

# Rhino<sup>®</sup>ceros

NURBS modeling for Windows

**Training Manual**  
**Level 1**

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Rhinoceros Level 1 Training Manual v3.0

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# **Part One: Introduction**

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# 1

## Before You Start

This course guide accompanies the Level 1 training sessions. Level 1 shows you how to produce 3-D models using NURBS geometry.

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 and the Help file for additional information.

### Duration:

3 days

### Course Objectives

In Level 1, you learn how to:

- Utilize the features of the Rhino user interface
- Customize your modeling environment
- Create basic graphic objects—lines, circles, arcs, curves, solids, and surfaces
- Model with precision using coordinate input and object snaps
- Modify curves and surfaces with edit commands
- Use control point editing to modify curves and surfaces
- Analyze your model
- Display any portion of the model
- Export and import models to and from different file formats
- Render the model

### Notes:

*The Rhino for Windows Interface  
Rhino uses NURBS for all curve  
and surface geometry.*





# 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.

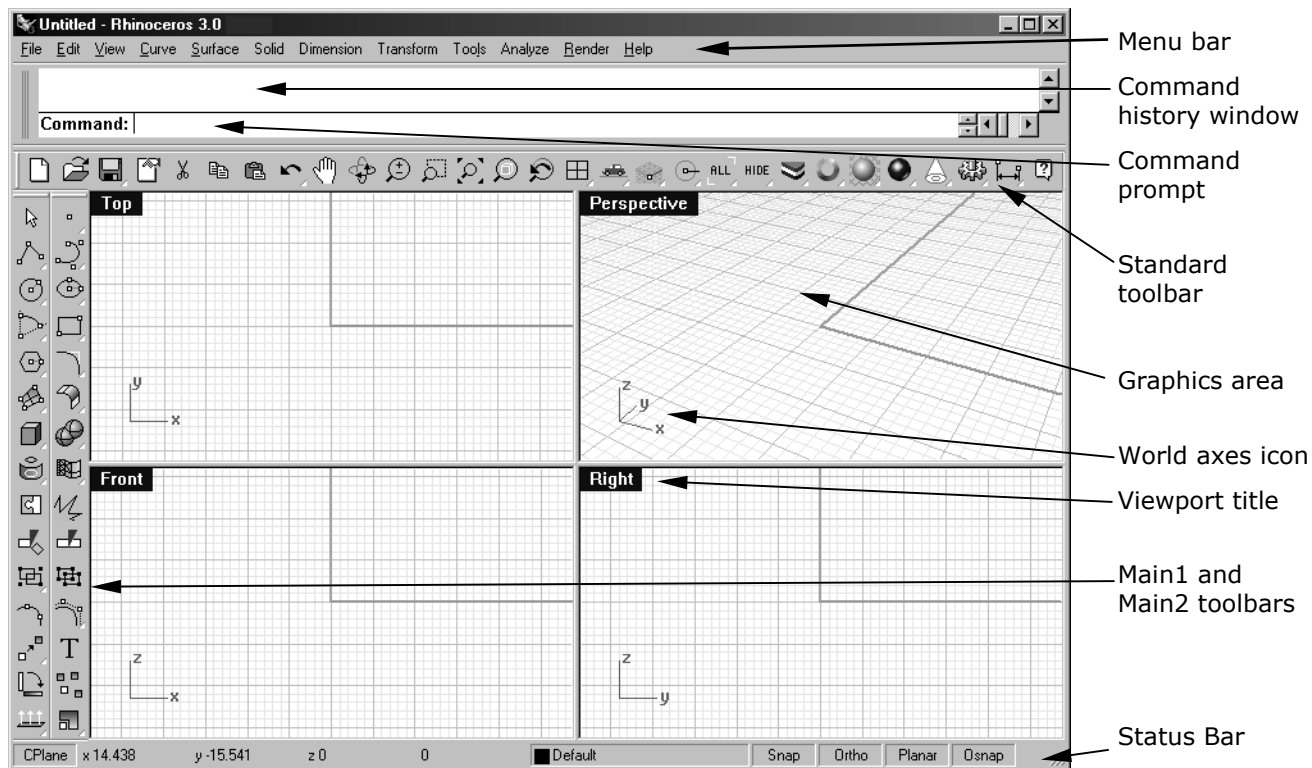


## Notes:

## 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.
Command area	Lists prompts, commands you enter, and information displayed by the command.
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 ).
Viewports	Displays different views of the model within the graphics area.
Status bar	Displays the coordinates of the pointer, the status of the model, options, and toggles.

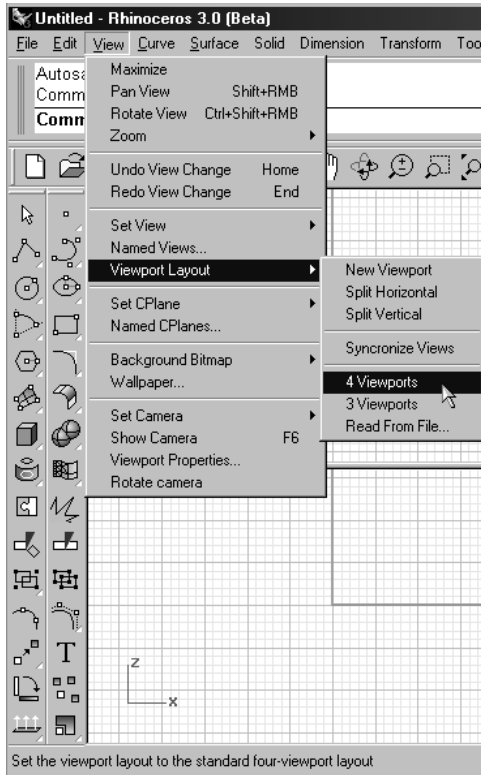


Watch the command line to find out what is happening.

Rhino screen

## Menus

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



*Rhino View menu*

## Toolbars

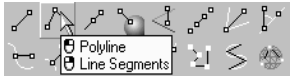
Rhino toolbars contain buttons that provide shortcuts to commands. You can float a toolbar anywhere on the screen, or dock it at the edge of the graphics area

Rhino starts up with the Standard toolbar docked above the graphics area and the Main1 and Main2 toolbars docked on the left.

## Notes:

### Tooltips

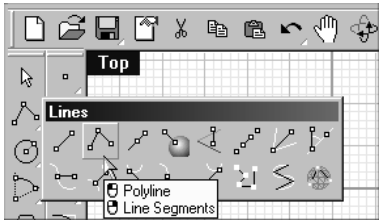
Tooltips tell what each button does. Move your pointer over a button without clicking it. A small yellow tag with the name of the command appears. In Rhino, many buttons can execute two commands. The tooltip indicates which buttons have dual functions.



### Flyouts

A button on a toolbar may include other command buttons in a flyout toolbar. Usually the flyout toolbar 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. To open the flyout toolbar, hold down the left button for a moment or press both buttons at the same time.



The Lines toolbar is linked to the Main1 toolbar.

If you hold the left mouse down over the Line button of the Main1 toolbar, the flyout Lines toolbar opens.

### Graphics Area

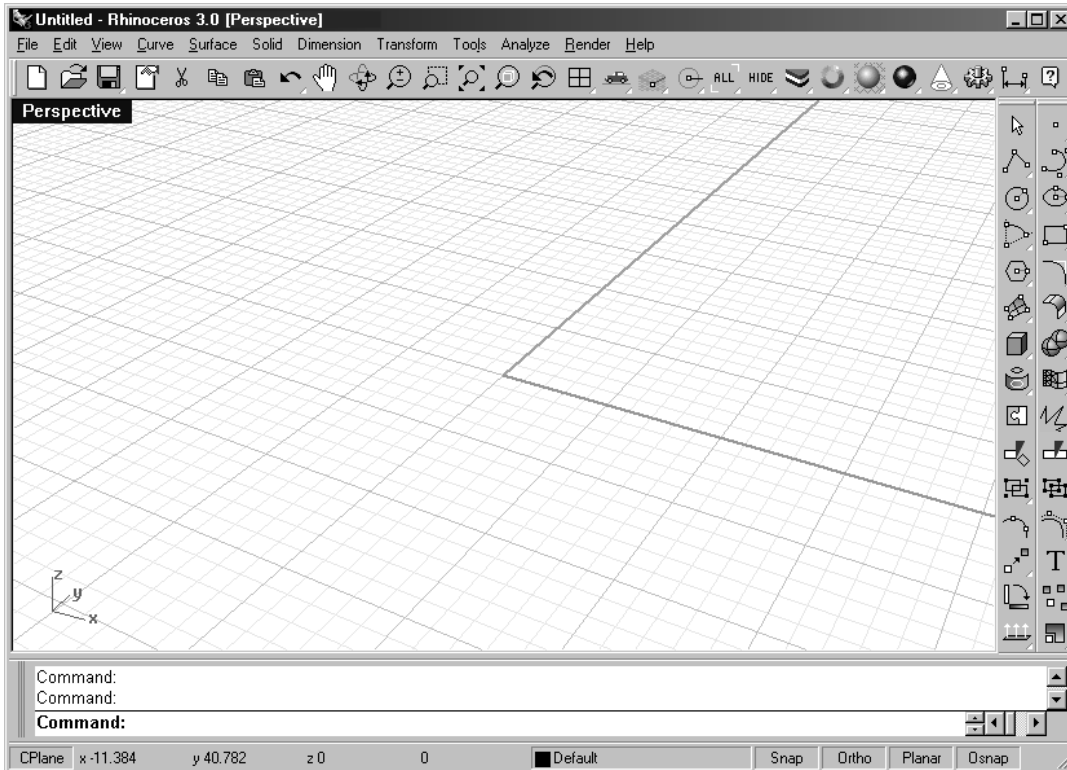
The Rhino graphics area holding the viewports can be customized to suit your preferences. The position of viewports can be arranged in different configurations.

## Notes:

## Viewports

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

To toggle between a small viewport and one that fills the graphics area, double-click the viewport title.



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

## Command Area

The command area displays commands and command prompts. It can be docked at the top or the bottom of the screen or it can float anywhere. The command window shows three lines by default. To open a window that displays the command history, press F2. The text in the Command History window can be selected and copied to the Windows clipboard.

## The Mouse

In a Rhino viewport, the left mouse button selects objects and picks locations. The right mouse button has several functions including panning and zooming, popping up a context-sensitive menu, and acting the same as pressing the **Enter** key. 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 commands from some toolbar buttons.

Drag with the right mouse button to pan and rotate in viewports. Use the mouse wheel or 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, pick 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 over a viewport.

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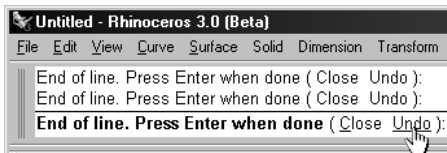
**Note:** **Enter** and **Spacebar** perform the same function.

---

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

## Clickable options

To use command options, click the option on the command line or type the underlined letter of the option and press Enter. (The interior capitalization is meaningless.)



## Notes:

### Autocomplete command name

Type the first few letters of a command name to activate the autocomplete command list. When enough letters of the command are typed so that it is unique, the command name completes on the command line. Press Enter to activate the command once the full command name appears. As you type command names, the autocomplete command list appears. As you type more letters, the list is narrowed down to the possible commands. Left click on the command in the list to start it.



### Repeating commands

To repeat the last command, right-click in a viewport, or press **Enter** or **spacebar**. To repeat previous commands, right-click in the command line window and select from a list.

### Canceling commands

To cancel a command, press **Esc** or enter a new command from a button or a menu.

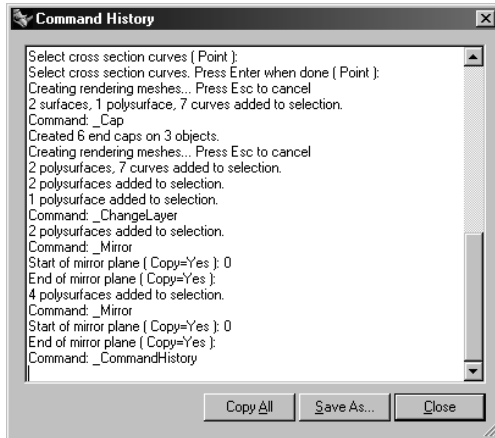
### Help

Press **F1** at any time to access Rhino Help. 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 are 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 **F1**.

## Notes:

### View the Command Line History

The command line history window lists the last 500 command lines from the current Rhino session. Press **F2** to view the command history.



### View Recent Commands

Right-click the command line to view recently used commands. To repeat the command, select it from the popup menu.

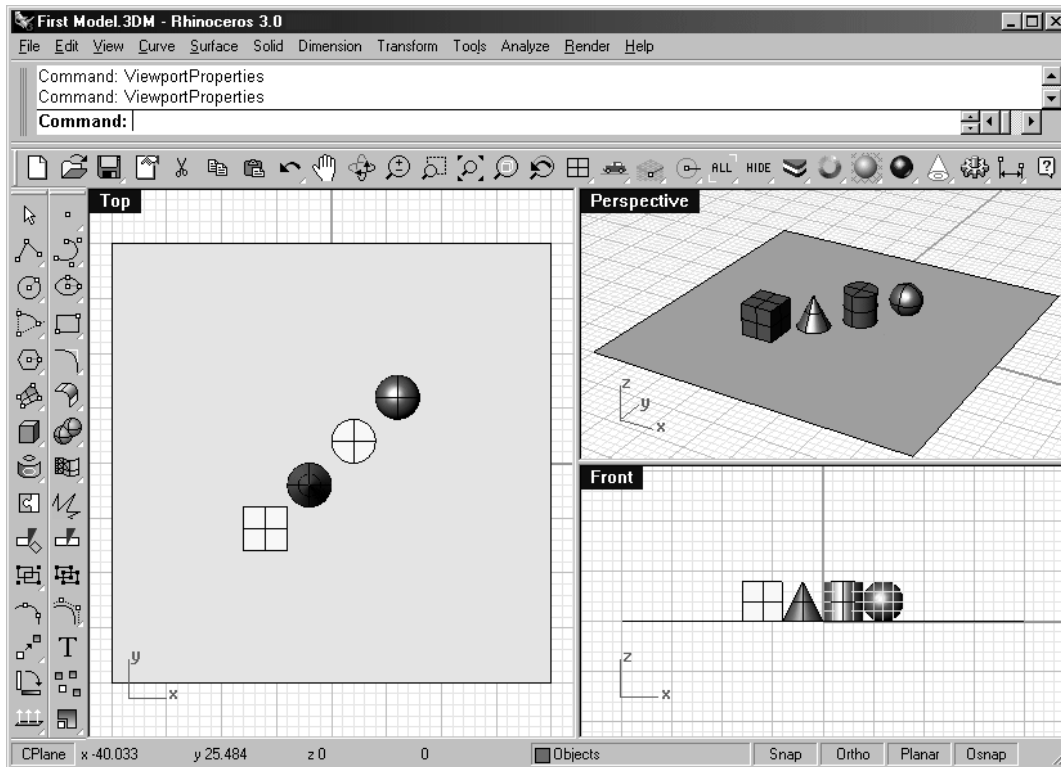




### Exercise 1—Rhino basics

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

You will find this model in the Training folder. If you haven't copied the files to your hard drive from the Training folder on the Rhino CD, you should do this before you proceed.



*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 will not 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.

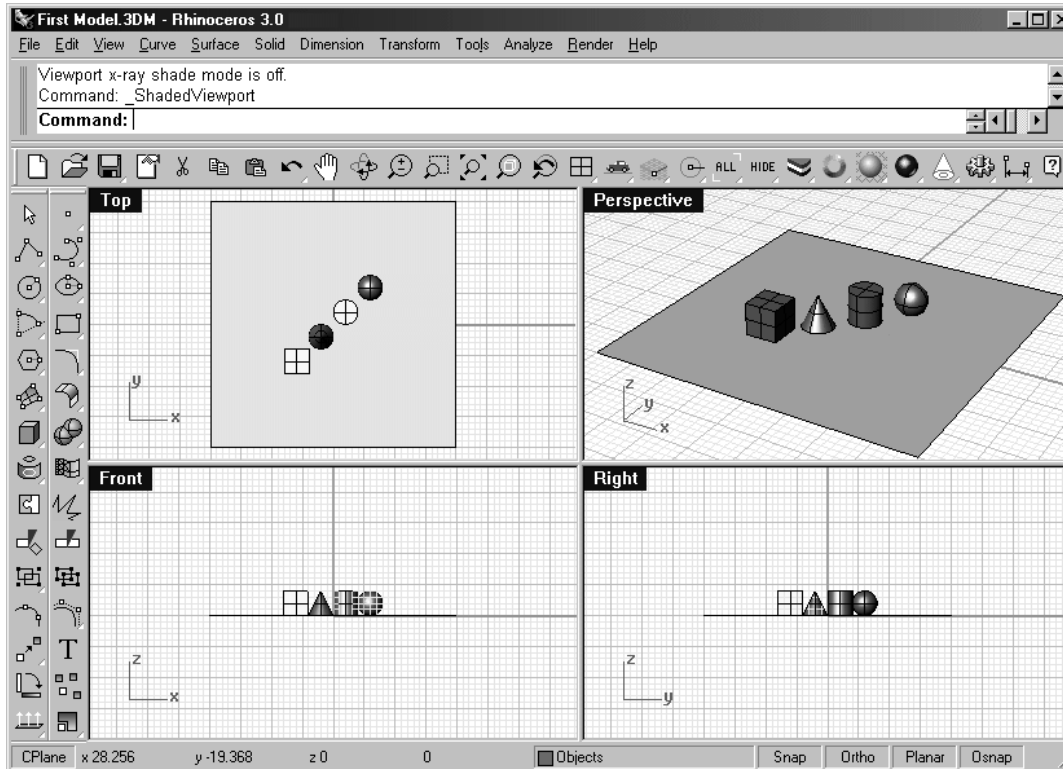
### Notes:



Open

## Notes:

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



*Three parallel viewports and one perspective viewport*

- 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 do not 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.

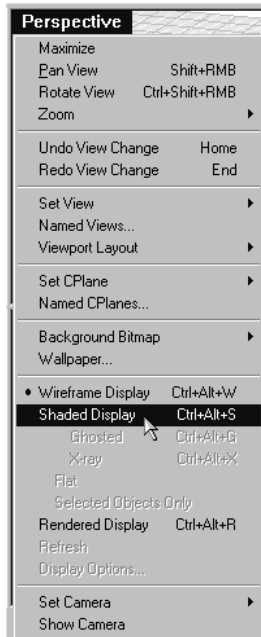
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- Click the mouse in the **Perspective** viewport to make it active.

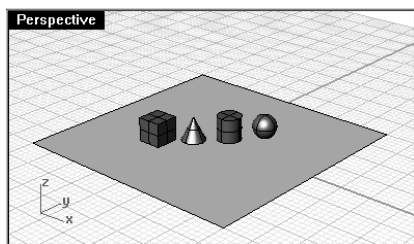
The viewport title highlights when it is active. The active viewport is the viewport where all your commands and actions take place.

## Notes:

- Click with the **Right Mouse Button** (RMB) on the **Perspective** viewport title, then click **Shaded Display**.



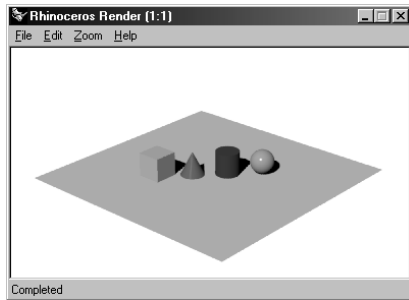
The objects appear shaded. Shaded display lets you preview the shapes. The viewport will remain shaded until you change it back to a wireframe view. You can change any viewport to shaded mode. Later we will discuss the other shaded display options.



*Shaded display.*

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

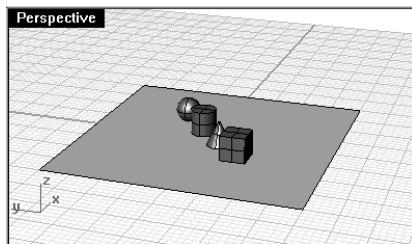
Rendering the model opens a separate render window. The model displays in render colors previously assigned to the objects. You can also set lights and a background color. You will learn about doing this later. You cannot manipulate the view in the render display window but the image can be saved to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



*Render.*

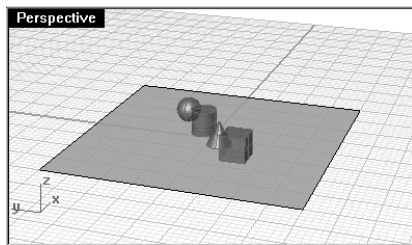
- 8 Close the render window.
- 9 In the **Perspective** viewport, click and drag with your right mouse button held down to rotate the view.

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



*Rotate the view in shaded display.*

- 10 Right click on the **Perspective** viewport title, then click **Ghosted Display**.



*Ghosted shade display.*

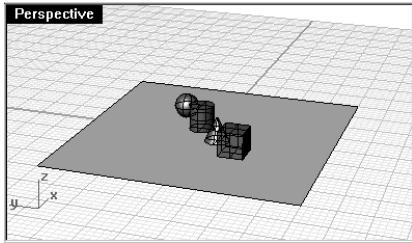
## Notes:



Render

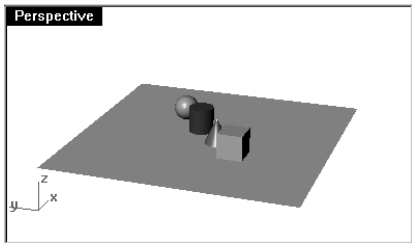
**Notes:**

- 11 Right click on the **Perspective** viewport title, then click **X-ray Display**.



*X-Ray shaded display.*

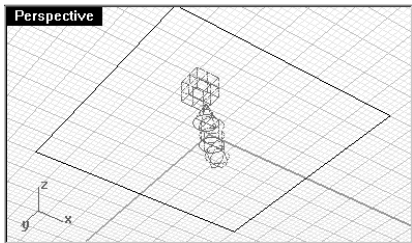
- 12 Right click on the **Perspective** viewport title, then click **Rendered Display**.



*Rendered display.*

- 13 Change to a **Wireframe Display** mode.

- 14 To rotate your view, drag from the bottom of the view toward the top.

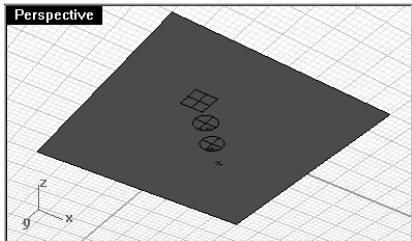


*Looking at the objects from the bottom in wireframe mode.*

You are now under the objects looking up.

15 Change to a **Shaded Display** mode.

The plane obscures the objects.



*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.

#### If you are “lost in space” in the perspective view:

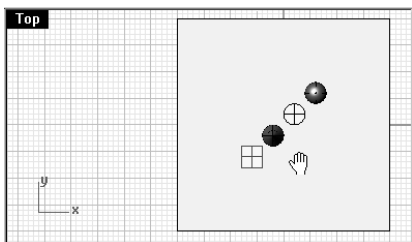
- ▶ From the **View** menu, click **Viewport Layout**, and then click **4 Viewports**.  
This takes you back to the default viewport settings.

### 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.

To pan in a viewport:

- 1 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.*

- 2 **Pan** the view in the other viewports.

### Notes:

*What if ...*

*Instead of panning or rotating, something funny happened.*

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

## Zooming in and out

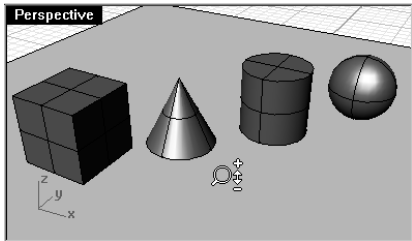
Sometimes you want to get closer to your objects or move back so you can see more. This is called **zooming**. As with many things in Rhino, there are several ways to do this. The easiest way is to hold down the **Ctrl** key and drag up and down in a viewport with the right mouse button. If you have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

### To zoom in and out:

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

Drag up to zoom in.

Drag down to zoom out.



*Zooming with ctrl and the right mouse button.*

## 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 is 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:



Zoom Extents

Left click this button.



Zoom Extents All Viewports  
Right click this button

## Move Objects

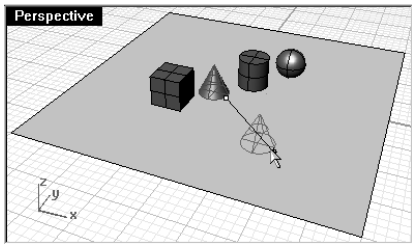
Dragging follows the construction plane of the current viewport.

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 highlights to show it is selected.

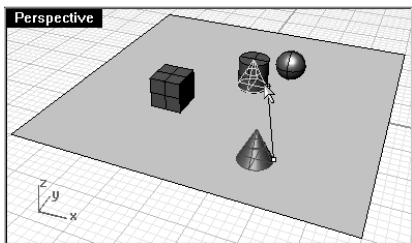


*The selected cone highlights.*

- 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.*

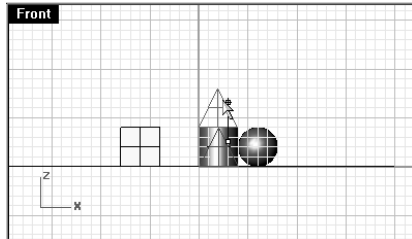


## Notes:

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

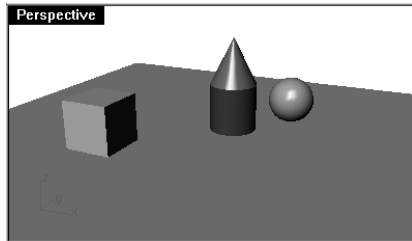
Watch what happens in the **Perspective** viewport.

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 Change the viewport to a **Rendered Display**.



*Rendered Display.*

### Try on Your Own

- 1 Re-open the model. Do not save changes.
- 2 Drag the objects around.

Use the **Front** viewport to move the objects vertically and the **Top** or **Perspective** viewport to move them horizontally.

## Copy Objects

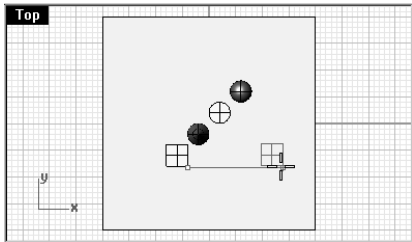
To create more objects, copy the shapes.

### To start with a new model:

- 1 From the **File** menu, click **Open**.
- 2 Do not save the changes.
- 3 In the **Open** dialog box, select **First Model.3dm**.

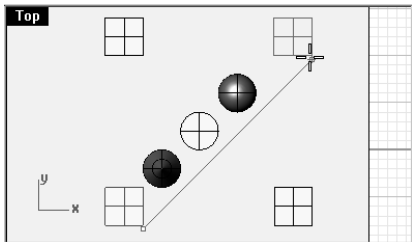
### To copy objects:

- 1 Click the box to select it.
- 2 From the **Transform** menu, click **Copy**.
- 3 At the **Point to copy from** ( Vertical=No 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.
- 5 At the next **Point to copy to** prompts, click other places to make more copies of the box.



*Make three copies*

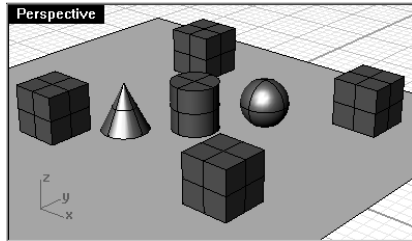
## Notes:



Copy

## Notes:

- 6 When you have enough copies, press **Enter**.



*Shaded display.*

### Try on Your Own

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

## Changing the View of Your Model

When you add detail to your models, you will 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 view:

- 1 Right-click the viewport title, click **Viewport Properties**.
- 2 In the **Viewport Properties** dialog box, click **Parallel** or **Perspective**, and then click **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 or use the mouse wheel.

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	
Home	Zoom previous	
End	Zoom next	

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

## Resetting Your View

If you get lost, four view techniques 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:

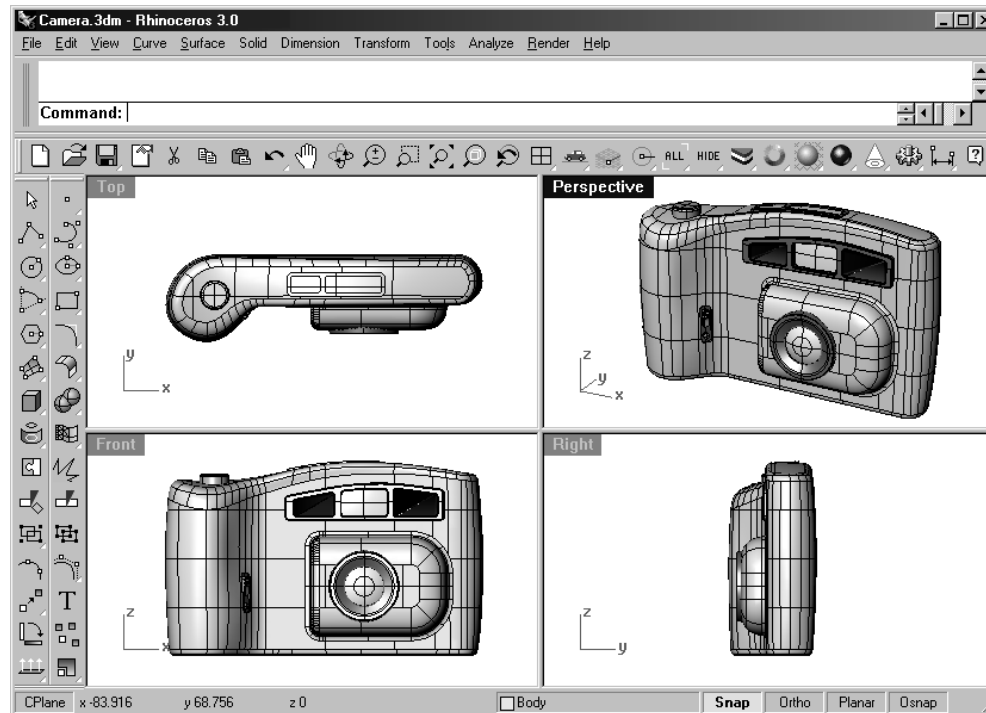
- ▶ From the **View** menu, click **Zoom**, and then click **Extents All**.

## Notes:

### Exercise 2—Display options

- ▶ Open the model **Camera.3dm**.

You will use this to practice changing views. You will create views from six directions including the oblique perspective view.

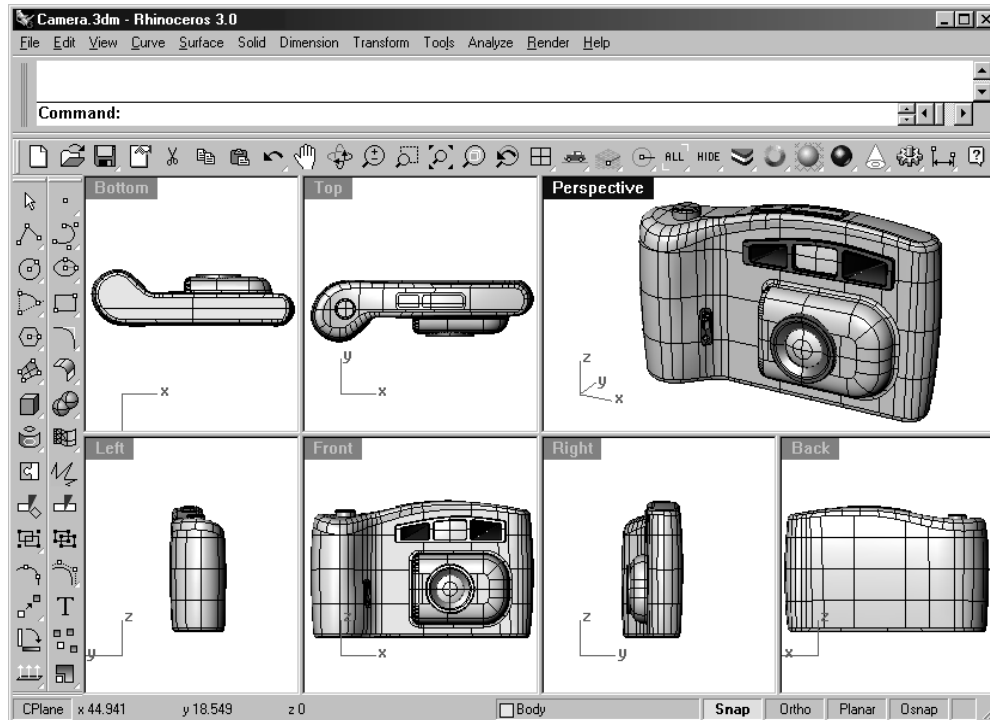


### To change the number of viewports:

- 1 Make the **Top** viewport active.
- 2 From the **View** menu, click **Viewport** Layout, and then click **Split Vertical**.
- 3 Repeat this step for the other parallel views.
- 4 Right click on the **Top** viewport title on the left, click **Set View**, then click **Bottom**.
- 5 Right click on the **Front** viewport title on the left, click **Set View**, then click **Left**.
- 6 Right click on the **Right** viewport title on the right, click **Set View**, then click **Back**.
- 7 From the **View** menu, click **Zoom**, and then click **Extents All**.

### To synchronize the viewports:

- 1 Right click on the **Front** viewport title, click **Zoom**, and then click **Synchronize Views**.



All the views are sized to the same scale as the active viewport and aligned with each other.

### Notes:



Split Vertical



Bottom View



Left View



Back View






Zoom Extents All



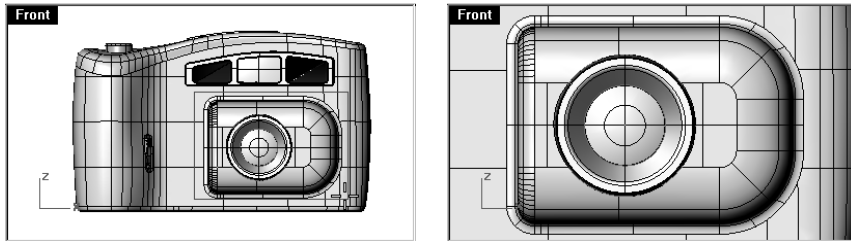
Synchronize Views

### To change the shape of viewports:

- 1 Move your cursor to the edge of a viewport until you see the resizing  or  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.

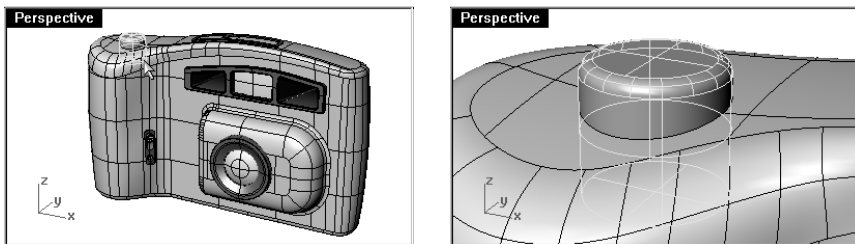
### To zoom to a window:

- 1 From the **View** menu, click **Zoom**, and then click **Window**.
- 2 At the **Drag a window to zoom** ( All Dynamic Extents Factor In Out Selected Target ) prompt, drag a window around a portion of the model.



### To zoom a selected object:

- 1 Select the shutter release.
  - 2 From the **View** menu, click **Zoom**, and then click **Selected**.
- The view zooms to the selected object.



### To rotate the view:

- ▶ In a perspective viewport, drag with right mouse button.
- ▶ In a parallel viewport, use the arrow keys.

### Notes:



Zoom Window



Zoom Selected

---

**Notes:****To maximize and restore a viewport:**

- 1 Double-click the viewport title to maximize it.
- 2 Double-click the title of the maximized viewport to restore it to its smaller size and reveal the other viewports.



# **Part Two: Creating Geometry**

---



# 3

## Creating Two-Dimensional Objects

### Drawing Lines

The **Line**, **Lines**, and **Polyline** commands draw straight lines. The **Line** command draws a single line segment. The **Lines** command draws multiple end-to-end line segments. The **Polyline** command draws a series of straight segments joined together (a single linear curve with multiple segments).

#### Exercise 3—Drawing lines

- 1 From the **File** menu, click **New**.  
Do not save changes.
- 2 In the **Template File** dialog box, click **Millimeters.3dm**.
- 3 From the **File** menu, click **Save As**.
- 4 In the **Save As** dialog box, type **Lines**, and then click **Save**.

#### To draw line segments:

- 1 From the **Curve** menu, click **Lines**, and then click **Line Segments** to begin the **Lines** command.
- 2 At the **Start of line** prompt, pick a point in a viewport.
- 3 At the **End of line** ( Undo ) prompt pick another point in a viewport.  
A line segment appears between the two points.
- 4 At the **End of line. Press Enter when done** ( Undo ) prompt, pick another point.  
You can press the right button instead of pressing the **Enter** key on your keyboard to terminate the command.

#### Notes:



Line Segments

Right-click for Line Segments

- 5 At the **End of line. Press Enter when done** ( Close Undo ) prompt, continue to pick points.  
Additional segments appear.  
Each segment meets but is not joined to 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. Press Enter when done** ( Close Undo ) prompt, press **Enter** to end the command.

#### To use the Close option:

- 1 From the **Curve** menu, click **Lines**, and then click **Line Segments** to begin the **Lines** command.
- 2 At the **Start of line** prompt, pick a point.
- 3 At the **End of line** ( Undo ) prompt pick another point.
- 4 At the **End of line. ...** ( Undo ) prompt, pick another point.
- 5 At the **End of line. ...** ( Close Undo ) prompt, click **Close**.  
The last line will end at the original start point.

#### To draw a polyline:

- 1 From the **Curve** menu, click **Lines**, and 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. ...** ( Undo ) prompt, pick another point.
- 5 At the **Next point of polyline. ...** ( Close Undo ) prompt, continue to pick points.

#### To use the Undo option:

- 1 From the **Curve** menu, click **Lines**, and 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. ...** ( Undo ) prompt, pick another point.
- 5 At the **Next point of polyline. ...** ( Close Undo ) prompt, click **Undo**.  
One segment of the polyline is removed.

#### Notes:



Polyline  
Left click for Polyline.

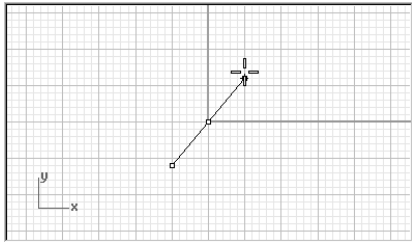
- 6 At the **Next point of polyline. ...**( Undo ) prompt, pick another point.
- 7 At the **Next point of polyline. ...** ( Close Undo ) prompt, press **Enter** to end the command.

#### To draw a single line segment:

- 1 From the **Curve** menu, click **Lines**, and then click **Single Line** to begin the **Line** command.
- 2 At the **Start of line** ( Normal Angled Vertical FourPoint Bisector Perpendicular Tangent Extend BothSides ) prompt, pick a point.
- 3 At the **End of line** ( BothSides ) prompt, pick another point.  
The command ends after one segment is drawn.

#### To use the BothSides option:

- 1 From the **Curve** menu, click **Lines**, and then click **Single Line** to begin the **Line** command.
- 2 At the **Start of line** ( Normal Angled Vertical FourPoint Bisector Perpendicular Tangent Extend BothSides ) prompt, click **BothSides**.
- 3 At the **Middle of line** (Normal Angled Vertical FourPoint Bisector Perpendicular Tangent Extend ) 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.



#### Notes:



Single Line

## Drawing Free-form Curves

The **InterpCrv** and **Curve** commands draw free-form curves. The **InterpCrv** command draws a curve through the points you pick. The **Curve** command uses control points to create a curve.

### Exercise 4—Drawing interpolated curves

- 1 From the **Curve** menu, click **Free-form**, and then click **Interpolate Points**.
- 2 At the **Start of curve** ( Degree=3 Knots=*SqrtChord* StartTangent ) prompt, pick a point.
- 3 At the **Next point of curve** ( Degree=3 Knots=*SqrtChord* EndTangent Undo ) prompt, pick another point.
- 4 At the next several **Next point of curve** ( Degree=3 Knots=*SqrtChord* EndTangent Close Sharp=*No* Undo ) prompts, pick more points.

Or, click an option, and press **Enter**.

Option	Description
<u>C</u> lose	Closes the shape by drawing a segment from the last point picked to the first point picked. This ends the command.
<u>E</u> ndTangent	After choosing a point on another curve, the next segment will be tangent to the point you picked and end the command.
<u>U</u> ndo	Deletes the last point picked.
<u>D</u> egree	You can set the degree of the curve.
<u>K</u> nots	Determines how the interpolated curve is parameterized.
	When you draw an interpolated curve, the points you pick are converted into knot values on the curve. The parameterization means how the intervals between knots are chosen:
<u>S</u> harp	When you make a closed curve, it will come to a point instead of making a smooth closure as it normally does.

### Exercise 5—Drawing curves from control points

- 1 From the **Curve** menu, click **Free-form**, and then click **Control Points**.
- 2 At the **Start of curve** ( Degree=3 ) prompt, pick a point.
- 3 At the **Next point** ( Degree=3 Undo ) prompt, pick another point.
- 4 At the **Next point** ( Degree=3 Close Sharp=*No* Undo ) prompts, pick more points.

Or, click an option, and press **Enter**.

## Notes:



Interpolated Points



Control Point Curve

*Notice that most of the points you pick are off the curve as control points.*

## Modeling Aids

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

Mode	Function Key	Description
Ortho	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, hold down the Shift key to toggle Ortho off. If Ortho is off, hold down the Shift key to toggle Ortho on.
Grid	F7	Hides or shows a reference grid in the current viewport of the graphics screen at the construction plane.
Snap	F9 or S	Forces the marker to snap on grid intersections.
Planar	P	This is a modeling aid similar to Ortho. This helps you model planar objects by forcing input to be on a plane parallel to the construction plane that passes through the last point that you picked.

Click the **Snap**, **Ortho**, and **Planar** panes on the status bar to toggle these modeling aids on and off.



### **Exercise 6—Drawing lines and curves using mode functions**

- 1 Toggle **Snap** on and draw some lines.  
The marker 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 at 90 degree intervals from the last point. Using Snap and Ortho toggles you can draw with precision. We will discuss other ways to get precision in a later session.

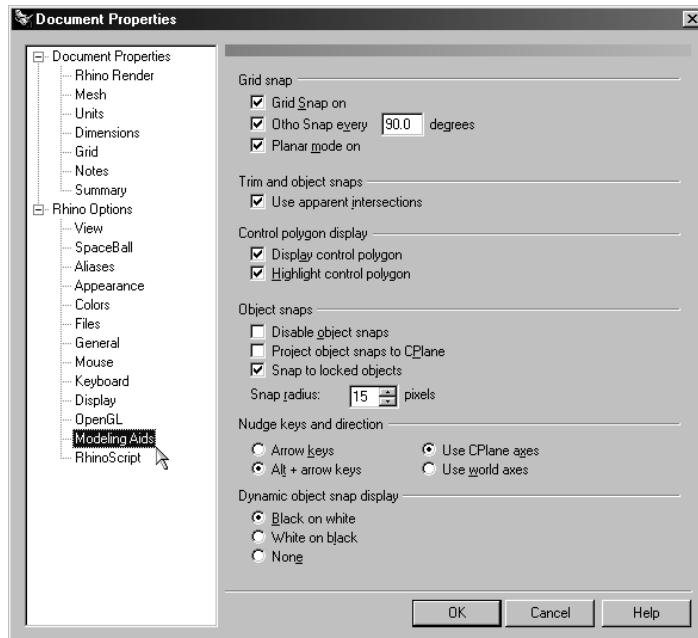
## Model Setup

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.

### To change the options:

- 1 From the **File** menu, click **Properties**.
- 2 In the **Document Properties** dialog box, under **Rhino Options**, click **Modeling Aids**.

**Modeling Aids** lets you control **Ortho**, **Object Snap**, **Grid Snap**, and other mode options.



- 3 Change the **Ortho** option to snap every **30** degrees.

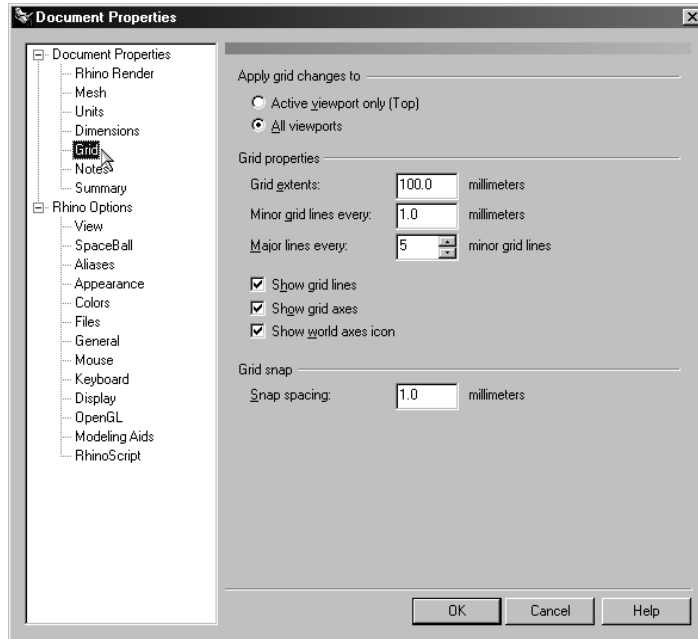
## Notes:



Document Properties



- 4 In the **Document Properties** dialog box, click **Grid**.



- 5 In **Grid** properties, change the following settings.

You can change the appearance of the modeling environment by changing the grid elements. The grid spacing, the frequency of the major lines, and the number of grid elements can be changed. The Grid dialog box lets you configure grid settings.

- 6 Change the **Grid Extents** setting to **10**.
- 7 Change the **Minor grid lines every** setting to **1**.
- 8 Change the **Major lines every** setting to **4**.
- 9 Change the **Snap Spacing** setting to **.25**, and click **OK**.
- 10 Draw some more lines and curves with **Snap** and **Ortho** on.

Notice that the marker now snaps between the grid intersections and that **Ortho** snaps at every 30 degrees.

## Notes:

*The value for Grid extents is for each quadrant.*



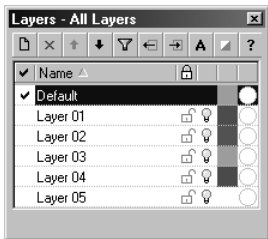
## Notes:

## 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 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).

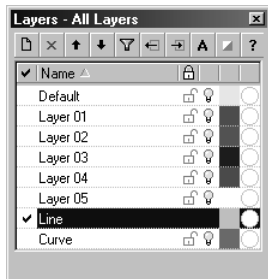
The **Layers** window manages layers. Use it to set up layers for your model.



## Exercise 7—Layers

### To create a new layer:

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



The **Default** layer is created automatically when you start a new model with no template. If you use a standard Rhino template, a few additional layers are also created.

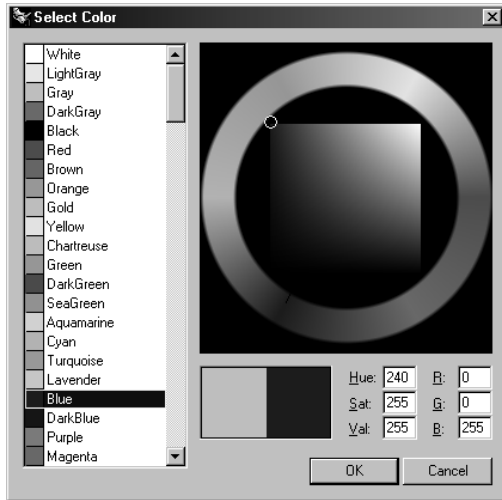
- 2 In the **Layers** window, click **New**.
- 3 The new **Layer 06** appears in the list, type **Line** and press **Enter**.
- 4 Click **New**.
- 5 The new **Layer 06** appears in the list, type **Curve** and press **Enter**.



**Layers**

### To assign a color to a layer:

- 1 Click the **Color** patch on the **Line** row in the list.



- 2 In the **Select Color** dialog box, click **Red** from the list.  
The right half of the sample rectangle turns red.  
Hue, Sat, Val are the hue, saturation and value components of the color.  
R, G, and B are the red, green and blue components of the color.
- 3 Click **OK**.
- 4 In the **Layers** window, 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 In the **Status Bar**, click the **Layer** pane.
- 2 In the Layer popup, click **Line**.  
Clicking the name or the checkbox sets the current layer.
- 3 Draw some lines.  
The lines are on the Line layer and they are colored red.

### Notes:

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

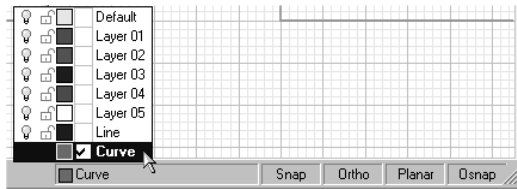
*Hue is the color that is referred to as a scale ranging from red through yellow, green and blue and then circularly back to red.*

*Saturation and Value are controlled by moving the small 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.*

## Notes:

- 4 To make a different layer current, click the **Layer** pane of the status bar.



- 5 Click **Curve**.
- 6 Draw some curves.  
They are on the Curve layer and are colored blue.
- 7 Draw more lines and curves on each layer.

### To lock a layer:

- 1 From the **Edit** menu, click **Layers**, and then click **Layers**.
- 2 In the **Layers** window, click the **Lock** icon in the row for **Line**, and click **OK**.  
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.

### To turn a layer off:

- 1 From the **Edit** menu, click **Layers**, and then click **Layers**.
- 2 In the **Layers** window, click the **On/Off** icon (light bulb) in the row for **Curve**, and click **OK**.  
Turning a layer off makes all objects on it invisible.

## Exercise 8—Selecting objects

### To select a single object:

- ▶ Move your pointer arrow over the object and left-click.  
The object turns yellow, which is the default highlight color.

### 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 left of the objects you want to select.
- 2 Hold your left mouse button down and drag diagonally to the right until you have several objects inside the selection box.  
The window selection box is a solid rectangle
- 3 Release your mouse button.  
All objects completely inside the selection box will be selected.
- 4 To add to your selection set, hold the **Shift** key down while making another selection.

### To select more than one object using a crossing window:

- 1 Move your pointer arrow into an open area to right of the objects you want to select.  
Hold your left mouse button down and drag diagonally to the left until you have several objects inside or touching the box.  
The crossing selection box is a dotted rectangle.
- 2 Release your mouse button.  
All objects inside or touching the box will be selected.
- 3 To add to your selection set, hold the **Shift** key down while making another selection.

### To hide an object:

- 1 Select an object.
- 2 From the **Edit** menu, click **Visibility**, and then click **Hide**.  
The object becomes invisible.

### To show hidden objects:

- ▶ From the **Edit** menu, click **Visibility**, and then click **Show**.  
The **Show** command redisplay all hidden objects.

### To lock an object:

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

## Notes:

HIDE

Hide  
Left click for hide.

HIDE

Show  
Right click this button.

LOCK

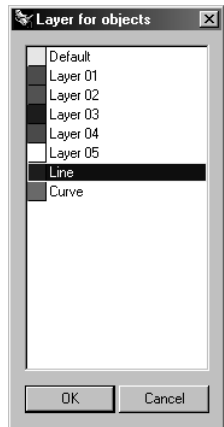
Lock

### To unlock locked objects:

- ▶ From the **Edit** menu, click **Visibility**, and then click **Unlock**.  
The **Unlock** command redisplay all locked objects.

### To change an object from one layer to another:

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



- 3 In the **Layer for object** dialog box, select the new layer for the object, and click **OK**.

### Notes:

LOCK

Unlock  
Right-click for unlock.



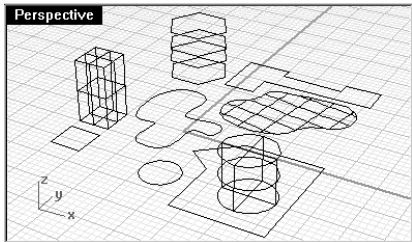
Change Layer

## Deleting Objects

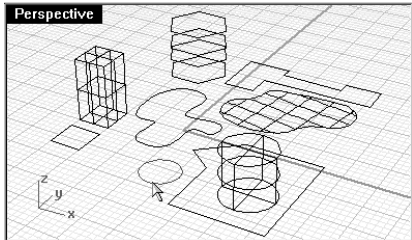
Delete removes selected objects from the model. Use **Delete** to practice selection.

### Exercise 9—Practice using delete and selection options

- 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.

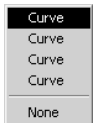


- 4 From the **Edit** menu, click **Delete** or press the **Delete** key.  
The objects disappear.

### To start the first practice:

- 1 Select one of the lines on the hexagon in the **Top** viewport.

Because there are several curves superimposed on each other, the selection menu appears, allowing you to select one of the curves.



- 2 Select the top curve from the list.

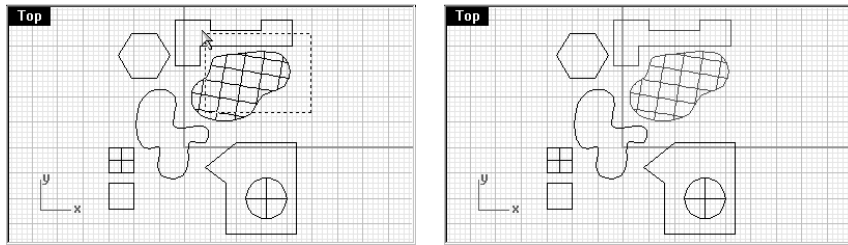


- 3 From the **Edit** menu, click **Delete**.

Observe one disappear in the **Perspective** viewport.

- 4 In the Top viewport, use a crossing box to select the surface and the polyline in the top right part of the drawing.

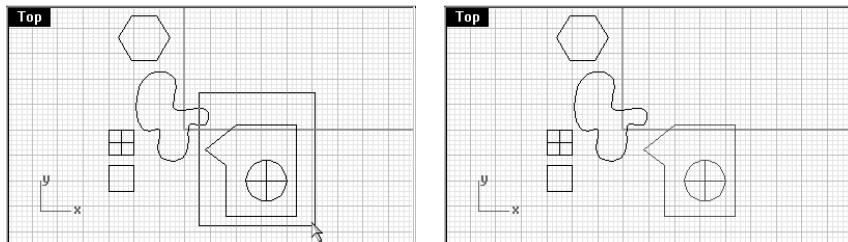
Both objects are selected.



- 5 From the **Edit** menu, click **Delete**.

- 6 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.



- 7 From the **Edit** menu, click **Delete**.

- 8 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, **Undo** takes you back one command.

#### To redo:

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

Each time you click, the previous **Undo** is reinstated.

#### Notes:



Undo  
Left click for undo.



Redo  
Right-click for redo.



# 4

## Precision Modeling

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

Whenever you draw a curve, 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 line**, **Start of polyline**, **Start of curve**, or **Next point** and the arrow-shaped cursor turns into a cross-shaped cursor.

You can enter a point two ways: pick a point in a viewport with the mouse, or type coordinates at 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 three-dimensional space.

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.

### Notes:

*Set the units and tolerance of the model before you begin.*

*You can change the tolerance after you start, but objects created before the change still have the old tolerance value.*



Cross-shaped cursor

## Absolute Coordinates

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

### Exercise 10—Setting up a model

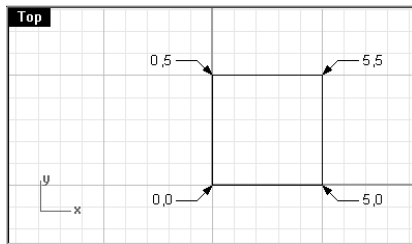
- 1 From the **File** menu, click **New**.
- 2 Click **Millimeters.3dm**, and then click **Open**.
- 3 From the **File** menu, click **Save As**.

Name the model **BOXES**.

Use the **BOXES.3dm** model to learn how to draw with absolute coordinates.

### Exercise 11—Entering absolute coordinates

- 1 Double-click the viewport title to maximize the **Top** viewport,.
- 2 From the **Curve** menu, click **Lines**, and 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, click **Close** to close the polyline.



## Notes:



Polyline

Left click for Polyline.

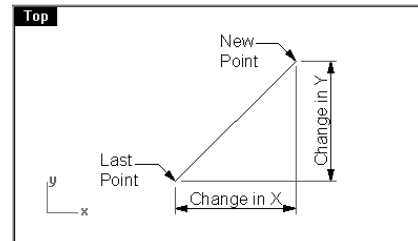
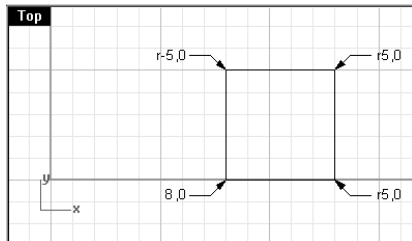
## Relative Coordinates

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 the last point entered, instead of on the origin (0,0,0) of the construction plane.

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

### Exercise 12—Entering relative coordinates

- 1 From the **Curve** menu, click **Lines**, and 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**.  
These are absolute coordinates.
- 4 At the **Next point of polyline**. Press Enter when done ( Undo ) prompt, type **R0,5** and press **Enter**.  
These are relative coordinates.
- 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, click **Close** to close the polyline.

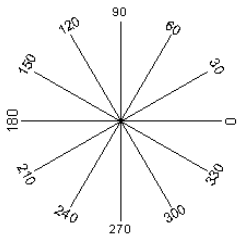
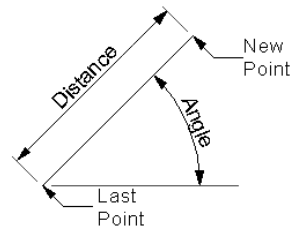
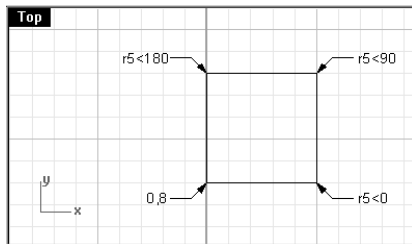


## Polar Coordinates

**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 **R**; absolute polar coordinates are not. Instead of using x-, y-, and z-coordinates, enter relative polar coordinates like this: *Rdistance<angle*.

### Exercise 13—Entering polar coordinates

- 1 From the **Curve** menu, click **Lines**, and 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, click **Close** to close the polyline.



## Distance and Angle Constraint Entry

Using distance constraint entry, you can specify a point by typing a distance and pressing **Enter**. Then as you move your cursor in any direction, the distance from the last point will be constrained. This is a good way to specify a line length quickly.

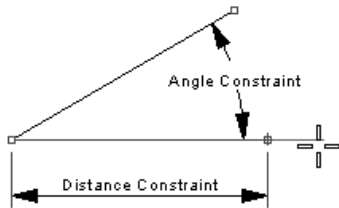
Using angle constraint entry, you can specify an angle by typing < followed by a value and pressing **Enter**. The next point is constrained to lines at multiples of the angle relative to the x-axis you specified.

### 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, draw a line 5 units long using distance constraints.

### Exercise 14—Distance constraint entry

- 1 From the **Curve** menu, click **Lines**, and 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**.
- 4 Hold the **Shift** key down and pick a point to the right.  
Ortho constrains the cursor to 0 degrees.



*Distance constraining value 5*

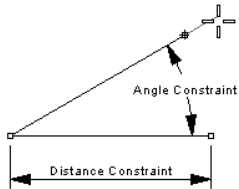
- 5 At the **Next point of polyline**. Press Enter when done ( Undo ) prompt, type **5** and press **Enter**.
- 6 Hold the **Shift** key down and pick a point up.  
Ortho constrains the cursor to 90 degrees
- 7 At the **Next point of polyline**. Press Enter when done ( Close Undo ) prompt, type **5** and press **Enter**.
- 8 Hold the **Shift** key down and pick a point to the left.  
Ortho constrains the cursor to 180 degrees.

## Notes:

- At the **Next point of polyline**. Press Enter when done ( Close Undo ) prompt, click **Close** to close the polyline.

### Exercise 15—Distance and angle constraint entry

- From the **Curve** menu, click **Lines**, and then click **Polyline**.
- At the **Start of polyline** prompt, type **16,5** and press **Enter**.
- 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 the marker snaps to a distance of 5 and an angle of 45 degrees.
- Pick a point down and to the right.  
The angle constraint sets the angle.



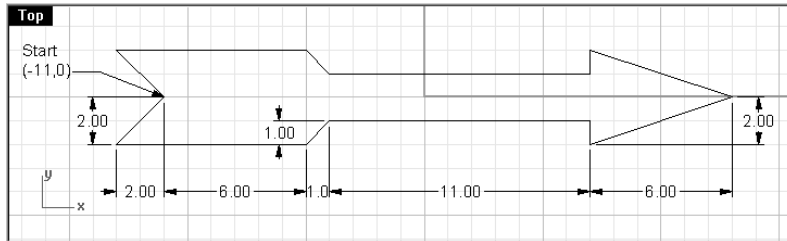
#### Angle constraint value 30

- 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 up and to the right.  
The angle constraint sets the angle.
- 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 up and to the left.  
The angle constraint sets the angle.
- At the **Next point of polyline**. Press Enter when done ( Close Undo ) prompt, click **Close** to close the polyline.
- Save your model. You will use this model for another exercise.



**Exercise 16—Practice using distance and constraint entry**

- 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 (Rx,y), polar coordinates (Rdistance<angle), and distance constraint.

Begin your model at **-11,0**. The following is an example of the command line input that you might use:

Next point of polyline ( Undo ): **r-2,-2**

Next point of polyline. Press Enter when done ( Undo ): **r8,0**

Next point of polyline. Press Enter when done ( Close Undo ): **r1,1**

Next point of polyline. Press Enter when done ( Close Undo ): **r11<0**

Next point of polyline. Press Enter when done ( Close Undo ): **r0,-1**

Next point of polyline. Press Enter when done ( Close Undo ): **r6,2**

Next point of polyline. Press Enter when done ( Close Undo ): **r-6,2**

Next point of polyline. Press Enter when done ( Close Undo ): **r0,-1**

Next point of polyline. Press Enter when done ( Close Undo ): **r11<180**

Next point of polyline. Press Enter when done ( Close Undo ): **r-1,1**

Next point of polyline. Press Enter when done ( Close Undo ): **r8<180**

Next point of polyline. Press Enter when done ( Close Undo ): **c**

- 3 Save your model.

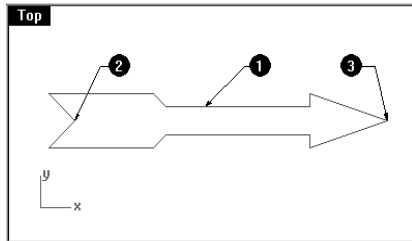
## Notes:



Revolve

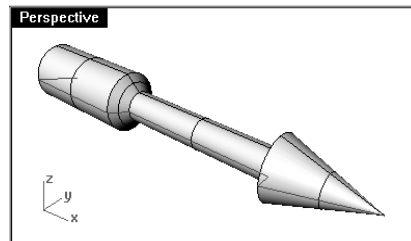
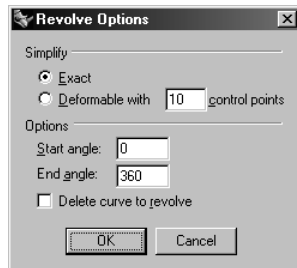
### To make it 3-D:

- 1 Select the polyline (1).
- 2 From the **Surface** menu and click **Revolve**.
- 3 Toggle **Snap** on.



- 4 At the **Start of revolve axis** prompt, select the end of the arrow along the centerline (2).
- 5 At the **End of revolve axis** prompt, select the other end of the arrow (3) along the centerline.
- 6 In the **Revolve Options** dialog box, click **OK**.

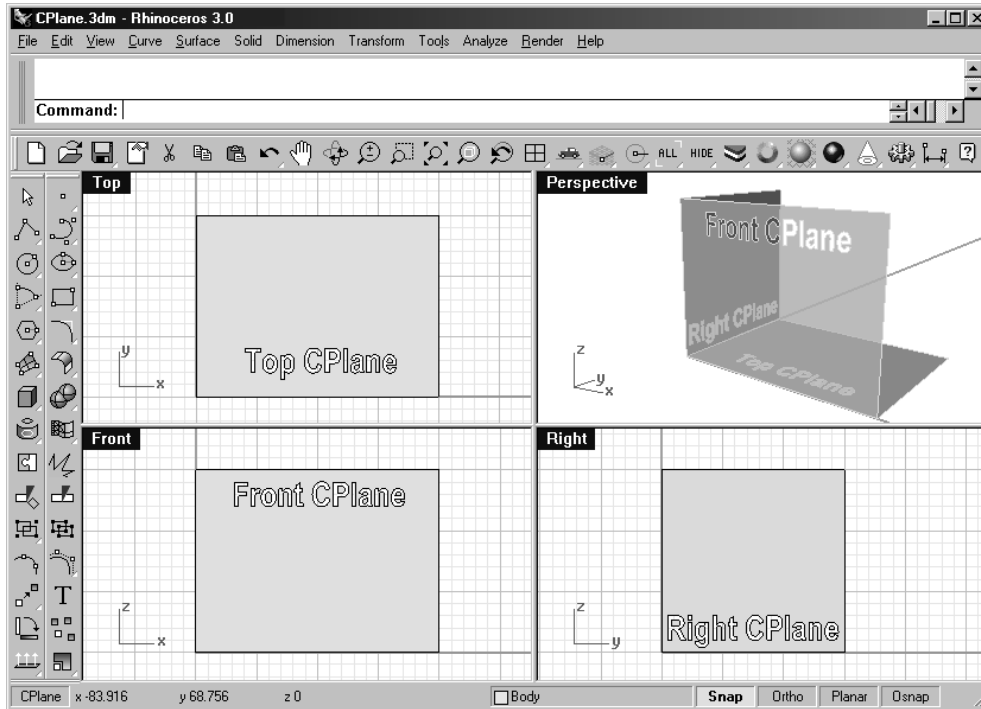
Your arrow is now a three-dimensional model.



## Notes:

## Viewports

Viewports are windows in the Rhino graphics area that show you a view of your model. To move and resize viewports, drag the viewport title 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.



## 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** viewport uses the **Top** construction plane.

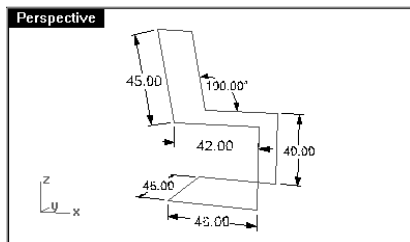
Each construction plane has a grid. The grid is a plane of perpendicular lines lying on the construction plane. On default grids, every fifth line is slightly thicker. The red line represents the construction plane x-axis. The green line represents the construction plane y-axis. The red and green lines 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 plane axes 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 17—Modeling in 3-D space

Rhino makes it easy to draw in 3-D space. You can draw on a different construction plane by simply moving your cursor into a different viewport. Another useful tool for modeling in 3-D space is **elevator mode**.

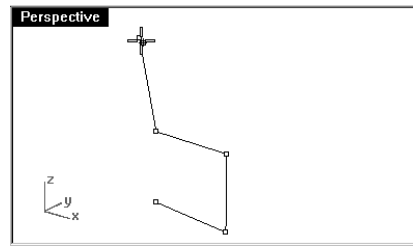
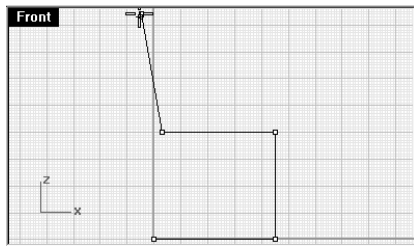
In the following exercise we will draw in different viewports and use elevator mode to move some points in 3-D space.



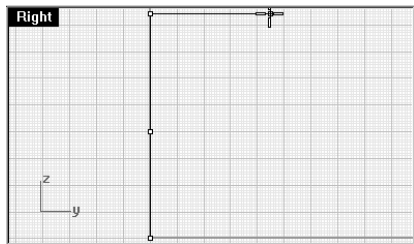
## Notes:

You will use Snap and Ortho to draw in different viewports.

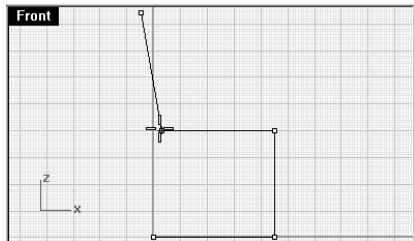
- 1 Open the model **Chair.3dm**.  
The model units are centimeters.
- 2 Toggle **Planar** and **Snap** on. Toggle **Ortho** on as needed.
- 3 From the **Curve** menu, click **Lines**, and then click **Polyline**.
- 4 Move your cursor into the **Front** viewport.
- 5 At the **Start of polyline** prompt, type **0,0** and press **Enter**.
- 6 At the **Next point of polyline** prompt, use coordinate input to draw the first part of the chair frame.



- 7 At the **Next point of polyline** prompt, move your cursor to the **Right** viewport to draw a horizontal line.

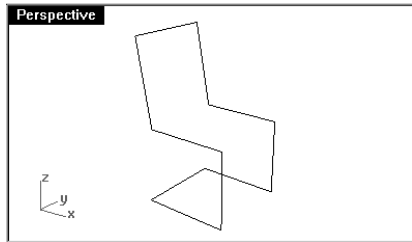


- 8 At the **Next point of polyline** prompt, move your cursor to the **Front** viewport hold down the **Ctrl** key, and pick the point at the lower end of the diagonal line.





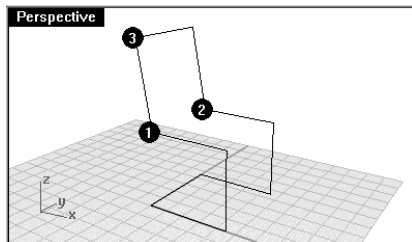
12 At the **Next point of polyline** prompt, click **Close**.



We now want to draw along the chair back, so we will change the construction plane.

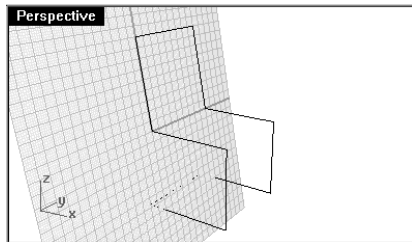
### To change the construction plane:

- 1 On the **Status Bar**, click **Osnap**, and check **End**.
- 2 From the **View** menu, click **Set CPlane**, then click **3 Points**.
- 3 At the **CPlane origin** prompt, move your cursor to the **Perspective** viewport, and pick the vertex (1) at the back of the chair.



- 4 At the **X axis direction** prompt, pick the vertex (2) at the other side of the back.
- 5 At the **CPlane orientation** prompt, pick the vertex (3) at the top of the chair.

The construction plane is now aligned with the back of the chair.

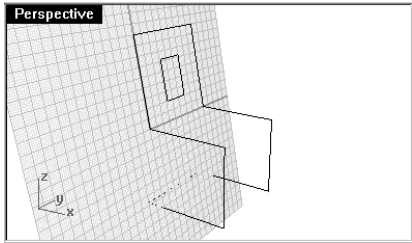


### Notes:



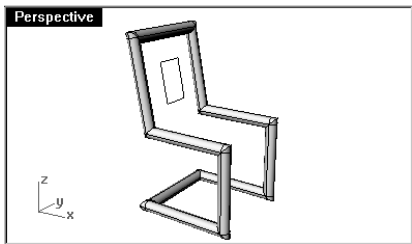
Set CPlane: 3 Points

- 6 Draw some lines on the new construction plane.



**To make it solid:**

- 1 Select the chair frame.
- 2 From the **Solid** menu, click **Pipe**.
- 3 At the **Radius for closed pipe <1> ( Diameter )** prompt, type **3** and press **Enter**.  
The chair has a solid frame.



- 4 Save the model.

**On your own:**

Try some variations and add some features.



**Notes:**



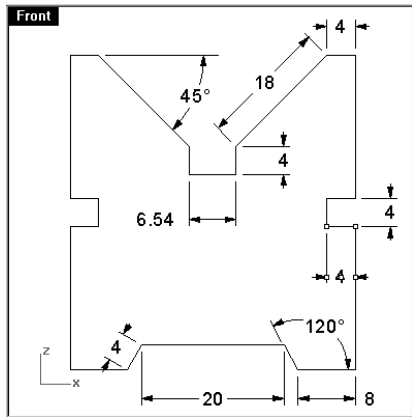
Pipe



## Notes:

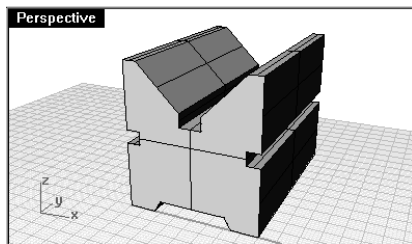
### Exercise 18—Practice using distance and angle constraints

- 1 Start a new model using the **Millimeters** template. Save as **V-Block**.
- 2 Double-click the viewport title in the **Front** viewport to maximize it.  
Create the following model in the front construction plane.
- 3 Draw the object below using a combination of absolute coordinates (x,y), relative coordinates (rx,y), and relative polar coordinates (rdistance<angle).
- 4 Start your model at **0,0** in the **Front** viewport.  
Try to create the model using a single polyline.



- 5 Double-click the viewport title of the **Front** viewport to restore your views.
- 6 Select the polyline.
- 7 From the **Solid** menu, click **Extrude Planar Curve**.
- 8 At the **Extrusion distance <1>** ( Direction Cap=Yes BothSides Tapered ) prompt, type **60** and press **Enter**.

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



- 9 Save your model.



Extrude Planar Curve

## Object Snaps

**Object Snaps** are tools for specifying points on existing objects. Use them for precision modeling and to get accurate data. Object snaps are often referred to as *osnaps*. In Rhino, reliable modeling and easy editing depends on objects actually meeting at specified points. Object snaps give you precision you cannot get using the “eyeball” method











### To open the Osnap toolbar

- ▶ Click the **Osnap** pane 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 an eligible point on an object causes the marker to jump to that point and a tooltip to appear.

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

Command	Button	Description
<input type="checkbox"/> End <input type="checkbox"/> Near <input type="checkbox"/> Point <input type="checkbox"/> Mid <input type="checkbox"/> Cen <input type="checkbox"/> Int <input type="checkbox"/> Perp <input type="checkbox"/> Tan <input type="checkbox"/> Quad <input type="checkbox"/> Knot <input type="checkbox"/> Project <input type="checkbox"/> Disable		
End		End snaps to the end of a curve, surface edge or polyline segment.
Near		Near snaps to the nearest point on an existing curve or surface edge.
Point		Point snaps to a control point or point object.
Mid		Midpoint snaps to the midpoint of a curve or surface edge.
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 makes a perpendicular to the last selected point. Works only when you are selecting a series of points.
Tan		Tangent To snaps to the point on a curve that makes a tangent to the last selected point. This only works when you are selecting a series of points.
Quad		Quad snaps to the quadrant point. The quadrant point is the maximum or minimum direction on a curve in the x or y construction plane direction.
Knot		Knot snaps to knot points on curves or surface edges.

Command	Button	Description
Project		Projects the snap point to the construction plane.
Disable		Temporarily turns off running object snaps, retaining settings

### Exercise 19—Using object snaps

- 1 **Open** the model **Osnap.3dm**.
- 2 Toggle **Snap** and **Ortho** off.

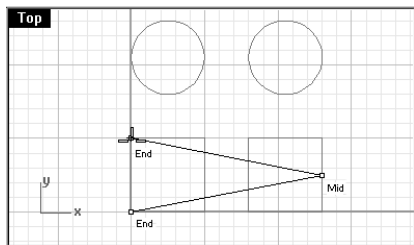
### Using End and Midpoint object snaps:

- 1 Click the **Osnap** pane on the status bar.  
You can leave the **Osnap** toolbar displayed.



*Osnap bar with End and Mid on.*

- 2 Check **End** and **Mid**.  
You can check and clear individual object snaps to make modeling with precision easy.
- 3 From the **Curve** menu, click **Lines**, and 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 square and pick when the marker snaps to the end of the line.
- 5 Pick points to draw the polyline.  
The line starts exactly at that corner.



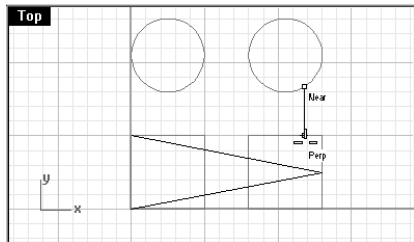
- 6 At the **Next point of polyline** prompt, snap to the midpoint of the right vertical edge of the second square.  
The marker snaps to the midpoint of the line that the cursor touches, making the new line cross to the exact midpoint of that side.
- 7 Pick to draw the polyline.

## Notes:

- 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, and press **Enter**.

### Using Near and Perpendicular to object snaps:

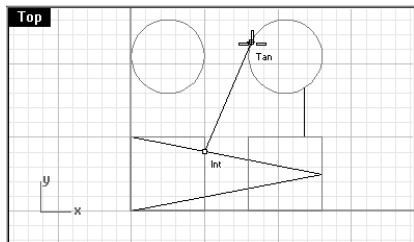
- In the **Osnap toolbar** check **Near** and **Perp**, clear **End** and **Mid**.
- From the **Curve** menu, click **Lines**, and then click **Polyline**.
- At the **Start of polyline** prompt, pick on the lower edge the circle at the top right.  
The marker snaps to the point on the circle nearest to where the cursor is positioned.



- At the **Next point of polyline** prompt, pick the top horizontal edge of the second square.  
The marker snaps to a point making a perpendicular to the previous point.
- Pick to draw the polyline, and press **Enter**.

### Using Intersection and Tangent to object snaps:

- In the **Osnap toolbar** check **Int** and **Tan**; clear **Near** and **Perp**.
- From the **Curve** menu, click **Lines**, and then click **Polyline**.
- At the **Start of polyline** prompt, pick the intersection where the diagonal line crosses the vertical line on the first square.  
The marker snaps to the intersection between the two lines.

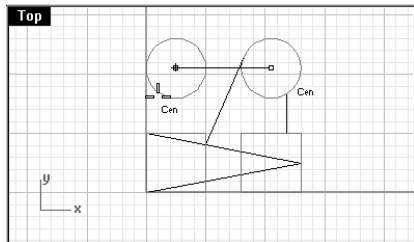


## Notes:

- 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 Pick to draw the polyline segment, and press **Enter**.

### Using Center of object snap:

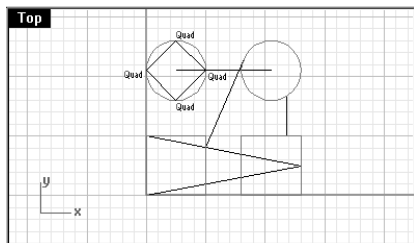
- 1 In the **Osnap toolbar** check **Cen**, clear **Int** and **Tan**.
- 2 From the **Curve** menu, click **Lines**, and then click **Polyline**.
- 3 At the **Start of polyline** prompt, pick on the edge of a circle.  
The marker snaps to the center of the circle.



- 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 Pick to draw the polyline segment, and press **Enter**.

### Using Quadrant of object snap:

- 1 In the **Osnap toolbar** check **Quad**, clear **Cen**.
- 2 From the **Curve** menu, click **Lines**, and then click **Polyline**.
- 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, click **Close**.

## Analysis Commands

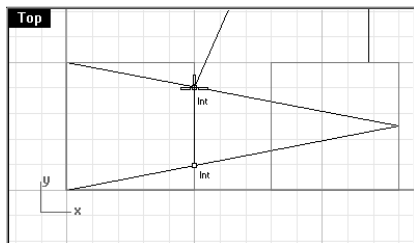
Rhino provides analysis tools for finding lengths, angles, areas, distances, and the volume and centroid of solids. Additional commands let you analyze curve curvature, determine continuity between curves, and find unjoined edges.

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.
EvaluatePt	The Cartesian coordinates of the point, in both world and construction plane 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 **Measure 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.

Use object snaps.



## Notes:



Distance

- 4 Press **F2** to display the information.

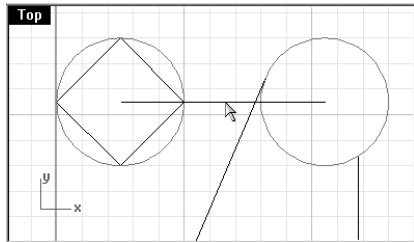
CPlane angles and deltas:  $xy = 90$  elevation = 0  $dx = 0$   $dy = 3.077$   $dz = 0$

World angles and deltas:  $xy = 90$  elevation = 0  $dx = 0$   $dy = 3.077$   $dz = 0$

Distance = 3.077 millimeters

#### To find the length of a line:

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

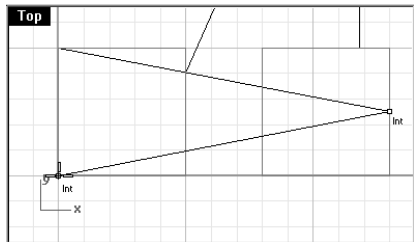


Length = 8.000 millimeters

#### To measure the angle between two lines:

- 1 From the **Analyze** menu, click **Measure Angle**.
- 2 At the **Start of line** prompt, select a point that defines the start of an angle line.
- 3 At the **End of first line** prompt, select a point that defines the end of an angle line.

Use object snaps as appropriate.



- 4 At the **Start of second line** prompt, select a point that defines the beginning of the second angle line.

#### Notes:

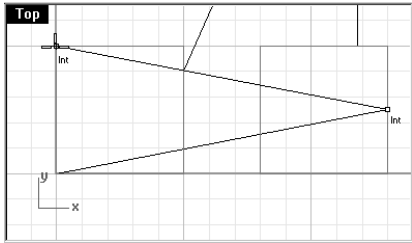


Length



Angle

- 5 At the **End of second line** prompt, select a point that defines the end of the second angle line.

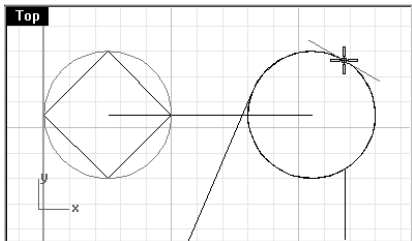


The angle displays on the command line in the following format:  
Angle is 21.7711 degrees

#### To measure the radius of a circle:

- 1 From the **Analyze** menu, click **Measure Radius**.
- 2 At the **Select point on curve for radius measurement** prompt, select one of the circles.

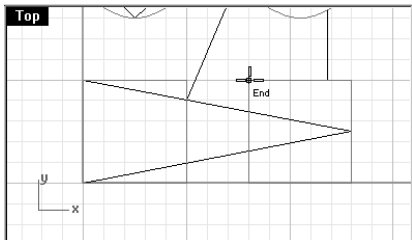
This also measures a radius on a curve.



The radius displays on the command line in the following format:  
Radius at pick point is 2.5

#### To evaluate a point:

- 1 From the **Analyze** menu, click **Analyze Point**.
- 2 At the **Point to evaluate** prompt, snap to the end point of one of the squares.



#### Notes:



Radius



Point








## Notes:

The x,y,z point is displayed for the world coordinate plane and the current construction plane.  
Point in world coordinates = 8.000,5.000,0 CPlane coordinates = 8.000,5.000,0

## 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, and tangent points to two coplanar curves and a radius.

Button	Command	Description
	Circle	Draw a circle using the center and radius.
	Circle 3Point	Draw a circle through three points on its circumference
	Circle Diameter	Draw a circle using two diameter points.
	Circle Tangent	Draw a circle tangent to two or three curves.
	Circle AroundCurve	Draw a circle perpendicular to a curve at the chosen point.

## Circle options

Option	Description
Deformable	Draws an approximation of a circle with a defined number of control points..
Vertical	Draw a circle perpendicular to the construction plane.

## Exercise 20—Drawing circles

### To draw a center, radius circle:

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



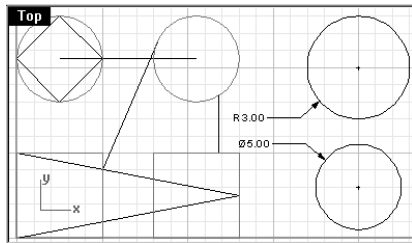
Circle

## Notes:

### To draw a center, diameter circle:

- 1 From the **Curve** menu, click **Circle**.
- 2 At the **Center of circle** ( Deformable Vertical 2Point 3Point Tangent AroundCurve ) prompt, type **20,3** and press **Enter**.
- 3 At the **Radius <3>** ( Diameter ) prompt, click **Diameter**.
- 4 At the **Diameter <6.00>** ( Radius ) prompt, type **5** and press **Enter**.

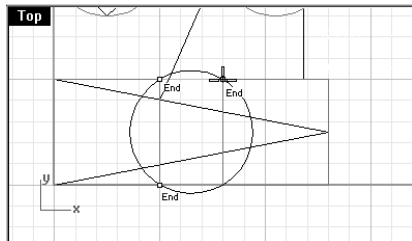
A circle is created that is based on a center point and a diameter. The Diameter option will be the default until you change it.



### To draw a 3-point circle:

- 1 From the **Curve** menu, click **Circle**.
- 2 At the **Center of circle** ( Deformable Vertical 2Point 3Point Tangent AroundCurve ) prompt, click **3Point**.
- 3 At the **First point on circle** prompt, snap to an **End** on one of the squares.
- 4 At the **Second point on circle** prompt, snap to another vertex on the same square.
- 5 At the **Third point on circle** prompt, snap to a third vertex on the other square.

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

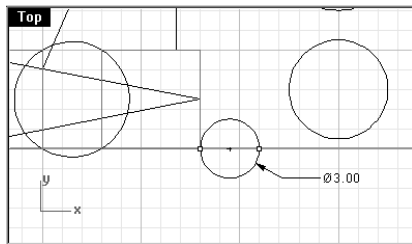


Circle 3Pt

### To draw a diameter circle:

- 1 From the **Curve** menu, click **Circle**.
- 2 At the **Center of circle** ( Deformable Vertical 2Point 3Point Tangent AroundCurve ) prompt, click **2Point**.
- 3 At the **First point on circle** prompt, pick a point on the screen.
- 4 At the **Second point on circle** prompt, type **3** and press **Enter**, then turn **Ortho** on and pick to the right.

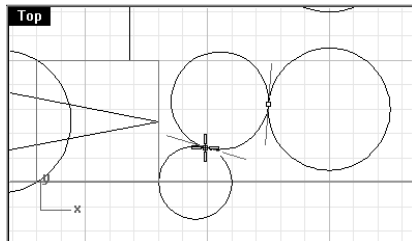
A circle is created with the two points you picked as the diameter, and the diameter was constrained to a value of 3.



### To draw a tangent, tangent, radius circle:

- 1 From the **Curve** menu, click **Circle**.
- 2 At the **Center of circle** ( Deformable Vertical 2Point 3Point Tangent AroundCurve ) prompt, click **Tangent**.
- 3 At the **First point** ( Point ) prompt, select the circle you just created near the top edge.
- 4 At the **Second point or radius <5>** ( Point FromFirstPoint ) prompt, type **2** and press **Enter**.
- 5 At the **Second point or radius <2>** ( Point FromFirstPoint ) prompt, pick the other circle.

A circle is created that is tangent to the two circles you selected with a radius of 2.



### Notes:



Circle Diameter

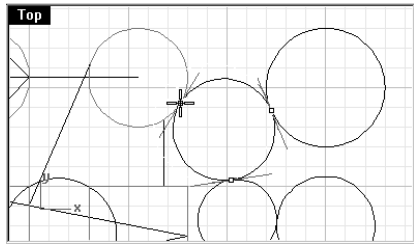


Circle Tangent, Tangent, Radius

### To draw a tangent, tangent, tangent circle:

- 1 From the **Curve** menu, click **Circle**.
- 2 At the **Center of circle** ( Deformable Vertical 2Point 3Point Tangent AroundCurve ) prompt, click **Tangent**.
- 3 At the **First point** ( Point ) prompt, select a circle you created.
- 4 At the **Second point or radius <2>** ( Point FromFirstPoint ) prompt, select another circle or line.
- 5 At the **Third point** ( Point ) prompt, pick another piece of geometry.

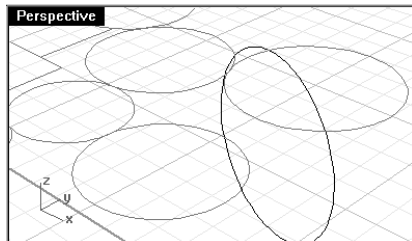
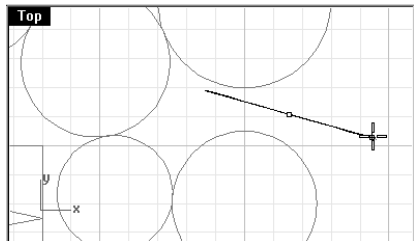
A circle is created that is tangent to the three pieces of geometry that you selected.



### To draw a circle vertical to the construction plane:

- 1 From the **Curve** menu, click **Circle**.
- 2 At the **Center of circle** ( Deformable Vertical 2Point 3Point Tangent AroundCurve ) prompt, click **Vertical**.
- 3 At the **Center of circle** ( 2Point ) prompt, pick a point.
- 4 At the **Radius <4>** ( Diameter ) prompt, type **3** and press **Enter**.

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



### Notes:



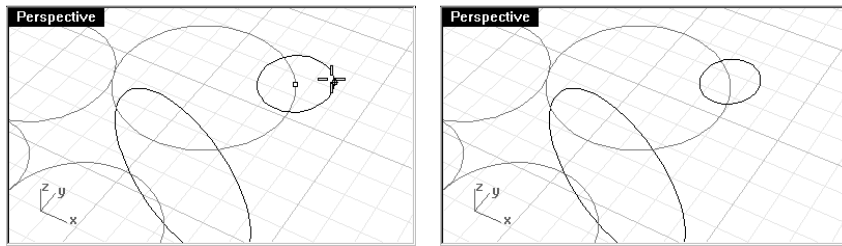
Circle Tangent to 3 Curves

## Notes:

### To draw a circle around a curve:

- 1 From the **Curve** menu, click **Circle**.
- 2 At the **Center of circle** ( Deformable Vertical 2Point 3Point Tangent AroundCurve ) prompt, click **AroundCurve**.
- 3 At the **Center of circle** prompt, pick a point.
- 4 At the **Radius <2>** ( Radius ) prompt, click **Radius**.
- 5 At the **Radius <2>** ( Diameter ) prompt, type **1** 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.

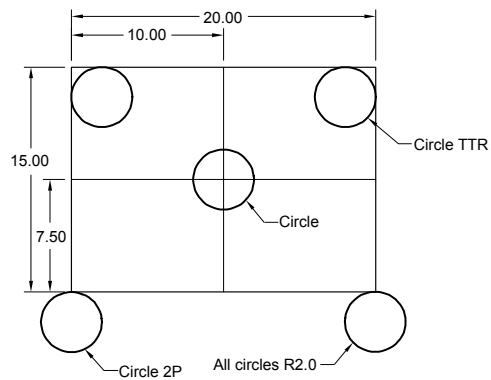


## Notes:

### Exercise 21—Practice drawing circles

- 1 Start a new model. Save as **Circles**.
- 2 From the **Edit** menu, click **Layers**, and then click **Edit**.
- 3 In the **Layers** window, click the **New** icon 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.

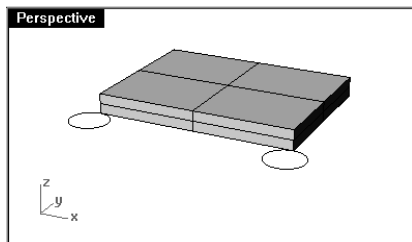
Use **Line Segments** for the box and **Single Line** for the centerlines. You will use several of the circle options and object snaps to complete this model.



### To make it 3-D:

- 1 Select the lines that form the rectangle.
- 2 From the **Surface** menu, click **Extrude**.
- 3 At the **Distance** ( Direction BothSides=No Cap=No Mode=Straight ) prompt, click **Cap**.
- 4 At the **Distance** ( Direction BothSides=No Cap=Yes Mode=Straight ) prompt, type **2** and press **Enter**.

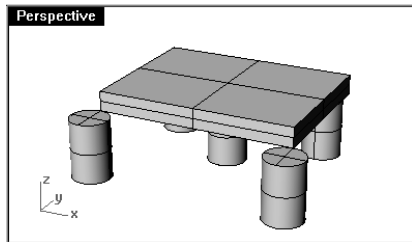
The rectangle generates a box.



## Notes:

- 5 Select the circles.
- 6 From the **Surface** menu, click **Extrude**.
- 7 At the **Distance** ( Direction BothSides=No Cap=Yes Mode=Straight ) prompt, type **-6** and press **Enter**.
- 8 At the **Distance** ( Direction BothSides=No Cap=Yes Mode=Straight ) prompt, drag the mouse in the Front view and click.

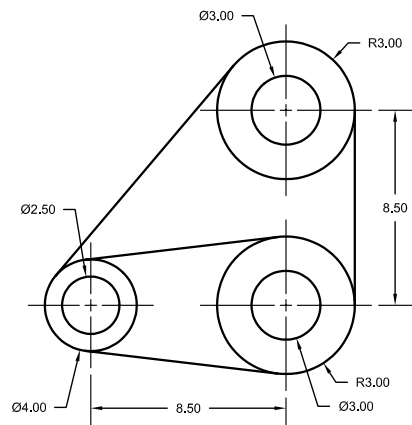
The circles generate cylinders.



### Exercise 22—Using circle-related object snaps

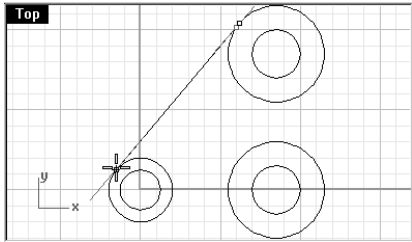
- 1 Start a new model. Save as **Link**.
- 2 Complete the model as shown.
- 3 Draw the three large circles first.
- 4 Draw the small holes next.

Use object snaps to snap to the centers of the large circles.



**To draw the tangent lines:**

- 1 From the **Curve** menu, click **Lines**, and then click **Single Line**.
- 2 At the **Start of line** ( Normal Angled Vertical FourPoint Bisector Perpendicular Tangent Extend BothSides ) prompt, click **Tangent**.
- 3 At the **Start of line** prompt, pick on the edge of one of the circles near where you want the tangent line to attach.
- 4 At the **End of line** prompt, pick on the edge of another circle it will find the tangent points for you.



- 5 Continue to use this command to complete the model.
- 6 Save your model.

**Notes:**



Line Tangent to 2 Curves










## Notes:

## Drawing Arcs

You can create arcs using various points on the arc and construction geometry.

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

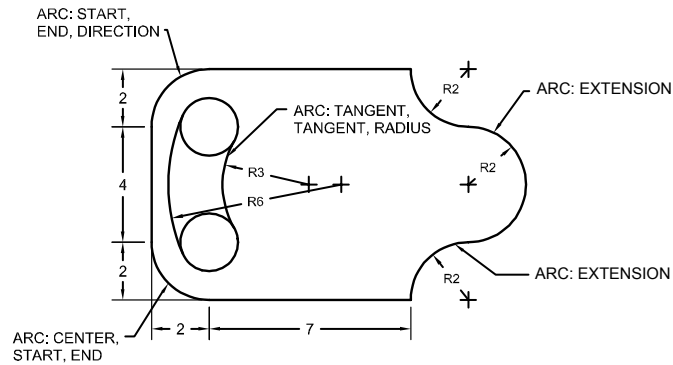
Button	Command	Description
	Arc	Draw an arc from the center, start, and angle.
	Arc 3Points	Draw an arc from three points.
	Arc Start, End, Direction	Draw an arc from start point, end point, and the direction from the start point. The direction can be entered after the start point is entered or after the end point is entered.
	Arc Tangent, Tangent, ThroughPoint	Creates an arc from tangents and radius.
	Arc Start End Radius	Creates an arc from start point, end point, and radius.
	Convert Output=arcs	Converts a curve to arc segments that are joined together.
	CurveThroughPt Convert Output=arcs	Creates an interpolated curve through selected points and then converts the curve to arc segments.

## Arc options

Option	Description
Deformable	Creates uniform cubic splines. You can enter the number of control points..
Extension	Extend a curve with an arc.

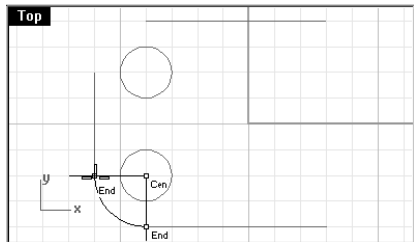
## Exercise 23—Practice drawing arcs (1)

### ► Open the model **Arc1.3dm**.



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

- 1 From the **Curve** menu, click **Arc**.
- 2 At the **Center of arc** ( Deformable StartPoint Tangent Extension ) 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 of arc** prompt, snap to the end of the other line.



### To draw a start, end, direction arc:

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

## Notes:



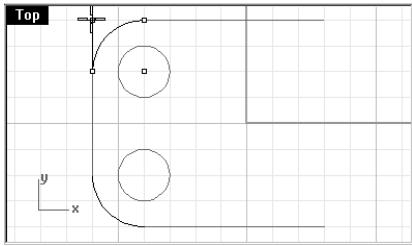
Arc



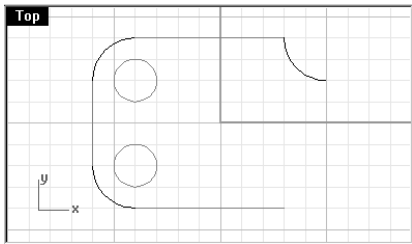
Direction Arc

## Notes:

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

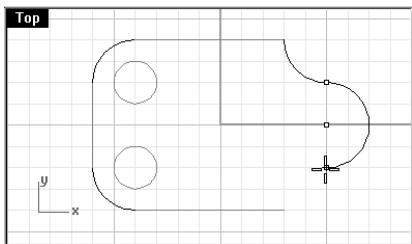


- 5 Make another **Direction Arc** at the upper right.



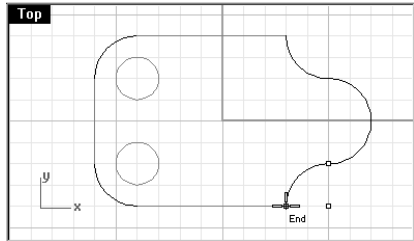
### To add more arc segments:

- 1 From the **Curve** menu, click **Arc**.
- 2 At the **Center of arc** ( Deformable StartPoint Tangent Extension ) prompt, type **E** and press **Enter**.
- 3 At the **Select curve near end** prompt, pick near the end of the arc you just made.
- 4 At the **End of arc** ( Center ) prompt, type and press **Enter**.
- 5 At the **End of arc** ( Center ) prompt, with Ortho on pick a point below the first point.  
The arc will be tangent to the curve you chose.



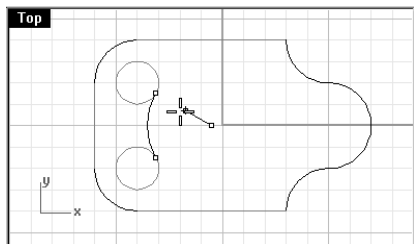
- 6 From the **Curve** menu, click **Arc**.

- 7 At the **Center of arc** ( Deformable StartPoint Tangent Extension ) prompt, type **E** and press **Enter**.
- 8 At the **Select curve near end** prompt, pick near the end of the arc you just made.
- 9 At the **End of arc** ( Center ) prompt, snap to the end of the line.



#### To draw a tangent, tangent, radius arc:

- 1 From the **Curve** menu, click **Arc**, and then click **Tangent, Tangent, Radius**.
- 2 At the **First tangent curve** ( Point ) prompt, pick the lower right side of the top circle.
- 3 At the **Second tangent curve or radius <1>** ( Point FromFirstPoint ) prompt, type **3** and press **Enter**.
- 4 At the **Second tangent curve or radius <3>** ( Point FromFirstPoint ) prompt, pick the upper right side of the lower circle.
- 5 At the **Choose arc** prompt, move your cursor and pick when the correct arc is displayed.



- 6 From the **Curve** menu, click **Arc**, and then click **Tangent, Tangent, Radius**.
- 7 At the **First tangent curve** ( Point ) prompt, pick the upper left side of the top circle.
- 8 At the **Second tangent curve or radius <1>** ( Point FromFirstPoint ) prompt, type **6** and press **Enter**.
- 9 At the **Second tangent curve or radius <6>** ( Point FromFirstPoint ) prompt, pick the lower left side of the lower circle.

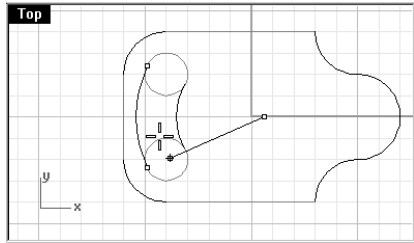
#### Notes:



Arc Tangent, Tangent, Radius

**Notes:**

10 At the **Choose arc** prompt, move your cursor and pick when the correct arc is displayed.

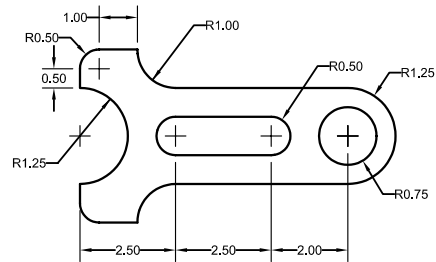


11 Save your model.

**Exercise 24—Practice drawing arcs (2)**

Draw construction centerlines first: use their intersection point to draw your arcs and circles.

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

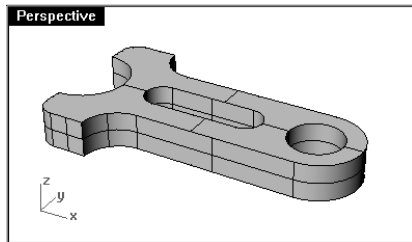


## Notes:

### To make it solid:

- 1 Select the curves.
- 2 From the **Surface** menu, click **Extrude**.
- 3 At the **Distance** ( Direction BothSides=No Cap=No Mode=Straight ) prompt, click **Cap**.
- 4 At the **Distance** ( Direction BothSides=No Cap=Yes Mode=Straight ) prompt, type **1** and press **Enter**.





The curves have been extruded and capped.



## 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.

### Ellipses

Button	Command	Description
	Ellipse	Draw an ellipse by specifying the center point and axis ends.
	Ellipse Diameter	Draw an ellipse by specifying the axis ends.
	Ellipse FromFoci	Draw the ellipse from the focus points
	AroundCurve	Draw an ellipse whose axis is perpendicular to a curve.

## Notes:






### Polygons

Button	Command	Description
	Polygon	Creates a polygon from its center and radius.
	Polygon Edge	Draw a polygon by specifying the ends of one edge.
	Polygon Star	Draw a star from a polygon.

### Polygon options

Option	Description
NumSides	Specify the number of sides for the polygon.
Circumscribed	Draw a polygon that is circumscribed about a radius. The default is to draw a polygon that is inscribed in a specified radius.

### Rectangles

Button	Command	Description
	Rectangle	Draw a rectangle using opposite corners.
	Rectangle Center	Draw a rectangle from center and a corner.
	3Point	Draw a rectangle through three points.
	Vertical	Draw a rectangle that is perpendicular to the construction plane.
	Rounded	Create a rectangle with rounded corners (arc or conic).





- 6 At the **Rho or point for conic corner to pass through <1>** ( Corner=Arc ) prompt type **1**, and press **Enter**.

If you are creating a rounded rectangle, select 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.

- 7 Repeat these steps for a second rounded rectangle with a length of **18** and a width of **8** with **.5** radius corners.

#### To draw an ellipse from its center and axis ends:

- 1 From the **Curve** menu, click **Ellipse**, and then click **From Center**.
- 2 At the **Center of ellipse** ( Corner Diameter FromFoci AroundCurve ) prompt, type **0,0** and press **Enter**.
- 3 At the **End of First axis** ( Corner ) prompt, type **4**, and press **Enter**.
- 4 At the **End of First axis** ( Corner ) prompt, toggle **Ortho** on and pick to the right.
- 5 At the **End of Second axis** prompt, type **3**, and press **Enter**.
- 6 At the **End of second axis** prompt, pick a point.

#### To draw a polygon from its center and radius:

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

#### Notes:



Ellipse

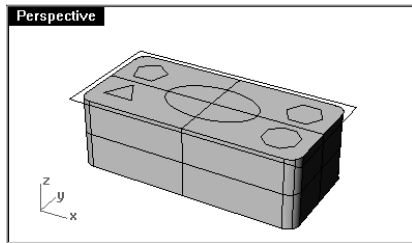


Polygon

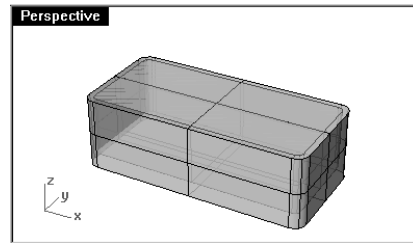
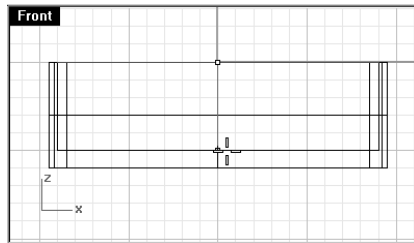
## Notes:

### To make the rounded rectangles solid:

- 1 Select the larger rounded rectangle.
- 2 From the **Surface** menu, click **Extrude**.
- 3 At the **Distance** ( Direction BothSides=*No* Cap=*No* Mode=*Straight* ) prompt, click **Cap**.
- 4 At the **Distance ...** prompt, drag it down to set the thickness and click.



- 5 Select the smaller rounded rectangle.
- 6 From the **Surface** menu, click **Extrude**.
- 7 At the **Distance** ( Direction BothSides=*No* Cap=*Yes* Mode=*Straight* ) prompt, click **Cap**.
- 8 At the **Distance ...** prompt, drag it down until it is slightly above the previous solid and click.

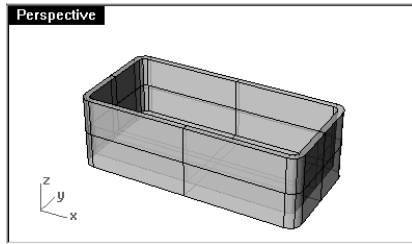


### To make the rounded rectangle hollow:

- 1 Select the outer rounded rectangle.
- 2 From the **Solid** menu, click **Difference**.

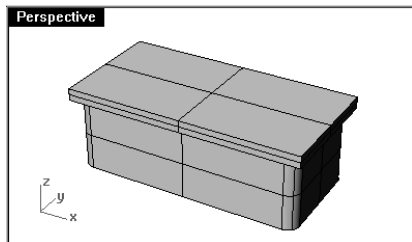
## Notes:

- 3 At the **Select second set of surfaces or polysurfaces. ...** prompt, select inner rounded rectangle, and press **Enter**.



### To make the rectangle solid:

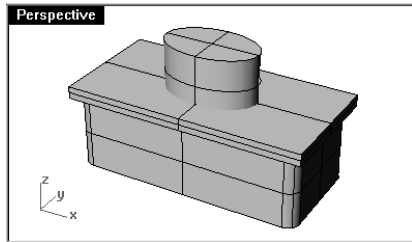
- 1 Select the rectangle.
- 2 From the **Surface** menu, click **Extrude**.
- 3 At the **Distance** ( Direction BothSides=No Cap=Yes Mode=Straight ) prompt, click **Cap**.
- 4 At the **Distance ...** prompt, drag it up to set the thickness and click.



### To make the ellipse solid:

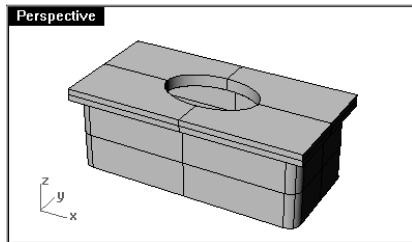
- 1 Select the ellipse.
- 2 From the **Surface** menu, click **Extrude**.
- 3 At the **Distance** ( Direction BothSides=No Cap= Yes Mode=Straight ) prompt, click **Cap**.
- 4 At the **Distance** ( Direction BothSides=No Cap= Yes Mode=Straight ) prompt, click **Bothsides**.

- 5 At the **Distance** ... prompt, pick to set the thickness.



#### To cut the ellipse solid from the rectangle:

- 1 Select the solid rectangles.
- 2 From the **Solid** menu, click **Difference**.
- 3 At the **Select second set of surfaces or polysurfaces. ...** prompt, select the solid ellipse, and press **Enter**.



#### To extrude the polygons:

- 1 Select the polygons.
- 2 From the **Surface** menu, click **Extrude**.
- 3 At the **Distance** ( Direction BothSides=No Cap=No Mode=Straight ) prompt, click **Cap**.
- 4 At the **Distance** ( Direction BothSides=No Cap=Yes Mode=Straight ) prompt, click **Bothsides**.
- 5 At the **Distance** ... prompt, pick to set the thickness.

#### To cut holes with the solid polygons:

- 1 Select the solid rectangles.
- 2 From the **Solid** menu, click **Difference**.
- 3 At the **Select second set of surfaces or polysurfaces. ...** ( DeleteInput=Yes ) prompt, type **D** and press **Enter**.

#### Notes:

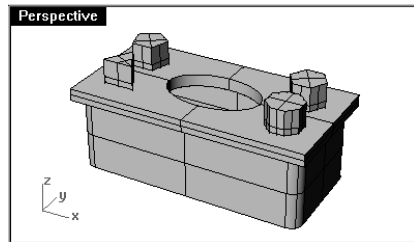


Boolean Difference

## Notes:




- At the **Select second set of surfaces or polysurfaces. ...** ( DeleteInput=No ) prompts, select the solid polygons and press **Enter**.

Holes will be cut, but the objects will remain.



## Modeling Free-Form Curves

The use of free-form curves allows more flexibility to create complex shapes.

Button	Command	Description
	Curve	Curve by control points draws a curve from specified control points. Control points mostly do not lie on the curve, but they determine its shape.
	InterpCrv	Interpolated curve creates a curve going through specified interpolate points. These points lie on the curve and determine its curvature.
	Conic	Draw a conic curve, which is part of an ellipse, parabola, or hyperbola.

Option	Description
Undo	Allows you to back up one point.
Close	Makes a closed curve.
Sharp	If Yes, makes a closed curve with a sharp point.
Degree	Allows you to set the degree of the curve.

### Exercise 26—Practice drawing curves (1)

- Open** the model **Curve.3dm**.

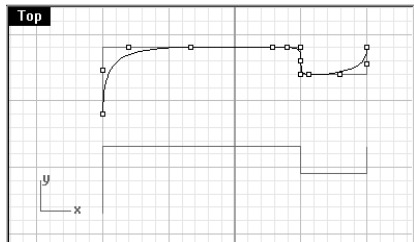
In this exercise, you will learn how to make a curve by control points, an interpolated curve, and a conic curve to compare the differences between the three methods.

A common method to create with free-form curves is to draw lines that are accurately measured to use as guidelines. In this exercise the guidelines have been created for you.

- 2 In the **Osnap** toolbar, check **End** and **Near**, clear all others.  
If you click **End** with your right mouse, it will clear all the others.
- 3 Toggle **Ortho** and **Snap** off.

**To draw a curve by control points:**

- 1 From the **Curve** menu, click **Free-form**, and then click **Control Points**.
  - 2 At the **Start of curve** ( Degree=3 ) prompt, snap to the end point of the polyline guideline.
  - 3 At the **Next point** ( Degree=3 Undo ) prompt, snap on the polyline guideline using the **Near** object snap.
  - 4 At the **Next point** ( Degree=3 Undo ) prompt, snap on the polyline guideline using the **Near** object snap until you get to the end.
  - 5 At the **Next point** ( Degree=3 Close Sharp=Yes Undo) prompt, press **Enter**.
- A free-form curve is drawn. The control points, while on the guideline, were not on the curve itself except at the two ends.
- Control points control the curvature of the curve, but the curve is usually not on them.



**To draw a curve interpolate through points:**

- 1 Change to the **Interpolated Curve** layer.
- 2 From the **Curve** menu, click **Free-form**, and then click **Interpolate Points**.
- 3 At the **Start of curve** ( Degree=3 Knots= *SqrtChord* StartTangent ) prompt, snap to the end point of the polyline guideline.
- 4 At the **Next point of curve** ( Degree=3 Knots= *SqrtChord* EndTangent Undo ) prompt, snap on the polyline guideline using the **Near** object snap.
- 5 At the **Next point of curve** ( Degree=3 Knots= *SqrtChord* EndTangent Undo ) prompt, continue to snap to the guideline until you get to the end.

**Notes:**



Curve

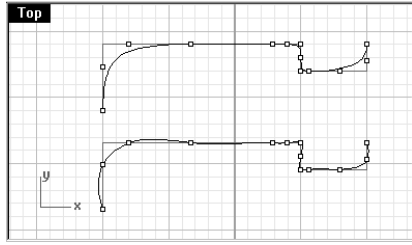


Interpolated Curve

- 6 At the **Next point of curve** ( Degree=3 Knots= *SqrtChord* EndTangent Close Sharp=Yes Undo ) prompt, press **Enter**.

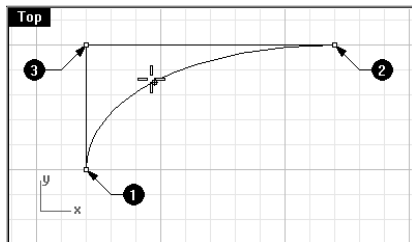
A free-form curve is created from specified interpolate points. These points lie on the curve and determine its curvature.

Note how difficult it is to make the curve follow the guideline exactly.



#### To draw a conic curve:

- 1 Change to the **Conic** layer.
- 2 From the **Curve** menu, click **Conic**.
- 3 At the **Start of conic** ( Tangent Perpendicular ) prompt, snap to the point (1) at the lower left.
- 4 At the **End of conic** ( Apex Tangent Perpendicular ) prompt, snap to the point (2) above and to the right from the previous point.
- 5 At the **Apex** prompt, snap to the point (3) in between the previous points.
- 6 At the **Curvature point or rho** prompt, pick a point for the desired curvature.



#### Notes:





Conic

## Notes:

## Modeling Helix and Spiral

The use of free-form 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
	Helix	Draw a helix. User can specify a radius, the number of turns, the length and direction of the axis.
	Spiral	Draw a spiral. User can specify two radii, the number of turns, the length and direction of the axis.

Option	Description
Vertical	The helix or spiral axis will be perpendicular to the construction plane in the active viewport.
AroundCurve	Select a curve for the helix or spiral to wind around to create "phone cord" shapes.
Flat	Lets you draw a planar spiral.
Mode	Determines whether the number of turns or the distance between turns is used to create the helix or spiral Turns    Allows you to set the number of turns along the axis. Pitch    Allows you to set the distance between turns along the axis.
ReverseTwist	Allows you to reverse the twist direction of the helix or spiral.

### To draw a helix

- 1 Change to the **Helix** layer.
- 2 Turn on the **End** and **Point** object snap.
- 3 From the **Curve** menu, click **Helix**.
- 4 At the **Start of axis** ( Vertical AroundCurve ) prompt, snap to the end of the vertical line (1) in the **Perspective** viewport..
- 5 At the **End of axis** prompt, snap to the end of the vertical line (2) in the **Perspective** viewport.

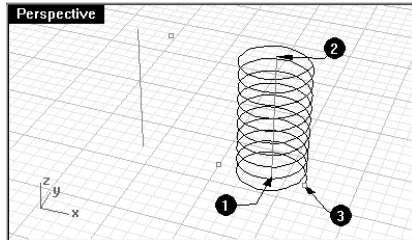


Helix



- At the **Radius and start point <1.000>** ( Diameter Mode=*Turns* Turns=*10* Pitch=*1* ReverseTwist=*No* ) prompt, pick the point (3) to the right of the axis line.

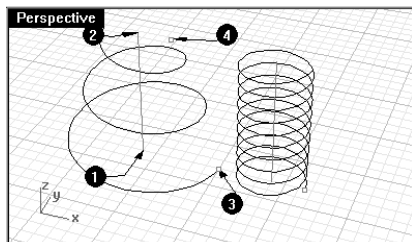
A helix with 10 turns and a radius of 2.5 is created.



### To draw a spiral:

- Change to the **Spiral** layer.
- From the **Curve** menu, click **Spiral**.
- At the **Start of axis** ( Flat Vertical AroundCurve ) prompt, snap to the end of the other vertical line (1) in the **Perspective** viewport.
- At the **End of axis** prompt, snap to the other end of the same line (2).
- At the **First radius and start point <1>** ( Diameter Mode=*Turns* Turns=*10* Pitch=*1* ReverseTwist=*No* ) prompt, click **Mode**.
- At the **First radius and start point <0.735>** ( Diameter Mode=*Pitch* Turns=*10* Pitch=*1* ReverseTwist=*No* ) prompt, click **Pitch**
- At the **Pitch <1>** prompt, type **4** and press **Enter**.
- At the **First radius and start point <1>** ( Diameter Mode=*Pitch* Turns=*10* Pitch=*4* ReverseTwist=*No* ) prompt, click **ReverseTwist**.
- At the **First radius and start point <1>** ( Diameter Mode=*Pitch* Turns=*10* Pitch=*4* ReverseTwist=*Yes* ) prompt, snap the point (3) for the radius of the base of the spiral.
- At the **Second radius <0>** ( Diameter Mode=*Pitch* Turns=*10* Pitch=*4* ReverseTwist=*Yes* ) prompt, snap to the other point (4) for the ending radius.

A spiral is created with a reverse twist and a pitch of 4.



### Notes:

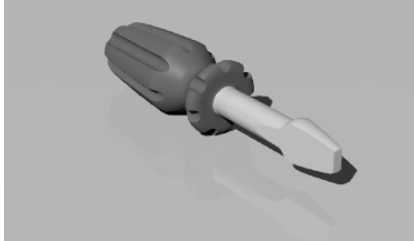


Spiral

## Notes:

### Exercise 27—Drawing free-form curves

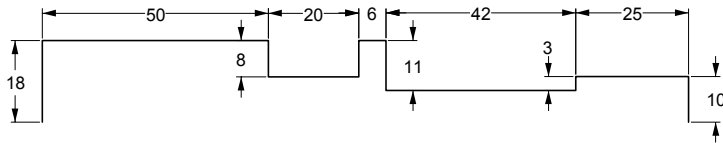
In the following exercise we will draw the guidelines and free-form curves for a toy screwdriver.



- 1 Start a new model. Save as **Screwdriver**.
- 2 Create **Construction** and **Curve** layers.  
Make them different colors.

#### To create the construction lines:

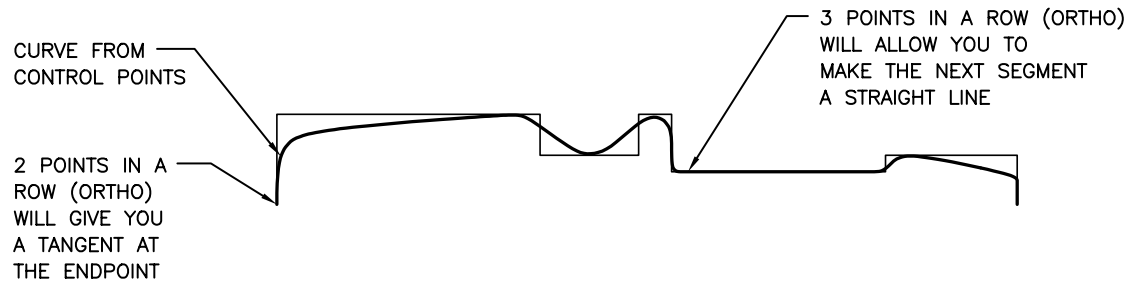
- 1 Change to the **Construction** layer.



- 2 Draw a polyline using the dimensions above for the guideline.

#### To create the curve by control points:

- 1 Change to the **Curve** layer.
- 2 Use the **Curve** command to draw the shape for the toy screwdriver.



- 3 **Save** your model.

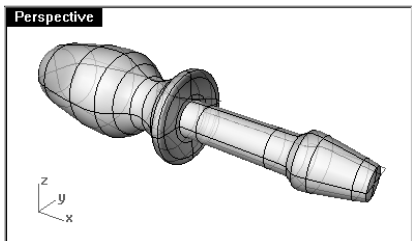
### To make it solid:

- 1 Toggle **Snap** and **Ortho** on.
- 2 From the **Surface** menu and click **Revolve**.
- 3 At the **Select curve to revolve** prompt, select the curve.



- 4 At the **Start of revolve axis** prompt, snap to the end of the curve.
- 5 At the **End of revolve axis** prompt, snap to the other end of the curve.
- 6 In the **Revolve Options** dialog box, click **OK**.

Your model is now a three-dimensional surface.



### Notes:



Revolve



# 5

## 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 touch 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 does not create a fillet.
Join	<b>Yes</b> joins the fillet to the curves. <b>No</b> does not join the fillet to the curves.
Trim	<b>Yes</b> trims the curves to the fillet arc. <b>No</b> does not trim the curves.

### Exercise 28—Fillet

► **Open** the model **Fillet.3dm**.

#### 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 Trim=Yes ) prompt, click **Radius** to change the radius.
- 3 At the **Fillet radius <1>** prompt, type **0**, and press **Enter**.  
This Radius option remains the default until you change it.
- 4 At the **Select first curve to fillet** ( Radius=0 Join=No Trim=Yes ) prompt, select an outer vertical line.

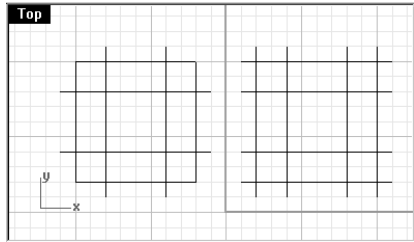
### Notes:



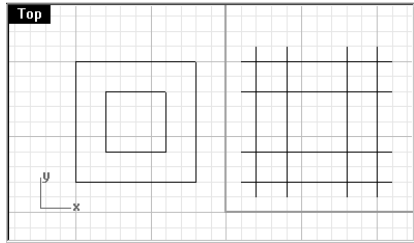
Fillet

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

The ends of the lines are trimmed to a corner.



- Press **Enter** to repeat the command.
- Fillet** the other corners, as shown.



#### To fillet lines using an arc:

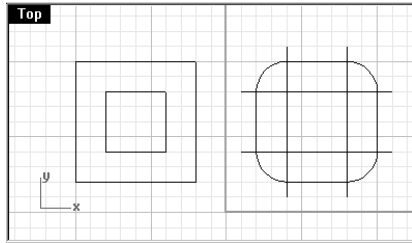
- From the **Curve** menu, click **Fillet**.
- At the **Select first curve to fillet** ( Radius=0 Join=No Trim=Yes ) prompt, type **2** and press **Enter** to change the radius.  
This is another method to change the radius.
- At the **Select first curve to fillet** ( Radius=2 Join=No Trim=Yes ) prompt, click **Join** to change the Join option to Yes.  
This option joins the curves as they are filleted.
- At the **Select first curve to fillet** ( Radius=2 Join=Yes Trim=Yes ) prompt, select an outer vertical line.
- At the **Select second curve to fillet** ( Radius=2 Join=Yes Trim=Yes ) prompt, select an adjacent horizontal line.  
The ends of the lines are trimmed to a radius.
- Press **Enter** to repeat the command.

#### Notes:

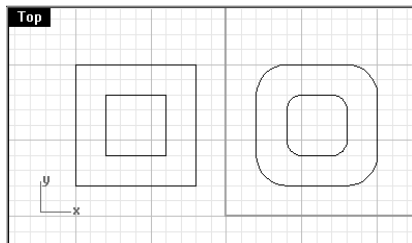
*Remember to pick on the part of the line that you want to keep.*

## Notes:

- 7 Fillet the other corners as shown.



- 8 Press **Enter** to repeat the command.
- 9 At the **Select first curve to fillet** ( Radius=2 Join=Yes Trim=Yes ) prompt, type **1** and press **Enter**.  
This radius will be used for the smaller object.
- 10 At the **Select first curve to fillet** ( Radius=1 Join=Yes Trim=Yes ) prompt, select an outer vertical line.
- 11 At the **Select second curve to fillet** ( Radius=1 Join=Yes Trim=Yes ) prompt, select an adjacent horizontal line.
- 12 **Fillet** the other corners as shown.

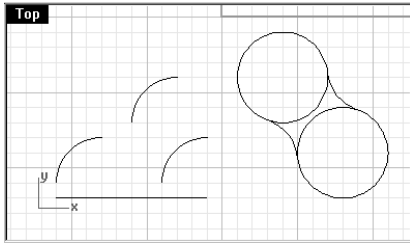


### To fillet circles:

- 1 From the **Curve** menu, click **Fillet**.
- 2 At the **Select first curve to fillet** ( Radius=1 Join=Yes Trim=Yes ) prompt, type **3** and press **Enter**.
- 3 At the **Select first curve to fillet** ( Radius=3 Join=Yes Trim=Yes ) prompt, click **Trim**.  
This also disables the Join option.
- 4 At the **Select first curve to fillet** ( Radius=3 Trim=No ) prompt, select the right edge of a circle.
- 5 At the **Select second curve to fillet** ( Radius=3 Trim=No ) prompt, select the right edge of the other circle.

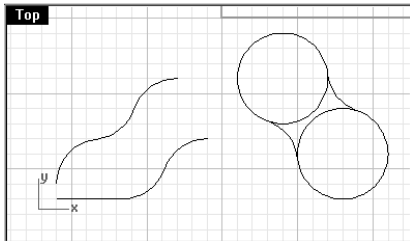
## Notes:

- 6 Repeat the command for the left side of the circles.



### To fillet and join arcs and lines:

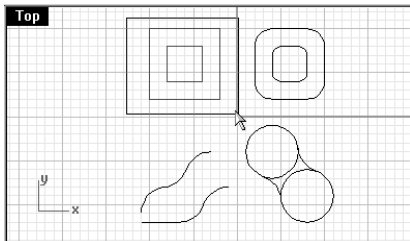
- 1 From the **Curve** menu, click **Fillet**.
- 2 At the **Select first curve to fillet** ( Radius=3 Trim=No ) prompt, click **Trim**.
- 3 At the **Select first curve to fillet** ( Radius=3 Join=Yes Trim=Yes ) prompt, select one of the arcs in the middle of the viewport.
- 4 At the **Select second curve to fillet** ( Radius=3 Join=Yes Trim=Yes ) prompt, select the other arc in the middle of the viewport.



- 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.



Join



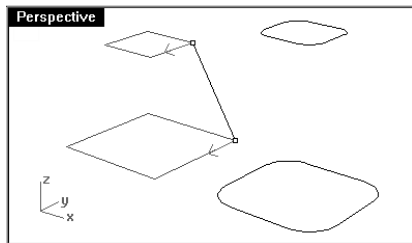
- From the **Edit** menu, click **Join**.

The objects are joined together. Curves join only if they touch.

#### To make a lofted surface:

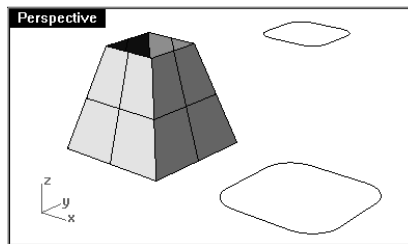
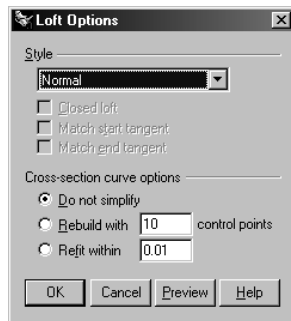
- Change to the **Surfaces** layer.
- Select the two squares in the upper left part of the **Top** viewport.
- Change to the **Perspective** viewport.
- From the **Surface** menu, click **Loft**.

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



- At the **Select seam point to adjust, press Enter when done ( FlipDirection Automatic Natural )** prompt, press **Enter**.
- In the **Loft Options** dialog box, click **OK**.

A surface is generated between the two closed polylines.



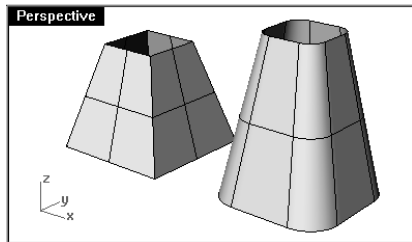
#### Notes:

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



Loft

- 7 Repeat the procedure for the rounded squares.
- 8 In the **Loft Options** dialog box, click **OK**.



- 9 Save your model.

## 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 curves.

Option	Description
Distances	<p>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.</p> <p>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 that far from the intersection, and the curve is extended to the chamfer line. If you enter 0 for both distances, the curves are trimmed or extended to their intersection, but no chamfer line is created.</p>

### Exercise 29—Chamfer

- ▶ **Open** the model **Chamfer.3dm**.

#### To chamfer lines:

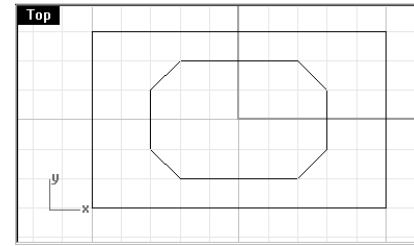
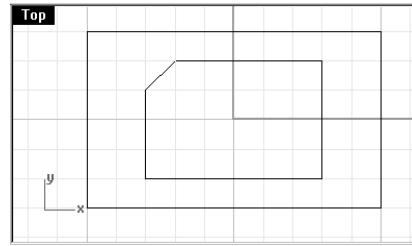
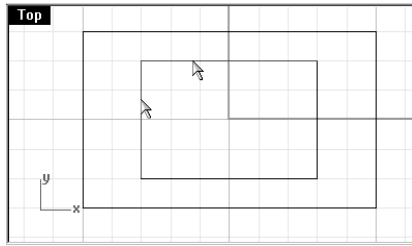
- 1 From the **Curve** menu, click **Chamfer**.
- 2 At the **Select first curve to chamfer** ( Distances=*1,1* Join=*No* Trim=*Yes* ) prompt, type **1,1**, and press **Enter**, to set the distances.
- 3 At the **Select First Curve to chamfer** ( Distances=*1,1* Join=*No* Trim=*Yes* ) prompt, select one of the inner vertical lines.

## Notes:



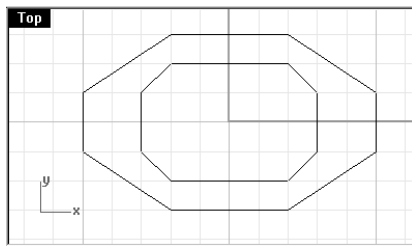
Chamfer

- At the **Select Second Curve to chamfer** ( Distances=1,1 Join=No Trim=Yes ) prompt, select an adjacent horizontal line.



- Continue creating chamfers on all of the corners as shown.
- Press **Enter** to repeat the command.
- At the **Select First Curve to chamfer** ( Distances=1,1 Join=No Trim=Yes ) prompt, type **3,2** and press **Enter**.
- At the **Select First Curve to chamfer** ( Distances=3,2 Join=No Trim=Yes ) prompt, select one of the outer horizontal lines.
- At the **Select Second Curve to chamfer** ( Distances=3,2 Join=No Trim=Yes ) prompt, select an adjacent vertical line.

The first value is the distance along the first curve selected, the second value is the distance along the second line selected.



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

#### To join the chamfered objects:

- From the **Edit** menu, click **Select**, and then click **Curves**.
- From the **Edit** menu, click **Join**.

The curves have been joined together into closed polylines.

#### To turn the curves into surfaces:

- Change to the **Surfaces** layer.

#### Notes:

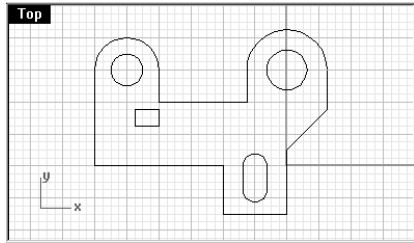


Select Curves

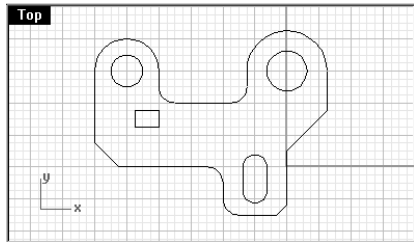


## Notes:

- 1 **Open** the model **Filletex.3dm**.

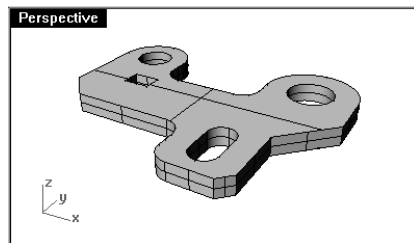


- 2 Use **Fillet** and **Chamfer** to edit the drawing as shown.  
All fillets and rounds use a radius of 0.5 units.



### To make it solid:

- 1 From the **Edit** menu, click **Select**, and then click **Curves**.
- 2 From the **Solid** menu, click **Extrude Planar Curves**.
- 3 At the **Extrusion distance** ( Direction Cap=Yes BothSides Tapered ) prompt, type **.5** and press **Enter**.



## Notes:

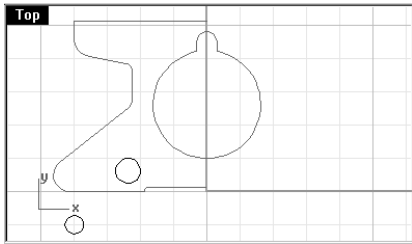
## Move

Use Move to move objects without changing orientation or size.

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

### Exercise 31—Move

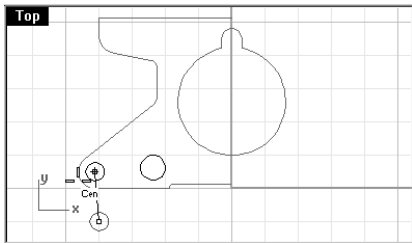
- 1 **Open** the model **Move.3dm**.



- 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** viewport.
- 2 From the **Transform** menu, click **Move**.
- 3 At the **Point to move from** ( Vertical ) prompt, snap to the center of the small circle.
- 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:

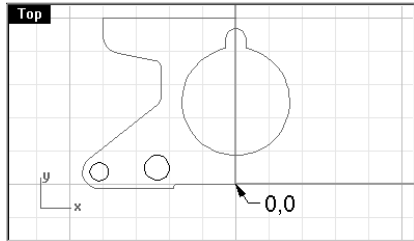
- 1 From the **Edit** menu, click **Select**, and then click **Curves**.
- 2 From the **Transform** menu, click **Move**.



Move

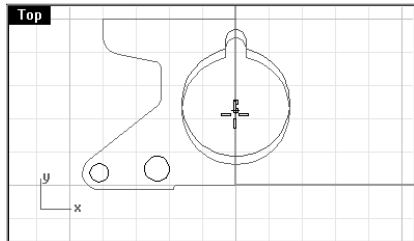
## Notes:

- 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** viewport.



### To move using relative coordinates:

- 1 Select the large slotted circle in the middle of the object.  
You 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** viewport.  
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.



## Notes:

### Copy

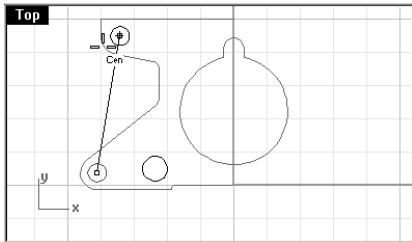
Copy duplicates selected objects and places them in a new location. The command can repeat to create more than one copy in the same command sequence.

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

### Exercise 32—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 Pick to place the object, and press **Enter**.

#### 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.

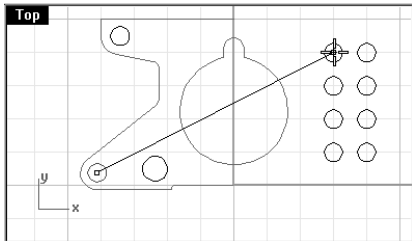


Copy



## Notes:



- 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.



- Press **Enter** to end the command.

## Undo and Redo

If you make a mistake or do not like the results of a command, use **Undo**. If you decide you want those undone results back after all, use **Redo** command. Redo restores the last thing undone.

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

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

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

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

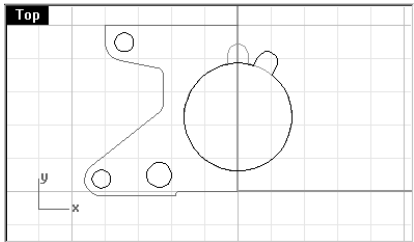
## Rotate

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 33—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=No ) prompt, snap to the center of the slotted circle.



- 4 At the **Angle or first reference point** ( Copy=No ) prompt, type **-28** and press **Enter**.

## Notes:








Rotate

## Notes:

## Group

Grouping objects allows all members of the group to be selected as one. You can then apply commands to the entire group.

Rhino Button	Mouse Button	Command	Description
	Left-click or <b>Ctrl+G</b>	Group	Makes a group from selected objects.
	Left-click or <b>Ctrl+Shift+G</b>	Ungroup	Removes grouping.
	Left-click	AddToGroup	To add objects to a group.
	Left-click	RemoveFromGroup	To remove objects from a group.
	Left-click	SetGroupName	To name groups

### Exercise 34—Grouping

#### To group selected objects:

- 1 Select the two circles that you placed.
- 2 From the **Edit** menu, click **Groups**, then click **Group**.

#### To add objects to a group:

- 1 At the command prompt type **AddT**.  
When the **AddToGroup** command completes, press **Enter**.
- 2 At the **Select objects** prompt, select the polyline on the left, the original circle, and the slotted circle in the center.
- 3 At the **Select objects** prompt, press **Enter**.
- 4 At the **Select group** prompt, select one of the circles in the group.  
The objects are now part of the group.

## Notes:

### To remove an object from a group:

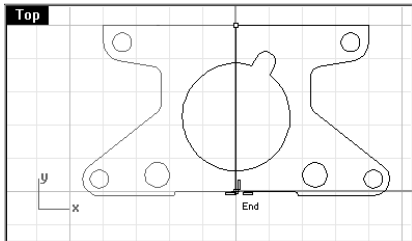
- 1 At the command prompt, type **RemoveFr** and press the **Tab** key, then press **Enter**.  
When the **RemoveFromGroup** command completes, press **Enter**.
- 2 At the **Select objects to extract from group** prompt, select the slotted circle.
- 3 At the **Select objects to extract from group** prompt, press **Enter**.

## Mirror

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

### Exercise 35—Mirror

- 1 Select the group.
- 2 From the **Transform** menu, click **Mirror**.
- 3 At the **Start of mirror plane** ( Copy=Yes ) prompt, type **0,0** or snap to the end of the line at the lower right of the part.
- 4 At the **End of mirror plane** ( Copy=Yes ) prompt, toggle **Ortho** on and pick directly above the previous point.



- 5 Select the two groups.
- 6 From the **Edit** menu, click **Groups**, then click **Ungroup**.



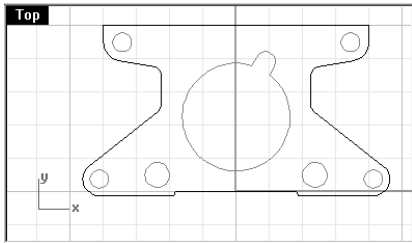
Mirror

## Join

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

### Exercise 36—Join

- 1 Select the two polylines.
- 2 From the **Edit** menu, click **Join**.



## Scale

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

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.

### Exercise 37—Scale

- 1 From the **Edit** menu, click **Select**, and then click **Curves**.
- 2 From the **Transform** menu, click **Scale**, and then click **Scale 2-D**.
- 3 At the **Origin point** ( Copy=No ) prompt, type **0,0** and press **Enter**.

## Notes:



Join

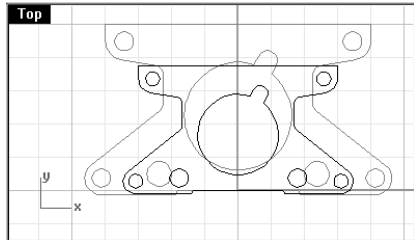


Scale2D  
Right-click for Scale2D

## Notes:

- 4 At the **Scale factor or first reference point <1>** ( Copy=No ) prompt, type **.75** and press **Enter**.

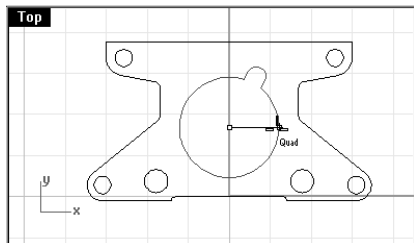
The whole part has been scaled to 75 percent of its original size.



### To scale using the reference point option:

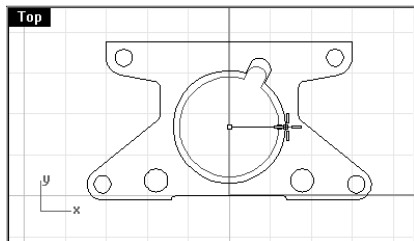
- 1 From the **Edit** menu, click **Select**, and then click **Curves**.
- 2 From the **Transform** menu, click **Scale**, and then click **Scale 2-D**.
- 3 At the **Origin point** ( Copy=No ) prompt, snap to the center of the slotted circle.
- 4 At the **Scale factor or first reference point <.75>** ( Copy=No ) 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=No ) prompt, type **1.375** and press **Enter**.

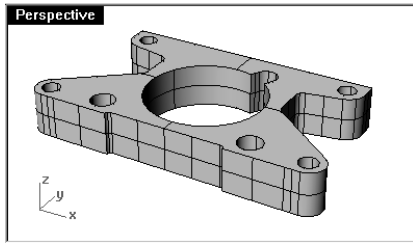
The slotted circle now has a larger radius of 1.375. Everything else was scaled proportionally.



## Notes:

### To make it solid:

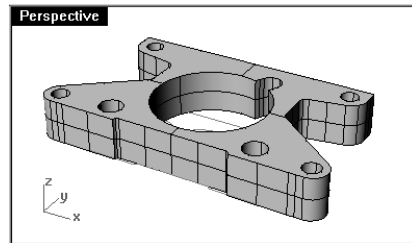
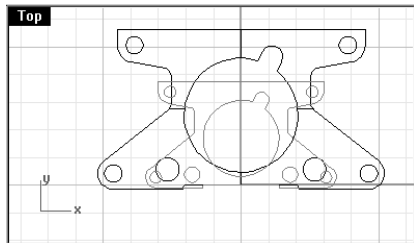
- 1 From the **Edit** menu, click **Select**, and then click **Curves**.
- 2 From the **Solid** menu, click **Extrude Planar Curves**.
- 3 At the **Extrusion distance** ( Direction Cap=Yes BothSides Tapered ) prompt, type **1** and press **Enter**.



### To scale in 3-D:

- 1 From the **Edit** menu, click **Select**, and then click **Polysurfaces**.
- 2 From the **Transform** menu, click **Scale**, and then click **Scale**.
- 3 At the **Origin point** ( Copy=No ) prompt, type **0,0** and press **Enter**.
- 4 At the **Scale factor or first reference point** <1> ( Copy=No ) prompt, type **1.5** and press **Enter**.

The solid is larger in every dimension.



### To scale in one dimension:

- 1 From the **Edit** menu, click **Select**, and then click **Polysurfaces**.
- 2 From the **Transform** menu, click **Scale**, and then click **Scale 1-D**.
- 3 At the **Origin point** ( Copy=No ) prompt, type **0,0** and press **Enter**.
- 4 At the **Scale factor or first reference point** <1.500> ( Copy=No ) prompt, type **.5** and press **Enter**.



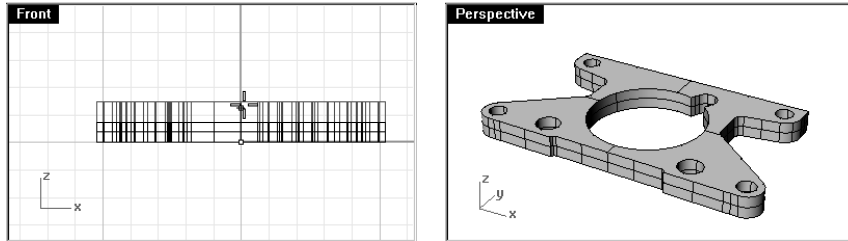
Scale



Scale1D

- At the **Scaling direction** ( Copy=No ) prompt, move your cursor into the **Front** viewport, toggle **Ortho** on, and pick a point above the previous point.

The object is half as thick.



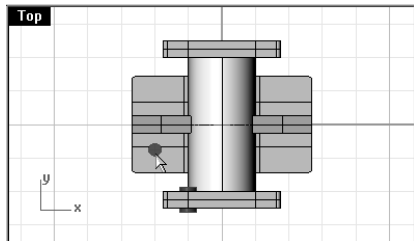
## Array

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:

- Open** the model **T-Flange.3dm**.
- Select the small cylinder (1) that is visible in the **Top** viewport.



- From the **Transform** menu, click **Array**, and then click **Rectangular**.
- At the **Number in x direction <1>** prompt, type **2** and press **Enter**.
- At the **Number in y direction <1>** prompt, type **2** and press **Enter**.
- At the **Number in z direction <1>** prompt, type **1** and press **Enter**.
- At the **Unit cell or x distance** prompt, type **4** and press **Enter**.

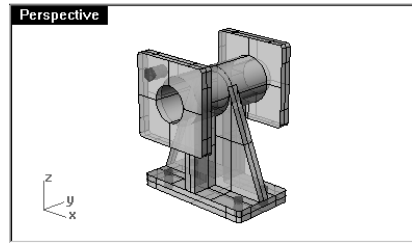
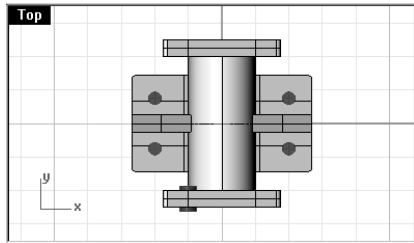
## Notes:



Array Rectangular

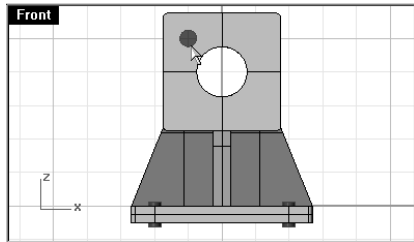


- 8 At the **Y distance or first reference point** prompt, type **1.5** and press **Enter**.  
The cylinders are arrayed on the base of the flange.



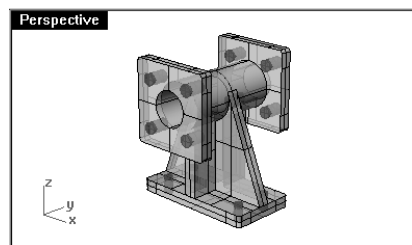
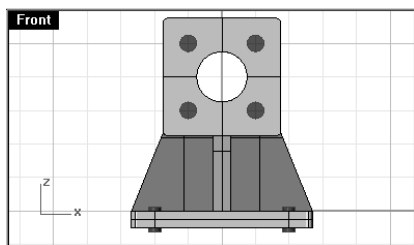
### To create a 3-D array:

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



- 2 From the **Transform** menu, click **Array**, and 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 distance** prompt, type **2** and press **Enter**.  
7 At the **Y distance or first reference point** prompt, type **-2** and press **Enter**.  
8 At the **Z distance or reference first point** prompt, type **-4.5** and press **Enter**.

The cylinders are arrayed on both flanges.



### Notes:



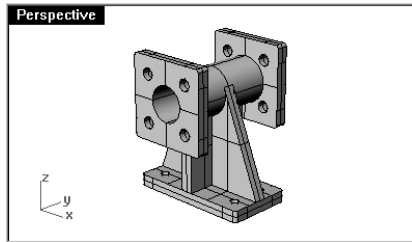
Array Rectangular

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

### To make holes in the flange:

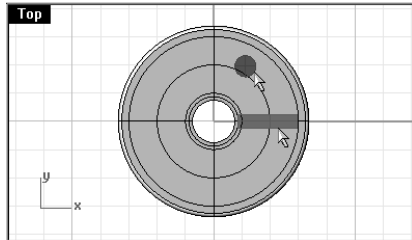
- 1 Save your model.
- 2 From the **Solid** menu, click **Difference**.
- 3 At the **Select first set of surfaces or polysurfaces. ...** prompt, pick the flange (1).
- 4 At the **Select first set of surfaces or polysurfaces. ...** prompt, press **Enter**.
- 5 At the **Select second set of surfaces or polysurfaces. Press Enter when done.** ( DeleteInput=Yes ) prompt, pick all of the cylinders (2) for the holes and press **Enter**.

The cylinders are subtracted from the flange.



### To create a polar array:

- 1 **Open** the model **Wheel.3dm**.
- 2 Select the cylinder and the box in the **Top** viewport.



- 3 From the **Transform** menu, click **Array**, and then click **Polar**.
- 4 At the **Center of polar array** prompt, snap to the center of the wheel.
- 5 At the **Number of items <2>** prompt, type **3** and press **Enter**.

### Notes:



Boolean Difference

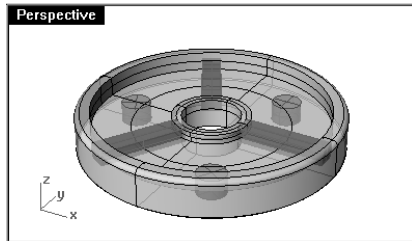


Array Polar

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

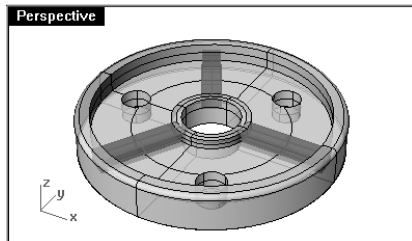
- 6 At the **Angle to fill <360>** prompt, press **Enter**.

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



#### To remove the holes from the wheel:

- 1 Save your model.
  - 2 From the **Solid** menu, click **Difference**.
  - 3 At the **Select first set surfaces or polysurfaces** prompt, pick the wheel.
  - 4 At the **Select first set of surfaces or polysurfaces. Press Enter for second set** prompt, press **Enter**.
  - 5 At the **Select second set of surfaces or polysurfaces. Press Enter when done.** ( DeleteInput=Yes ) prompt, pick all of the cylinders for the holes and press **Enter**.
- The cylinders are subtracted from the flange.



#### To add the ribs to the wheel:

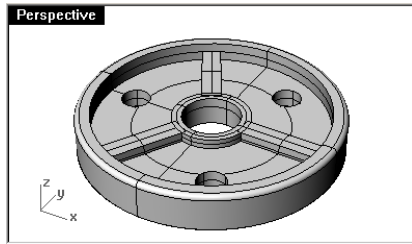
- 1 From the **Solid** menu, click **Union**.
- 2 At the **Select surfaces or polysurfaces to union** prompt, pick the wheel.

#### Notes:



Boolean Union

- 3 At the **Select surfaces or polysurfaces to union. Press Enter when done** prompt, pick each of the ribs and press **Enter**.

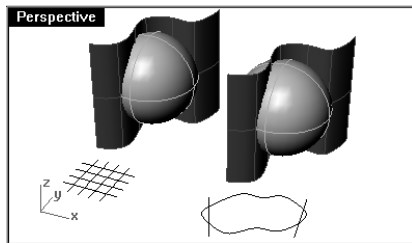


## Trim

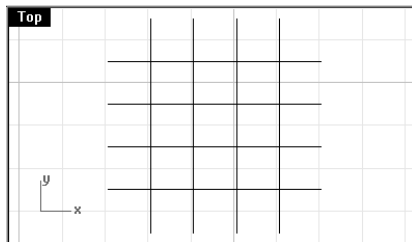
Trim cuts and deletes portions of an object to make it end precisely at its intersection with another object.

### Exercise 39—Trim

- 1 **Open** the model **Trim-Split.3dm**.



- 2 From the **View** menu, click **Zoom**, and then click **Window**.
- 3 Make a window around the grid in the lower left corner of the **Top** viewport.

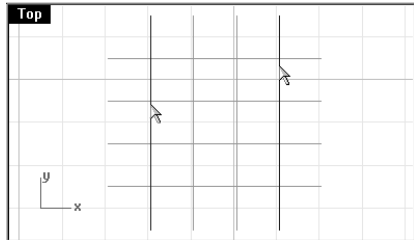


## Notes:



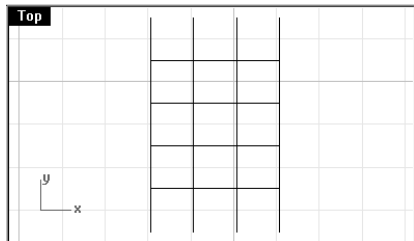
Zoom Window

- 4 From the **Edit** menu, click **Trim**.
- 5 At the **Select cutting objects** ( *ExtendLines=Off UseApparentIntersections=Off* ) prompt, select the two outside vertical lines in the grid.



- 6 At the **Select cutting objects** ( *ExtendLines=Off UseApparentIntersections=Off* ) prompt, press **Enter** to go to the next stage of the command.
- 7 At the **Select object to trim. Press Enter to clear selection and start over** ( *ExtendLines=Off UseApparentIntersections=Off* ) prompt, select each of the horizontal lines at their left and right ends.

The lines are trimmed to the cutting edges.



- 8 At the **Select object to trim. Press Enter to clear selection and start over** ( *ExtendLines=Off UseApparentIntersections=Off* ) prompt, press **Enter**.
- 9 From the **View** menu, click **Zoom**, and then click **Extents All**.
- 10 From the **Edit** menu, click **Trim**.
- 11 At the **Select cutting objects** ( *ExtendLines=Off UseApparentIntersections=Off* ) prompt, select the surface that intersects the sphere in the **Perspective** viewport.

## Notes:



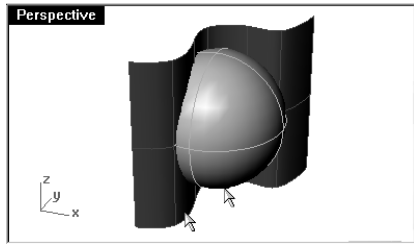
Trim



Zoom Extents All

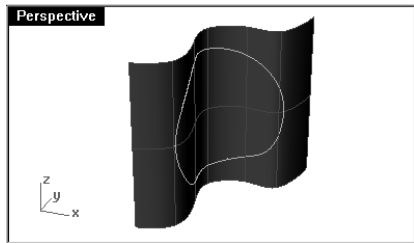
## Notes:

- 12 At the **Select cutting objects** ( *ExtendLines=Off UseApparentIntersections=Off* ) prompt, press **Enter**.



- 13 At the **Select object to trim. ...** ( *ExtendLines=Off UseApparentIntersections=Off* ) prompt, select the right side of the sphere.

The sphere is trimmed at the surface.



- 14 At the **Select object to trim. ...** ( *ExtendLines=Off UseApparentIntersections=Off* ) prompt, press **Enter**.

## Split

Splits one object with another into two objects splits a curve at a point you specify. The Split command breaks the object where it intersects the cutting object but does not delete anything.

### Exercise 40—Split

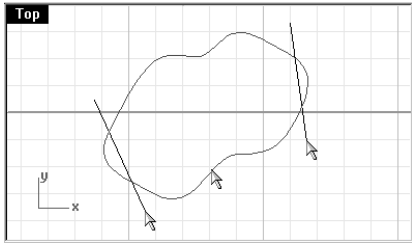
- 1 From the **View** menu, click **Zoom**, and then click **Window**.
- 2 Make a window around the closed curve in the lower right corner of the **Top** viewport.
- 3 From the **Edit** menu, click **Split**.
- 4 At the **Select objects to split** prompt, select the closed curve.
- 5 At the **Select objects to split** prompt, press **Enter** to go to the next stage of the command.



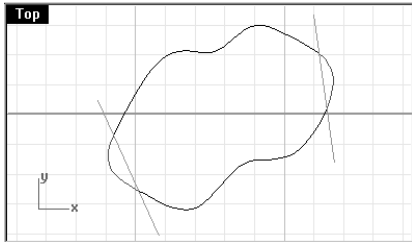
Split

## Notes:

- 6 At the **Select cutting objects** ( Point ) prompt, select the lines and press **Enter**.

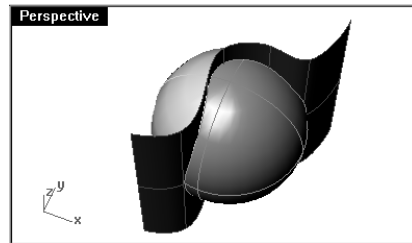
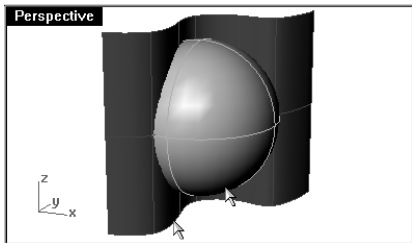


- 7 At the **Select cutting objects** ( Point ) prompt, press **Enter**.  
The curve is separated into four curves precisely where the lines intersect it.



- 8 From the **View** menu, click **Zoom**, and then click **Extents All**.  
9 From the **Edit** menu, click **Split**.  
10 At the **Select objects to split** prompt, select the sphere.  
11 At the **Select cutting objects** ( Isocurve ) prompt, select the surface that intersects the sphere and press **Enter**.

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

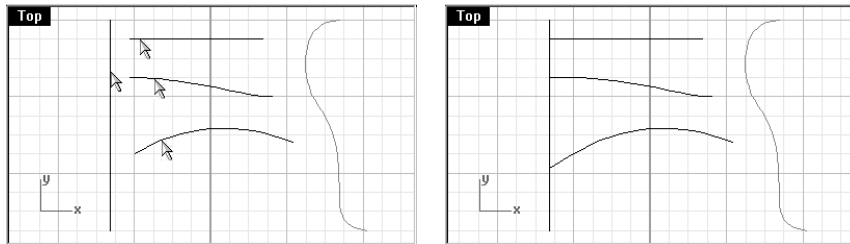


## Extend

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 41—Extend

- 1 **Open** the model **Extend**.
- 2 From the **Curve** menu, click **Extend**, and then click **Extend Curve**.
- 3 At the **Select boundary objects or enter extension length. Press Enter for dynamic extend** prompt, select the line at the left.
- 4 At the **Select boundary objects or enter extension length. Press Enter for dynamic extend** prompt, press **Enter**.
- 5 At the **Select curve to extend ( Type=Line )** prompt, select the left ends of the three curves.  
The line and curves extend to touch the boundary edge on the left.



- 6 At the **Select curve to extend ( Type=Line )** prompt, press **Enter** to end the command.

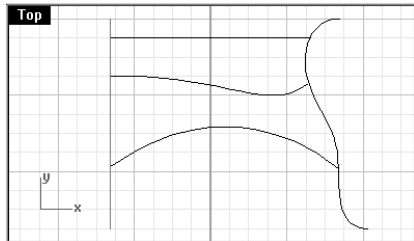
### To extend with some of the other options:

- 1 From the **Curve** menu, click **Extend**, and then click **Extend Curve**.
- 2 At the **Select boundary objects or enter extension length. Press Enter for dynamic extend** prompt, select the curve at the right.
- 3 At the **Select boundary objects or enter extension length. Press Enter for dynamic extend** prompt, press **Enter**.
- 4 At the **Select curve to extend ( Type=Line )** prompt, select the right end of the line.  
The line is extended straight.
- 5 At the **Select curve to extend ( Type=Line )** prompt, type **T** and press **Enter**.
- 6 At the **Type <Line> ( Line Arc Smooth )** prompt, type **A** and press **Enter**.
- 7 At the **Select curve to extend ( Type=Arc )** prompt, select the right end of the curve.  
The curve continues with a tangent arc to the boundary edge.



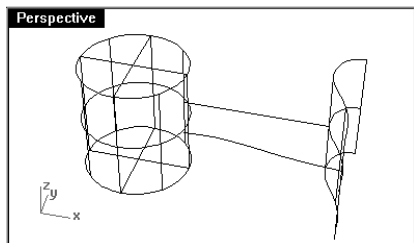
## Notes:

- 8 At the **Select curve to extend** ( Type=*Arc* ) prompt, type **S** and press enter.
- 9 At the **Type <Arc>** ( Line Arc Smooth ) prompt, type **S** and press **Enter**.
- 10 At the **Select curve to extend** ( Type=*Smooth* ) prompt, select the right end of the arc.  
The arc continues with a tangent curve.



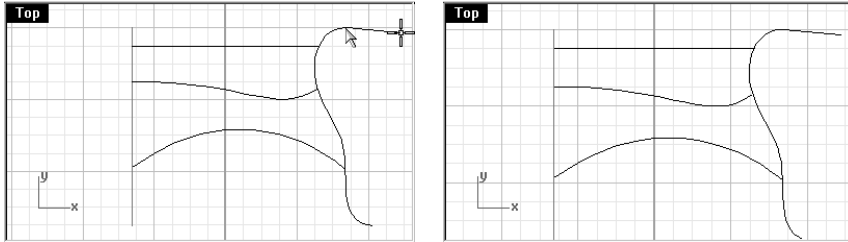
### To extend to a surface:

- 1 From the **Curve** menu, click **Extend**, and then click **Extend Curve**.
- 2 At the **Select boundary objects or enter extension length. ...** prompt, select the cylinder on the left.
- 3 At the **Select boundary objects or enter extension length. ...** prompt, select the surface on the right.
- 4 At the **Select boundary objects or enter extension length. ...** prompt, press **Enter**.
- 5 At the **Select curve to extend** ( Type=*Smooth* ) prompt, type **T** and press **Enter**.
- 6 At the **Type <Smooth>** ( Line Arc Smooth ) prompt, type **L** and press **Enter**.
- 7 At the **Select curve to extend** ( Type=*Line* ) prompt, select both ends of the line and the curve.  
The curves extend to the surface of the sphere and to the surface.

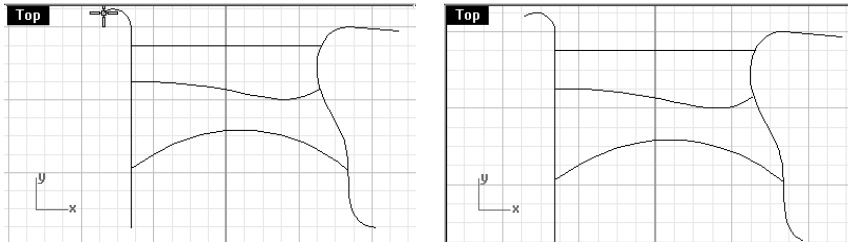


### To extend a curve without boundary edges:

- 1 From the **Curve** menu, click **Extend**, and then click **By Line**.
- 2 At the **Select curve to extend** ( Type=*Line* ) prompt, select near the curve in the upper right side.
- 3 At the **End of extension** prompt, pick a point.  
The curve is extended as a tangent line to the point you picked.



- 4 From the **Curve** menu, click **Extend**, and then click **By Arc**.
- 5 At the **Select curve to extend** ( Type=*Arc* ) prompt, select near the upper end of the line on the left.
- 6 At the **Center of arc extension** prompt, type **1** and press **Enter** to set the radius.
- 7 At the **Center of arc extension** prompt, pick a point to the left of the line.
- 8 At the **End of extension** prompt, pick an end for the arc.



### Notes:



Extend by Line



Extend by Arc

## Notes:

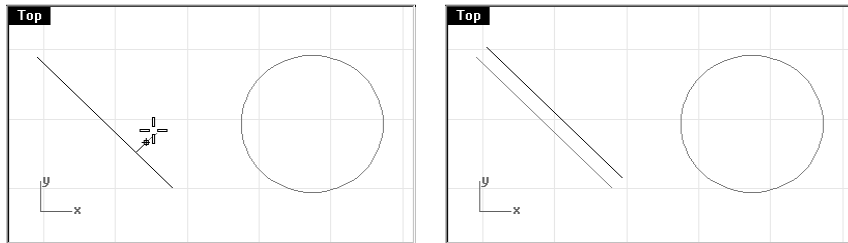
### Offset

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 42—Offset

- 1 **Open** the model **Offset.3dm**.
- 2 Maximize the **Top** viewport.
- 3 From the **Curve** menu, click **Offset**.
- 4 At the **Select curve to offset** ( Distance=*1* Corner=*Sharp* ThroughPoint Tolerance ) prompt, select the line.
- 5 At the **Through point** ( Distance=*1* Corner=*Sharp* ThroughPoint Tolerance ) prompt, pick on the upper right side of the line.

A parallel line is created.



#### To offset with the through point option:

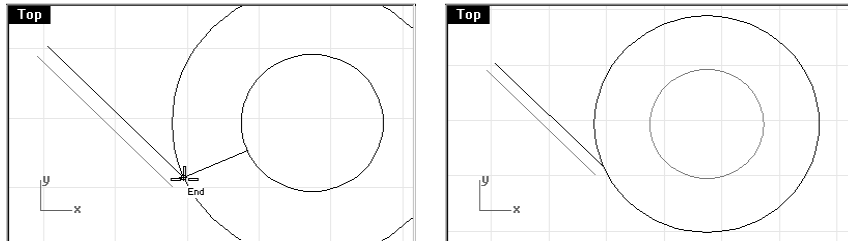
- 1 From the **Curve** menu, click **Offset**.
- 2 At the **Select curve to offset** ( Distance=*1* Corner=*Sharp* ThroughPoint Tolerance ) prompt, select the circle.
- 3 At the **Through point** ( Distance=*1* Corner=*Sharp* ThroughPoint Tolerance ) prompt, type **T** and press **Enter**.



Offset

## Notes:

- 4 At the **Side to offset** ( Corner=*Sharp* Tolerance ) prompt, snap to the end of the line you offset.  
A concentric circle is created that goes through the end point of the line.

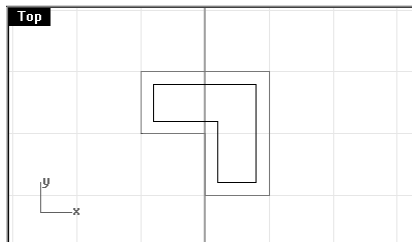


### Offset Using Corner Options

#### Sharp

- 1 From the **Curve** menu, click **Offset**.
- 2 At the **Select curve to offset** ( Distance= $4.82974$  Corner=*Sharp* ThroughPoint Tolerance ) prompt, select the polyline.
- 3 At the **Through point** ( Distance= $4.82974$  Corner=*Sharp* ThroughPoint Tolerance ) prompt, type **1** and press **Enter**.
- 4 At the **Through point** ( Distance= $1$  Corner=*Sharp* ThroughPoint Tolerance ) prompt, pick inside the polyline.

The polyline is offset with sharp corners.



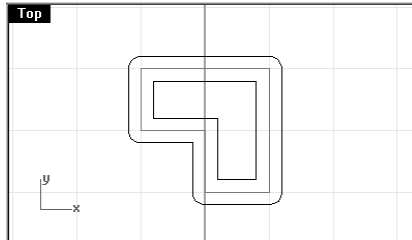
#### Round

- 1 Press **Enter** to repeat the command.
- 2 At the **Select curve to offset** ( Distance= $1$  Corner=*Sharp* ThroughPoint Tolerance ) prompt, select the polyline.
- 3 At the **Through point** ( Distance= $1$  Corner=*Sharp* ThroughPoint Tolerance ) prompt, type **C** and press **Enter**.
- 4 At the **Corner <Sharp>** ( Sharp Round Smooth Chamfer ) prompt, type **R** and press **Enter**.

## Notes:

- 5 At the **Side to offset** ( Distance=1 Corner=Round ThroughPoint Tolerance ) prompt, pick outside the polyline.

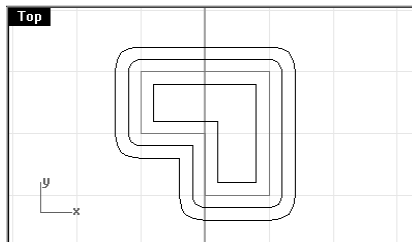
The polyline is offset, but the corners are rounded with arcs.



### Smooth

- 1 Press **Enter** to repeat the command.
- 2 At the **Select curve to offset** ( Distance=1 Corner=Round ThroughPoint Tolerance ) prompt, select the polyline.
- 3 At the **Through point** ( Distance=1 Corner=Round ThroughPoint Tolerance ) prompt, type **2** and press **Enter**.
- 4 At the **Through point** ( Distance=2 Corner=Round ThroughPoint Tolerance ) prompt, type **C** and press **Enter**.
- 5 At the **Corner <Round>** ( Sharp Round Smooth Chamfer ) prompt, choose the **Smooth** option, and press **Enter**.
- 6 At the **Through point** ( Distance=2 Corner=Smooth ThroughPoint Tolerance ) prompt, pick the outside of the polyline.

The polyline is offset, but the corners are rounded with curvature continuous blend curves.

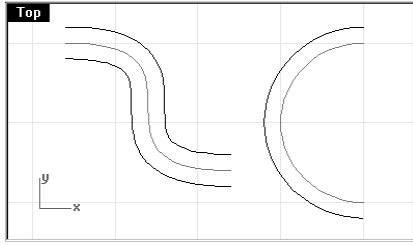


### Distance

- 1 At the **Select curve to offset** ( Distance=2 Corner=Smooth ThroughPoint Tolerance ) prompt, select the free-form curve.
- 2 At the **Through point** ( Distance=2 Corner=Smooth ThroughPoint Tolerance ) prompt, type **1** and press **Enter**.

## Notes:

- 3 At the **Through point** ( Distance=*1* Corner=*Smooth* ThroughPoint Tolerance ) prompt, pick on the one side of the curve.
- 4 Press **Enter** to repeat the command.
- 5 At the **Through point** ( Distance=*1* Corner=*Smooth* ThroughPoint Tolerance ) prompt, pick on the other side of the curve.



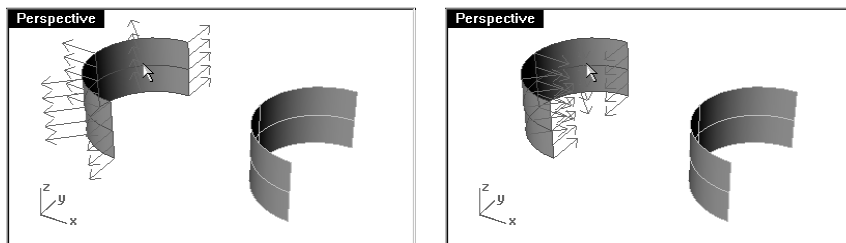
- 6 Select the arc.
- 7 Press **Enter** to repeat the offset command.
- 8 At the **Through point** ( Distance=*1* Corner=*Smooth* ThroughPoint Tolerance ) prompt, pick on the left side of the arc.

### To offset a surface:

- 1 From the **Surface** menu, click **Offset**.
- 2 At the **Select surface or polysurface to offset** prompt, select one of the purple surfaces, and press **Enter**.
- 3 At the **Offset distance <1>** ( Flip Solid Loose Tolerance ) prompt, place your cursor over the surface and click your left mouse button.

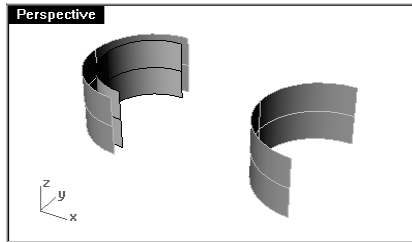
Notice that the direction of the arrows changed. The normal direction of the surface was flipped by the mouse click. If you have a positive number for offset distance, the surface will be offset in the direction of the normal. If you have a negative number it will offset the opposite direction

The normals should be pointed toward the concave side of the surface.



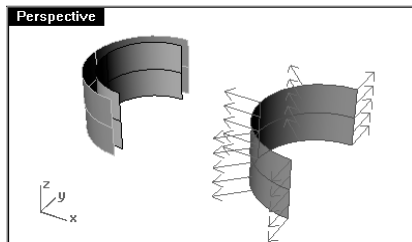
## Notes:

- 4 At the **Offset distance <1>** ( Flip Solid Loose Tolerance ) prompt, press **Enter**.  
The surface is offset in the direction of the normals.

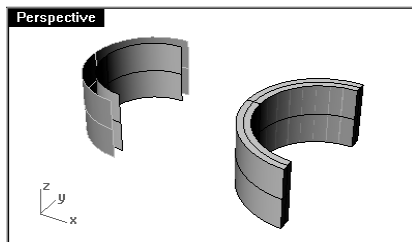


### To offset a surface to a solid:

- 1 Select the other purple surface.
- 2 From the **Surface** menu, click **Offset**.
- 3 At the **Offset distance <1>** ( Flip Solid Loose Tolerance ) prompt, click on the surface to change the normal direction if necessary.



- 4 At the **Offset distance <1>** ( Flip Solid Loose Tolerance ) prompt, choose the **Solid** option and press **Enter**.
- 5 At the **Offset distance <1>** ( Flip Loose Tolerance ) prompt, press **Enter** to create the offset surface and the surfaces needed to make the solid.

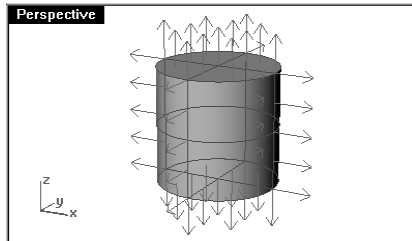


## Notes:

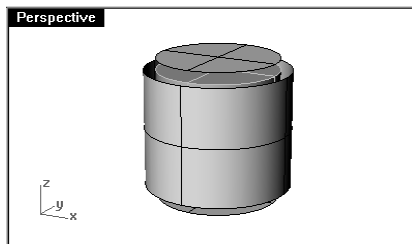
### To offset a polysurface:

- 1 Select the cylinder.
- 2 From the **Surface** menu, click **Offset**.

The normals on a closed polysurface will always point to the outside.



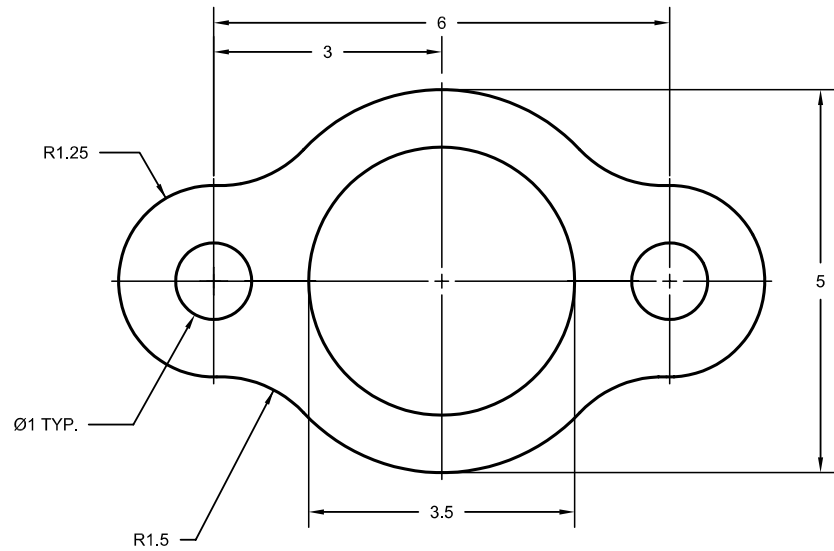
- 3 At the **Offset distance <1>** ( Flip Solid Loose Tolerance ) prompt, press **Enter**.  
Each surface of the polysurface is offset as a separate piece.





**Exercise 43—Practice**

- 1 Start a new model. Save as **Gasket1**
- 2 Use **Circle**, **Arc**, **Trim**, **Fillet**, and **Join** to create the part shown.



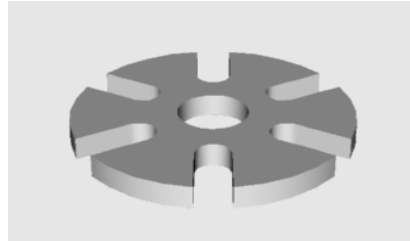
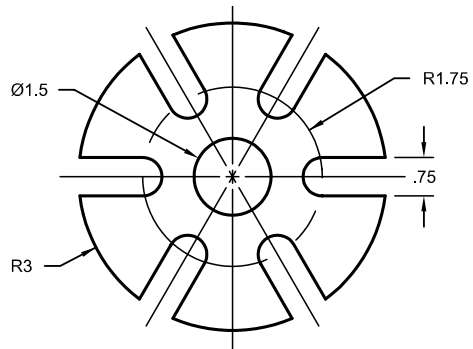
- 3 Use **Extrude Planar Curve** on the **Solid** menu to create the 3-D part. The extrusion thickness is **.125**.

**Notes:**

## Notes:

### Exercise 44—Practice

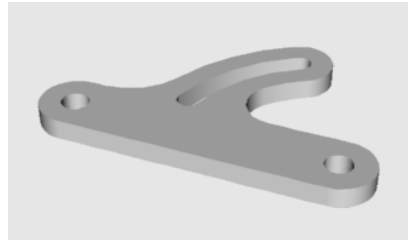
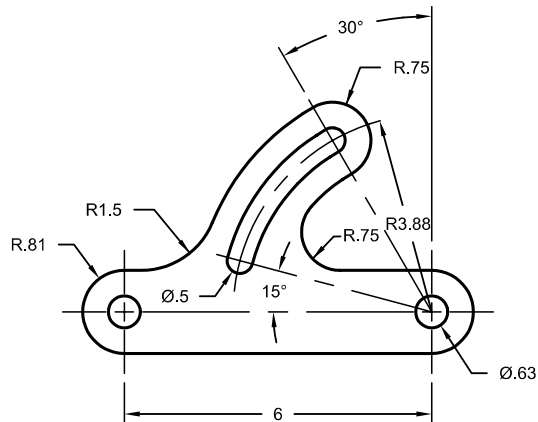
- 1 Start a new model. Save as **Cam**.
- 2 Use **Circle**, **Arc**, **Line**, **Trim**, **Join**, and **Array** to draw the part shown.



- 3 Use **Extrude Planar Curve** on the **Solid** menu to create the 3-D part. The extrusion thickness is 0.5.

### Exercise 45—Practice

- 1 Start a new model. Save as **Link**.
- 2 Use **Line**, **Arc**, **Trim**, **Offset**, **Join**, **Fillet** and **Circle** to draw the part shown.

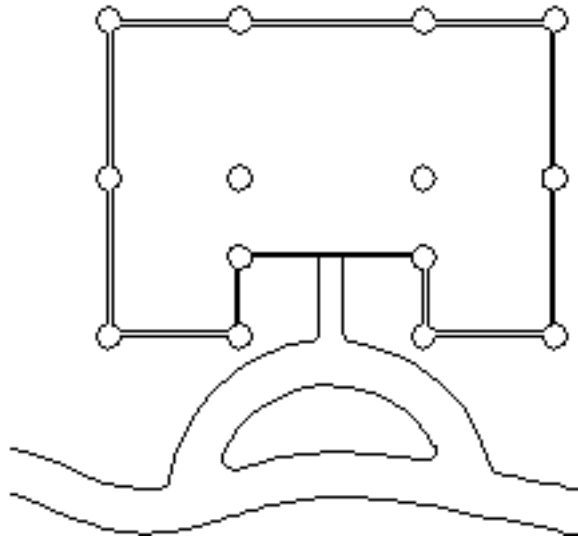


- 3 Use **Extrude** to create the 3-D part. The extrusion thickness is **.5**.

**Notes:**

**Exercise 46—Practice**

- 1 Start a new model. Save as **Building**.
- 2 Use **Line**, **Arc**, **Trim**, **Offset**, **Curve**, **Fillet** and **Circle** to draw the layout of the building shown.





# 6

## 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 **control point editing**.

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

Rhino's curves are represented internally with non-uniform rational B-splines ( NURBS ). Three things determine the shape of a NURBS curve:

- 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.

### A few facts about control points, edit points, and knots

- Control points do not have to be on the curve.
- Edit points are always on the curve.
- Rhino lets you edit curves and surfaces by moving control points and edit points.
- Knots are parameters (that is, numbers, not points).
- Adding knots to a curve or surface lets you control the movement of the object during control-point editing.

## Notes:

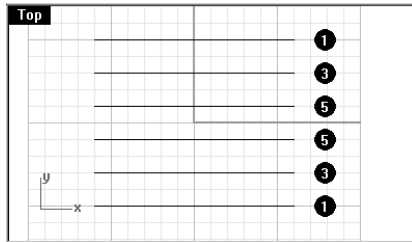
### Exercise 47—Control point editing

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.

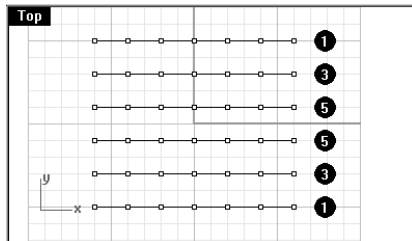
#### To edit control points:

- 1 **Open** the model **Control Point.3dm**.

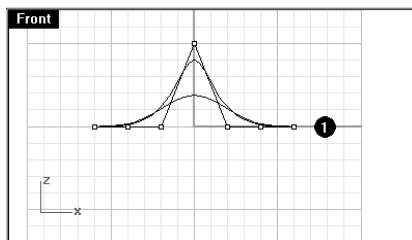
There are pairs of curves with different degrees in the model.



- 2 Turn **Ortho** and **Snap** on.
- 3 From the **Edit** menu, click **Select**, and then click **Curves**.
- 4 From the **Edit** menu, click **Point Editing**, and then click **Control Points On**. (Press **F10**.)



- 5 In the **Front** viewport, select the middle row of points.
- 6 Drag the points vertically, **5** units.



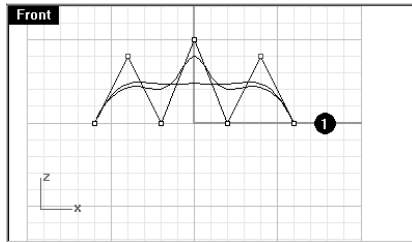
- 7 In the **Front** viewport, select two rows of points on either side of the center.



PointsOn

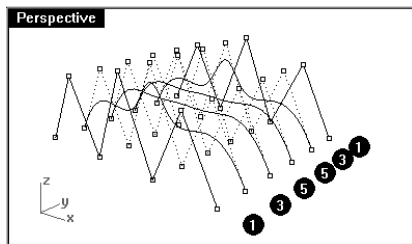
## Notes:

- 8 Drag the points vertically, 4 units.

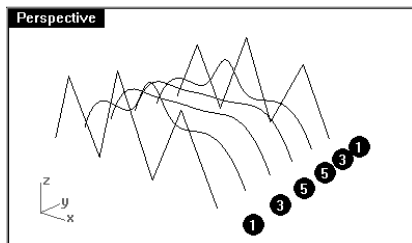


Notice that the degree 1 curves (polylines) come to a point at each control point and the control points are exactly on the curve. When a curve or polyline bends at a point like this it is called a kink. If you create a surface from a curve that has a kink, it will have a seam at the kink.

The degree 3 and 5 curves are smooth. The degree 3 curves have more curvature than the degree 5 curves. Individual points have more influence on a small area of the curve with degree 3 curves, while points have greater influence over a wider span of the curve with degree 5 curves.



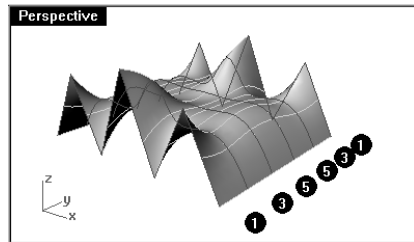
- 9 Press **Esc** twice to turn off the control points.



- 10 Select the curves.
- 11 From the **Surface** menu, click **Loft**.

- 12 In the **Loft options** dialog box, click **OK**.

Because the degree 1 curves were included in the loft, a polysurface is created with a seam at each kink.



- 13 Select the surface.
- 14 Turn on the control points.

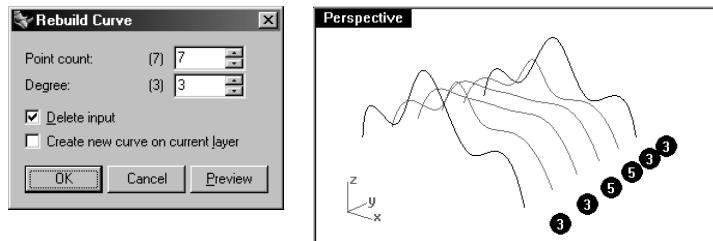
The points do not turn on and the following message is displayed on the command line:

**Cannot turn on points for polysurfaces.**

- 15 **Undo** the loft.

#### To change the polylines into curves without kinks:

- 1 From the **Curve** menu, click **Edit Tools**, and then click **Rebuild**.
- 2 At the **Select curves to rebuild** prompt, select both polylines.
- 3 At the **Select curves to rebuild** prompt, press **Enter**.
- 4 In the **Rebuild Curve** dialog box change the point count to **7** and the degree to **3**, and click **OK**.  
A degree 3 curve cannot have kinks. The curve smoothes and changes shape.



#### To loft a surface over the curves:

- 1 Select all of the curves.
- 2 From the **Surface** menu, click **Loft**.

#### Notes:

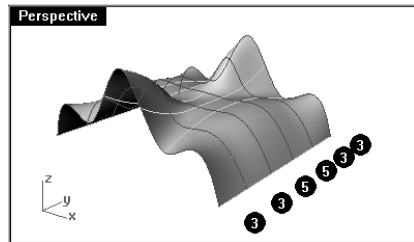


Rebuild



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

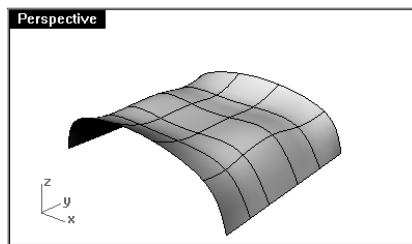
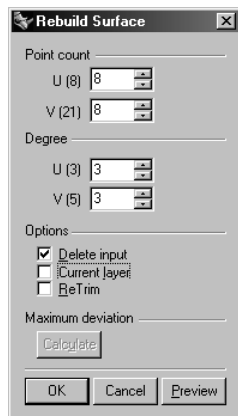
A single surface appears over the curves. The surface can be edited with control points.



### To rebuild a surface:

- 1 Select the surface.
- 2 From the **Surface** menu, click **Edit Tools**, then click **Rebuild**.
- 3 In the **Rebuild Surface** dialog, change the point count to **8** in both the U and V direction. Change the degree to **3** for both U and V.

The surface is smoother with fewer control points.



### Nudge Controls

Another method to move control points and other geometry in a more subtle way is to use the Nudge keys. The nudge keys are the arrow keys on the keyboard activated with the Alt, Ctrl, and Shift keys.

### Notes:

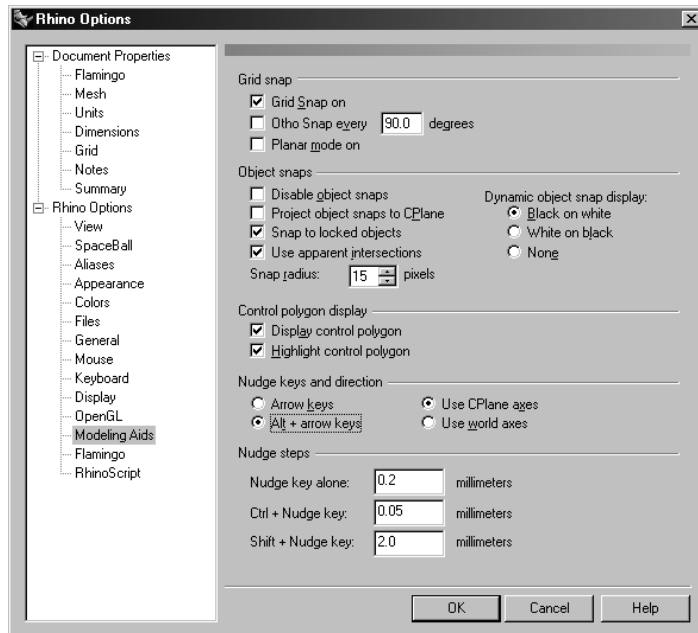


Rebuild Surface

## Notes:

### To change the nudge settings:

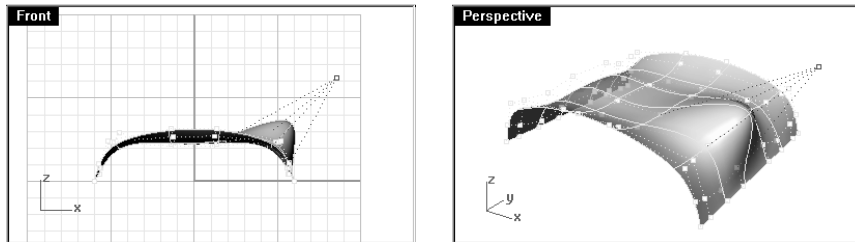
- 1 From the **Tools** menu, click **Options**.
- 2 In the **Options** dialog box, on the **Modeling Aids** page, note the **Nudge settings**. Any of these values can be changed.



### To use Nudge commands to move control points:

- 1 Select a control point in the **Front** viewport.
- 2 Hold down the **Alt** key down and press an arrow key. Notice that it moves (nudges) the point a small amount.
- 3 Hold the **Alt** and the **Ctrl** key down and press another arrow key. The movement is much smaller.

- 4 Hold the **Alt** and the **Shift** key down and press another arrow key.  
The movement is magnified.



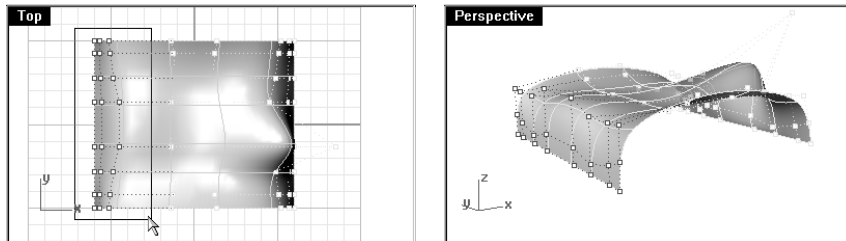
- 5 Hold the **Alt** and press the **PageUp** or **PageDown** key to nudge in the **Z** direction.

#### To use set points to adjust points:

- 1 Select all the points along the left edge of the surface.
- 2 From the **Transform** menu, click **Set Points**.
- 3 In the **Set Points** dialog box, check **Set X**, and uncheck **Set Y** and **Set Z**.



- 4 In the **Front** viewport move the points and click.  
The control points are aligned.



#### Notes:

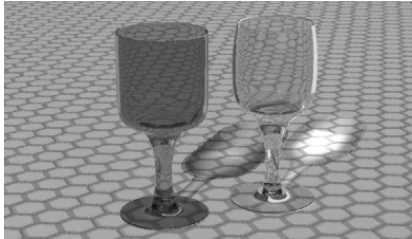


Set Points

## Notes:

### Exercise 48—Practice with curves and control point editing

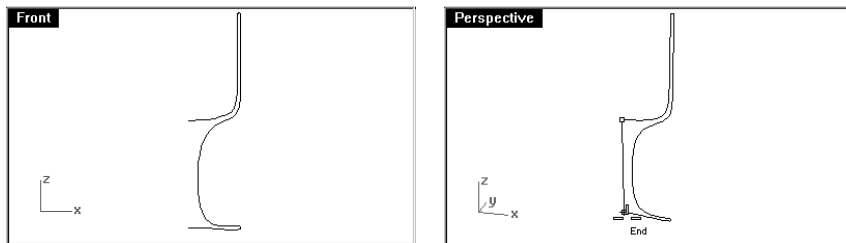
- 1 Start a new model. Save as **Glass**.



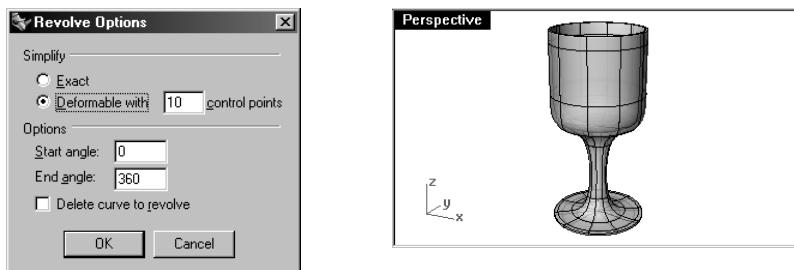
- 2 Use the **Curve** command to create the half cross-section of the glass.

#### To make it 3-D:

- 1 From the **Surface** menu, click **Revolve**.
- 2 At the **Select curves to revolve** prompt, select the curve you created, and press **Enter**.
- 3 At the **Start of revolve axis** prompt, pick one end of the curve.
- 4 At the **End of revolve axis** prompt, pick the other end of the curve.



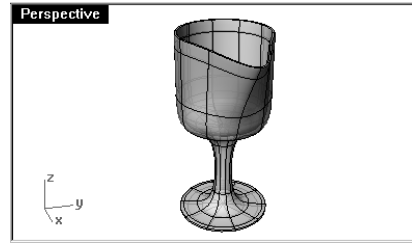
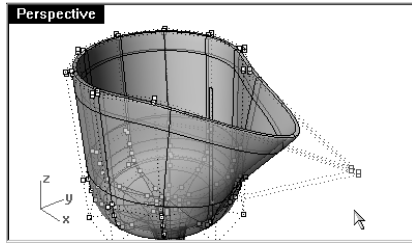
- 5 In the **Revolve Options** dialog box, check **Deformable**, then click **OK**.



- 6 Save your model.

**Notes:**

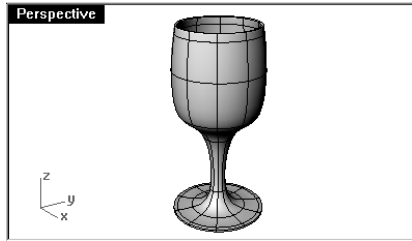
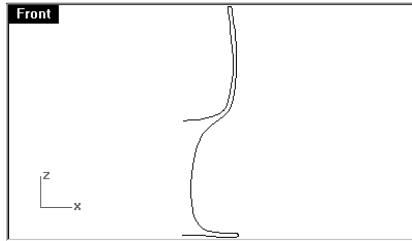
7 Experiment with adjusting the control points to see what happens.



8 **Incremental Save** your model.

9 Delete the surface.

10 Change the original curve and make another revolved surface.



11 **Incremental Save** your model.



# **Part Three: 3-D Modeling and Editing**

---





# 7

## Creating Deformable Shapes

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

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. You will create spheres that will be deformed to make the shapes.

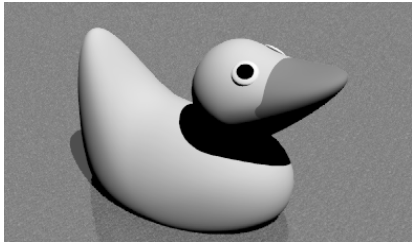
If you need to know more about control points and surfaces, search the *Rhino Help* index for “control points.”

## Notes:

### Exercise 49—Creating a rubber duck

- 1 Start a new model. Save as **Duck**.
- 2 You can use layers to separate your parts, but for this model, it is not necessary.

For more information about layers, look up “layer” the *Rhino Help* index.

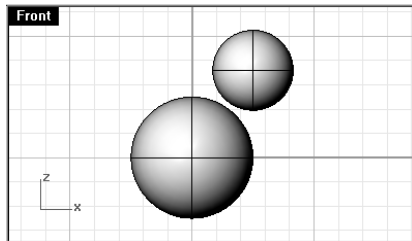


### Create the body and head shapes

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

#### To create the basic shapes:

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



#### To make the spheres deformable:

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



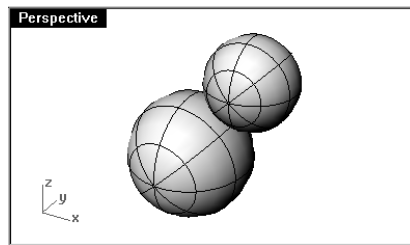
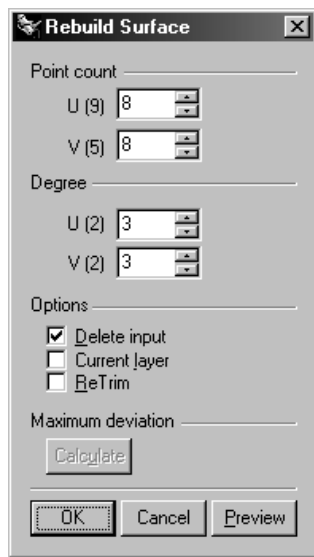
Sphere

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

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

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

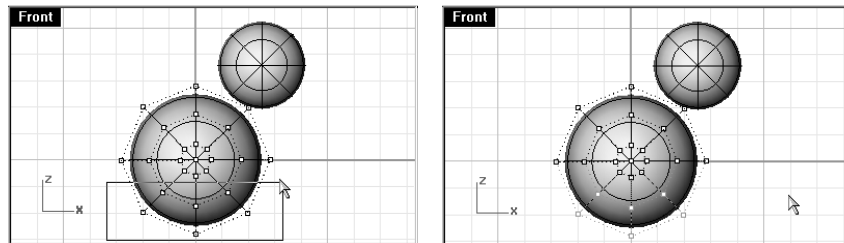
The spheres are now deformable. Having more control points allows more control over smaller parts of the surface. A degree-three surface will have a smoother shape when deformed.



### To modify the body shape:

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

To window select, drag a box left to right around the control points.



### Notes:



Rebuild Surface



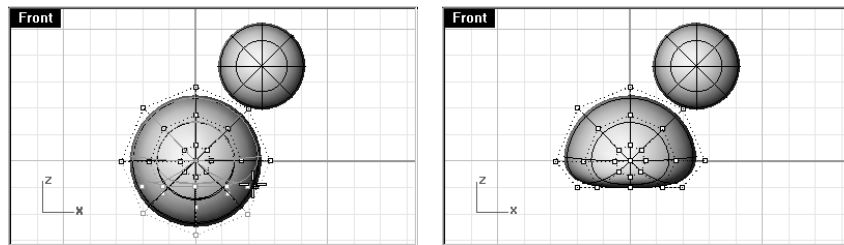
Control Points On

- 4 From the **Transform** menu, click **Set Points**.
- 5 Make the **Set Points** dialog box look like this, and click **OK**.



- 6 Drag the selected control points up.

This aligns all of the selected control points to the same world z-value (vertical in **Front** viewport), flattening the surface.



#### To scale the sphere shape:

- 1 Turn off control points and select the body shape.
- 2 From the **Transform** menu, click **Scale**, then click **Scale1D**.
- 3 At the **Origin point** ( Copy=No ) prompt, type **0** and press **Enter**.
- 4 At the **Scale factor or first reference point <1>** ( Copy=No ) prompt, with **Ortho** on, pick a point to the right in the **Front** viewport.

#### Notes:

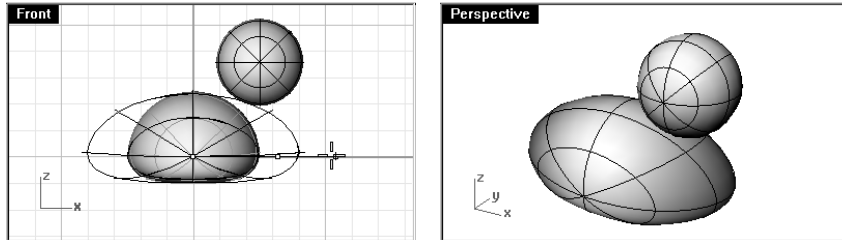


Set Points

## Notes:

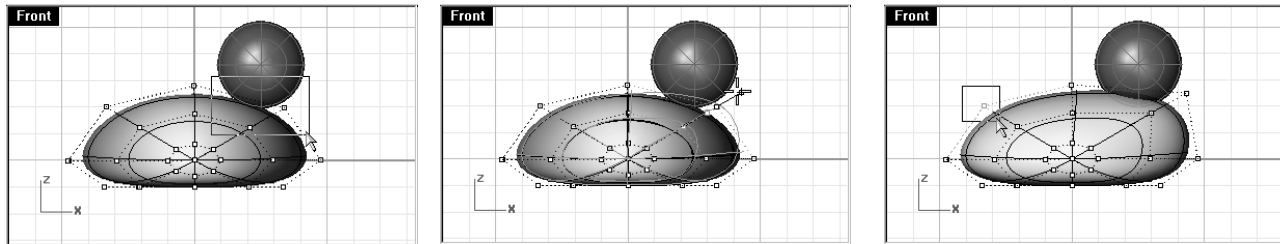
- At the **Second reference point** ( Copy=No ) prompt, pick a point further to the right in the **Front** viewport.

The body will be shaped more like an ellipsoid.



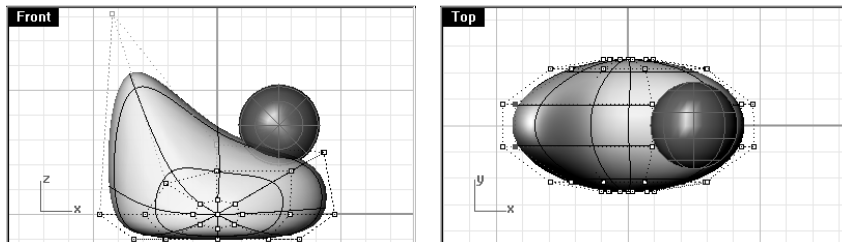
### To reshape the chest and tail

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



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

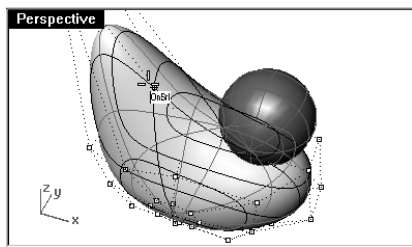


**To add control for further shaping the tail:**

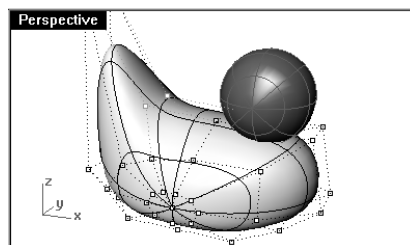
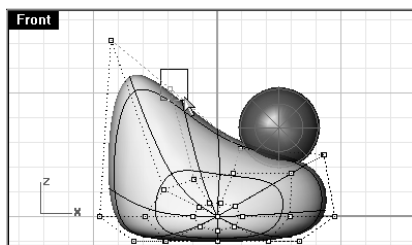
Before we start to edit the tail further, we will add an additional set of points to the tail portion of the body.

- 1 From the **Edit** menu, click **Control Points**, and then click **Insert Knot**.
- 2 At the **Select curve or surface for knot insertion** prompt, pick the body surface.  
A surface isocurve will be displayed. It will either be in the U or the V direction.
- 3 At the **Point on surface to add knot** ( Automatic Symmetrical=*No* Direction=*U* ) prompt, type **V** and press **Enter** to change the direction if necessary.
- 4 At the **Point on surface to add knot** ( Automatic Symmetrical=*No* Direction=*V* ) prompt, pick a point midway between the tail and the center of the body.
- 5 At the **Point on surface to add knot** ( Automatic Symmetrical=*No* Direction=*V* ) prompt, press **Enter**.

A new set of points in the V direction have been added to the body.



- 6 Window select the control points at the upper part of the new isocurve, and drag them down to further form the tail and the body.

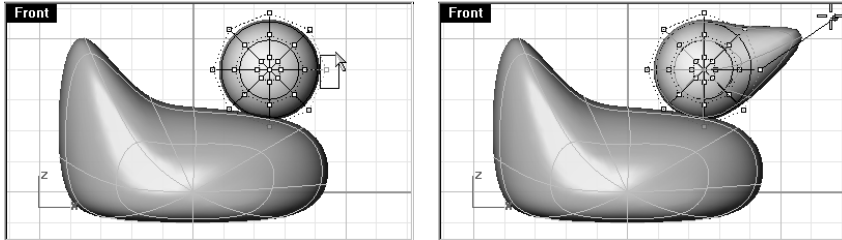


- 7 You can adjust the control points further until you get the shape you want.
- 8 **Save** your model.

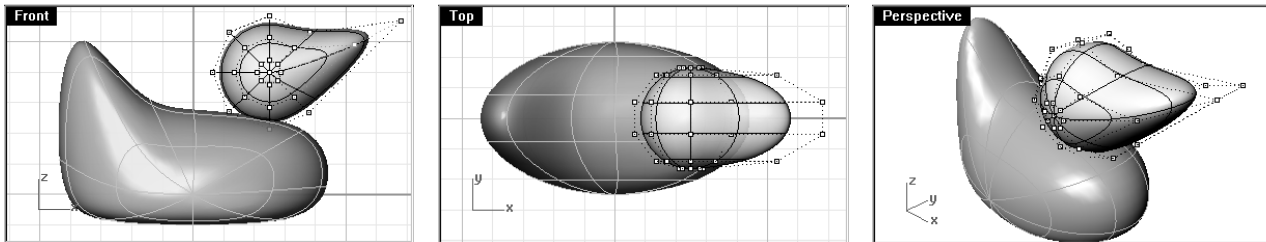
## Notes:

### To create the head:

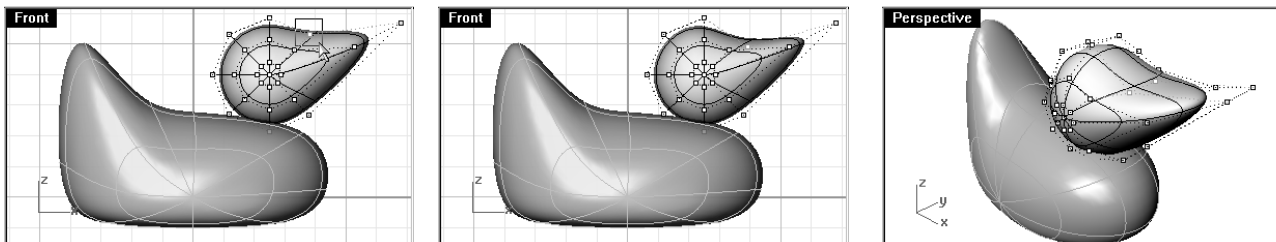
- 1 In the **Front** viewport, select the small sphere.
- 2 From the **Edit** menu, click **Control Points**, and then click **Control Points On**.
- 3 Select the control points on the right side and drag them to begin forming the bill.



- 4 Window select the control points further back on the same isocurve and drag them forward to widen the bill.



- 5 Window select control points at the top of the bill and drag them down as shown below.



- 6 Press **Esc** to turn control points off.

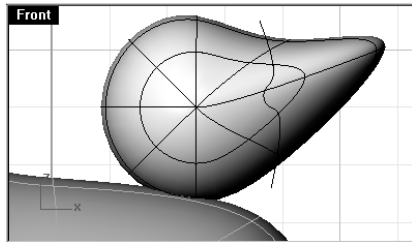
*Remember to use a window to select the control points. There may be multiple control points in the same location in this view.*

## 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.

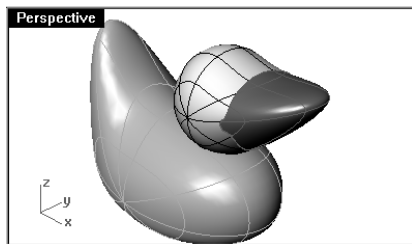
### To split a surface with a curve:

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



- 2 Select the head.
- 3 From the **Edit** menu, click **Split**.
- 4 At the **Select cutting objects. Press Enter to clear selection and start over ( Isocurve )** prompt, select the curve you just created.
- 5 At the **Select cutting objects. Press Enter when done ( Isocurve )** prompt, press **Enter**.

The bill and head are now separate surfaces so they can be rendered with different colors.





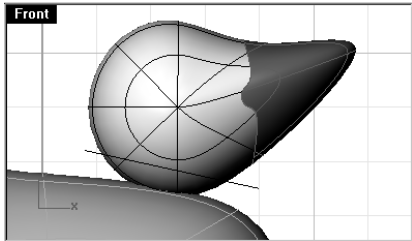
## Notes:

### Create the duck's neck

The ducky needs a neck. You will first make an edge on the surface of the head and a corresponding edge on the surface of the body so you can create a blended surface between the two edges.

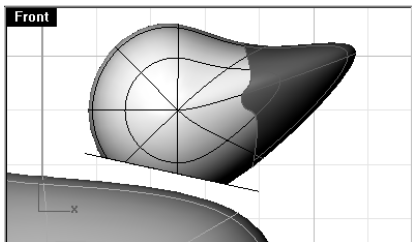
#### To trim the head:

- 1 Draw a **Line** across the bottom of the head.



- 2 Select the line you just created.
- 3 From the **Edit** menu, click **Trim**.
- 4 At the **Select object to trim. Press Enter to clear selection and start over** ( `ExtendLines=No` `UseApparentIntersections=No` ) prompt, select the bottom edge of the head.

The bottom of the head is trimmed.



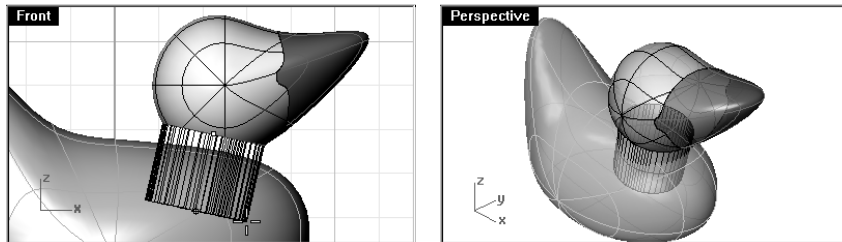
## Notes:



Extrude

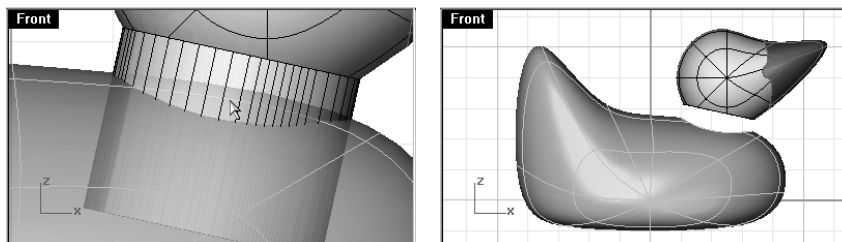
### To cut a hole in the body that matches the opening in the bottom of the head:

- 1 From the **Surface** menu, click **Extrude**, and then click **Straight**.
- 2 At the **Select curves or surfaces to extrude** prompt, select the edge curve at the bottom of the head.
- 3 At the **Select curves or surfaces to extrude** prompt, press **Enter**.
- 4 At the **Extrusion distance** ( Direction BothSides=No Cap=No Mode=Straight ) prompt, drag the extruded surface until it intersects the top of the duck body and click.



- 5 Select the cylindrical surface you extruded.
- 6 From the **Edit** menu, click **Trim**.
- 7 At the **Select object to trim. ...** ( ExtendLines=No UseApparentIntersections=No ) prompt, select the part of the body on the inside of the extruded surface.
- 8 At the **Select object to trim. ...** prompt, press **Enter**.

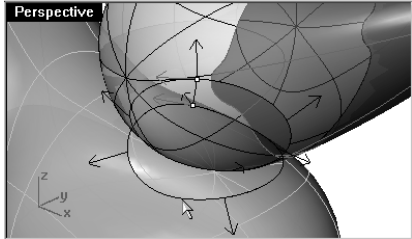
A hole will be created in the body.



- 9 **Delete** the extruded surface.
- 10 **Save** your model.

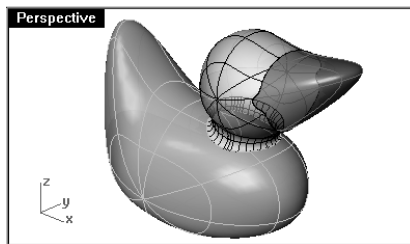
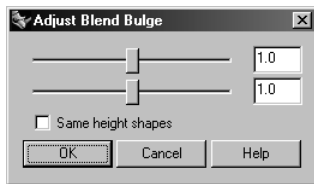
**To create the blend surface between the head and body:**

- 1 From the **Surface** menu, click **Blend Surface**.
- 2 At the **Select segment for first edge** ( PlanarSections Continuity=*Curvature* ) prompt, select the edge curve at the bottom of the head.
- 3 At the **Select segment for second edge ...** prompt, press select the edge of the hole in the body.



- 4 At the **Adjust curve seams ...** prompt, press **Enter**.
- 5 In the **Blend Bulge** dialog box, click **OK**.

A surface is blended between the body and the head.



- 6 **Save** your model.

**Notes:**

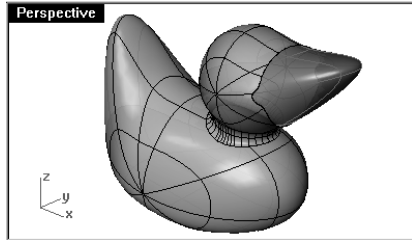


Blend Surface

### To join the parts:

- 1 Select the body the blend surface and the back of the head.
- 2 From the **Edit** menu, click **Join**.

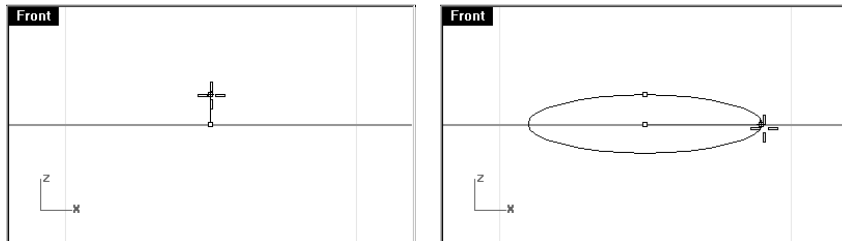
The three surfaces are joined into one. The bill is left separate for rendering purposes.



### To make an eye:

- 1 From the **Solid** menu, click **Ellipsoid**.
- 2 Toggle **Ortho** and **Snap** on to help.
- 3 At the **Ellipsoid center ...** prompt, pick a point in the **Front** viewport.
- 4 At the **End of first axis ...** prompt, pick a point in the vertical direction.
- 5 At the **End of second axis** prompt, pick a point that creates an elliptical shape.

The order of the first two points is important to correctly orient the isocurves on the surface.



### Notes:



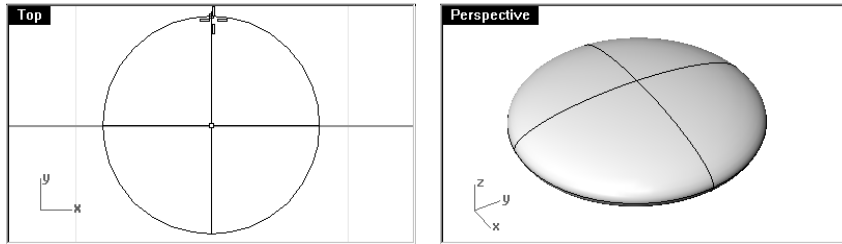
Join



Ellipsoid

## Notes:

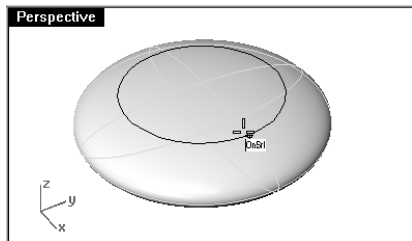
- At the **End of third axis** prompt, pick a point in the **Top** viewport that creates a round, flat ellipsoid.



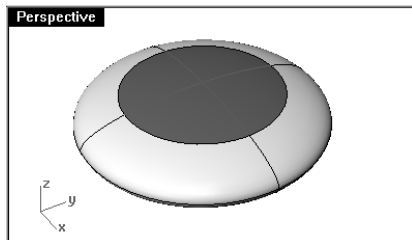
### To make the pupil for the eye:

To be able to assign a different color for the pupil of the eye, the ellipsoid surface has to be split into two parts. In this example we will use an isocurve to split the ellipsoid into two parts.

- Select the ellipsoid.
- From the **Edit** menu, click **Split**.
- At the **Select cutting objects. ...** ( Isocurve ) prompt, click Isocurve.



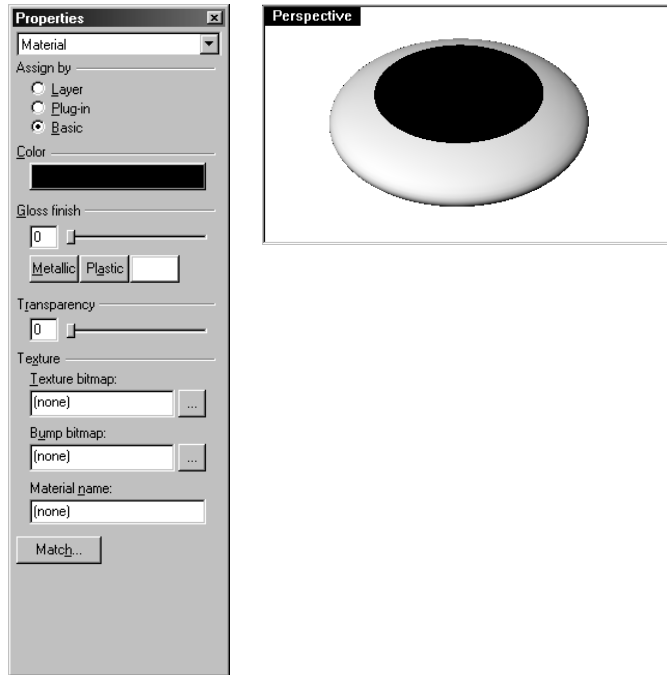
- At the **Split point** ( Direction= $U$  ) prompt, pick on the top part of the surface to define an isocurve for the split.
- At the **Split point** ( Direction= $U$  ) prompt, press **Enter**.  
The surface is split at the isocurve.



- Select the surface at the top of the ellipsoid.

7 From the **Edit** menu, click **Object Properties**.

8 In the **Properties** window, on the **Material** page, click **Basic**, click on the **Color** button, and select a color for the pupil of the eye, like black.



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

#### To group the parts of the eye:

- 1 Select the both eye surfaces.
- 2 From the **Edit** menu, click **Groups** and then click **Group**.  
The eye parts are grouped as one object.

#### To move the eyes to the surface of the head:

- 1 Select the eye group.
- 2 From the **Transform** menu, click **Orient**, and then click **On Surface**.

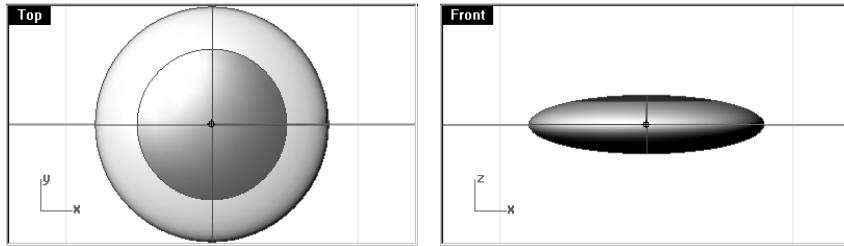
#### Notes:



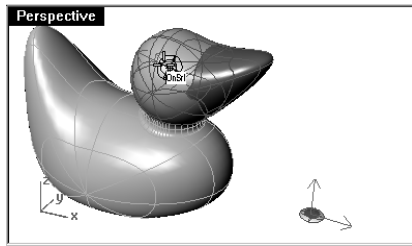
Properties

**Notes:**

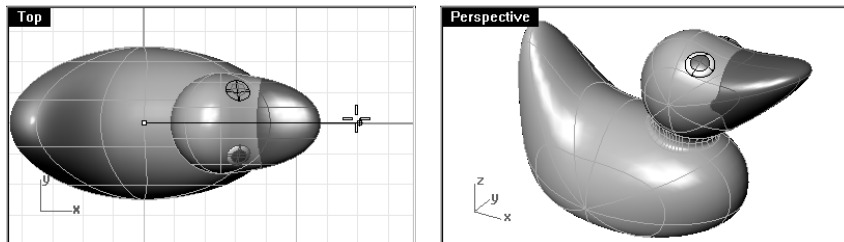
- 3 At the **Point to orient from** ( OnSurface ) prompt, snap to the center of the eye in the **Top** viewport.



- 4 At the **Surface to orient on** prompt, click on the head.
- 5 At the **Point on surface to orient to** ( Align=*U* Copy=*Yes* FlipNormal=*No* Mirror=*None* SwapUV=*No* ) prompt, change the Copy option to **No**.
- 6 At the **Point on surface to orient to ...** prompt, click the location for the eye.



- 7 **Mirror** the eye to the other side of the head.

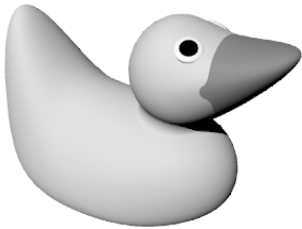


## Render a picture of the ducky

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 and shaded mode.

To render the ducky:

- 1 Select the beak.
- 2 From the **Edit** menu, click **Object Properties**.
- 3 In the **Properties** window, on the **Material** page, click **Basic** and click the color swatch.
- 4 In the **Select Color** dialog box, select a color for the beak, like orange.
- 5 Select the body.
- 6 From the **Edit** menu, click **Object Properties**.
- 7 In the **Properties** window, on the **Material** page, check **Basic** and select a color for the body, like yellow.
- 8 From the **Render** menu, click **Render**.



## Notes:

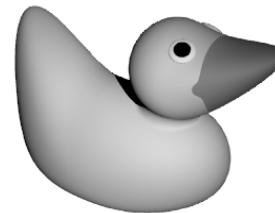
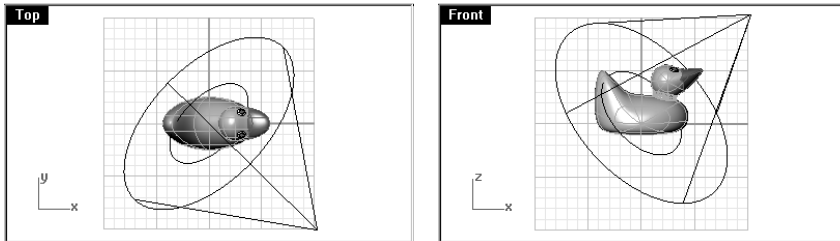


Render



### To place lights:

- 1 From the **Render** menu, click **Create Spotlight**.
- 2 At the **Base of cone** ( Vertical Diameter 3Point Tangent AroundCurve ) prompt, select a point in the middle of the model.
- 3 At the **Radius <1.000> ( Diameter )** prompt, drag the radius until it is approximately three times as large as the model.
- 4 At the **End of cone** prompt, pick a point in the **Top** viewport while holding the **Ctrl** key down to activate elevator mode.  
In the **Front** viewport pick a point slightly above the object.
- 5 From the **Render** menu, click **Render**.



### Notes:



Spotlight










# 8

## Modeling with Solids













Modeling solids in Rhino is easy. There are several commands that enable you create and edit solid objects.

Solids in Rhino are closed surfaces or polysurfaces that enclose a volume. Some of the solid primitives are closed single-surfaces carefully edge-matched, others are polysurfaces.

Rhino's polysurface objects are not deformable. You cannot turn control points on and use them to deform the polysurface.

Button	Command	Description
	Box	Draw a rectangular box from two diagonal corners and a height.
	Box 3Point	Draw a box from two adjacent corners, a point on the edge across from them, and a height.
	Sphere	Draw a sphere from a center point and a radius.
	Sphere Diameter	Draw a sphere from the two ends of a diameter.
	Sphere 3Point	Draw a sphere from three points on its surface.
	Cylinder	Draw a solid cylinder from a center point, a radius, and a height.
	Tube	Draw a solid tube from a center point, two radii, and a height.

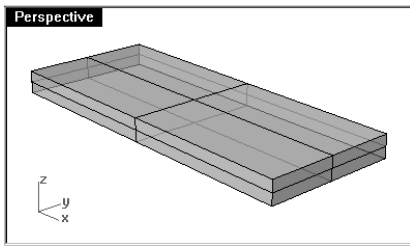
**Notes:**

Button	Command	Description
	Cone	Draw a solid cone from a base point, a base radius, and a height.
	TCone	Draw a truncated cone that ends in a circular plane from a base point, two radii, and a height.
	Ellipsoid	Draw an ellipsoid from a center point, and three axis endpoints.
	Torus	Draw a solid Torus from a center point, a radius for the center of the tube, and a tube radius.
	Pipe	Draw a pipe around an existing curve. The pipe has a circular cross-section and optional caps at the ends. 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.
	TextObject	Draw text as outline curves, surfaces, or solids.
	Extrude	Draw a solid by extruding a closed planar curve vertically.
	Extrude	Creates a solid by extruding a surface vertically.
	Cap	Creates planar surfaces to close planar holes in a surface or partial solid.
	BooleanUnion	Boolean operation in Rhino to combine solids.
	BooleanDifference	Boolean operation in Rhino to subtract surfaces and solids from one another.
	BooleanIntersection	Boolean operation in Rhino to create the intersection of two surfaces or solids.

### Exercise 50— Model a bar with text

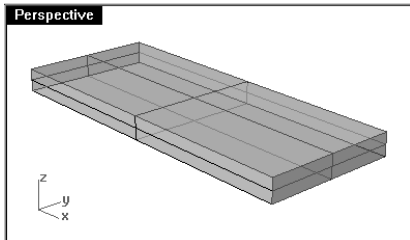
In the following exercise we will make a solid primitive, extract some surfaces, rebuild a surface and deform it, join the new surfaces into a solid, fillet the edges, add text to a surface, and do a Boolean operation on the solid.

- 1 Start a new model using the **Millimeters** template. **Save as Bar.**
- 2 From the **Solid** menu, click **Box.**
- 3 At the **First corner of base** ( 3Point Vertical Center ) prompt, type **0,0** and press **Enter.**
- 4 At the **Other corner of base or length** prompt, type **15** and press **Enter.**
- 5 At the **Width. Press Enter to use length** prompt, type **6** and press **Enter.**
- 6 At the **Height. Press Enter to use length** prompt, type **1** and press **Enter.**



#### To edit the box:

- 1 From the **Solid** menu, click **Extract Surface.**
- 2 At the **Select surfaces to extract** ( Copy ) prompt, select the top and both ends.
- 3 At the **Select surfaces to extract. Press Enter when done** ( Copy ) prompt, press **Enter.**



#### Notes:

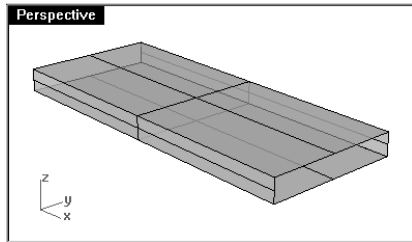


Box

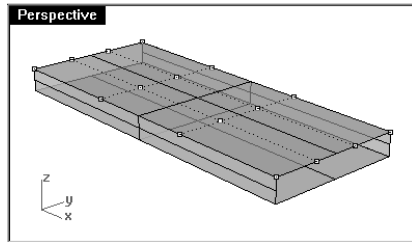
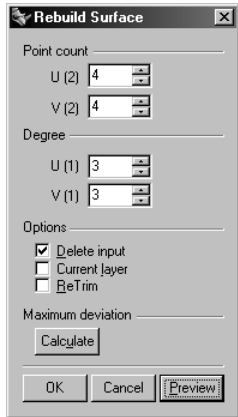


Extract Surface

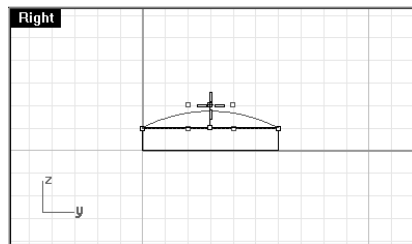
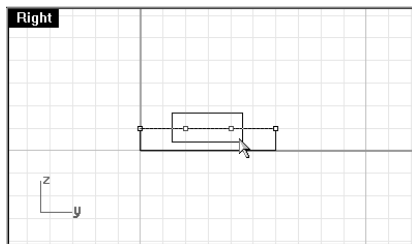
- 4 Select the two ends and delete them.



- 5 Select the top extracted surface.  
6 From the **Surface** menu, click **Edit Tools**, then click **Rebuild**.  
7 In the **Rebuild Surface** dialog box, set the **Point count** to **4** and the **Degree** to **3** for both U and V.



- 8 From the **Edit** menu, click **Point Editing**, then click **Control Points On**.  
9 In the **Right** viewport, window select the middle points and drag them up approximately one unit.



- 10 Turn off control points.  
11 Select all of the surfaces.

## Notes:



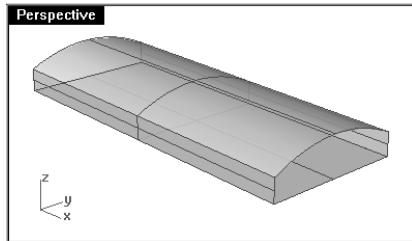
Rebuild Surface



Control Points On

- 12 From the **Edit** menu, click **Join**.

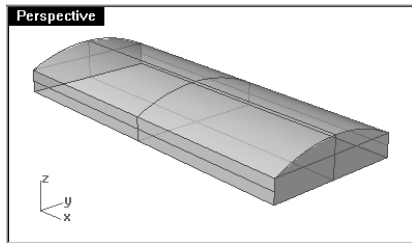
The surfaces are joined making an open polysurface.



- 13 Select the polysurface.

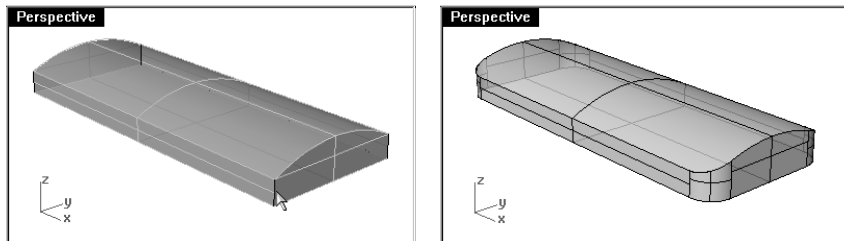
- 14 From the **Solid** menu, click **Cap Planar Holes**.

Two end caps are created.



#### To fillet the edges:

- 1 From the **Solid** menu, click **Fillet Edge**.
- 2 At the **Select edges to fillet** ( Radius=1 ) prompt, select the vertical edges.
- 3 At the **Select edges to fillet. Press Enter when done** ( Radius=1 ) prompt, press **Enter**.



- 4 From the **Solid** menu, click **Fillet Edge**.
- 5 At the **Select edges to fillet** ( Radius=1 ) prompt, type **.2** and press **Enter**.
- 6 At the **Select edges to fillet** ( Radius=0.2 ) prompt, window select the entire bar to get the horizontal edges.

#### Notes:

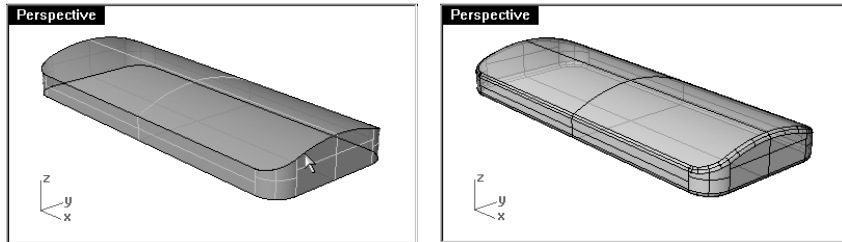


Cap Planar Holes



Fillet Edge

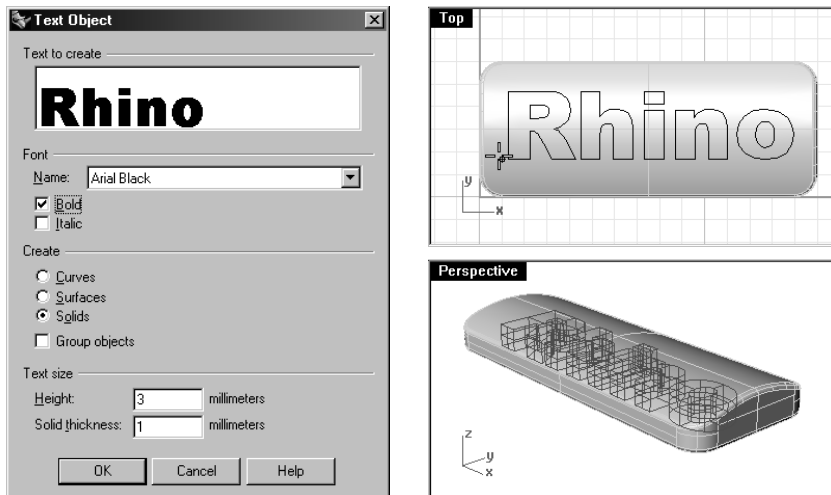
- 7 At the **Select edges to fillet. ...** ( Radius=0.2 ) prompt, press **Enter**.



- 8 Make a copy of the bar and lock it.  
We will use the second bar for another technique later in the exercise.

### To make solid text:

- 1 From the **Solid** menu, click **Text**.
- 2 In the **Text Object** dialog box, select a **Font**.  
Under **Create** click **Solids**.  
Under **Text size**, set the **Height** to **3.00** and the **Solid thickness** to **1.00**.  
Place the text in the **Top** viewport.

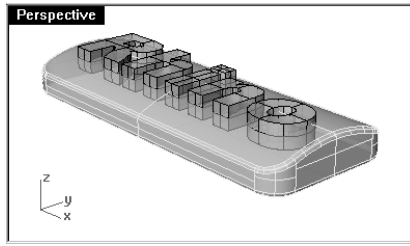
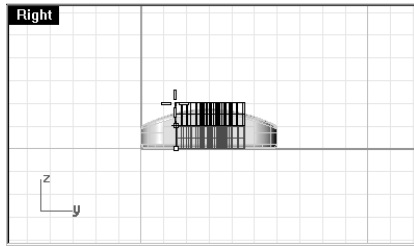


- 3 Drag the text to the middle of the bar and click.
- 4 In the **Front** or **Right** viewport, drag the text until it protrudes through the top surface.

### Notes:

**T**  
TextObject  
Look for this button

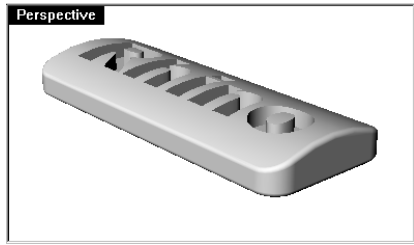




### To engrave the text in the bar:

- 1 Select the bar.
- 2 From the **Solid** menu, click **Difference**.
- 3 At the **Select second set of surfaces or polysurfaces** ( DeleteInput=Yes ) prompt, window select all of the text.
- 4 At the **Select second set of surfaces or polysurfaces. Press Enter when done** ( DeleteInput=Yes ) prompt, press **Enter**.

The text is embossed into the bar.

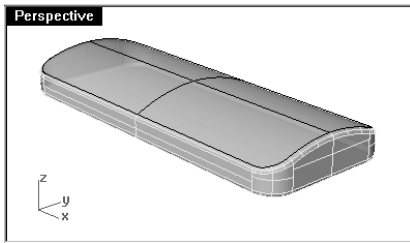


### Notes:

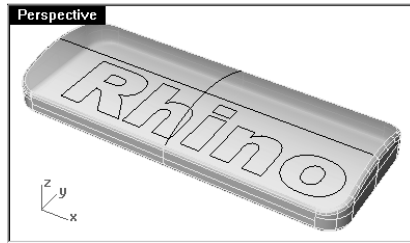
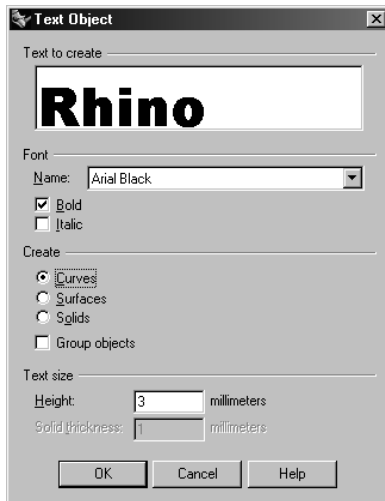
## Notes:

### To make a label:

- 1 **Hide** the finished bar and **Unlock** the copy of the bar.
- 2 From the **Solid** menu, click **Extract Surface**.
- 3 At the **Select surfaces to extract** ( Copy ) prompt, select the top surface.
- 4 At the **Select surfaces to extract** ( Copy ) prompt, press **Enter**.



- 5 Select the lower part of the bar and lock it.
- 6 Make the **Top** viewport active.
- 7 From the **Solid** menu, click **Text**.
- 8 In the **Text Object** dialog box, under **Create**, click **Curves**, and click **OK**.

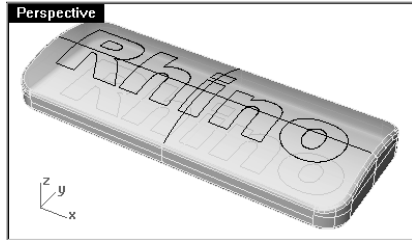


- 9 Place the text in the **Top** viewport.
- 10 Drag the text to the middle of the bar and click.

### To project the text curves through the bar:

- 1 Select the text (curves).
- 2 From the **Curve** menu, click **Curve From Objects**, and then click **Project**.
- 3 At the **Select surfaces or polysurfaces to project onto** prompt, select the top surface in the **Top** viewport.

The **Project** command projects objects perpendicular to the construction plane.



- 4 Delete the original text curves.
- 5 Select the top surface.
- 6 From the **Edit** menu, click **Split**.
- 7 At the **Select cutting objects. ...** prompt, window select the text curves, and press **Enter**.

The curves have split the surface. Each part of the text is a separate surface.



- 8 From the **Edit** menu, click **Select**, then click **Curves**.  
This will select the curves you used to split the surface.
- 9 From the **Edit** menu, click **Visibility**, then click **Hide** to hide the curves.

### To change the render color of the text:

- 1 Select the text surfaces.
- 2 From the **Edit** menu, click **Object Properties**.

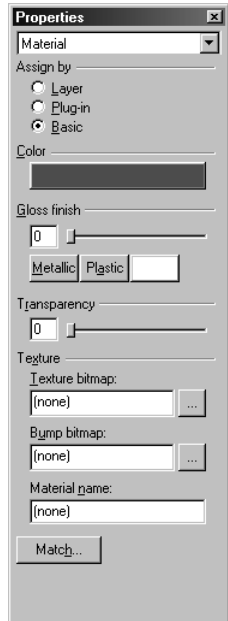
### Notes:



Project  
Look for this button

**Notes:**

- 3 In the **Properties** window, on the **Material** page, click **Basic** and select a color for the text, like red.



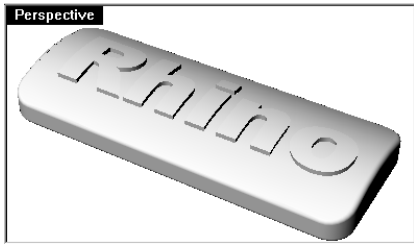
- 4 Right click on the **Perspective** viewport title.
  - 5 From the menu, click **Rendered Display**.
- The letters render in a different color.



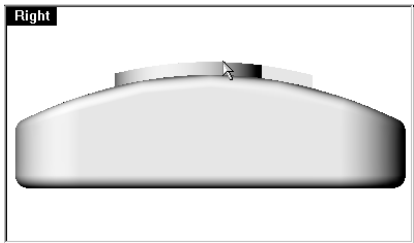
**To emboss the text in the bar:**

- 1 Click in the **Top** viewport.
- 2 From the **Edit** menu, click **Select**, then click **Previous Selection**.  
This selects the text surfaces again.
- 3 From the **Solid** menu, click **Extrude Solid**, then click **Straight**.
- 4 At the **Extrusion distance** ( Direction BothSides=*No* Cap=*Yes* Mode=*Straight* ) prompt, type **.2** and press **Enter**.

The surfaces are extruded perpendicular to the active construction plane.



Notice that the top of the text follows the curvature of the original surface.



**Notes:**



Extrude Surface  
Look for this button



# 9








## Creating 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 isoparametric curves (isocurves) on the surface.

Surfaces have an area, their shape can be changed by moving control points, and they can be meshed.

**Notes:**

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 approximates a set of curves and/or point objects.
	Revolve	Revolves a curve around an axis to create a surface.
	Loft	Creates a surface from shape curves; the normal, loose, and tight options make a surface with no creases as it passes over the shape curves. The StraightSections 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 follow along a rail curve that defines one edge of the surface.

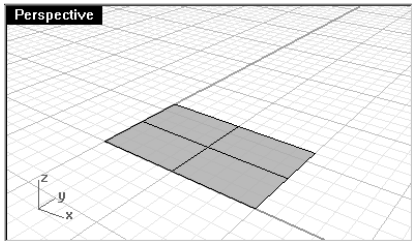
**Notes:**

Button	Command	Description
	Sweep2	Creates a surface from shape curves that follow along two rail curves that define two edges of the surface.
	FilletSrf	Creates a fillet or round between two surfaces.
	BlendSrf	Makes a smooth surface between two existing surfaces
	RailRevolve	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.
	Extrude AlongCurve	Extrudes a curve following along a second curve.
	Extrude ToPoint	Extrudes a curve to a point.
	Plane	Creates a rectangular planar surface parallel to the construction plane from two diagonal points.
	Plane 3Point	Creates a rectangular planar surface from 3 points.
	Plane Vertical	Creates a rectangular planar surface from 3 points that is vertical to the construction plane.



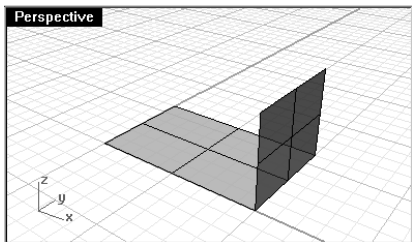
### Exercise 51—Basic techniques for making surfaces

- 1 Start a new model. Save as **Surfaces**.  
In this exercise, you will model some simple surfaces.
- 2 Turn on **Snap** and **Planar**.
- 3 From the **Surface** menu, click **Plane**, then click **Corner to Corner**.
- 4 In the **Top** viewport, at the **First corner of plane** ( 3Point Vertical Center Deformable ) prompt, pick a point.
- 5 At the **Other corner or length** prompt, pick another point to make a rectangular plane.



#### To create a vertical plane:

- 1 From the **Surface** menu, click **Plane**, and 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.
- 4 At the **Height. Press Enter to use width** prompt, drag your cursor up and pick.



#### Notes:



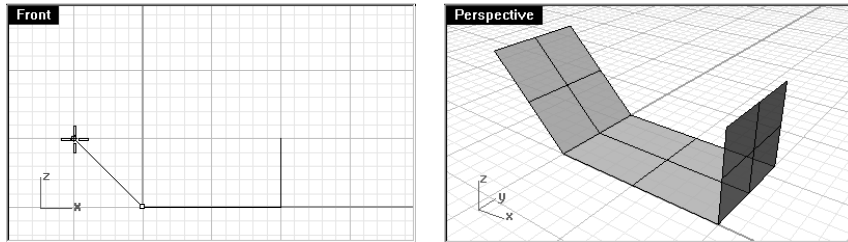
Plane  
Look for this button



Plane Vertical  
Look for this button

### To create a plane from 3 points:

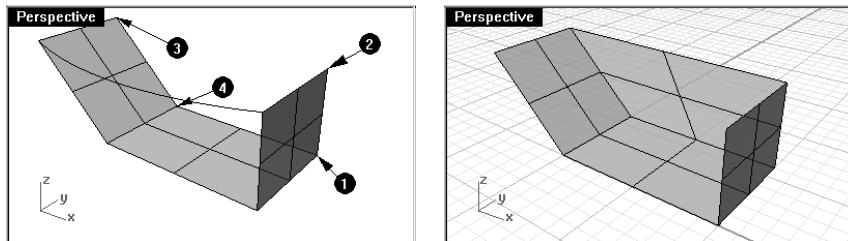
- 1 From the **Surface** menu, click **Plane**, and then click **3 Points**.
- 2 At the **Start of edge ...** prompt, snap to the endpoint at the left of the first surface.
- 3 At the **End of edge** prompt, snap to the other endpoint at the left side of the first surface.
- 4 At the **Width. Press Enter to use length** prompt, pick a point in the **Front** viewport that is angled to the left and the same height as the vertical surface.



### To create a plane from corner points:

- 1 From the **Surface** menu, click **Corner Points**.
- 2 At the **First corner of surface** prompt, snap to an endpoint at the edge of the first surface (1).
- 3 At the **Second corner of surface** prompt, snap to the endpoint at the edge of the first surface (2).
- 4 At the **Third corner of surface** prompt, snap to an endpoint at the edge of the third surface (3).
- 5 At the **Fourth corner of surface** prompt, snap to the other endpoint at the edge of the third surface (4).

A surface will be created with corners at the points you selected.



### Notes:



Plane 3 Points  
Look for this button

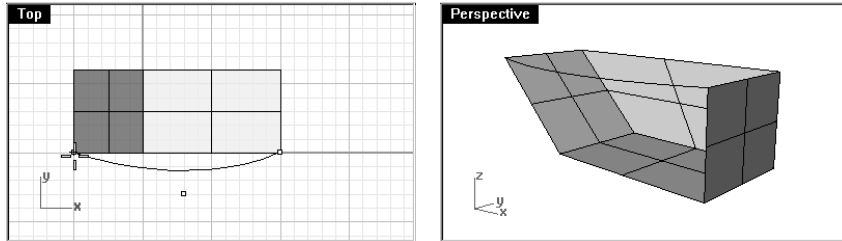


Surface from 3 or 4 Corner Points  
Look for this button

**To create a plane from edge curves:**

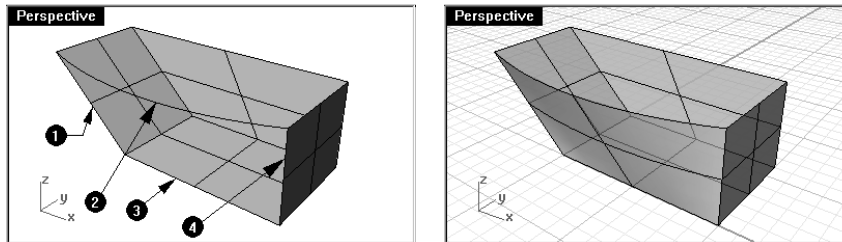
- 1 Draw a curve in the **Top** viewport that starts and ends at the top of the two vertical surfaces as shown below.

Planar mode keeps this curve on the same plane as the surface corners.



- 2 From the **Surface** menu, click **Edge Curves**.
- 3 At the **Select 2, 3, or 4 curves** prompt, select the three surface edges and the curve you created.

A surface is created.



**Notes:**

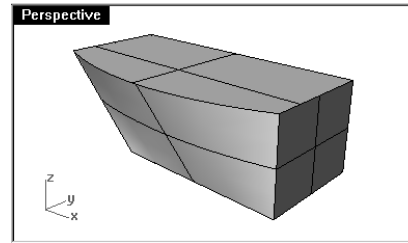
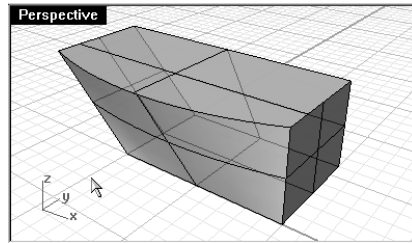
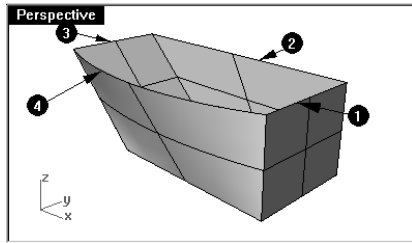


Surface from Edge Curves  
Look for this button

**To create a surface from planar curves:**

- 1 From the **Surface** menu, click **Planar Curves**.
- 2 At the **Select planar curves to build surface** prompt, select the top edge of a surface you just created.
- 3 At the **Select planar curves to build surface. ...** prompt, select the other three edges and press **Enter**.

A surface is created.



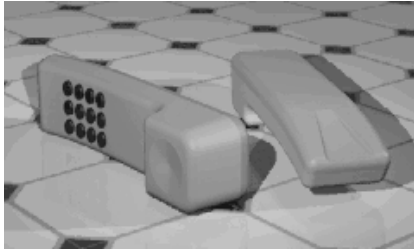
**Notes:**



Surface from Planar Curves  
Look for this button

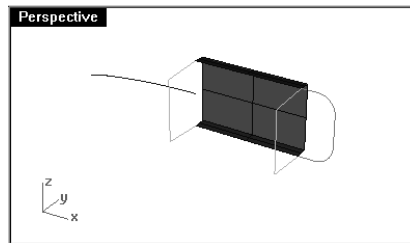
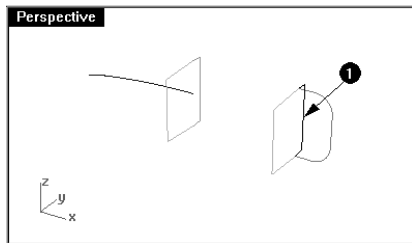
## Exercise 52—Extruding surfaces

In this exercise, you will be creating a cordless phone using extrusions. To aid in organizing the model, surface and curve layers have been created. Make sure you change layers as you are making the extrusions.

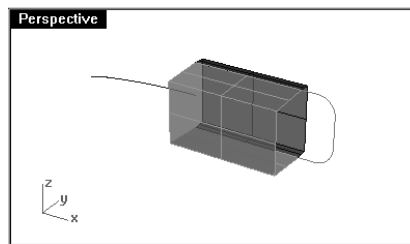
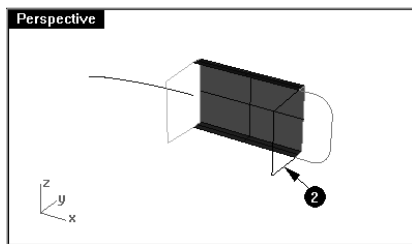


- 1 **Open** the model **Extrude.3dm**.
- 2 Select the curve (1) as shown below.
- 3 From the **Surface** menu, click **Extrude**, and then click **Straight**.
- 4 At the **Extrusion distance** ( Direction BothSides=No Cap=No Mode=Straight ) prompt, type **-3.5** and press **Enter**.

If the object being extruded is a planar curve, the curve is extruded perpendicular to the plane of the curve.



- 5 Press **Esc** to unselect the curve.
- 6 Change to the **Bottom Surface** layer.
- 7 Repeat this process for the other curve (2).



## Notes:



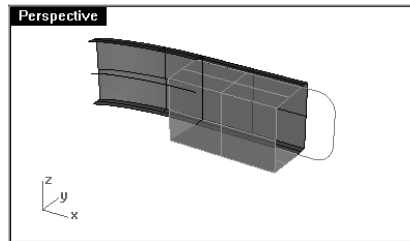
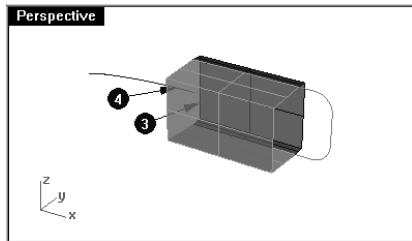
Extrude  
Look for this button

*If you do not get the answer you expected, undo and try picking near the other end of the path curve.*

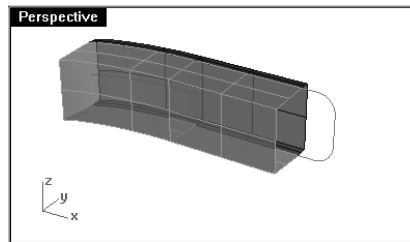
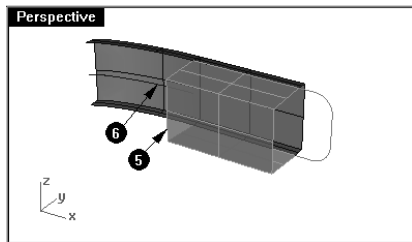
### To extrude a curve along another curve:

- 1 Change to the **Top Surface** layer.
- 2 Select the curve on the left (3).
- 3 From the **Surface** menu, click **Extrude**, and then click **Along Curve**.
- 4 At the **Select path curve** ( *Cap=No Mode=AlongCurve* ) prompt, select the path curve (4) near its right end.

The curve is extruded along the path of the secondary curve.



- 5 Change to the **Bottom Surface** layer.
- 6 Repeat this process for the other curve (5).



### To extrude a curve with a taper ( draft angle ) :

- 1 Change to the **Top Surface** layer.
- 2 Select the curve on the right (1).
- 3 From the **Surface** menu, click **Extrude**, and then click **Tapered**.
- 4 At the **Extrusion distance** ( *Direction DraftAngle=0 Cap=No Corners=Sharp Mode=Tapered* ) prompt, click **DraftAngle**.
- 5 At the **Draft angle <0>** prompt, type **3** and press **Enter**.

### Notes:

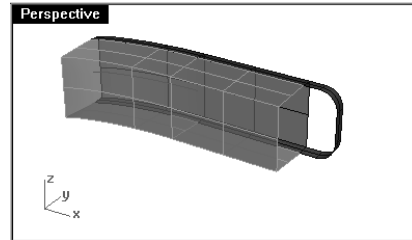
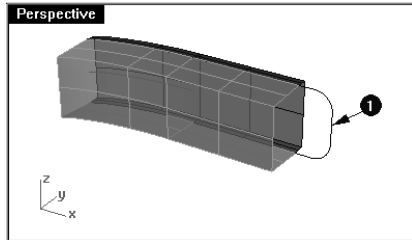


Extrude Along Curve  
Look for this button

## Notes:

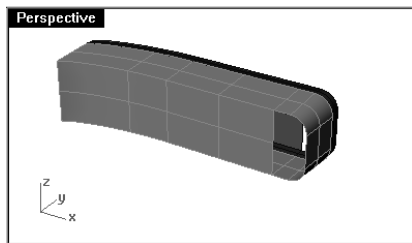
- At the **Extrusion distance** ( Direction DraftAngle=3 Cap=No Corners=Sharp Mode=Tapered ) prompt, type **.375** and press **Enter**.

The curve is extruded with a three-degree draft angle in the positive direction on the y-axis.



- Change to the **Bottom Surface** layer.
- Select the same curve.
- From the **Surface** menu, click **Extrude**, and then click **Tapered**.
- At the **Extrusion distance** ( Direction DraftAngle=3 Cap=No Corners=Sharp Mode=Tapered ) prompt, type **-1.375** and press **Enter**.

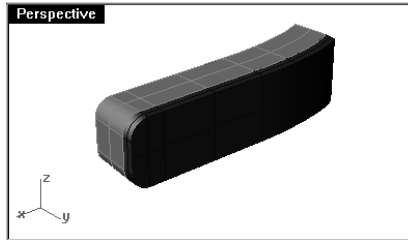
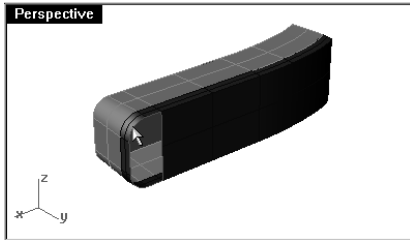
The curve is extruded with a three-degree draft angle in the negative direction from the previous extrusion.



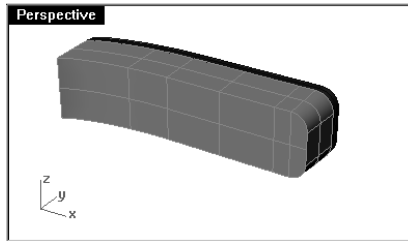
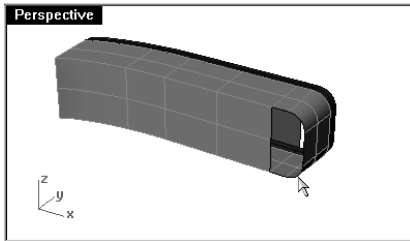
**To create a surface from a planar curves:**

- 1 Change to the **Top Surface** layer.
- 2 From the **Surface** menu, click **Planar Curves**.
- 3 At the **Select planar curves to build surface** prompt, select the edge curves that bound the openings of the tapered extrusion at the top.
- 4 At the **Select planar curves to build surface. ...** prompt, press **Enter**.

A surface is created at the end.



- 5 Change to the **Bottom Surface** layer.
- 6 Repeat this process for the other end.



- 7 Select all of the top surfaces.
- 8 From the **Edit** menu, click **Join**.  
4 surfaces or polysurfaces joined into 1.
- 9 Repeat this for the bottom surfaces.

**Notes:**



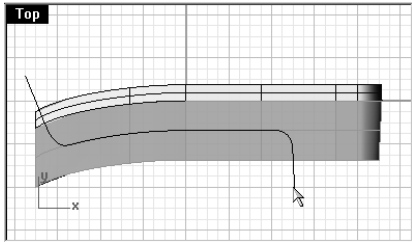
Surface from Planar Curves  
Look for this button



## Notes:

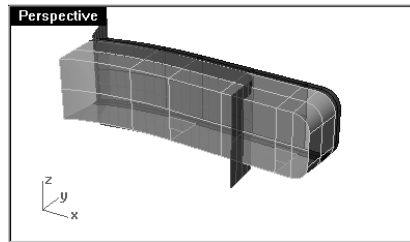
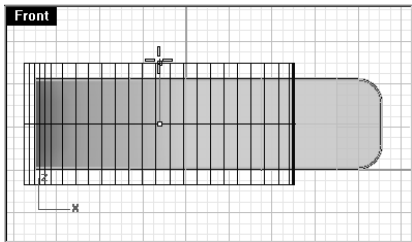
### To create an extruded surface on both sides of a curve:

- 1 Turn on the **Extrude Straight-bothsides** layer.
- 2 Select the freeform curve as shown.



- 3 From the **Surface** menu, click **Extrude**, and then click **Straight**.
- 4 At the **Extrusion distance** ( Direction BothSides=No Cap=No Mode=Straight ) prompt, click **Bothsides**.
- 5 At the **Extrusion distance** ( Direction BothSides=Yes Cap=No Mode=Straight ) prompt, in the **Front** view drag so that it extends past the object and pick.

The curve is extruded symmetrically from the curve.

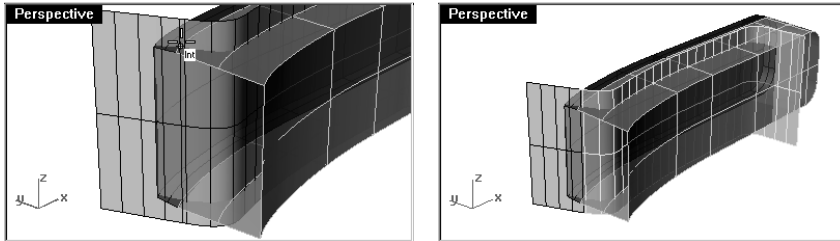


### To split the surface:

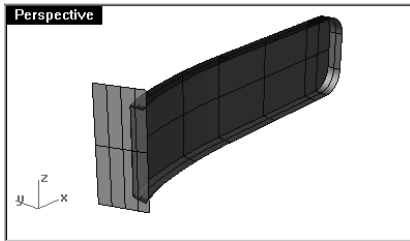
- 1 Select the extruded surface.
- 2 From the **Edit** menu, click **Split**.
- 3 At the **Select cutting objects. Press Enter to clear selection and start over** ( Isocurve ) prompt, click Isocurve.
- 4 At the **Split point** ( Direction=U ) prompt, click **Direction** to change the direction of the isocurve if necessary.
- 5 At the **Split point** ( Direction=V ) prompt, snap to the intersection where all three surfaces touch in the **Top** viewport.

## Notes:

- 6 At the **Split point** ( Direction= $V$  ) prompt, press **Enter**.



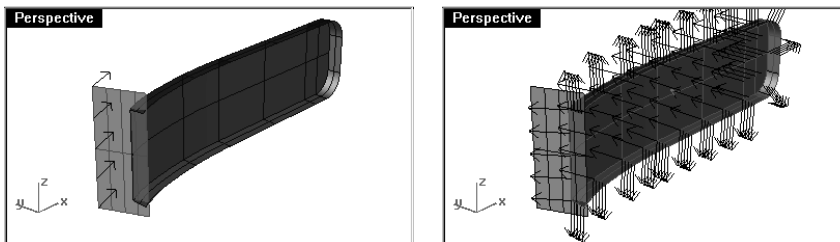
- 7 Select the left part of the split surface.
- 8 From the **Edit** menu, click **Layers**, then click **Change Object Layer**.
- 9 Make the **Top Surface** layer current and turn off the **Bottom Surface** layer.



### To join the surfaces using Boolean difference:

Next you will use a Boolean operation to join and trim the surface and the polysurface. Before you start, you should analyze the surface to see which direction the surface normals are pointing and adjust them if necessary. As a general rule, if the normals are pointing toward each other the Boolean operations will work as expected.

- 1 Select the surface.
- 2 From the **Analyze** menu, click **Direction**.
- 3 At the **Press Enter when done** ( UReverse VReverse SwapUV Flip ) prompt, click to change the direction or press **Enter** when it is pointing the direction as illustrated below.

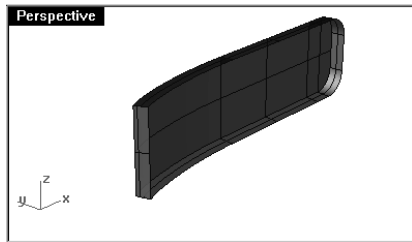


- 4 Repeat this process for the polysurface.

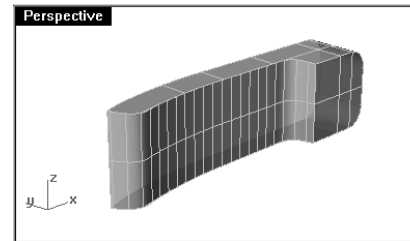
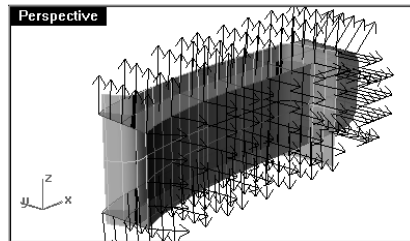
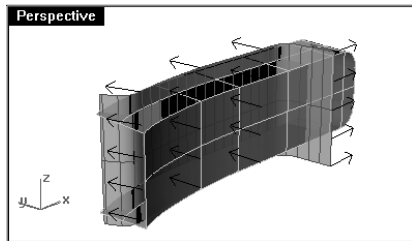
## Notes:

- 5 Select the polysurface.
- 6 From the **Solid** menu, click **Difference**.
- 7 At the **Select second set of surfaces or polysurfaces** ( DeleteInput=No ) prompt, click **DeleteInput**.
- 8 At the **Select second set of surfaces or polysurfaces. Press Enter when done** ( DeleteInput=Yes ) prompt, pick the surface and then press **Enter**.

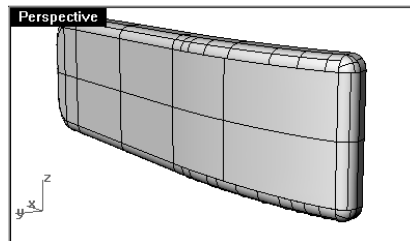
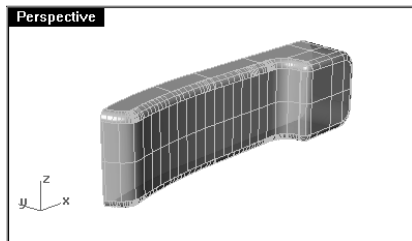
The surfaces are trimmed and joined.



- 9 Repeat this process for the bottom surfaces.



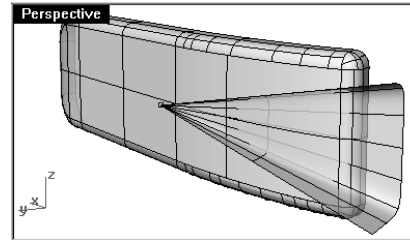
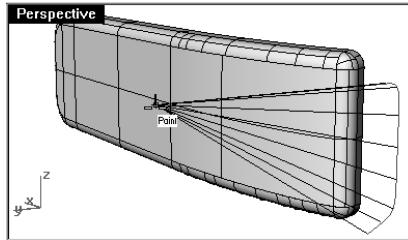
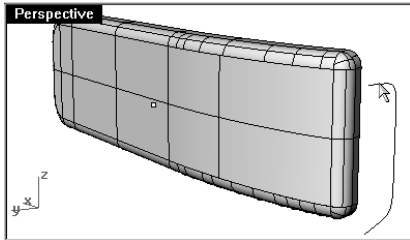
- 10 Use the **FilletEdge** command (Solid > Fillet Edge) with a radius of **.2** to round the edges.



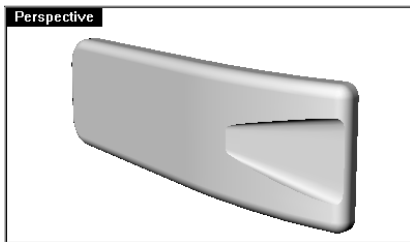
**To create an extruded surface from a curve to a point:**

- 1 Turn on the **Extrude to a point** layer.
- 2 Select the U-shaped curve.
- 3 From the **Surface** menu, click **Extrude**, and then click **To Point**.
- 4 At the **Point to extrude to** ( Mode=*ToPoint* ) prompt, select the point object near the top surface.

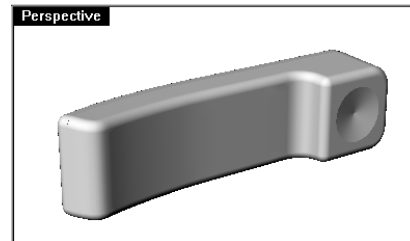
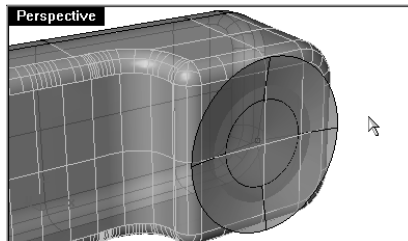
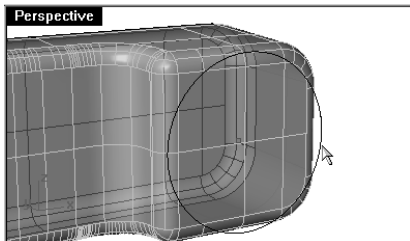
The curve is extruded to the point.



- 5 Use the **BooleanDifference** command (**Solid** menu: **Difference**) to remove the surface from the top of the phone.



- 6 Repeat these steps for the bottom part of the phone.



- 7 **Save** your model as **Phone**.

**Notes:**



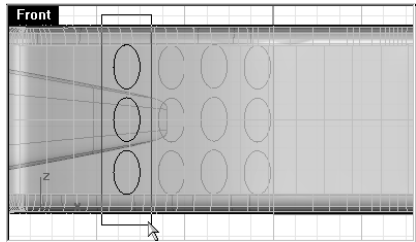
Extrude To Point  
Look for this button

## Notes:

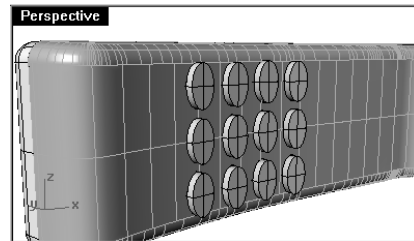
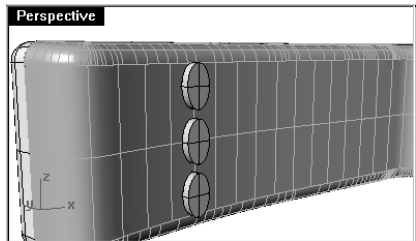
### To create the buttons:

- 1 Turn on the **Curves for Buttons** layer.
- 2 In the **Front** view, window select the first column of buttons.

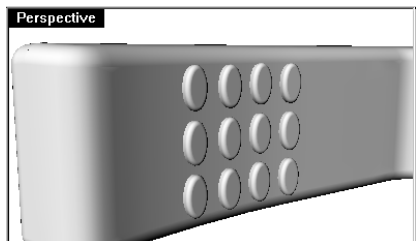
Three curves are selected.



- 3 From the **Solid** menu, click **Extrude Solid**, then click **Straight**.
- 4 At **Extrusion distance** ( Direction BothSides=*No* Cap=*Yes* Mode=*Straight* ) prompt, type **-.2** and press **Enter**.
- 5 Repeat these steps for the other columns of buttons.

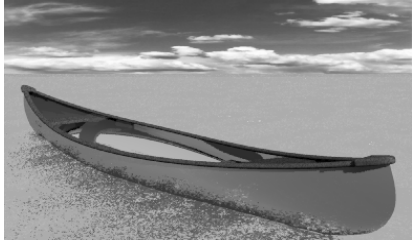


- 6 Use the **FilletEdge** command (Solid > Fillet Edge) with a radius of **.05** to round the edges.
- The button edges are rounded.

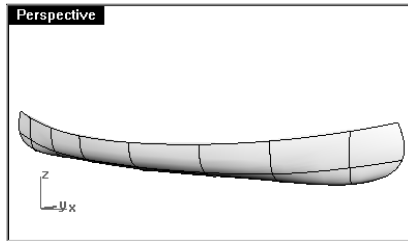
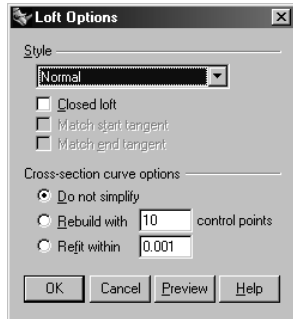


- 7 Save your model.

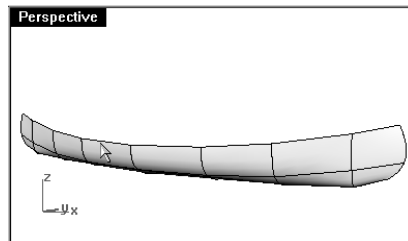
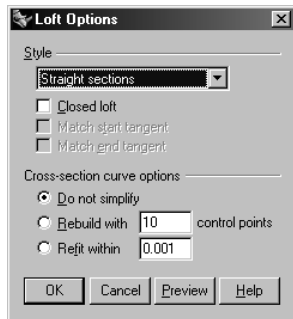
### Exercise 53—Lofted surfaces



- 1 **Open** the model **Loft.3dm**.
  - 2 Window select all of the curves.
  - 3 From the **Surface** menu, click **Loft**.
- A surface is fitted over the curves.



- 4 In the **Loft Options** dialog box, switch **Style** to **Straight sections**, then click **Preview**.
- A surface is fitted through the curves, but the sections are straight between the curves.



### Notes:



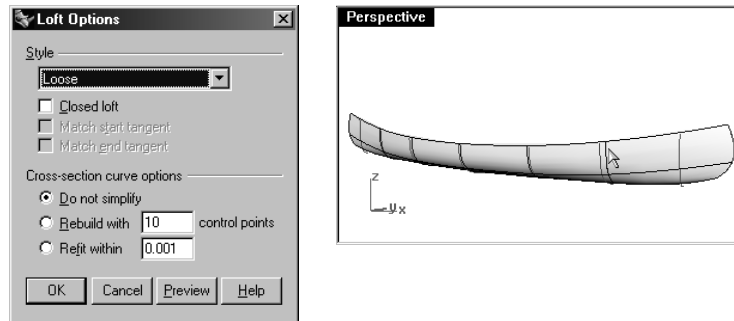
Loft  
Look for this button

## Notes:

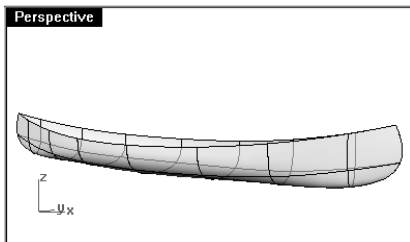
- 5 In the **Loft Options** dialog box, switch **Style** to **Loose**, then click **Preview**.

A surface is created that uses the same control points as the curves. The surface follows the curves more loosely.

Use this option when you want the surface to conform to the control points of the input curve.

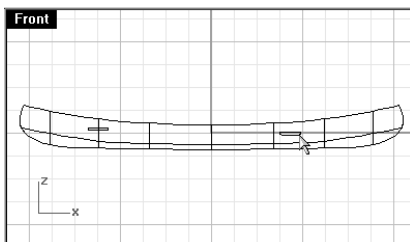


- 6 In the **Loft Options** dialog box, switch **Style** to **Normal**, then click **OK**.
- 7 **Mirror** the surface to create the other half and **Join** the two halves.



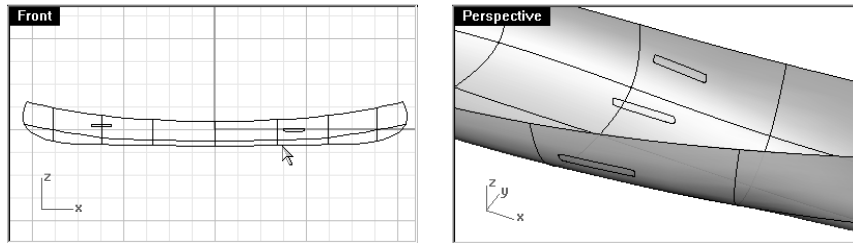
To make a seat:

- 1 Turn the **Hull Curves** layer off and turn the **Seat Curves**, and **Seat** layers on.
- 2 Make the **Seat Curves** layer current.
- 3 In the **Front** view, select the rounded rectangles.

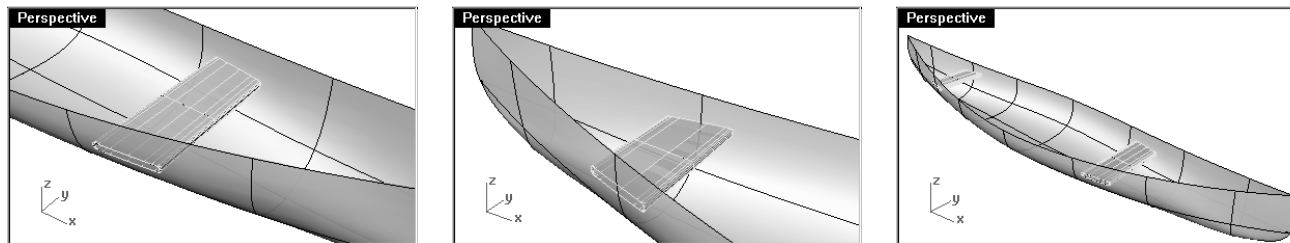


## Notes:

- 4 From the **Curve** menu, click **Curve From Objects**, and then click **Project**.
- 5 At the **Select surfaces or polysurfaces to project onto** prompt, select the hull.  
The curves will be projected to both sides of the hull surface.



- 6 Select the curves on the surface.
- 7 From the **Surface** menu, click **Loft**.
- 8 At the **Adjust curve seams** ( Flip Automatic Natural ) prompt, press **Enter**.  
A surface is fitted over the curves that fits exactly with the shape of the hull.



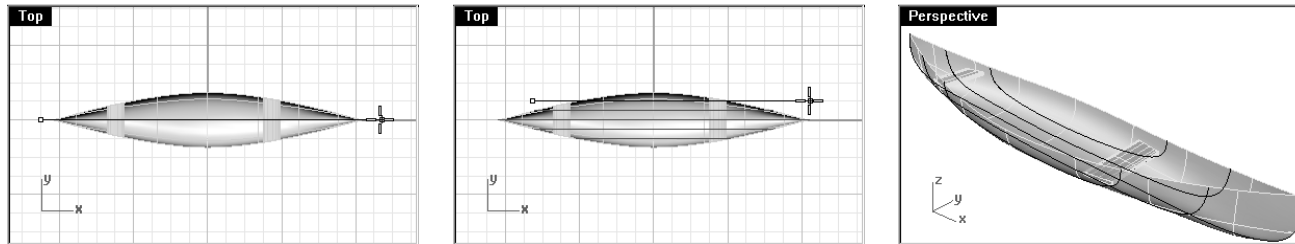
### To create section curves from the surfaces:

- 1 Select the hull.
- 2 Change to the **Section** layer.
- 3 From the **Curve** menu, click **Curve From Objects**, and then click **Section**.
- 4 At the **Start of section** prompt, in the **Top** viewport, pick a point to the left at the center of the hull.



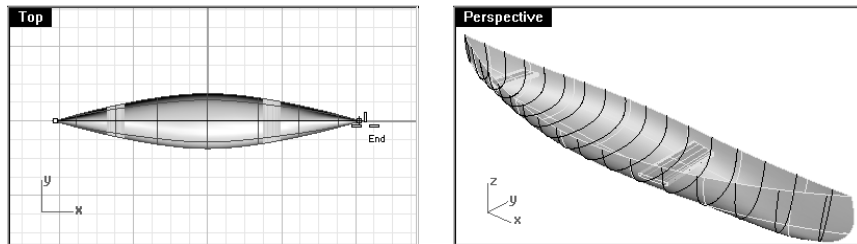
## Notes:

- 5 At the **End of section** prompt, with **Ortho** on, drag a line to the right and pick.  
A curve is generated on the surface. Repeat this at various locations.



### To create contour curves across the hull surfaces:

- 1 Select the hull.
  - 2 Change to the **Contour** layer.
  - 3 From the **Curve** menu, click **Curve From Objects**, and then click **Contour**.
  - 4 At the **Contour plane base point** ( Range ) prompt, snap to the left end of the canoe.
  - 5 At the **Direction perpendicular to contour planes** prompt snap to the right end of the canoe.
  - 6 At the **Distance between contours <1>** prompt, press **Enter**.
- A curve is generated every foot along the hull.

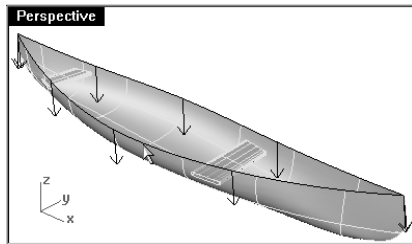


## Notes:

### To create an edge curve from the surfaces:

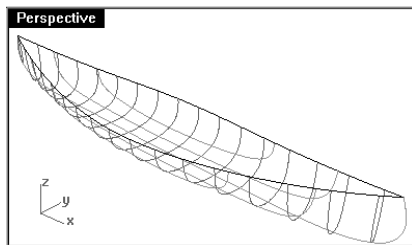
- 1 Change to the **Top Rail** layer.
- 2 From the **Curve** menu, click **From Objects**, and then click **Duplicate Edge**.
- 3 At the **Select edges to duplicate** prompt, pick the top edge of the hull.
- 4 At the **Select edges to duplicate. Press Enter when done** prompt, pick the other top edge and press **Enter**.

A curve is generated at the edge of the surface.



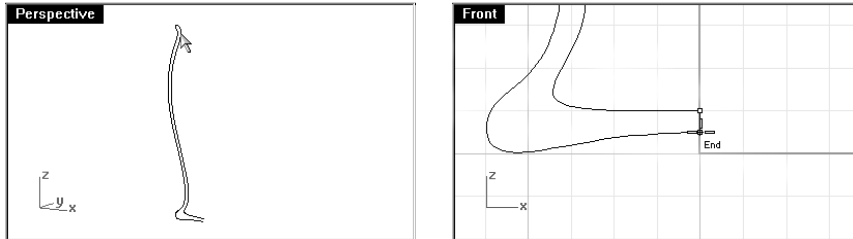
- 5 Turn off the **Hull** layer.

You have a wireframe of curves that duplicate the hull surface.

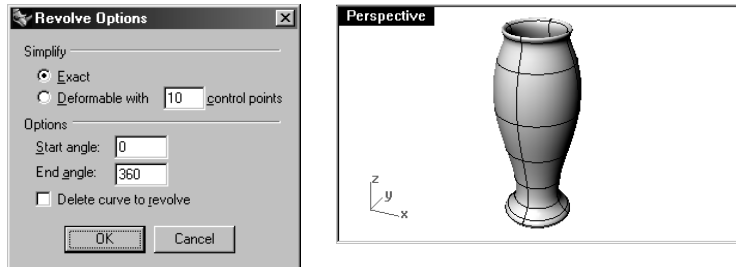


### Exercise 54—Revolved surfaces

- 1 **Open** the model **Revolve.3dm**.
- 2 Select the free-form curve
- 3 From the **Surface** menu, click **Revolve**.
- 4 At the **Start of revolve axis** prompt, select one end of the curve.
- 5 At the **End of revolve axis** prompt, select the other end of the curve.



- 6 In the **Revolve Options** dialog box, click **OK**.  
A surface is revolved around the axis line.



### Notes:



Revolve  
Look for this button

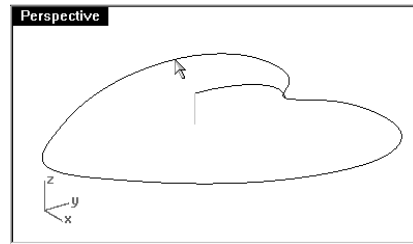
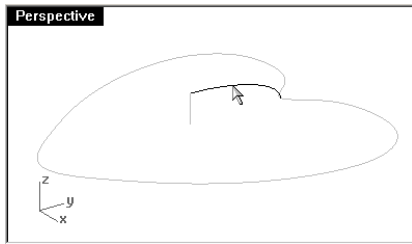
## Notes:

### Exercise 55—Using a rail revolve

Rail Revolve lets you revolve around an axis and along a path curve.

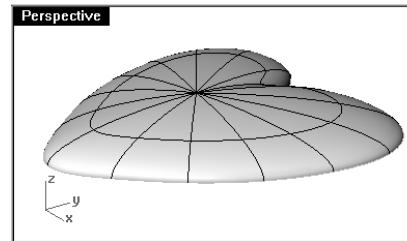
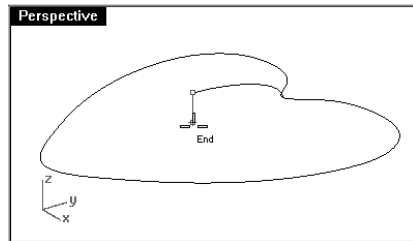
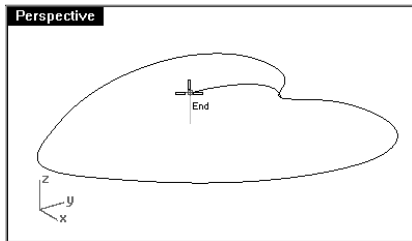
#### To create a rail revolve:

- 1 **Open** the model **Rail Revolve.3dm**.
- 2 Select the conic shaped curve.
- 3 From the **Surface** menu, click **Rail Revolve**.
- 4 At the **Select rail curve** ( ScaleHeight=*No* ) prompt, select the path curve.

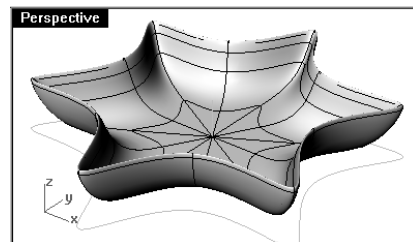
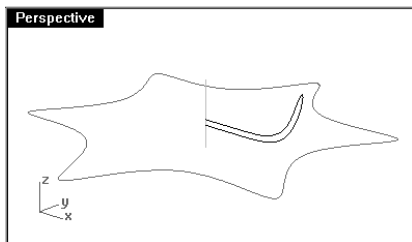


- 5 At the **Start of RailRevolve axis** prompt, select one end of the axis line.
- 6 At the **End of RailRevolve axis** prompt, select the other end of the axis line.

A surface is revolved around the axis that follows along the curve at the end of the object.



- 7 Turn the **Bowl** layer on and other layers off.
- 8 Use **Rail Revolve** to make a bowl.

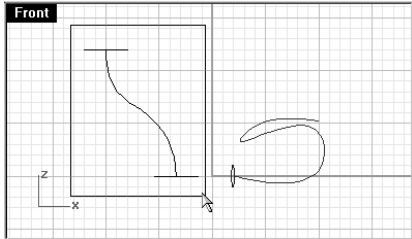


Rail Revolve

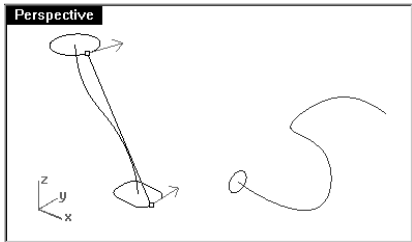
## Notes:

### Exercise 56—Using 1-rail sweeps to create surfaces

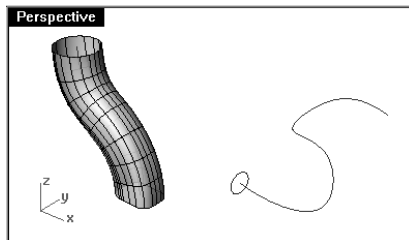
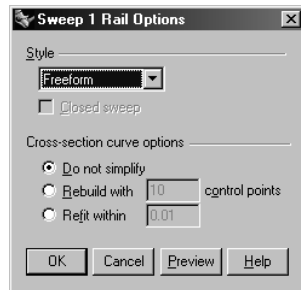
- 1 **Open** the model **1 Rail Sweep.3dm**.
- 2 Select the three curves on the left.



- 3 From the **Surface** menu, click **Sweep 1 Rail**.
- 4 At the **Adjust curve seams** ( Flip Automatic Natural ) prompt, press **Enter**.



- 5 In the **Sweep 1 Rail Options** dialog box, click **OK**.
- The two cross section shapes are blended along the rail curve to form the surface.



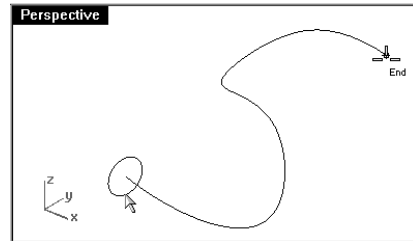
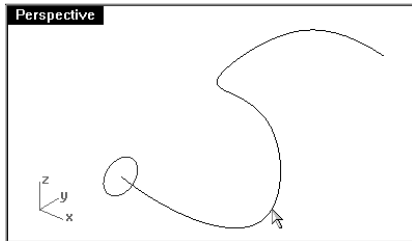
Sweep along 1 Rail  
Look for this button

### To create a 1-rail sweep to a point:

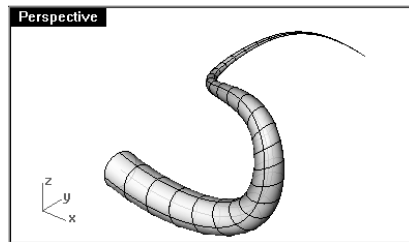
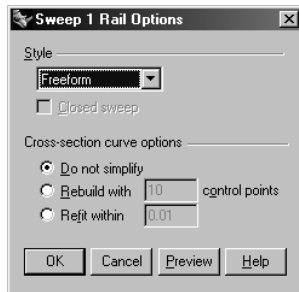
- 1 From the **Surface** menu, click **Sweep 1 Rail**.
- 2 At the **Select rail curve** prompt, select the free-form curve on the right.
- 3 At the **Select cross section curves** ( Point ) prompt, select the circle.

**Notes:**

- 4 At the **Select cross section curves** ( Point ) prompt, click **Point**.
- 5 At the **Pick end point** prompt, select the other end of the free-form curve.



- 6 In the **Sweep 1 Rail Options** dialog box, then click **OK**.  
The cross-section will sweep to a point to form the surface.



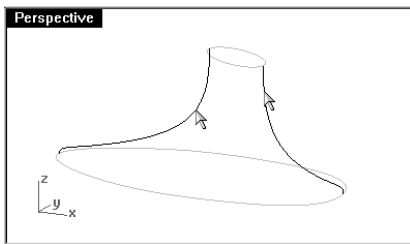
### Exercise 57—Using 2-rail sweeps to create surfaces

- ▶ **Open** the model **2 Rail Sweep.3dm**.

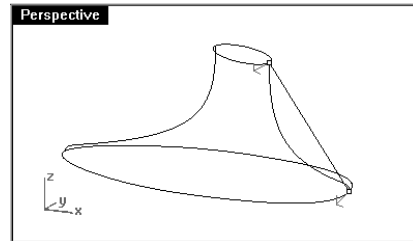
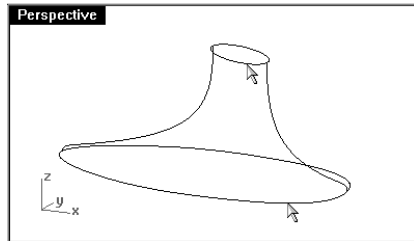


#### To create the base part:

- 1 Change to the **Base Surface** layer.
- 2 From the **Surface** menu, click **Sweep 2 Rails**.
- 3 At the **Select rail curves** prompt, select the two profile curves.



- 4 At the **Select cross section curves** ( Point ) prompt, select the cross-section curves.
- 5 At the **Select cross section curves** ( Point ) prompt, press **Enter**.
- 6 At the **Adjust curve seams** ( Flip Automatic Natural ) prompt, press **Enter**.



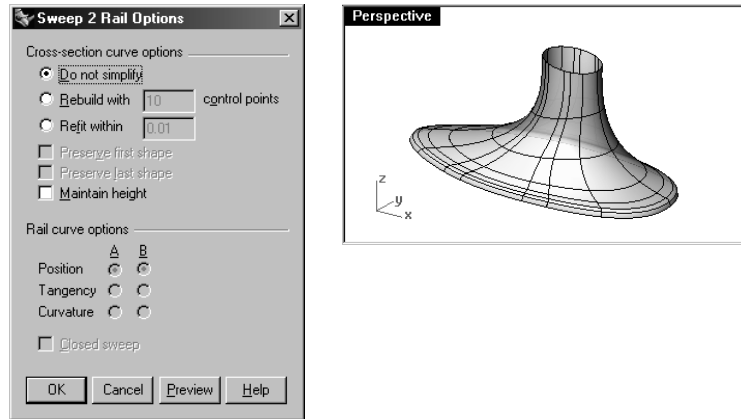
#### Notes:



Sweep along 2 Rails  
Look for this button

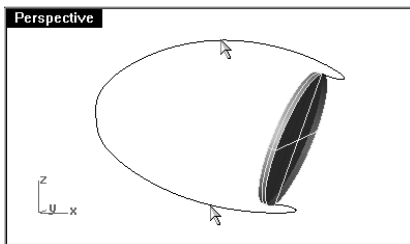
## Notes:

- 7 In the **Sweep 2 Rails Options** dialog box, click **OK**.  
A surface is created whose edges match the rail curves.



### To create the housing:

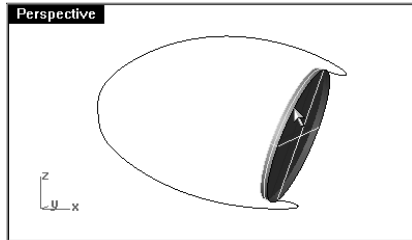
- 1 Turn the **Housing Surface**, **Housing Curves**, and **Mirror** layers on.
- 2 Make the **Housing Surface** layer current.
- 3 From the **Surface** menu, click **Sweep 2 Rails**.
- 4 At the **Select rail curves** prompt, select the two rail curves.



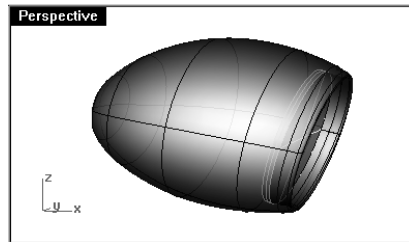
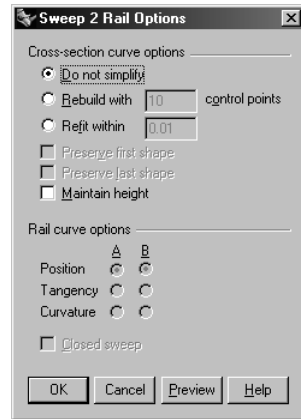


**Notes:**

- At the **Select cross section curves** ( Point ) prompt, select the outer edge of the cylinder, and then press **Enter**.
- At the **Adjust curve seams** ( Flip Automatic Natural ) prompt, press **Enter**.



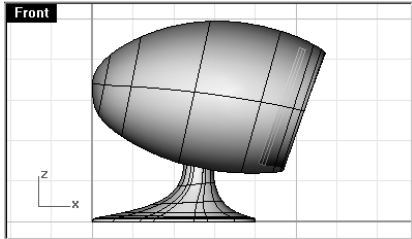
- In the **Sweep 2 Rails Options** dialog box, click **OK**.  
A surface is created.



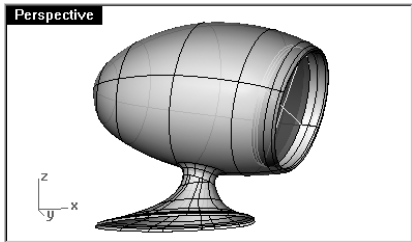
## Notes:

### To join the two parts:

- 1 Select the base and the housing surfaces.
- 2 From the **Solid** menu, click **Union**.  
The two parts are joined and trimmed.



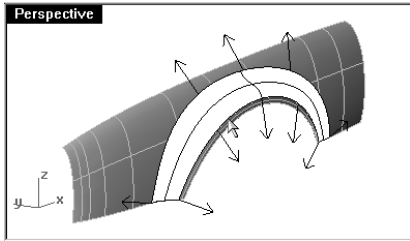
- 3 Use the **FilletEdge** command (**Solid** menu: **Fillet Edge**) with a radius of **.25** to round the intersecting edge.



## Notes:

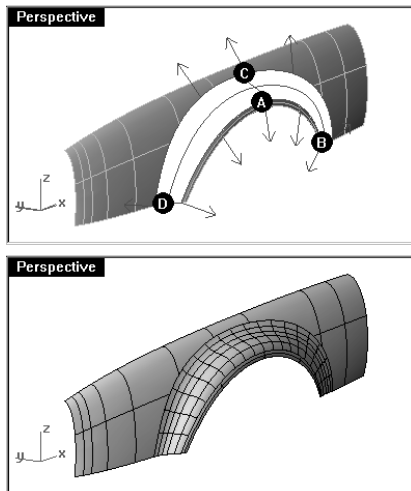
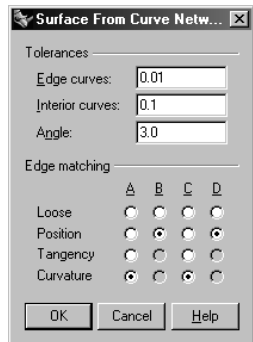
### Exercise 58—Using a network of curves to create surfaces

- 1 **Open** the model **Networksurf.3dm**.
- 2 From the **Surface** menu, click **Curve Network**.
- 3 At the **Select curves in network** ( NoAutoSort ) prompt, select the two edge curves and the cross-section curves, and press **Enter**.



- 4 In the **Surface From Curve Network** dialog box, change the edge matching to **Curvature**, click **OK**.

A surface is created that has curvature continuity with the other two surfaces.



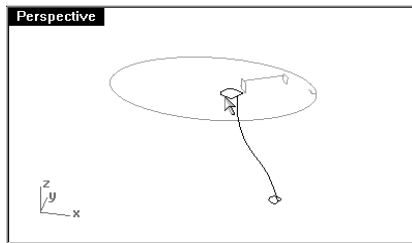
## Notes:

### Exercise 59— Practice using one-rail sweeps:

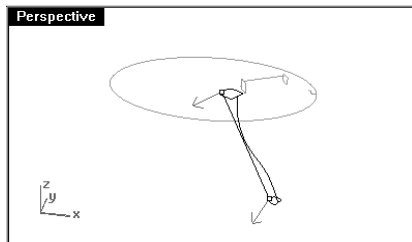
In this exercise you use one rail sweeps to make an end table with free-form tapered legs.

#### To create the legs:

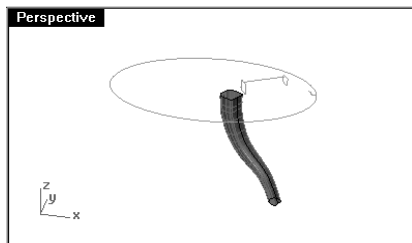
- 1 **Open** the model **Table.3dm**.
- 2 From the **Surface** menu, click **Sweep 1 Rail**.
- 3 At the **Select rail curve** prompt, select the path curve for the leg.
- 4 At the **Select cross section curves** ( Point ) prompt, select the shape curve for both ends of the leg.
- 5 At the **Select cross section curves. Press Enter when done** ( Point ) prompt, press **Enter**.



- 6 At the **Adjust curve seams** ( Flip Automatic Natural ) prompt, press **Enter**.



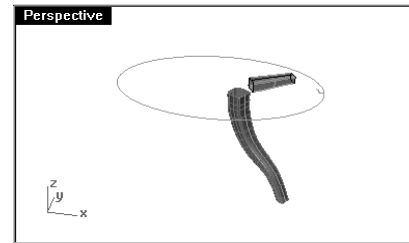
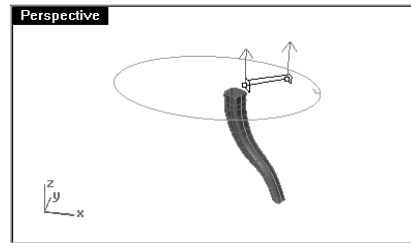
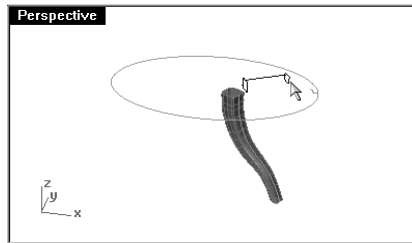
- 7 In the **Sweep 1 Rail Options** dialog box, click **OK**.  
The table leg is created. Note the nice transition from one cross-section curve to the other.



## Notes:

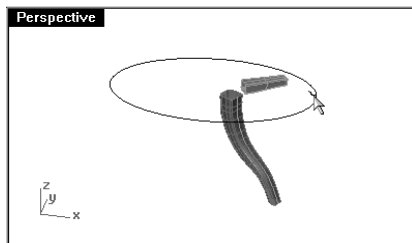
### To create the brace:

- 1 Change to the **Braces** layer.
- 2 Repeat the previous process to create the brace.

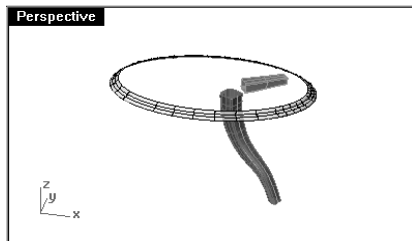


### To create the top:

- 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.
- 5 At the **Select cross section curves. Press Enter when done** ( Point ) prompt, press **Enter**.



- 6 In the **Sweep 1 Rail Options** dialog box, click **OK**.  
The surface for the rim of top is created.

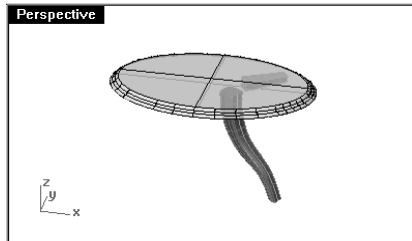


## Notes:

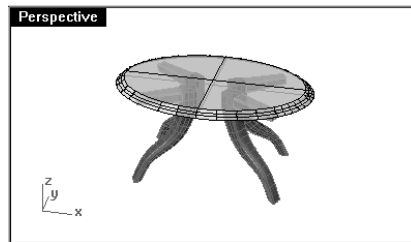
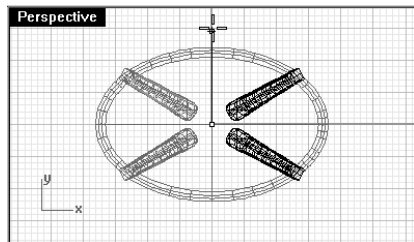
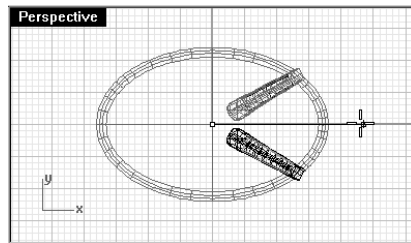
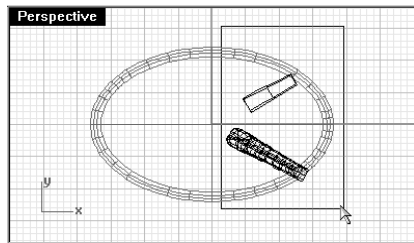
### To finish the table:

- 1 Select all of the surfaces you created.
- 2 From the **Solid** menu, click **Cap Planar Holes**.

Six caps were created.



- 3 Use **Mirror** to copy the brace and the leg to finish the model.  
Mirror them around **0,0** in the **Top** viewport.

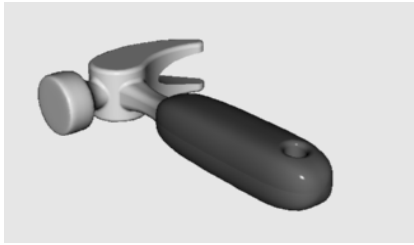


## Notes:

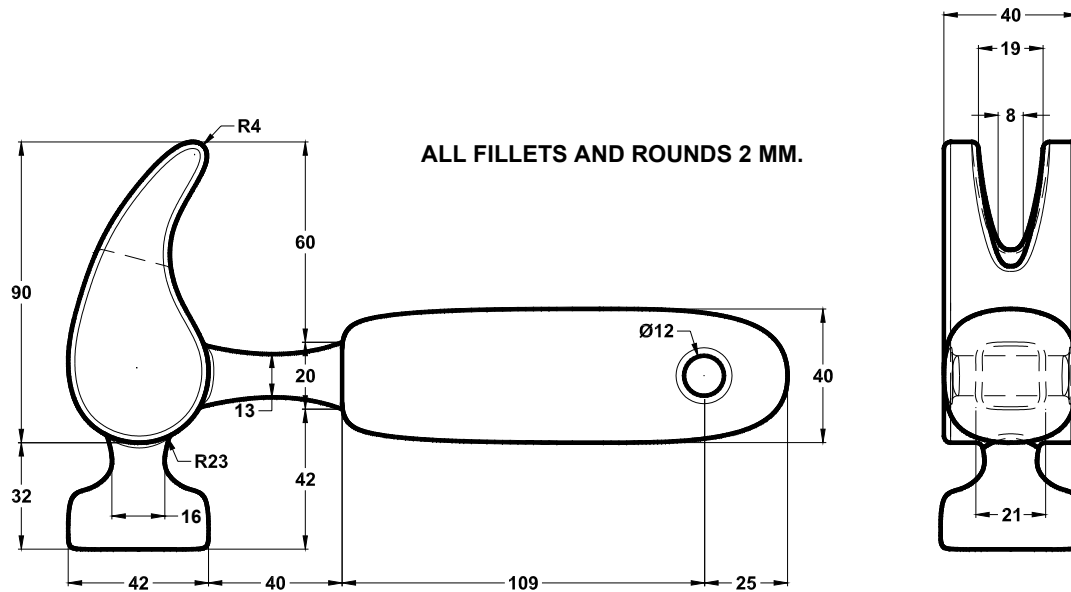
### Exercise 60— Creating a toy hammer:

In this exercise you will use most of the techniques that you've learned in the previous sessions.

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.



- 1 Open the model **Hammer**.
- 2 The following layers have been created: **Construction, Curve, Handle, Tang, Head, Hole, and Claw**.



- 3 Draw outlines for the hammer in the **Top** viewport.

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, arcs, and curves. You can trim the circles and arcs and then join them together to create a closed curve. You can rebuild the curve and adjust the control points to get a more sculptural shape.

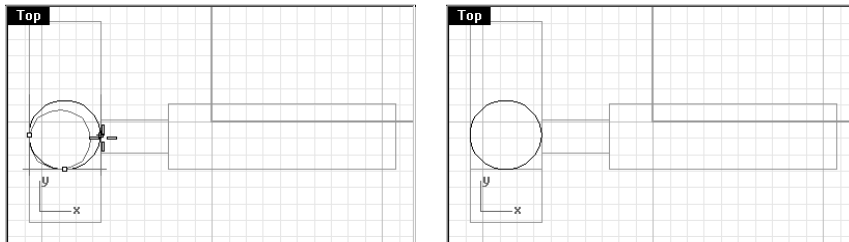
- 1 Change to the **Curve** layer
- 2 Draw a curve defining the shape of the claw in the **Top** viewport.

You can use a free-form curve or use a combination of arcs and circles that 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 using arcs and circles.

Start by drawing two circles.

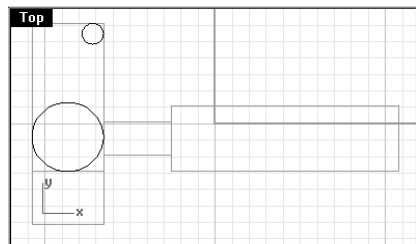
- 3 Use the **Circle** command (Curve menu: Circle > Tangent to 3 curves) to create a circle at the lower end of the claw.

Draw the circles tangent to the construction geometry.



- 4 Use the **Circle** command (Curve menu: Circle > Tangent, Tangent, Radius) to create a circle at the upper end of the claw that is tangent to the upper right corner with a 4 mm radius.

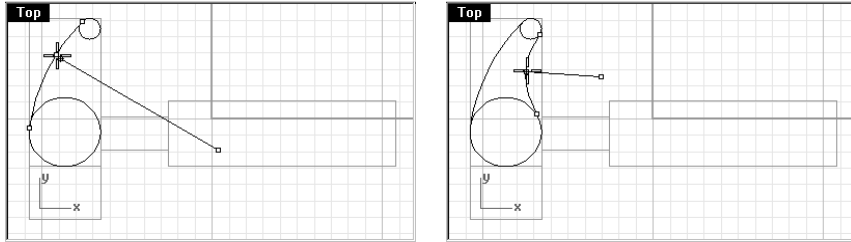
Draw the circles tangent to the construction geometry.



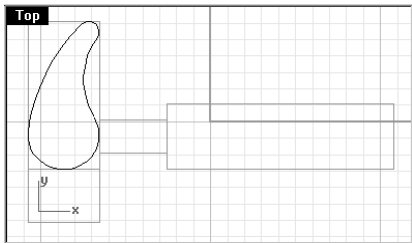


## Notes:

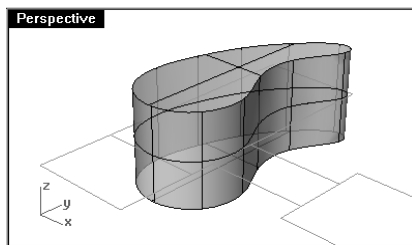
- 5 Use the **Arc** command (Curve menu: Arc > Tangent, Tangent, Radius) to create arcs that are tangent to the two circles.



- 6 Use the **Trim** command (Edit menu: Trim) to trim the inside part of the circles.



- 7 Use the **Join** command (Edit menu: Join) to join the arc segments.
- 8 Change to the **Claw** layer.
- 9 Select the joined segments.
- 10 Use the **Extrude** command (Solid menu: Extrude > Straight) to extrude the curve on both sides of the construction plane.

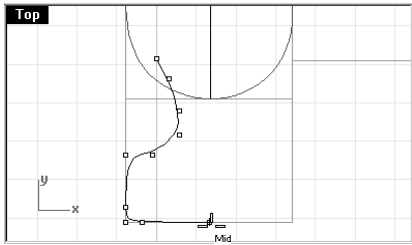


## Notes:

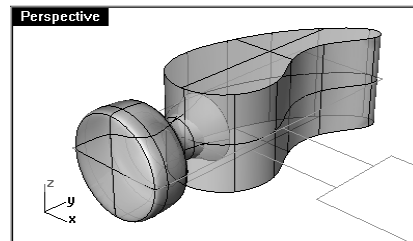
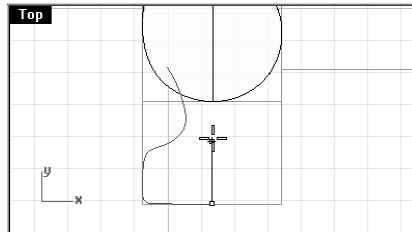
### To create the head:

- 1 Change to the **Curve** layer.
- 2 Use the **Curve** command (Curve menu: Free-form > Control Points) to create the curve for the cross-section of the head.

Make sure the curve intersects the claw part. This makes joining the two pieces easier.



- 3 Change to the **Head** layer.
  - 4 Use the **Revolve** command (Surface menu: Revolve) to revolve the curve.
- Use the midpoint of the construction line for the revolve axis.

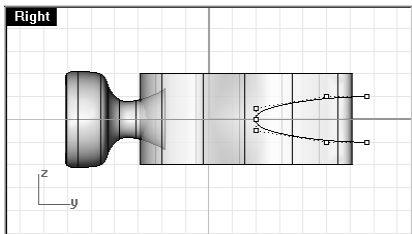


- 5 Save your model.

### To create slot for the claw part of the hammer:

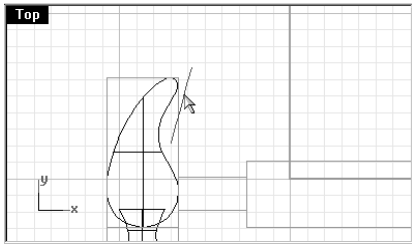
- 1 Use the **Curve** command (Curve menu: Free-form > Control Points) to draw a curve for the slotted part of the claw.

Make sure the curve is symmetrical.

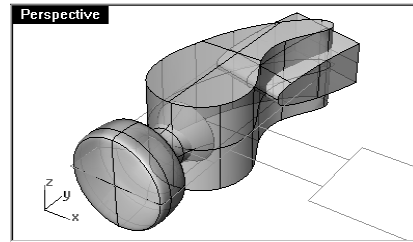
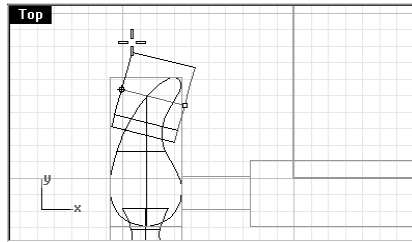


## Notes:

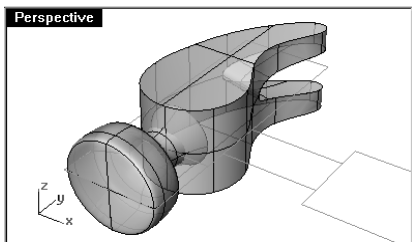
- 2 Use the **Line** command (Curve menu: Line > Single Line) to draw a line between the endpoints.
- 3 Use the **Join** command (Edit menu: Join) to join the curve and the line.
- 4 Drag the closed curve closer to the claw.
- 5 Use the **Rotate** command (Transform menu: Rotate) to rotate the curve to align more closely with the curve of the claw.



- 6 Change to the **Claw** layer.
- 7 Use the **Extrude** command (Solid menu: Extrude > Straight) to extrude the curve through the claw.

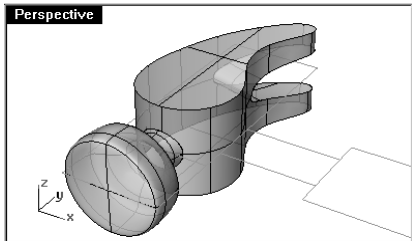


- 8 Save your model.
- 9 Use the **BooleanDifference** command (Solid menu: Difference) to subtract the slot from the claw.

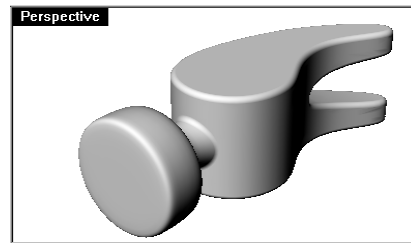
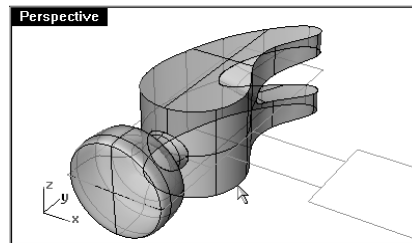


**To finish the claw:**

- 1 Use the **BooleanUnion** command (Solid menu: Difference) to join the head with the claw.



- 2 Use the **FilletEdge** command (Solid menu: Fillet Edge) to make the fillets around the top and bottom of the claw, the slot, and the intersection between the head and the claw.



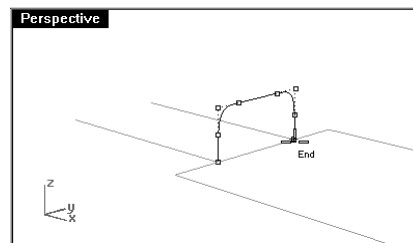
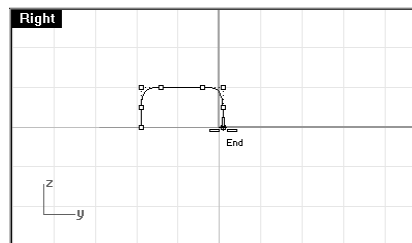
- 3 Save your model.

**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 Change to the **Curve** layer and turn on **Ortho**.
- 2 Use the **Curve** command (Curve menu: Free-form > Control Points) to draw a curve for the upper cross-section of the tang.

Make sure the curve is symmetrical.

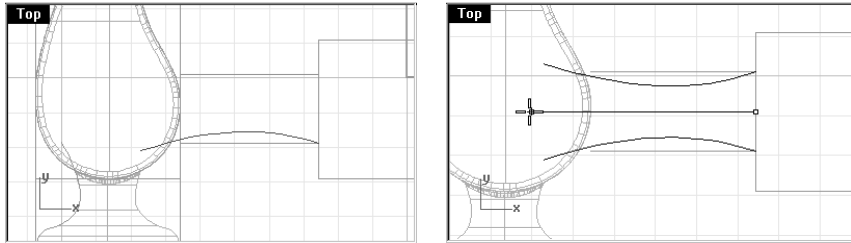


- 3 Save your model

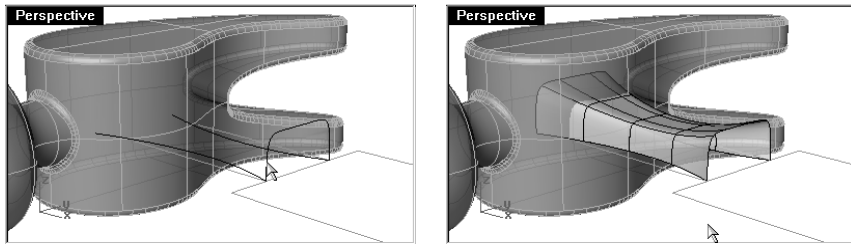
**To create the tang:**

- 1 Use the **Curve** command (Curve menu: Free-form > Control Points) to draw one of the curves for the tang of the hammer.

Make sure that it intersects the claw.

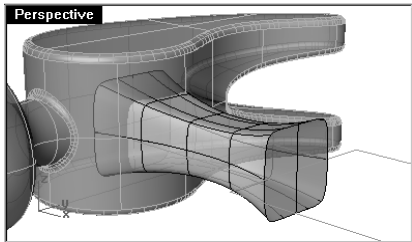


- 2 Use the **Mirror** command (Transform menu: Mirror) to create the other curve.
- 3 Change to the **Tang** layer.
- 4 Use the **Sweep2** command (Surface menu: Sweep 2 Rails) to make the surface.



**To finish the tang:**

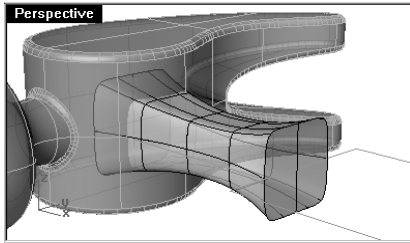
- 1 Use the **Mirror** command (Transform menu: Mirror) to make the other half of the tang.



- 2 Select both halves.
- 3 Use the **Join** command (Edit menu: Join) to join the two surfaces.
- 4 Select the joined surfaces.

## Notes:

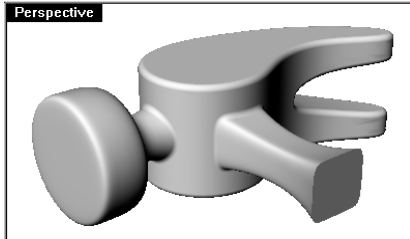
- 5 Use the **Cap** command (Solid menu: Cap Planar Holes) to make the tang a closed polysurface.



- 6 Save your model.

### To finish the hammer head:

- 1 Select the tang and the claw.
- 2 Use the **BooleanUnion** command (Solid menu: Difference) to join the tang with the claw and the head.
- 3 Use the **FilletEdge** command (Solid menu: Fillet Edge) to make the fillets at the intersection between the tang and the claw.  
The edge has a round on it.

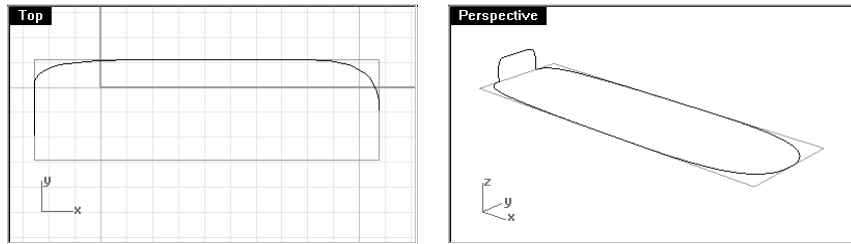


- 4 Save your model.

**To create the handle:**

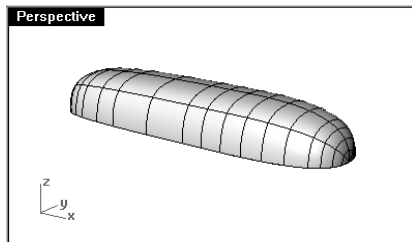
- 1 Change to the **Curve** layer.
- 2 Use the **Curve** command (Curve menu: Free-form > Control Points) to draw a curve for the top edge of the handle.

Make it start at the endpoint of the tang profile curve and end on the centerline.

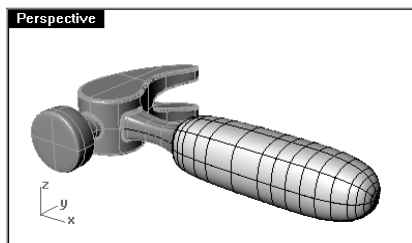


- 3 Use the **Mirror** command (Transform menu: Mirror) to make the other half.
- 4 Change to the **Handle** layer.
- 5 Use the **Sweep2** command (Surface menu: Sweep 2 Rails) to make the surface using the tang curve as the profile curve.

A surface is created.



- 6 Select the surface.
- 7 Use the **Mirror** command (Transform menu: Mirror) to make the other half.
- 8 Select both halves.
- 9 Use the **Join** command (Edit menu: Join) to join the two surfaces.



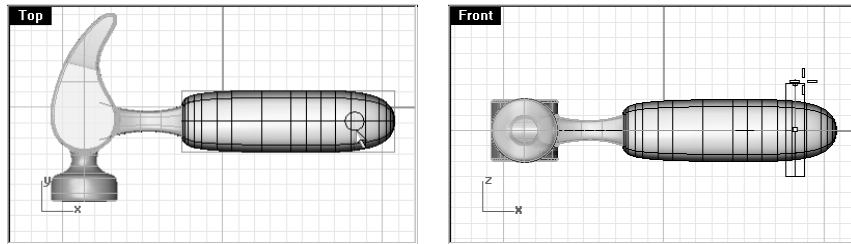
## Notes:

- 10 Use the **Cap** command (Solid menu: Cap Planar Holes) to cap the open end.
- 11 Save your model.

### To create the hole for the handle:

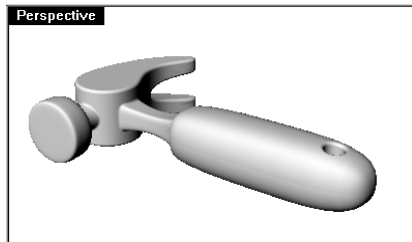
- 1 Use the **Circle** command (Curve menu: Circle > Center, Radius) to make a circle 25mm from the end of the handle.  
You may have to draw a construction line to help you position the circle.
- 2 Use the **Extrude** command (Solid menu: Extrude > Straight) to extrude the curve on both sides of the construction plane.

Make sure the extrusion intersects both sides of the handle.



- 3 **Save** your model.
- 4 Use the **BooleanDifference** command (Solid menu: Difference) to subtract the hole from the handle.
- 5 Use the **FilletEdge** command (Solid menu: Fillet Edge) to make the fillets at the edges of the hole.

The edges have rounds on them.



- 6 **Save** your model.

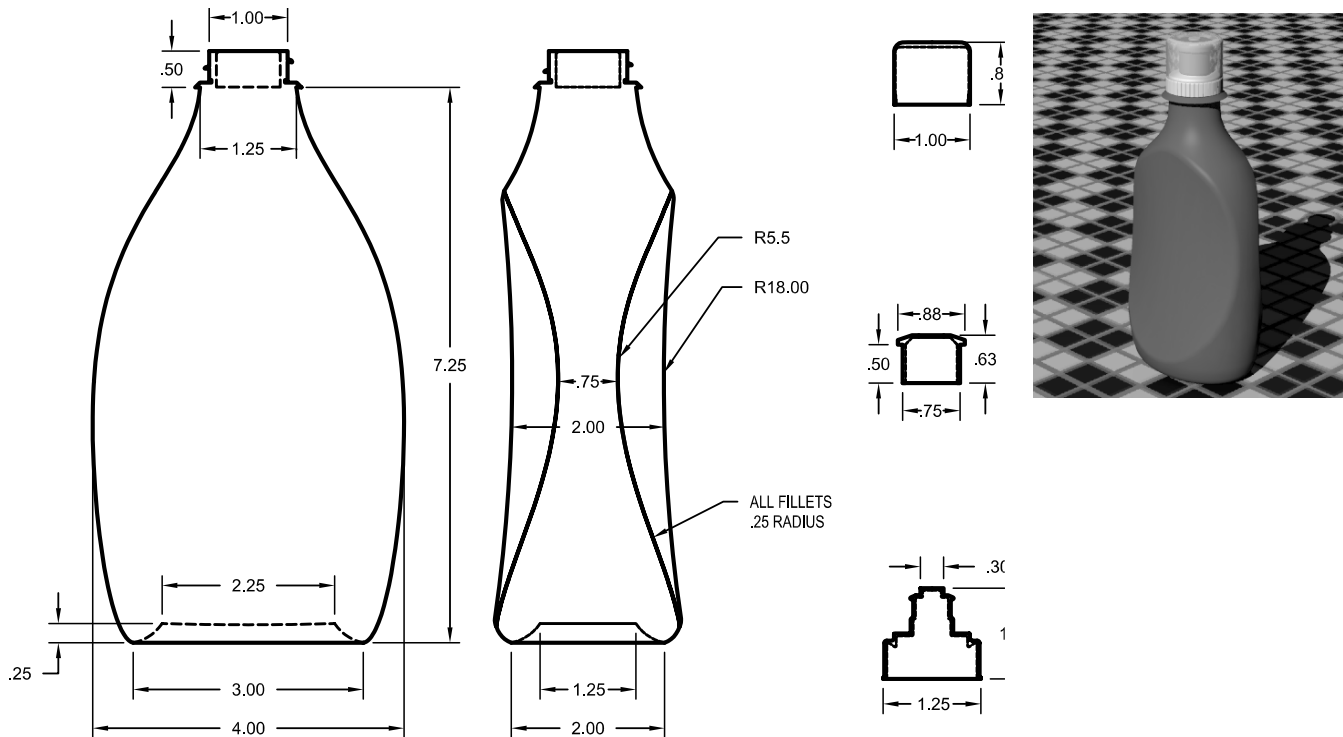


## Notes:

### Exercise 61— Creating 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.



- 1 Start a new model. Use the **Inches.3dm** template.
- 2 Save as **Bottle**.
- 3 Make the following layers: **Construction**, **Curve**, **Bottle**, **BottleTop**, **Threads**, **Cap1**, **Cap2**, and **CapTop**.
- 4 Change to the **Construction** layer.

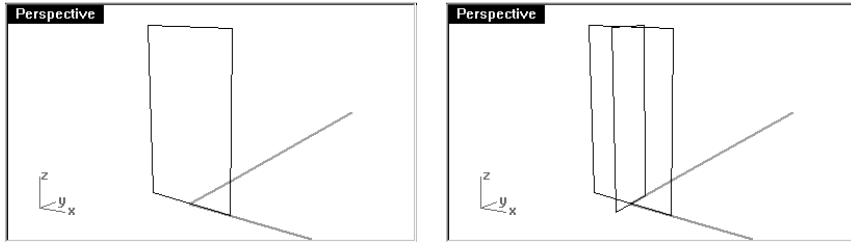
## Create the bottle shape

To create the bottle shape, you will draw curves that defines the bottom and top edge, and then draw a curve that represents the side shape. You will then create a surface from these defining curves.

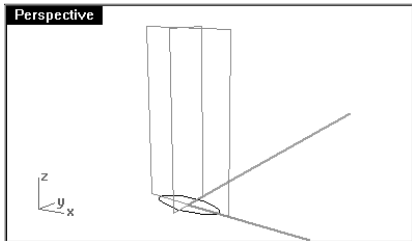
### To draw the construction curves:

- 1 Use the **Rectangle** command (Curve menu: Rectangle > Corner to Corner) to make rectangles in the Front and Right views that define the overall size of the bottle shape of the bottle.

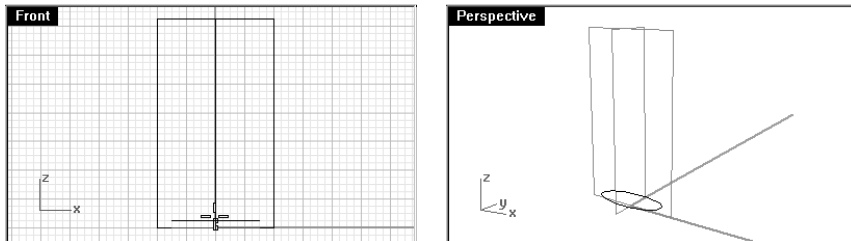
The rectangles will be used as guides for building the bottle curves.



- 2 Use the **Lock** command (Edit menu: Visibility > Lock) to lock the two rectangles.
- 3 Use the **Ellipse** command (Curve menu: Ellipse > From Center) to make the shape for the bottom of the bottle.
- 4 Snap to the intersection of the rectangles for the center of the ellipse.



- 5 Use the **Move** command (Transform menu: Move) to move the ellipse up **.25** in the **Front** view.



### Notes:



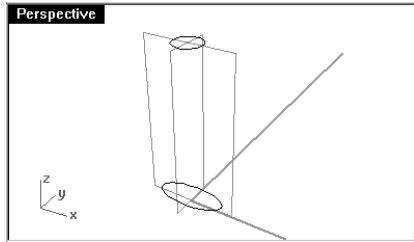
Rectangle



Ellipse

## Notes:

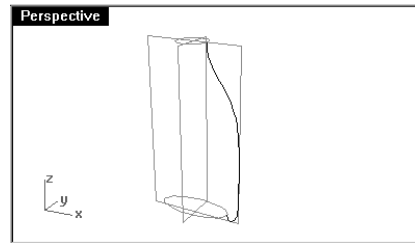
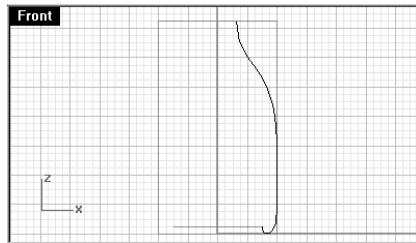
- 6 Use the **Circle** command (Curve menu: Circle > Center, Radius) to draw a circle for the top shape of the bottle.
- 7 Snap to the intersection at the top of the rectangles for the center of the circle.



### To draw the edge curve:

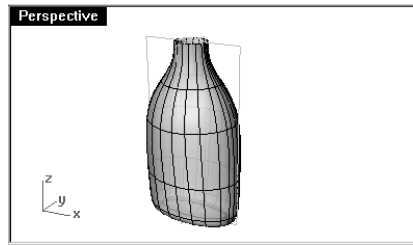
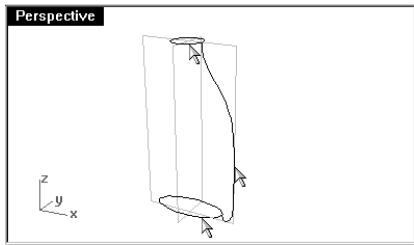
- 1 Change to the **Curve** layer.
- 2 Use the **Curve** command (Curve menu: Free-Form > Control Points) to define the edge of the bottle in the **Front** viewport.

Use the rectangle and object snaps to help you establish the correct size. 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 Use the **Sweep2** command (Surface menu: Sweep 2 Rails) to make the bottle surface. The ellipse and the circle will be the rails. The curve you created will be the shape curve.
- 3 In the **Sweep 2 Rails Options** dialog box, click **OK**.



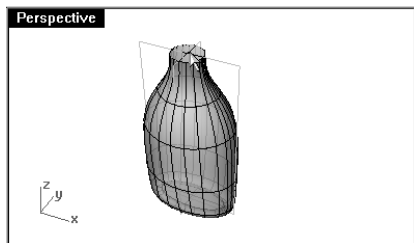
### Cap the top and bottom

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 specified volume.

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 Select the surface.
- 2 Use the **Cap** command (Solid menu: Cap Planar Holes) to close the holes.



### Notes:



Two-Rail Sweep



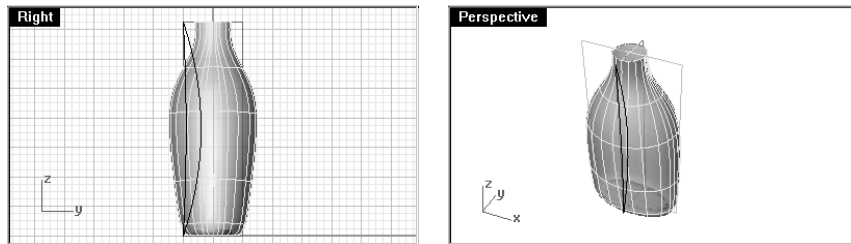
Cap Planer Holes

## Flatten the sides

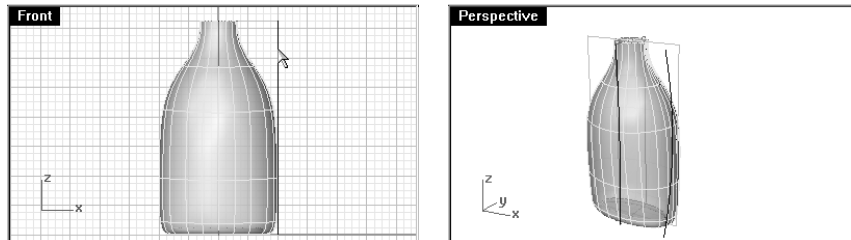
You will notice in the **Right** viewport that the bottle bulges too much. You will create custom surfaces to trim the bulges away.

### To create the trimming surface:

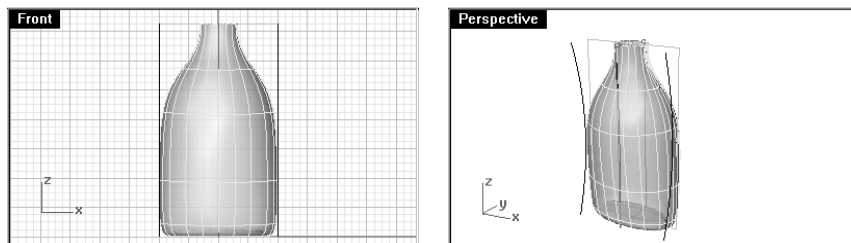
- 1 Change to the **Curve** layer.
- 2 In the **Right** viewport, draw two curves to approximate the shape at the edge of the bottle and the middle of the bottle.
- 3 The technical drawing gives dimensions for these curves, but for this exercise you can use the construction rectangles to help. If you 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 more curvature to align with the edge of larger rectangle.

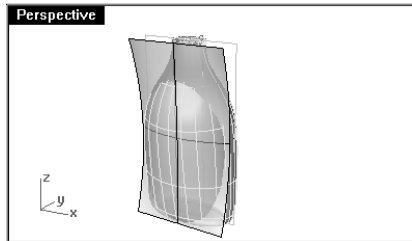


- 5 **Mirror** the edge curve to the opposite side.

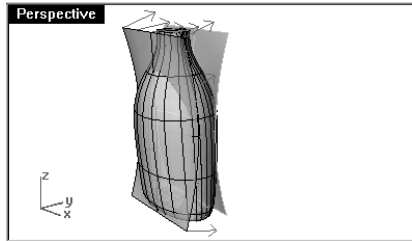


- 6 Select the three curves you just created.
- 7 Use the **Loft** command (Surface menu: Loft) to make the cutting surface.
- 8 In the **Loft Options** dialog box, click **OK**.

A lofted surface intersects the bottle.



- 9 **Mirror** the surface to the other side of the bottle.

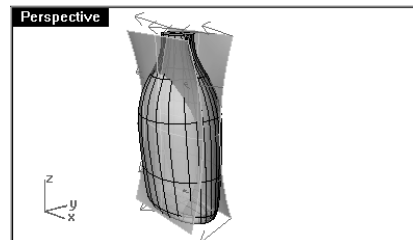
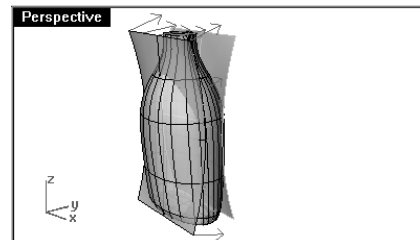


- 10 Save your model.

#### To remove the surface from the bottle:

- 1 Change to the **Bottle** layer.
- 2 Use the **Dir** command (Analyze menu: Direction) to check the surface normal direction. Flip the normals if necessary.

The normals should be pointing toward the center of the bottle.



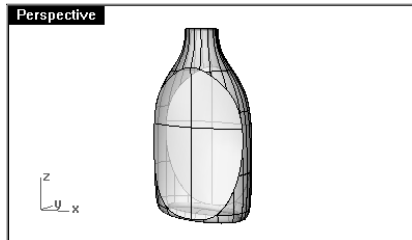
#### Notes:



Loft

- 3 Select the bottle.
- 4 Use the **BooleanDifference** command (Solid menu: Difference) to subtract the two lofted surfaces from the bottle.

The bottle is now a closed solid.



- 5 Use the **FilletEdge** command (Solid menu: Fillet Edge) to round off the sharp edges by adding a smooth radius between the edges.



## Notes:



Boolean Difference

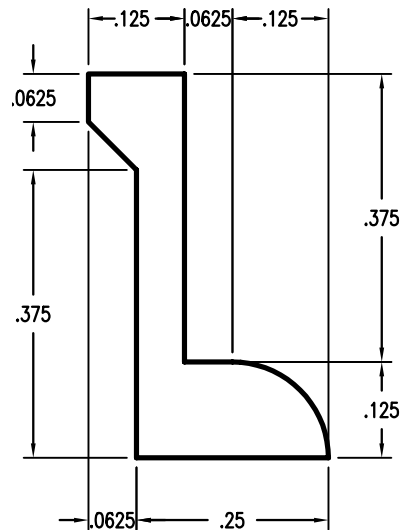
## Notes:

### Create the bottle's top

To create the bottle throat, you are going to revolve a profile curve to create the surface, and then subtract threads.

#### To create the profile curve:

- 1 Change to the **Curve** layer.
- 2 In the **Front** viewport, use the **Lines** command (Curve menu: Line > Line Segments) and the **Arc** command (Curve menu: Arc > Center, Start, Angle) to create a profile curve for the outside and the inside of the top.
- 3 Use the drawing below to draw the correct dimensions.



- 4 Begin the drawing anywhere in the viewport.  
You will move it to a precise location after it is made.
- 5 Use the **Join** command (Edit menu: Join) to join the segments together.

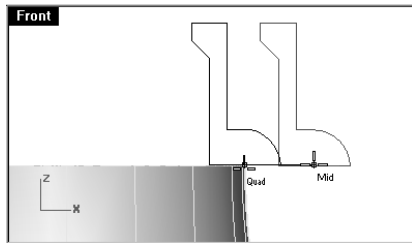




## Notes:

**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 and click. The line stops 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.

- 6 Use the **Move** command (Transform menu: Move) to move the profile from the midpoint of the bottom edge to the quadrant of the top of the bottle.

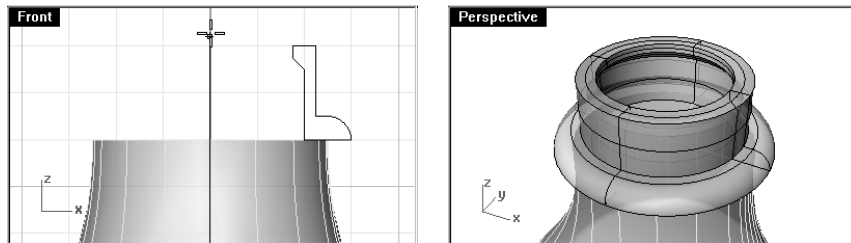


- 7 Change to the **Bottle Top** layer.

### To create the top surface:

- 1 Select the profile curve.
- 2 Use the **Revolve** command (Surface menu: Revolve) to make the surface.
- 3 In the **Revolve Options** dialog box, click **Delete curve to revolve**, and click **OK**.

The top is created.



**Note:** Practice additional surface modeling techniques by creating threads on the bottle top and the caps for the model. Use the technical drawing to help.



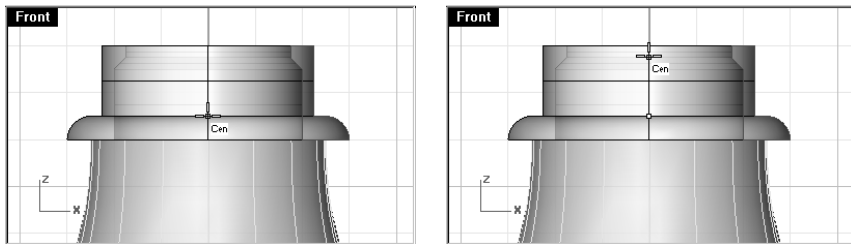
Revolve

## Adding the threads

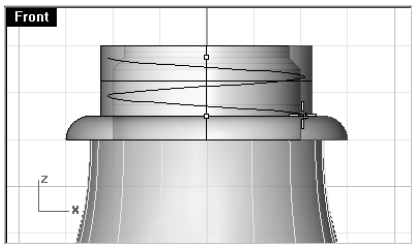
To add the threads, you will sweep a profile shape along a path drawn with the Helix command.

### To create the path curve helix:

- 1 Change to the **Curve** layer.
- 2 From the **Curve** menu, click **Helix**.
- 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 At the **Radius and start point <1.000>** ( Diameter Mode=*Pitch* Turns=*1.25* Pitch=*0.25* ReverseTwist=*No* ) prompt, click **Turns**.
- 6 At the **Number of turns <1.250>** prompt, type **1.5** and press **Enter**.
- 7 At the **Radius and start point <1.000> ...** prompt, pick a point near the edge that is slightly smaller than the outside edge of the area to be threaded.



## Notes:

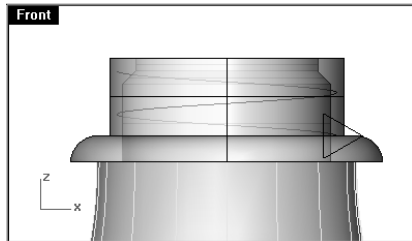


Helix

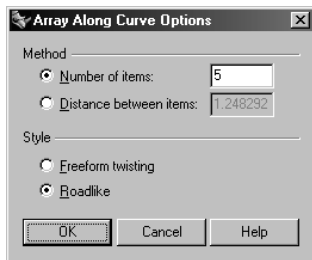
## Notes:

### To draw the thread profile:

- 1 Draw a triangle using the **Polygon** command for the thread as shown below.

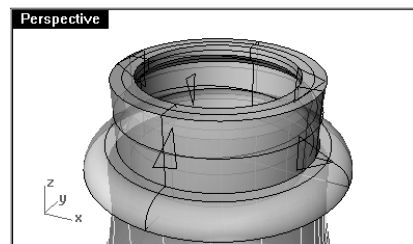
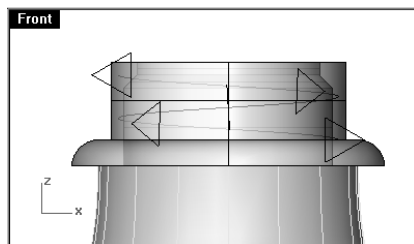


- 2 Select the triangle you created.
- 3 From the **Transform** menu, click **Array**, and then click **Along Curve**.
- 4 At the **Select path curve** ( Basepoint ) prompt, select the helix near the bottom.
- 5 In the **Array Along Curve Options** dialog, change the number of items to **5** and click **Roadlike**.

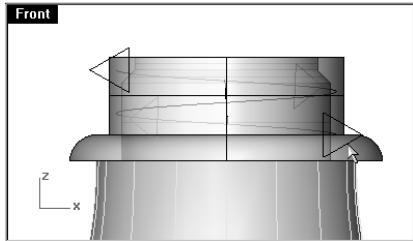


- 6 At the **Click in a viewport to choose a CPlane** prompt, click in the **Top** viewport.

It might be a good idea at this point to change your perspective view so you can see the profile curves more easily.



- 7 Delete the first and last profile curve.



You will use the remaining profile curves to create a surface.

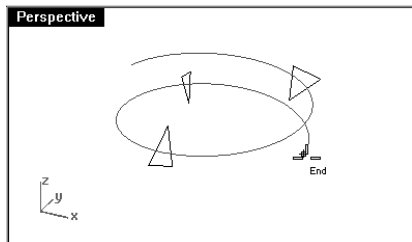
#### To create the thread surface:

- 1 Change to the **BottleTop** layer.

You can see the helix and profile curves easier if you **Hide** the bottle top at this point.

Use the **Sweep1** command to create a surface that starts at a point at one end of the helix, goes through each profile, and ends at a point at the other end of the helix.

- 2 From the **Surface** menu, click **Sweep 1 Rail**.
- 3 At the **Select rail curve** prompt, select the helix.
- 4 To start the surface at a point, at the **Select cross-section curves** ( Point ) prompt, type **P**, and press **Enter**.
- 5 At the **Pick start point** prompt, use the **End** object snap to pick one end of the helix.



- 6 At the **Select cross-section curves** ( Point ) prompt, select the profile curves in order.
- 7 To end the surface at a point, at the **Select cross-section curves** ( Point ) prompt, type **P**, and press **Enter**.

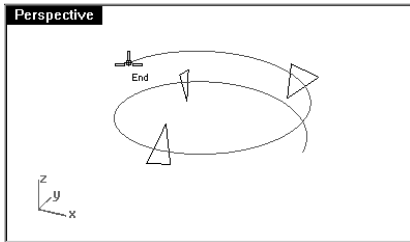
#### Notes:



Sweep Along One Rail

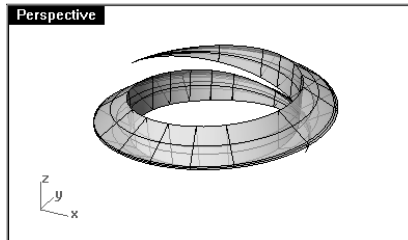
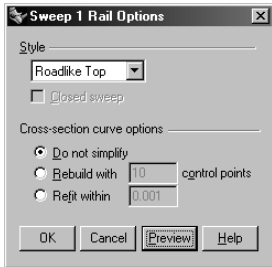
**Notes:**

- 8 At the **Pick end point** prompt, use the **End** object snap to pick the other end of the helix.

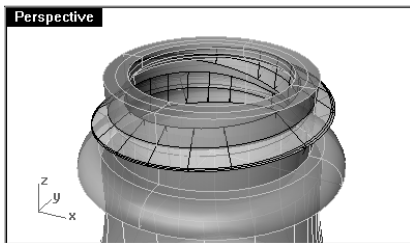


- 9 At the **Adjust curve seams** ( Flip Automatic Natural ) prompt, press **Enter**.

- 10 In the **Sweep 1 Rail Options** dialog box, change the **Style** to **Roadlike Top**, and click **OK**.

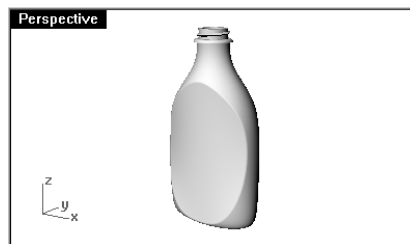
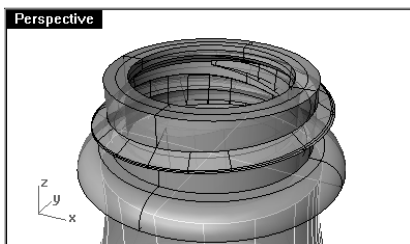


- 11 **Show** the bottle top.



- 12 Select the helical thread and the bottle top.

- 13 Use the **BooleanUnion** command (Solid menu: Union) to join the thread and the bottle top.





# 10

## 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. Search the index for “import and export.”

### Importing and Exporting Rhino File Information

When you export to a format like 3DS, STL 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 object and export it, 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 you will use the “Save As” method to export three of the most common file formats.

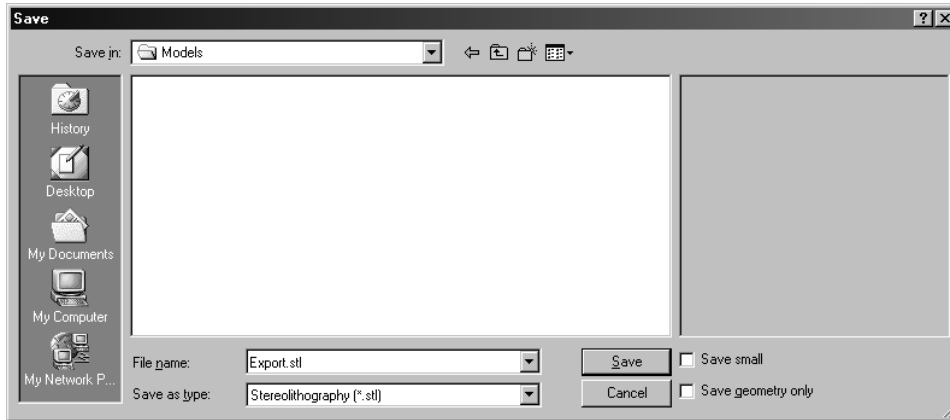
### *Exercise 62— Exporting models*

#### **To export a model to a mesh format:**

- 1 **Open** the model **Export.3dm**.
- 2 From the **File** menu, click **Save As**.

**Notes:**

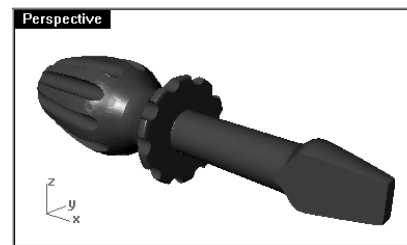
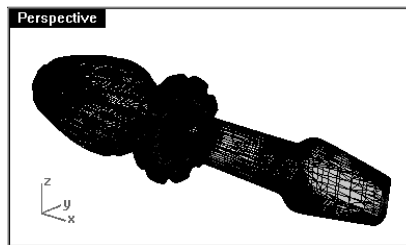
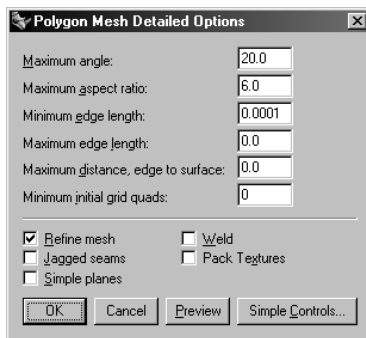
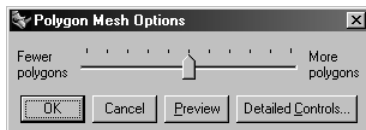
- 3 In the **Save As** dialog box, change the **Save as type** to **Stereolithography (\*.stl)**.



- 4 In the file name box type **Export** and click **Save**.  
5 In the **STL Export Options** dialog box, click **Binary**, check **Export open objects** and click **OK**.



- 6 In the **Polygon Mesh Options** dialog box, click **Preview**.

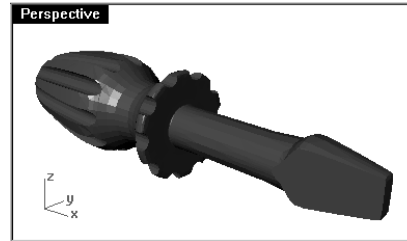
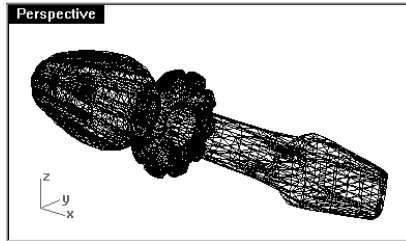
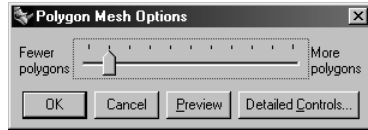


*Flat shade of the render mesh.*



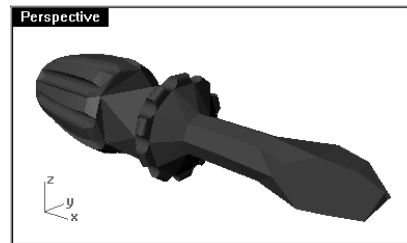
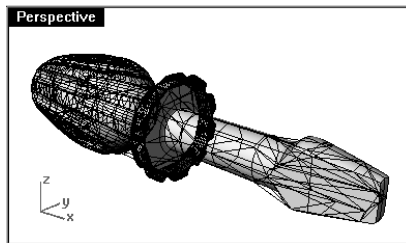
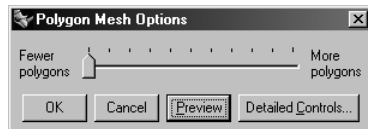
**Notes:**

- 7 Adjust the slider to the left for **Fewer polygons** and click **Preview**.



*Flat shade of the mesh.*

- 8 Move slider all the way to the left and **Preview**.



*Flat shade of the mesh.*

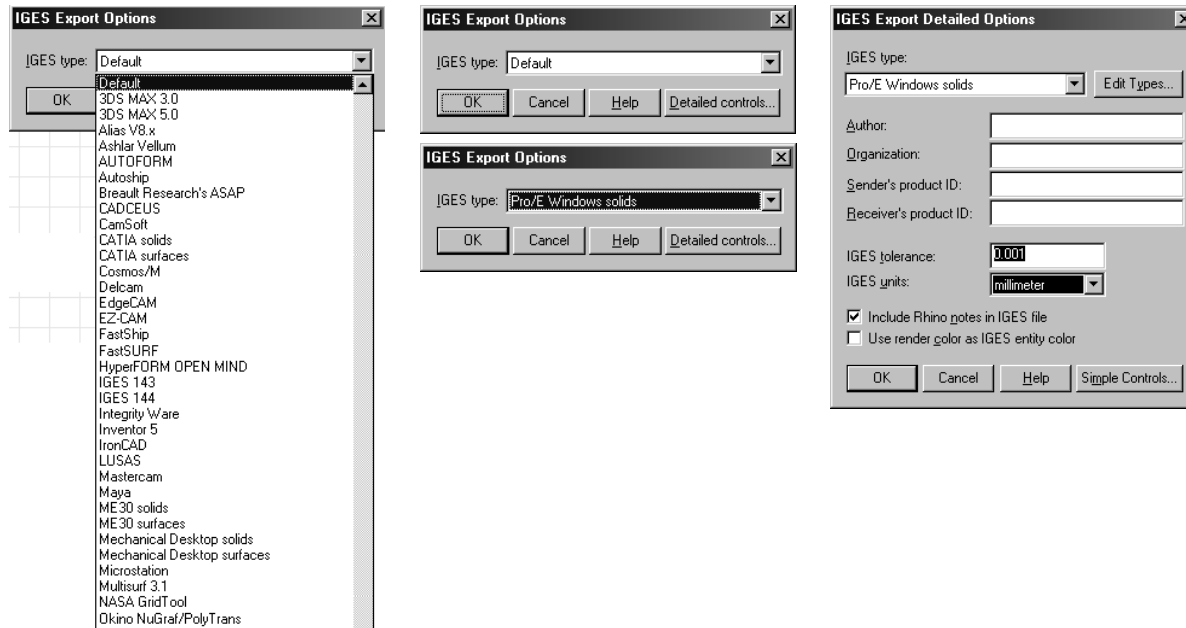
- 9 Adjust the slider until you get the desired mesh, then click **OK**.  
Detailed mesh controls are discussed in more depth in the Level 2 training class.

## Notes:

### To export a model to IGES:

- 1 From the **File** menu, click **Save As**.
- 2 In the **Save As** dialog box, change the **Save as type** to **IGES (\*.igs)**.
- 3 In the **IGES Export Options** dialog box, select **Pro E Windows solids** as the IGES type, click **Detailed Controls**.

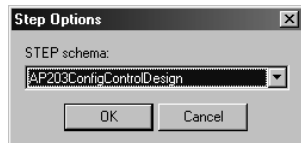
Detailed controls allows the user more input.



- 4 Click **Cancel** to terminate, or **OK** to create the IGES file.

### To export a model to STEP:

- 1 From the **File** menu, click **Save As**.
- 2 In the **Save As** dialog box, change the **Save as type** to **STEP (\*.stp, \*.step)**.
- 3 In the **STEP Options** dialog box, use the default setting.



# 11

## Rendering

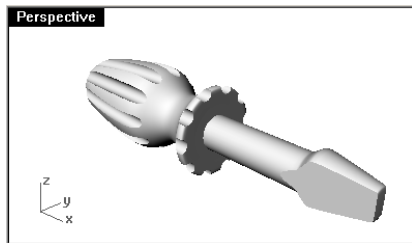
Rendering is available for showing your model as if it was photographed. While the Rhino renderer may be good enough for much of your work, use another rendering program such as Rhino's Flamingo plug-in for higher quality results. Flamingo is available as a plug-in to Rhino 2.0. Visit [www.flamingo3d.com](http://www.flamingo3d.com) for more information.

Rhino's renderer uses color, spotlights, displays shadows, and does antialiasing. It also allows the attachment of textures and bump maps. In this exercise we will focus on the full rendering capability.

### **Exercise 63— Practice rendering a model**

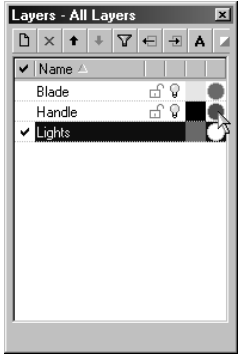
- 1 **Open** the model **Render.3dm**.
- 2 From the **Render** menu, click **Current Renderer**, then click **Rhino Render**.
- 3 Right click on the **Perspective** title bar, then click **Rendered display**.

The viewport mimics but does not exactly duplicate what you will get in a Render.



## To assign a color to the handle:

- 1 In the **Layers** dialog box, click on the **Materials** column for the **Handle** layer.



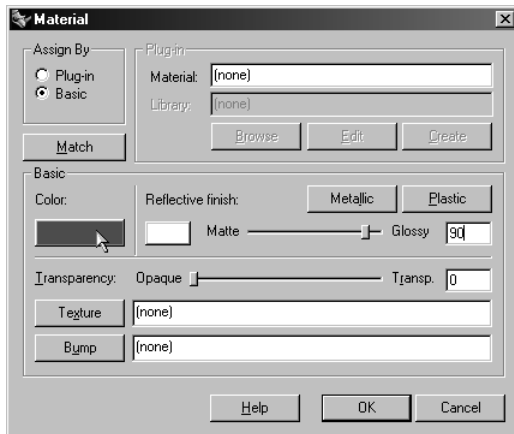
- 2 In the **Material** dialog box, click **Basic**,

To give the handle a highlight, change the Reflective finish setting.

A highlight of 0 means that the object is not shiny at all and it will not have a specular highlight. A low value of highlight makes the shiny spot to be small, and the object will look glossier. As the 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.

- 3 Change the **Reflective finish** setting to **90**, then click the **Color** swatch.



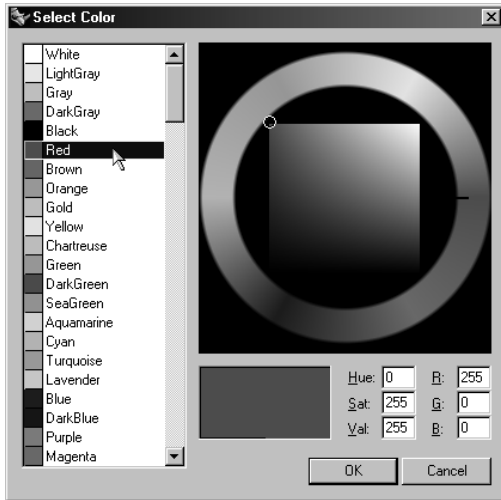
## Notes:



Render

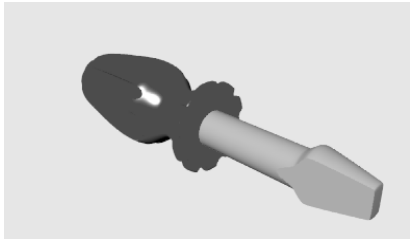
## Notes:

- 4 In the **Select Color** dialog box, select a color, like **Red**, and click **OK**.



- 5 Repeat the steps above to assign a material to the Blade layer.
- 6 From the **Render** menu, click **Render**.

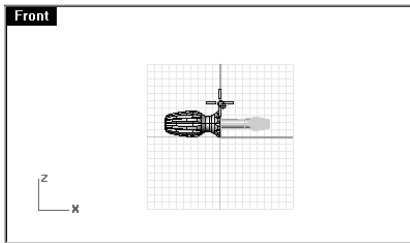
A display window appears with the current viewport rendered in colors, but it will probably lack detail. You can close the Display Window without disturbing your model. Placing lights will add depth and detail to the rendered image.



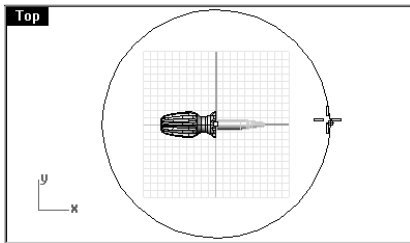
### To place a light:

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 **Lights** layer.
- 3 From the **Render** menu, click **Create Spotlight**.
- 4 At the **Base of cone** ( Vertical Diameter 3Point Tangent AroundCurve ) prompt, pick a point near the center of the screwdriver and slightly above it in the **Front** viewport.

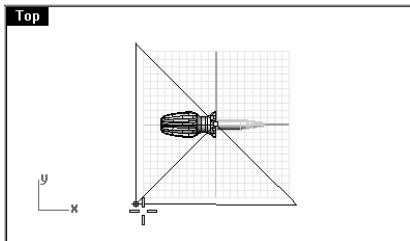


- 5 At the **Radius <1>** ( Diameter ) prompt, pick a point so that the radius is larger than the entire screwdriver in the **Top** viewport.



- 6 At the **End of cone** prompt, hold down the control key, and pick a point below and to the left in the **Top** viewport.

This starts elevator mode.



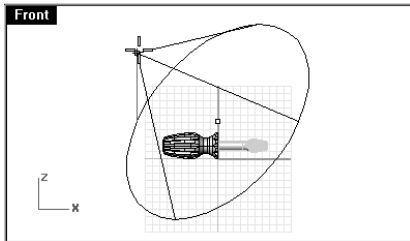
### Notes:



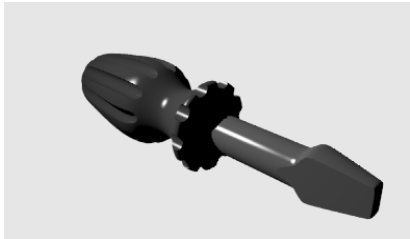
Spotlight

## Notes:

- 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.
- 9 From the **Render** menu, click **Render**.  
The image has some highlights and shadows.

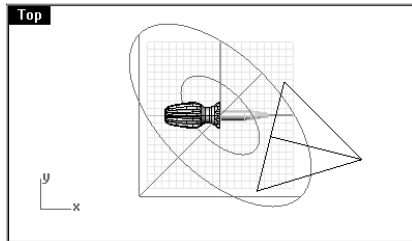


### To place a second light:

- 1 Zoom out in the **Top** and the **Front** viewport.
- 2 From the **Render** menu, click **Create Spotlight**.
- 3 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.
- 4 At the **Radius <1> ( Diameter )** prompt, pick a point so that the radius is larger than the entire screwdriver in the **Top** viewport.

## Notes:

- 5 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.



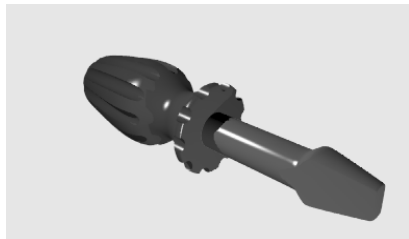
### To assign properties to the light:

- 1 Select the new light.
- 2 From the **Edit** menu, click **Object Properties**.
- 3 On the **Light** page, change the **Shadow intensity** to **30** and the **Spotlight hardness** to **60**.

Experiment with these settings to get the desired effect.



- 4 Click in the **Perspective** viewport.
- 5 From the **Render** menu, click **Render**.

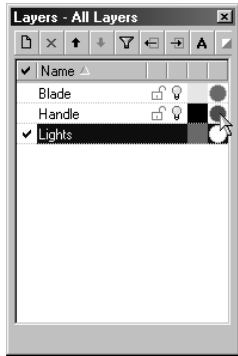




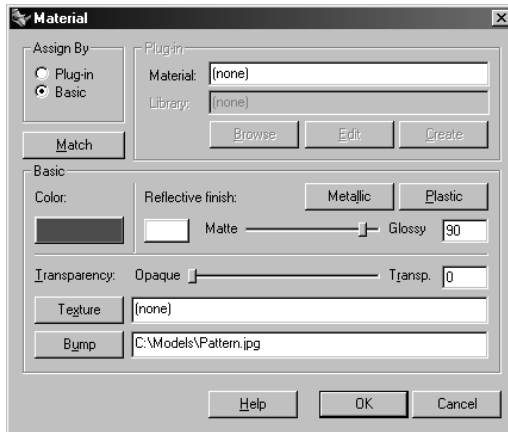
## Notes:

### To add a bump surface to the handle:

- 1 In the **Layers** dialog box, click on the **Materials** column for the **Handle** layer.



- 2 In the **Material** dialog box, click the **Bump** button.

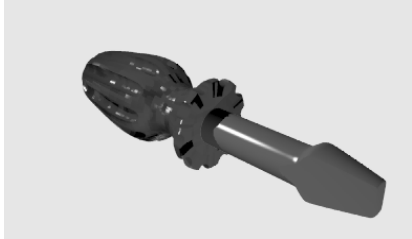


You can use any bitmap file for a bump. The bumps come from the pattern of light and dark in the bitmap image.

- 3 In the **Open Bitmap File** dialog box, select **Pattern.jpg**, then click **Open**.
- 4 Click **OK** to close the **Material** dialog box.

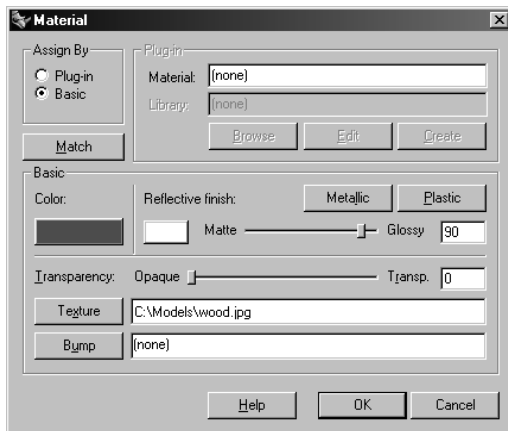
## Notes:

- 5 From the **Render** menu, click **Render**.  
The surface of the handle has a bumpy appearance.



### To add a texture to the handle:

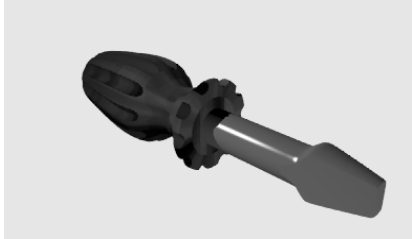
- 1 In the **Layers** dialog box, click on the **Materials** column for the **Handle** layer.
- 2 In the **Material** dialog box, clear the **Bump** field.
- 3 In the **Material** dialog box, click the **Texture** button.



- 4 In the **Open Bitmap File** dialog box, select **Wood.jpg**, then click **Open**.
- 5 Click **OK** to close the **Material** dialog box.

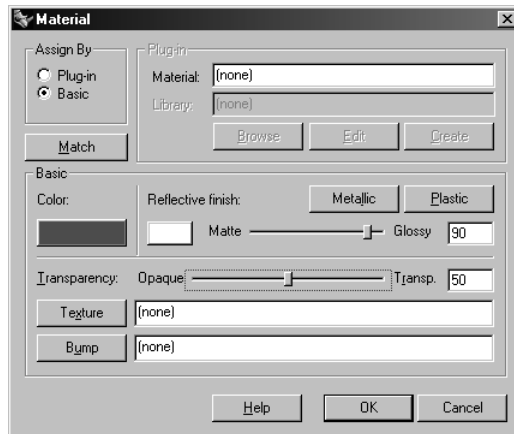
- 6 From the **Render** menu, click **Render**.

The surface of the handle is textured with a wood grain texture.



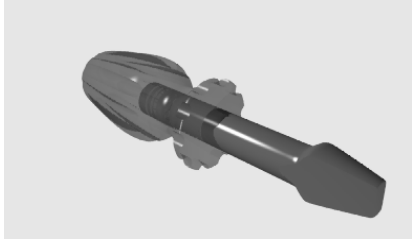
### To make the handle transparent:

- 1 In the **Layers** dialog box, click on the **Materials** column for the **Handle** layer.
- 2 In the **Material** dialog box, clear the **Texture** field.
- 3 In the **Material** dialog box, change the **Transparency** setting to **50**.



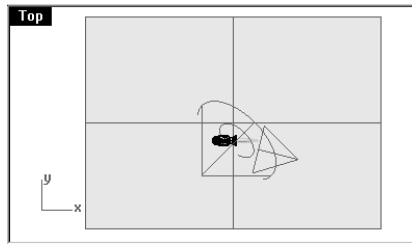
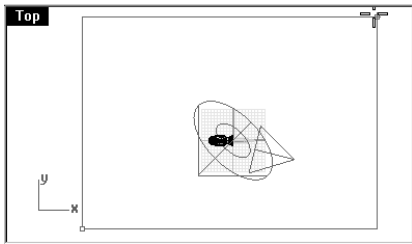
### Notes:

- From the **Render** menu, click **Render**.  
The handle will look transparent.



#### To add a surface for a ground plane:

- Use the **Plane** command (Surface menu: Plane > Corner to Corner) to draw a flat surface in the **Top** viewport.

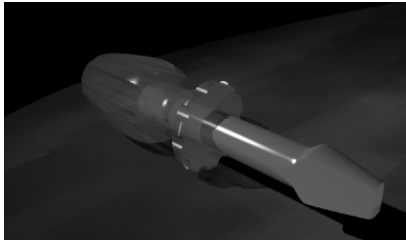
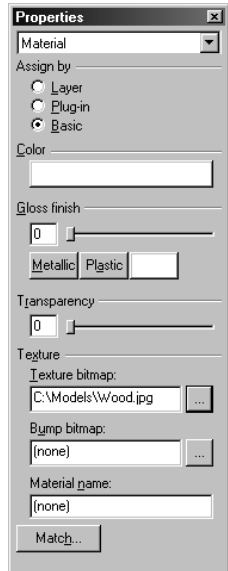


- Select the surface.
- Use the **Properties** command (Edit menu: Object Properties...) on the **Material** page, click **Basic**.

#### Notes:

## Notes:

- 4 In the **Texture** section assign the **Wood.jpg** to the plane.



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

## Rendering with Flamingo

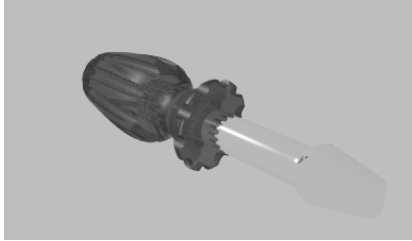
In this part of the exercise you will use Flamingo to assign materials from the material library and set up an environment. You will use the same lights and model.

### To assign a material:

- 1 Select the flat surface.
- 2 Use the **Hide** command (Edit menu: Visibility > Hide) to hide it.
- 3 From the **Render** menu, click **Current Renderer**, then click **Flamingo Raytrace**.
- 4 In the **Layers** dialog box, click on the **Materials** column for the **Handle** layer.
- 5 In the **Material** dialog box, click **Plug-in**, then click **Browse**.
- 6 In the **Material Library** dialog box, in the **Plastics** library, in the **Transparent** folder, select a material, then press **OK**.
- 7 In the **Material** dialog box, press **OK**.
- 8 Repeat this process for the **Blade** layer.
- 9 Choose a shiny metal material, like **Metal\Steel\Polished\Plain**.

## Notes:

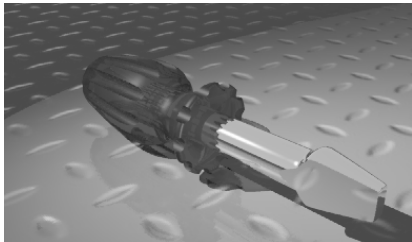
10 **Render** the model.



### To set up an environment:

- 1 Use the **Options** command (Tools menu: Options...) to setup the Flamingo environment options
- 2 In the **Rhino Options** dialog box, on the **Flamingo** page, click the **Environment** button.
- 3 On the **Environment** dialog box, change to a **3 Color Gradient**.  
Use the default setting.
- 4 Check the **Ground Plane** check box.
- 5 On the **Ground Plane** page, click the **Material** button, and select a material for the ground plane, like **Metal\Aluminum\Satin\Checker Plate**.
- 6 On the **Environment** dialog box, press **OK**.
- 7 On the **Rhino Options** dialog box, press **OK**.
- 8 **Render** the model.

Notice that you also get reflections.



Flamingo rendering is covered in more depth in Level 2 training.

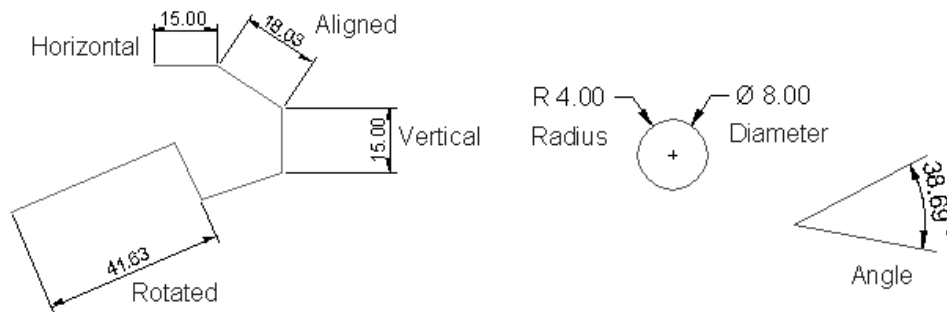
For extra practice use the canoe you made in the loft exercise.




# 12 Dimensions

## Dimensions









You can create simple dimensions that are visible when you are looking at the plan view of the construction plane.

## Dimension Types



Button	Command	Description
	Dim	Creates a horizontal or a vertical dimension.
	DimAligned	Creates an aligned dimension.
	DimRotated	Creates a rotated dimension.

**Notes:**

Button	Command	Description
	DimAngle	Creates an angular dimension.
	DimRadius	Creates a radius dimension.
	DimDiameter	Creates a diameter dimension.
	Text	Creates 2-D annotation text.
	Leader	Draw an arrow leader.
	Properties	Edit dimensions and text.
	DimRecenterText	Returns text that has been moved away from its default position to its original location.
	Make2-D	Creates curves from the selected objects as silhouettes relative to the active construction plane. The silhouette curves are projected flat and then placed on the world x,y-plane.

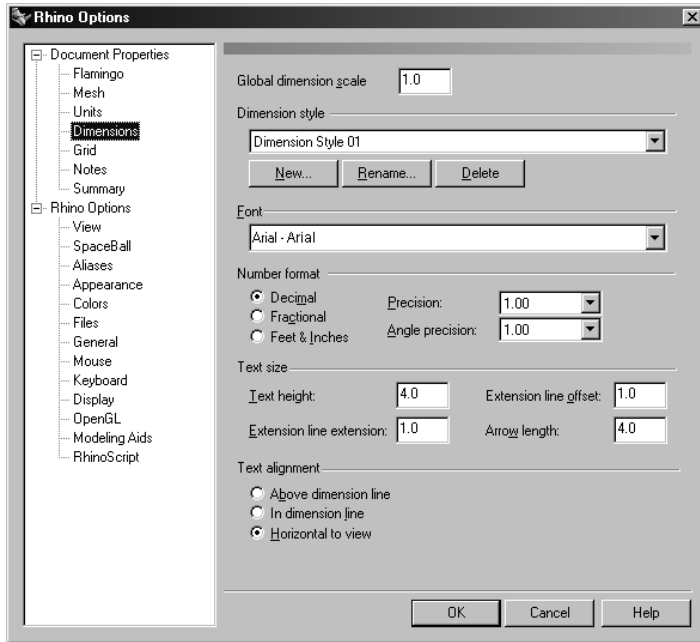
**Exercise 64— Practice dimensioning**

- 1 **Open** the model **Dimension.3dm**.
- 2 From the **File** menu, click **Properties**.

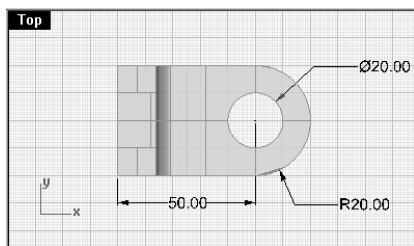


## Notes:

- 3 In the **Rhino Options** dialog box, on the **Dimensions** page, make the following changes.



- 4 From the **Dimension** menu, click **Linear Dimension**.
- 5 At the **First dimension point** ( Style ) prompt, snap to the lower left end of the part in **Top** viewport.
- 6 At the **Second dimension point** prompt, snap to the lower right end of the part in the **Top** viewport.
- 7 At the **Dimension location** ( Vertical Horizontal ) prompt, pick a point below the part in the **Top** viewport.

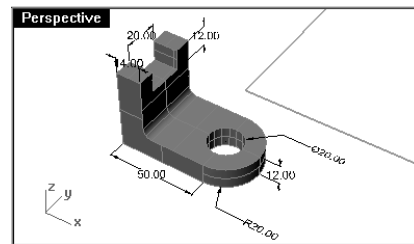
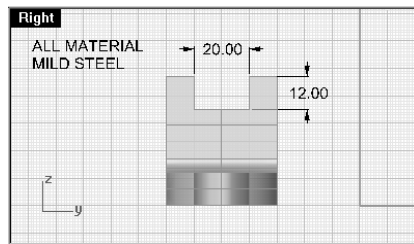


*Use object snaps to locate the extension line origins.*

- 8 From the **Dimension** menu, click **Radial Dimension**.
- 9 At the **Select curve for radius dimension ( Style )** prompt, select the lower left quadrant of the arc in the **Front** viewport.
- 10 At the **Dimension location** prompt, click to place the dimension text.



- 11 Dimension the rest of the drawing using leaders, text blocks, horizontal, vertical, radius, and diameter dimensions.



- 12 **Save** your model.

## Making a 2-D Drawing from a 3-D 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 aligning the views. Options for first angle projection or third angle projection are available. In addition to the three orthographic views, a two-dimensional perspective drawing is also generated. Hidden lines are removed and placed on a separate layer.

Options for creating four views: three parallel viewports and a perspective viewport, or single views of individual viewports are supported.

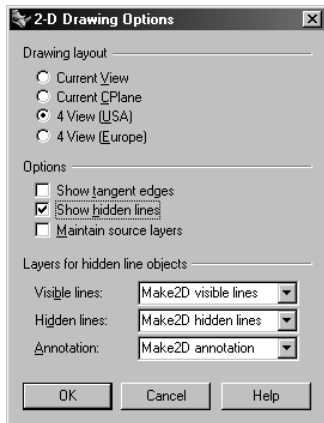
The Make2D command makes a two-dimensional drawing for all four views.

### Notes:

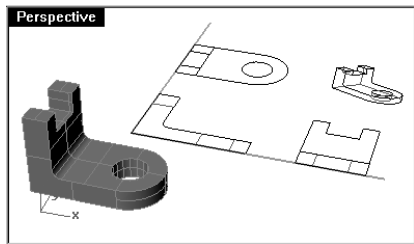
*To move extension line or text locations for dimensions, turn on control points for the dimension and move the control points.*

**Exercise 65— Practice making a 2-D drawing for export**

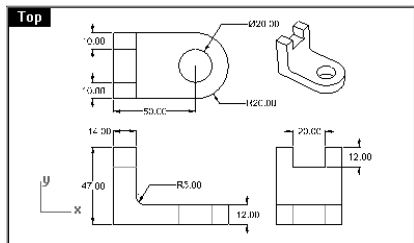
- 1 **Open** the model **Make2D.3dm**.
- 2 From the **Dimension** menu, click **Make 2-D Drawing**.
- 3 In the **Make2D Options** dialog box, click **4-view (USA)** and check **Show hidden lines**, and then click **OK**.



The 2-D drawings are created on the **Top** CPlane near the origin on the world xy plane. View them in the **Top** viewport.



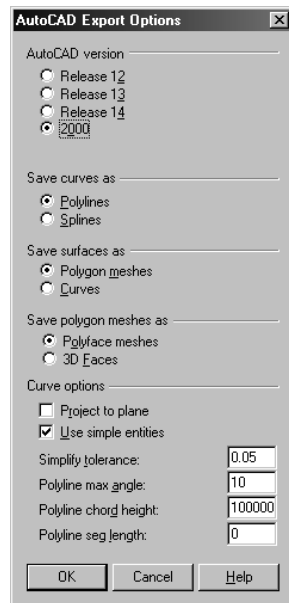
- 4 **Dimension** the 2D drawing.



## Notes:

### To export the 2-D drawing to AutoCAD:

- 1 Select the 2-D geometry and the dimensions.
- 2 From the **File** menu, click **Export Selected**.
- 3 Change the **Save as type** to **AutoCAD DWG**, the **File name** to **Bracket.dwg**, then click **Save**.
- 4 In the **AutoCAD Export Options** dialog box, **AutoCAD Version 2000**, **Save Curves as Polylines**, **Save Surfaces as Polygon meshes**, **Save Polygon meshes as Polyface meshes**, check **Use Simple Entities**, click **OK**.



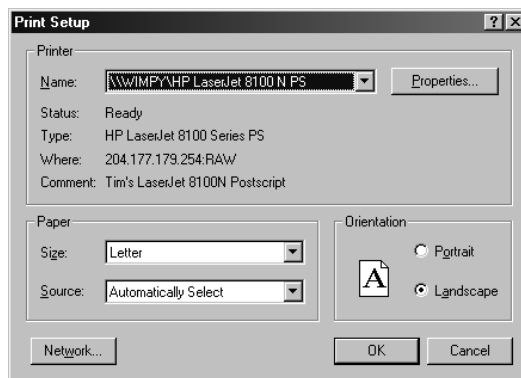
The 2-D geometry, layers and dimensions are translated to AutoCAD's .DWG format.

# 13 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, hidden line removal is not supported. Use the **Make2D** command to generate hidden-line views to print.

## Exercise 66— Practice printing

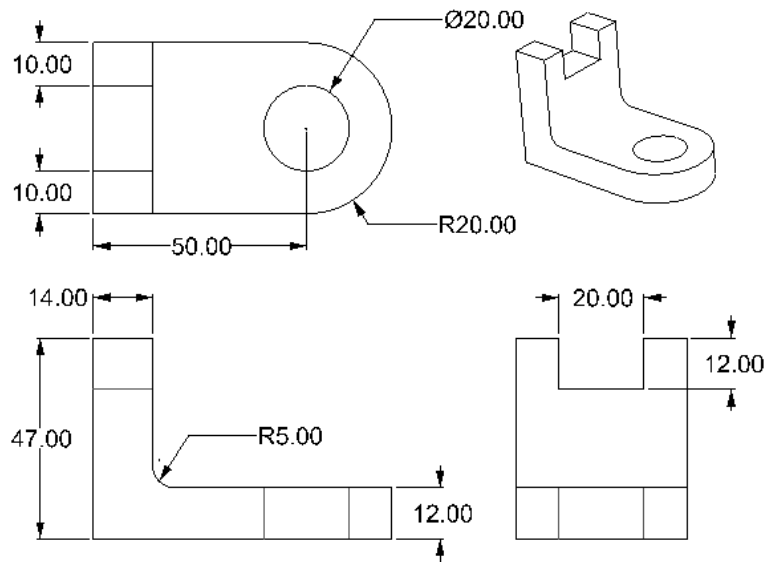
- 1 **Open** the model **Printing.3dm**.
- 2 Click in the **Top** viewport.
- 3 From the **File** menu, click **Print Setup**.
- 4 In the **Print Setup** dialog box, click **Landscape**, adjust any other options appropriate for your printer or plotter, then click **OK**.



- 5 From the **File** menu, click **Print**.

**Notes:**

- 6 In the **Print** dialog box, choose the following settings:  
Under **Print area**, click **View**.  
Under **Print scale**, select **Scaled to Fit**.  
Under **Print offset** click **Center objects on paper**.
- 7 Click **OK**.  
The **Top** viewport will be printed.



## Notes:

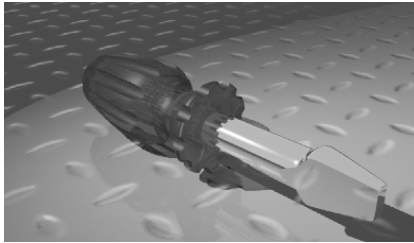
### To print the current viewport to a scale:

- 1 From the **File** menu, click **Print**.
- 2 In the **Print** dialog box, change **Print scale** to **1:20**, then click **OK**.

### To print a rendered image:

- 1 **Open** the model **Render.3dm**.
- 2 From the **Render** menu, click **Render**.
- 3 In the render window, from the **File** menu, 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**.







# Part Four: Customizing Workspaces and Toolbars

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# 14

## Rhino Settings

### Options

The **Rhino Options** dialog box displays most of the settings that affect your modeling environment. Use it to toggle the settings or redefine them.

The **Document Properties** settings are saved with the Rhino model. The **Rhino Options** settings are saved in the Windows registry and affect every Rhino model.

### *Exercise 67— Practice with options*

#### To change modeling aids:

- 1 From the **Tools** menu, click **Options**.
- 2 In the **Rhino Options** dialog box, click the **Modeling Aids** page.

The Modeling Aids page controls Grid Snap, Ortho, Planar mode, and Object Snap options.

These options can be toggled on and off from the dialog box or from the status bar. To change the options for Ortho or object snap, type new values in the boxes. In an earlier exercise you changed the Ortho setting so that it snapped every 30 degrees.

In the object snap area, **Project object snap to CPlane** projects the snap point to the construction plane even though the object snap point might be in 3-D space.

#### To change Rhino's screen appearance:

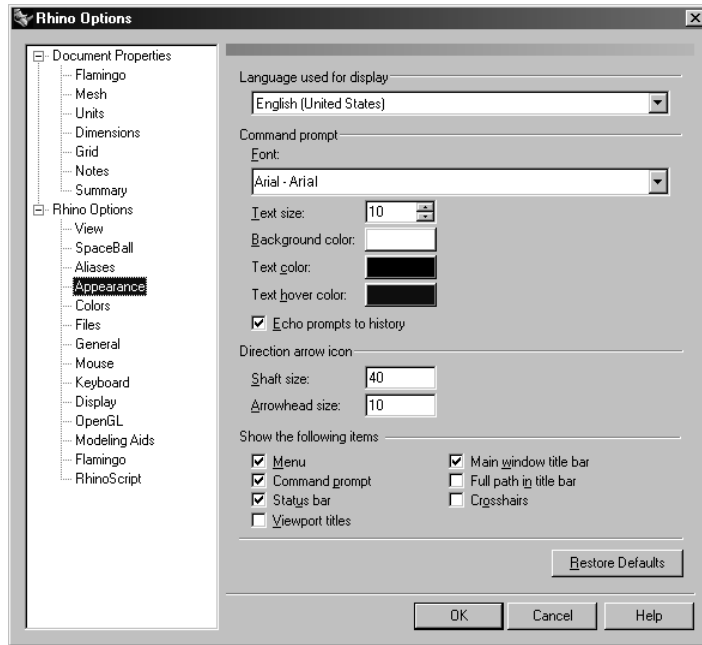
- 1 Click the **Appearance** page.  
The Appearance page controls the appearance of the Rhino window.

### Notes:



Options

- 2 Click the gray rectangle next to **Background color** and change it to a different color.
- 3 Check the **Crosshairs** box.

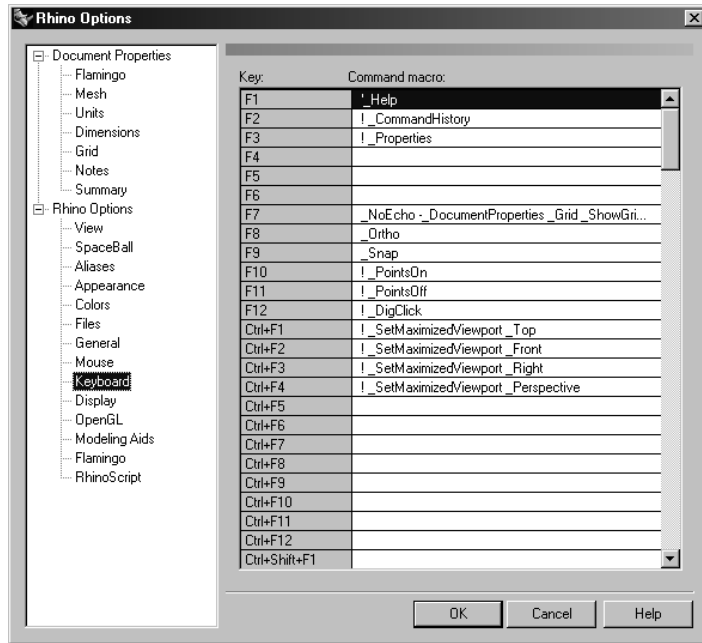


## Notes:

**To change or create shortcuts:**

- 1 Open the **Keyboard** page.

The Keyboard page sets up shortcut keys for Rhino commands.



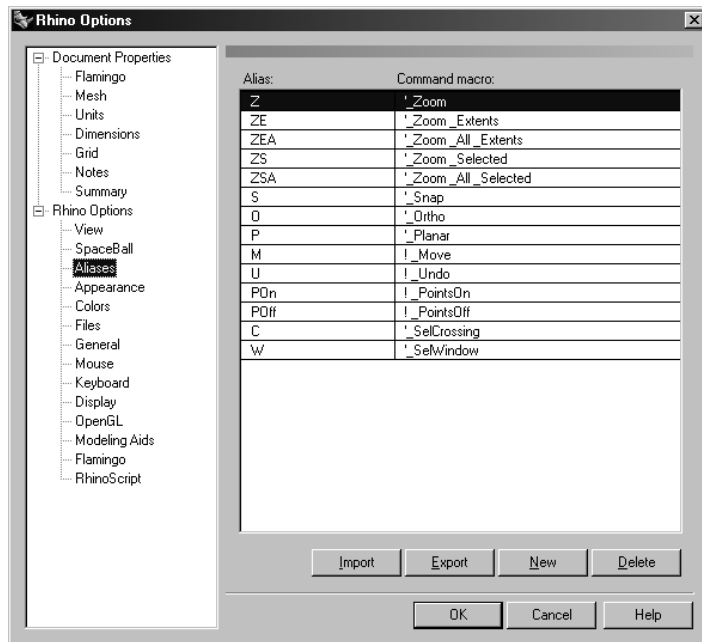
- 2 Type **DisableOsnap** in the box next to the **F4** key.

This gives you a key that toggles your persistent object snaps between on and disabled.

## Notes:

### To change or create command aliases:

- 1 Click the **Aliases** page.  
The command Aliases page allows the user to create custom aliases for Rhino commands.
- 2 Click **New**.  
A cursor moves into the white area of the dialog box.



- 3 Type **L**, then press the tab key.
- 4 Type **! Lines**.  
The exclamation point acts as a cancel.  
Now you will be able to type **L** and press **Enter** at the command line, to begin the **Lines** command.

### To change view options:

- Click the **View** page.  
The View page has pan, zoom, rotate, and redraw controls.

## Notes:

### To change general options:

Click the **General** page.

The General page the number of undos retained in memory, startup commands, recent command list limit, AutoSave control, and do not repeat commands, default isocurve density for new surfaces.

### To set file options:

- 1 Click the **Files** page.

The Files page sets the location of template files and the autosave files.

- 2 Click **OK** to close the **Options** dialog box.
- 3 Try the **Lines** alias and the **DisableOsnap** toggle.

Note how your cursor has changed.

## Document Properties

Document properties are all of the settings that are saved with the Rhino model.

### *Exercise 68— Practice with document properties*

#### To change the grid:

- 1 From the **File** menu, click **Properties**.
- 2 Click the **Grid** page.

The Grid page configures grid, grid axes, and axis icons. You changed the snap spacing in an earlier exercise.

#### To change render options:

- 1 Click the **Rhino Render** or **Flamingo** page.  
The Render page controls most of the render options.
- 2 Click the **Mesh** page.

This page controls the mesh properties. It will also affect performance.

#### To set units:

Click the **Units** page.

The Units page controls the unit system and the tolerance settings.



Document Properties





# 15

## Custom Toolbar Layouts

The toolbar layout is the arrangement of toolbars containing command buttons on the screen. The toolbar layout is stored in a toolbar collection 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 collection file.

### ***Exercise 69— Customizing a toolbar layout***

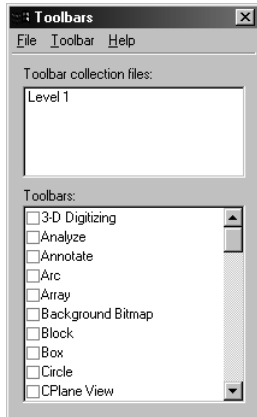
- 1 Begin a new model.
- 2 From the **Tools** menu, click **Toolbar Layout**.
- 3 In the **Toolbars** dialog box, click **Save As**.
- 4 In the **Save Workspace As** dialog box, in the **File name** box type **Level 1**, and click **Save**.

A new toolbar collection file has been created. Toolbar collection files are saved with a .ws3 extension. You will use this new toolbar collection to do some customization.

## Notes:

### To show another toolbar:

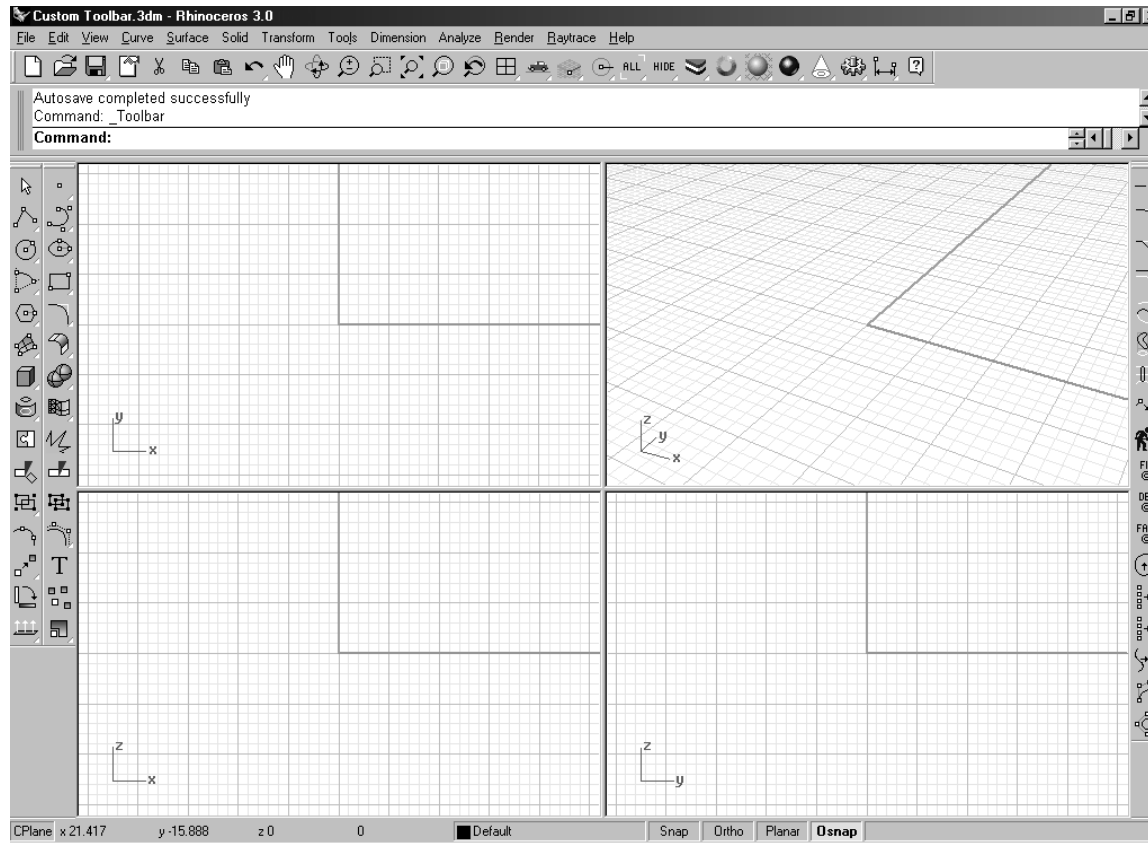
- 1 From the **TOOLS** menu, click **Toolbar Layout**.



- 2 In the **Toolbars** dialog box, check **Curve Tools** to show the toolbar.
- 3 In the **Toolbars** dialog box, clear **Curve Tools** to hide the toolbar.
- 4 Show the **Curve Tools** toolbar again, then click **Close**.

## Notes:

- To dock the **Curve Tools** toolbar at the right, drag it to the right edge until it changes to a vertical shape, then releasing your mouse button.



### To save your toolbar layout:

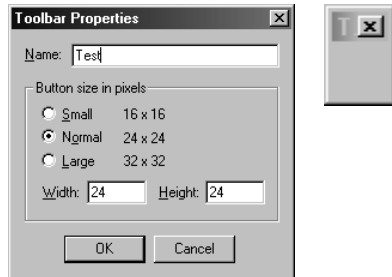
In the **Toolbars** dialog box, from the **File** menu, click **Save**.

## Notes:

### To create a new toolbar:

- 1 From the **Tools** menu, click **Toolbar Layout**.
- 2 In the **Toolbars** dialog box, from the **Toolbar** menu, click **New**.
- 3 In the **Toolbar Properties** dialog box, in the **Name** box, type **Test**, and click **OK**.

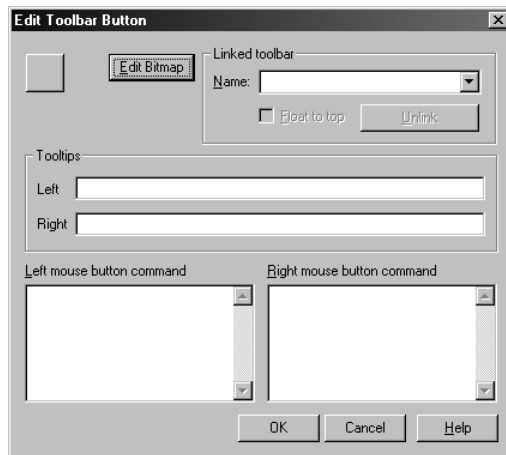
A new toolbar is created with one blank button in it.



- 4 Close the **Toolbars** dialog box.

### To edit the new button:

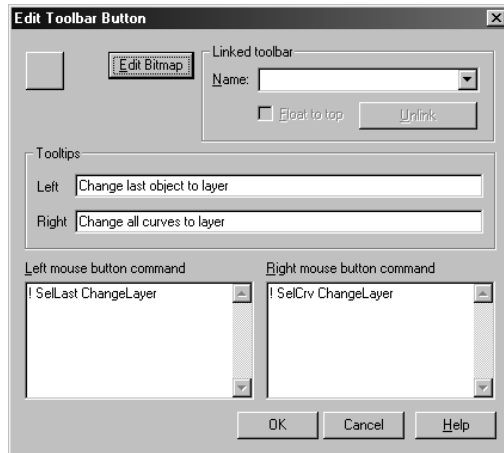
- 1 Hold down the **Shift** key and right-click the blank button.



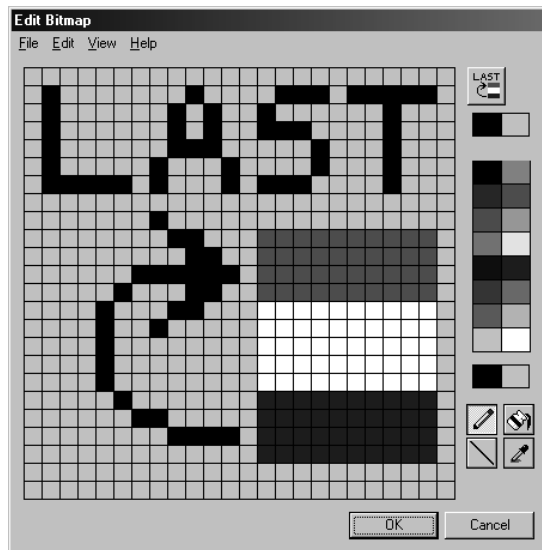
- 2 To enter the tooltips for the left and right mouse buttons, in the **Edit Button** dialog box, under **Tooltips**, in the **Left** edit box, type **Change Last Object to Layer**.  
In the **Right** edit box, type **Change All Curves to Layer**.

## Notes:

- 3 In the **Left mouse button command** box, type **! SelLast ChangeLayer**.
- 4 In the **Edit Button** dialog box, in the **Right mouse button command** box, type **! SelCrv ChangeLayer**.



- 5 Click **Edit Bitmap**.
- 6 In the **Edit Bitmap** dialog box, make a picture for the button face image, and click **OK**.



- 7 In the **Edit Button** dialog box, click **OK**.

## Notes:

### To copy a button from one toolbar to another:

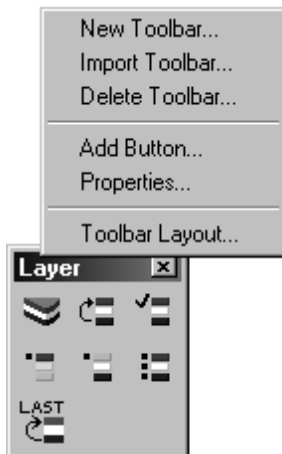
- 1 From the **Standard** toolbar, flyout the **Layer** toolbar, and tear it off.
- 2 While holding down the **Ctrl** key, drag the button you just created to the layer toolbar and drop it.



### To add a button to a toolbar:

- 1 From the **Tools** menu, click **Toolbar Layout**.
- 2 In the **Toolbar** dialog box, check the **Test** toolbar in the list to show the toolbar.
- 3 Click **New Button**.  
A blank button is added to the **Test** toolbar.
- 4 **Close** the dialog box.

Many toolbar functions are available by right-clicking a toolbar's titlebar.



---

**Notes:**

**To open the standard toolbar layout:**

- 1 From the **Tools** menu, click **Toolbar Layout**.
- 2 In the **Toolbars** dialog box, from the **File** menu, click **Close**.
- 3 In the **Save Changes** dialog box, click **Yes**.
- 4 In the **Toolbars** dialog box, from the **File** menu, click **Open**.
- 5 Select **Default.ws3**, and click **Open**.