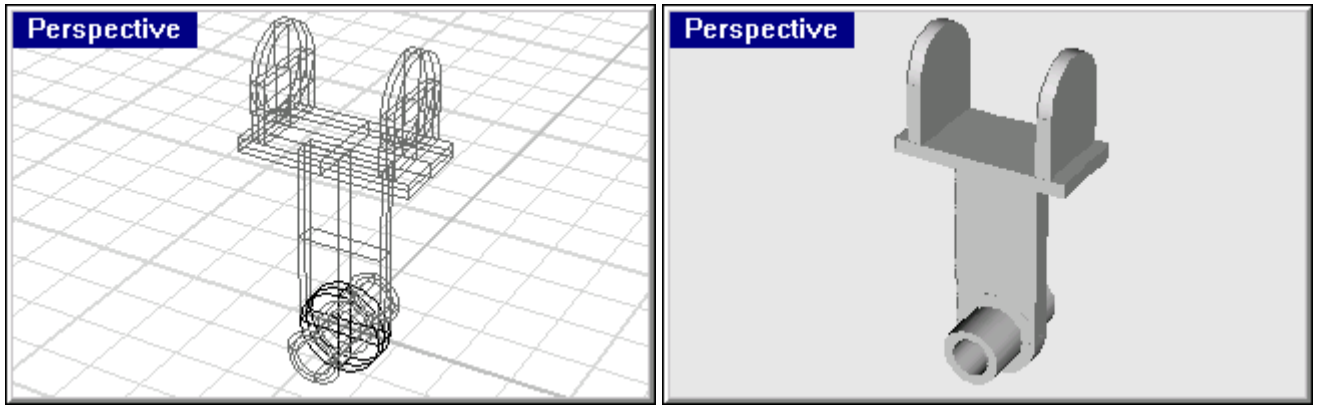
5 At the **Diameter <2> (Radius)** type **1** and press ENTER.

6 At the **End of tube** prompt, type **2** and press ENTER.

Notes:

Tube
Look for this button.

7 At the **End of tube** prompt, move your cursor into the **Top** viewport, toggle **Ortho** on and pick a point directly above the previous point.



8 **Save** your model.

To join the parts:

1 From the **Solid** menu, click **Union.**

2 At the **Select surface or polysurface** prompt, select one of the parts you created.

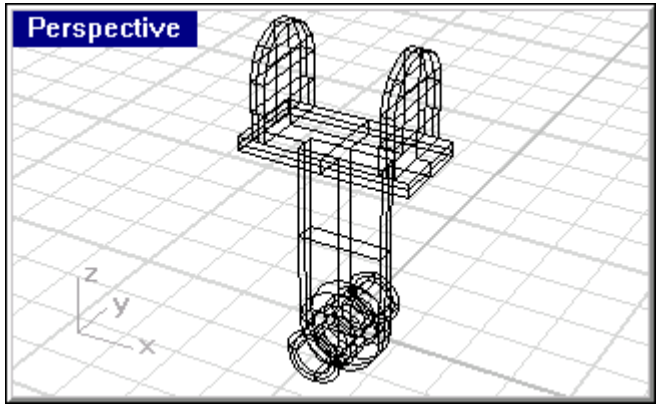
3 At the **Select surface or polysurface to union** prompt, select an adjacent part.

Notes:

Always save your model before you do a boolean operation.

Boolean Union
Look for this button.

4 Continue to **Union** parts together until it becomes one piece.



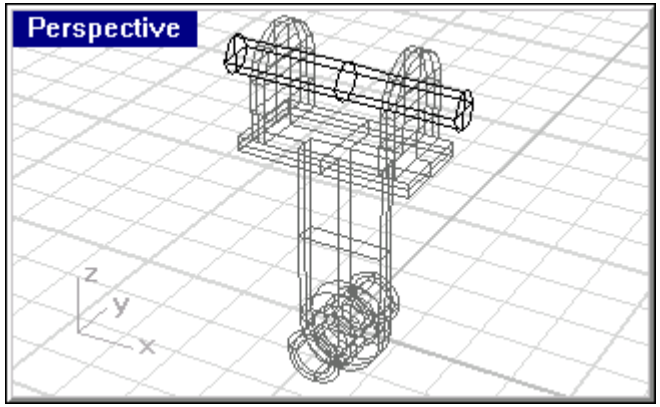
To make the hole through the top part of the bracket:

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

2 At the **Base of cylinder (Vertical)** prompt, move your cursor into the **Top** viewport, type **2,0,4.5** and press ENTER.

3 At the **Diameter <0.75> (Radius)** prompt, type **.63** and press ENTER.

4 At the **End of cylinder** prompt, toggle **Ortho** on and drag the cylinder to a point that passes through the object.

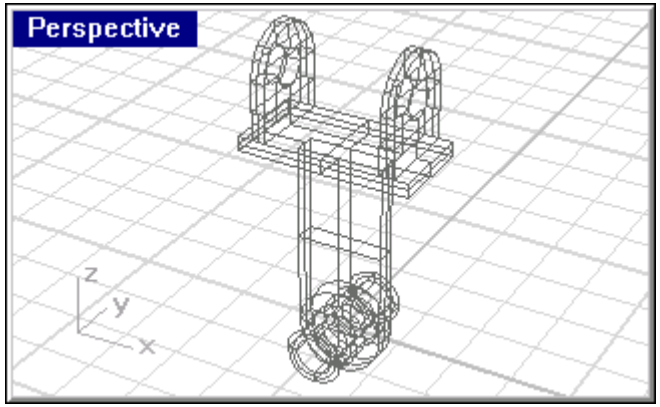


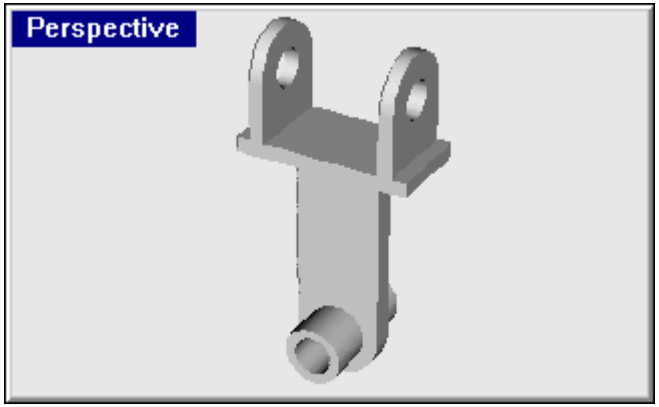
Notes:

Cylinder
Look for this button.

To subtract the cylinder to create the holes:

1. From the **Solid** menu, click **Difference.**
2. At the **Select surface or polysurface** prompt, select joined part.
3. At the **Select surface or polysurface to subtract** prompt, select the cylinder you just created.





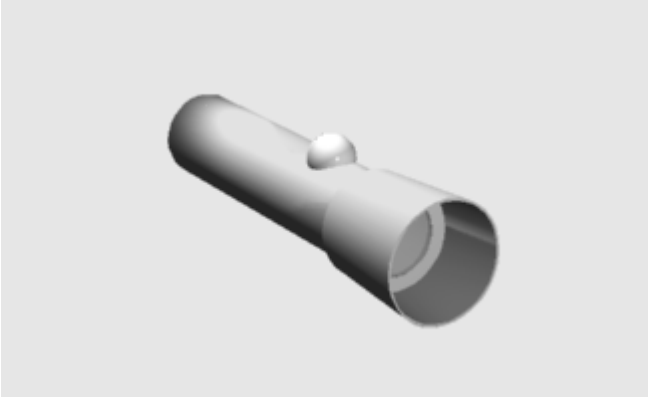
Notes:

Boolean Difference
Look for this button.

Modeling a Flashlight

When this model is finished you will have three distinct parts—the flashlight body, the lens, and the switch.

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 shaded view of the finished flashlight.

Exercise 51— Model a Flashlight

To draw the body:

1. Start a new model. Save as **Flashlight**.
2. From the **Solid** menu, click **Cylinder**.
3. Move your cursor to the **Right** viewport.

   When you are in the middle of a command, you don't have to click in the viewport, just move the cursor there to make it active. The viewport title highlights in blue.
4. At the **Base point** (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.
5. At the **Radius** (Diameter) prompt, type **.75** and press ENTER.

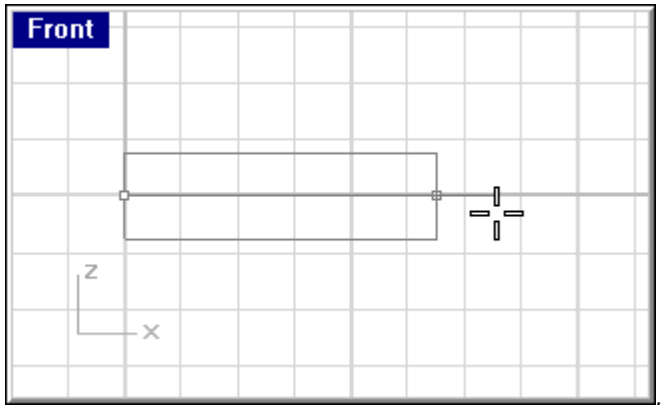
Notes:

Cylinder
Look for this button.

6 At the **End of cylinder** prompt, type **5.5** and press ENTER.

7 Turn **Ortho** on.

8 At the next **End of cylinder** prompt, in the **Front** viewport, pick a point to show the rotation angle so the cylinder points to the right.



9 From the **View** menu click **Zoom**, then click **Extents All**.

Or, right-click the **Zoom Extents** button.

10 Click in the **Perspective** viewport.

When you are not in the middle of a command, you have to click in a viewport to make it the active viewport.

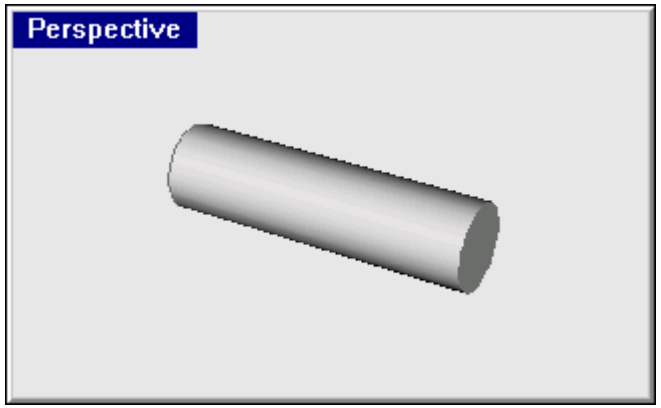
11 From the **Render** menu, click **Shade**.

Notes:

Zoom Extents
Look for this button.

Shade
Look for this button.

Since you will be using **Shade** frequently, this might be a good opportunity to find the toolbar button for **Shade**. It is in the toolbar that runs across the top of the viewports. The icon is a gray sphere.

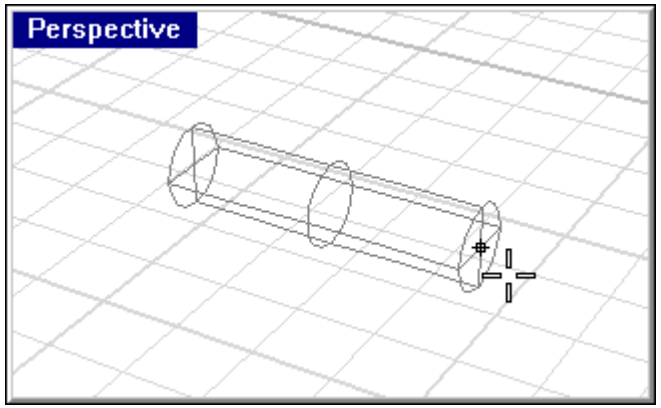


To draw the flashlight reflector shield:

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.



Notes:

Truncated Cone
Look for this button.

3 At the **Radius <.75>** (Diameter) prompt, type **.875** and press ENTER.

A truncated cone has two radius values, the base and top.

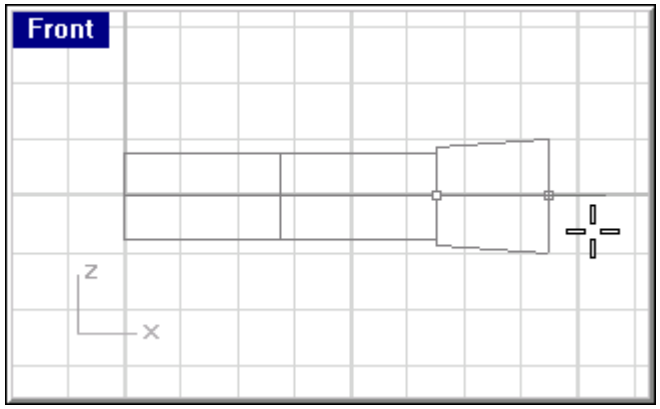
4 At the next **Radius <1.00>** (Diameter) prompt, type **1** and press ENTER.

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 to rotate it into place.

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 rotation angle won't be entered until you click with the left mouse button.

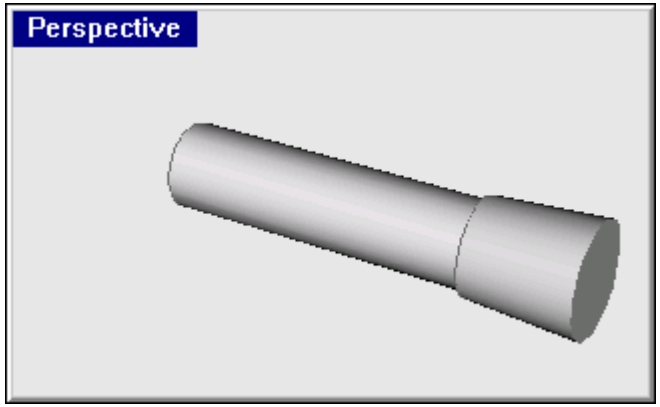


Being able to zoom and pan inside a command makes moving around in Rhino easy.

7 From the **View** menu, click **Zoom**, then click **Extents All**.

Notes:

8 **Shade** the **Perspective** view.



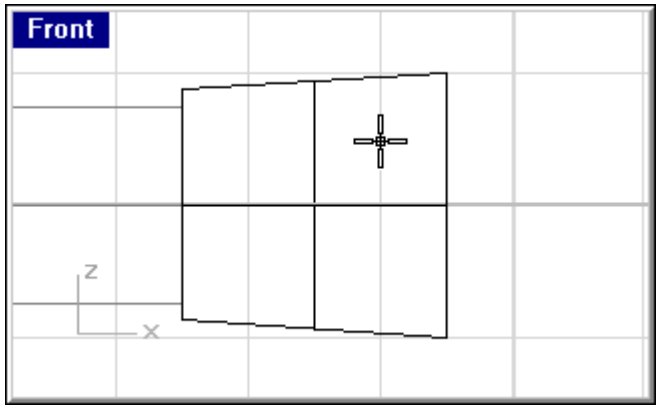
To create the cutting cone:

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.

1 Select the truncated cone.

2 From the **Transform** menu, click **Copy**.

3 At the **Base point** (Vertical InPlace) prompt, in the **Front** viewport pick any point.



Notes:

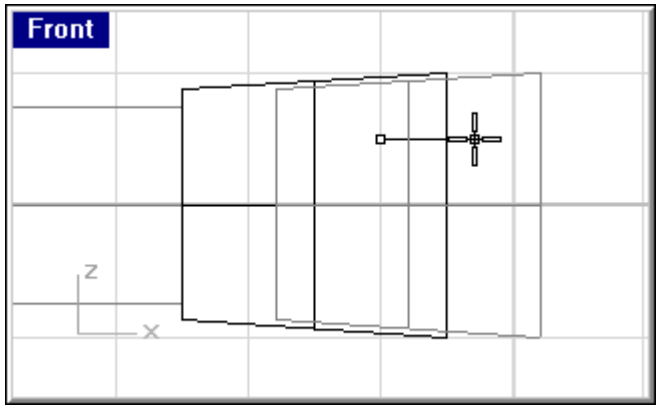
Copy
Look for this button.

4 At the **Offset point** prompt, turn on **Ortho** and drag the cone to the right until the sides of the cone copy are inside the first cone.

5 Pick a point to place the copy, and press ENTER.

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

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



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 boolean commands. Starting the command cancels your selection. You can only select one object at each prompt.

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.

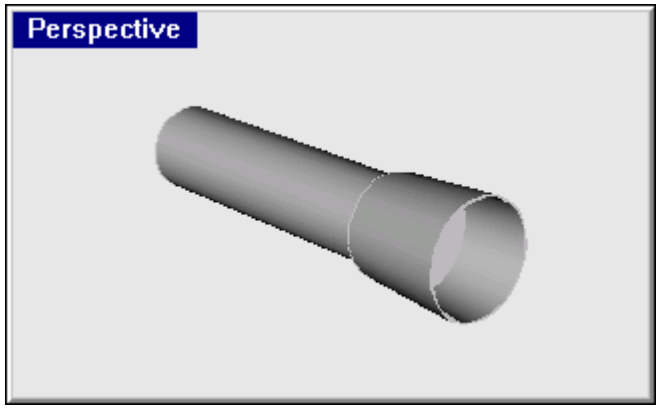
Notes:

Boolean Union
Look for this button.

Boolean Difference
Look for this button.

4 **Shade** the **Perspective** view.

5 Rotate the view a bit so you can see inside the flashlight.

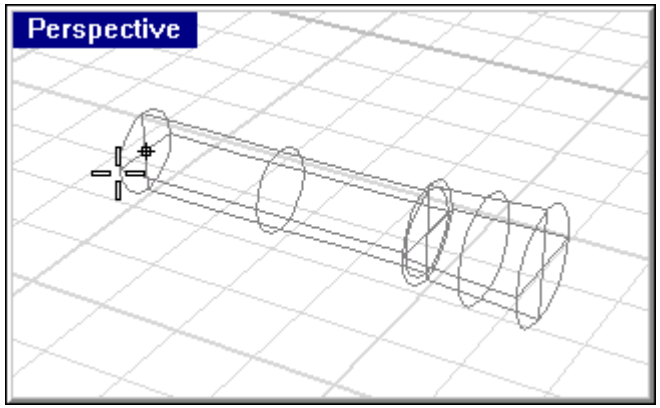


To draw the cutting cylinder:

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



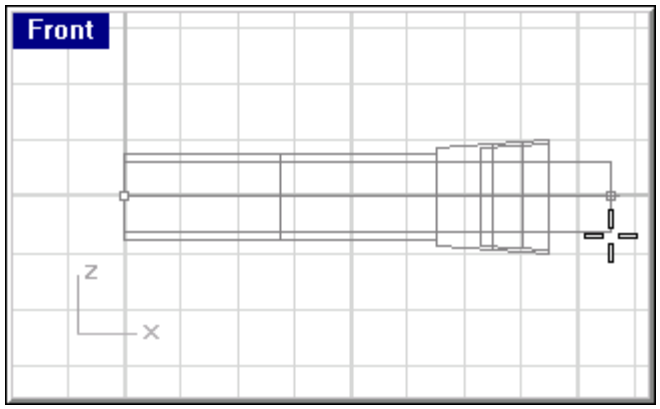
Notes:

Cylinder
Look for this button.

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 reflection hood.

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



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.

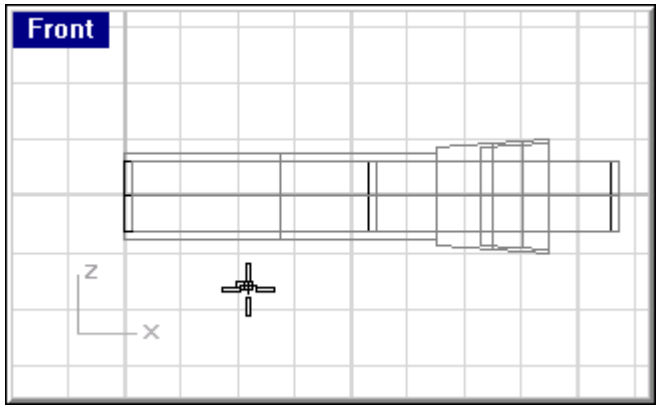
2 Turn **Ortho** on, if it is not already.

3 Turn off the Center object snap.

Sometimes having object snaps on interferes with your ability to click and drag freely.

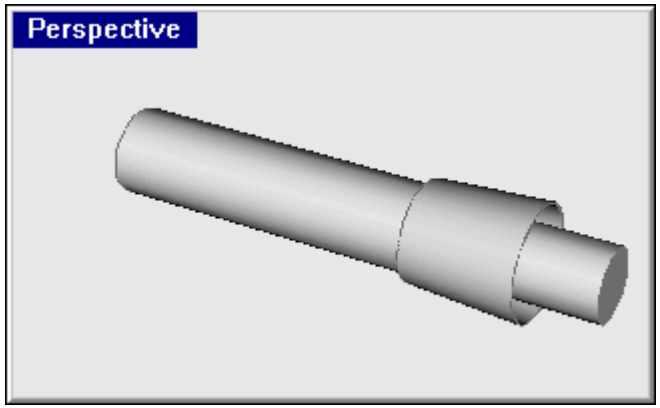
Notes:

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.



5 From the **View** menu, click **Zoom**, then click **Extents All**.

6 **Shade** the **Perspective** view.



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.

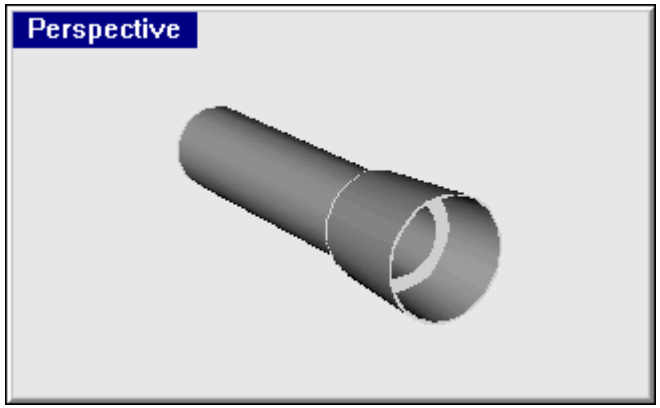
Notes:

Zoom Extents All Viewports
Look for this button.

Boolean Difference
Look for this button.

3 At the **Select surface or polysurface to subtract** prompt, select the new cylinder.

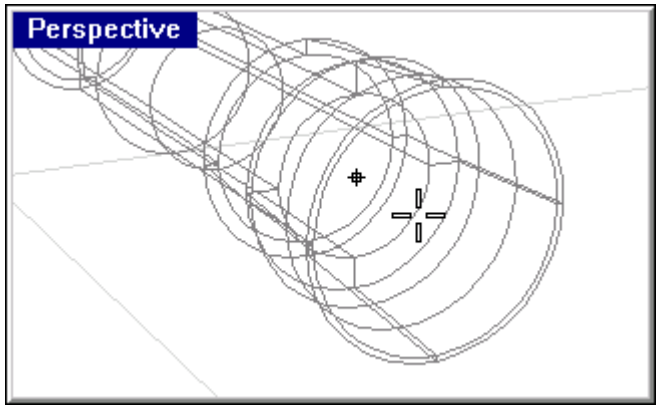
4 **Shade** the **Perspective** view.



To draw the lens:

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

2 At the **Base point** (Vertical) prompt, snap to the center of the inside edge of the truncated cylinder.



3 At the **Radius <0.625>** (Diameter) prompt, type **.625** and press ENTER.

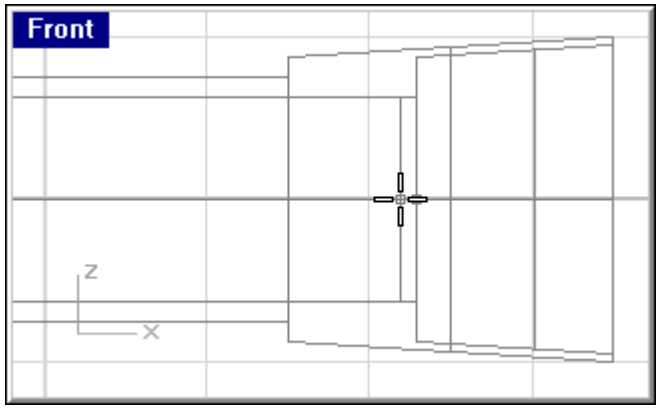
4 At the **Choose a point** prompt, in the **Front** viewport pick a point slightly to the left of the center point.

Notes:

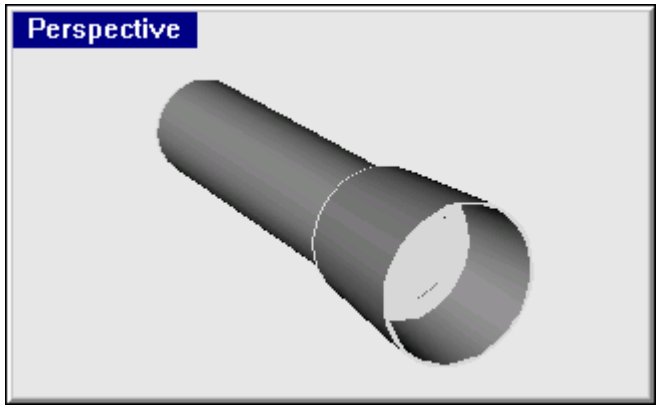
Cylinder
Look for this button.

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 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 partially transparent.



To draw the switch button:

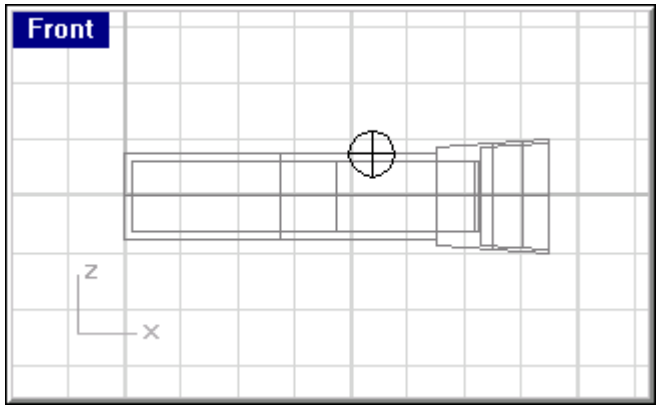
1. From the **Solid** menu, click **Sphere**, and then click **Center, Radius**.
2. At the **Center point** prompt, in the **Front** viewport pick a point near the top of the cylinder.

Notes:

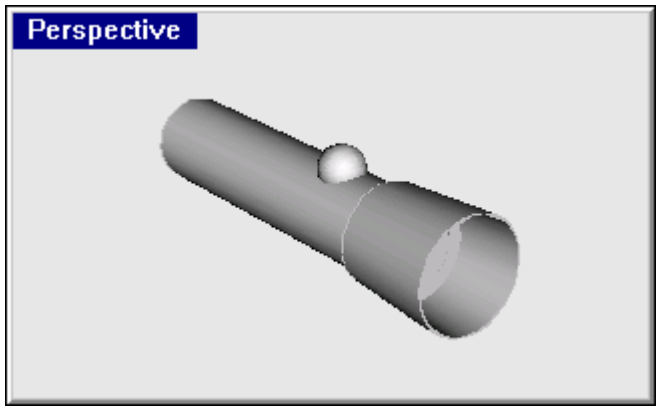
Sphere
Look for this button.

3 At the **Radius** (Diameter) prompt, pick a radius.

You can type **.4**, or just drag out a size that looks good to you.



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



Render the Flashlight

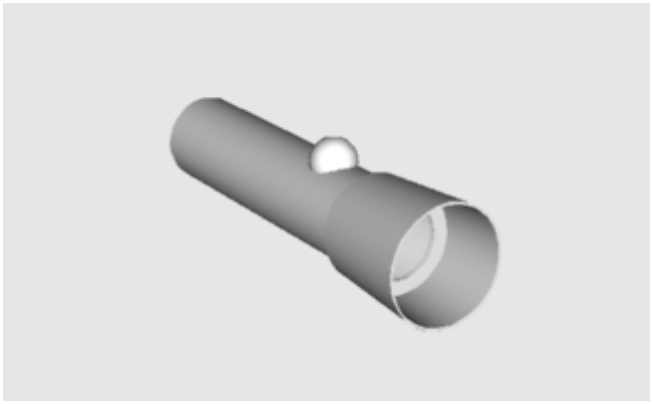
You can use Rhino to add color, texture, and lights to your model to make simple presentations. In this exercise you will add only color to the flashlight parts and use Rhino's rendering capability to create a more realistic picture.

Notes:

To render the model:

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

   A separate **Display Window** appears with the model rendered in default gray.



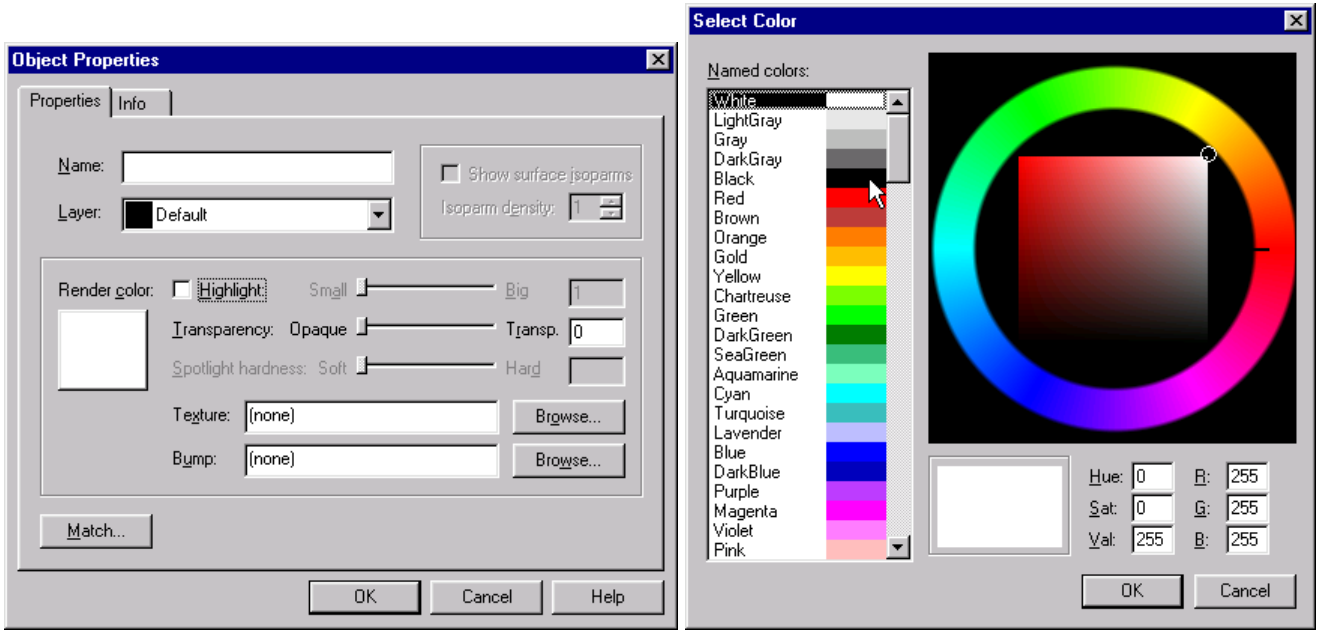
To set the color for the flashlight body:

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.

Notes:

Render
Look for this button.

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



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**.

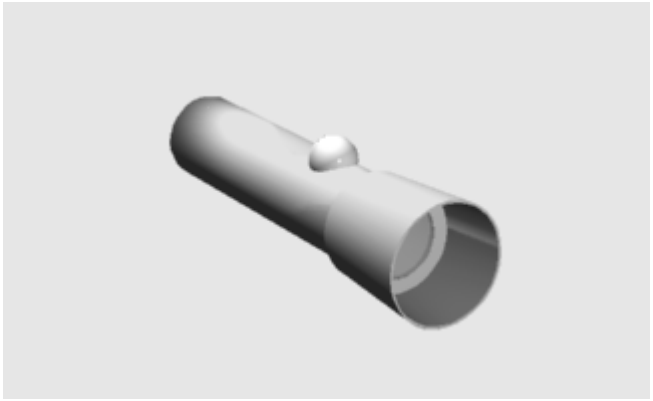
3 In the Object Properties dialog box, drag the **Transparency** slider to the right.

Don't move it all the way to the right or the lens will disappear completely. You want it to be a little opaque so you can still see it.

Notes:

Render
Look for this button.

4 From the **Render** menu, click **Render**.



Notes:

Notes:

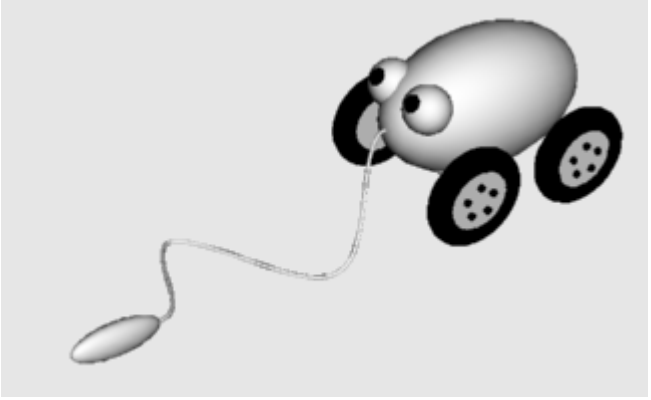
Modeling a Pull Toy

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

Drawing 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 your cursor 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. The advantage this gives you is that you can enter your points without restricting them to the construction plane.



The finished Pull Toy

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

Exercise 52—Creating solids.

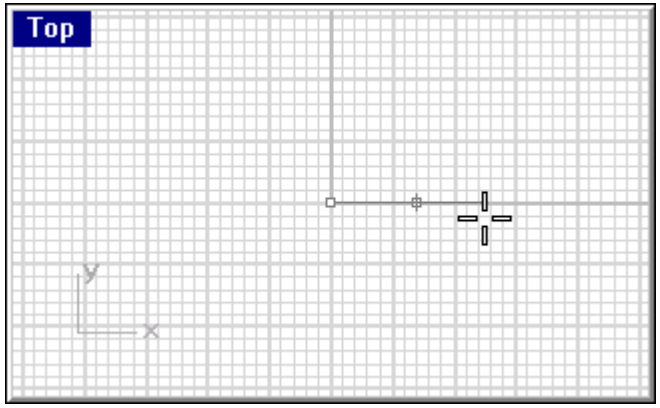
To start the model:

Start a new model. Save as **Pull Toy**.

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 pick to show the direction.



6. At the **End of second axis** prompt, type **4** and press ENTER.

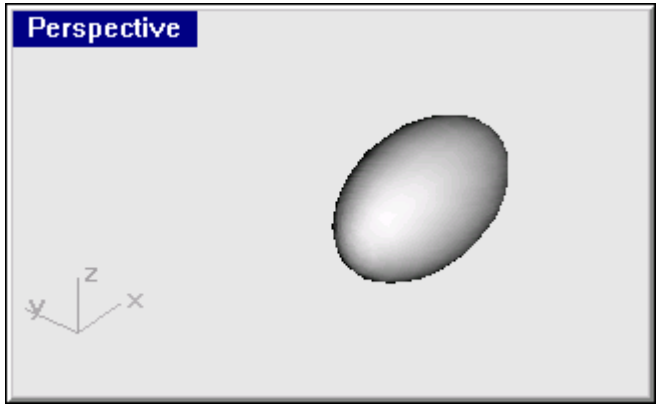
   This sets the width of the ellipsoid.
7. At the **End of third axis** prompt, type **4.5** and press ENTER.

   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.

Notes:

Ellipsoid
Look for this button.

9 **Shade** the **Perspective** view.



Drawing 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 point (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.

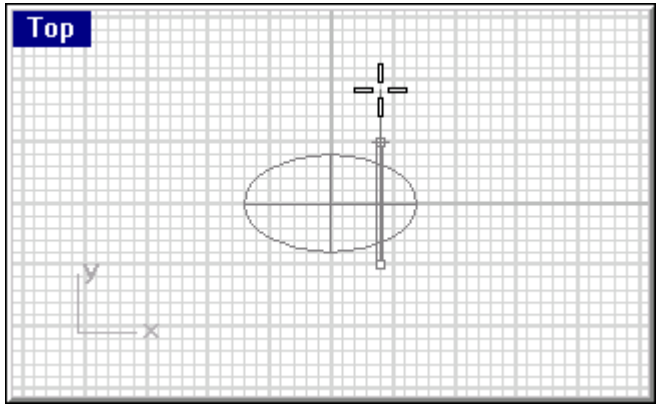
3 At the **Radius (Diameter)** prompt, type **.25** and press ENTER.

4 At the **End of cylinder** prompt, type **10** and press ENTER.

Notes:

Cylinder
Look for this button.

5 In the **Top** viewport drag the cylinder as shown to place the axle.



To create a wheel hub:

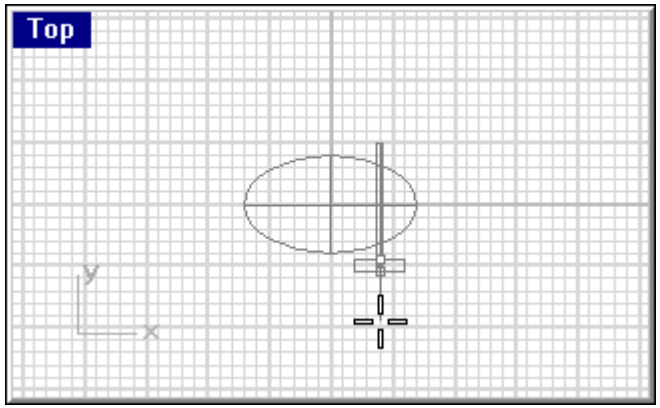
1 From the **Solid** menu, click **Cylinder**.

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.

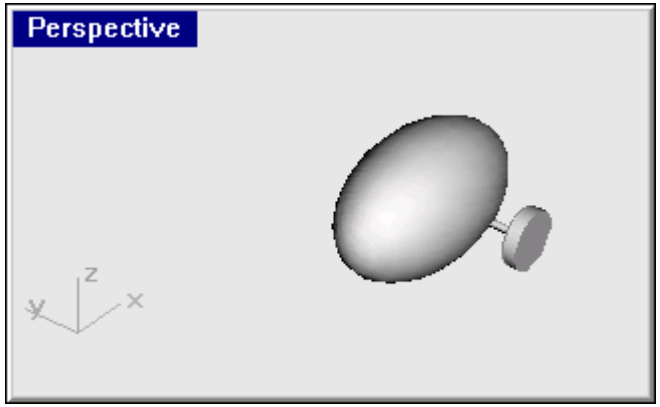
5 At the next **End of cylinder** prompt, pick in the **Top** viewport to place the hub.



Notes:

Cylinder
Look for this button.

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



Draw the Lug Nuts

You will make the lug nuts by extruding a hexagonal polygon. Polygons are ***curves*** in Rhino.

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.

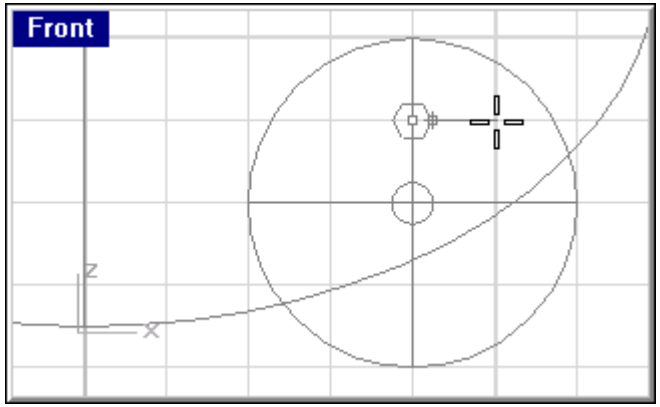
3 In the Front viewport, at the **Center of polygon (NumSides=6 Circumscribed)** prompt, type **4,4,5.5** and press ENTER.

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

Notes:

Polygon
Look for this button.

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



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, make sure **Cap=yes**.

If it does not, type **C** and press ENTER.

This is an example of a ***command option***. Many commands have options. You will learn how to change and use them as you learn to use the commands. Take a moment and look at the options available for the Extrude command.

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

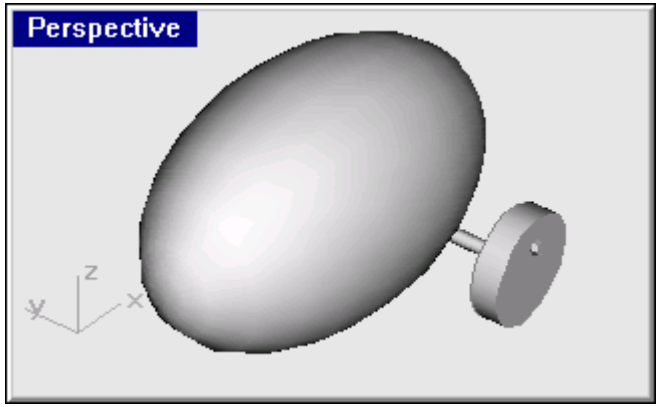
4 At the **Extrusion distance (Direction Cap=yes Bothsides Tapered)** prompt, type **.25** and press ENTER.

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.

Notes:

Extrude
Look for this button.

5 **Shade** the **Perspective** viewport.



Assign Colors

Now that you have the basic parts built, you are going to assign render 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 render colors now, the color property will be copied when we copy the parts.

To assign render color to the parts:

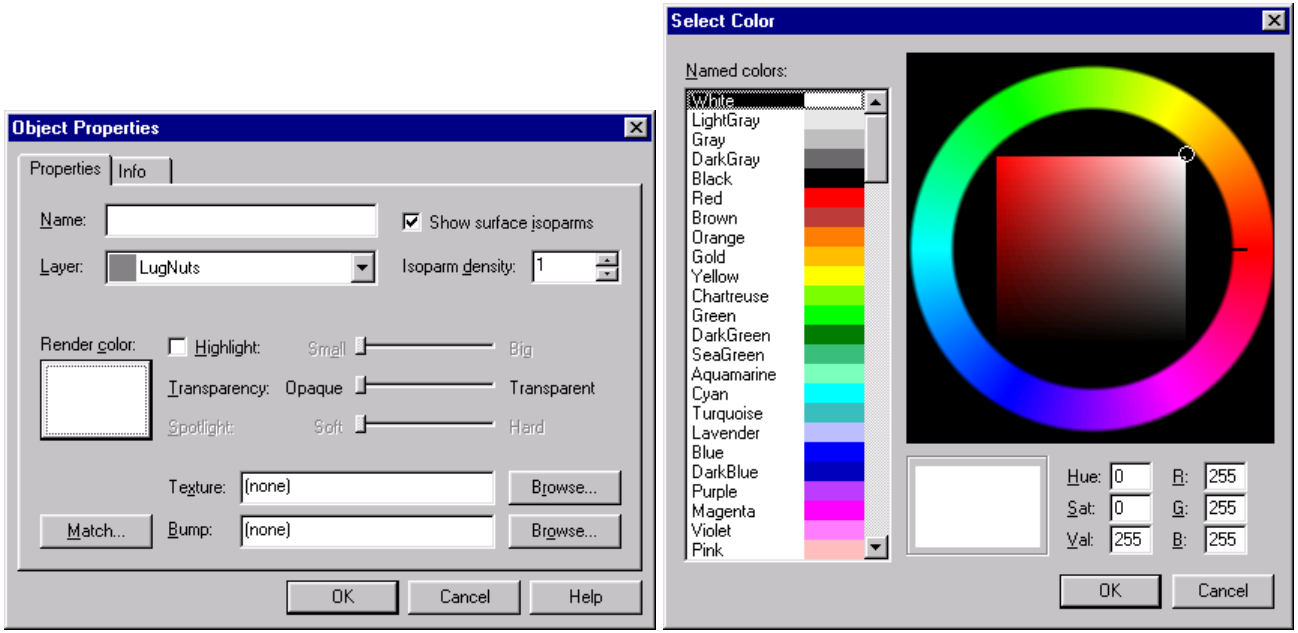
1 Select the lug nut.

2 From the **Edit** menu, click **Object Properties**.

3 In the Object Properties dialog box, under **Render color**, click the color swatch as shown.

Notes:

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



The Select Color dialog box.

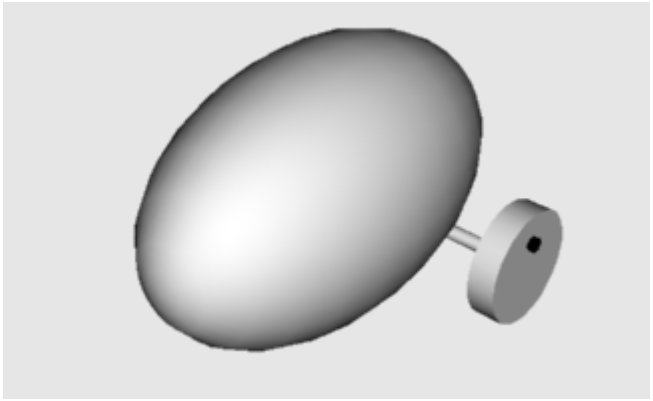
5 In the Object Properties dialog box, click **OK**.

6 Follow steps 1 through 6 to set a render color for the toy body.

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

Notes:

7 **Render** the **Perspective** viewport.



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. They can be rotated as they are copied.

To array the nuts:

1 Select the lug nut.

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***.)

2 From the **Transform** menu, click **Array**, and then click **Polar**.

3 At the **Center point** prompt, snap to the center of the hub.

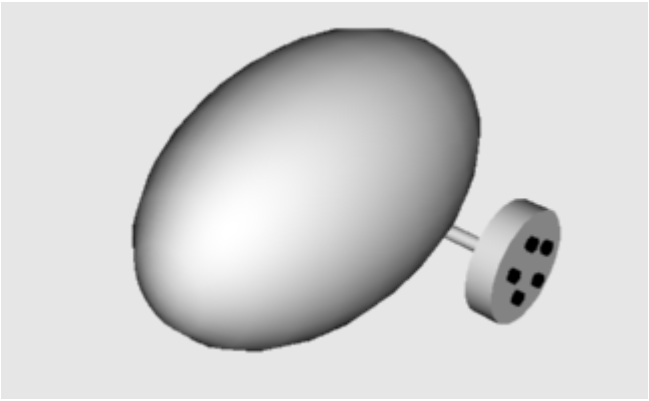
4 At the **Number of elements <1>** prompt, type **5** and press ENTER.

5 At the **Angle to fill <360>** prompt, press ENTER.

Notes:

Polar Array
Look for this button.

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



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 radius of the torus 0.5 units larger than the wheel hub.

4 At the **Second radius (Diameter)** prompt, type **.75** and press ENTER.

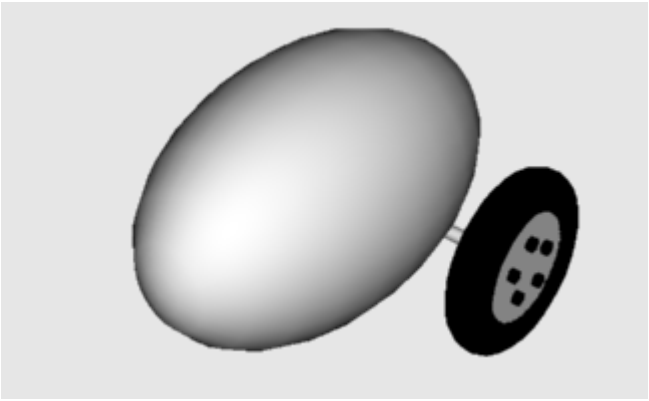
This makes the hole radius .25 units smaller than the wheel hub.

5 Set the render **Color** of the tire to **Black**.

Notes:

Torus
Look for this button.

6 **Render** the **Perspective** viewport.

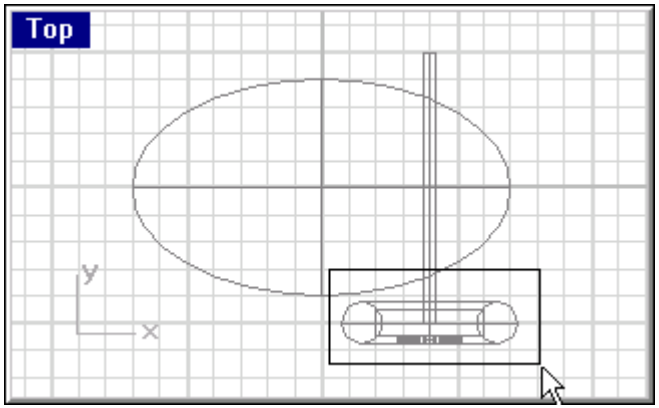


Mirror the Wheels

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, tire, and lug nuts as shown.



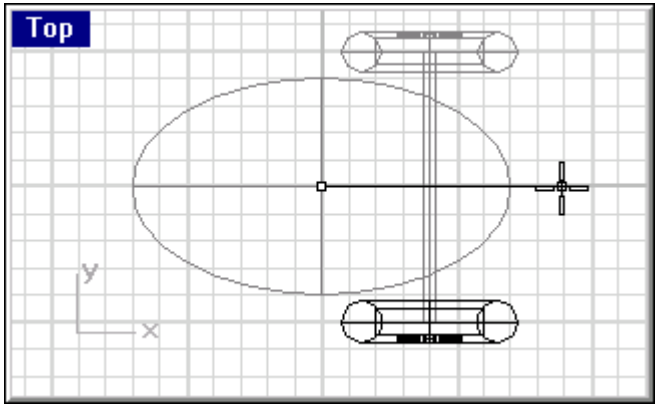
2 From the **Transform** menu, click **Mirror**.

3 At the **Start of mirror plane** prompt, type **0,0,0**.

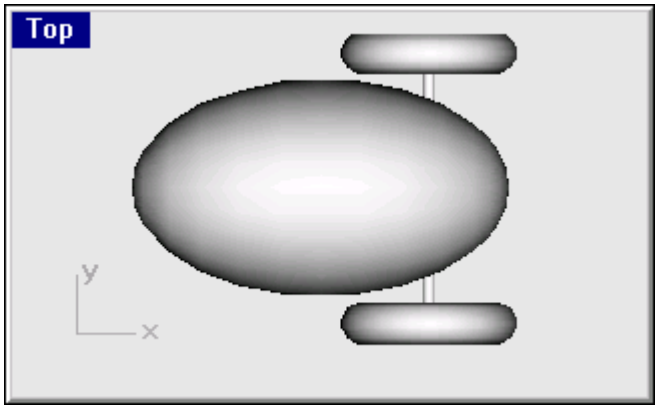
Notes:

Mirror
Look for this button.

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



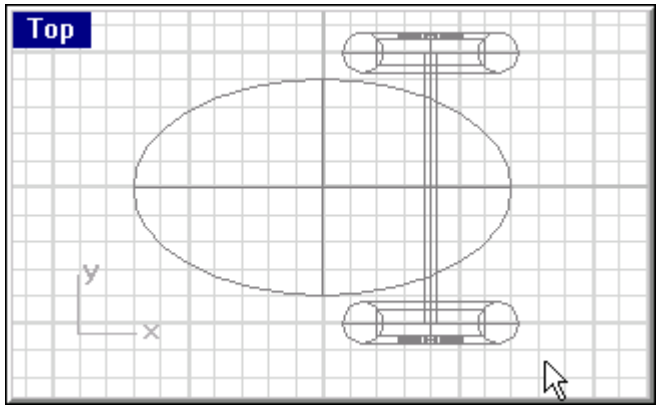
5 **Shade** the **Top** viewport.



Notes:

To mirror the front wheels and axle:

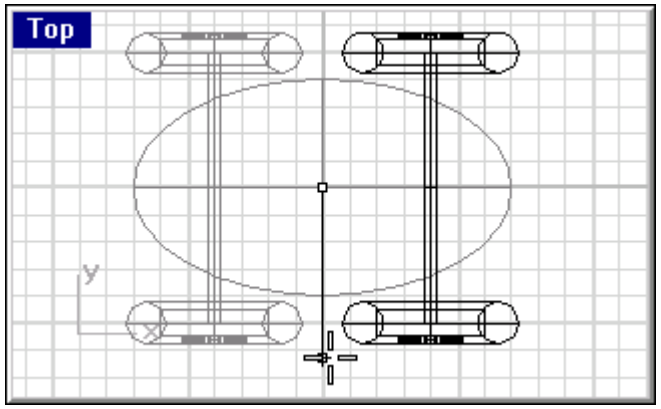
1 In the **Top** viewport, use a window to select the back wheels and axle as shown.



2 From the **Transform** menu, click **Mirror**.

3 At the **Start of mirror plane** prompt, type **0,0,0**.

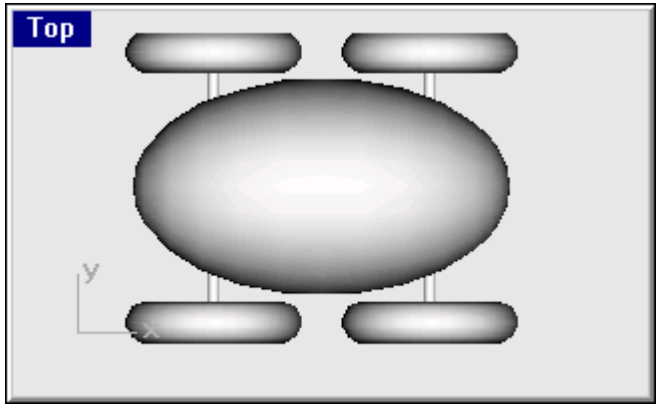
4 In the **Top** viewport, with **Ortho** on, drag down as shown and click.



Notes:

Mirror
Look for this button.

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



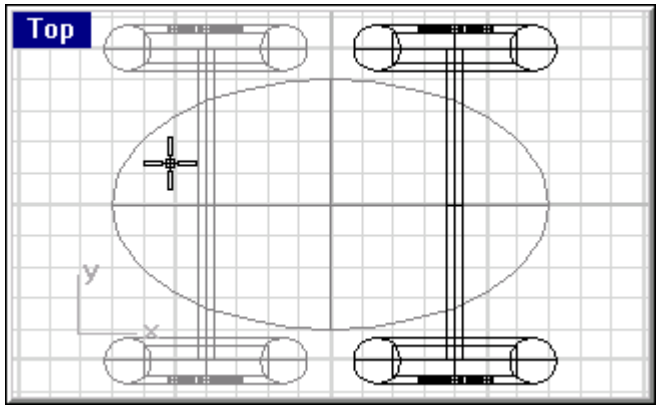
Draw the Eyes

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 of sphere** prompt, in the **Top** viewport hold CTRL and pick near the front edge of the ellipsoid.

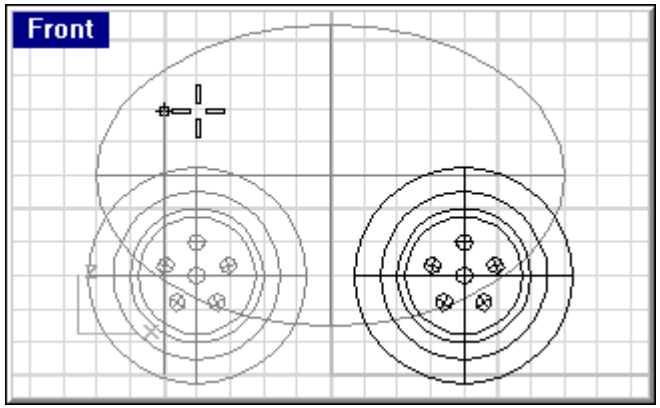


Notes:

Sphere
Look for this button.

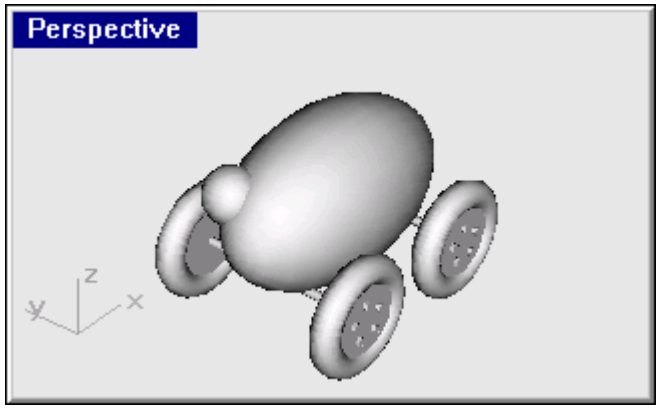
This will start elevator mode.

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



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

5 **Shade** the **Perspective** viewport.

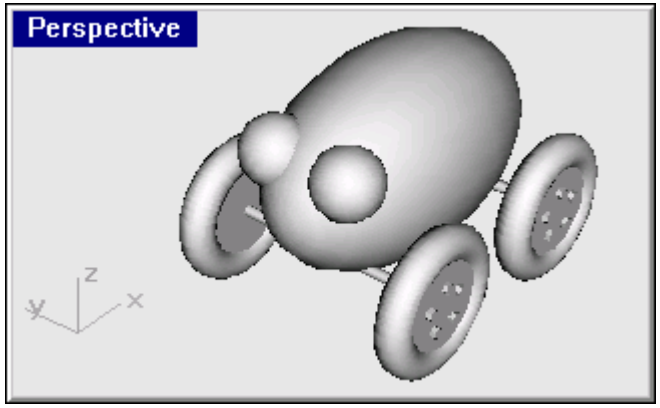


6 In the **Top** viewport, **Mirror** the sphere to the other side of the ellipsoid.

Use the same procedure you used to mirror the wheel hub.

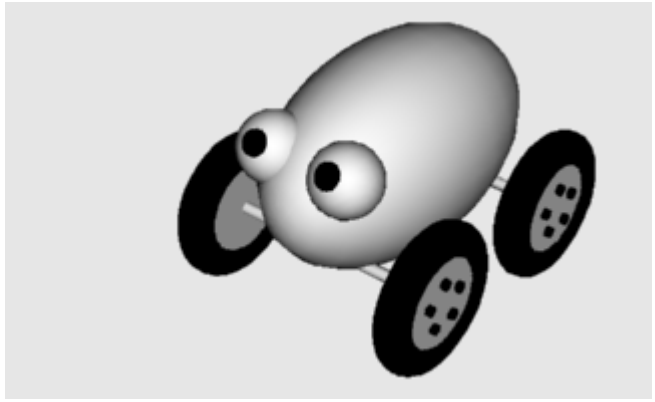
Notes:

7 **Shade** the **Perspective** viewport.



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.

9 **Render** the **Perspective** viewport.



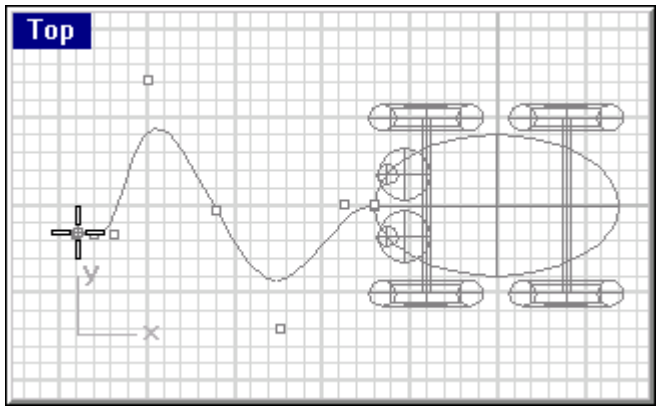
Make the Pull Cord

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

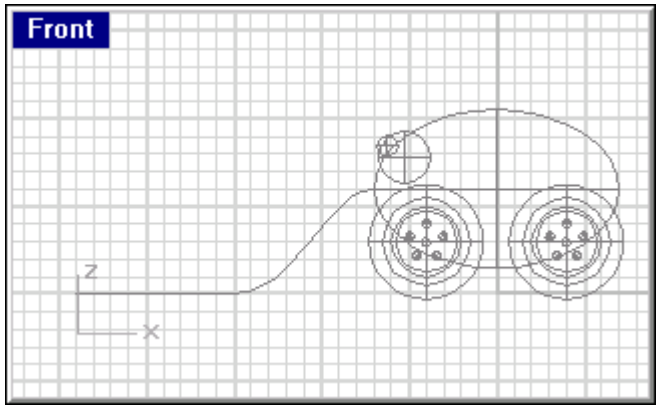
Notes:

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 **Freeform**, and then click **Control Points**.
3. At the **Start of curve** prompt, in the **Top** viewport, snap **Quad** to the front end of the ellipsoid.



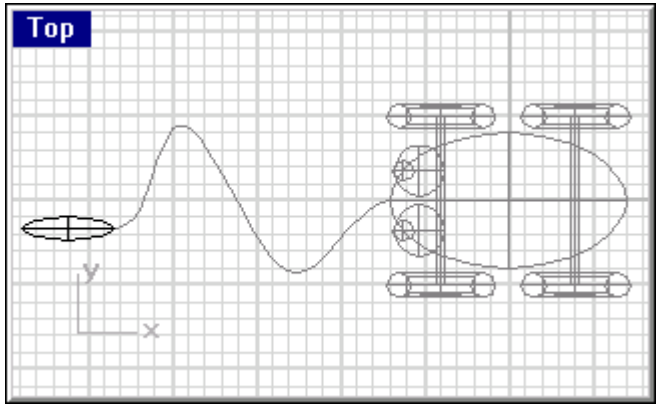
4. Continue picking points until you create a curve to represent the cord.



Notes:

Curve
Look for this button.

5 Draw an **Ellipsoid** to represent a handle at the end of the curve.



Make the cord fat:

1 From the **Solid** menu, click **Pipe**.

2 At the **Select curve to create pipe around** prompt, choose the curve you just created.

3 At the **Starting radius <1>** (Diameter Cap=yes Thick=No) prompt, type **.125** and press ENTER.

4 At the **End radius <1>** (Diameter) prompt, press ENTER.

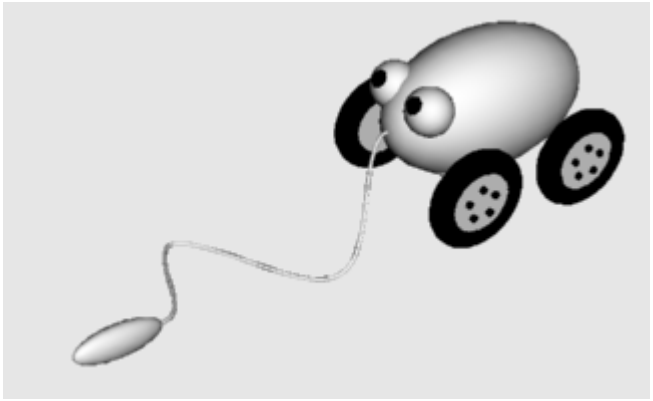
The pipe will be the same diameter for the full length of the curve.

5 Change the cord color to black and the handle color to any color you like.

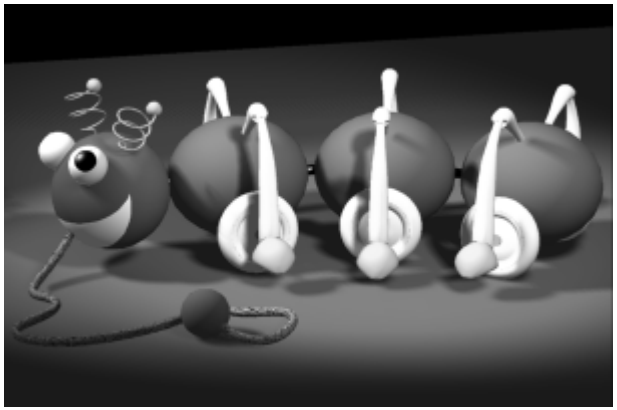
Notes:

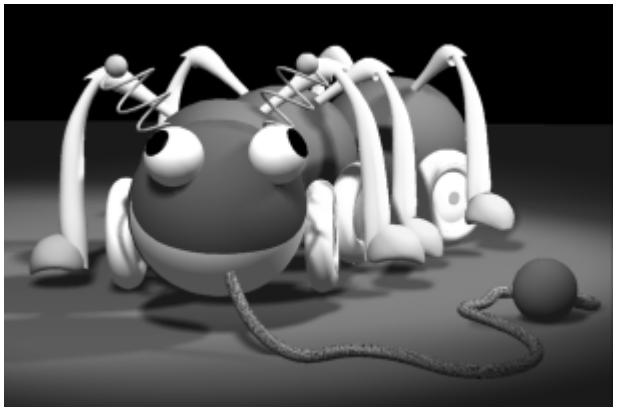
Pipe
Look for this button

6 **Render** the **Perspective** view.



Another example of a Pull Toy created with solids and surfaces. We will learn more about surfaces in another chapter.





Insect Pull Toy, modeled by Grant Lange.

Notes:

Making Solid Text

Creates two or three-dimensional text from True Type fonts, in the form of curves, surfaces, or solids.

To create text using 3-D solids:

1. Start a new model. Save as **Text**.
2. Click in the **Top** viewport.
3. From the **Solid** menu, click **Text.**

   The Add Text dialog appears.

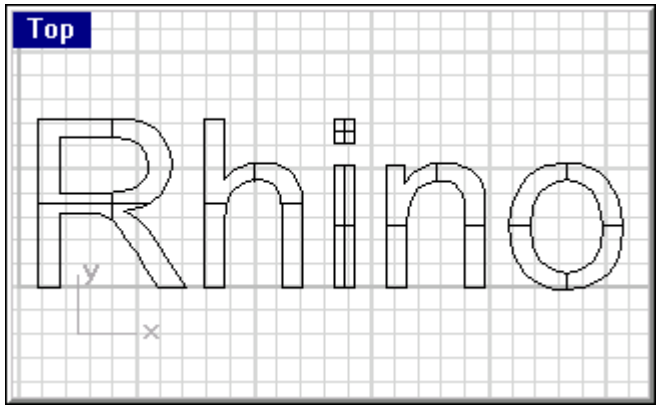
4. In the **Add Text** dialog box, in the **Font** box, select a font.
5. In the **Font Style** box, select a style.
6. In the **Text** box, type some new text.
7. Click **Create Solids**.
8. In the **Height** box, type the height of the text.
9. In the **Extrude Depth** box, type a thickness for your solid text, then click **OK**.

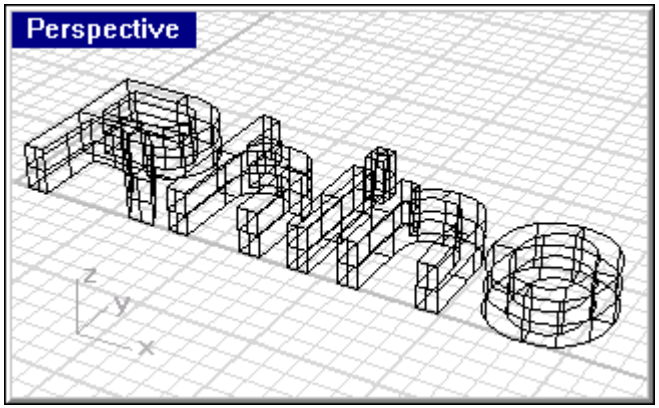
Notes:

Text
Look for this button

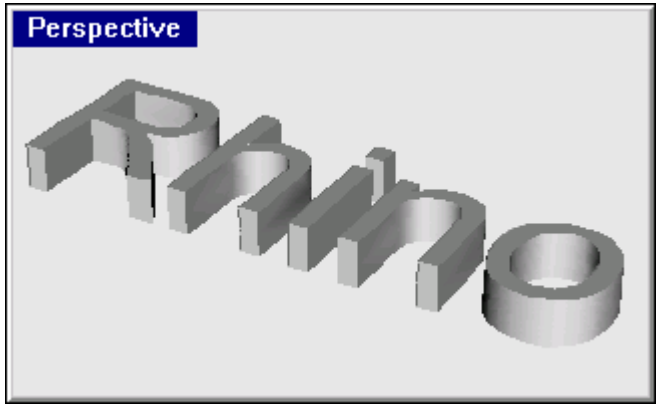
10 From the **View** menu, click **Zoom**, then click **Extents All.**

The text is inserted perpendicular to the active viewport, starting at 0,0.





11 From the **Render** menu, click **Shade**.



Notes:

10 Creating Surfaces

Surfaces

A Rhino surface is similar to a piece of stretchy fabric. It can take on many different shapes.

Surfaces are bounded by curves called edges. To visualize the surface shape Rhino displays a grid of isoparms on the surface.

Surfaces have an area, their shape can be changed by moving control points, and they can be meshed.

Button	Command	Description
	SrfPt	Creates a surface by picking three or four points in space for corners.
	EdgeSrf	Creates a surface by selecting two, three or four existing curves whose ends meet exactly.
	PlanarSrf	Creates a surface from planar curves that enclose an area.
	Patch	Creates a surface that goes through a set of curves and/or point objects
	Revolve	Revolve a curve around an axis to create a surface.
	Loft	Creates a surface from shape curves; the normal option makes a surface with no creases as it passes over the shape curves.

Notes:

Button	Command	Description
		The Straight Sections option creates a surface with creases at each shape curve and straight sections between the shapes curves.
	Sweep1	Creates a surface from shape curves that follows along a rail curve. This command controls one edge of the surface.
	Sweep2	Creates a surface from shape curves that follows along two rail curves. The two-rail sweep allows you to control the surface's edges.
	FilletSrf	Creates a fillet or round between two surfaces.
	FilletEdge	Creates a fillet or round on the edge of a polysurface or solid.
	BlendSrf	Makes a smooth surface between two existing surfaces
	RailRev	Revolves a shape curve holding one end along a rail curve. This command is very useful for putting a smooth end cap on an irregularly shaped surface.
	Extrude	Extrudes a curve perpendicular to the construction plane with option to taper the surface with a draft angle.
	ExtrudeAlongCrv	Extrudes a curve following along a second curve.
	ExtrudeToPt	Extrudes a curve to a point.
	Plane	Creates a rectangular planar surface parallel to the construction plane from two diagonal points.
	Plane3Pt	Creates a rectangular planar surface from 3 points.
	PlaneV	Creates a rectangular planar surface from 3 points that is vertical to the construction plane.

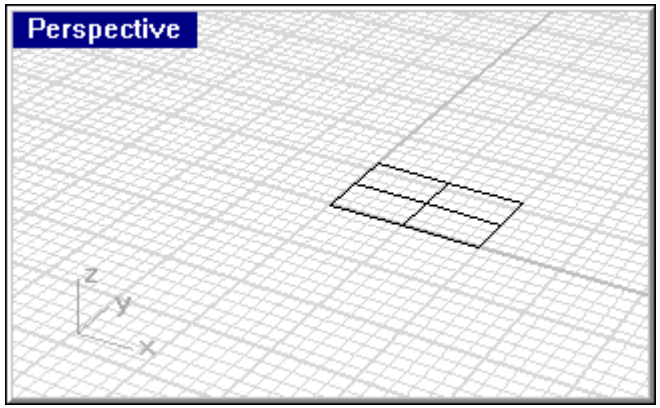
Notes:

Exercise 53—Basic techniques for making surfaces

- Start a new model. Save as **Surfaces**.
  In this exercise we will model some simple surfaces.

To create a rectangular plane:

1. From the **Surface** menu, click **Rectangle**, then click **Corner to Corner.**
2. In the **Top** viewport, at the **First corner of plane** prompt, type **0,0** and press ENTER.
3. At the **Other corner or length** prompt, type **10** and press ENTER.
4. At the **Width** prompt, type **6** and press ENTER.



To create a vertical plane:

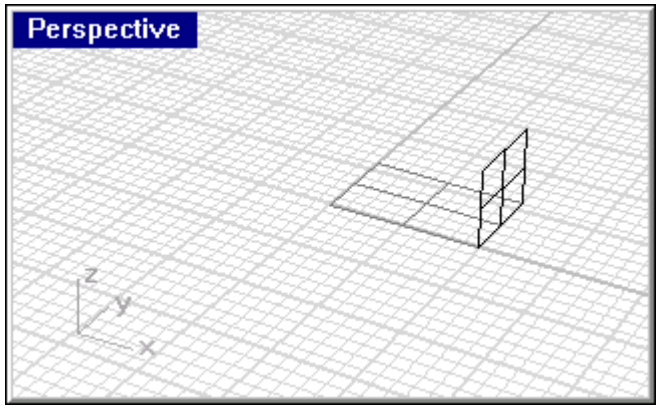
1. From the **Surface** menu, click **Rectangle**, then click **Vertical.**
2. At the **Start of edge** prompt, snap to the **End** point at the right side of the surface.
3. At the **End of edge** prompt, snap to the other **End** point at the right side of the surface.

Notes:

Plane
Look for this button

Plane Vertical
Look for this button

4 At the **Height of rectangle** prompt, type **5** and press ENTER.



To create a plane from 3 points:

1 From the **Surface** menu, click **Rectangle**, then click **3 Points.**

2 At the **Start of edge** prompt, snap to the endpoint at the top of the second surface.

3 At the **End of edge** prompt, snap to the other endpoint at the top side of the second surface.

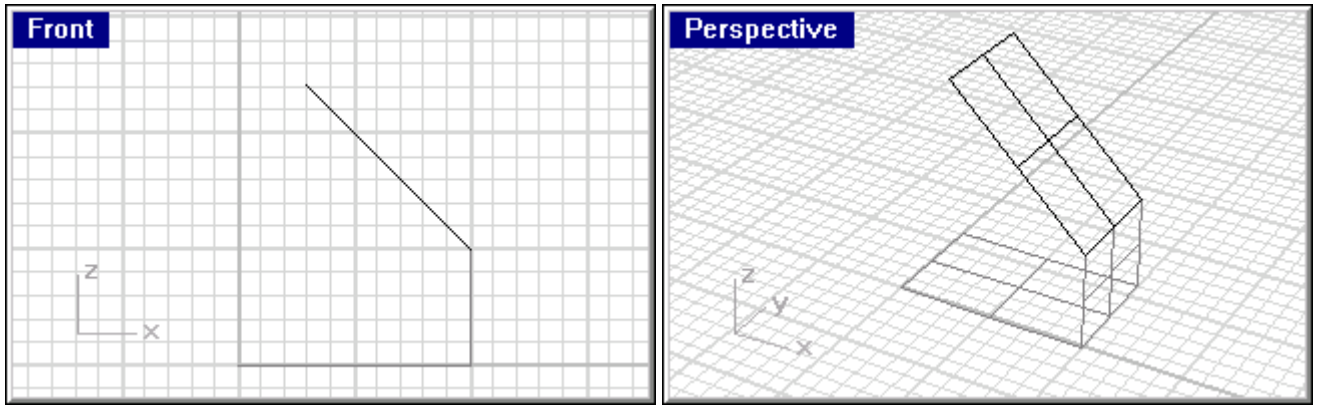
4 At the **Width** prompt, type **10** and press ENTER.

5 At the **Width** prompt, type **<45** and press ENTER.

Notes:

Plane 3 Points
Look for this button

6 At the **Width** prompt, pick a point in the front viewport.



To create a plane from points:

1 From the **Surface** menu, click **Corner Points.**

2 At the **First corner of surface** prompt, snap to an **End** point at the left edge of the first surface.

3 At the **Second corner of surface** prompt, snap to the other **End** point at the left edge of the first surface.

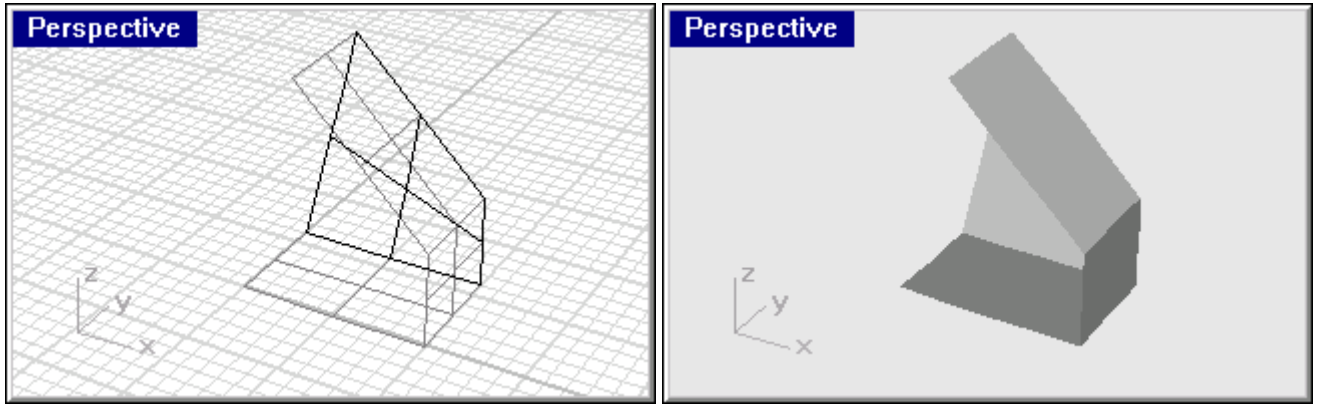
4 At the **Third corner of surface** prompt, snap to an **End** point at the left edge of the third surface.

Notes:

Surface from 3 or 4 Corner Points
Look for this button

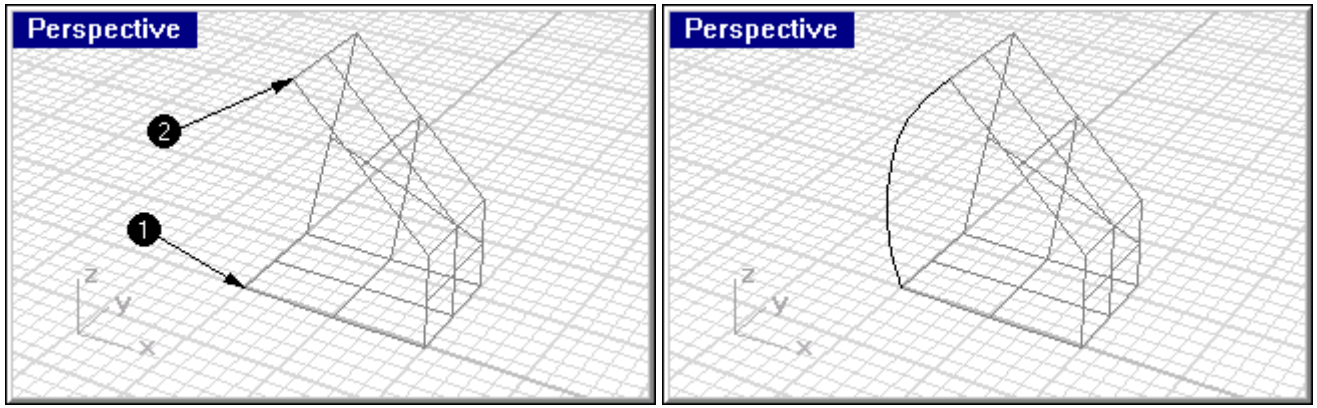
5 At the **Fourth corner of surface** prompt, snap to the other **End** point at the left edge of the third surface.

A surface will be created with corners at the points you selected.



To create a plane from curves:

1 Draw a curve between the two open corners in the **Front** viewport.



2 From the **Surface** menu, click **Edge Curves.**

3 At the **Choose 2, 3, or 4 curves** prompt, select the near edge of the first surface you created.

4 At the **Choose 2, 3, or 4 curves** prompt, select the of the curve you created.

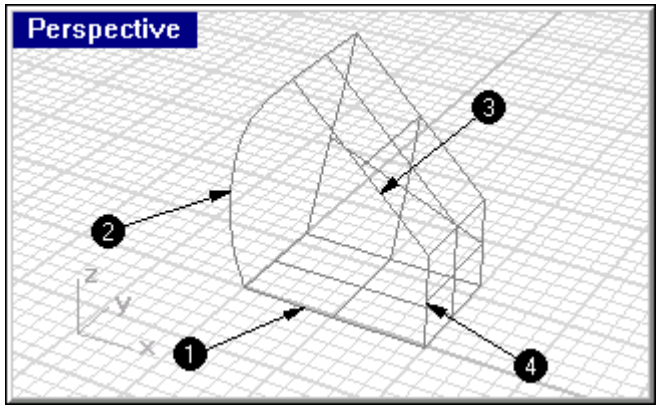
Notes:

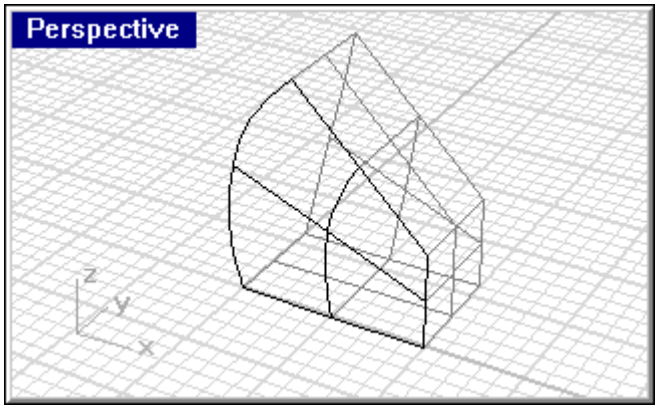
Surface from 2, 3 or 4 Edge Curves
Look for this button

5 At the **Choose 2, 3, or 4 curves** prompt, select another edge.

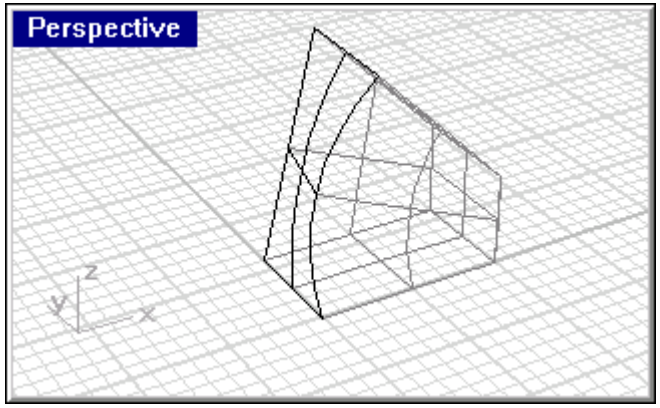
6 At the **Choose 2, 3, or 4 curves** prompt, select another edge.

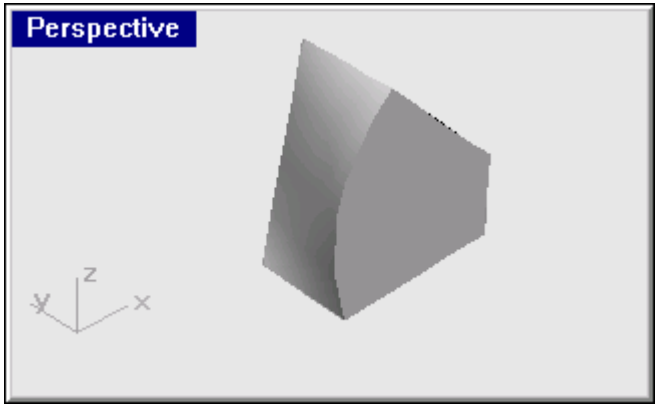
A surface is created.





7 Repeat this process for the open end by picking four edges.

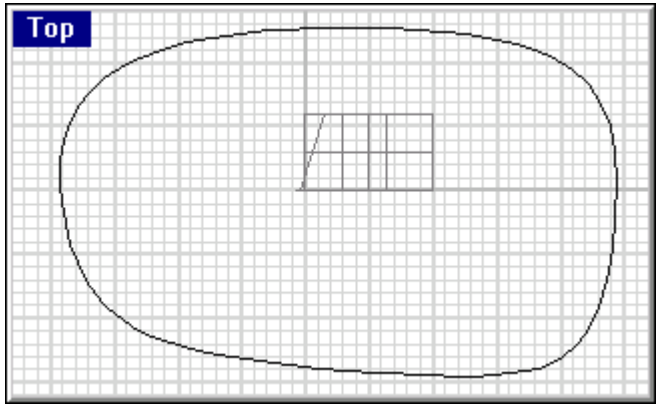




Notes:

To create a surface from a closed planar curve:

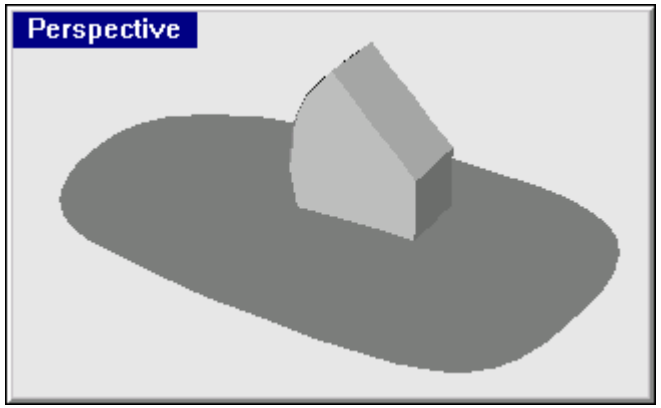
1 Draw a **Curve from Control Points** in the **Top** viewport. Make sure it is closed.



2 Select the curve.

3 From the **Surface** menu, click **From Planar Curves.**

A surface is created.



Notes:

Curve
Look for this button

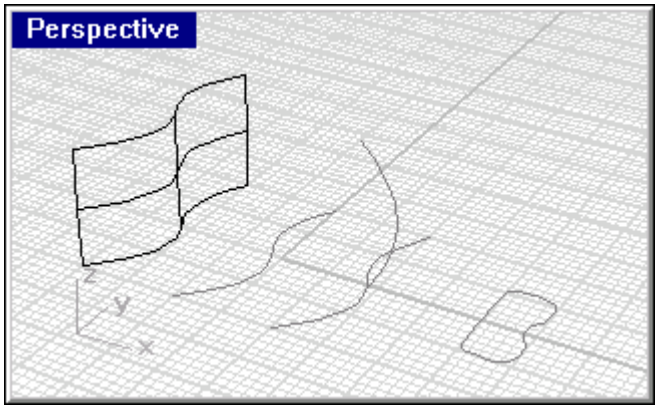
Surface from Planar Curves
Look for this button

Exercise 54—Extruding Surfaces

To create an extruded surface from a curve:

1. Open the **Surface Extrude** model.
2. Select the curve on the left.
3. From the **Surface** menu, click **Extrude**, then click **Straight.**
4. At the **Extrusion distance (Direction BothSides Tapered)** prompt, type **12** and press ENTER.

   The curve is extruded straight up from the construction plane.



5. Press ESC to unselect the curve.

To create an extruded surface from a curve to a point:

1. From the **Surface** menu, click **Extrude**, then click **To Point.**
2. At the **Select curve to extrude** prompt, pick the next curve.

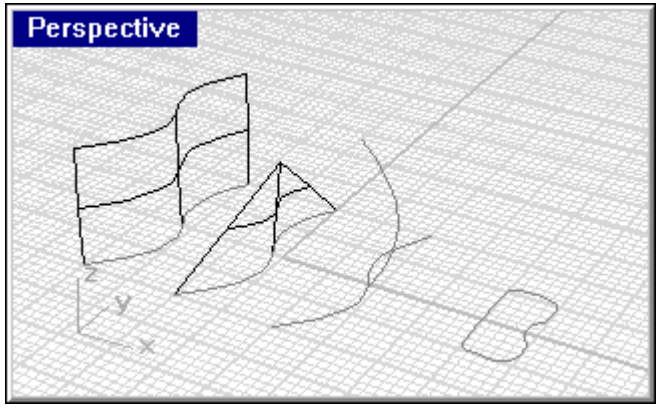
Notes:

Extrude
Look for this button

Extrude To Point
Look for this button

3 At the **Point to extrude to** prompt, pick a point in the front viewport above the x plane.

The curve is extruded to a point.

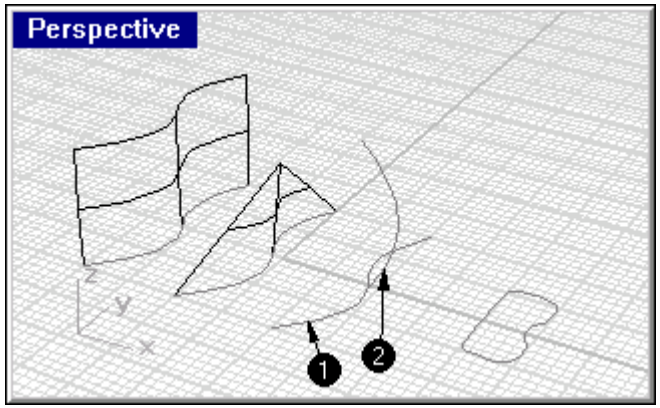


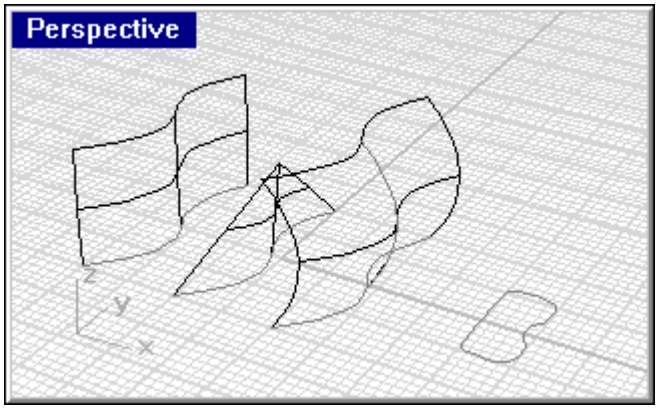
To extrude a curve along another curve:

1 From the **Surface** menu, click **Extrude**, then click **Along Curve.**

2 At the **Select shape curve** prompt, pick the next curve in the **top** view.

3 At the **Select path curve** prompt, pick the path curve near its bottom end in the **Front** viewport.





Notes:

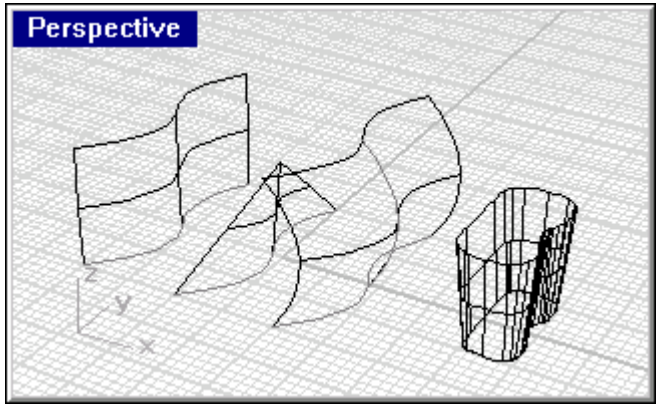
Extrude Along Curve
Look for this button

If you don't get the answer you expected, undo and try picking near the other end of the path curve.

To extrude a curve with a taper (draft angle):

1. Select the closed curve on the right.
2. From the **Surface** menu, click **Extrude**, then click **Straight.**
3. At the **Extrusion distance <12> (Direction Cap=no Bothsides Tapered)** prompt, type **t** and press ENTER.
4. At the **Draft angle <0>** prompt, type **5** and press ENTER.
5. At the **Extrusion distance <12> (Direction Cap=no)** prompt, type **10** and press ENTER.

   The curve is extrude with a ten degree draft angle.



Exercise 55—Lofted surfaces

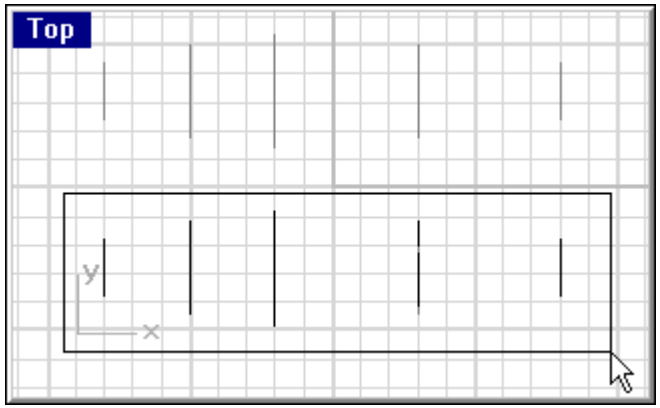
To create a lofted surface:

1. Open the **Loft** model.

Notes:

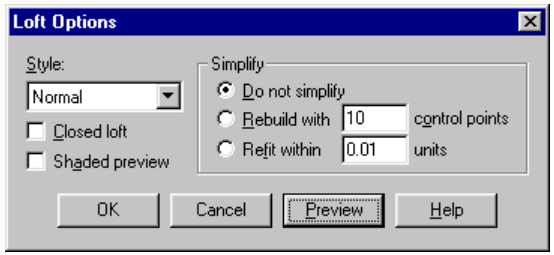
Extrude
Look for this button

2 Window select all of the curves at the lower part of the Top viewport.



3 From the **Surface** menu, click **Loft.**

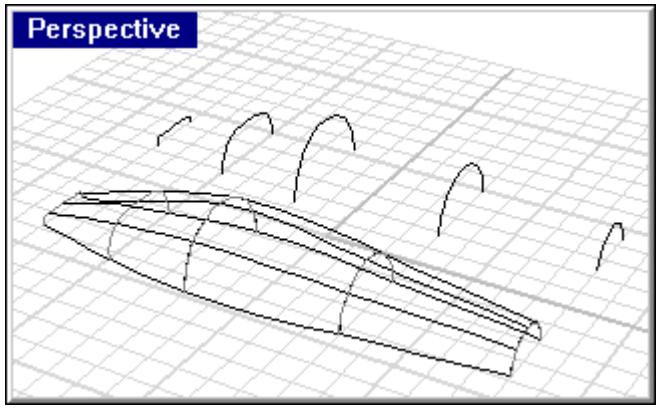
4 In the **Loft Options** dialog, **OK**.



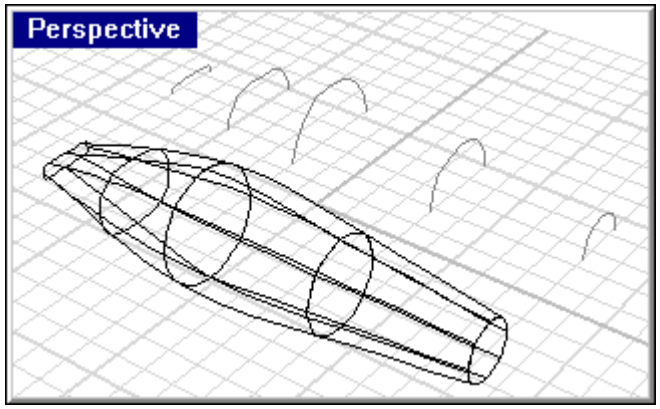
Notes:

Loft
Look for this button

A surface is fitted over the curves.

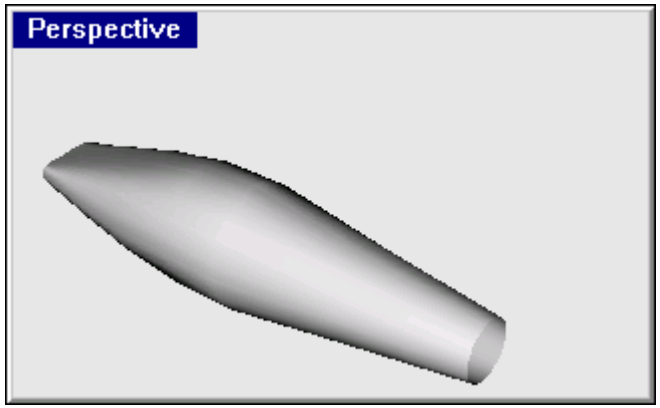


5 **Mirror** the surface to create the other half.



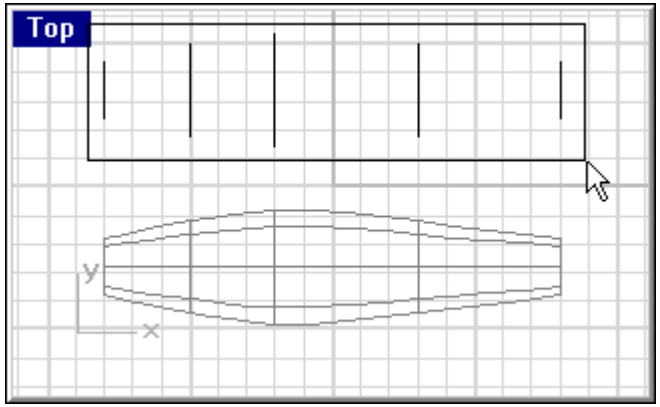
Notes:

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



To create a lofted surface with straight sections:

1 Window select all of the curves at the top part of the Top viewport.

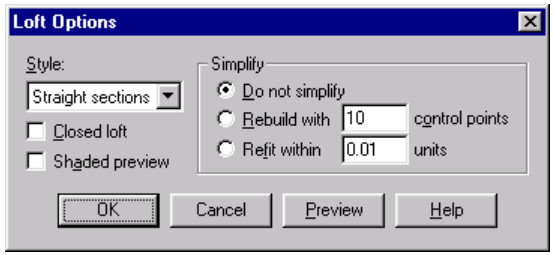


2 From the **Surface** menu, click **Loft.**

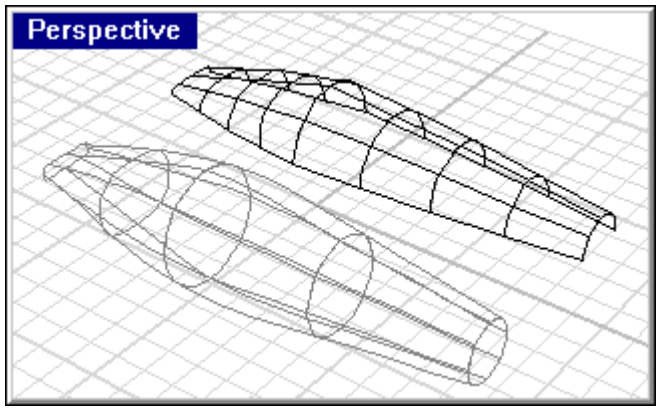
Notes:

Loft
Look for this button

3 In the **Loft Options** dialog, switch **Style** to **Straight sections**, then click **OK**.

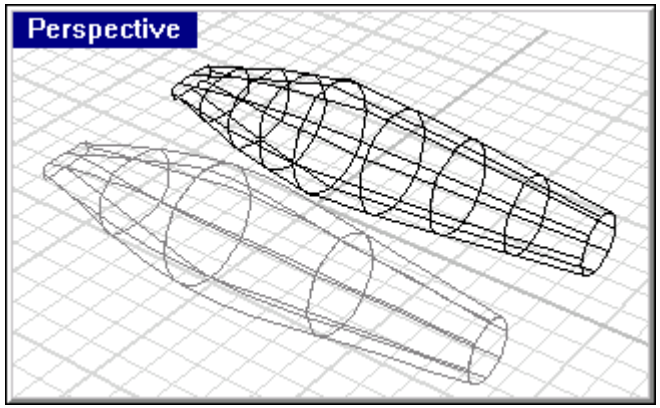


A surface is fitted over the curves.

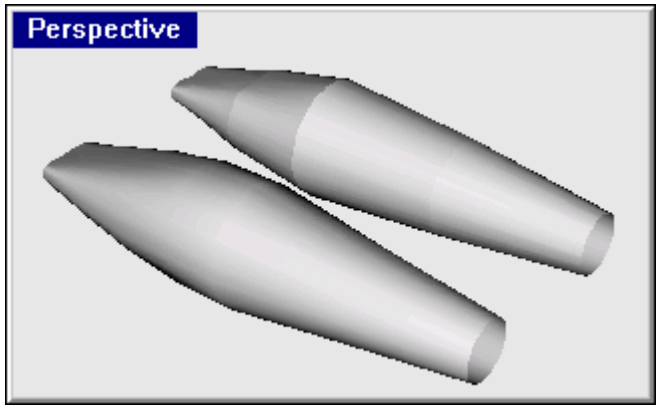


Notes:

4 **Mirror** the surface to create the other half.



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



Exercise 56—Revolved surfaces

To create a revolved surface:

1 Open the **Revolve** model.

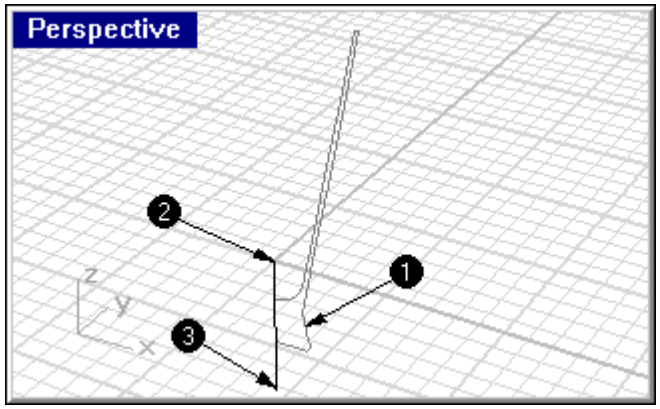
Notes:

2 From the **Surface** menu, click **Revolve.**

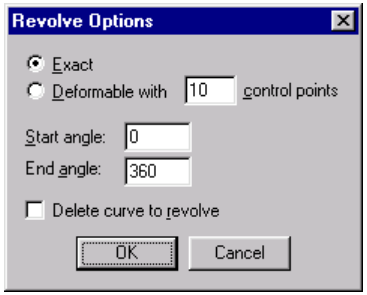
3 At the **Select curve to revolve** prompt, select the curve.

4 At the **Start of revolve axis** prompt, pick the end of the red vertical line.

5 At the **End of revolve axis** prompt, pick the other end of the red vertical line.



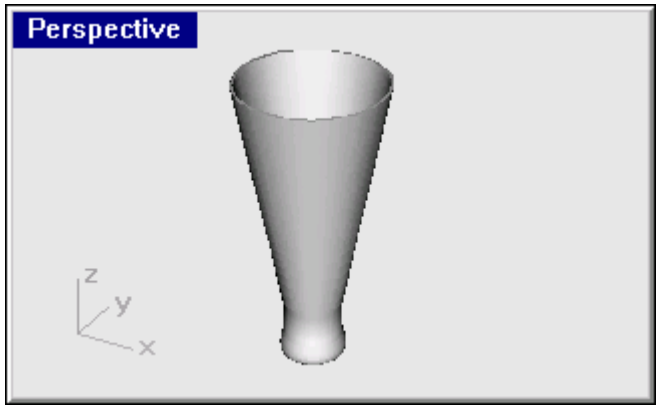
6 In the **Revolve Options** dialog, click **OK**.



Notes:

Revolve
Look for this button

A surface is revolved around the axis line.



Exercise 57—Using a rail revolve

There are times when you need to put a cap on the end of an irregular shaped object. Revolve lets you revolve a profile curve in a circular shape. Rail revolve allows you to revolve around an axis and along a path curve.

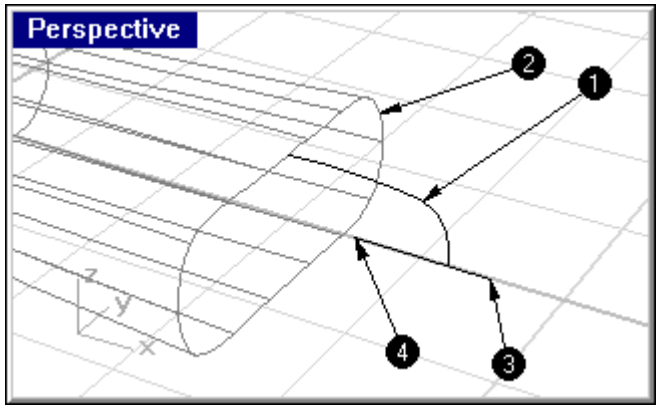
To create a Rail Revolve:

1. Open the **Rail Revolve** model.
2. From the **Surface** menu, click **Rail Revolve**.

Notes:

Rail Revolve
Look for this button.

3 At the **Select profile curve** prompt, select the curve at the end of the object (1).

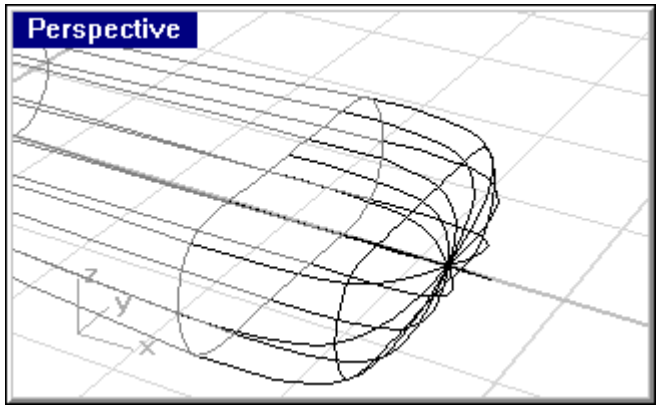


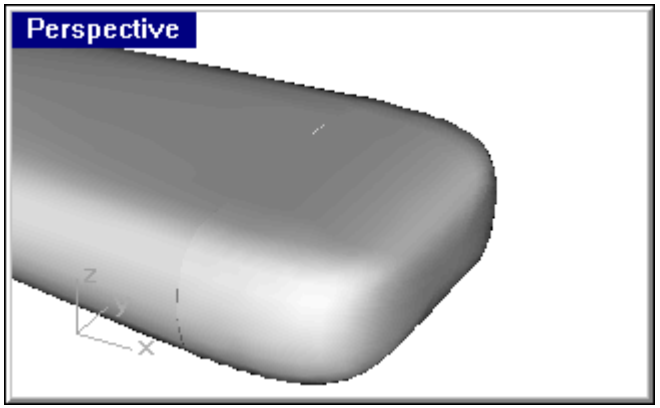
4 At the **Select path curve** prompt, select the end of the object (2).

5 At the **Start of revolve axis** prompt, select one end of the axis line (3).

6 At the **End of revolve axis** prompt, select the other end of the axis line (4).

A surface is revolved around the axis that follows along the curve at the end of the object.





7 Repeat this procedure for the other end of the object.

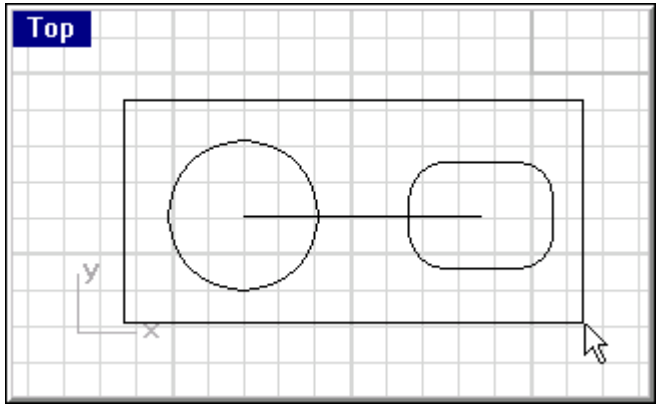
Notes:

Exercise 58—Using sweeps to create surfaces

To create a 1 Rail Sweep:

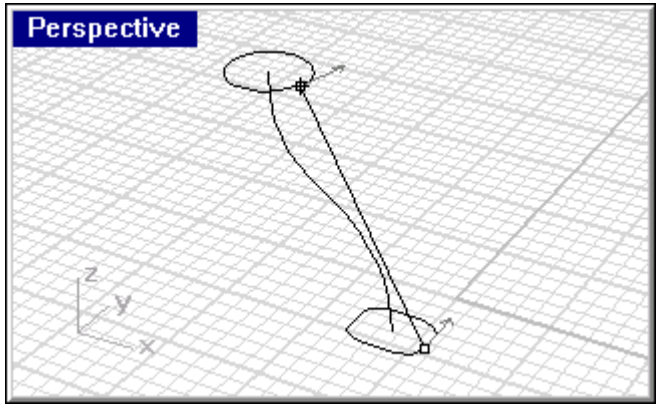
1 Open the **1 Rail Sweep** model.

2 Select all of the curves.



3 From the **Surface** menu, click **Sweep 1 Rail.**

4 At the **Select seam point to adjust, press Enter when done (FlipDirection Automatic Natural)** prompt, press ENTER.

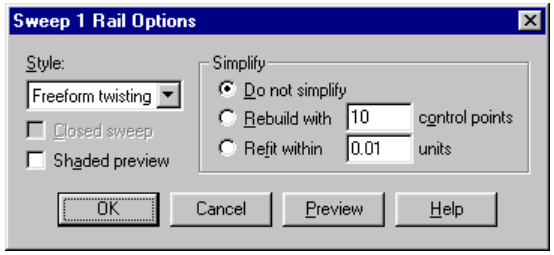


Notes:

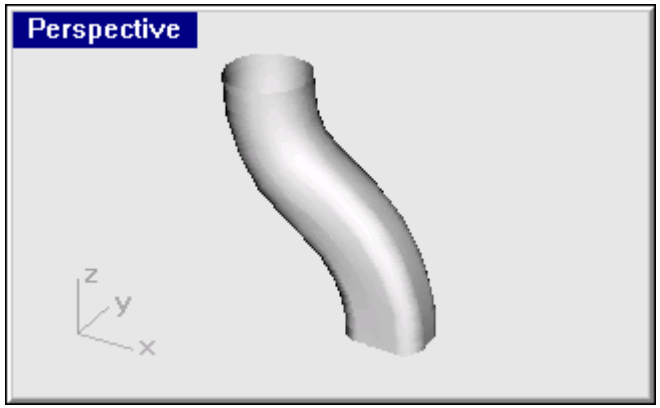
Sweep along 1 Rail
Look for this button

5 In the **Sweep 1 Rail Options** dialog, then click **OK**.

The two cross section shapes are blended together to form the surface.



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



Exercise 59—Using sweeps to create surfaces

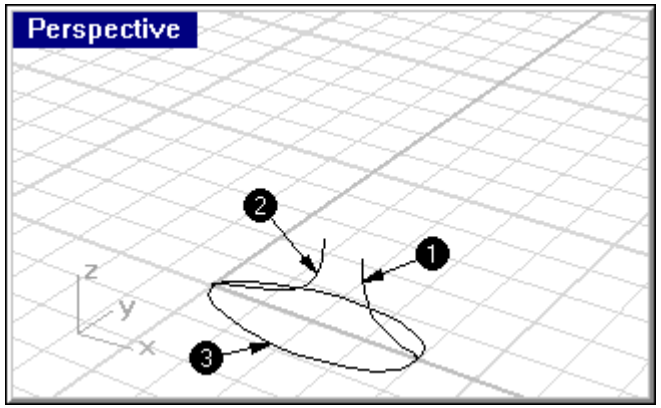
To create a 2 Rail Sweep:

- Open the **2 Rail Sweep** model.

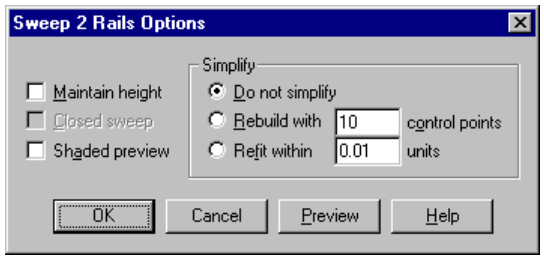
Notes:

To create the base part:

1. Change to the **Base Surface** layer.
2. From the **Surface** menu, click **Sweep 2 Rails.**



3. At the **Select 2 rail curves** prompt, select the first rail curve (1).
4. At the **Select 2 rail curves** prompt, select the next rail curve (2).
5. At the **Select cross-section curves (Point)** prompt, select the cross-section curve (3).
6. At the **Select cross-section curves (Point)** prompt, press ENTER.

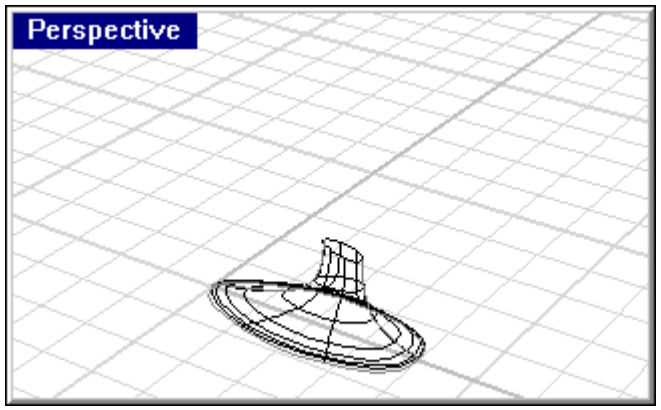


Notes:

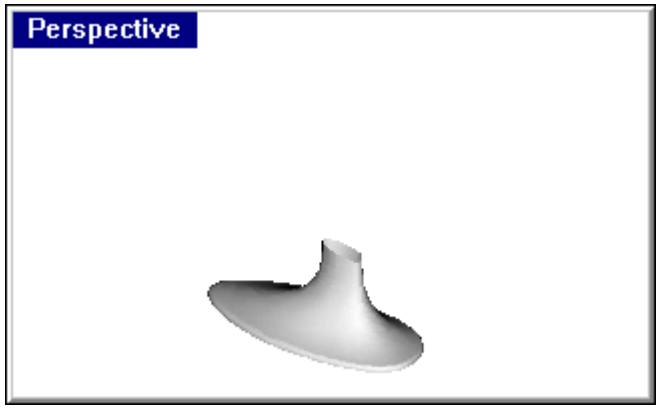
Sweep along 2 Rails
Look for this button

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

A surface is created that whose edges match the rail curves along the rail curves.



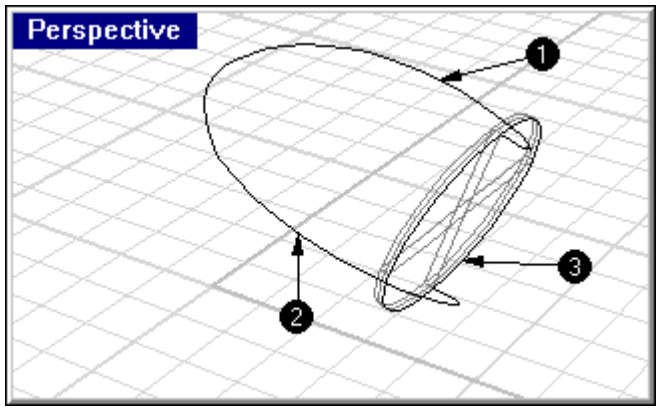
8 From the **Render** menu, click **Shade.**



Notes:

To create the housing:

1. From the **Edit** menu, click **Layers**, then click **Edit**.
2. On the **Edit Layers** dialog, turn the **Housing Surface**, **Housing Curves**, and **Mirror** layers on. Lock the **Base Curves** and **Base Surface** layers.
3. Make the **Housing Surface** layer current.
4. Click **OK.**
5. From the **Surface** menu, click **Sweep 2 Rails.**

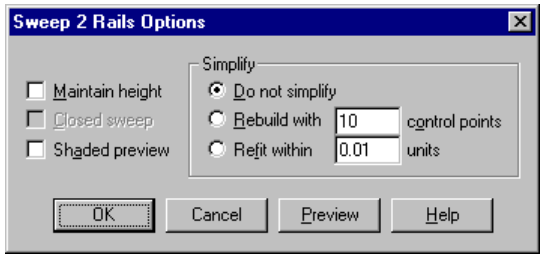


6. At the **Select 2 rail curves** prompt, select the first rail curve (1).
7. At the **Select 2 rail curves** prompt, select the next rail curve (2).
8. At the **Select cross-section curves (Point)** prompt, select the curve at the front of the cylinder (3).

Notes:

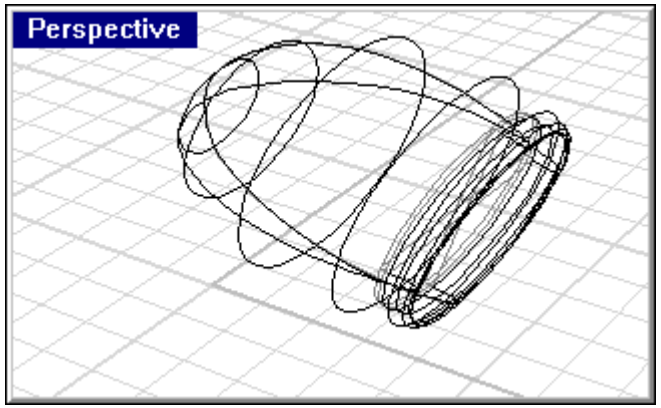
Sweep along 2 Rails
Look for this button

9 At the **Select cross-section curves (Point)** prompt, press enter.



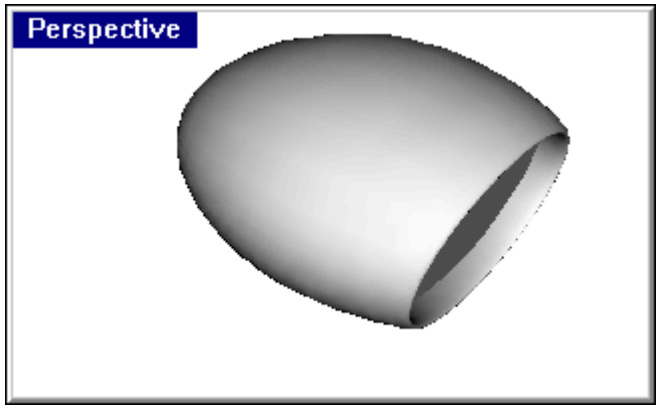
10 In the **Sweep 2 Rails Options** dialog, click **OK**.

A surface is created that wraps around the rail curves and finished with a smooth end where the rail curves come together.



Notes:

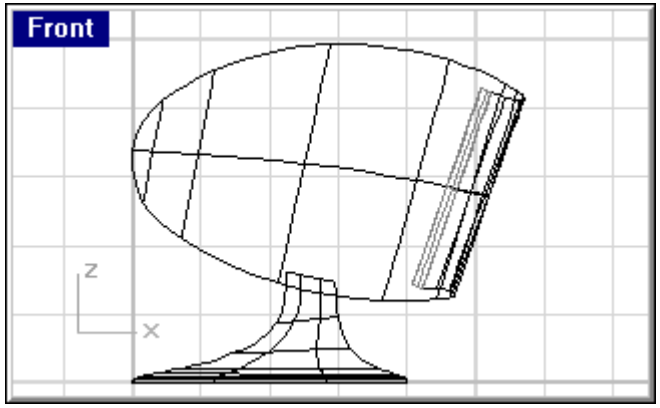
11 From the **Render** menu, click **Shade.**



To join the two parts:

1 From the **Edit** menu, click **Layers**, then click **Edit**.

2 In the **Edit Layers** dialog, make sure the **Housing Surface,** the **Base Surface**, and **Mirror** layers are on. All other layers should be turned off.

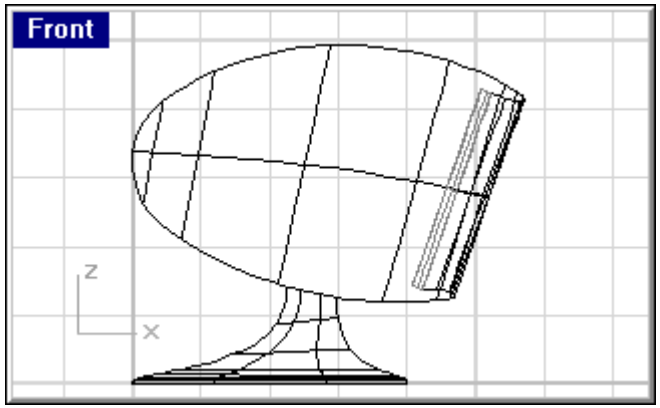


3 From the **Solid** menu, click **Union.**

4 At the **Select surface or polysurface** prompt, select the base surface.

Notes:

5 At the **Select surface or polysurface to union** prompt, select the housing surface.



To make a fillet between the two surfaces:

1 From the **Surface** menu, click **Fillet**.

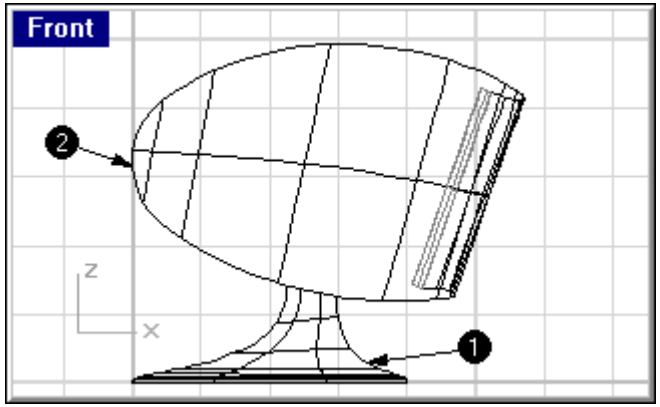
2 At the **Select first surface to fillet (Radius=1 Extend=Yes Trim=Yes)** prompt, type **.25** and press ENTER.

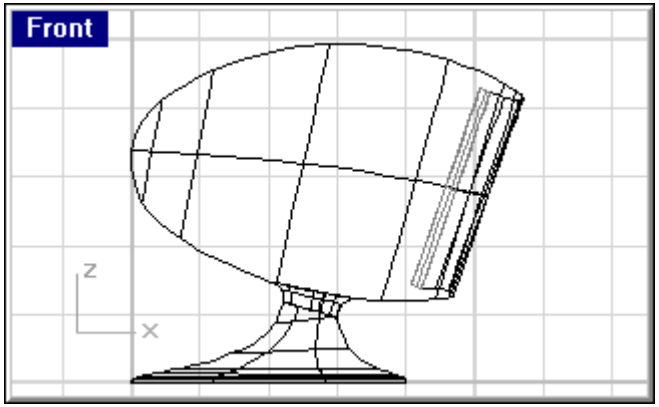
3 At the **Select first surface to fillet (Radius=0.25 Extend=Yes Trim=Yes)** prompt, select the base surface (1).

Notes:

4 At the **Select second surface to fillet (Radius=0.25 Extend=Yes Trim=Yes)** prompt, select the housing surface (2).

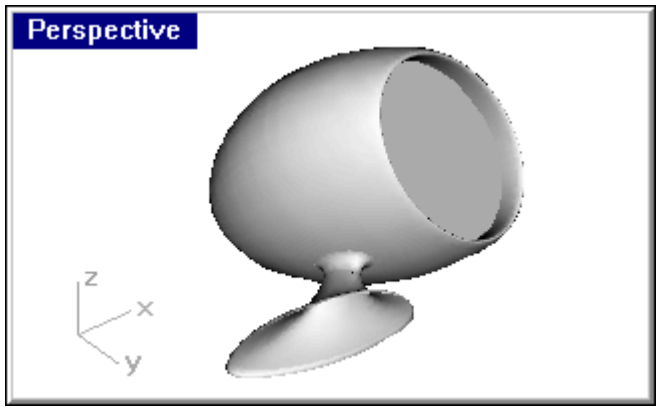
A fillet is created between the two surfaces.





5 Rotate the object in the perspective viewport.

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

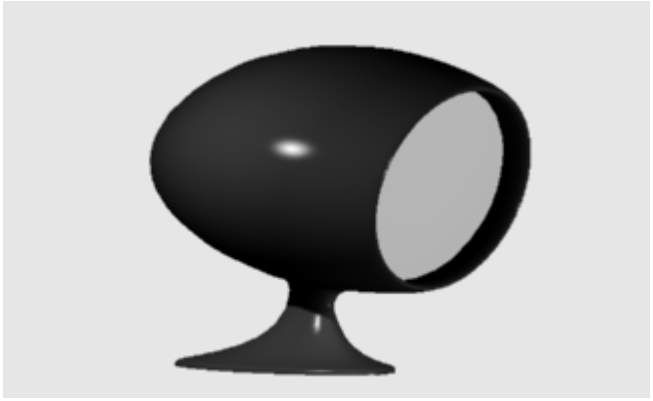


7 Rotate the object in the perspective viewport.

8 **Join** the base, the fillet, and the housing surfaces.

Notes:

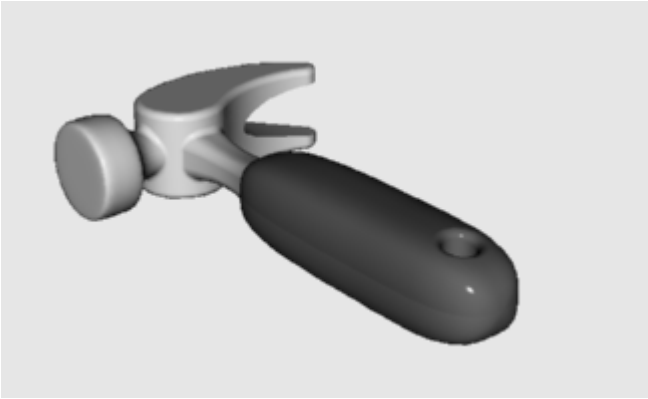
9 From the **Render** menu, click **Render.**



Notes:

Modeling a Toy Hammer

Some models require more attention to detail. This is an example of a model that requires precise modeling techniques. This exercise also requires a number of different surface creation techniques. The technical drawing is included to help you create a very precise model.

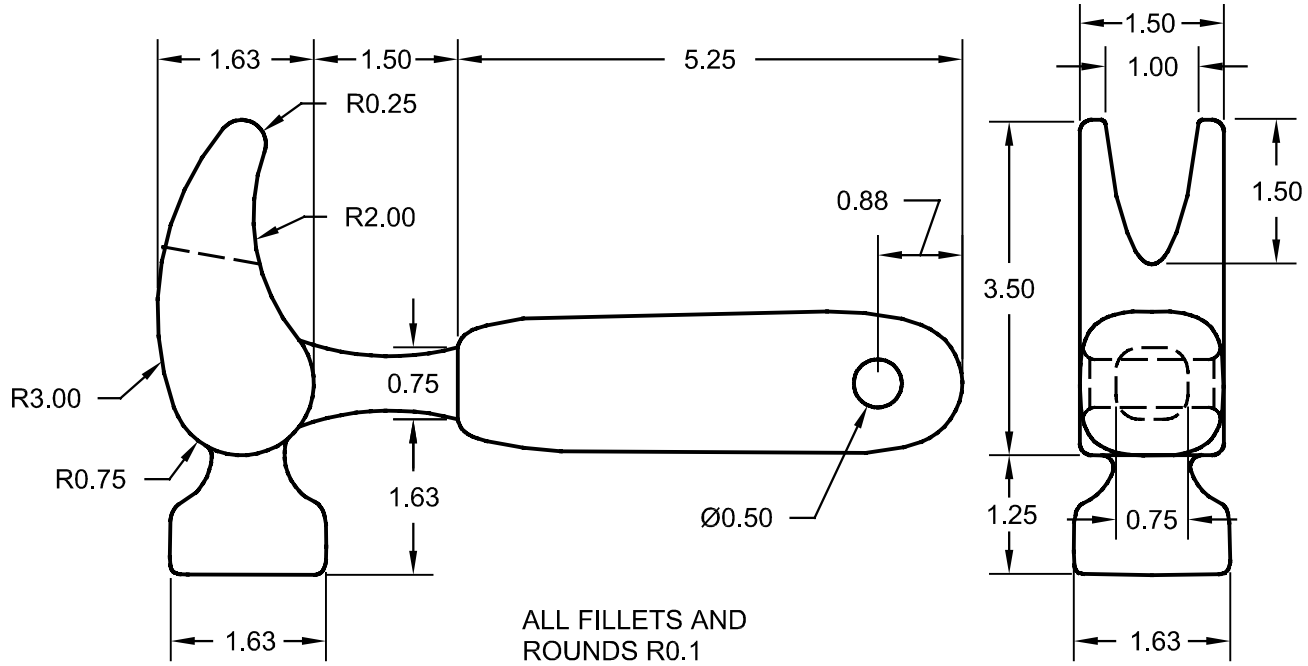


Notes:

Exercise 60— Creating a toy hammer:

1 Start a new model. Save as **Hammer**.

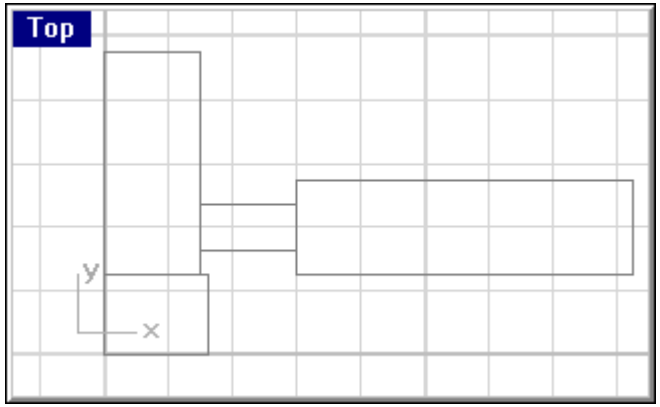
2 Create the following layers: **Curve, Handle, Tang, Head, Claw.**



Notes:

3 Draw the outlines for the hammer.

Drawing outlines helps while drawing the curves. You can either draw lines, polylines, or rectangles to create the outlines. Use the dimensions on the technical drawing to get accurate outlines.



To create the claw:

When modeling the shape of the claw you will use circles and arcs. You can trim the circles and arcs and then join them together to create the closed curve.

1 Change to the **Curve** layer

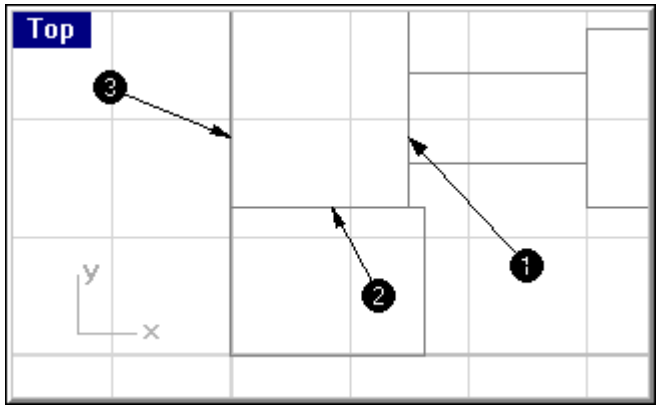
2 Draw a curve defining the shape of the claw in the top viewport.

You can use a freeform curve or use a combination of arcs and circles which are trimmed and joined to create the curve. Following is a step by step approach to creating the curve for the claw part of the hammer.

Start by drawing two circles.

Notes:

3 From the **Curve** menu, click **Circle**, then click **Tangent to 3 Curves.**

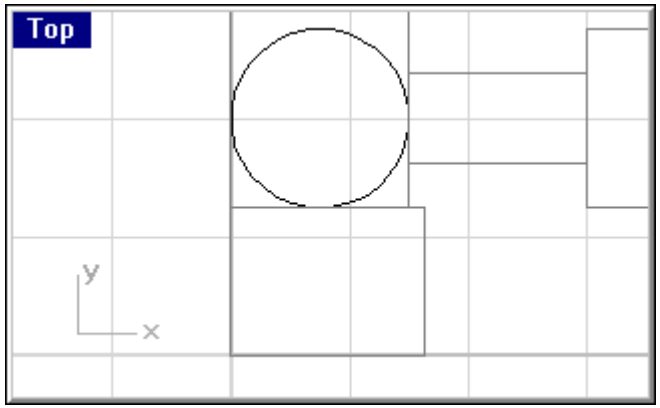


4 At the **Select first curve** prompt, select the curve at point one.

5 At the **Select second curve** prompt, select the curve at point two.

6 At the **Select third curve** prompt, select the curve at point three.

The circle will be created tangent to all three curves you selected.



7 From the **Curve** menu, click **Circle**, then click **Diameter.**

8 At the **Start of diameter (Vertical)** prompt, snap to the midpoint of the upper claw construction line.

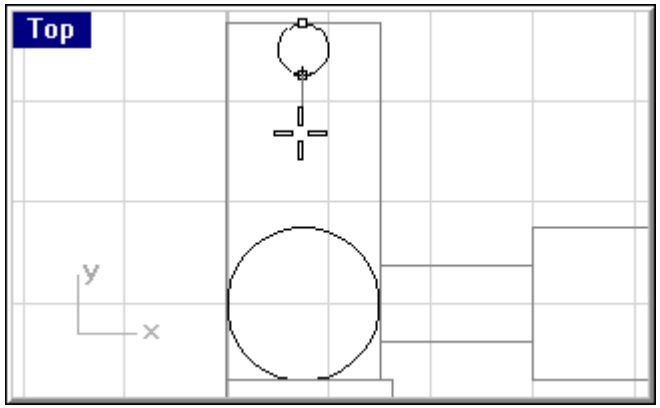
9 At the **End of diameter** prompt, type **.5** and press ENTER.

Notes:

Circle: Tangent, Tangent, Tangent
Look for this button.

Circle Diameter
Look for this button.

10 At the **End of diameter** prompt, pick a point directly below the first point.



To create the tangent arcs:

1 From the **Curve** menu, click **Arc**, then click **Tangent, Tangent, Radius.**

2 At the **Select first curve for ArcTTR** prompt, select a circle near the right edge.

3 At the **Select second curve for ArcTTR** prompt, select the other circle near the right edge.

4 At the **Radius <1>** prompt, type **2** and press ENTER.

5 Delete the parts of the arcs that you don't need.

6 From the **Curve** menu, click **Arc**, then click **Tangent, Tangent, Radius.**

7 At the **First curve** prompt, select a circle near the left edge.

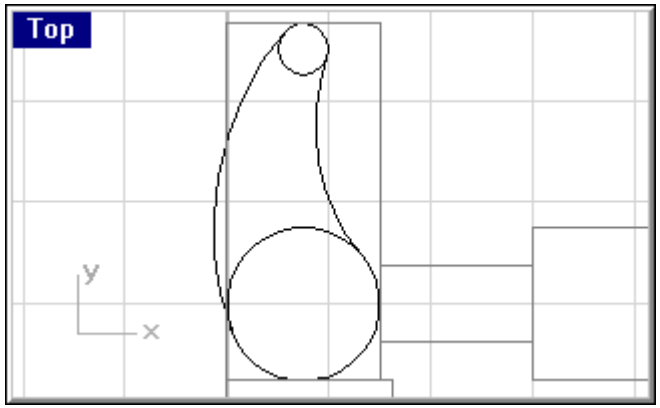
8 At the **Second curve** prompt, select the other circle near the left edge.

9 At the **Radius <1>** prompt, type **3** and press ENTER.

Notes:

Arc: Tangent, Tangent, Radius
Look for this button.

10 Delete the parts of the arcs that you don't need.



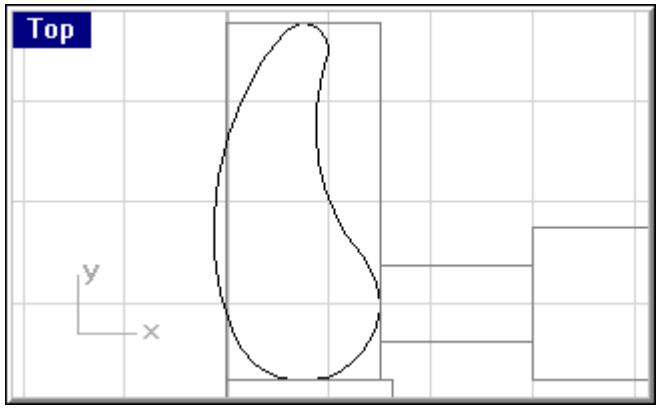
To trim the circles:

1. From the **Edit** menu, click **Trim.**
2. At the **Select cutting edges (Options)** prompt, select one of the arcs.
3. At the **Select cutting edges (Options)** prompt, select the other arc.
4. At the **Select cutting edges (Options)** prompt, press ENTER.
5. At the **Select object to trim (Options)** prompt, select the insides of the circles.

Notes:

Trim
Look for this button.

6 At the **Select object to trim (Options)** prompt, press ENTER to end the command.



7 **Join** the curve segments.

The command terminates when the curve closes.

To extrude the claw part of the hammer:

1 Change to the **Claw** layer.

2 Select the joined segments.

3 From the **Surface** menu, click **Extrude**, then click **Straight.**

4 At the **Extrusion distance (Direction Cap=no Bothsides Tapered)** prompt, type **C** and press ENTER.

This makes it a solid.

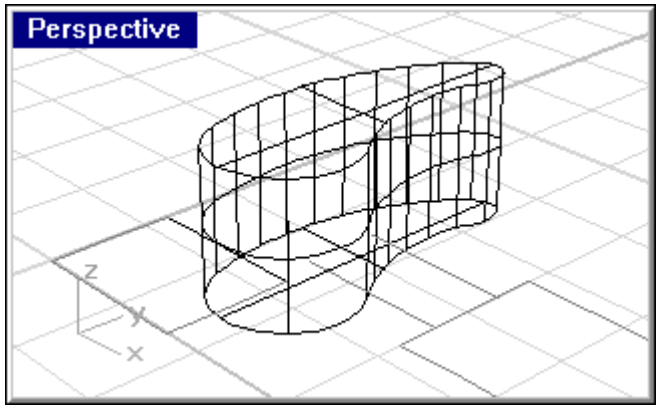
5 At the **Extrusion distance (Direction Cap=yes Bothsides Tapered)** prompt, type **B** and press ENTER.

This extends the extrusion on both sides of the construction plane.

Notes:

Extrude Straight
Look for this button.

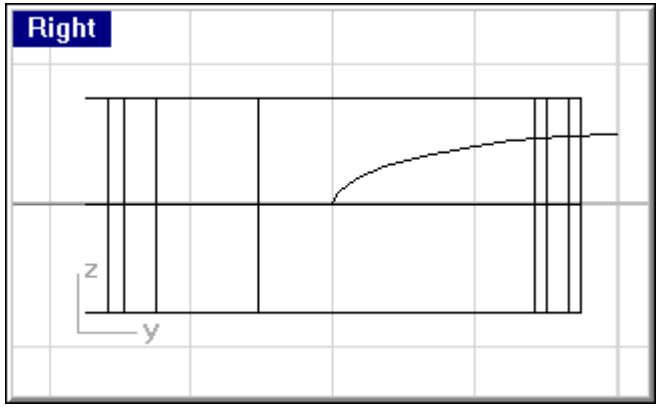
6 At the **Extrusion distance (Direction Cap=yes)** prompt, type **.75** and press ENTER.



To create slot for the claw part of the hammer:

1 Change to the **Curve** layer.

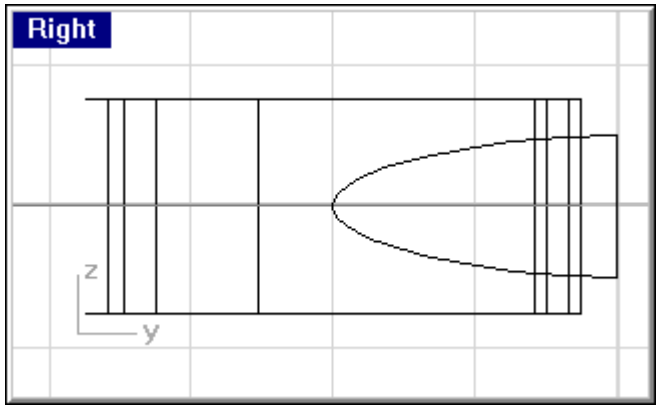
2 Draw a **Curve** in the **Right** viewport, that starts in the middle of the extruded object, and extends to the right beyond the extrusion.



3 **Mirror** the curve.

Notes:

4 Draw a **Line** between the end points, then **Join** the three curves.



5 Select the joined segments.

6 Change to the **Claw** layer.

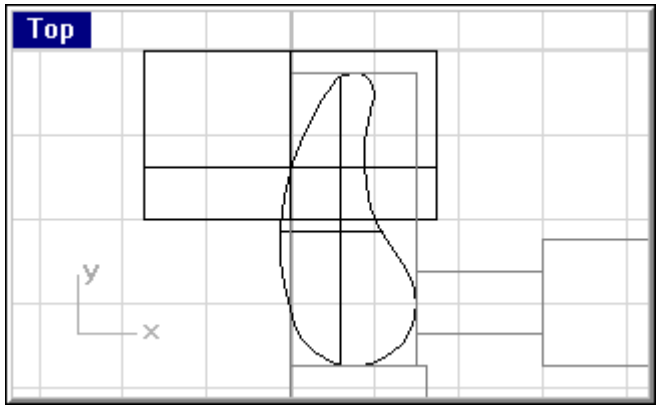
7 From the **Surface** menu, click **Extrude**, then click **Straight.**

8 At the **Extrusion distance <.75> (Direction Cap=yes Bothsides Tapered)** prompt, type **B** and press ENTER.

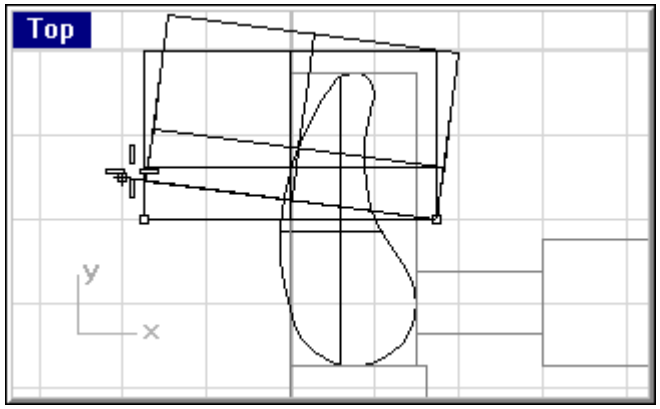
Notes:

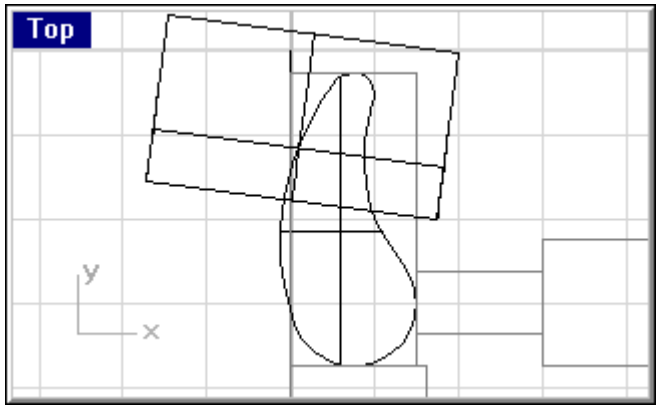
Extrude Straight
Look for this button.

9 At the **Extrusion distance (Direction Cap=yes)** prompt, type **2** and press ENTER.



10 Rotate the slot in the **Top** viewport





11 **Save** your model.

To remove the slot from the claw:

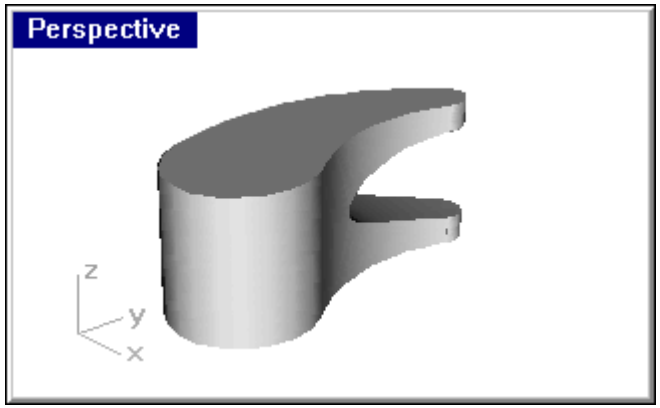
1 From the **Solid** menu, click **Difference.**

2 At the **Select surface or polysurface** prompt, select the claw extrusion.

Notes:

Boolean Difference
Look for this button.

3 At the **Select surface or polysurface to subtract** prompt, select the Slot extrusion.



4 **Save** your model.

To make the fillets on the claw:

1 From the **Solid** menu, click **Fillet Edge.**

2 At the **Select edges to fillet (Radius=1)** prompt, select the top edge of the claw.

3 At the **Select edges to fillet (Radius=1)** prompt, select the bottom edge of the claw.

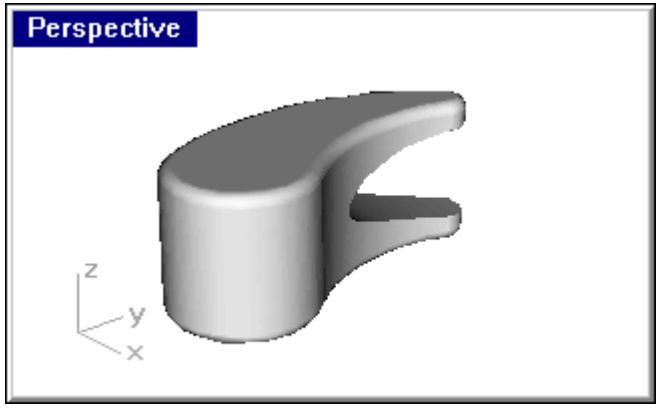
4 At the **Select edges to fillet (Radius=1)** prompt, type **.1** and press ENTER.

Notes:

Fillet Edge
Look for this button.

5 At the **Select edges to fillet (Radius=0.1)** prompt, press ENTER.

The edges have a round on them.

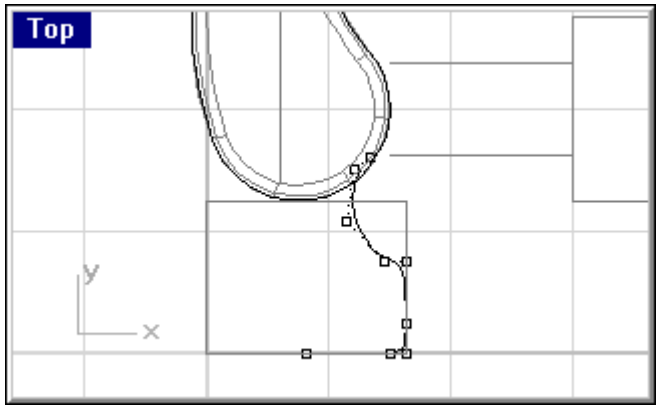


6 **Save** your model.

To create the Head:

1 Change to the **Curve** layer.

2 Create the curve for the cross section of the head, using the **Curve** command**.**



Make sure the curve intersects the claw part. This will make joining the two pieces easier.

Notes:

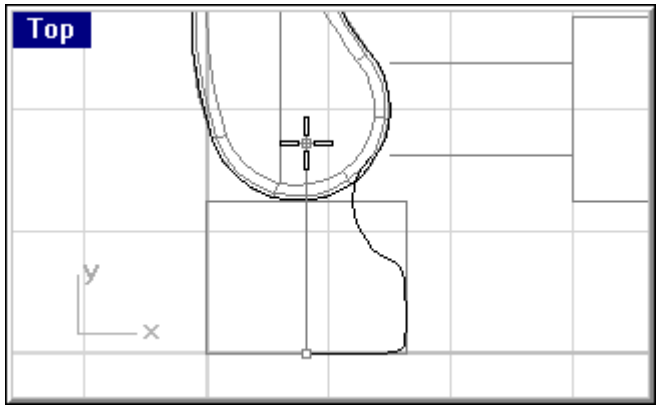
3 Change to the **Head** layer.

4 From the **Surface** menu, click **Revolve.**

5 At the **Select curve to revolve** prompt, choose the polyline for the cross-section of the head.

6 At the **Start of revolve axis** prompt, pick the end of the curve.

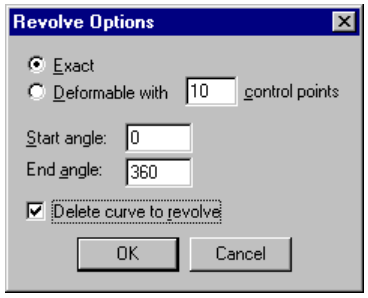
7 At the **End of revolve axis** prompt, toggle ortho on and pick a point above the first point.



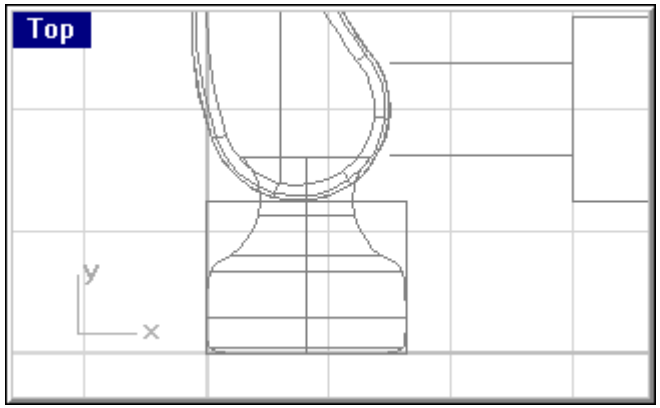
Notes:

Revolve
Look for this button.

8 In the **Revolve Options** dialog, click **OK**.



A surface is created.



9 **Save** your model.

To join the head and the claw:

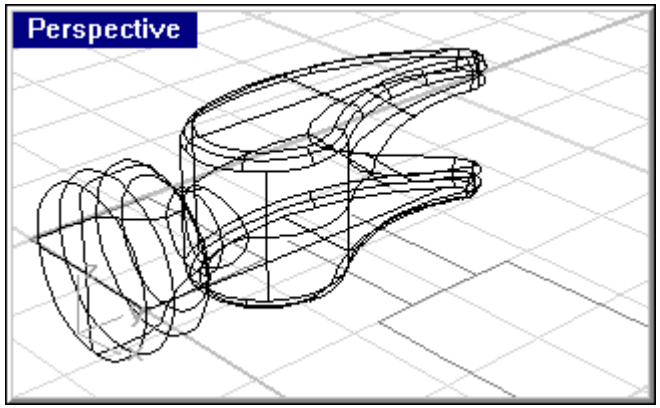
1 From the **Solid** menu, click **Union.**

2 At the **Select surface or polysurface** prompt, select the **Head**.

Notes:

Boolean Union
Look for this button.

3 At the **Select surface or polysurface to union** prompt, select the **Claw**.

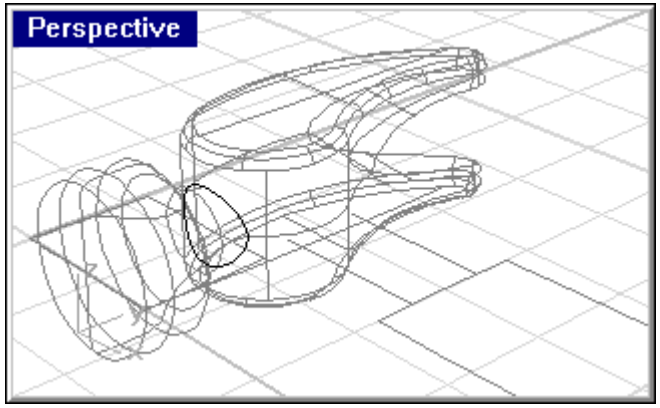


4 **Save** your model.

To make the fillet where the head and the claw meet:

1 From the **Solid** menu, click **Fillet Edge.**

2 At the **Select edges to fillet (Radius=0.1)** prompt, select the edge where the two parts join.

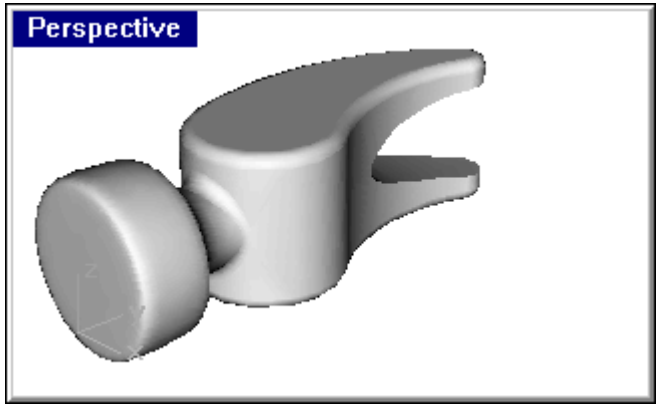


Notes:

Fillet Edge
Look for this button.

3 At the **Select edges to fillet (Radius=0.1)** prompt, press ENTER.

The edge has a round on it.



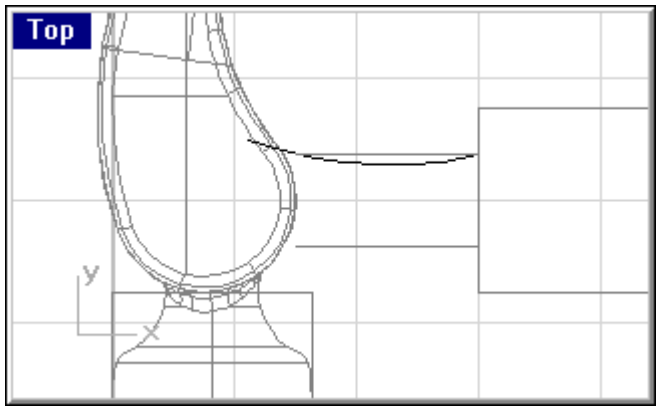
4 **Save** your model.

To create the Tang:

1 Change to the **Curve** layer.

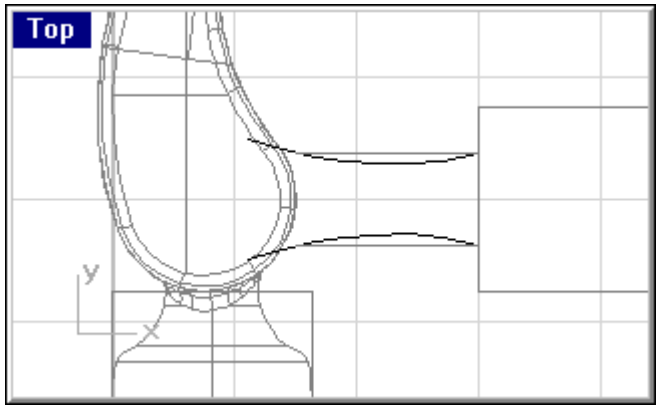
2 Create one of the curves for the tang of the hammer.

Make sure that it intersects the claw.



Notes:

3 **Mirror** to create the other curve.

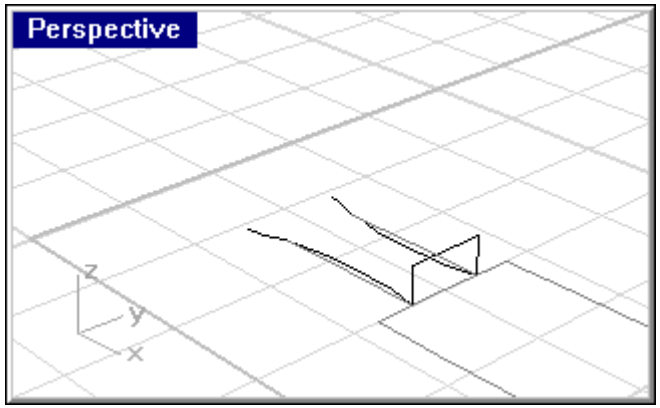


To create the shape curve for the tang and the handle:

Create the shape curve for the tang in the right viewport. This curve will also be used for the handle.

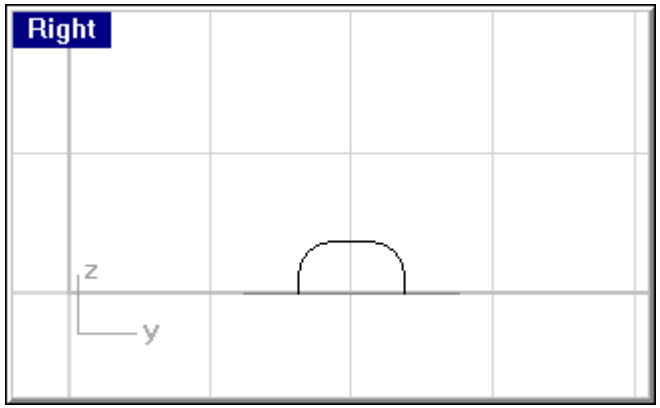
1 Turn on ortho. Use the **Lines** command to create the curve for the tang cross-section curve. This curve will also be used for the handle

The height of the line is **.375** and the width is **.75**.



Notes:

2 **Fillet** the corners with a radius of **.25**.



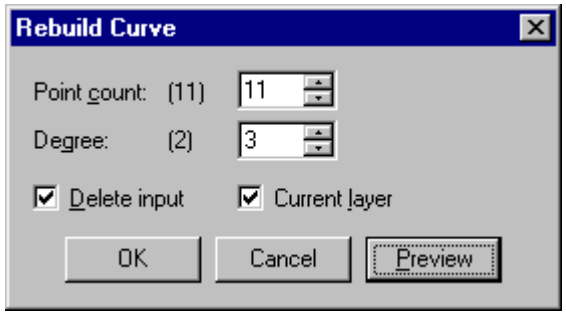
3 **Join** the curves

4 Select the joined curve.

5 From the **Curve** menu, click **Edit Tools**, then click **Rebuild.**

Rebuilding the curve eliminates any kinks, which will make a smooth surface with fewer potential problems later in the modeling process.

6 In the **Rebuild Curve** dialog, click **Preview**. If the geometry matches the curve, then click **OK**. If the geometry doesn't match the curve, change the point count and click **Preview**.



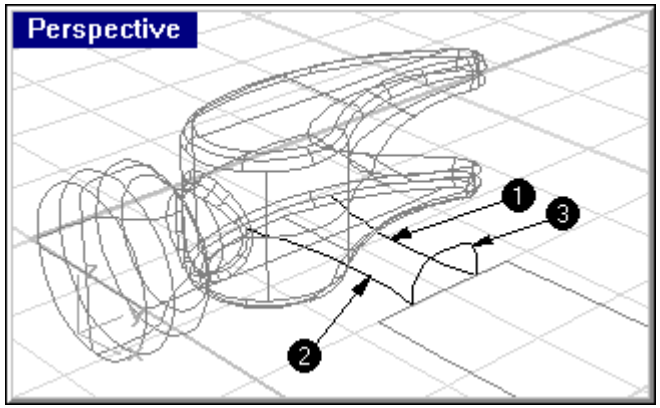
To create the surface for the tang:

1 Change to the **Tang** layer.

Notes:

Rebuild Curve
Look for this button.

2 From the **Surface** menu, click **Sweep 2 Rails.**



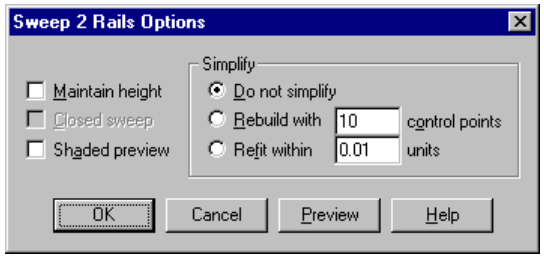
3 At the **Select 2 rail curves** prompt, select one of the rail curves (1).

4 At the **Select 2 rail curves** prompt, select the other rail curve (2).

5 At the **Select cross-section curves (Point)** prompt, select the shape curve (3).

6 At the **Select cross-section curves (Point)** prompt, press ENTER.

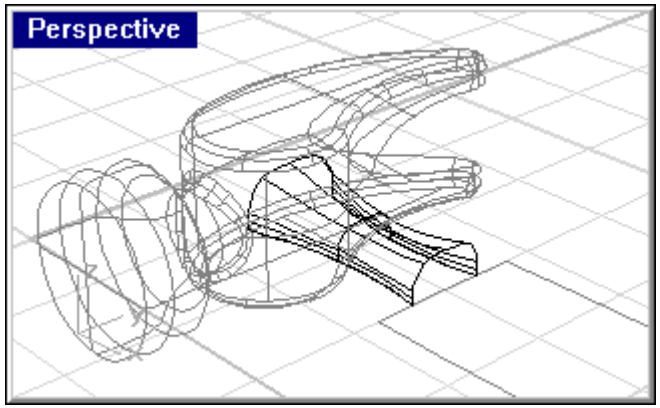
7 In the **Sweep 2 Rails Options** dialog, click **OK**.



Notes:

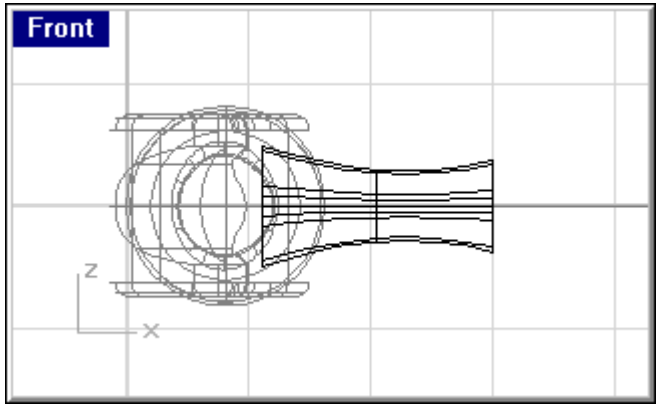
Sweep 2 Rails
Look for this button.

A surface is created.



To finish the tang:

1. From the **Transform** menu, click **Mirror**.
2. At the **Select objects to mirror** prompt, choose the tang surface you created.
3. At the **Start of mirror plane** prompt, in the **Front** viewport snap to the endpoint at the lower left corner of the tang surface.
4. At the **End of mirror plane** prompt, toggle ortho on and pick a point to the right in the **Front** viewport.



Notes:

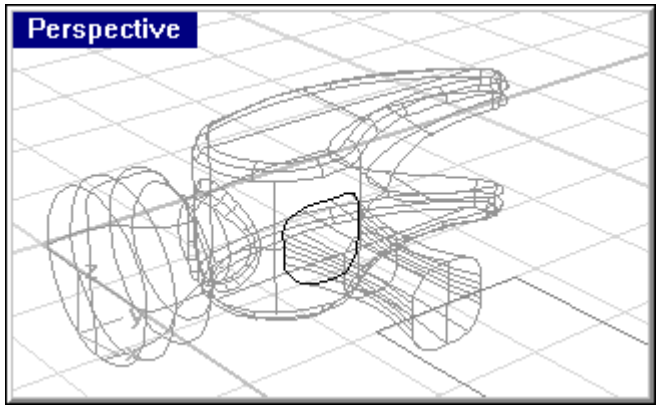
Mirror
Look for this button.

5 **Join** the two surfaces together.

6 From the **Solid** menu, click **Cap Planar Holes**.

7 At the **Select surfaces or polysurfaces to cap** prompt, choose the tang.

8 At the **Select surfaces or polysurfaces to cap** prompt, press ENTER.



9 **Save** your model.

To join the tang and the claw:

1 From the **Solid** menu, click **Union.**

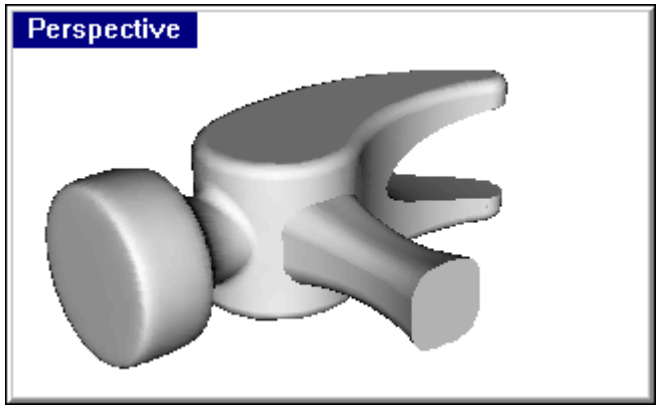
2 At the **Select surface or polysurface** prompt, select the tang.

Notes:

Cap Planar Holes
Look for this button.

Boolean Union
Look for this button.

3 At the **Select surface or polysurface to union** prompt, select the claw.

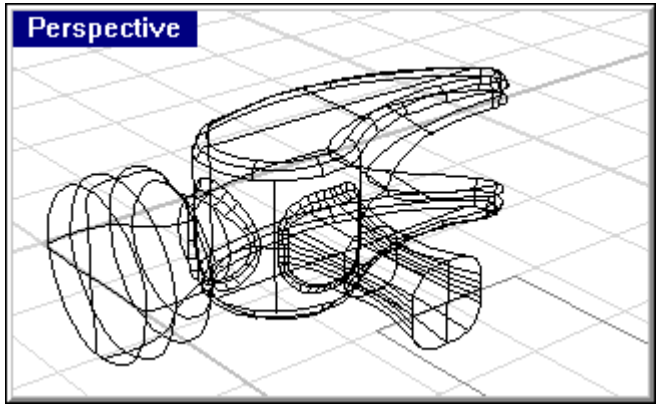


4 **Save** your model.

To make the fillet where the tang and the claw meet:

1 From the **Solid** menu, click **Fillet Edge.**

2 At the **Select edges to fillet (Radius=0.1)** prompt, select the edge where the two parts join.

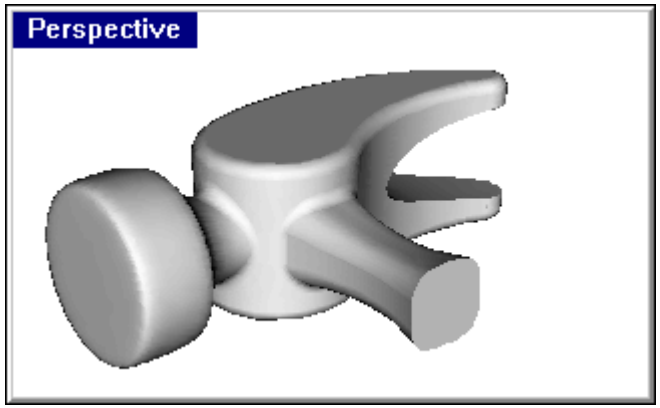


3 At the **Select edges to fillet (Radius=0.1)** prompt, press ENTER.

Notes:

Fillet Edge
Look for this button.

The edge has a round on it.

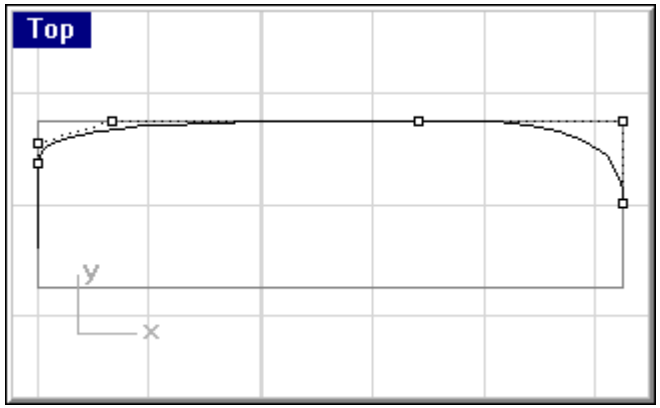


4 **Save** your model.

To create the Handle:

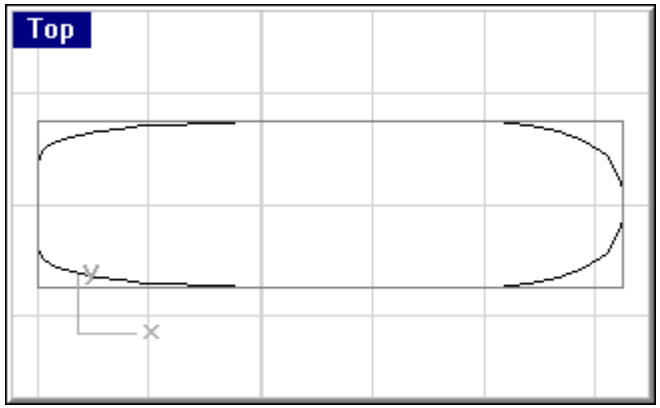
1 Change to the **Curve** layer.

2 Create one of the curves for the handle of the hammer. Make it start at the endpoint of the tang profile curve and end on the centerline.



Notes:

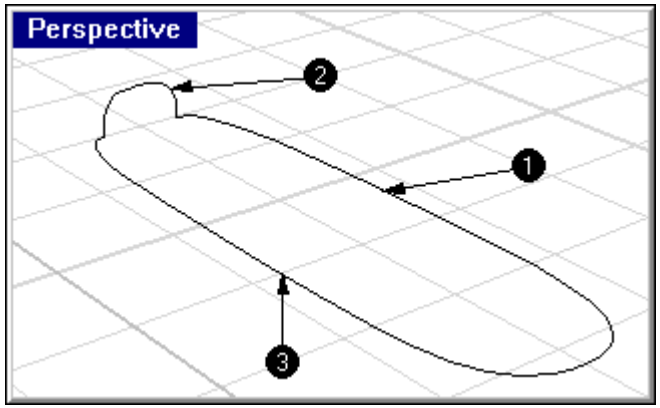
3 **Mirror** to create the other curve.



To create the surface for the handle:

1 Change to the **Handle** layer.

2 From the **Surface** menu, click **Sweep 2 Rails.**



3 At the **Select 2 rail curves** prompt, select one of the rail curves.

4 At the **Select 2 rail curves** prompt, select the other rail curve.

5 At the **Select cross-section curves (Point)** prompt, select small the curve you drew for the tang.

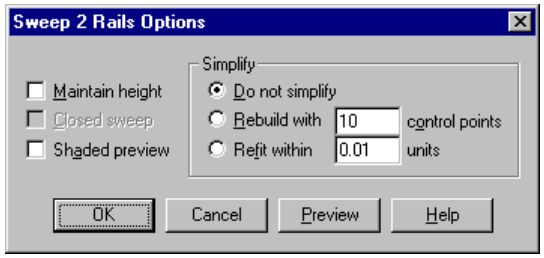
Notes:

Sweep 2 Rails
Look for this button.

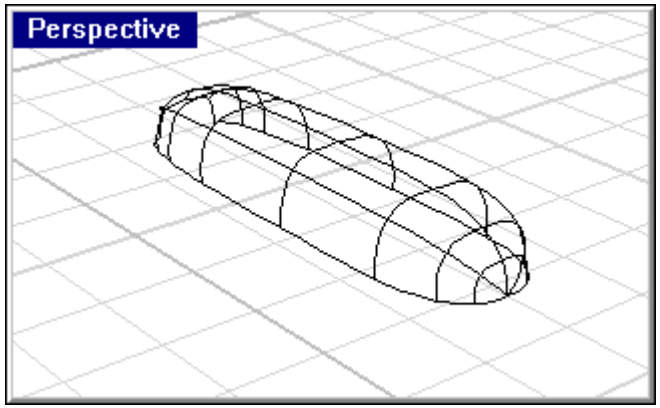
This is the same curve you used to create the tang of the hammer.

6 At the **Select cross-section curves (Point)** prompt, press ENTER.

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



A surface is created.



To finish the handle:

1 From the **Transform** menu, click **Mirror**.

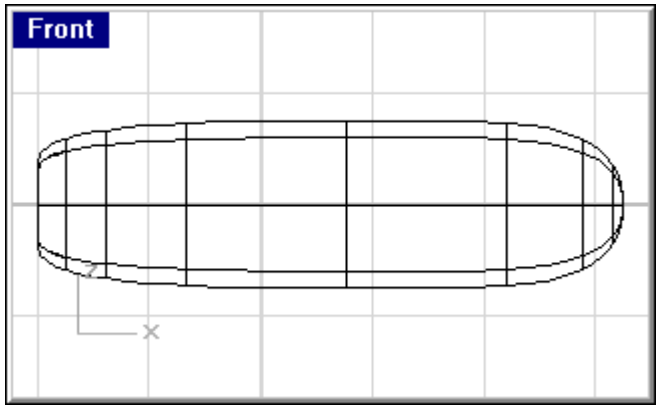
2 At the **Select objects to mirror** prompt, choose the surface you created.

3 At the **Start of mirror plane** prompt, in the **Front** viewport snap to the one end of the handle edge.

Notes:

Mirror
Look for this button.

4 At the **End of mirror plane** prompt, toggle ortho on and pick a point to the right in the **Front** viewport.

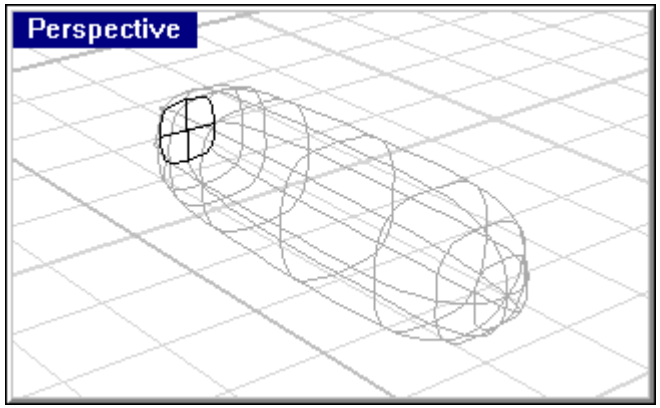


5 **Join** the two surfaces together.

6 From the **Solid** menu, click **Cap Planar Holes**.

7 At the **Select surfaces or polysurfaces to cap** prompt, choose the handle.

8 At the **Select surfaces or polysurfaces to cap** prompt, press ENTER.



9 **Save** your model.

Notes:

Cap Planar Holes
Look for this button.

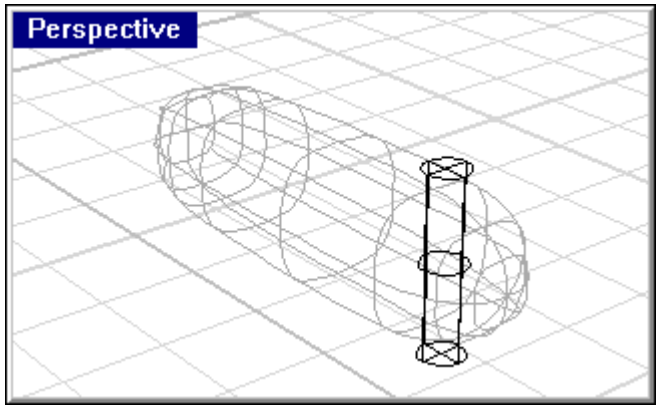
To create the hole for the handle:

1 Draw a **Circle** with the center .88 from the end of the handle. You may have to draw a construction line to help you position the circle.

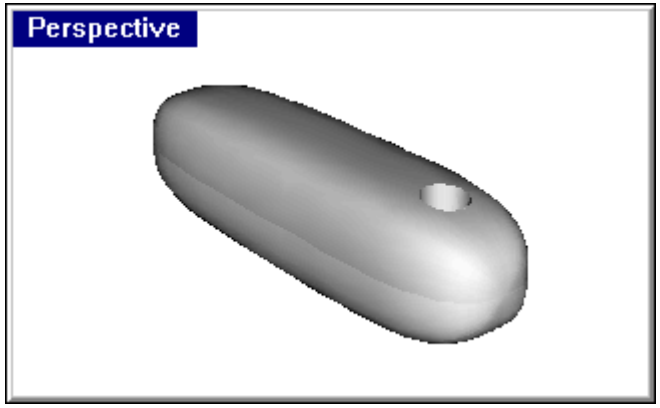
2 From the **Surface** menu, click **Extrude**, then click **Straight.**

3 At the **Extrusion distance <2.00> (Direction Cap=yes Bothsides Tapered)** prompt, type **B** and press ENTER.

4 At the **Extrusion distance (Direction Cap=yes)** prompt, type **2** and press ENTER.



5 Use **Difference** to remove the cylinder from the handle.



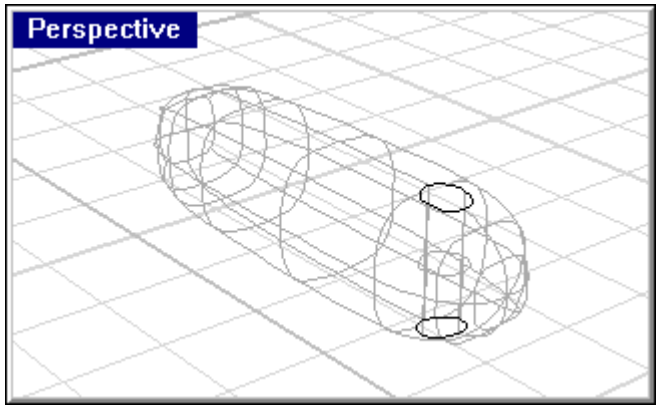
6 **Save** your model.

Notes:

Extrude Straight
Look for this button.

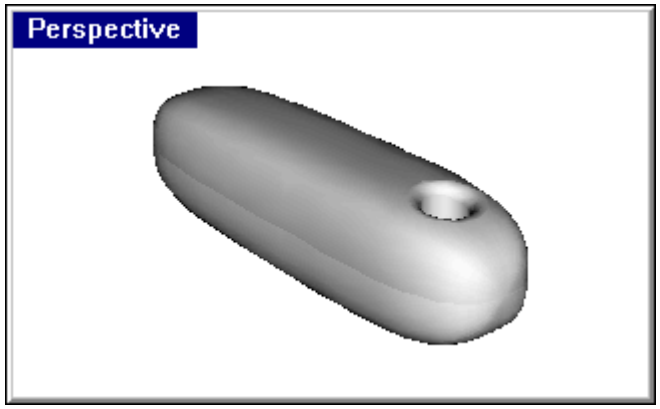
To create the fillets for the hole:

1. From the **Solid** menu, click **Fillet Edge.**
2. At the **Select edges to fillet (Radius=0.1)** prompt, type **.1** and press ENTER.
3. At the **Select edges to fillet (Radius=0.1)** prompt, select both edges of the hole.



4. At the **Select edges to fillet (Radius=0.1)** prompt, press ENTER.

   The edge has a round on them.

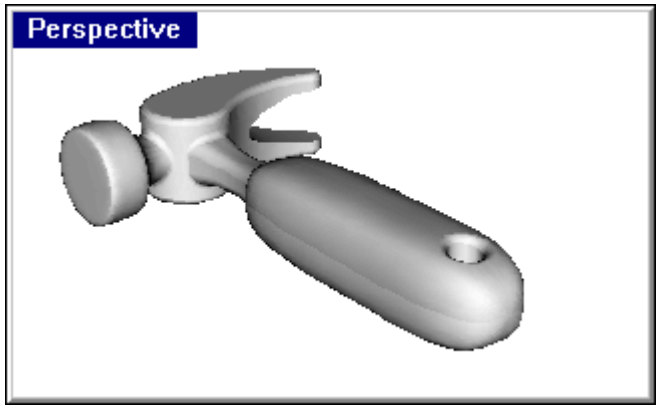


5. **Save** your model.
6. Click in the **Perspective** viewport.

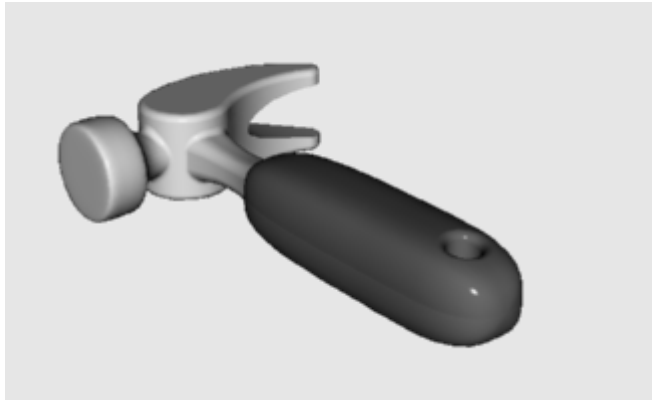
Notes:

Fillet Edge
Look for this button.

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



8 From the **Render** menu, click **Render.**



Notes:

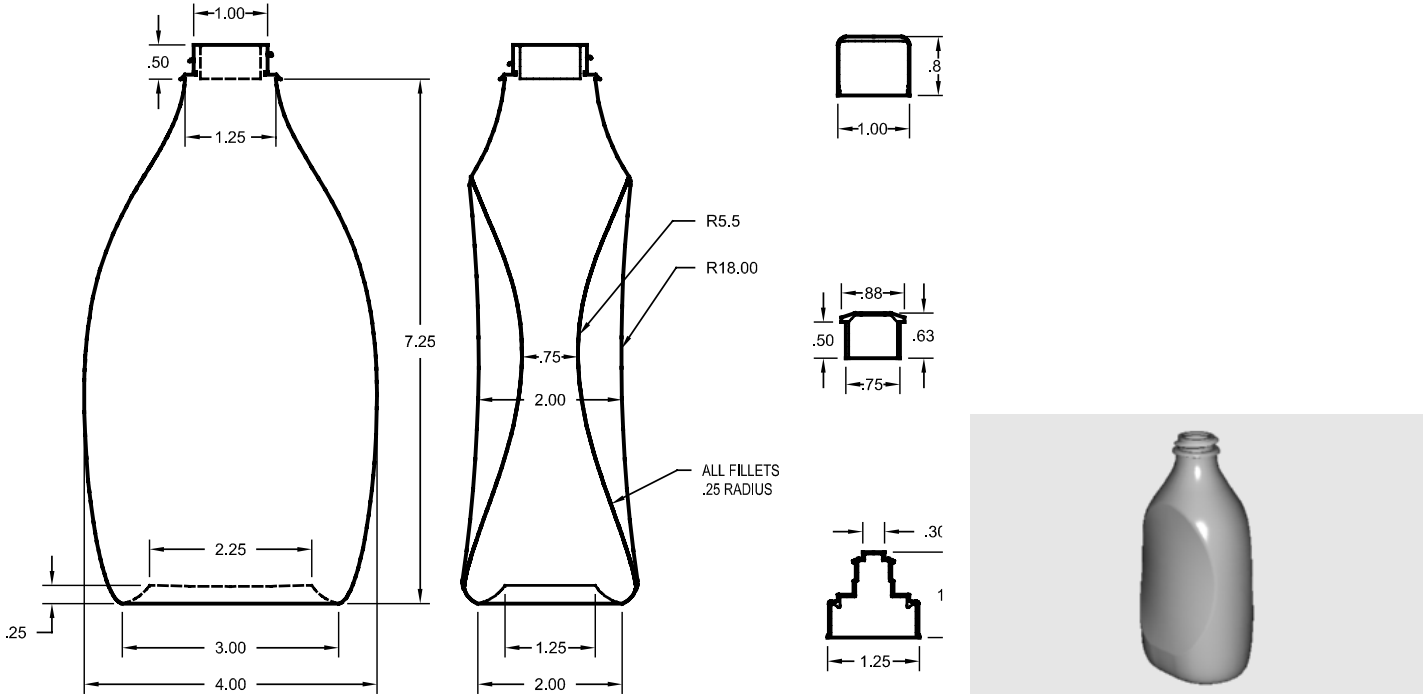
Shade
Look for this button.

Render
Look for this button.

Modeling a Squeeze Bottle

Some models require more attention to detail. This is an example of a model that requires precise modeling techniques. This exercise also requires a number of different surface creation techniques.

The technical drawing is included to help you create a very precise model.



Exercise 61— Creating a squeeze bottle:

To start the model:

1. Start a new model. Save as **Bottle**.
2. Make the following layers: **Construction, Curve, Bottle, BottleTop, Threads, Cap1, Cap2,** and **CapTop.**
3. Change to the **Construction** layer.

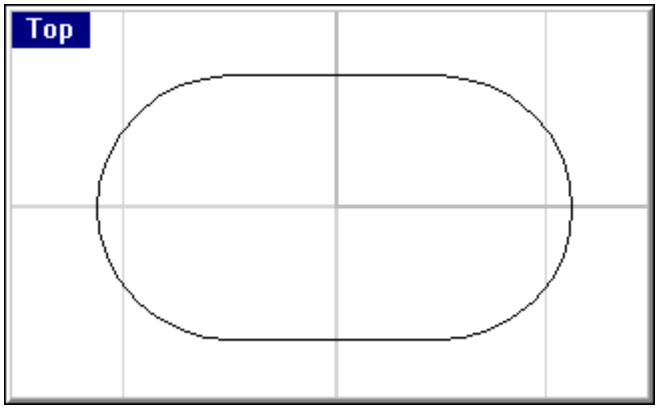
Notes:

Create the Bottle Shape

To create the bottle shape, you will draw curves that define the bottom and top edge, then draw a curve that represents the side shape. You will then create a surface from these defining curves.

To draw a curve that defines the bottom of the bottle:

1. From the **Curve** menu, click **Rectangle**, then click **Center, Corner.**
2. At **Center of rectangle (Rounded)** prompt, type **R**, and press ENTER.
3. At the **Center of rectangle** prompt, move your cursor into the Top viewport then type **0,0,.25** and press ENTER.
4. At the **Corner or length** prompt, type **2.25** and press ENTER.
5. At the **Width** prompt, type **1.25** and press ENTER.
6. At the **Point for rounded corner to pass through (Corner=Arc)** prompt, type **.625** and press ENTER.



To draw a circle for the top of the bottle:

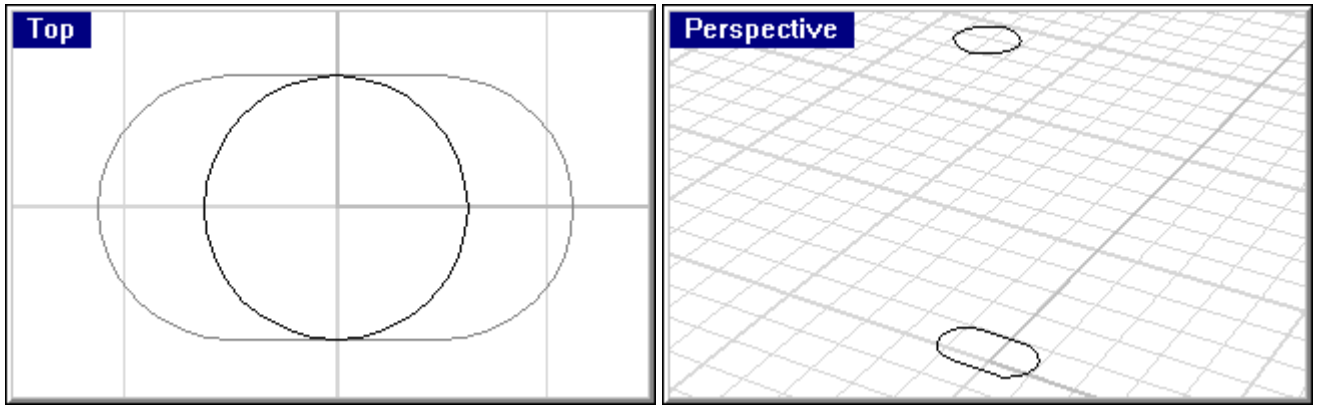
1. From the **Curve** menu, click **Circle**, then click **Center, Radius.**
2. To start the circle 7.25" above the construction plane, at the **Center of circle (Vertical OnCurve)** prompt, move your cursor into the Top viewport, then type **0,0,7.25**, and press ENTER.

Notes:

Centered Rectangle
Look for this button.

Circle
Look for this button.

3 At the **Radius** prompt, type **.625** and press ENTER.



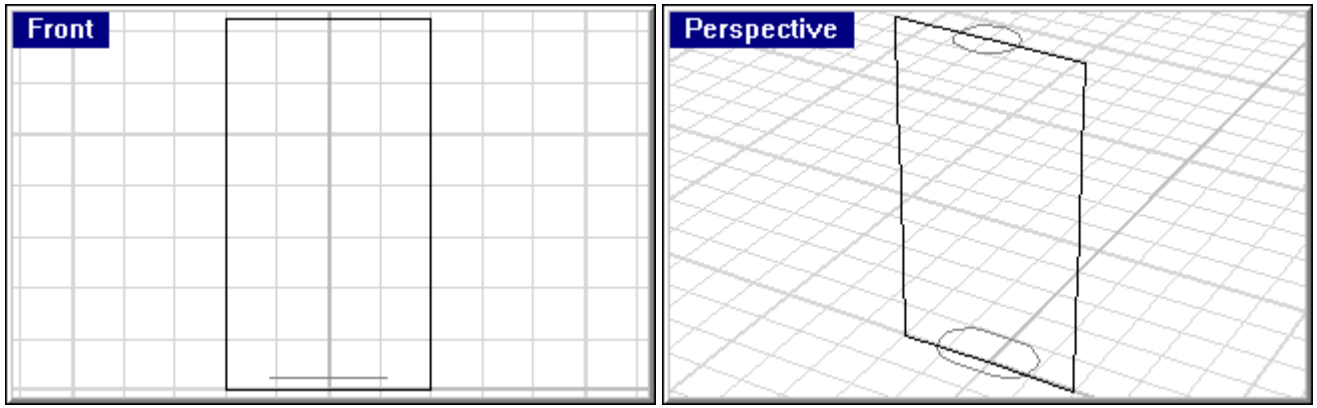
To draw a rectangle that approximates the overall dimension of the bottle:

1 From the **Curve** menu, click **Rectangle**, then click **Corner, Corner.**

2 In the **Front** view, at the **First corner of rectangle (Rounded)** prompt, type **–2,0** and press ENTER.

3 At the **Other corner or length** prompt, type **4** and press ENTER.

4 At **Width** prompt, type **7.25** and press ENTER.

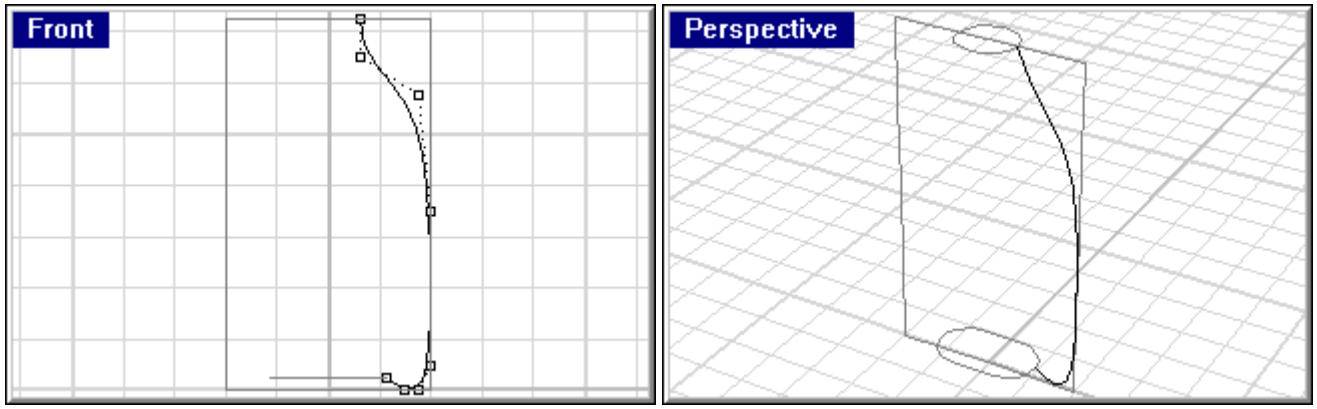


Notes:

Rectangle
Look for this button.

To draw the edge curve:

1. Change to the **Curve** layer.
2. Draw a **curve by control points** to define the edge of the bottle in the **Front** view.
3. Use the rectangle and object snaps to help you establish the correct size.



4. The curve will be used to create the surface of the bottle.

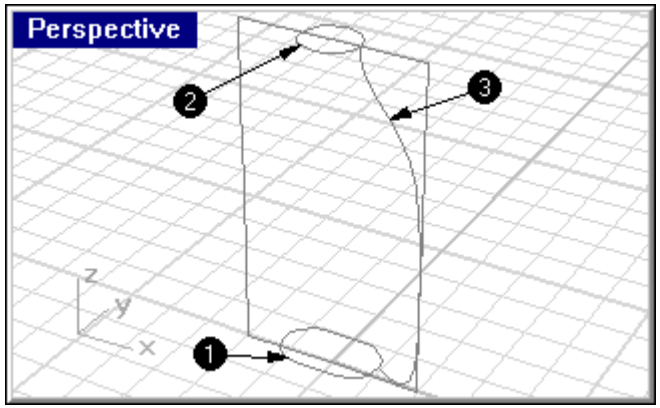
To create the surface for the bottle:

1. Change to the **Bottle** layer.
2. From the **Surface** menu, click **Sweep 2 Rails.**

Notes:

Two-Rail Sweep
Look for this button.

The curve you created will be the shape curve (3). The rounded rectangle (1) and the circle (2) will be the rails.



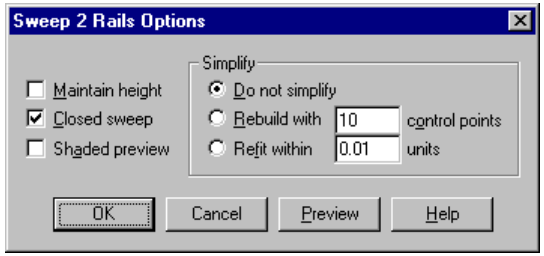
3 At the **Select 2 rail curves** prompt, select the rounded rectangle.

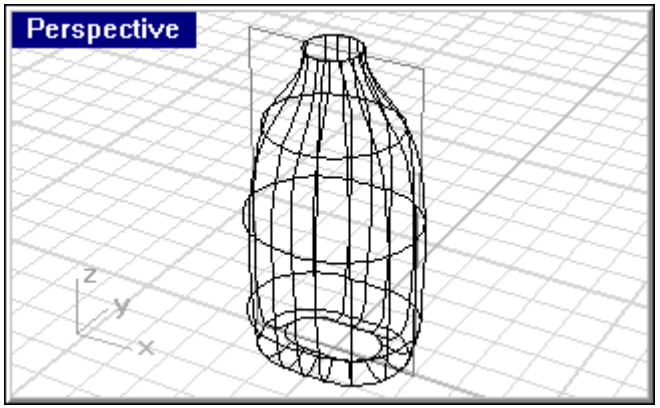
4 At the **Select 2 rail curves** prompt, select the circle.

5 At the **Select cross-section curves (Point)** prompt, select the curve.

6 At the **Select cross-section curves (Point)** prompt, press ENTER.

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





Notes:

Cap the Top and Bottom

If you close the bottle, thereby creating a solid, Rhino can calculate the bottle's volume. If you were creating this bottle in real life, knowing the volume would be important. Normally, a bottle would have to be designed to hold a certain amount of stuff.

If the edges of the remaining open surfaces are planar curves, you can use the Cap command to close them. The open edges on the bottle are the top circle and the bottom rounded rectangle, and they are planar.

To cap the top and bottom:

1. From the **Solid** menu, click **Cap Planar Holes.**
2. At the **Select surfaces or polysurfaces to cap** prompt, choose the bottle.
3. At the **Select surfaces or polysurfaces to cap** prompt, press ENTER.

Cap Planer Holes
Look for this button.

Flatten the Sides

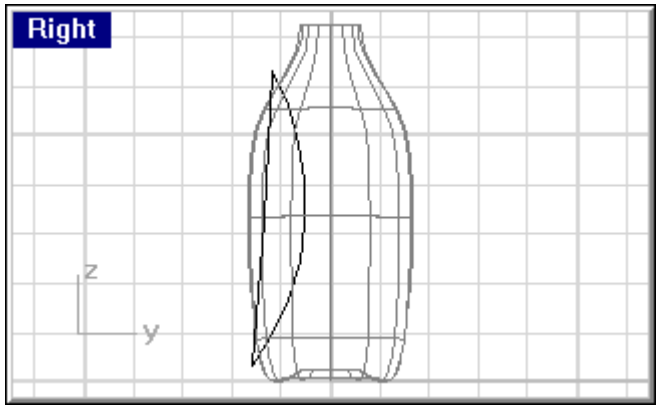
You will notice that the bottle bulges too much in the **Right** view. We will create a custom surface to trim the bulge away.

To create the trimming surface:

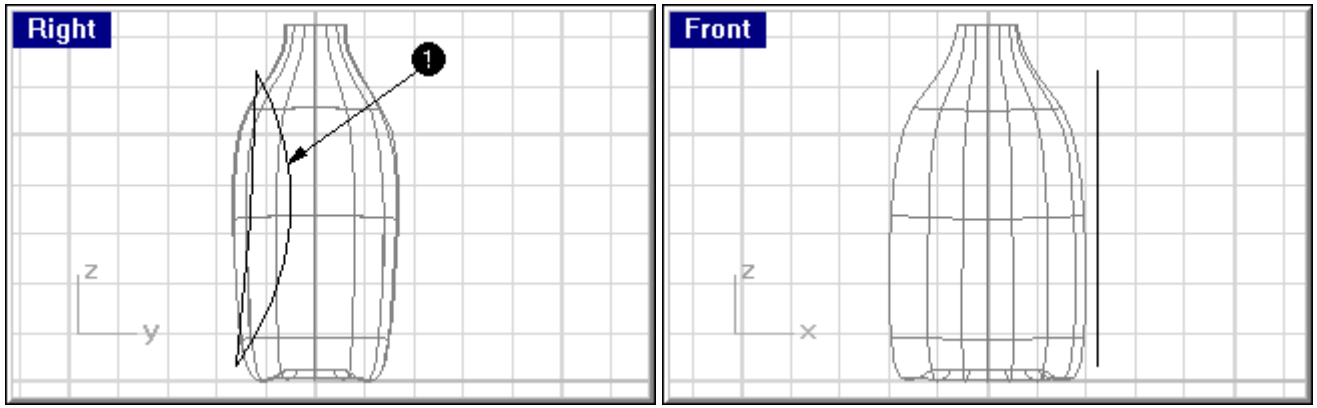
1. Change to the **Curve** layer.
2. In the **Right** view, draw to curves to approximate the shape at the edge of the bottle and the middle of the bottle.

Notes:

3 The technical drawing gives dimensions for these curves, but for this exercise you can just approximate these. If you would like to go further, try to figure out on your own how to draw the curves to the exact specifications.

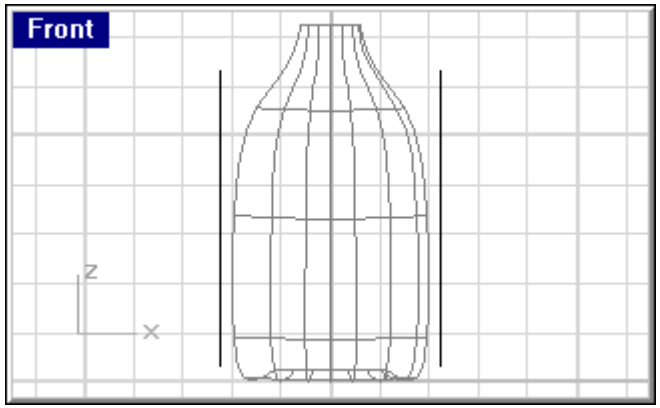


4 In the **Front** viewport move the curve with the smaller radius just outside the edge of the bottle.



Notes:

5 **Mirror** the edge curve to the opposite side.



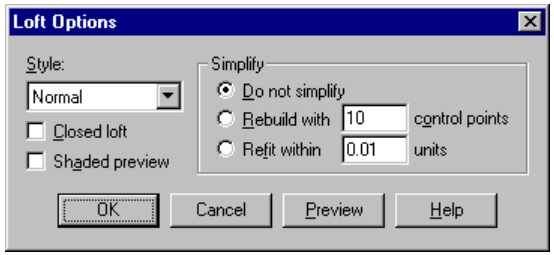
6 Select the three curves you just created.

7 To create a surface defined by the curves you created, from the **Surface** menu, click **Loft.**

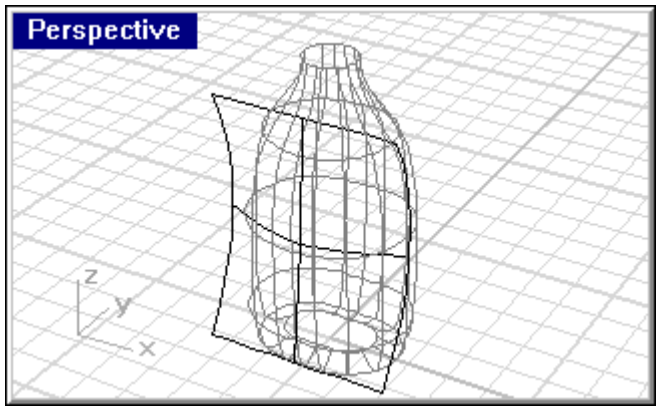
Notes:

Loft
Look for this button.

8 In the **Loft Options** dialog, click **OK**.

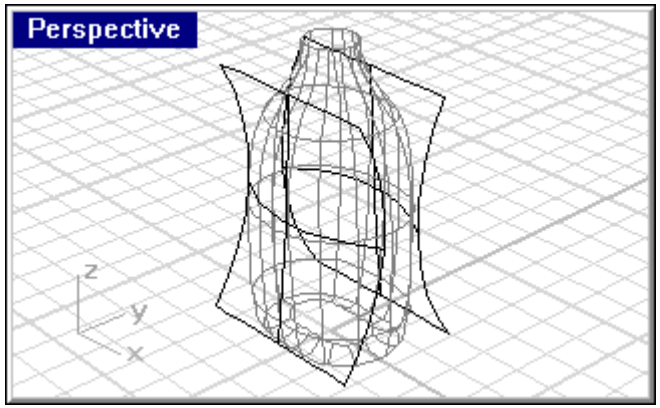


A lofted surface intersects the bottle.



Notes:

9 **Mirror** the surface to the other side of the bottle.



10 **Save** your model.

To remove the surface from the bottle:

1 Change to the **Bottle** layer.

2 From the **Solid** menu, click **Difference.**

3 At the **Select surface or polysurface** prompt, select the bottle.

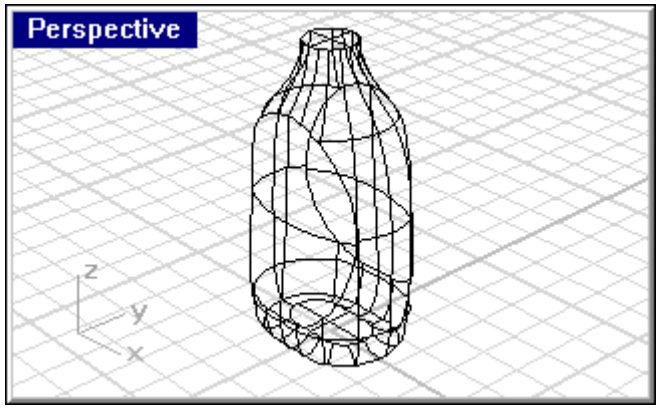
4 At the **Select surface or polysurface to subtract** prompt, select the lofted surface.

Notes:

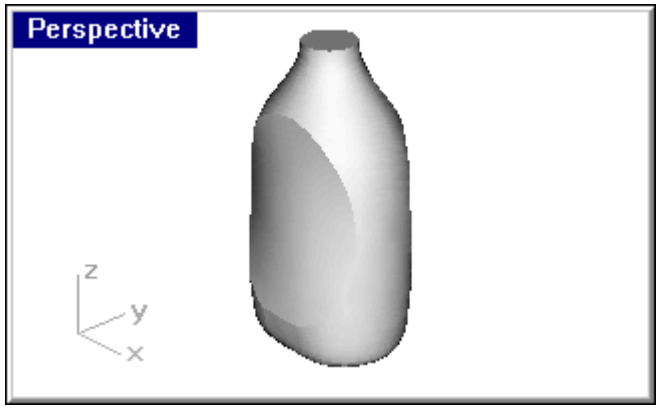
Boolean Difference
Look for this button.

If Difference doesn't work, try Union, if Union doesn't work try Intersection. If one boolean operation doesn't give you the right answer, you can generally try another one until you get the right answer.

5 Repeat this process for the other side of the bottle.



6 **Shade** the bottle to see what it looks like.



Round the Edges of the Shapes

The bottle is now a closed solid. We now want to round off the sharp edges by adding a smooth radius between the edges.

To fillet the edges:

1 From the **Solid** menu, click **Fillet Edge.**

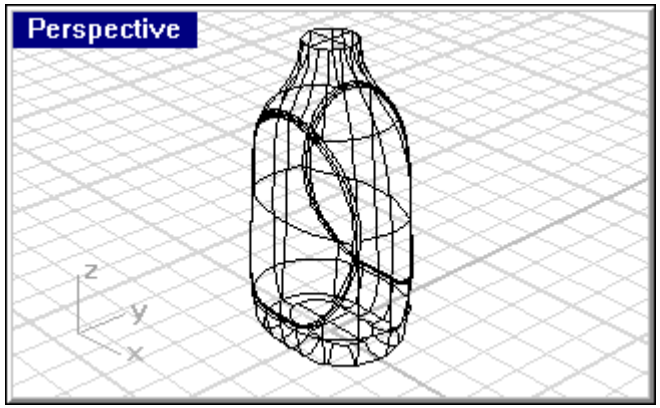
2 At the **Select edges to fillet (Radius=1.0)** prompt, select an edge that was trimmed.

Notes:

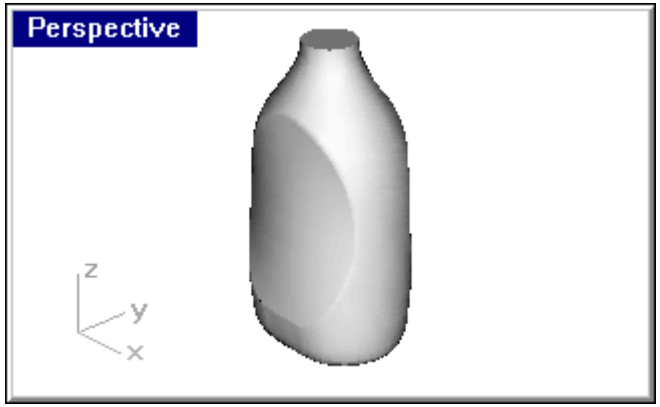
3 At the **Select edges to fillet (Radius=1.0)** prompt, select the other edge that was trimmed.

4 At the **Select edges to fillet (Radius=1.0)** prompt, type **.25** and press ENTER.

5 At the **Select edges to fillet (Radius=0.25)** prompt, press ENTER.



6 **Shade** the bottle to see the results.



Create the Bottle's Top

To create the bottle throat, you are going to **Revolve** a profile curve to create surface, and then add threads.

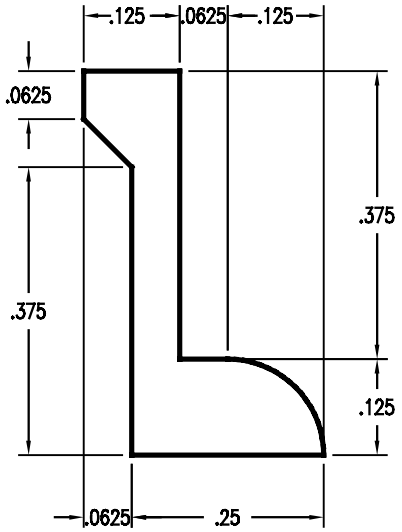
Notes:

Notes:

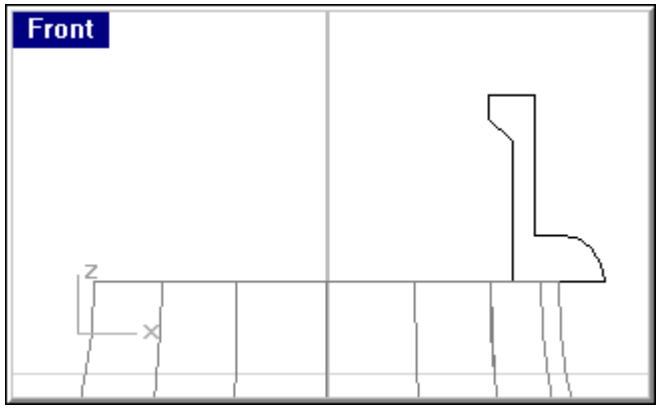
To create the profile curve:

1. Change to the **Curve** layer.
2. In the **Front** view, use the **Line Segments** command and an **Arc** to create a profile curve for the outside and the inside of the top.
3. Begin the first point at **.5,7.25** so it aligns with the profile on the edge of the bottle.
4. Use the technical drawing below to draw the correct dimensions.

   Tip: When drawing line segments, you can use ***object snaps***, the ***distance constraint*** and ***ortho*** to draw precisely. For example, for the first line type **.5,7.25** for the starting point, then simply type **.25** to constrain the line to .25 units. Turn ortho on and drag the line to the right. The line will stop at .25 units from the beginning of the line. For the vertical line on the left, use the End object snap to pick the end of the first line, type .375 to constrain the distance, and drag the line up.



5 **Join** the segments together.



6 Change to the **Bottle Top** layer.

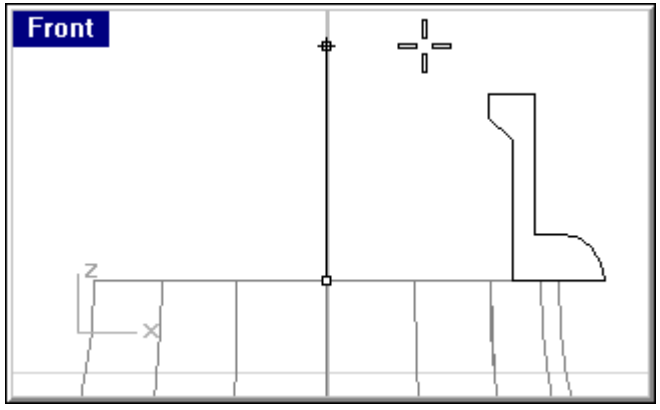
To create the top surface:

1 From the **Surface** menu, click **Revolve.**

2 At the **Select curve to revolve** prompt, select the profile curve.

3 At the **Start of revolve axis** prompt, snap to the center of the circle at the top of the bottle.

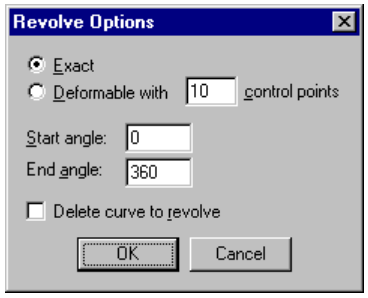
4 At the **End of revolve axis** prompt, pick a point directly above the previous point.



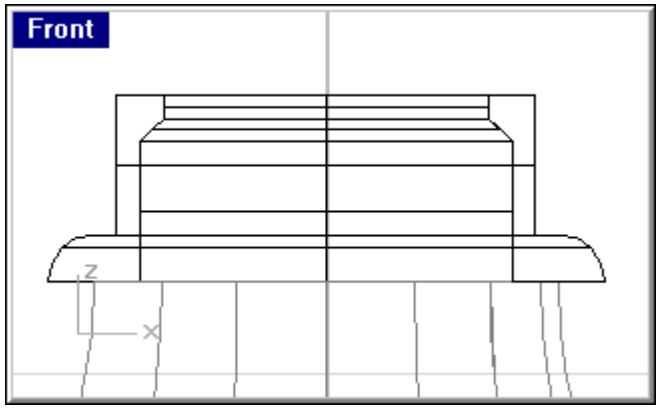
Notes:

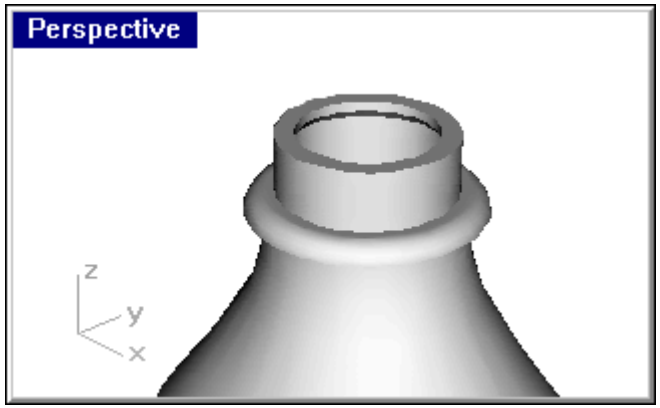
Revolve
Look for this button.

5 In the **Revolution Options** dialog box, click **Delete curve to revolve**, and click **OK**



The top is created.





Adding the threads

To add the threads, you will **Sweep** a profile shape along a path drawn with the **Helix** command.

To create the path curve helix:

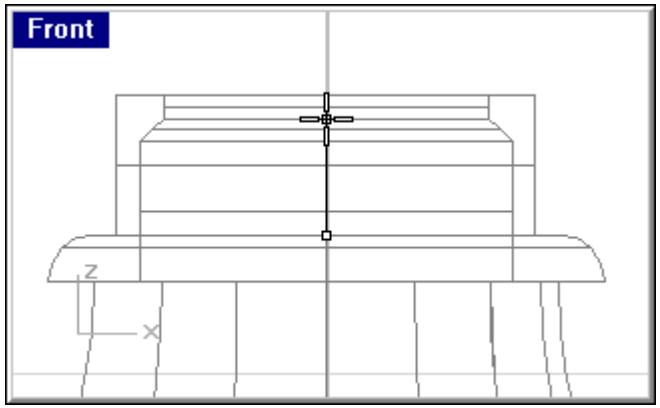
1 Change to the **Curve** layer.

2 From the **Curve** menu, click **Helix.**

Notes:

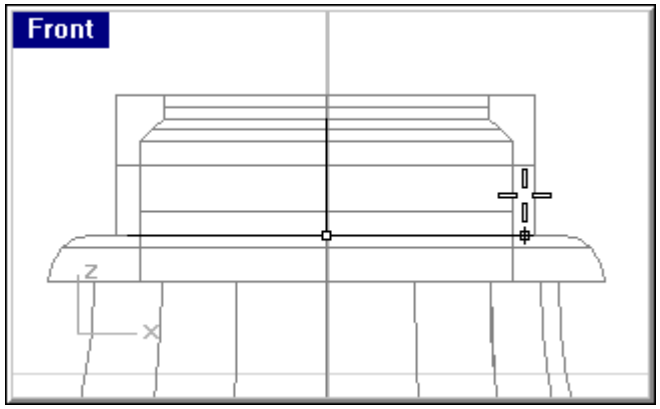
Helix
Look for this button.

3 At the **Start of axis (Vertical AroundCurve)** prompt, snap to the center point at the bottom of the area to be threaded.



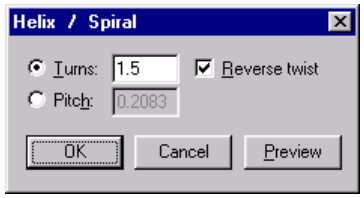
4 At the **End of axis** prompt, snap to the center at the top of the area to be threaded..

5 To set the radius, at the **Radius** prompt, pick a point near the edge that is slightly smaller than the outside edge of the area to be threaded.



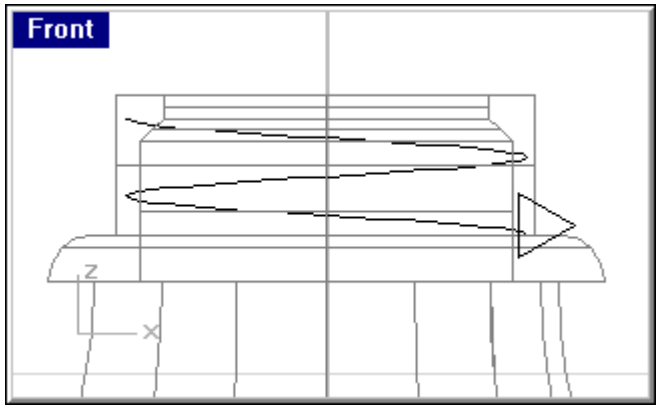
Notes:

6 In the **Helix / Spiral** dialog, change the turns to **1.5**, check **Reverse twist**, then click **OK**.



To draw the thread profile:

1 Draw a triangle using the **Polygon** command for the thread as shown below.



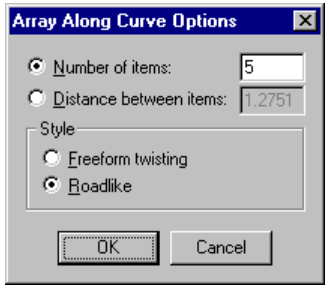
2 Select the triangle you created.

3 From the **Transform** menu, click **Array**, then click **Along Curve.**

4 At the **Select path curve** prompt, select the helix near the bottom.

Notes:

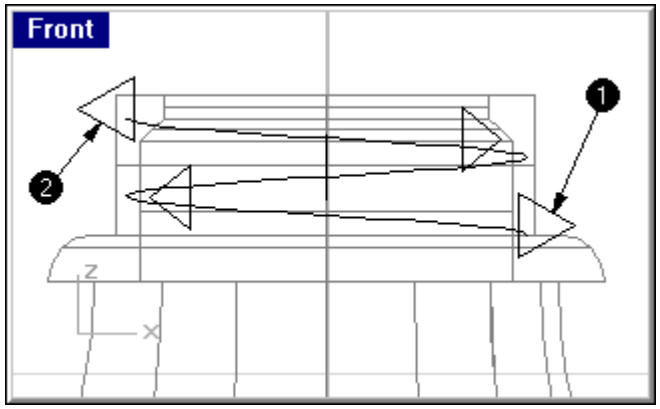
5 In the **Array along curve options** dialog box, in the **Number of items** box, type **5**, under **Style**, click **Roadlike**, and then click **OK.**



6 At the **Click in a viewport to choose a Cplane**, click in the **Top** viewport.

7 It might be a good idea at this point to change your perspective view so you can see the profile curves easier.

8 Delete the first and last profile curve.



9 The remaining profile curves will be used to create a surface.

To create the thread surface:

1 Change to the **Bottle** layer.

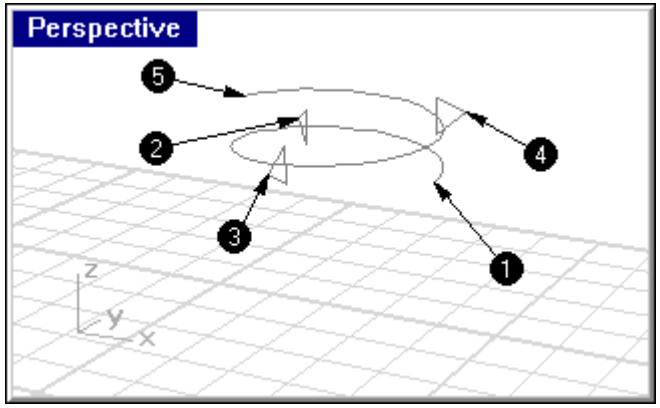
2 You can see the helix and profile curves easier if you **Hide** the bottle top at this point.

Notes:

Use the sweep along path command to create a surface that starts a point at one end of the helix, goes through each profile and ends at a point at the other end of the helix.

3 From the **Surface** menu, click **Sweep 1 Rail.**

4 At the **Select rail curve** prompt, select the helix.



5 To start the surface at a point, at the **Select cross-section curves (Point)** prompt, type **p**, and press ENTER.

6 At the **Start of 1-rail sweep** prompt, use the **End** object snap to pick one end of the helix (1).

7 At the **Select cross-section curves (Point)** prompt, select the first profile curve (2) along the helix.

8 At the **Select cross-section curves (Point)** prompt, select the next profile curve (3) along the helix.

9 At the **Select cross-section curves (Point)** prompt, select the next profile curve (4) along the helix.

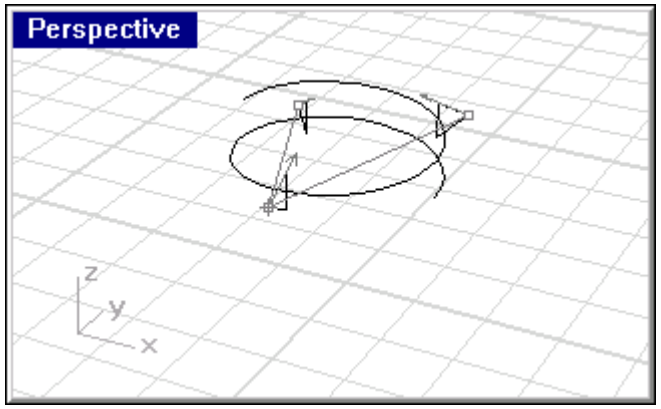
10 To end the surface at a point, at the **Select cross-section curves (Point)** prompt, type **p**, and press ENTER.

11 At the **End of 1-rail sweep** prompt, use the **End** object snap to pick the other end of the helix (5).

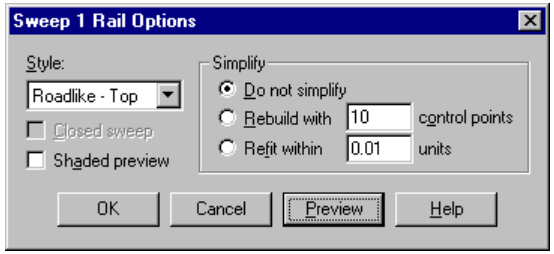
Notes:

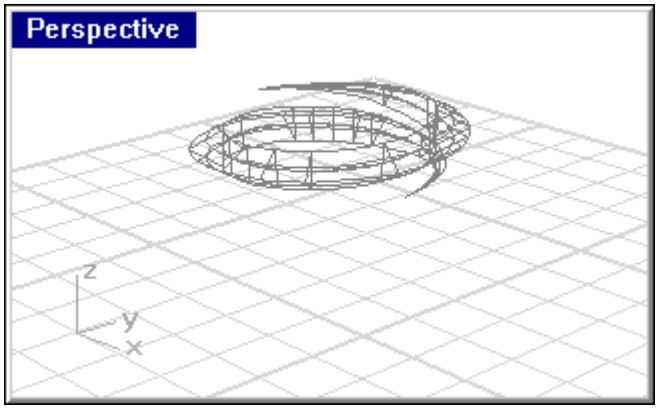
Sweep Along One Rail
Look for this button.

12 At the **Matching seams and directions...Select seam point to adjust. Press Enter when done (FlipDirection Automatic Natural)**prompt, press ENTER.



13 In the **Sweep 1Rail Options** dialog, change the **Style** to **Roadlike - Top**, then click **OK**.





Notes:

14 **Shade** the model.

15 **Render** the model.

Bottle Caps

If the class is not ready to move on, you can practice additional surface modeling techniques by creating the caps for the model. Use the technical drawing to help.

Notes:

Modeling an End Table

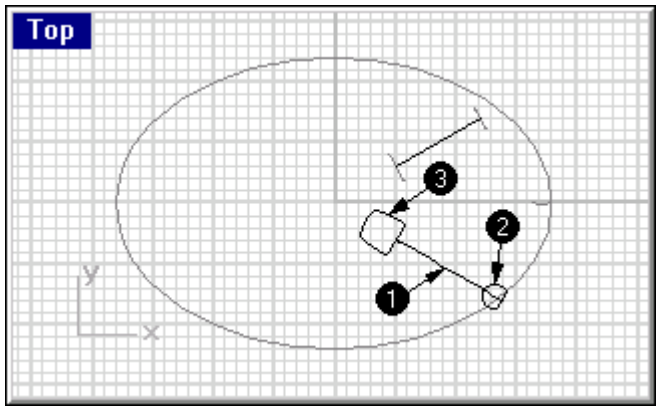
In this exercise you use one rail sweeps to make an end table with freeform tapered legs.

Exercise 62— Practice using One-Rail Sweeps:

- Open the **Table** model.

To create the legs:

1. From the **Surface** menu, click **Sweep 1 Rail.**
2. At the **Select rail curve** prompt, select the path curve (1) for the leg.
3. At the **Select cross-section curves (Point)** prompt, select the shape curve for both ends of the leg (2,3).
4. At the **Select cross-section curves (Point)** prompt, press ENTER.

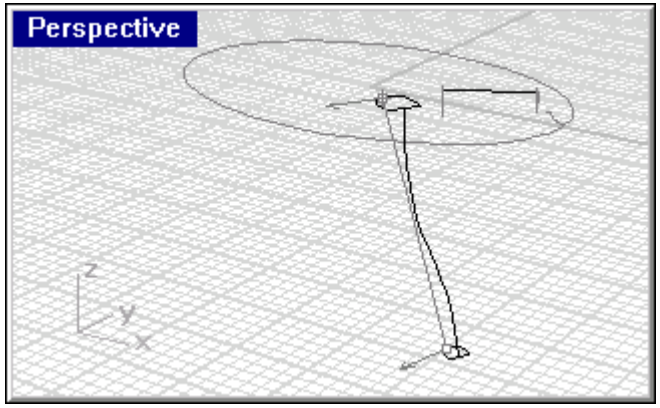


Notes:

Sweep along 1 Rail
Look for this button

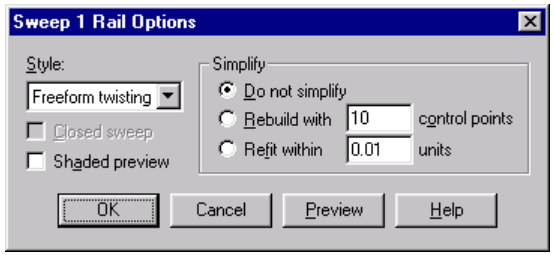
5 At the **Matching seams and directions...**

Select seam point to adjust. Press ENTER when done (FlipDirection Automatic Natural): prompt, press ENTER.

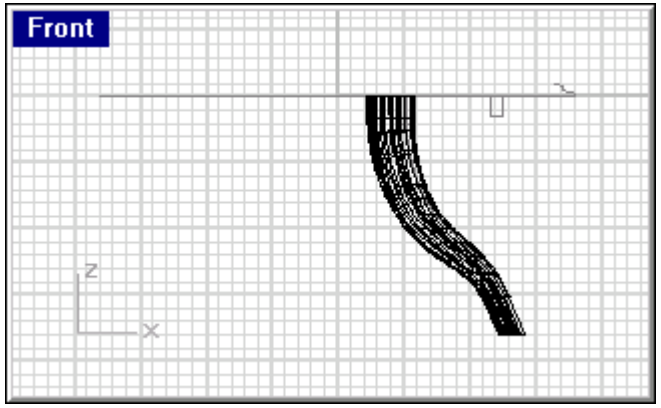


Notes:

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



The table leg is created. Note the nice transition from one cross-section curve to the other.



To create the brace:

1 Change to the **Braces** layer.

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

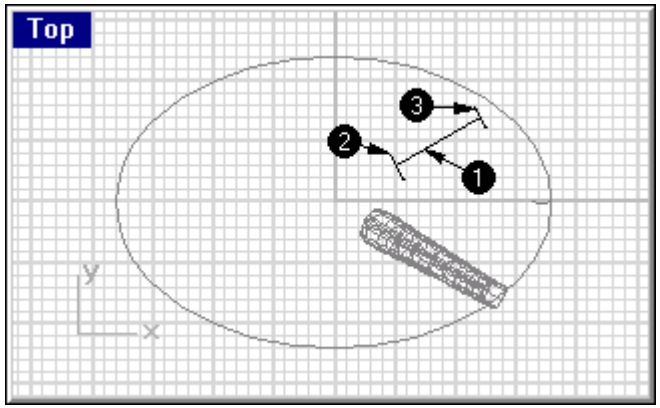
3 At the **Select rail curve** prompt, select the path curve (1) for the brace.

4 At the **Select cross-section curves (Point)** prompt, select the shape curve for both ends of the brace (2,3).

Notes:

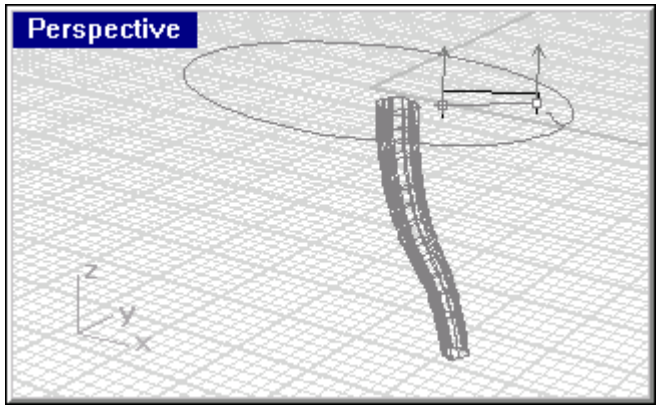
Sweep along 1 Rail
Look for this button

5 At the **Select cross-section curves (Point)** prompt, press ENTER.



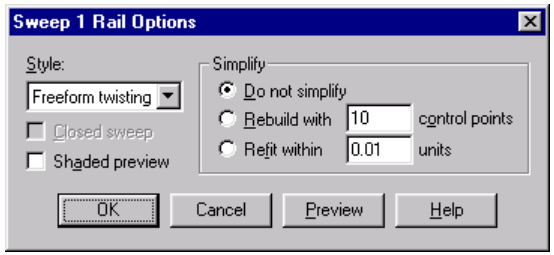
6 At the **Matching seams and directions...**

Select seam point to adjust. Press ENTER when done (FlipDirection Automatic Natural): prompt, press ENTER.

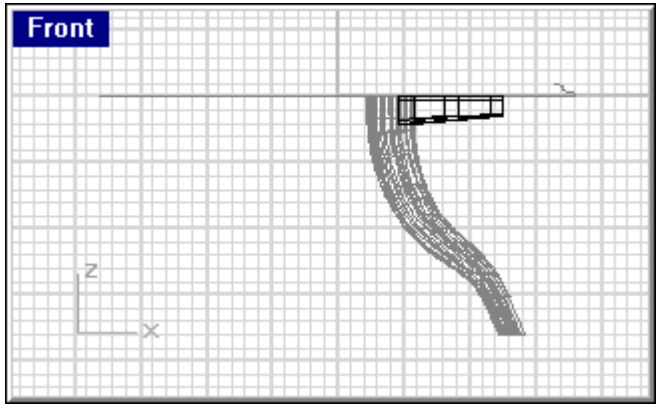


Notes:

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



The table brace is created.



To create the top:

1 Change to the **TOP** layer.

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

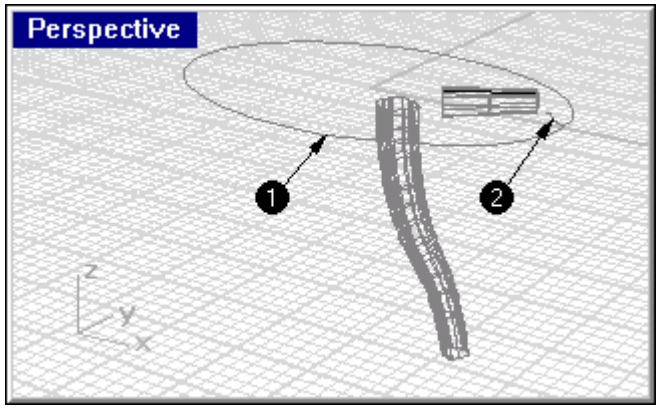
3 At the **Select rail curve** prompt, select the ellipse.

4 At the **Select cross-section curves (Point)** prompt, select the shape curve for the top.

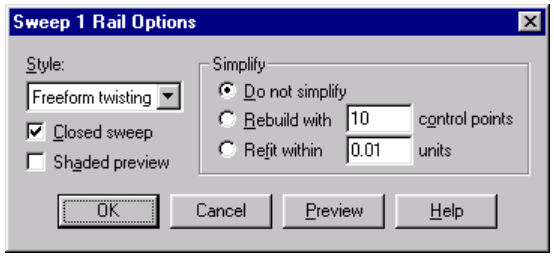
Notes:

Sweep Along Path
Look for this button.

5 At the **Select cross-section curves (Point)** prompt, press ENTER.

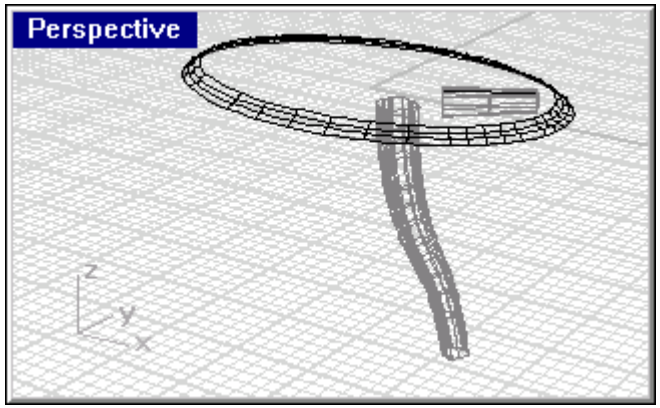


6 In the **Sweep 1 Rail Options** dialog, click **OK**.



Notes:

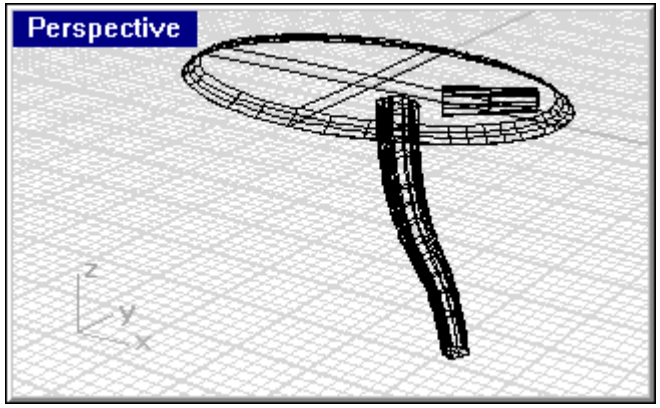
The surface for the rim of top is created.



To finish the table:

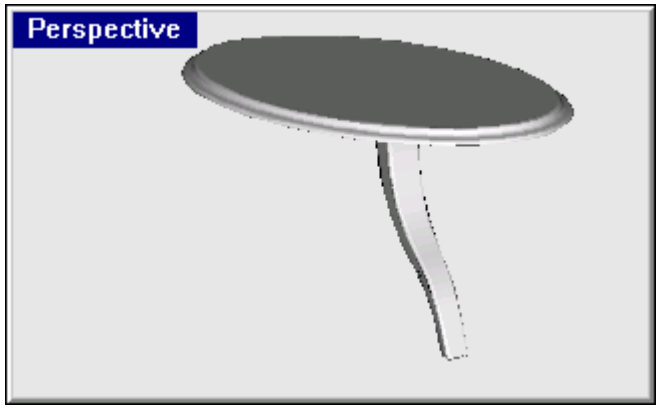
1. Select all of the surfaces you created.
2. From the **Solid** menu, click **Cap Planar Holes.**

   Six caps were created.

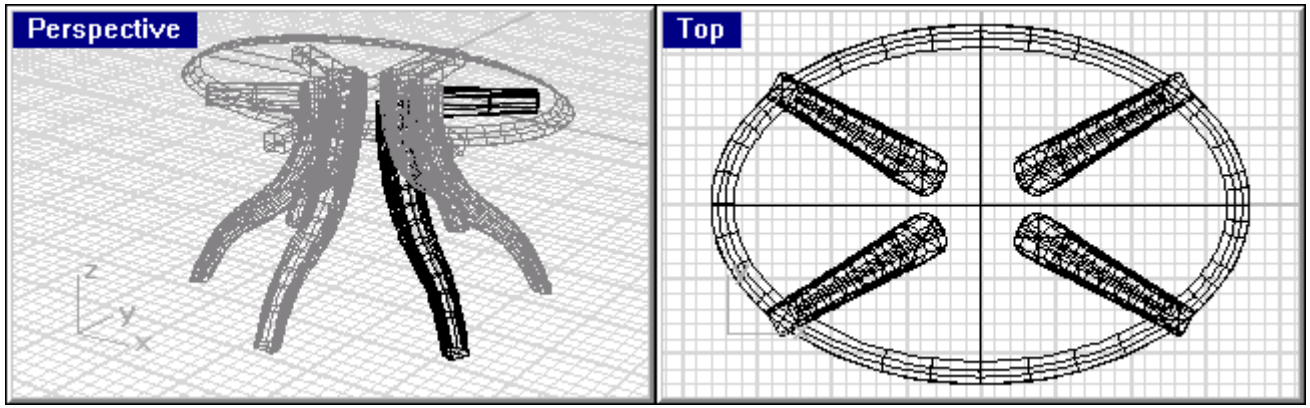


Notes:

3 **Shade** the **Perspective** viewport.

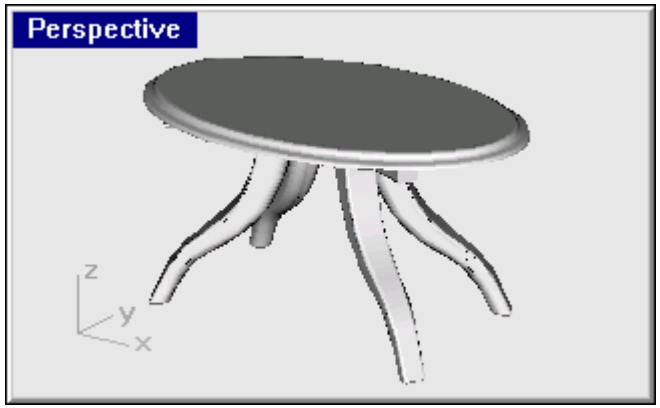


4 Polar array the brace and the leg to finish the model. You will need to array them around 0,0 in the Top viewport. Array six items, but you will only need four legs and braces. Delete the extras.



Notes:

5 **Shade** the **Perspective** viewport.



6 **Render** the **Perspective** viewport.

7 **Save** your model. We will use this in Exercise 63.

Notes:

11 Importing and Exporting Models

Rhino supports many different import and export formats, making it possible to model in Rhino and then export your model to downstream processes. For a complete list of import and export options refer to Rhino Help Topic "Importing and exporting Rhino file information."

Supported formats	Description
3DS MAX	Rhino has free plug-ins for 3DS MAX that will read Rhino mesh objects into 3DS MAX. The plug-in for MAX2 will import some NURBS objects.
	Rhino also writes 3DS files.
AutoCAD DWG/DXF	Reads and writes R12, R13, and R14.
IGES	IGES files contain NURBS geometry, object property, layer, and notes information.
	Export options are included for CamSoft, Catia, FastSURF, Mastercam, HP ME30, Mechanical Desktop, MicroStation, Pro/E NT, Pro/E UNIX, SCRC's I-DEAS, SOFTIMAGE, Solid Edge, SolidWorks, TekSoft, Unigraphics, and custom IGES types (click New Type).
	Rhino's IGES importer will read all IGES files.
Adobe Illustrator	Exports the curves as a 2-D snapshot from the current view.
Lightwave LWO	All Rhino objects are exported as polygonal mesh objects.
POV-Ray	
Wavefront OBJ	Supports meshes and NURBS.
RenderMan RIB	Exports nurbs objects as RenderMan .rib files.

Notes:

Notes:

Supported formats	Description
STL	Reads and writes STL files.
Windows Metafile	A 2-D representation of the current viewport is exported to the WMF file.

Importing and exporting Rhino file information

When you export to a format like 3DS or DWG, Rhino has to convert from smooth NURBS surfaces to a polygon mesh representation made of triangles. To do a good approximation of the curved surfaces, Rhino can sometimes use a lot of polygons. The density of triangles can be adjusted when exporting. You can create a mesh to export or Rhino can create the mesh during the export process.

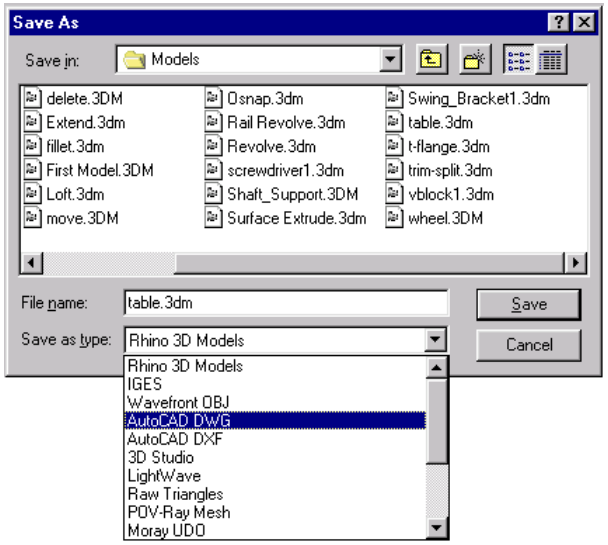
There are two methods to export models to other formats. You can "Save As" and choose a specific export format to export an entire model. You can select some objects and "Export Selected" then choose a specific export format to export a portion of the model. In the following exercise both methods will be used.

Exercise 63— Exporting Models:

To export a model to AutoCAD:

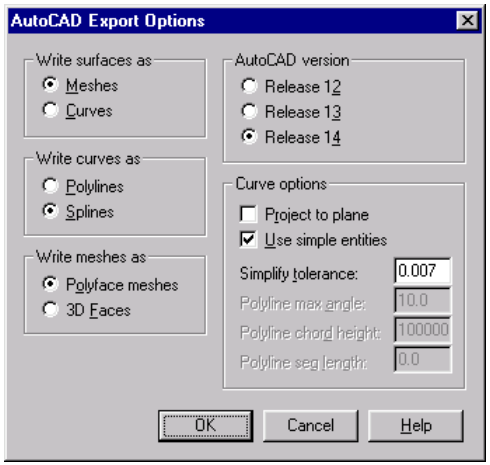
1. Open the **Table** model.
2. From the **File** menu, click **Save As**.

3 In the **Save As** dialog, change the **Save as type** to **AutoCAD DWG**.



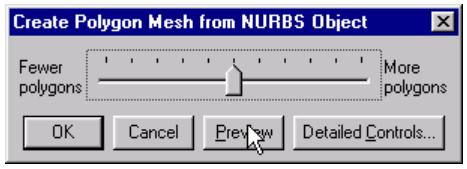
4 In the file name box type **Table** and click **Save**.

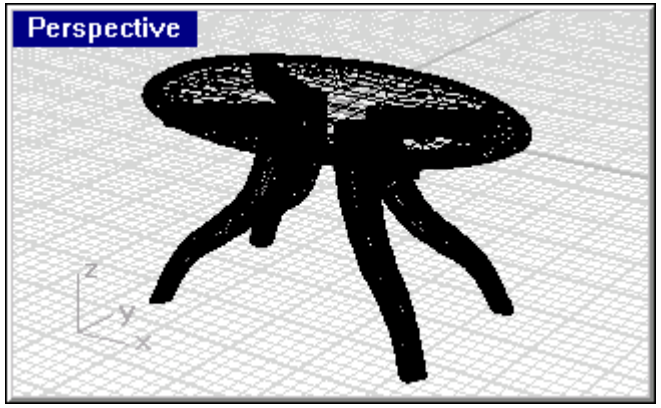
5 In the **AutoCAD Export Options** dialog, **Write Surfaces** as **Meshes**, **Write Curves** as **Splines**, **Write Meshes** as **Polyface Meshes**, AutoCAD Version **Release 14**, check **Use Simple Entities**, and click **OK**.



Notes:

6 In the **Create polygon mesh from NURBS object** dialog, click **Preview**.

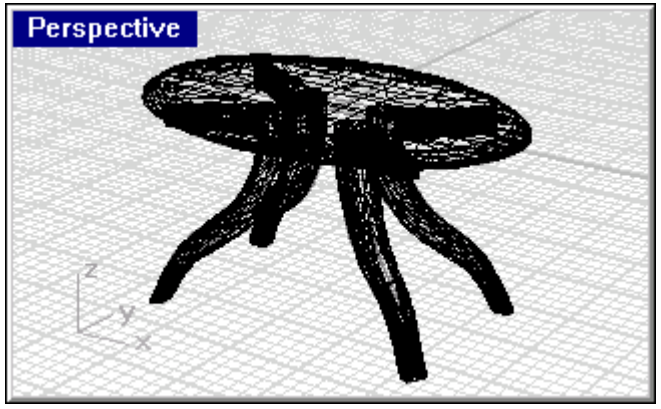




7 Adjust the slider all the way to the left for **Fewer polygons** and click **Preview**.

Note that the table top mesh is very angular, not a good approximation.

8 Move slider to the right and **Preview** until the mesh is good, then click **OK**.



Notes:

9 When this drawing is opened in AutoCAD it will be made up of meshes.

This would be useful for rendering with a third-party program like AccuRender®.

To export a model to IGES:

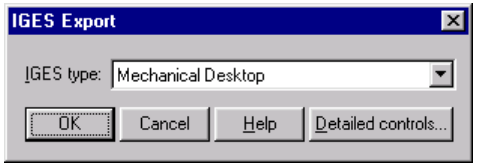
1 Select one of the table legs.

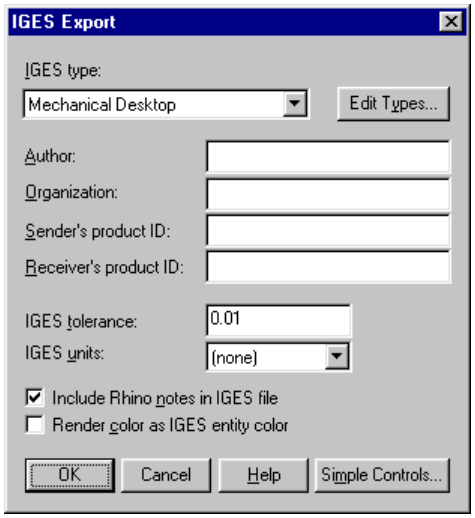
2 From the **File** menu, click **Export Selected**.

3 In the **Export Selected Objects** dialog, change the **Save As Type** to **IGES**, type **LEG** in the **File N**ame window, then click **Save**.

4 In the **IGES Export** dialog, select **Mechanical Desktop** as the **IGES type**, click **Detailed Controls**, the click **OK**.

Detailed controls allows the user more input.





5 Click **Cancel** to terminate, or **OK** to create the IGES file.

Notes:

Creating mesh objects

Although Rhino uses NURBS as its primary construction tool, you may want to turn some of your objects into triangle meshes.

If you are exporting objects to another application that uses meshes like 3D Studio, it is often best to convert the object to a mesh first so you have more control.

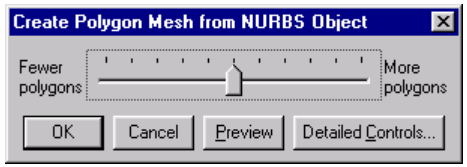
Button	Command	Description
	Mesh from NURBS object	Creates approximating meshes from an NURBS objects.
	Mesh Plane	Creates a rectangular mesh plane.
	Mesh Box	Creates a rectangular mesh box.
	Mesh Cylinder	Creates a mesh cylinder.
	Mesh Cone	Creates a mesh cone.
	Mesh Sphere	Creates a mesh sphere.
	Apply Mesh to NURBS Surface	Creates a morph target for animation programs. Creates a mesh from a surface.
	Explode Mesh	Explodes a mesh into component mesh surfaces.
	Hide Control Points	Hides part of a mesh by hiding selected control points.
	Show Control Points	Shows all hidden control points of a mesh.

Notes:

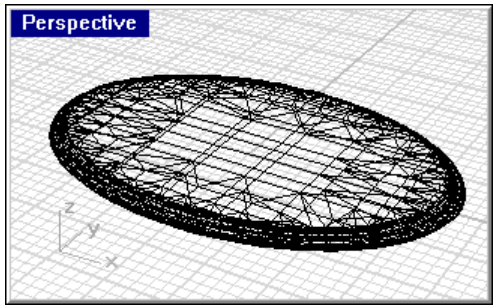
Exercise 64— Practice creating a mesh for export:

To Create a Mesh:

1. Select the Table Top.
2. From the **Tools** menu, click **Polygon Mesh**, then click **From NURBS Object.**.
3. In the **Create polygon mesh from NURBS object** dialog, click **OK**.



The mesh can be exported. This gives you more direct control over the meshes you export to programs that require meshes.



Notes:

Mesh from NURBS object
Look for this button.

Notes:

Making a 2D drawing from a 3D model

Rhino has the ability to generate a two-dimensional drawing from a three-dimensional model, by projecting the geometry to the world coordinate plane, and align the views. Options for first angle projection or third angle projection are available. In addition to the three orthographic views, a two dimensional pictorial drawing is also generated. The geometry may have duplicate segments and hidden lines will not be removed.

Options for creating 4 views: three parallel views and a perspective view, or single views of individual viewports are supported.

Make2D creates a two-dimensional drawing from a model of one viewport. Select the model in the viewport that displays the view you want to make. Make2D4View makes a two-dimensional drawing for all four views. Select the model in the perspective viewport to get all four projections. If you select the model in a parallel viewport, you will get three views excluding the perspective view.

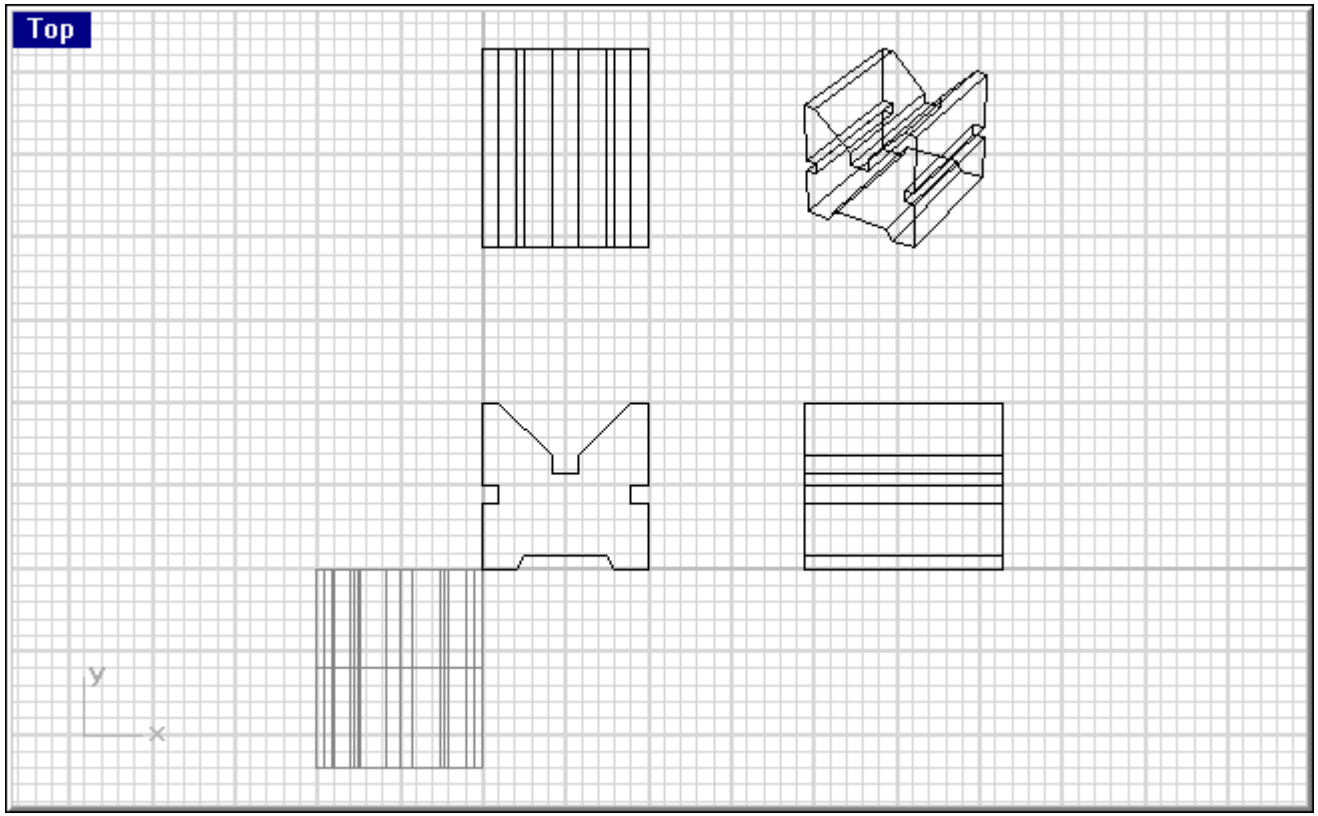
Exercise 65— Practice making a 2D drawing for export:

To create a 2D drawing from a 3D model:

1. Open the **Vblock1** model.
2. From the **Curve** menu, click **From Objects**, then click **2-D Utilities**, then click **Make 2-D Drawing with 4 Views.**
3. At the **Select objects to draw (ProjectionAngle=US)** prompt, select the V-Block in the perspective view.
4. At the **Select objects to draw (ProjectionAngle=US)** prompt, press ENTER.

   The 2D drawings are created in the Top viewport near the origin on the World xy plane. View them in the Top viewport.
5. Maximize the **Top** viewport.

6 **Zoom Extents**.



To export the 2D drawing to AutoCAD:

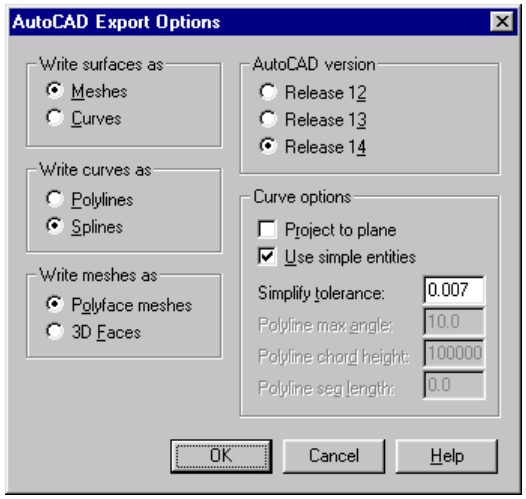
1 From the **Edit** menu, click **Select**, then click **All**, click **On Layer…**, then click **2d drawing**.
The 2D objects are highlighted.

2 From the **File** menu, click **Export Selected**.

3 Change the **Save as type** to **AutoCAD DWG,** the **File name** to **Vblock1.dwg,** then click **Save.**

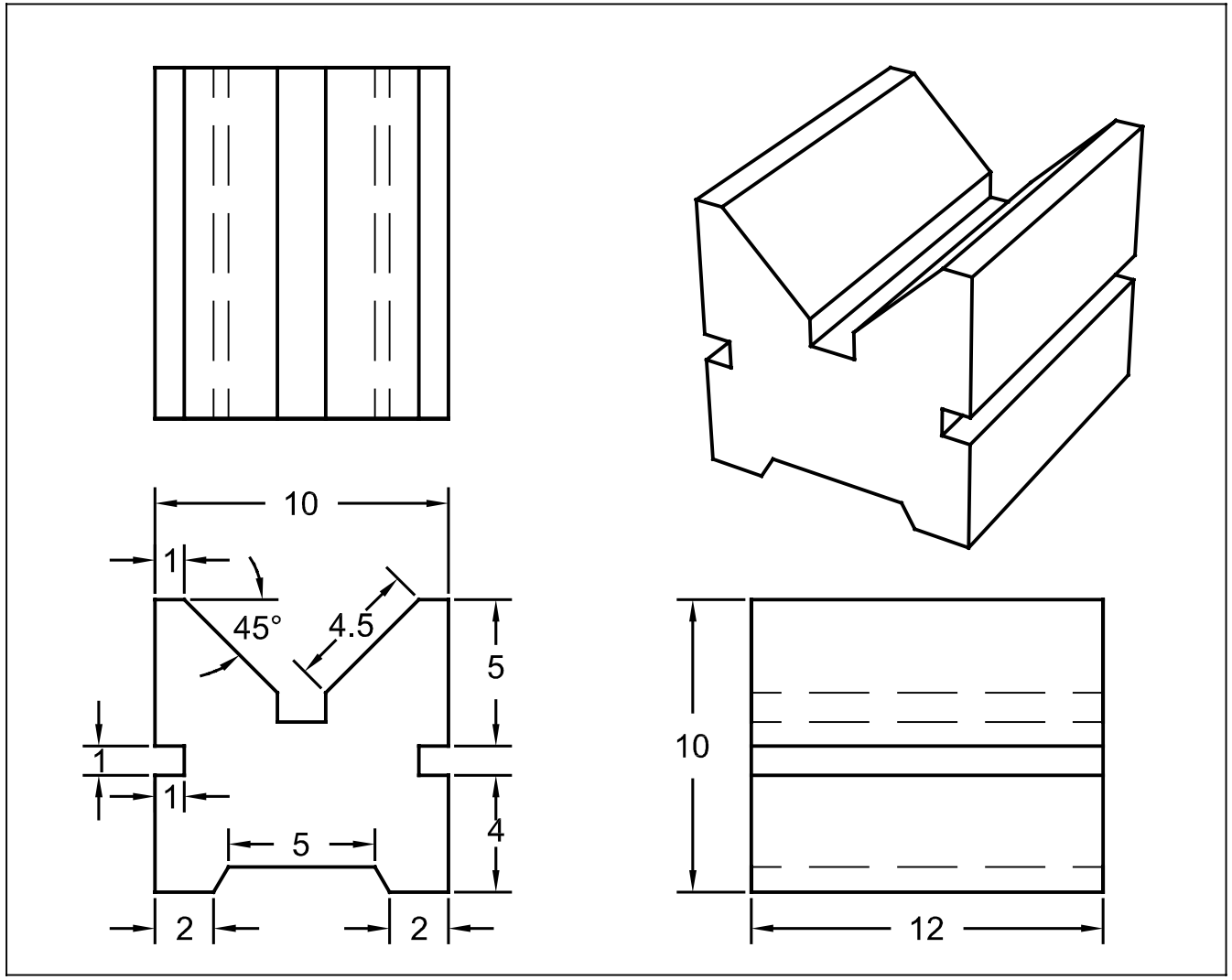
Notes:

4 In the **AutoCAD Export Options** dialog, **Write Surfaces** as **Meshes**, **Write Curves** as **Splines**, **Write Meshes** as **Polyface Meshes**, AutoCAD Version **Release 14**, check **Use Simple Entities**, Simplify tolerance **0.007**, click **OK**.



Notes:

5 Start AutoCAD and open the **Vblock1.DWG**. The drawing can be dimensioned, linetypes can be changed, and layers can be added for plotting.



Notes:

12 Rendering

Rendering your model

Rendering is available for showing your model as if it was photographed. While the Rhino renderer may be good enough for much of your work, use a separate rendering application for higher quality results.

There are two renderers in Rhino:

- A quick renderer, which does a gray-scale shaded preview image.
- A more sophisticated renderer, which uses color, uses spotlights, displays shadows and does anti-aliasing. It also allows the attachment of textures and bump maps. In this exercise we will focus on the full renderering capability.

Exercise 66— Practice rendering a model:

To render a model:

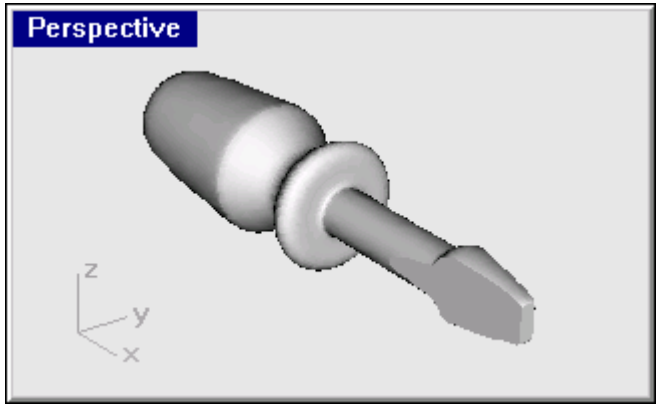
1. Open the **Screwdriver1** model.
2. Click in the **Perspective** viewport.

Notes:

Shade
Look for this button.

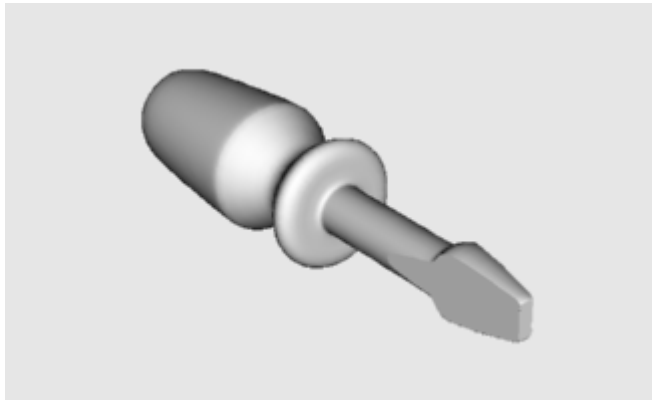
3 From the **Render** menu, click **Shade.**

The viewport is render in gray tones.



4 From the **Render** menu, click **Render.**

A display window appears with the current viewport rendered in gray tones. You can close the Display Window without disturbing your model. Now we will place some lights, then add color and shine to the model.



Notes:

Render
Look for this button.

To place a light:

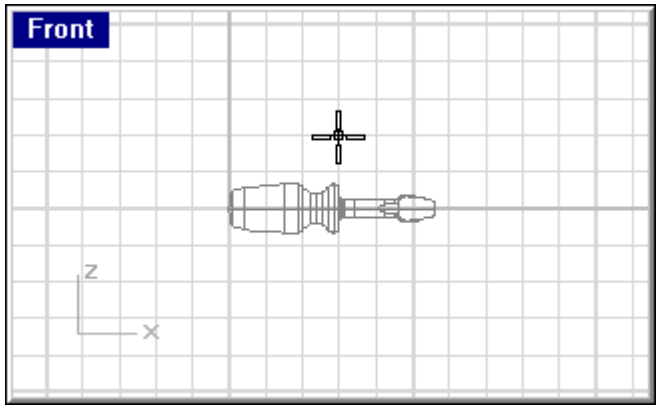
We will start with a standard lighting scheme. You can experiment to develop your own lighting schemes later.

1 **Zoom** out in the **Top** and the **Front** viewport.

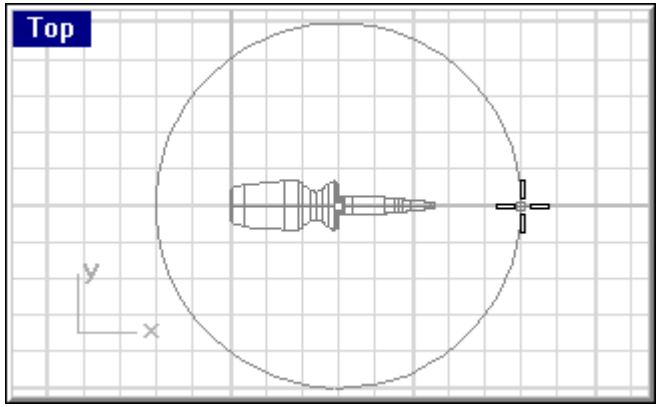
2 Change to the **Light 1** layer.

3 From the **Render** menu, click **Create Spotlight.**

4 At the **Base of cone (Vertical)** prompt, pick a point near the center of the screwdriver and slightly above it in the **Front** viewport.



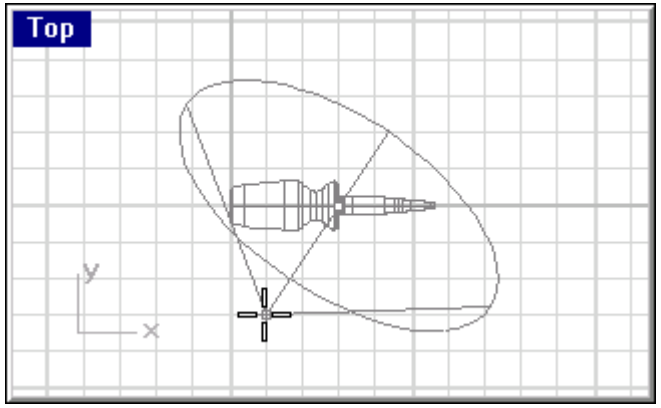
5 At the **Radius <1> (Diameter)** prompt, pick a point so that the radius is larger than the entire screwdriver in the **Top** viewport.



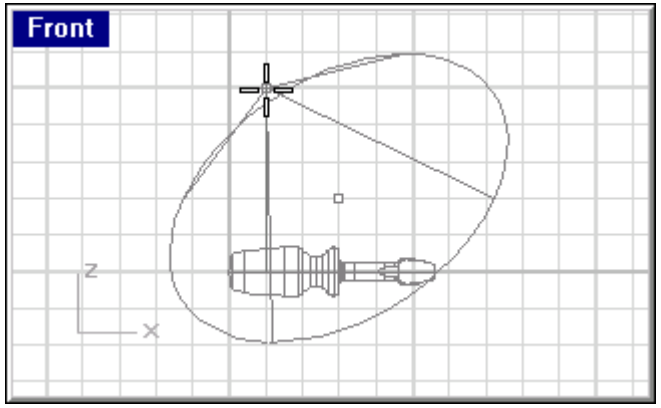
Notes:

Spotlight
Look for this button.

6 At the **End of cone** prompt, hold down the control key a pick a point below and to the left in the **Top** viewport.
This will start elevator mode.



7 At the **End of cone** prompt, click above the object in the **Front** viewport.
This will be your main light.

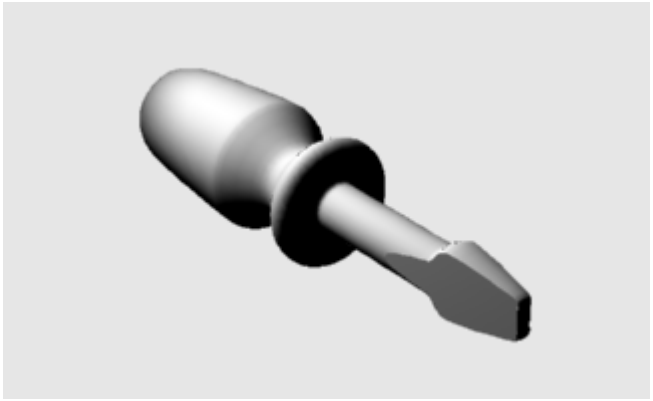


8 Click in the **Perspective** viewport.

Notes:

9 From the **Render** menu, click **Render.**

The image has some highlights and shadows.



To place a second light:

1 **Zoom** out in the **Top** and the **Front** viewport.

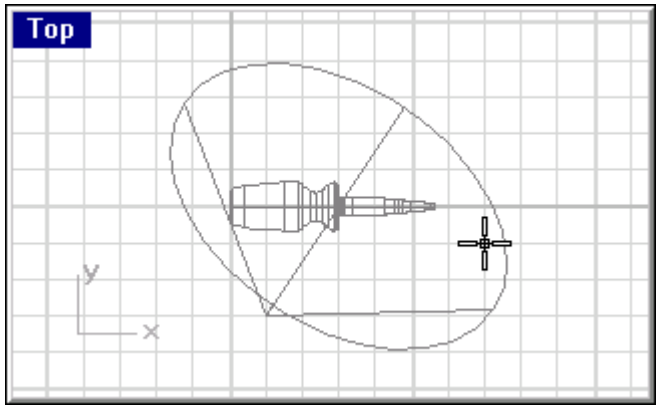
2 Change to the **Light 1** layer.

3 From the **Render** menu, click **Create Spotlight.**

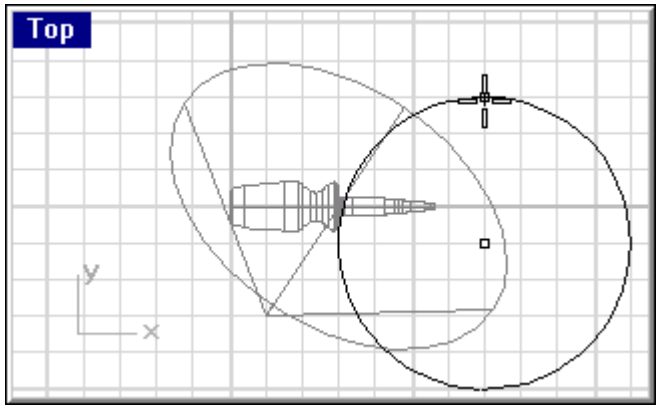
Notes:

Spotlight
Look for this button.

4 At the **Base of cone (Vertical)** prompt, pick a point to the right and slightly below the screwdriver and covers about half of it in the **Top** viewport.



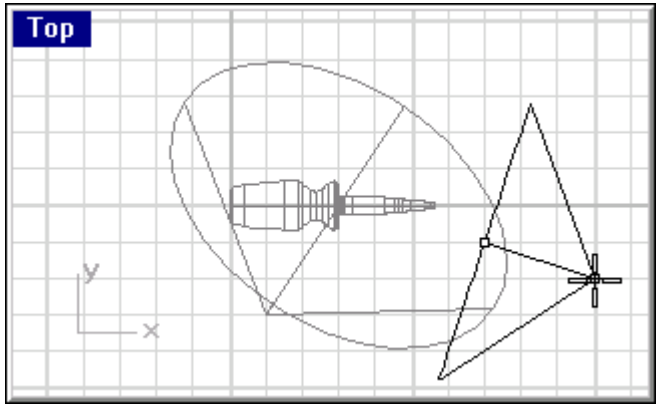
5 At the **Radius <1> (Diameter)** prompt, pick a point so that the radius is larger than the entire screwdriver in the **Top** viewport.



Notes:

6 At the **End of cone** prompt, click to the right and slightly below the object in the **Top** viewport.

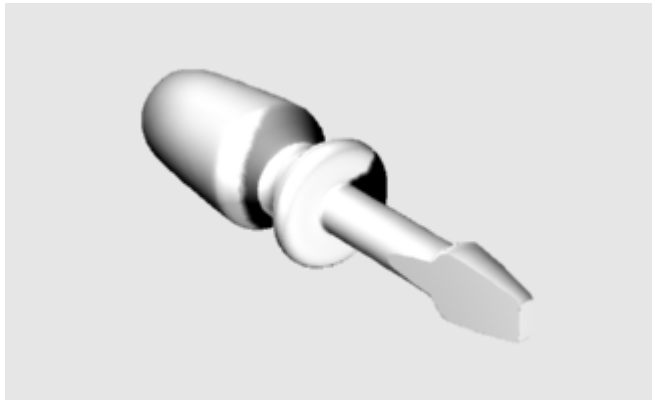
This will be your secondary (fill) light.



7 Click in the **Perspective** viewport.

8 From the **Render** menu, click **Render.**

The image is washed out on the front edge. Next, we will tone down the fill light to fix this problem.



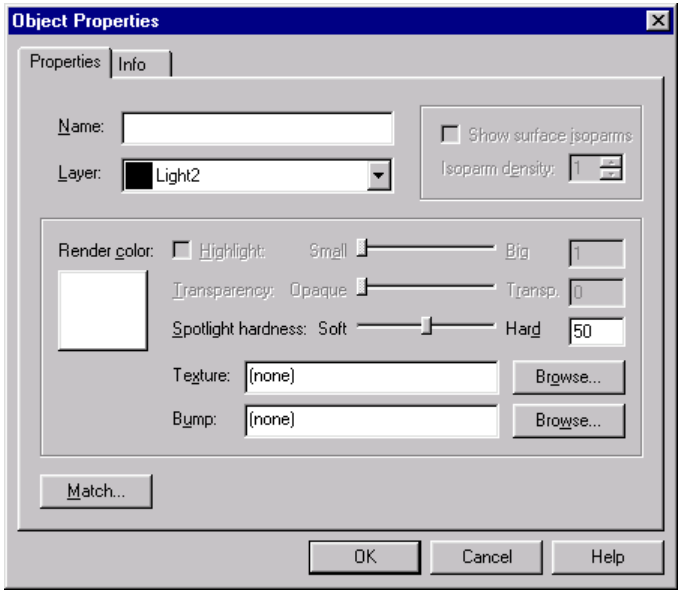
To assign a color to the light:

1 Select the secondary light cone.

Notes:

2 From the **Edit** menu, click **Object Properties.**

The **Object Properties** dialog appears.

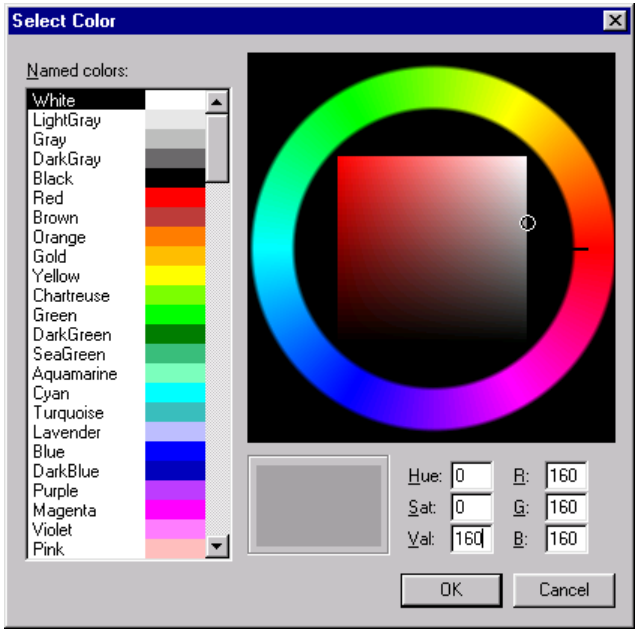


Notes:

Object Color
Look for this button.

3 In the **Object Properties** dialog, click the **Render color**.

The Choose a Color dialog box appears.



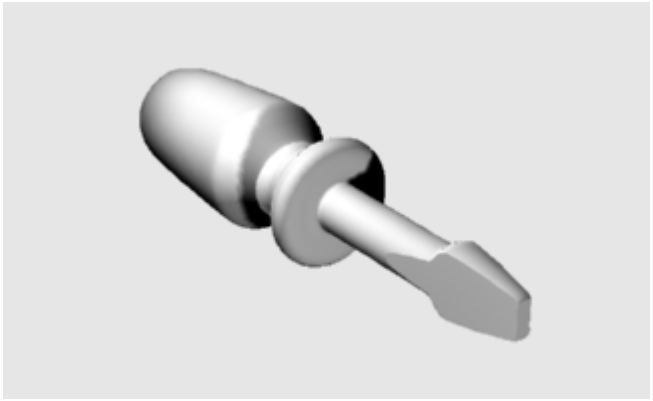
4 Type a **value** of **160** and click **OK**.

5 In the **Object Properties** dialog, click **OK**.

Notes:

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

The image is not washed out. The lighting can be adjusted as necessary.



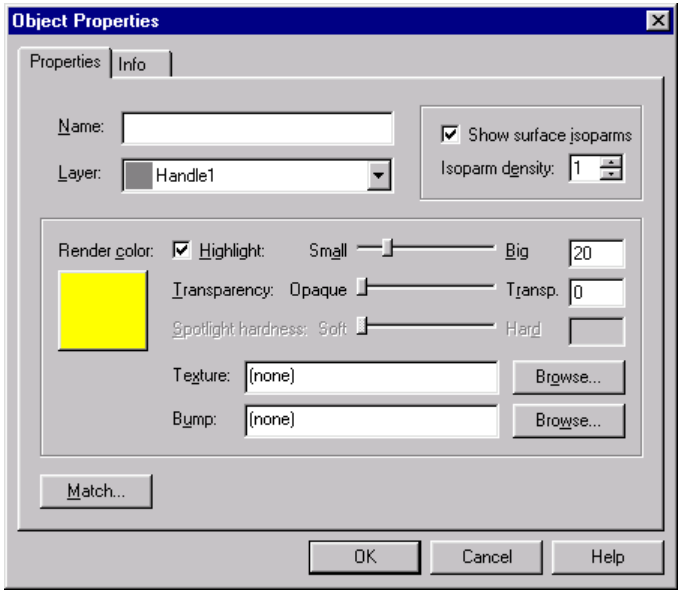
To assign a color to the handle:

1 Select the handle.

Notes:

2 From the **Edit** menu, click **Object Properties.**

The **Object Properties** dialog appears.

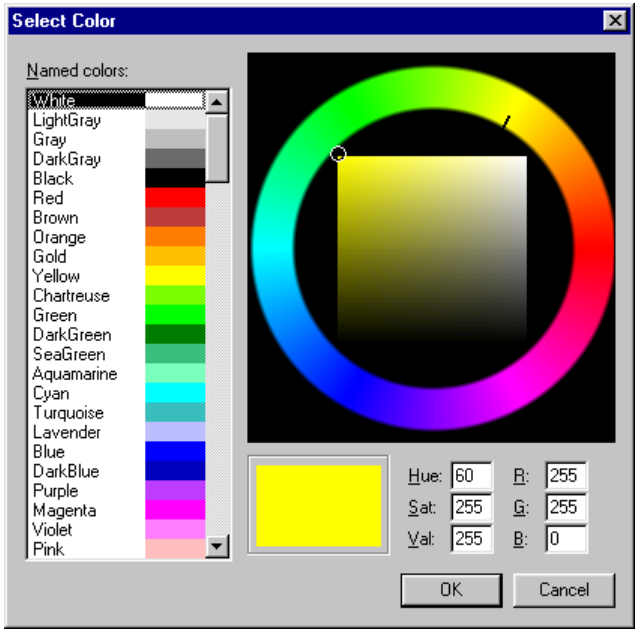


Notes:

Object Color
Look for this button.

3 In the **Object Properties** dialog, click the **Render color** swatch.

The Choose a Color dialog box appears.



4 Pick a color, like Yellow, then click **OK**.

5 To give the handle a highlight, check the **Highlight** box. Start with a value of 20.

A highlight of 0 means that the object isn't shiny at all and it won't have a specular highlight. A low value of highlight makes the shiny spot to be small, and the object will look more glossy. As highlight increases in value, the shiny spot gets large - this makes the object look more like it is made out of a reflective material.

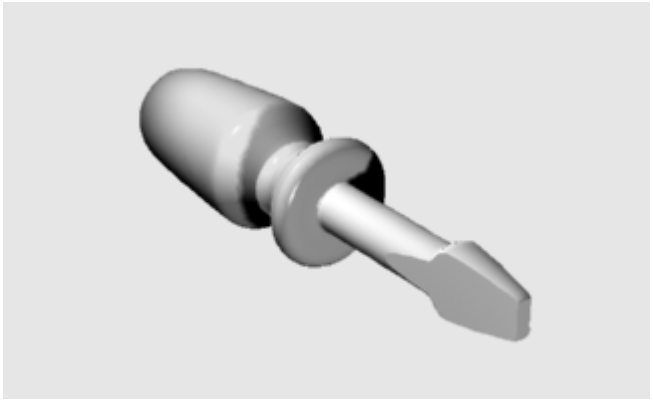
The shiny spot only appears when you are looking at an object at a certain angle relative to the angle of the light.

6 In the **Object Properties** dialog, click **OK.**

Notes:

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

The highlights should look better.



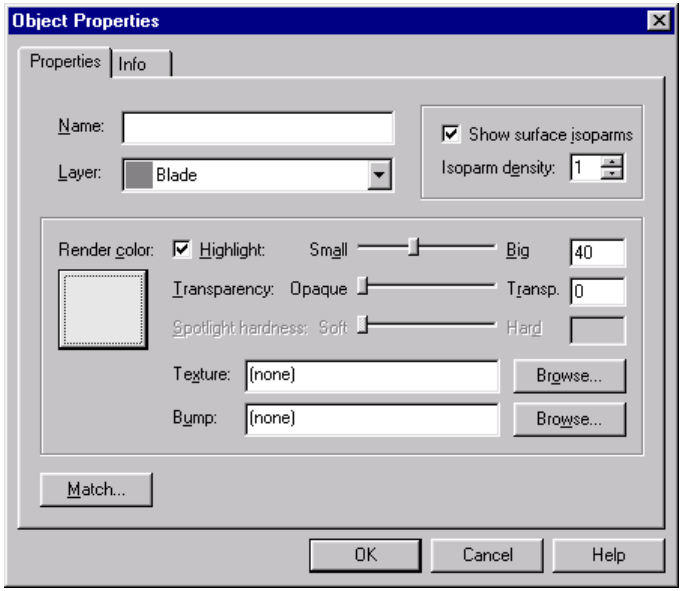
To assign a color to the blade:

1 Select the blade.

Notes:

2 From the **Edit** menu, click **Object Properties.**

The **Object Properties** dialog appears.

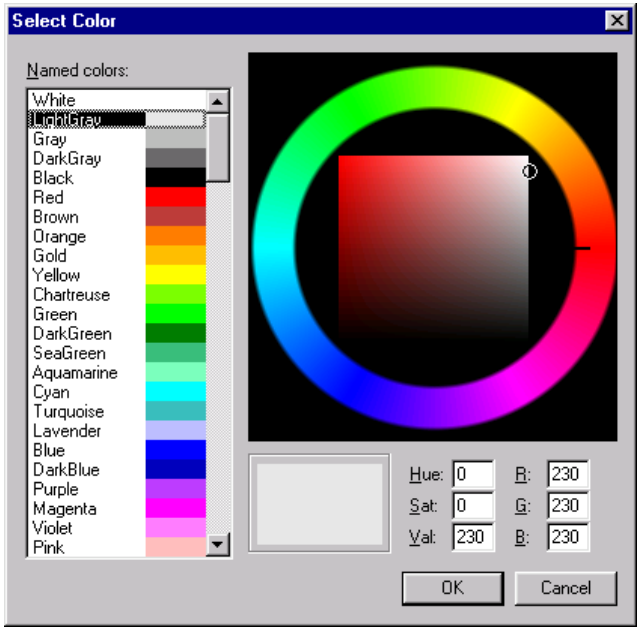


Notes:

Object Color
Look for this button.

3 In the **Object Properties** dialog, click the **Render color**.

The Choose a Color dialog box appears.



4 Pick a color, like Light Gray, then click **OK**.

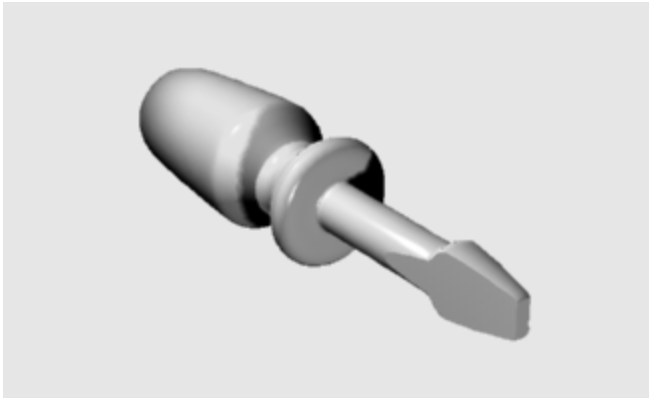
5 To give the blade a highlight, check the **Highlight** box. Start with a value of 40.

6 In the **Object Properties** dialog, click **OK.**

Notes:

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

The highlights should look better.



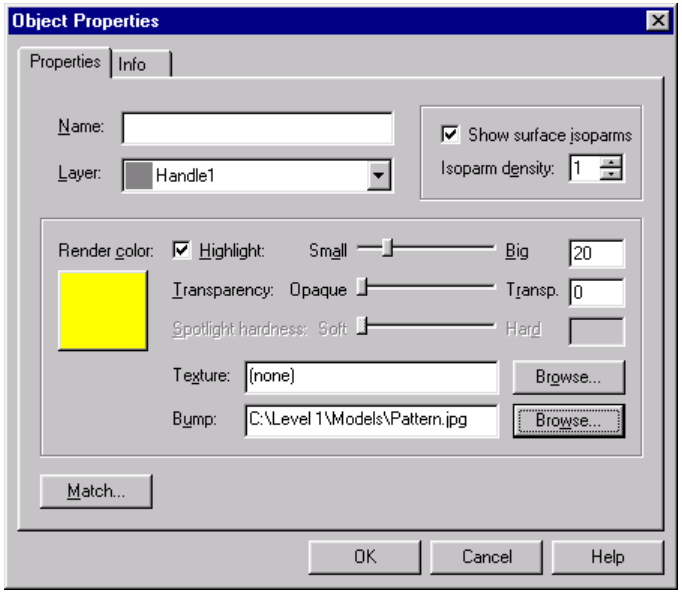
To add a bumpy surface to the handle:

1 Select the handle.

Notes:

2 From the **Edit** menu, click **Object Properties.**

The **Object Properties** dialog appears.



3 In the **Object Properties** dialog, click the **Browse** button next to **Bump**.

You can use any bitmap file for a bump. The bumps come from the pattern of light and dark in the bitmap image.

4 In the **Open bitmap file** dialog, select **Pattern.jpg**, then click **Open**.

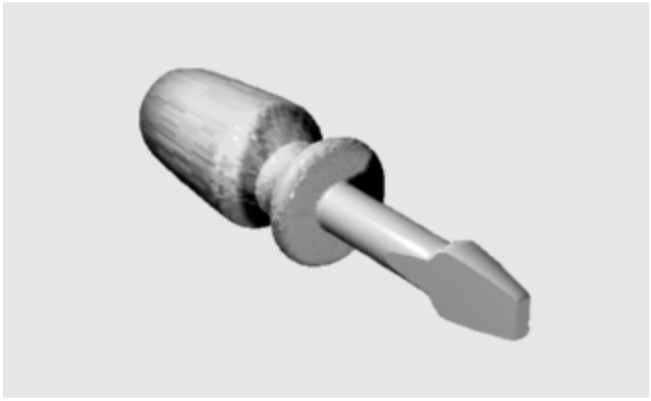
5 Click **OK** to close the **Object Properties** dialog.

Notes:

Object Color
Look for this button.

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

The surface of the handle has a bumpy appearance.



To add a texture to the handle:

For this part we are going to change layers to view a different handle.

1 From the **Edit** menu, click **Layers**, then click **Edit.**

2 Make the **Handle 2** layer current and turn off the **Handle 1** layer, then click **OK**.

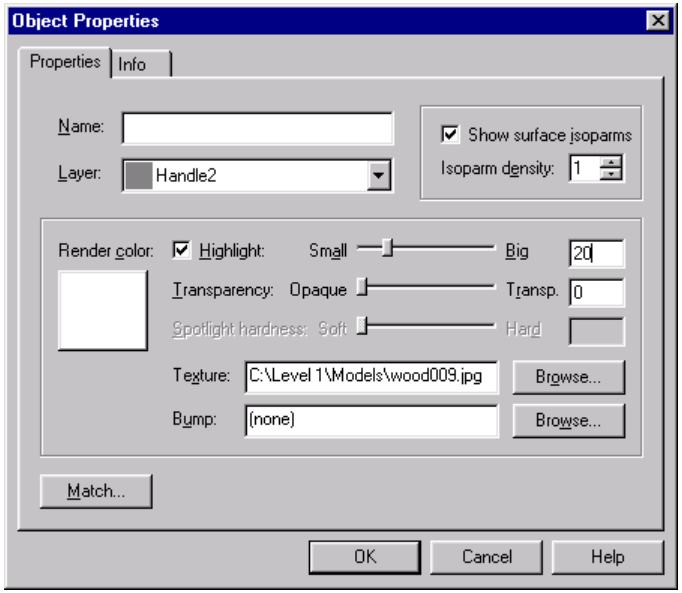
3 Select the new handle.

Notes:

Layer
Look for this button.

4 From the **Edit** menu, click **Object Properties.**

The **Object Properties** dialog appears.



5 In the **Object Properties** dialog, set the **Highlight** to **20**.

6 In the **Object Properties** dialog, click the **Browse** button next to **Texture**.

You can use any bitmap file. The color of the bitmap is rendered as applied to the object.

7 In the **Open bitmap file** dialog, select **Wood009.jpg**, then click **Open**.

8 Click **OK** to close the **Object Properties** dialog.

Notes:

Object Color
Look for this button.

9 From the **Render** menu, click **Render.**

The surface of the handle is textured with a wood grain texture.

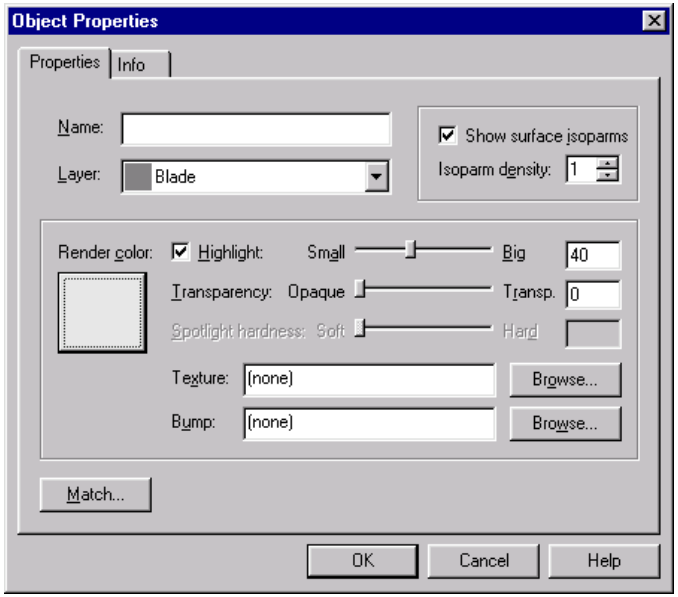
To make the blade transparent:

1 Select the blade.

Notes:

2 From the **Edit** menu, click **Object Properties.**

The **Object Properties** dialog appears.



3 In the **Object Properties** dialog, set the **Transparency** to **60**.

Transparent objects will look more like plastic than glass. Rhino does not have the kind of refraction capability needed to make realistic glass. In addition, shadow color for transparent objects matches the ambient light color and not the color of the transparent object.

4 Click **OK** to close the **Object Properties** dialog.

Notes:

Object Color
Look for this button.

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

The blade will appear transparent.

Notes:

13 Printing

Printing

Printing a wireframe image of your model directly from Rhino is supported. Options for scaling and printing in color are available. Images in the current viewport or all viewports can be printed. Rhino uses your windows printing device for output. While printing to scale is supported, there is no utility for hidden line removal.

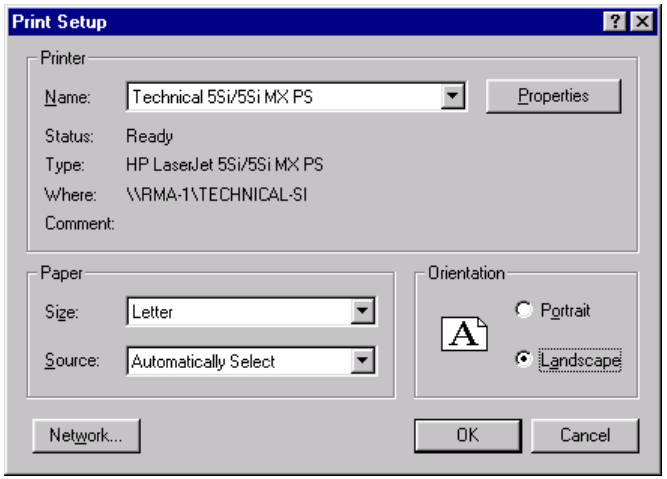
Exercise 67— Practice printing:

To print a wireframe image of the model:

1. Open the **Swing Bracket1** model.
2. From the **Tools** menu, click **Options**.
3. In the **Grid** tab, uncheck **Show Grid**, **Show Grid Axes**, and **World Axes Icon**, then click **OK**.
4. From the **File** menu, click **Print Setup**.

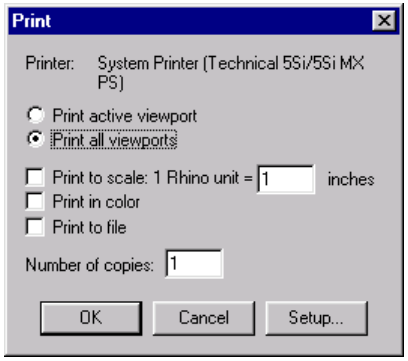
Notes:

5 In the **Print Setup** dialog, click **Landscape**, adjust any other options appropriate for your printer or plotter, then click **OK**.



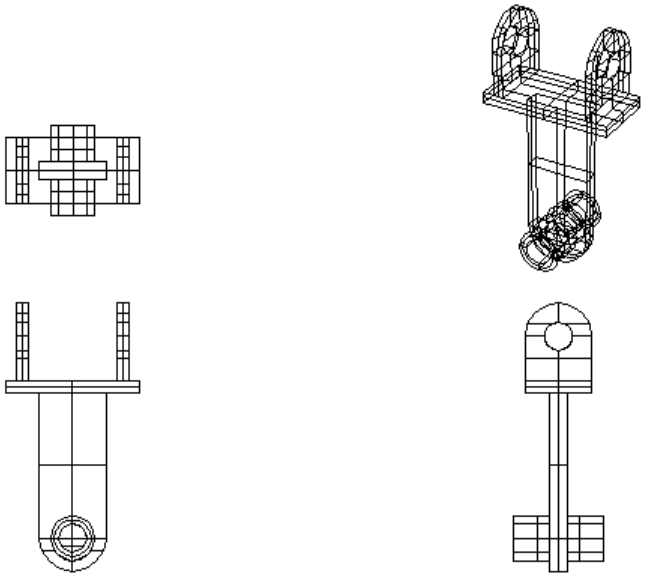
6 From the **File** menu, click **Print**.

7 In the **Print** dialog, check **Print All Viewports**, uncheck **Print to scale** and **Print in color** if necessary, then click **OK**.



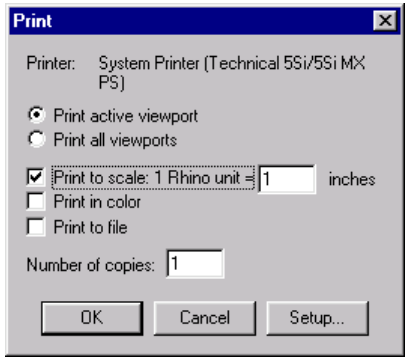
Notes:

The wireframe images in all viewports will be printed.



To print a wireframe image of the current viewport to a scale:

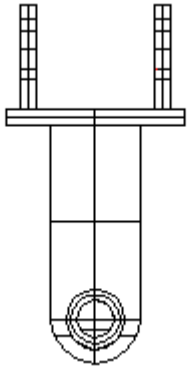
1. Click in the **Front** viewport.
2. From the **File** menu, click **Print**.



Notes:

3 In the **Print** dialog, check **Print Active Viewport** and **Print to scale**, change the **1 Rhino unit** = to **1**, then click **OK**.

The **Front** view image is printed to a scale of 1=1.



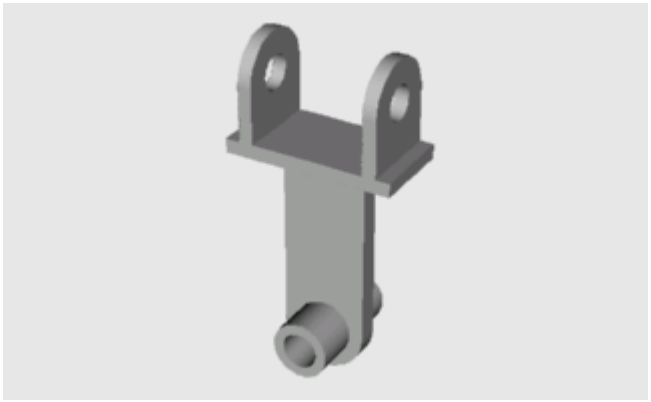
To print a rendered image:

1 Click in the **Perspective** viewport.

2 From the **Render** menu, click **Render**.

3 In the **Display Window**, click **File**, the click **Print**.

The rendered image will be printed to your default Windows printing device, which may be different from what you set in Print Setup.



Notes:

Part Four:
Customizing Workspaces and Toolbars

14 Rhino Options

Rhino Options

The **Options** dialog box shows most of the settings that affect your overall modeling environment. Use it to toggle them or redefine them.

Exercise 68— Practice changing Options:

To access the Rhino Options dialog box:

1. From the **Tools** menu, click **Options**. The **Options** dialog box appears.
2. Click on the **Modeling Aids** tab.

   The **Modeling Aids** tab controls,**Grid Snap, Ortho, Planar** mode, **Object Snap** and **Trim/Extend** Options.

   The Options can be toggled on and off from the dialog or from the status bar. To change the options for ortho or object snap type new values in the boxes. In an earlier exercise we changed the Ortho setting so that it snapped every 30 degrees.

   In the object snap area, **Project to Cplane** projects the snap point to the construction plane even though the object snap point might be in 3D space.

   In the Trim/Extend area, **Extend Lines** extends cutting edges or boundary edges to intersect the object to trim or extend. **Use Apparent Intersections** allows trim, extend and intersection object snap to work if visually intersecting objects are not actually touching.
3. Click on the **Appearance** tab.

Notes:

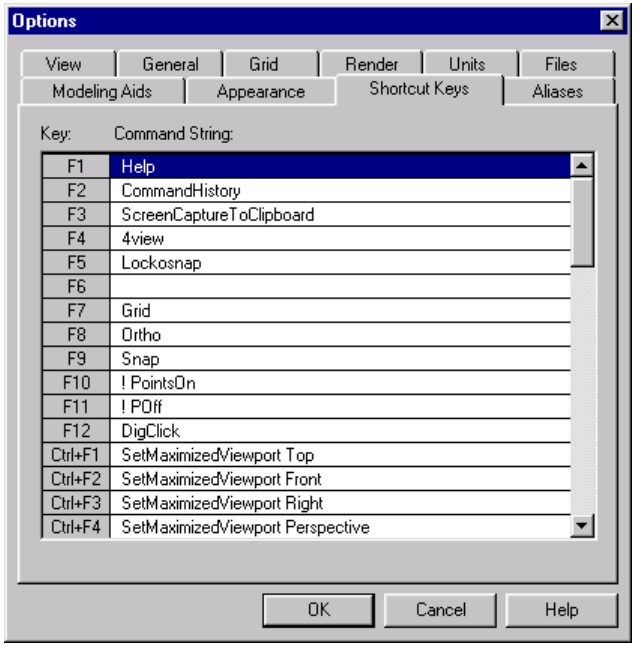
The **Appearance** tab controls the appearance of the Rhino window.

4 Click on the Gray rectangle next to **Viewport background** and change it to a different color.

5 Check the **Crosshairs** box.

6 Click on the **Shortcut Keys** tab.

The **Shortcut Keys** tab sets up shortcut keys for Rhino commands.



7 Type **Lockosnap** in the box next to the F5 key.

This gives you a key that toggles your persistent object snaps between on and locked.

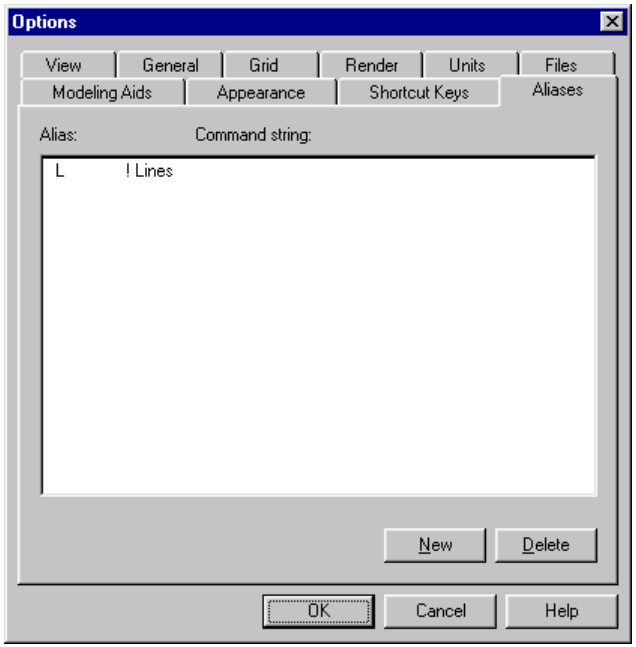
8 Click on the **Aliases** tab.

The command **Aliases** tab allows the user to create custom aliases for Rhino commands.

Notes:

9 Click **New**.

A cursor moves into the white area of the dialog.



10 Type **L**, then press the TAB key.

11 Type **! Lines**.

The apostrophe acts as a cancel.

By typing L and pressing ENTER at the command line, you can begin the Lines command.

12 Click on the **View** tab.

The **View** tab has pan, zoom, rotate, and redraw controls.

13 Click on the **General** tab.

The **General** tab configures mouse selection methods, the number of undos retained in memory, startup commands, recent command list limit, AutoSave control, and don't repeat commands.

Notes:

14 Click on the **Grid** tab.

The **Grid** tab configures grid, grid axes, and axis icons. We changed the snap spacing in an earlier exercise.

15 Click on the **Render** tab.

The **Render** tab controls render options.

16 Click on the **Units** tab.

The **Units** tab controls the unit system and the tolerance settings.

17 Click on the **Files** tab.

The **Files** tab sets the location of template files and the autosave files.

18 Click **OK** to close the **Options** dialog.

19 Try the **Lines** alias and the **Lockosnap** toggle. Note how your cursor has changed.

Notes:

15 Creating Custom Toolbar Layouts

The Toolbar Layout

The toolbar layout is the arrangement of toolbars containing command buttons on the screen. The toolbar layout is stored in a workspace file, that you can open and save. Rhino comes with a default toolbar layout, and automatically saves the active toolbar layout before closing. You can create your own custom layouts and save them for later use.

You must use an external file manager to delete a toolbar layout file.

Exercise 69— Customizing a Toolbar Layout:

To create a custom toolbar layout:

1. Begin a new model.
2. From the **Tools** menu, click **Toolbar Layout**, then **Save As**.
3. In the **Save Workspace As** dialog, type **Level 1** in the **File name** box, and click **OK**.

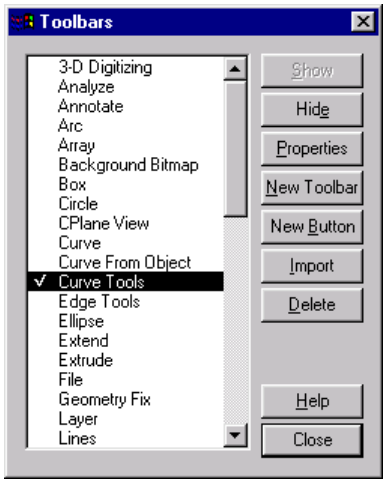
   A new toolbar layout file has been created. Toolbar layouts are saved with a **.ws** extension. We will use this new toolbar layout to do some customization.

Notes:

To show another toolbar:

1 From the **Tools** menu, click **Toolbar Layout**, then click **Edit.**

The toolbars dialog appears.



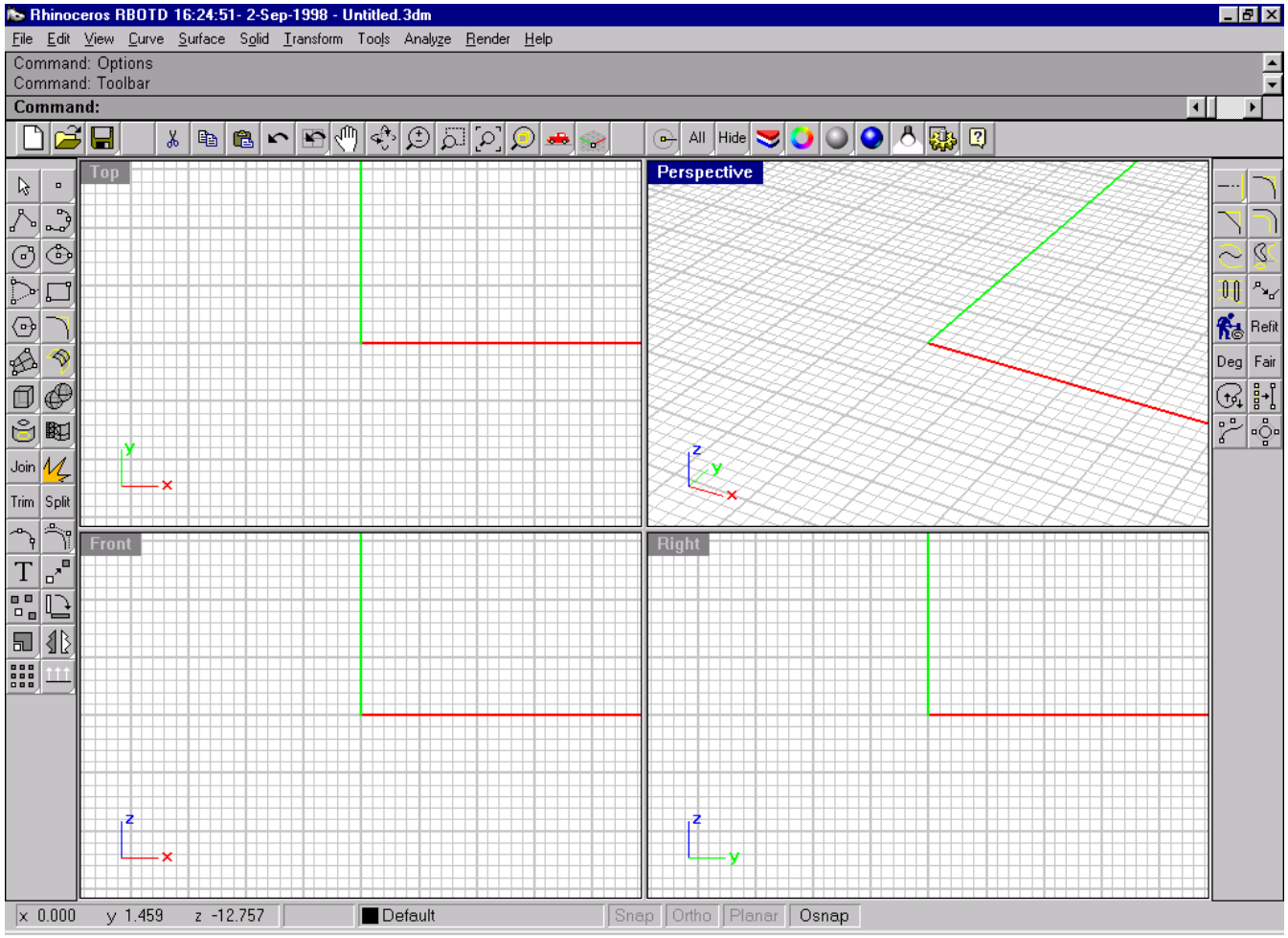
2 In the **Toolbars** dialog, double click **Curve Tools** to **show** the toolbar.

3 In the **Toolbars** dialog, double click **Curve Tools** to **hide** the toolbar

4 Show the **Curve Tools** toolbar again, then click **Close**.

Notes:

5 Dock the **Curve Tools** toolbar at the right by dragging it to the right edge until it changes to a vertical shape, then releasing your mouse button.



Notes:

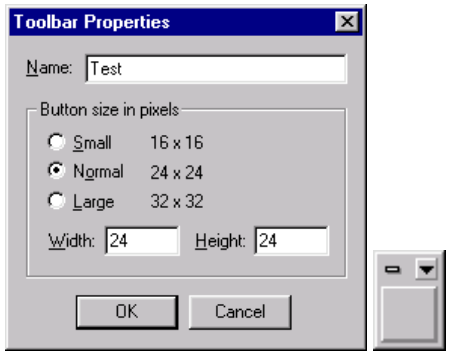
To save your toolbar layout:

- From the **Tools** menu, click **Toolbar Layout**, then click **Save.**

To create a new toolbar:

1. From the **Tools** menu, click **Toolbar Layout**, then click **Edit.**
2. In the **Toolbars** dialog, click **New Toolbar**.
3. In the **Toolbar Properties**, type **Test** in the **Name** box, then click **OK**.

   A new toolbar is created with one blank button in it.

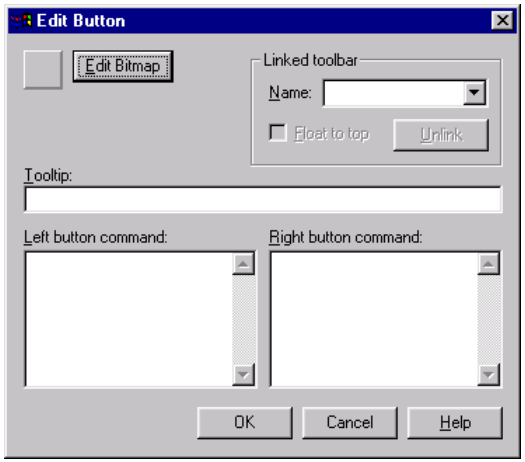


4. In the **Toolbars** dialog, click **Close**.

Notes:

To edit the new button:

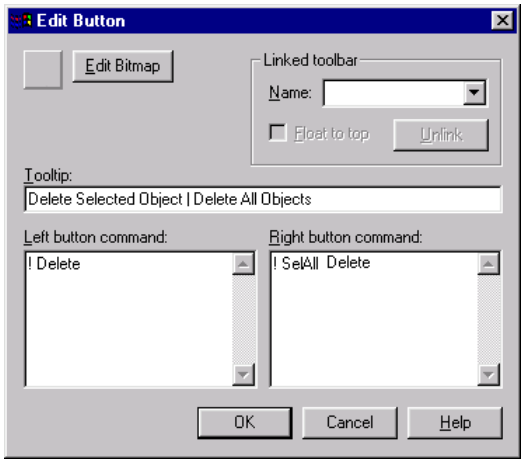
1 Hold down the SHIFT key and right click on the blank button.



2 In the **Edit Button** dialog, type **Delete Selected Objects | Delete All Objects** in the **Tooltip** box.

3 In the **Edit Button** dialog, type **! Delete** in the **Left Button Command** box.

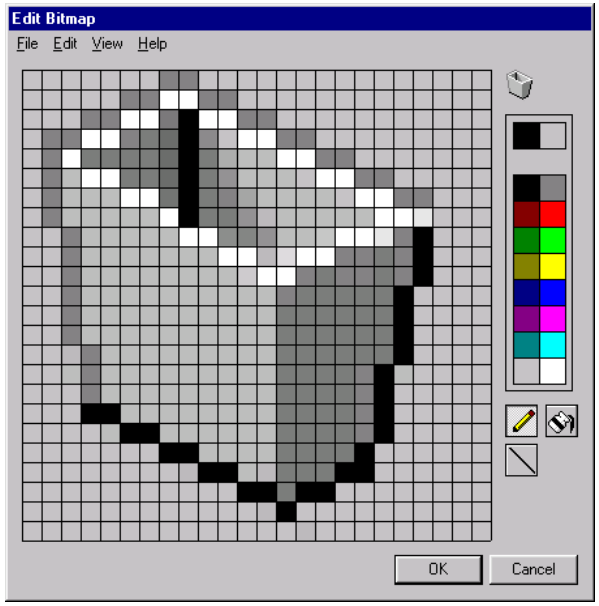
4 In the **Edit Button** dialog, type **! SelAll Delete** in the **Right Button Command** box.



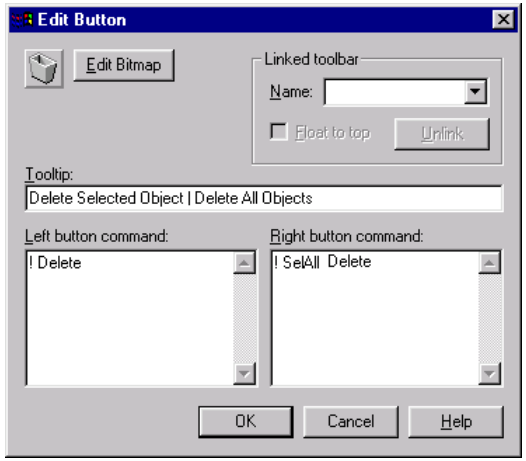
5 Click **Edit Bitmap**.

Notes:

6 In the **Edit Bitmap** dialog, make a picture to represent the button face image, then click **OK**.



7 In the **Edit Button** dialog, click **OK**.



Notes:

To copy a button from one toolbar to another:

1. From the **Main** toolbar, flyout the **Curve** toolbar, and tear it off.
2. While holding down the CTRL key, drag the **Curve From Control Points** button to the new toolbar and drop it.

To add a button to a toolbar:

1. From the **Tools** menu, click **Toolbar Layout**, then click **Edit.**
2. Highlight the **Test** toolbar in the list.
3. Click **New Button**.

   A blank button is added to the **Test** toolbar.
4. Click **Close** to close the dialog.

To open the standard toolbar layout:

1. From the **Tools** menu, click **Toolbar Layout**, then click **Open.**
2. In the **Save Changes** dialog, click **Yes**.
3. Select **Default.ws**, then click **Open**.

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