

# Rhino<sup>®</sup>ceros

NURBS modeling for Windows

**Training Manual  
Level 1**

**Version 4.0**

*R40TML1—Oct-2008*

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Rhinoceros Level 1 Training Manual v4.0

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Credits:

Phil Cook, Simply Rhino Limited, UK, [www.simplyrhino.co.uk](http://www.simplyrhino.co.uk) for the exercises on SmartTrack and Constraints.

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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, object snaps, and SmartTrack tools
- 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.*

## Example Schedule

<b>Day 1</b>	<b>Topic</b>
<b>8-10AM</b>	<b>Introduction, Rhino Interface</b>
<b>10AM-12PM</b>	<b>Rhino Interface, Panning &amp; Zooming</b>
12-1PM	Lunch
<b>1-3PM</b>	<b>Creating geometry</b>
<b>3 -5PM</b>	<b>Creating geometry</b>
<b>Day 2</b>	<b>Topic</b>
<b>8-10AM</b>	<b>Editing</b>
<b>10AM-12PM</b>	<b>Editing</b>
12-1PM	Lunch
<b>1-3PM</b>	<b>Editing</b>
<b>3 -5PM</b>	<b>Control point editing, modeling with solids</b>
<b>Day 3</b>	<b>Topic</b>
<b>8-10AM</b>	<b>Surfacing</b>
<b>10AM-12PM</b>	<b>Surfacing</b>
12-1PM	Lunch
<b>1-3PM</b>	<b>Modeling practice</b>
<b>3 -5PM</b>	<b>Import-Export, Rendering, Dimensioning, Printing, Customization</b>

**Notes:**

# 2

## Rhino Basics

### The Rhino for Windows Interface

Before learning individual tools, we will be 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 menus in this class.

#### To open Rhino:

- ▶ Double-click the Rhino icon from the Windows desktop.

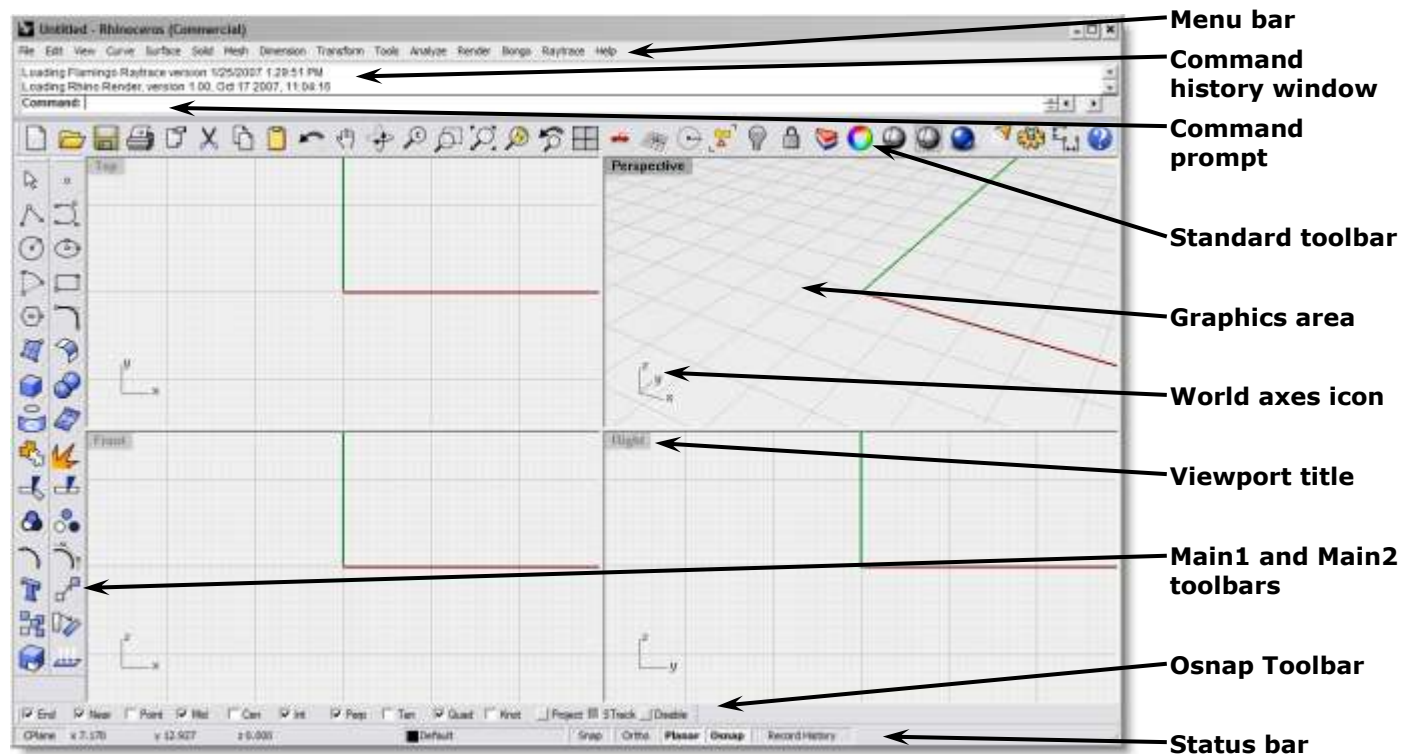
**Notes:**

**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, and 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.

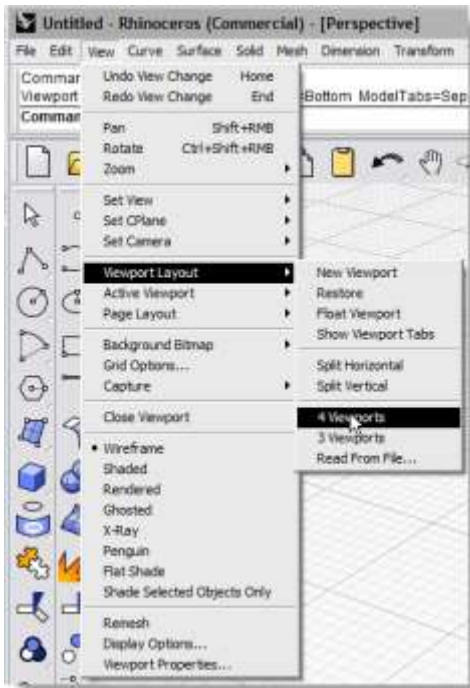


Watch the command line to find out what is happening.

Rhino screen

## Menus

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



Rhino View menu

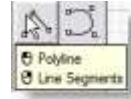
## Toolbars

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.

## 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.



*To start a Polyline, click the LMB, to start the Line Segments command click the RMB.*

## 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 mouse button for a moment or press the right mouse button.



*The Lines toolbar is linked to the Main1 toolbar. After the flyout is open you can pick any of the buttons on the toolbar to start a command.*

## Graphics Area

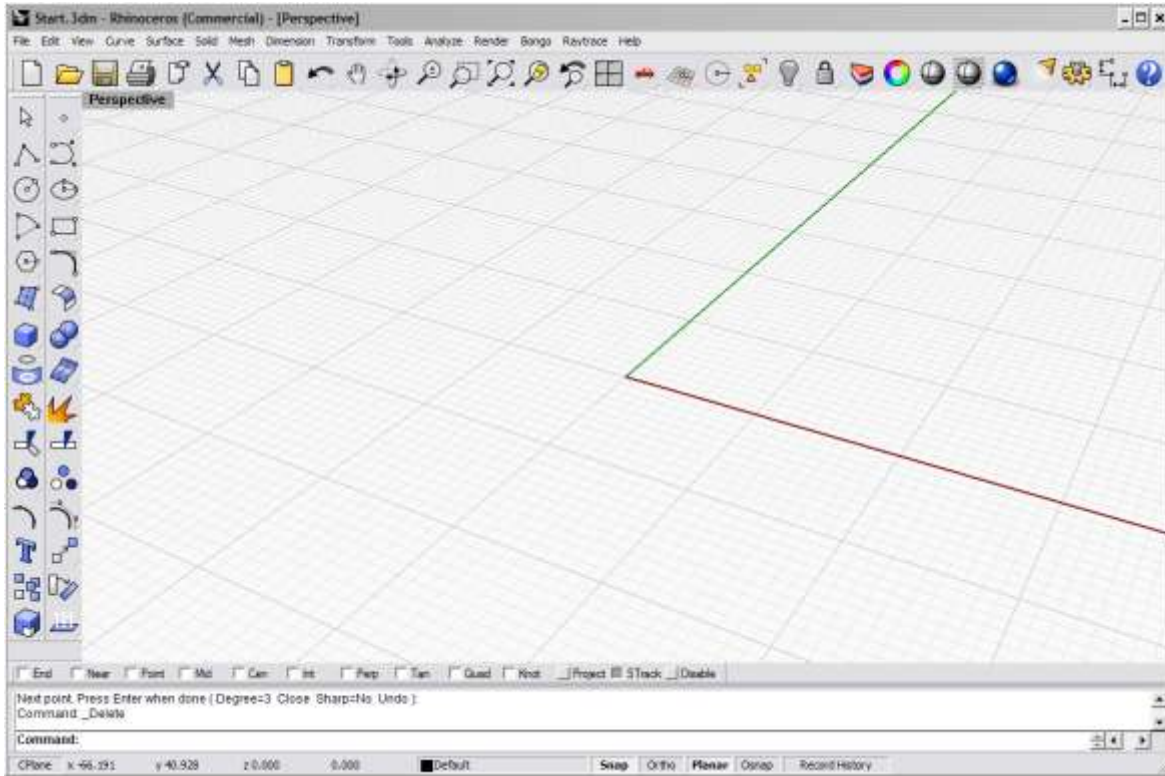
The Rhino graphics area, holding the viewports, can be customized to suit your preferences. The position of viewports can be arranged in different configurations.

## Notes:

## Viewports

Viewports are windows in the graphics 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.

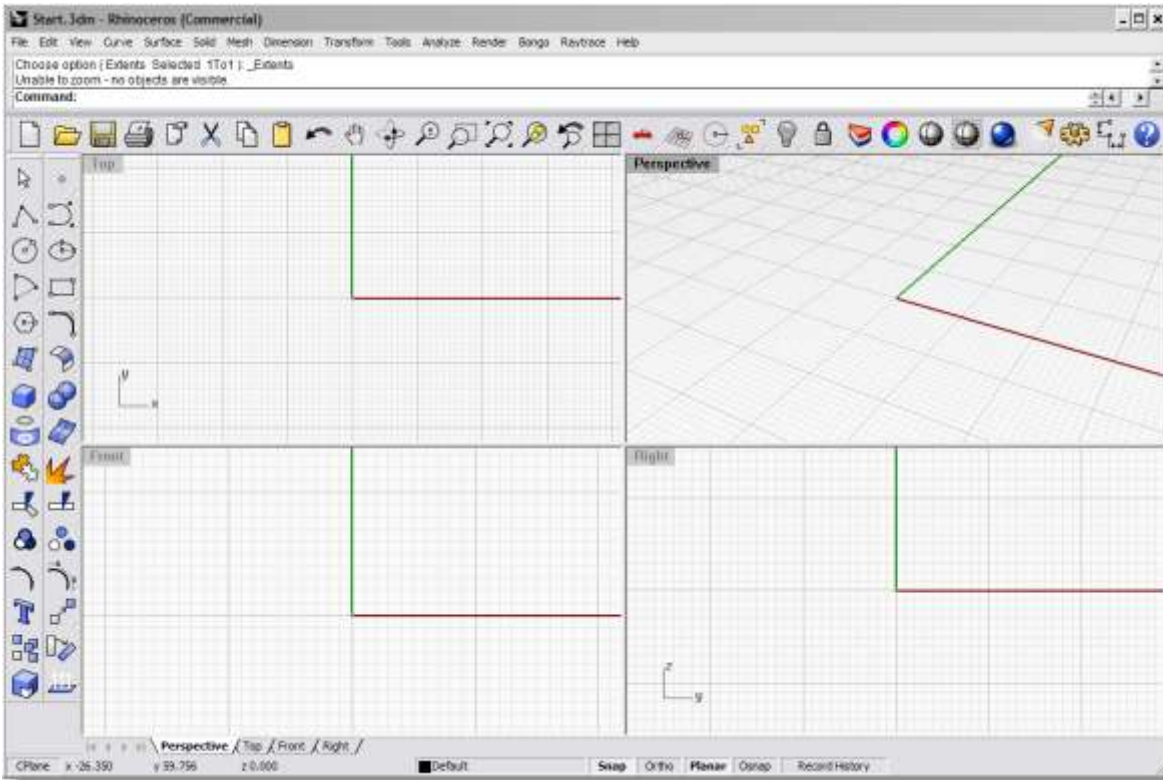


*Rearranged Rhino screen. Command line at the bottom, single maximized viewport, and toolbars docked in different locations.*

## Notes:

## Viewport tabs

Viewport titles can be shown in tabs. The bold face tab designates the active viewport. Tabs make it easy to switch between viewports when using maximized or floating viewports. To activate Viewport Tabs: From the **View** menu, click **Viewport Layout**, and then click **Show Viewport Tabs**.



*The tabs are located below the graphics area.*

**Notes:**



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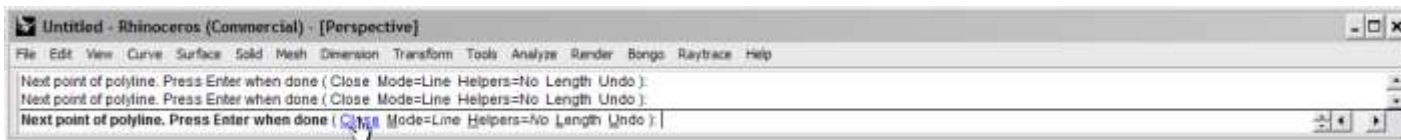
**Note:** **Enter** and **Spacebar** perform the same function.

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

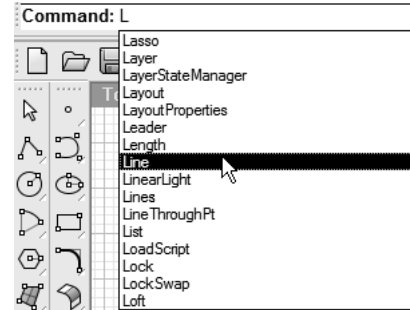
## Clickable options

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.)



## 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.



## Notes:

## Repeating commands

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

## Canceling commands

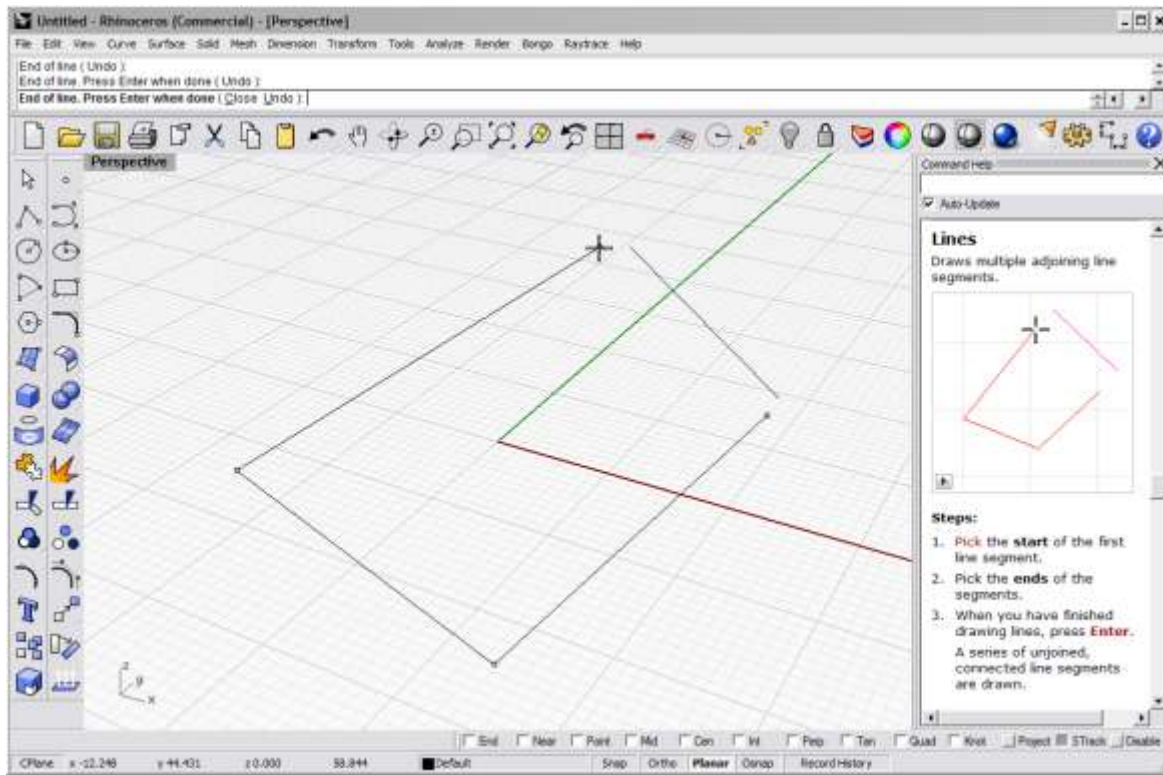
To cancel a command, press **Esc** or enter a new command from a button or a menu.

## 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 confused or unsure about what to do, the first place you should look is the help file. You can also access help for a specific command by starting the command and press **F1**.

In addition, the **CommandHelp** command displays the help topics in a dockable window and displays help for the current command.

Most of the commands include short video clips that show how the command and the options work.



*If Auto-update is checked, the help for the current command displays. If Auto-update is unchecked, you can type the name of the command that you want displayed and press enter to display the information.*

## Notes:

## View the Command Line History

The command 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.

The number of commands listed is set in Rhino Options. The default limit is twenty commands. When you use your twenty-first command the first one drops off the list.



**Notes:**

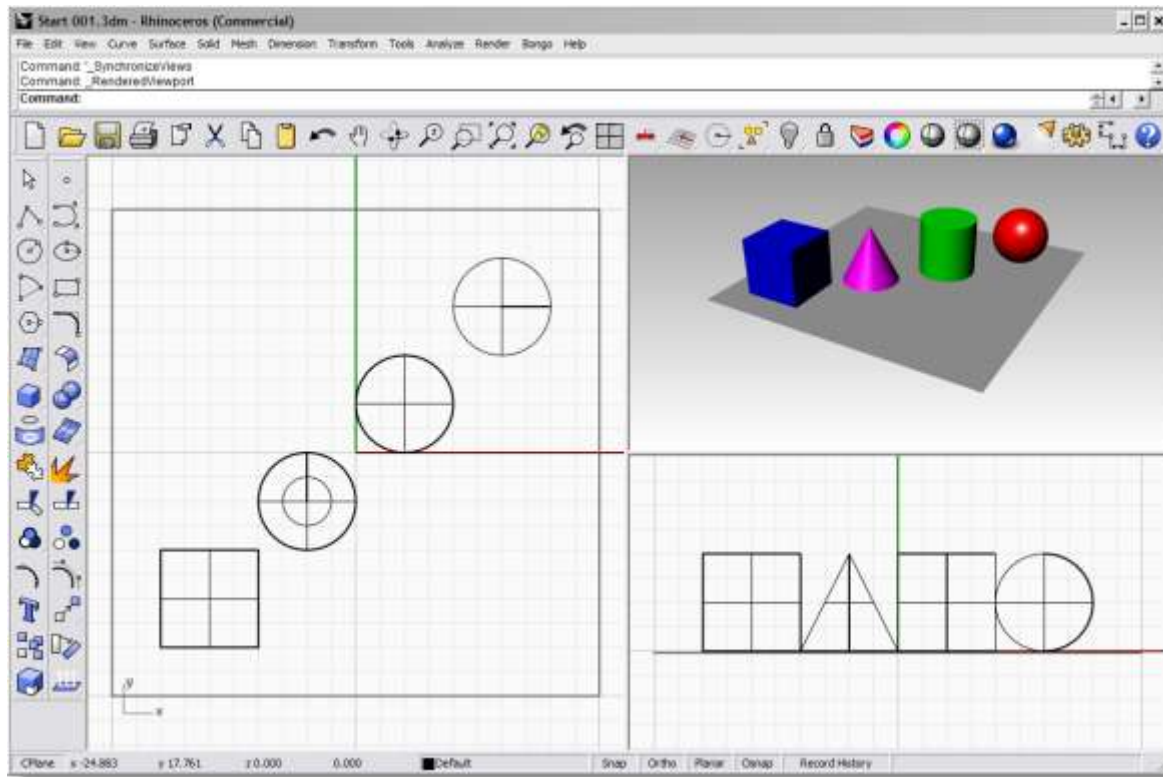
## Notes:

### Exercise 1—Rhino basics

#### To get started:

- 1 On the **Rhino Help** menu, click **Learn Rhino, Open Tutorial Models**.
- 2 **Double-click** the **Level 1** folder.
- 3 In the **Open** dialog box, select **Start.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.



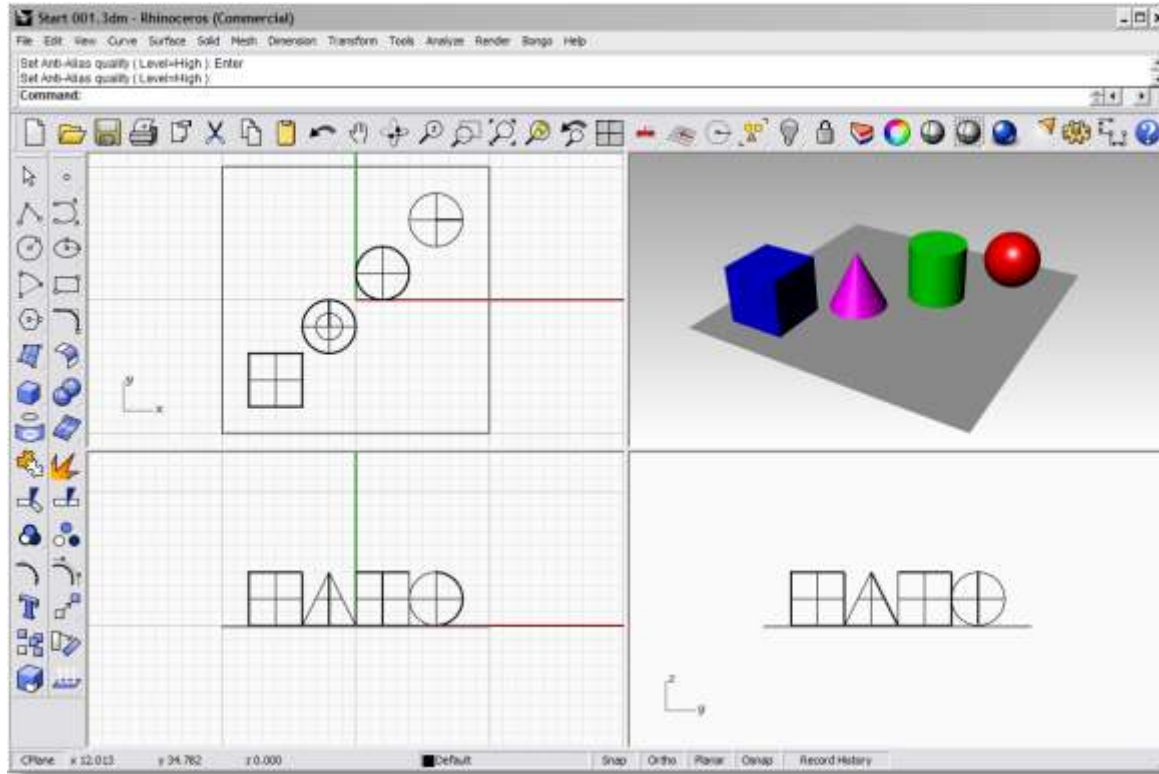
*Two parallel viewports and one perspective viewport.*

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



Open

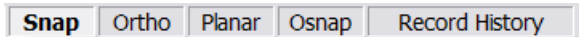
- 4 From the **View** menu, click **Viewport Layout**, and then click **4 Viewports**.



*Three parallel viewports and one perspective viewport*

- 5 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.

**Notes:**



4 Viewports

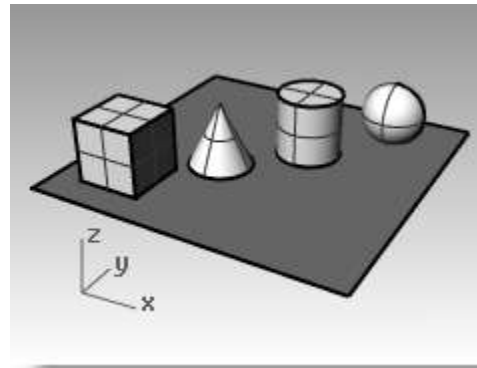
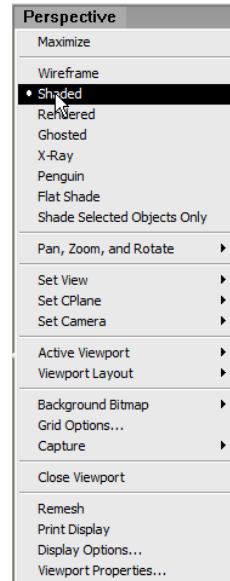
- 6 Click the mouse in the **Perspective** viewport to make it active.

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

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

The objects appear shaded. A shaded viewport 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 viewport display options.



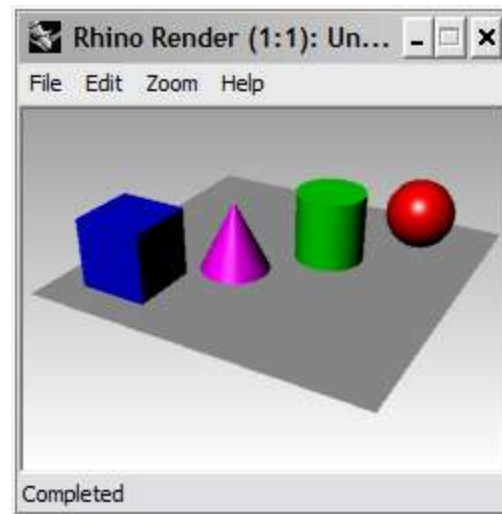
*Shaded display.*

- 8 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.

- 9 Close the render window.



*Render.*

## Notes:



Shaded Viewport

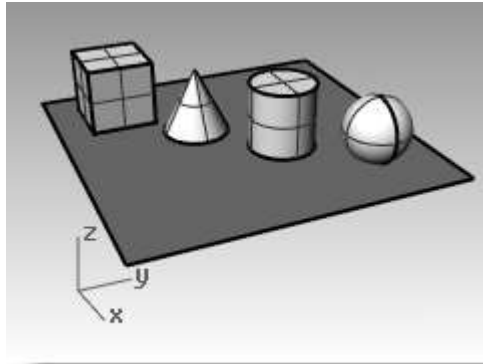


Render

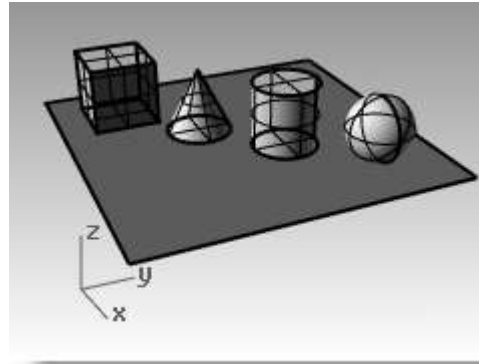
- 10** In the **Perspective** viewport, click and drag with your right mouse button held down to rotate the view.

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

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



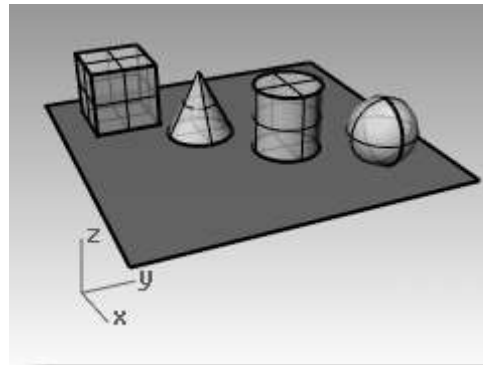
*Rotate the view in shaded display.*



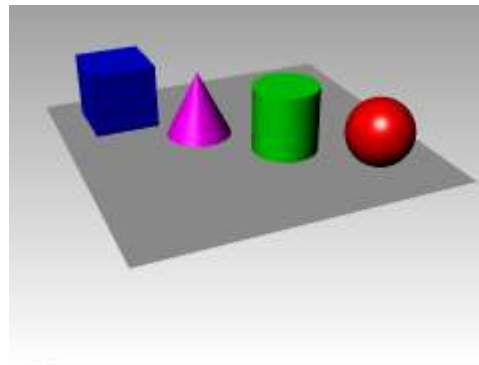
*X-Ray shaded display.*

- 12** Right click on the **Perspective** viewport title, and then click **Ghosted**.

- 13** Right click on the **Perspective** viewport title, and then click **Rendered**.



*Ghosted shade display.*



*Rendered display*

## Notes:



X-Ray Viewport



Ghosted Viewport



Rendered Viewport



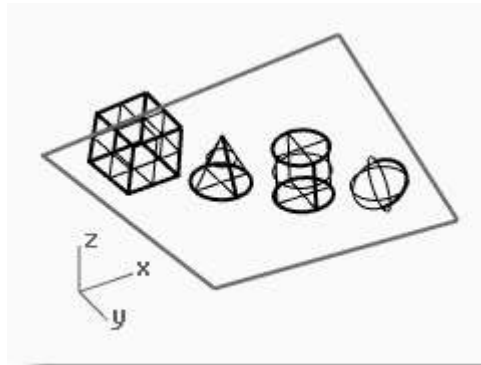
**14** Change to **Wireframe** mode.

**15** To rotate your view, drag from the bottom of the view toward the top.

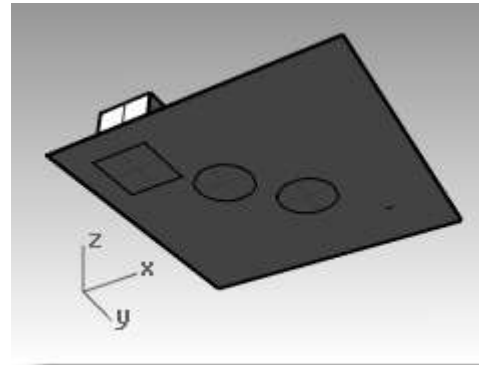
You are now under the objects looking up.

**16** Change to **Shaded** mode.

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



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



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

### Notes:



Wireframe Viewport  
Right Click

### 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** twice.  
This takes you back to the default viewport settings.

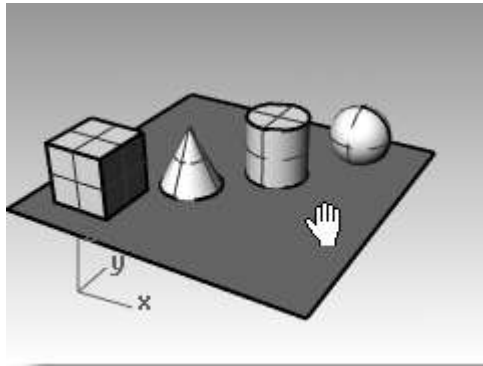
## Navigating Around the Model

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.

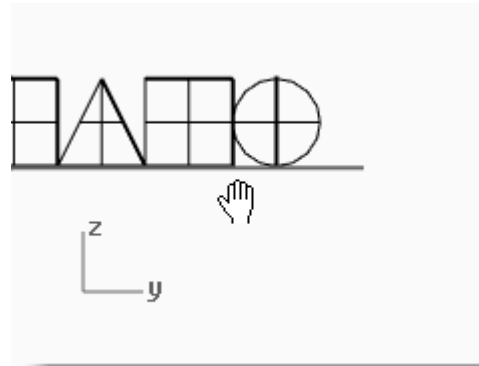
### To pan in a viewport:

- 1 In the **Perspective** viewport, hold the Shift and drag with the right mouse button to pan the view.
- 2 **Pan** the view in the parallel viewports by dragging with the right mouse button.

In the parallel viewports it's not necessary to press the Shift key.



*Panning with Shift and the right mouse button.*



*Panning in a parallel view with the right mouse button.*

### 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 turn the mouse wheel to zoom in and out. If you don't have a wheel mouse, hold down the **Ctrl** key and drag up and down in a viewport with the right mouse button.

### To zoom in and out:

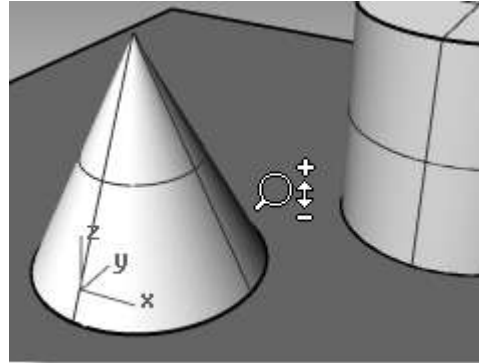
- 1 In the Perspective Viewport, roll the wheel on your mouse forward to zoom in, roll it backward to zoom out.

The camera zooms at cursor position.

- 2 In the **Perspective** 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, displays 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 **Zoom 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 **Zoom Extents All**.

## Notes:



Zoom Extents  
Left Click



Zoom Extents All Viewports  
Right Click

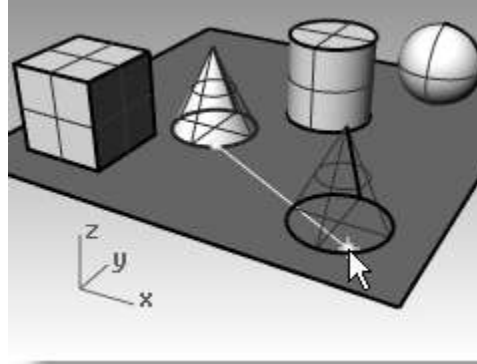
## Move Objects

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.

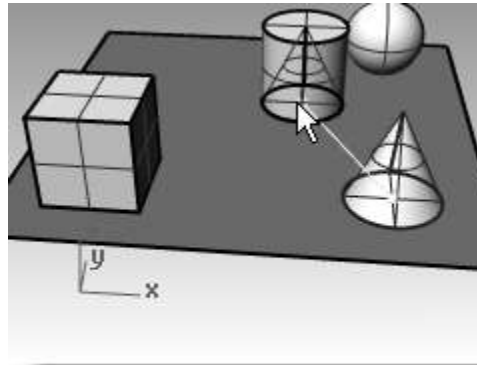
### To move objects:

- 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.



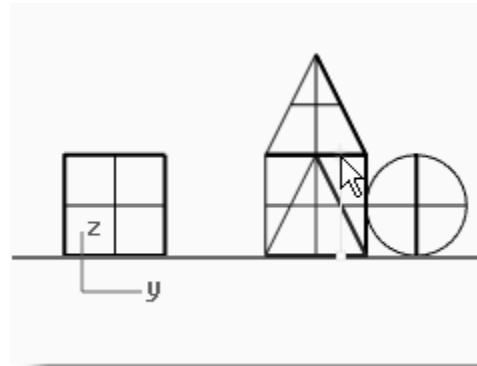
*Drag the cone to move it.*

**Notes:**

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

Watch what happens in the Perspective viewport.

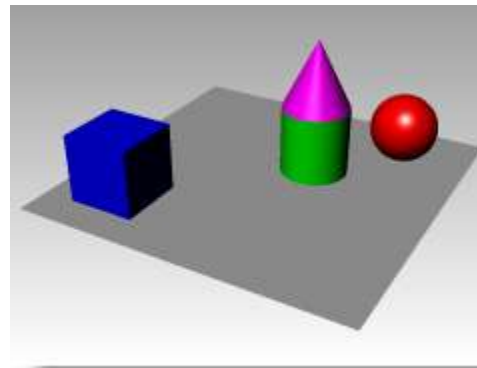
There are many times when you have to watch what is happening in other viewports to accurately place your objects.



*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.

**Notes:**

## Copy Objects

To create more objects, copy the shapes.

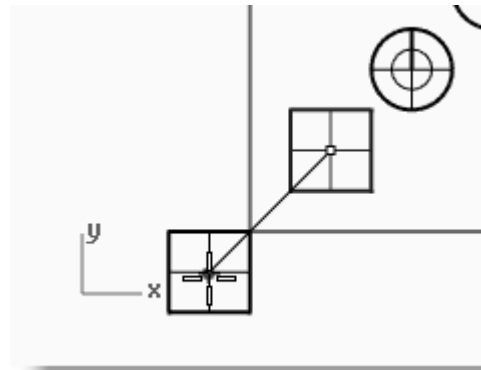
### To start with a new model:

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

### To copy objects:

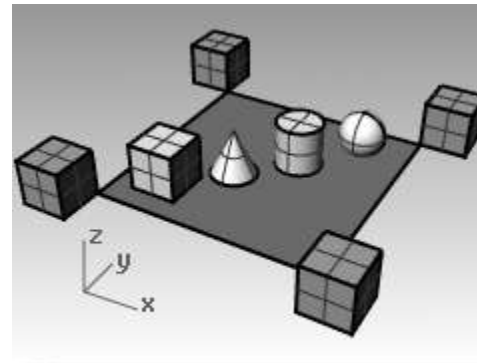
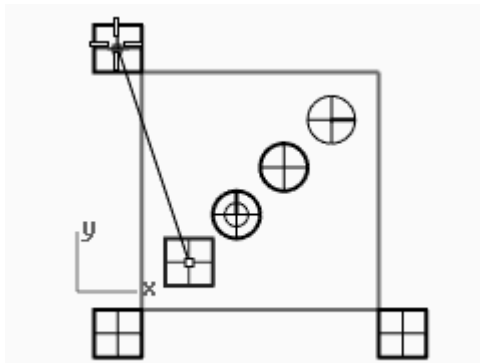
- 1 Click the box to select it.
- 2 From the **Transform** menu, click **Copy**.
- 3 Click somewhere in the **Top** viewport.

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



Select and copy the box.

- 4 Click where you want the first copy.  
Zoom in closer if you like.
- 5 Click other places to make more copies of the box.
- 6 When you have enough copies, press **Enter**.



## Notes:



Copy

### **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.

Most viewport controls can be accessed through the viewport popup menu.

To open the popup menu, right click the viewport title.

### **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 navigate:

<b>Key</b>	<b>Action</b>	<b>+ Ctrl</b>
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	Undo View Change	
End	Redo View Change	

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.

There are additional Zoom controls that will be discussed in other exercises.

## 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, 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:

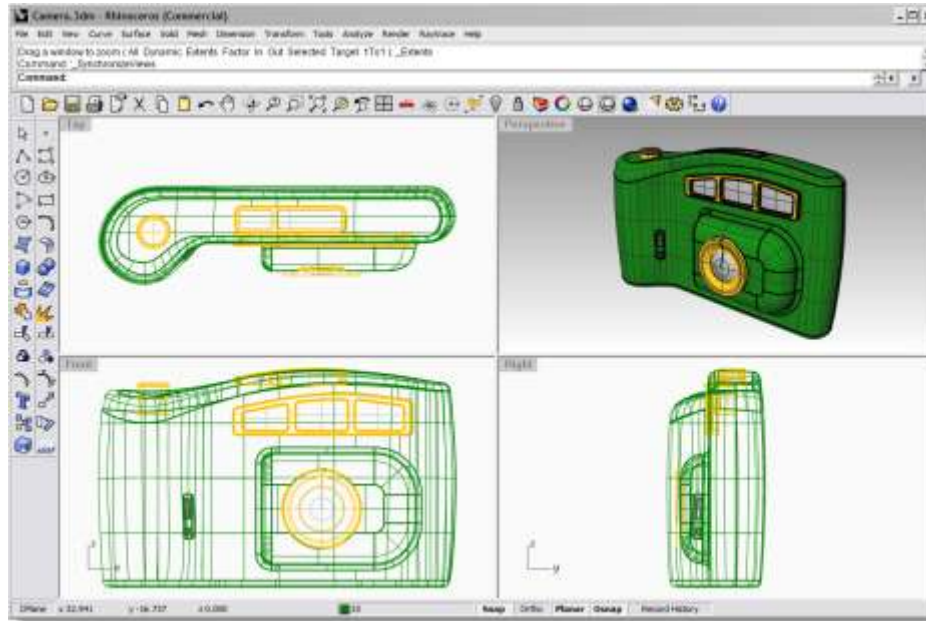
- ▶ From the **View** menu, click **Zoom**, and then click **Zoom Extents All**.



## Exercise 2—Display options

### ► Open the model **Camera.3dm.**

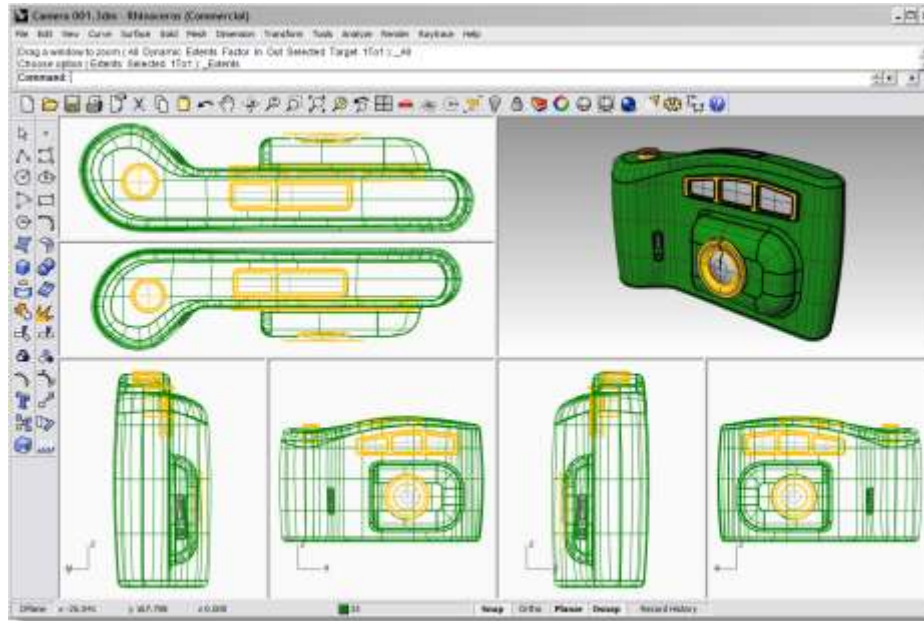
You will use this to practice changing views. You will create views from six directions and an oblique perspective view.



**Notes:**




### To change the number of viewports:

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



Each viewport is Split down the middle either horizontally or vertically.

### 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.

### Notes:



Split Viewport Horizontally



Split Viewport Vertically



Bottom View



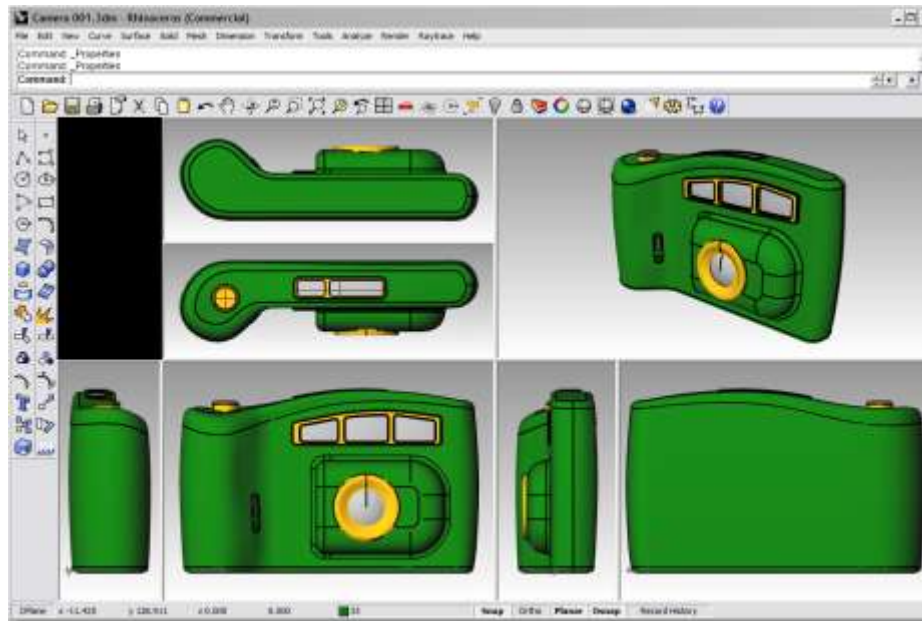
Left View



Back View

### To synchronize the viewports:

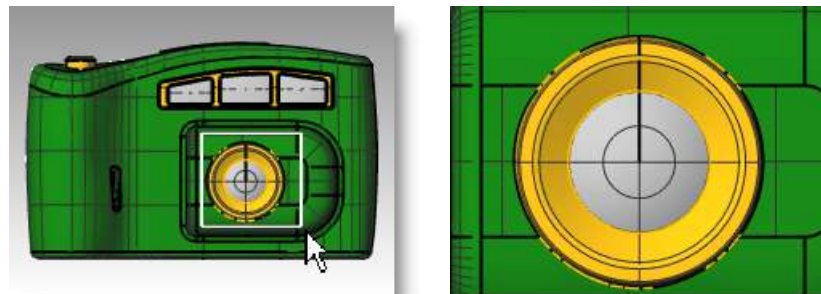
- 1 Adjust the size shape of the viewports.
- 2 Make the **Front** viewport active.
- 3 From the **View** menu, click **Zoom**, and then click **Zoom Extents**.
- 4 Right click on the **Front** viewport title, click **Set Camera**, and then click **Synchronize Views**.
- 5 Change the viewport displays to one of the shaded viewport settings.



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

### To zoom to a window:

- 1 From the **View** menu, click **Zoom**, and then click **Zoom Window**.
- 2 Click and drag a window around a portion of the model.



### Notes:



Synchronize Views

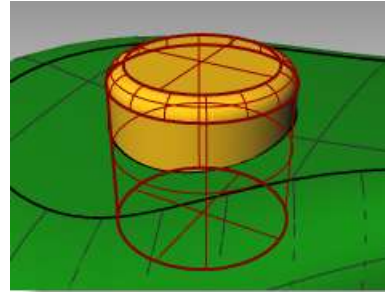


Zoom Window

### To zoom a selected object:

- 1 Select the shutter release.
- 2 From the **View** menu, click **Zoom**, and then click **Zoom Selected**.

The view zooms to the selected object.



### Notes:



Zoom Selected

### To rotate the view:

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

### 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**

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## 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 joined segments (a single linear curve with multiple segments).

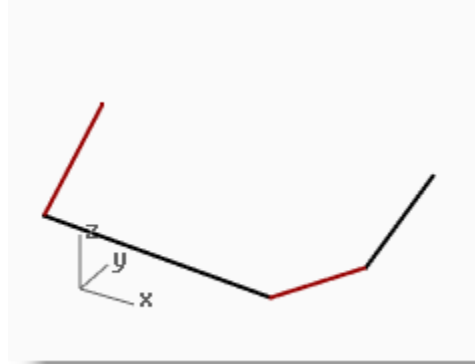
Option	Description
Close	Closes the shape by drawing a segment from the last point picked to the first point picked. This ends the command.
Undo	Deletes the last point picked.

### Exercise 3—Drawing lines

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

### To draw line segments:

- 1 From the **Curve** menu, click **Line**, and then click **Line Segments** to begin the **Lines** command.
- 2 Pick a point in a viewport.
- 3 Pick another point in a viewport.  
A line segment appears between the two points.
- 4 Pick another point.
- 5 Continue to pick points.  
Additional segments appear.  
Each segment meets but is not joined to the previous segment.
- 6 Press **Enter** to end the command.



### Notes:



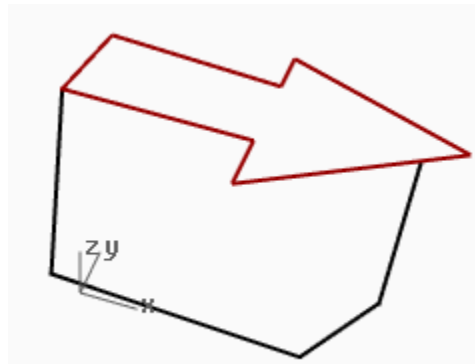
Line Segments  
Right Click

### To use the Close option:

- 1 Repeat the **Lines** command.
- 2 Pick a **Start** point.
- 3 Pick 3 or 4 more points.
- 4 Click **Close**.  
The last line will end at the original start point. Line segments are individual lines that meet at a common endpoint.

### To draw a polyline:

- 1 From the **Curve** menu, click **Polyline**, and then click **Polyline** to begin the **Polyline** command.
- 2 Pick a **Start** point.
- 3 Pick 3 or 4 more points.
- 4 Press **Enter** when done.  
This makes an open polyline. A polyline is made of line segments that are joined together. It is one object.



Polyline  
Left Click



### To use the Undo option:

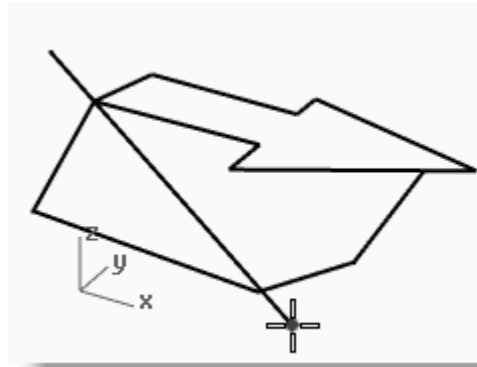
- 1 Repeat the **Polyline** command.
- 2 Pick a **Start** point.
- 3 Pick 3 or 4 more points.
- 4 Click **Undo** on the command line.

Notice that your cursor moves back to the previous point and one segment of the polyline is removed.

- 5 Continue to pick points.
- 6 Press **Enter** or click **Close** to end the command.

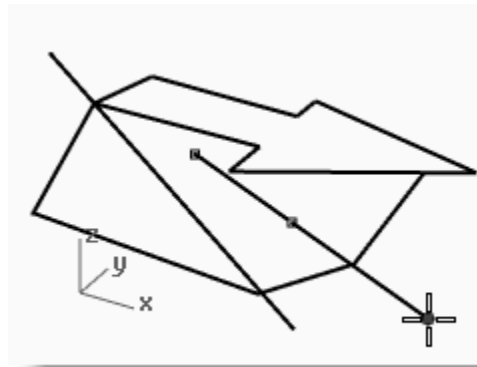
### To draw a single line segment:

- 1 From the **Curve** menu, click **Line**, and then click **Single Line** to begin the **Line** command.
  - 2 Pick a **Start** point.
  - 3 Pick an **End** point.
- The command ends after one segment is drawn.



### To use the Bothsides option:

- 1 From the **Curve** menu, click **Line**, and then click **Single Line** to begin the **Line** command.
  - 2 Click **Bothsides** on the command line.
  - 3 Pick a **Middle** point.
  - 4 Pick an **End** point.
- A segment is drawn with equal length on both sides of the middle point.



### Notes:



Line

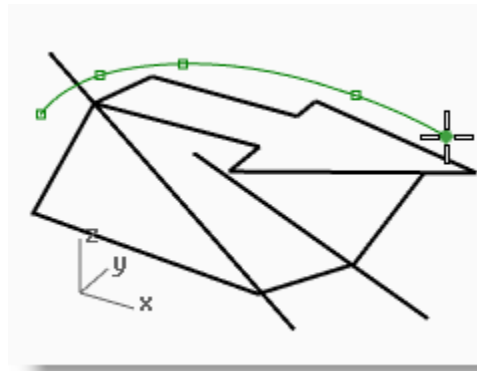
## Drawing Free-form Curves

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.

Option	Description
<u>C</u> lose	Closes the shape by drawing 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 4—Drawing interpolated curves

- 1 From the Curve menu, click **Free-form**, and then click **Interpolate Points**.
- 2 Pick a **Start** point.
- 3 Continue picking points.
- 4 Click **Close** to make a closed curve or, press **Enter** to end the command.



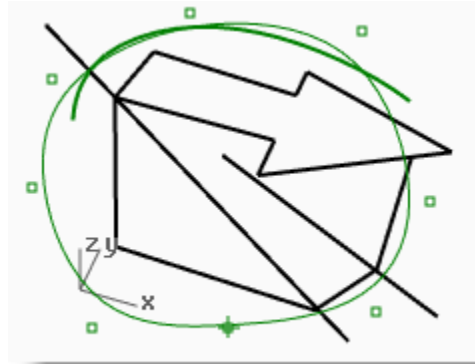
### Notes:



Curve: Interpolate Points

### Exercise 5—Drawing curves from control points

- 1 From the **Curve** menu, click **Free-form**, and then click **Control Points**.
- 2 Pick a **Start** point.
- 3 Continue picking points.
- 4 Click **Close** to make a closed curve or, press **Enter** to end the command.



### Notes:

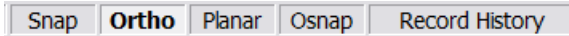


Control Point Curve

*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.



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

### Snap

Forces the marker to snap on grid intersections.

You can also toggle **Snap** on and off by pressing **F9** or typing the letter **S** and pressing **Enter**.

### Ortho

Restricts cursor movement to the points at a specified angle from the last point created. The default angle is 90 degrees.

You can also toggle **Ortho** on and off by pressing **F8** or pressing and holding the **Shift** key down.

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. F8 or Shift.

## Planar

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.

You can also toggle **Planar** On-Off by typing the letter **P** and pressing Enter.

## History

Records history and updates history-aware objects. With History recording and Update turned on, a lofted surface can be changed by editing the input curves.

In general, it is best to leave the **Record** option set to **No** and use the **Record History** status bar pane to selectively record history. Recording history uses computer resources and makes saved files larger.

## Grid

Pressing **F7** hides or shows a reference grid in the current viewport of the graphics screen at the construction plane.

### ***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

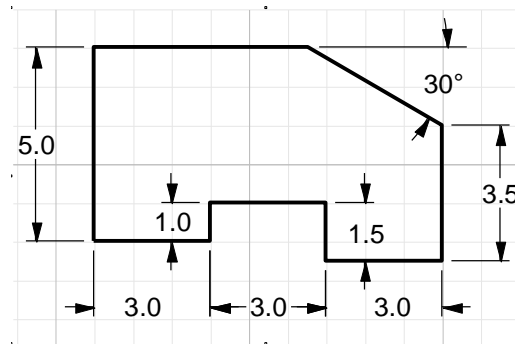
In Rhino you can create full-size models using precise measurements. You might need to change the modeling environment depending on the type of model you are creating; the default options may not always work.

### 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.
- 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.
- 11 Try to draw the closed polyline to the right with **Snap** and **Ortho** turned on.



## Notes:



Document Properties

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

### To reset the modeling aids options:

- 1 From the **Tools** menu, click **Options**.
- 2 In the **Rhino Options** dialog box, click **Modeling Aids**.
- 3 Change the **Ortho** options to snap every **90** degrees.

## Saving Your Work

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

### To save your model:

- ▶ From the **File** menu, click **Save**.  
Or, click one of the other options. You will have an opportunity to save your work.

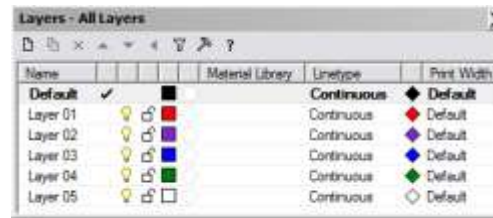
Command	Description
Save	Saves your model and keeps it open.
SaveSmall	Save your model without render or analysis meshes and preview image to minimize file size.
IncrementalSave	Save sequentially numbered versions of your model.
SaveAs	Saves your model to a specified file name, location, and format.
SaveAsTemplate	Save as a template.

## 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.



### Notes:



Save

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

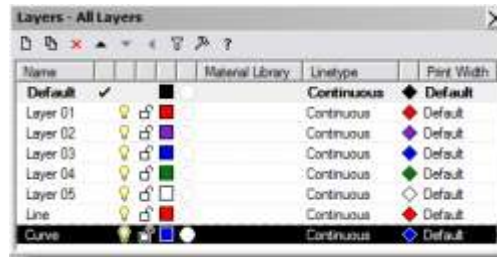


Edit Layers

## Exercise 7—Layers

### To create a new layer:

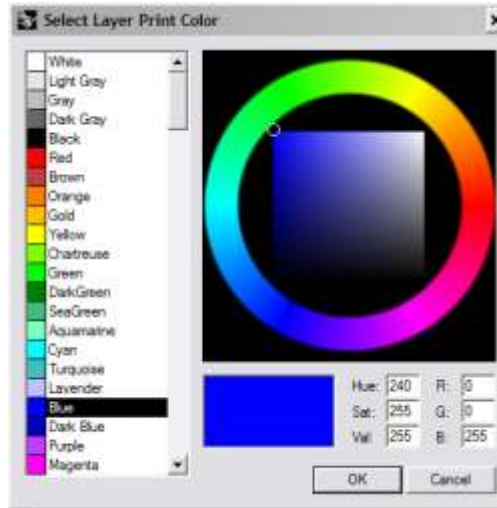
- 1 From the **Edit** menu, click **Layers**, and then click **Edit Layers**.
- 2 In the **Layers** window, click **New**.
- 3 The new layer, **Layer 06**, appears in the list. Type **Line** and press **Enter**.
- 4 Click **New**.
- 5 The new layer, **Layer 06**, appears again. Type **Curve** and press **Enter**.



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.

### 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,



## Notes:

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

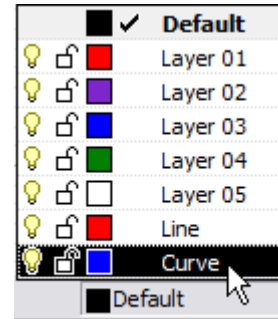
Hue is the color that is referred to as a scale ranging from red through yellow, green and blue and then circularly back to red.

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.

### To make a layer current:

- 1 In the **Status Bar**, click the **Layer** pane.
- 2 In the **Layer popup**, click **Line**.
- 3 Draw some lines.  
The lines are on the Line layer and they are colored red.
- 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.



*Clicking the name or the checkbox sets the current layer.*

### To lock a layer:

- 1 In the **Status Bar**, click the **Layer** pane.
- 2 In the **Layer popup**, click the **Lock** icon on the row for the **Line** layer.  
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 In the **Status Bar**, click the **Layer** pane.
- 2 In the **Layer popup**, click the **On/Off** icon (light bulb) in the row for **Curve**.  
Turning a layer off makes all objects on it invisible.

### Notes:



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**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 the right of the objects you want to select.
- 2 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.
- 3 Release your mouse button.  
All objects inside or touching the box will be selected.
- 4 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.

### 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:



Hide Objects



Show Objects



Lock Objects



Unlock Objects



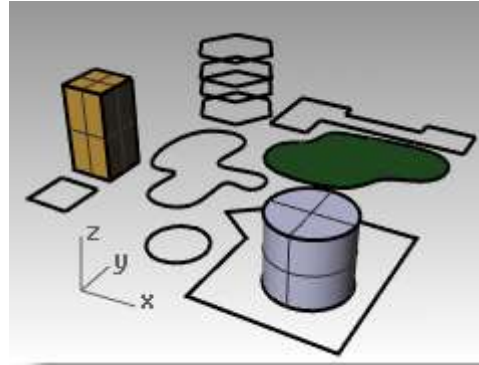
Change Object Layer

## Selecting Objects

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

### Exercise 9—Practice using selection options

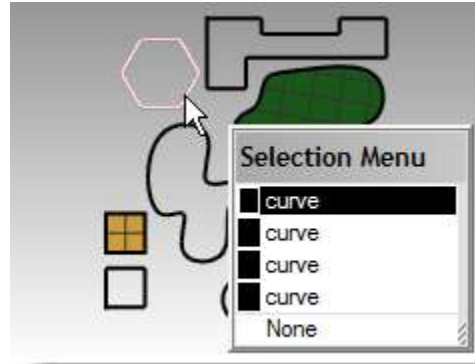
- 1 From the **File** menu, click **Open**.
- 2 In the **Open** dialog box click **Delete.3dm** and click **Open**, or double-click **Delete.3dm** to open the model.
- 3 Select the square and the circle.
- 4 From the **Edit** menu, click **Delete** or press the **Delete** key.  
The objects disappear.



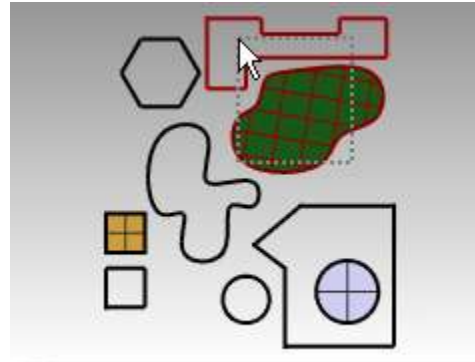
**Notes:**

### To start the first practice:

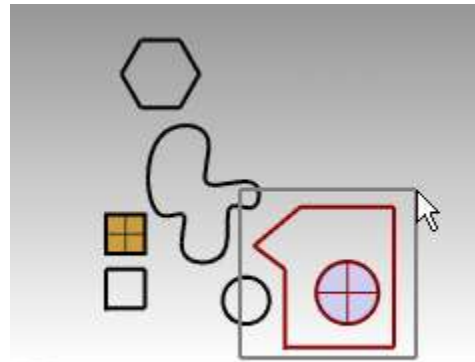
- 1 Select one of the lines on the hexagon in the **Top** viewport.  
Because there are several curves superimposed on each other, the selection menu appears, allowing you to select one of the curves.
- 2 Select the top curve from the list.
- 3 From the **Edit** menu, click **Delete**.  
Observe one 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 Hold the **Ctrl** key down and click on the cylinder to remove it from the selection set.
- 8 From the **Edit** menu, click **Delete**.
- 9 Continue deleting objects in the drawing.  
Practice using different selection methods to select and deselect objects. Use crossing and window. The **Shift** key while selecting will let you add to your selection set. The **Ctrl** key while selecting will let you to remove objects from your selection set.



### Notes:

## Notes:

### To undo and redo deletions:

- 1 From the **Edit** menu, click **Undo**.

Each time you click, **Undo** takes you back one command.











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

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

- 3 **Undo** all the deletions you made in the previous exercise.

### Additional selection options

In addition to the selection options we just practiced, there are several other useful tools for selecting objects. In the next exercise we will use some of these tools.

Command	Button	Menu Label	Description
SelAll		All Objects (Ctrl+A)	Select all objects.
SelNone		None (Esc)	Deselect all objects. Note: SelNone does not run inside a command to clear pre-selected objects.
Invert		Invert	Deselects all selected objects and selects all visible objects that were not previously selected.
SelPrev		Previous Selection	Re-selects the previous selection set.
SelLast		Last Created Objects	Selects the last changed objects.
SelPt		Points	Select all point objects.
SelCrv		Curves	Select all curves.
SelPolyline		Polylines	Select all polylines.
SelSrf		Surfaces	Select all surfaces.
SelPolysrf		Polysurfaces	Select all polysurfaces.

### To select objects using select tools:

- 1** From the **Edit** menu, click **Select Objects**, and then click **Curves**.  
All the curves are selected.
- 2** From the **Edit** menu, click **Select Objects**, and then click **Invert**.  
Everything but the previously selected curves is selected.
- 3** From the **Edit** menu, click **Select Objects**, and then click **None**.  
Everything is unselected.
- 4** From the **Edit** menu, click **Select Objects**, and then click **Polylines**.  
All of the polylines are selected.
- 5** From the **Edit** menu, click **Select Objects**, and then click **Surfaces**.  
The single surface is added to the selection set.
- 6** From the **Edit** menu, click **Select Objects**, and then click **Polysurfaces**.  
The polysurfaces are added to the selection set.
- 7** From the **Edit** menu, click **Select Objects**, and then click **None**.
- 8** Draw a few lines and curves.
- 9** From the **Edit** menu, click **Select Objects**, and then click **Last Created Objects**.  
The things you just created are selected.

### Notes:



Select Curves



Invert Selection



Select Polylines



Select Surfaces



Select Polysurfaces



Select None



Select Last Created Object

# 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.

### 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 **Small Object - 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.

#### Notes:

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

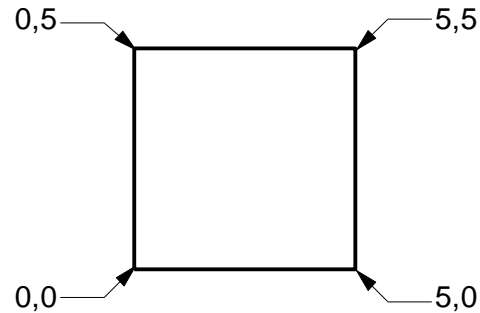
*You can change the tolerance after you start, but objects created before the change still have the old tolerance value.*



Cross-shaped cursor

### Exercise 11—Entering absolute coordinates

- 1 Double-click the viewport title to maximize the **Top** viewport.
- 2 From the **Curve** menu, click **Polyline**, and then click **Polyline**.
- 3 To **Start** type **0** and press **Enter**.  
If you are going to start at the origin of the sheet  $(0,0,0)$  you can simply type **0** as a shortcut.
- 4 For the **Next point** type **5,0** and press **Enter**.
- 5 For the **Next point** type **5,5** and press **Enter**.
- 6 For the **Next point** type **0,5** and press **Enter**.
- 7 Click **Close** to close the polyline.



### Notes:



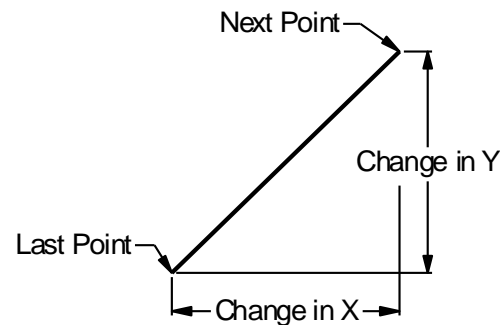
### Relative Coordinates

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*.

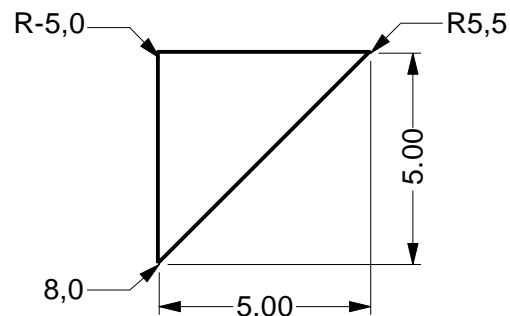
Relative coordinates are based on the last point entered, instead of on the origin  $(0,0,0)$  of the construction plane.

Precede the  $x,y,z$  coordinates with a single **R** (upper or lower case) to enter relative coordinates. Use the **@** symbol instead of an **R** to start relative coordinates if you prefer.



### Exercise 12—Entering relative coordinates

- 1 From the **Curve** menu, click **Polyline**, and then click **Polyline**.
- 2 To **Start** type **8,0** and press **Enter**.  
These are absolute coordinates.
- 3 For the **Next point** type **R5,5** and press **Enter**.  
These are relative coordinates.
- 4 For the **Next point** type **R-5,0** and press **Enter**.
- 5 Click **Close** to close the polyline.





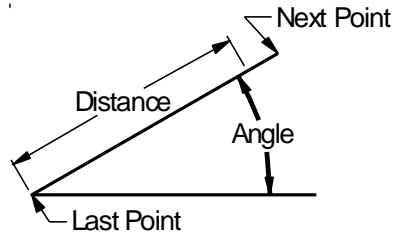
## Polar Coordinates

**Polar coordinates** specify a point that is a distance and direction away from 0,0 in the current construction plane.

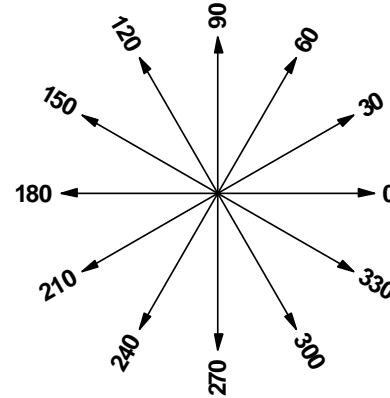
Vector directions in Rhino start with Zero degrees at 3 o'clock on a standard clock. They change in an anti-clockwise direction as illustrated below.

For example, if you want a point four units away from the construction plane origin, at a 45° angle anticlockwise from the construction plane x-axis, type 4<45, and press **Enter**.

Relative polar coordinates are preceded by **R** or **@**; absolute polar coordinates are not.

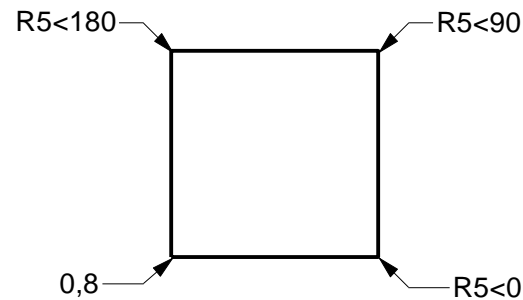


*Instead of using x-, y-, and z-coordinates, enter relative polar coordinates like this: Rdistance<angle.*



### Exercise 13—Entering polar coordinates

- 1 From the **Curve** menu, click **Polyline**, and then click **Polyline**.
- 2 To **Start** type **0,8** and press **Enter**.
- 3 For the **Next point** type **R5<0** and press **Enter**.
- 4 For the **Next point** type **R5<90** and press **Enter**.
- 5 For the **Next point** type **R5<180** and press **Enter**.
- 6 Click **Close** to close the polyline.



**Notes:**

## 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.

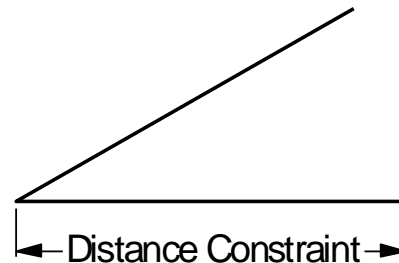
Using angle constraint entry, you can specify an angle by typing < followed by a value and pressing **Enter**. The next point is constrained to lines at multiples of the angle relative to the x-axis you specified.

### Using the Shift key to toggle Ortho on and off:

When Ortho is off, 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 five units long using distance constraints.

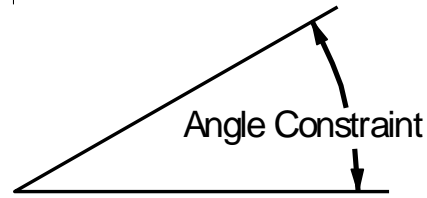
#### Exercise 14—Distance constraint entry

- 1 From the **Curve** menu, click **Polyline**, and then click **Polyline**.
- 2 To **Start** type **8,8** and press **Enter**.
- 3 For the **Next point** type **5** and press **Enter**.
- 4 Hold the **Shift** key down and pick a point to the right.  
Ortho constrains the marker to 0 degrees.
- 5 For the **Next point** type **5** and press **Enter**.
- 6 Hold the **Shift** key down and pick a point up.  
Ortho constrains the cursor to 90 degrees
- 7 For the **Next point** 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.
- 9 Click **Close** to close the polyline.



### Exercise 15—Distance and angle constraint entry

- 1 From the **Curve** menu, click **Polyline**, and then click **Polyline**.
- 2 To **Start** type **16,5** and press **Enter**.
- 3 For the **Next point** 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.
- 4 Pick a point down and to the right.  
The angle constraint sets the angle.
- 5 For the **Next point** type **5** and press **Enter**, then type **<45** and press **Enter**.
- 6 Pick a point up and to the right.  
The angle constraint sets the angle.
- 7 For the **Next point** type **5** and press **Enter**, then type **<45** and press **Enter**.
- 8 Pick a point up and to the left.  
The angle constraint sets the angle.
- 9 Click **Close** to close the polyline.
- 10 Save your model. You will use this model for another exercise.

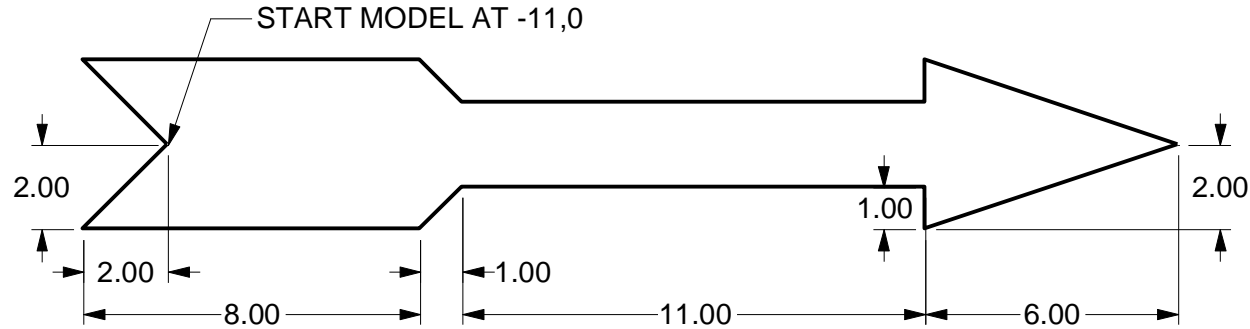


**Notes:**

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

- 1 Start a new model, use the **Small Objects - Millimeters.3dm** template. Save as **Arrow**.

Since the object is symmetrical, you will only draw the lower half of the model.



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.

The following is an example of the command line input that you might use:

**Absolute x,y**

- 2 From the **Curve** menu, click **Polyline**, then click **Polyline**.
- 3 For the **Start of Polyline** type **-11,0**.

**Relative x,y**

- 4 For the **Next point**, type **r-2,-2**.

**Distance constraint**

- 5 For the **Next point**, type **8** and press Enter, turn **Ortho** on and then pick to the right.

**Relative x,y**

- 6 For the **Next point**, type **r1,1**.

**Relative polar**

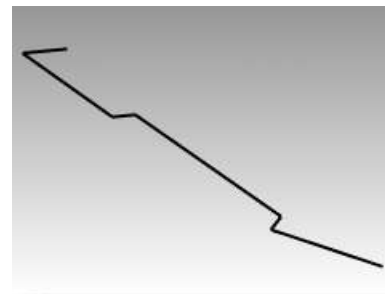
- 7 For the **Next point**, type **r11<0**.

**Distance constraint**

- 8 For the **Next point**, type **1** and press Enter, turn **Ortho** on and then pick down.

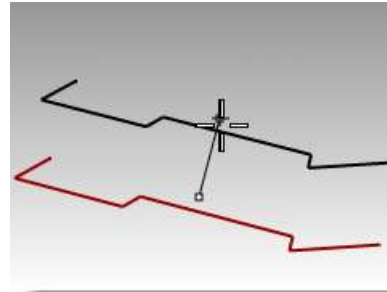
**Relative x,y**

- 9 For the **Next point**, type **r6,2**.
- 10 For the **Next point**, press **Enter** to end the command.
- 11 Save your model.



### To make a copy of the polyline:

- 1 Select the polyline.
- 2 From the **Transform** menu, click **Copy**.
- 3 For the **Point to copy from**, pick a point near the polyline.
- 4 For the **Point to copy to**, type **6** and press **Enter**, turn **Ortho** on and pick above the selected polyline.



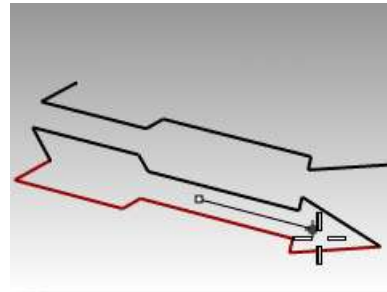
### Notes:



Copy

### To make a mirror image of the polyline:

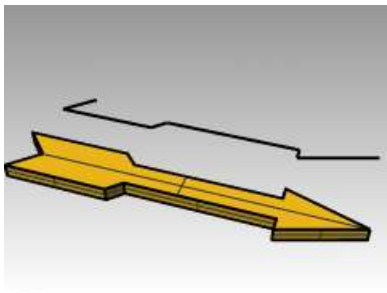
- 1 Select the original polyline.
- 2 From the **Transform** menu, click **Mirror**
- 3 For the **Start of mirror plane**, type **0** and press **Enter**.
- 4 For the **End of mirror plane**, turn **Ortho** on and then pick to the right.



Mirror

### To make it 3-D:

- 1 Select the original polyline and the mirror copy.
- 2 From the **Solid** menu and click **Extrude Planar Curve**, click **Straight**.
- 3 For the **Extrusion distance**, type **1** and press **Enter**.

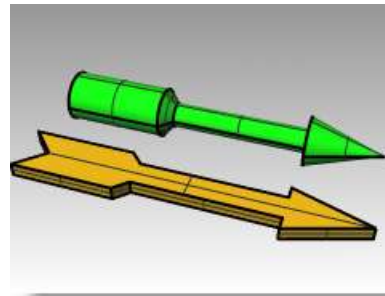
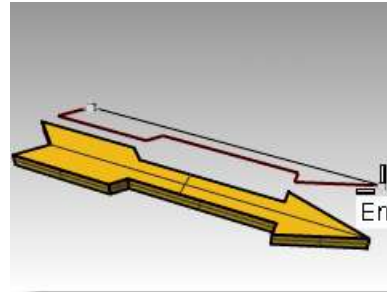


Revolve

**To make it 3-D (alternate):**

- 1** Select the copy of the polyline.
- 2** On the **Status bar**, click **Osnap**.
- 3** On the **Osnap toolbar** check **End**.
  
- 4** From the **Surface** menu, click **Revolve**.
- 5** For the **Start of revolve axis**, select the end of the polyline.
- 6** For the **End of revolve axis**, select the other end of the polyline along the centerline.
  
- 7** Press **Enter** to use the default **Start angle**.
- 8** Press **Enter** to use the default **Revolution angle**.

*Your arrow is now a three-dimensional model.*

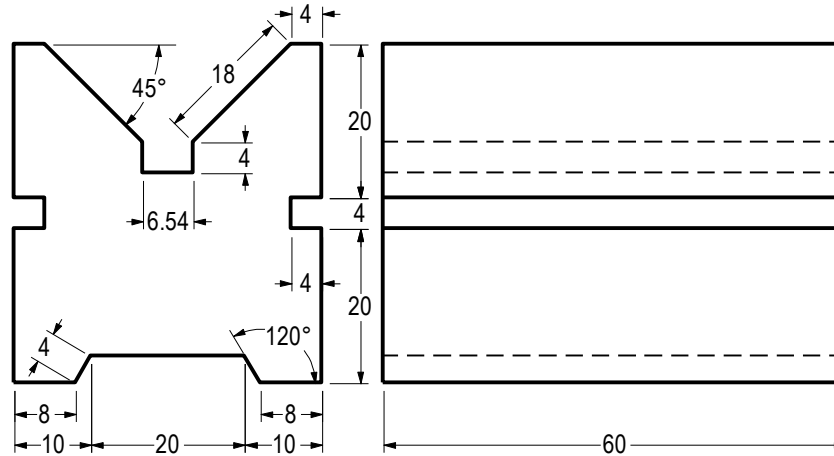


*Your arrow is now a three-dimensional model.*

**Notes:**

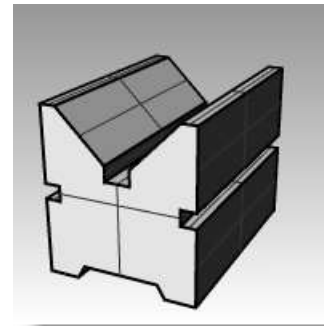
**Exercise 17—Practice using distance and angle constraints**

- 1 Start a new model using the **Small Objects – Millimeters.3dm** 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** 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**, and then click **Straight**.
- 8 For the **Extrusion Distance** type **60** and press **Enter**.  
You can view the model as a three-dimensional object in the **Perspective** viewport.
- 9 Save your model.



**Notes:**



Extrude Planar Curve

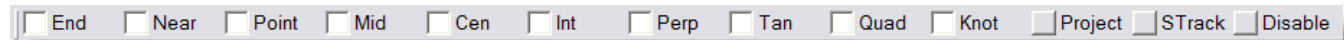
## Object Snaps

**Object Snaps** are tools for specifying points on existing objects. Use osnaps 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. Objects 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
End		End snaps to the end of a curve, surface edge corner or polyline segment end.
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.
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. It doesn't work on the first point that a command prompts you to pick.
Tan		Tangent To snaps to the point on a curve that makes a tangent to the last selected point. It doesn't work on the first point that a command prompts you to pick.
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.
Project		Projects the snap point to the construction plane.
SmartTrack		SmartTrack is a system of temporary reference lines and points that is drawn in the Rhino viewport using implicit relationships among various 3-D points, other geometry in space, and the coordinate axes' directions.
Disable		Temporarily turns off persistent object snaps, retaining settings

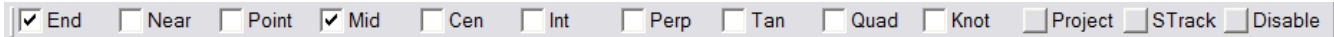


**Exercise 18—Using object snaps**

- 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.*

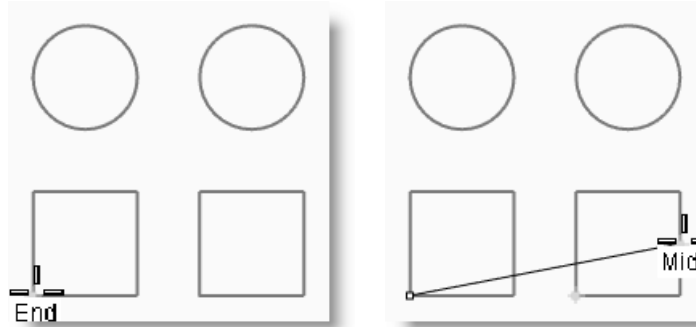
- 2 Check **End** and **Mid**.  
You can check and clear individual object snaps to make modeling with precision easy.

- 3 From the **Curve** menu, click **Polyline**, and then click **Polyline**.

- 4 For the **Start of Polyline**, 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.

The line starts exactly at that corner.

- 5 For the **Next point**, move your cursor close to the midpoint of the vertical line at the right side of the square on the right, pick when the marker snaps to the midpoint.

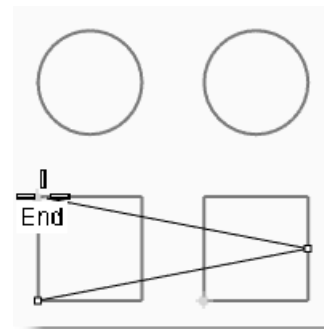


The marker snaps to the midpoint of the line that the cursor touches, making the new line cross to the exact midpoint of that side.

- 6 For the **Next point**, move your cursor close to the end of the line at the upper left corner of the first square, pick when the marker snaps to the end of the line.

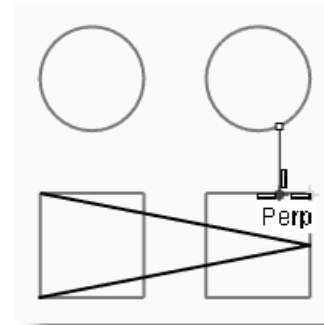
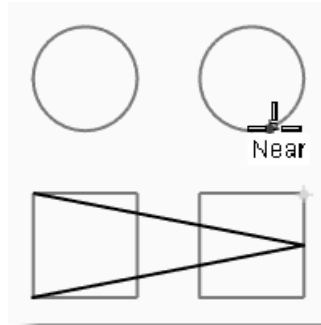
The marker snaps to the end of the line.

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



**Using Near and Perpendicular to object snaps:**

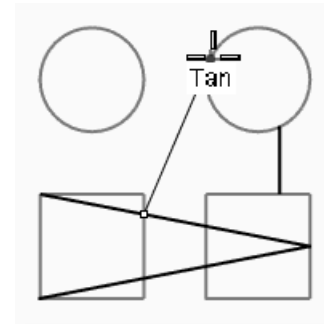
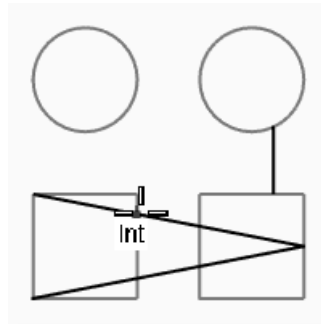
- 1 In the **Osnap** toolbar check **Near** and **Perp**, clear **End** and **Mid**.
- 2 From the **Curve** menu, click **Line**, and then click **Single Line**.
- 3 For the **Start of line**, pick on the lower edge of the circle at the top right.  
The marker snaps to the point on the circle nearest to where the cursor is positioned.
- 4 For the **End of line**, pick the top horizontal edge of the second square.



The marker snaps to a point making a perpendicular to the previous point.

**Using Intersection and Tangent to object snaps:**

- 1 In the **Osnap** toolbar check **Int** and **Tan**; clear **Near** and **Perp**.
- 2 From the **Curve** menu, click **Line**, and then click **Single Line**.
- 3 For the **Start of line**, 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.



- 4 For the **End of line**, pick the top, left edge of the circle on the right.  
The marker snaps to a point tangent to the circle.

### Using Center of object snap:

- 1 In the **Osnap** toolbar check **Cen**, clear **Int** and **Tan**.

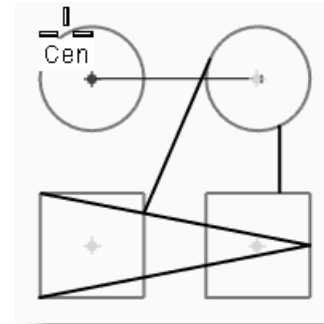
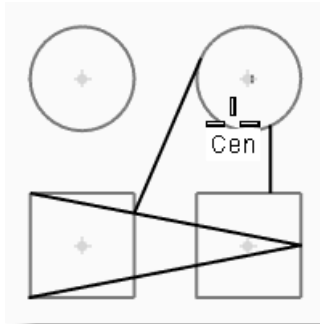
- 2 From the **Curve** menu, click **Line**, and then click **Single Line**.

- 3 For the **Start of line**, pick on the edge of a circle.

The marker snaps to the center of the circle.

- 4 For the **End of line**, pick on the edge of the other circle.

The marker snaps to the center of the circle.



### Using Quadrant of object snap:

- 1 In the **Osnap** toolbar check **Quad**, clear **Cen**.

- 2 From the **Curve** menu, click **Polyline**, and then click **Polyline**.

- 3 For the **Start of Polyline**, pick a point on the top edge of the first circle.

The marker snaps to the quadrant point of the circle.

- 4 For the **Next point**, pick the left edge of the circle.

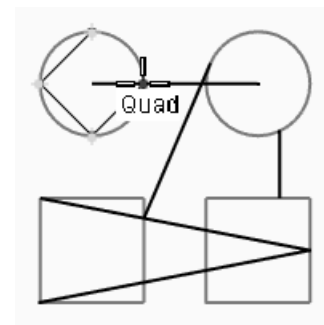
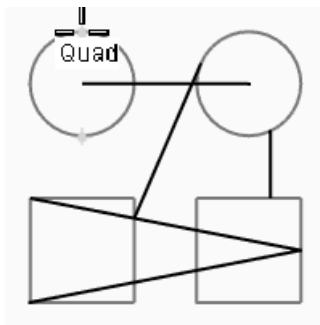
The marker snaps to the quadrant point of the circle.

- 5 For the **Next point**, pick the bottom edge of the circle.

- 6 For the **Next point**, pick the right edge of the circle.

- 7 Click **Close** to finish.

- 8 Use the **SaveAs** command to save the model. Name it **Analyze**. We will use it for a later exercise.



**Notes:**

## Other modeling aids

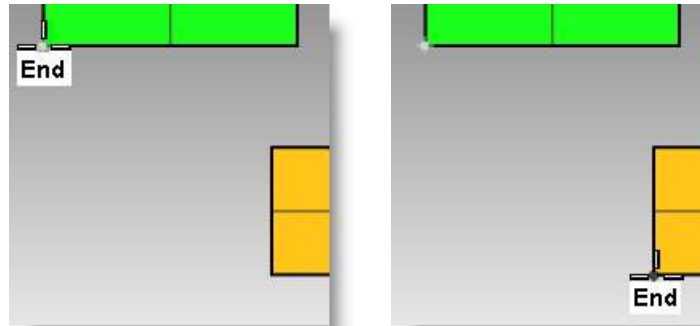
As well as allowing the user to work in a completely unconstrained free-form manner Rhino has a number of modeling aids and constraints that help with precision modeling. This section outlines these aids and constraints.

### Exercise 19—Smart Track

Smart Track creates a set of temporary reference lines and points that work in conjunction with Rhino's object snaps. Using Smart Track obviates the need to create dummy reference lines and points. Smart Track works on 2-D and 3-D objects. It can be used in conjunction with the Project and Planar constraints described later in this section.

#### To use the Smart Track tool:

- 1 Open the **Constraints.3dm** model and maximize the **Top** Viewport.
- 2 Make sure that the following object snaps are turned on: **End, Near, Point, Mid, Cen** and **Int**.
- 3 Turn on **Smart Track** on the **Osnap** toolbar.
- 4 From the **Curve** menu, click **Point Object**, then click **Single Point**.
- 5 Hover the cursor over the lower left corner of the green rectangle, the **End** snap will be displayed and a white marker point becomes visible.
- 6 Repeat this process on the bottom left hand corner of the gold rectangle.

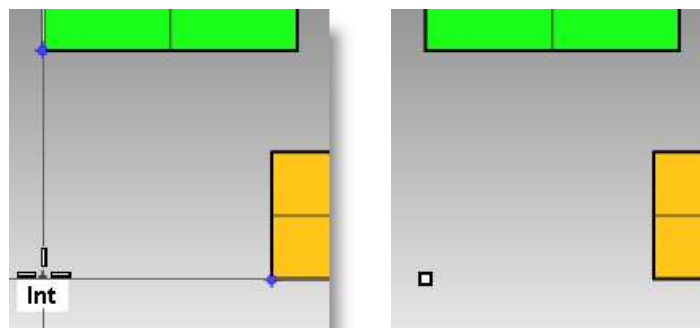


- 7 Now move the cursor towards the apparent intersection point of these two corners. Two temporary construction lines are displayed.

The point is positioned on the intersection of these two construction lines.

- 8 Click to place the **Point Object**.

Smart Track will work with any of the available object snaps. Experiment with some alternatives.



## Exercise 20 – Tab Constraint

The Tab constraint allows the user to fix a direction to a reference point and thereby constrain cursor movement. The following example shows a simplistic use of the Tab constraint.

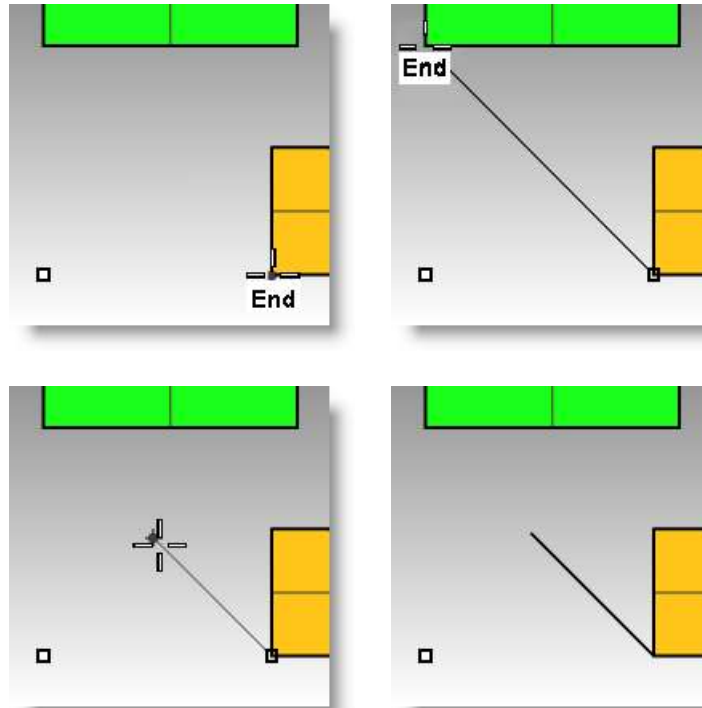
### To use the Tab Constraint:

- 1 We will work with the **Constraints.3dm** file as previous and again maximize Top view.
- 2 Turn off **Smart Track**.
- 3 From the **Curve** menu click **Line**, then click **Single Line**.
- 4 For the **Start of Line** snap to the **End** of the bottom left hand corner of the gold rectangle.
- 5 For the **End of Line**, hover the mouse over the bottom left hand corner of the green rectangle, when the **End** snap appears press the **Tab** key.

Note that the line now becomes white and the direction is constrained.

- 6 For the **End of Line**, drag your mouse to the desired location and click.

The **Tab** constraint will work in conjunction with all object snaps and all tools that require a directional input, for example **Move**, **Copy**, and **Rotate**.



Notes:

## Exercise 21 – Project Constraint

By default 2-D geometry is created on the active construction plane. Object snaps override this behavior and snapping to objects that are not on the construction plane will cause the geometry to be non-planar. The Project constraint ignores object snaps and pushes all the geometry onto the active construction plane.

### To use the Project Constraint:

1 We will again work with the **Constraints.3dm** file as previous.

2 Make sure that the **Ortho** constraint is **On**.

3 Turn **Layer 01** off and turn **Layer 02** on.

The surfaces on Layer 02 are located at different elevations.

4 Working in **Top** viewport, draw a **Polyline** around the perimeter of the three rectangles.

Notice how the how the planar nature of the polyline is overridden by the object snaps.

5 **Delete** the Polyline.

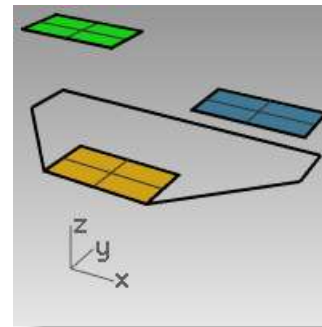
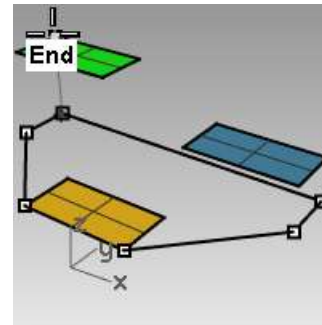
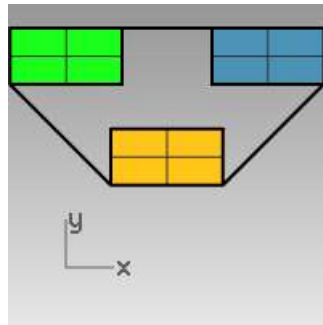
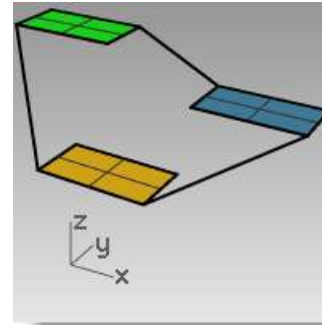
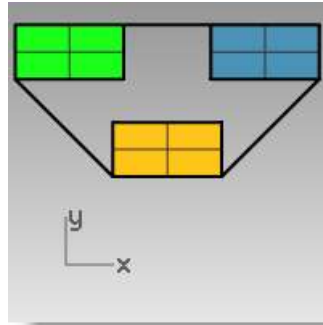
6 Turn on the **Project** constraint on the **Osnap** toolbar.

7 Again, working in **Top** viewport, draw a **Polyline** around the perimeter of the three rectangles.

Look in the Perspective viewport as you draw the polyline and note how the object snaps for the endpoints of the green and blue rectangles are projected to the construction plane.

The **Project** constraint forces all the segments of the polyline onto the construction plane. The resulting polyline is planar.

8 **Save** the File



### Notes:

## Exercise 22 – Planar Constraint

The Planar constraint limits successive picked locations to the same construction plane elevation as the previous location. For example a command such as Polyline can be started off the construction plane and the Planar constraint will override Rhino's default behavior of snapping back to the construction plane.

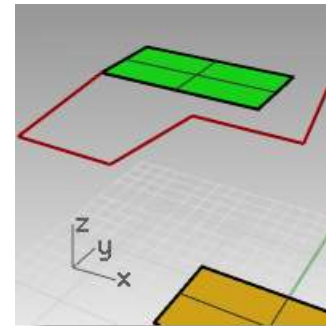
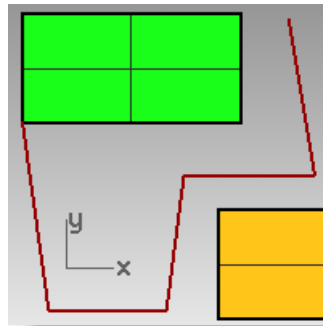
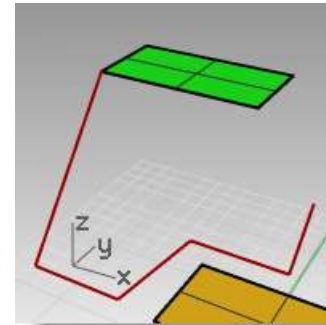
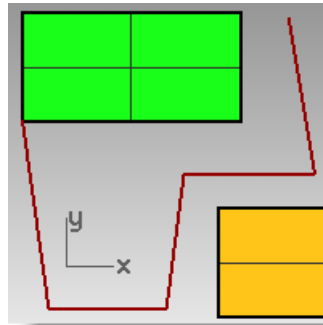
### To use the Planar Constraint:

We will first look at what happens with the Planar constraint Off. Then, we will turn the Planar constraint On to see what changes.

- 1 We will again work with the **Constraints.3dm** file as previous.
- 2 Turn off the **Ortho** and **Planar** constraints.
- 3 Start the **Polyline** command.
- 4 Working in the **Top** Viewport, start the **Polyline** at the lower left corner of the **Green** rectangle.
- 5 Add some more segments without snapping to any of the objects.

Look at the Perspective viewport and notice how the polyline pulls back onto the construction plane after the initial point.

- 6 To override this behavior and create a planar curve **delete** the **Polyline** and turn on the **Planar** constraint.
- 7 Draw the **Polyline** again.  
Note how it now remains planar.
- 8 **Delete** the **Polyline**.



**Notes:**

**To create a Planar curve that starts above the active construction plane and references object snaps at different heights:**

First we will draw a polyline without the Project constraint activated. Then we will use the Planar constraint in conjunction with the Project constraint to see what changes.

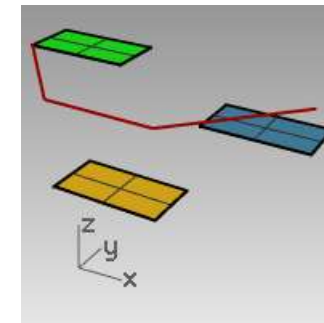
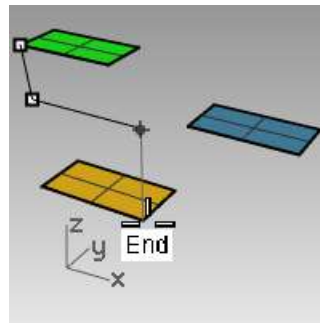
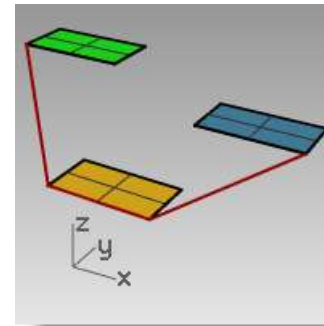
- 1 Turn the **Planar** constraint **On**.
- 2 In the **Top** viewport, draw a new **Polyline** starting again at one of the corners of the **Green** rectangle.
- 3 Create some additional points this time snapping to some of the **Blue** and **Gold** rectangle corner points.

Look at the Perspective view and note how the object snaps override the Planar constraint.

- 4 **Delete** the **Polyline**.

- 5 In the **Perspective** viewport, draw a new **Polyline** starting again at one of the corners of the **Green** rectangle.
- 6 After making the first point, turn the **Project** constraint **On**.
- 7 Create some additional points, snapping to some of the **Blue** and **Gold** rectangle corner points.

Note that the points stay planar to the first point, even though you are snapping to points that are at different elevations.



**Notes:**



## Viewports and Construction Planes

In the following exercise, we will practice using Osnaps while learning more about Viewports and Construction planes.

### Exercise 23—Viewports and construction planes

- ▶ Open the **Chair.3dm** model.

### 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.

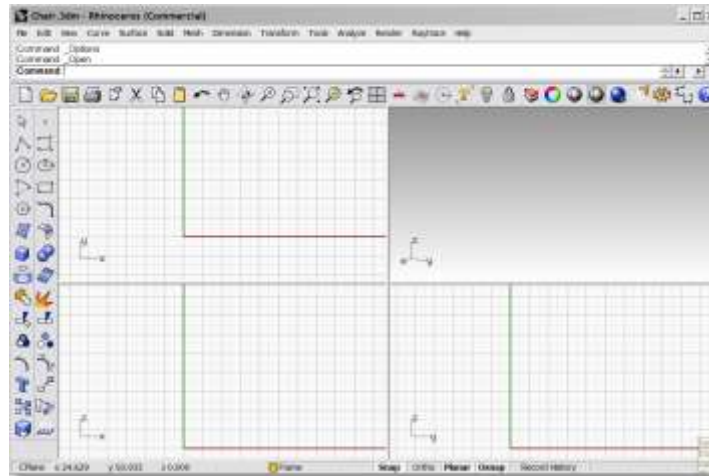
Each construction plane has its own axes, a grid, and an orientation relative to the world coordinate system.

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 icon in the left hand corner always shows world coordinates, which are different from the construction plane axes.

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.

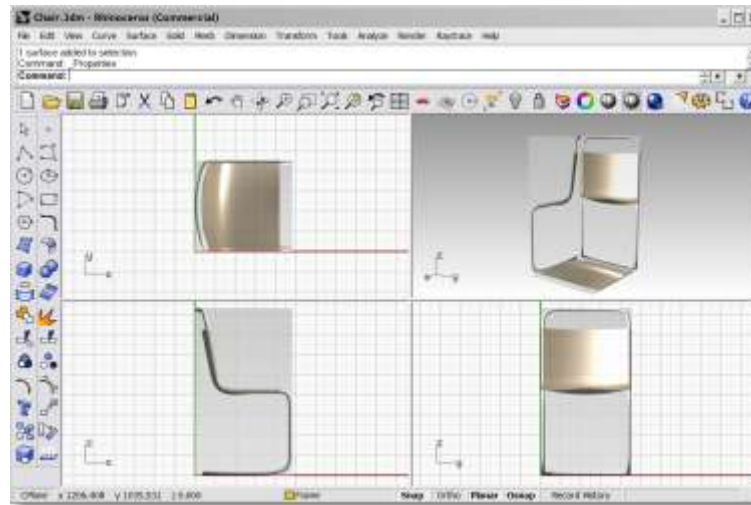
Next, we will turn on layers that illustrate how the default construction planes relate to each other. Each layer contains an image for a chair. Later in this exercise we will create our own chair.

- ▶ Turn the CP-Top, CP-Front, CP-Right layers on.

Each of the layers contain an image of a view of a tubular chair.

Notice that all the Chair images intersect at the origin of the model (0,0,0).

When you draw on any viewport the geometry will be located on the construction plane, unless you use Osnaps, Planar mode, or Elevator mode.



## Notes:

## Exercise 24—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**.

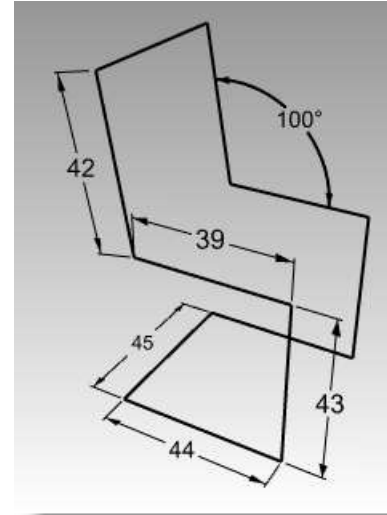
In the following exercise we will draw in different viewports and use elevator mode to move some points in 3-D space.

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

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

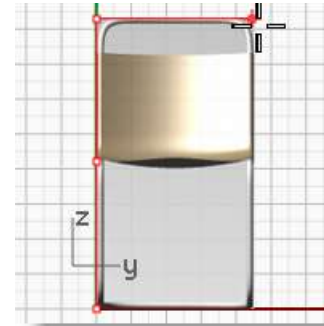
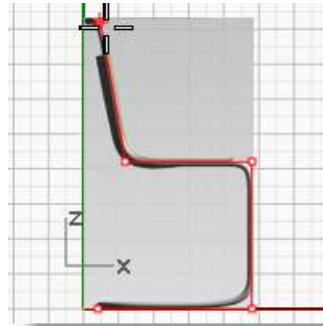
Pick a second point to specify the coordinate of the desired point. You can pick a point with the mouse, or type a single number to specify the height above the construction plane. Positive numbers are above the construction plane; negative numbers are below.

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



## Notes:

- 1 Toggle **Planar** off and **Snap** on. Toggle **Ortho** on as needed.
- 2 Turn on the **Point** osnap.
- 3 From the **Curve** menu, click **Polyline**, and then click **Polyline**.
- 4 Move your cursor into the **Front** viewport.
- 5 To **Start** click at the lower left side of the tubular chair image.
- 6 Use coordinate input and the image to draw the first part of the chair frame.
- 7 After you complete the profile lines, move your cursor to the **Right** viewport to draw a horizontal line.



**8** Move your cursor to the **Front** viewport until it snaps to the point at the lower end of the diagonal line. Don't click at this time.

**9** While the cursor is locked onto the point, hold down the **Ctrl** key, and pick.

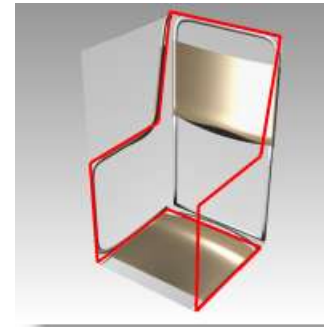
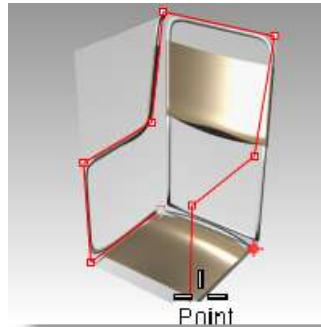
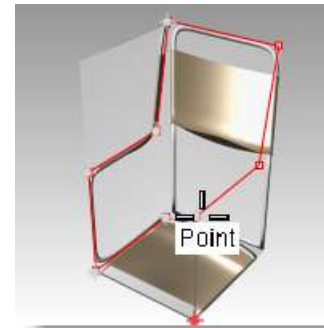
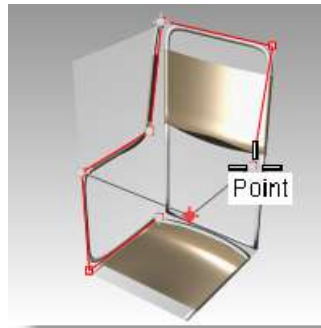
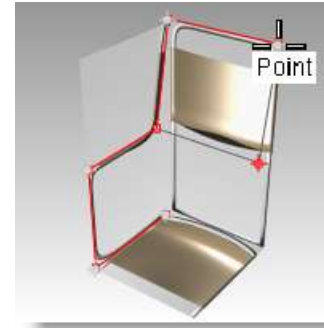
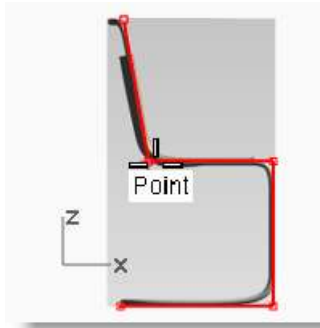
Holding the **Ctrl** key while clicking with the left mouse button activates elevator mode.

**10** Release the **Ctrl** key, move your cursor to the **Perspective** viewport and drag the point until it snaps to the last point, and pick.

**11** Continue drawing the rest of the chair frame using the same technique.

Snap to the next point in the Front viewport, activate elevator mode, then move your mouse to the Perspective viewport to position the point.

**12** For the last segment you can click **Close** on the command line or snap to the start point and click.

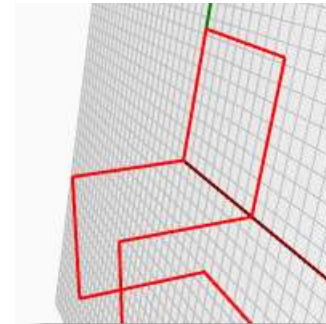
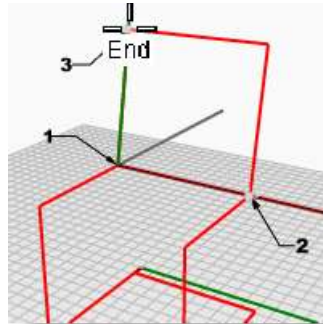
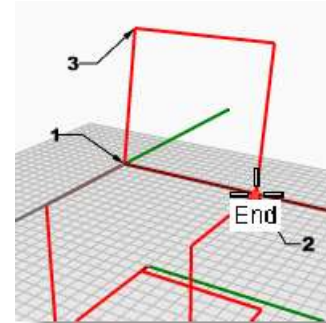
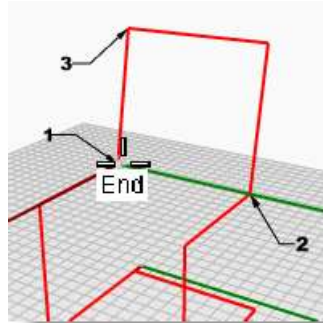


**Notes:**

### To make a custom construction plane:

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

- 1 On the **Status Bar**, click **Osnap**, and check **End**.
- 2 Change to the **Perspective** viewport and turn on the **Grid**, if necessary, by pressing **F7**.
- 3 From the **View** menu, click **Set CPlane**, and then click **3 Points**.
- 4 For the **CPlane origin**, pick vertex (1).
- 5 For X axis direction, pick vertex (2).
- 6 For the **CPlane orientation**, pick vertex (3).  
The construction plane is aligned with the back of the chair.



### To create a named construction plane

- 1 From the **View** menu, click **Set CPlane**, then click **Named CPlanes**.
- 2 In the **Named CPlanes** dialog, click **Save**.
- 3 In the **Save CPlanes** dialog, type **ChairBack**, then click **OK**.

You have a custom cplane that can be restored as needed. This custom cplane is saved to the file.

### Notes:



Set Cplane by 3 points



Named CPlanes

### To create a named view:

- 1 From the **View** menu, click **Set View**, click **Plan**.

The view changes. You are looking straight down on the new construction plane.

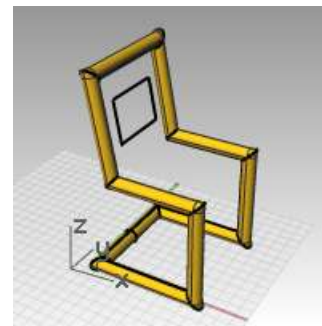
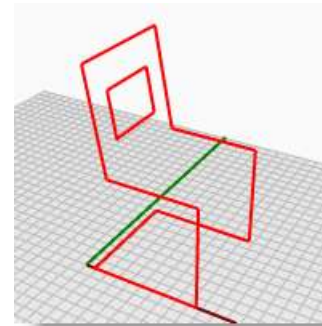
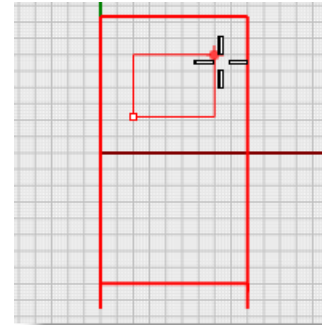
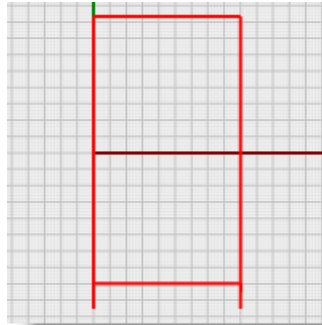
- 2 From the **View** menu, click **Set View**, click **Named Views**.

- 3 In the **Named Views** dialog, click **Save**.

- 4 In the **Save Views** dialog, type **ChairBack**, then click **OK**.

This creates a custom view that can be restored as needed.

- 5 Change back to the **Perspective** view.
- 6 From the **View** menu, click **Set View**, then click **Perspective**.
- 7 Draw some lines on the new construction plane.



### To make it solid:

- 1 Change to the **Chair Frame** layer.
- 2 Select the chair frame.
- 3 From the **Solid** menu, click **Pipe**.
- 4 For the **Start** and **End Radius**, type **3** and press **Enter**.  
The chair has a solid frame.
- 5 **Save** the model.

### Notes:



Plan View of CPlane



Edit Named Views



Perspective View

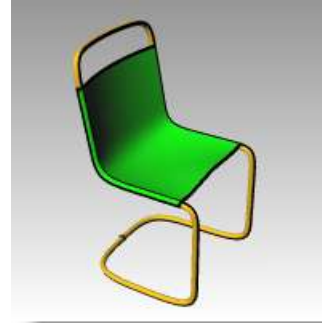


Pipe

---

**On your own:**

Try some variations and add some features.



**Notes:**

## 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 a curve object or surface edge.
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.

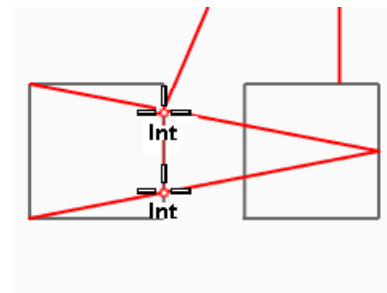
### To find the distance between two points:

- 1 Open the **Analyze.3dm** that you saved from an earlier exercise.  
If you didn't get a chance to save the model, open the Analyze-01.3dm instead.
- 2 From the **Analyze** menu, click **Distance**.
- 3 Pick the intersection where a diagonal line intersects a vertical line.
- 4 Pick the intersection where the other diagonal line intersects the same vertical line.  
Use object snaps.
- 5 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  
 $dx = 0$   $dy = 3.077$   $dz = 0$

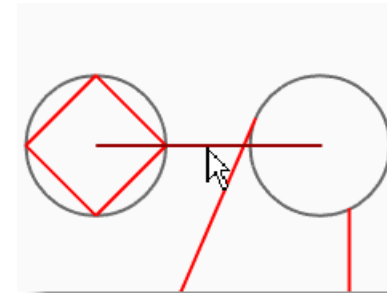
**Distance** = 3.077 millimeters



Distance

### To find the length of a line:

- 1 From the **Analyze** menu, click **Length**.
- 2 Select the line between the centers of the circles.  
**Length** = 8.000 millimeters



Length



### To measure the angle between two lines:

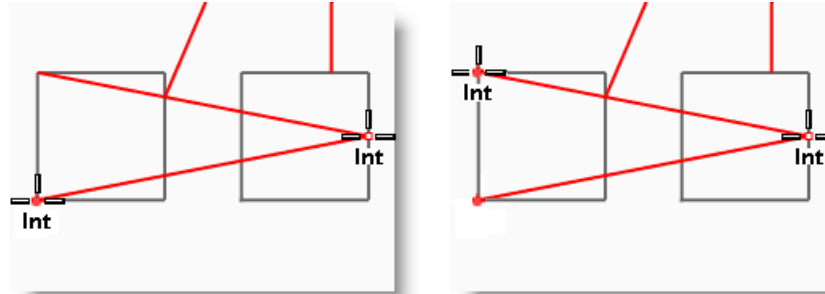
- 1 From the **Analyze** menu, click **Angle**.
- 2 Select a point that defines the vertex of an angle line.
- 3 Select a point that defines the end of an angle line.

Use object snaps as appropriate.

- 4 Select a point that defines the vertex of the second angle line.

- 5 Select a point that defines the end of the second angle line.

The angle displays on the command line in the following format: **Angle** = 21.7711



### Notes:



Angle

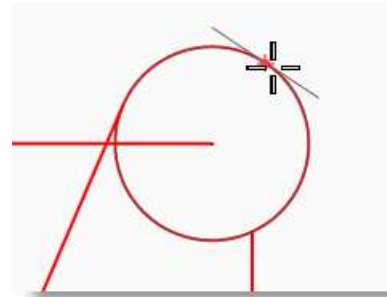
### To measure the radius of a circle:

- 1 From the **Analyze** menu, click **Radius**.
- 2 Select one of the circles.

This also measures the radius of a point on a curve.

The radius displays on the command line in the following format:

**Radius** = 2.5



Radius

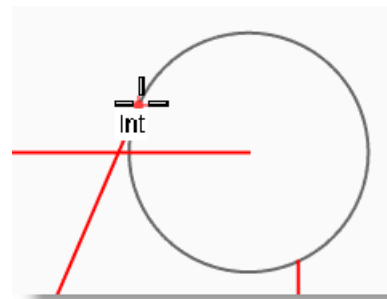
### To evaluate a point:

- 1 From the **Analyze** menu, click **Point**.
- 2 Snap to the end point of the tangent line.

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

**Point in world coordinates** = 8.203,11.488,0.000

**CPlane coordinates** = 8.203,11.488,0.000



Evaluate Point

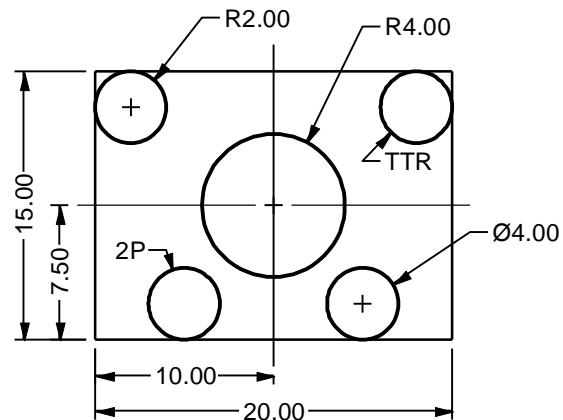
## Drawing Circles

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, Tangent, Radius	Draw a circle tangent to two curves with a given radius.
	Circle Tangent to 3 Curves	Draw a circle tangent to three curves.
	Circle AroundCurve	Draw a circle perpendicular to a curve at the chosen point.
	Deformable	Draws an approximation of a circle with a defined number of control points..
	Vertical	Draw a circle perpendicular to the construction plane.

### Exercise 25—Drawing circles

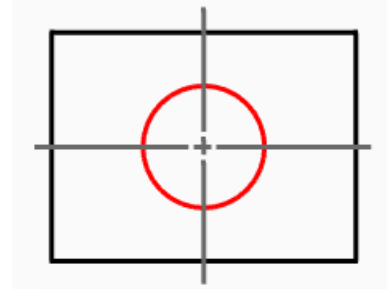
- ▶ **Open the Circles.3dm.**



### To draw a center, radius circle:

- 1 From the **Curve** menu, click **Circle**, and then click **Center, Radius**.
- 2 For the **Center of circle** snap to the intersection of the centerlines.
- 3 For the **Radius** type **4** and press **Enter**.

A circle is created around the centerline intersection.

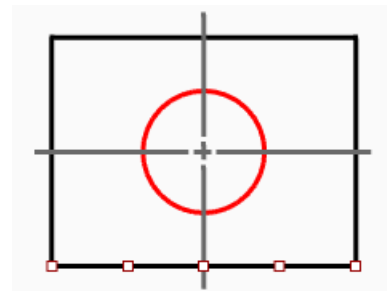


### To divide a curve by the number of segments:

For the next part of this exercise, we need to find some specific points to locate the two of the circles. We will use the **Divide** command to create the points.

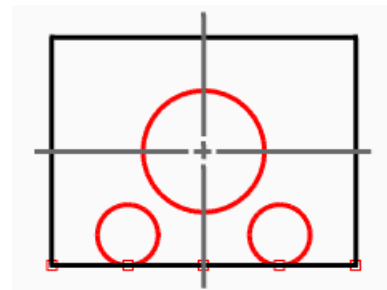
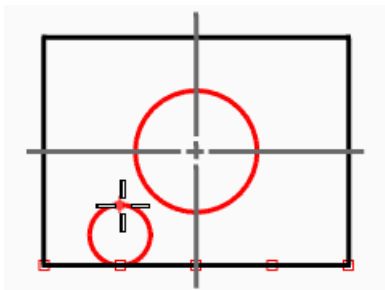
- 1 From the **Curve** menu, click **Point Object**, then click **Divide Curve by**, and then click **Number of segments**.
- 2 Select the **bottom line** as the **curve to divide**, press **Enter**.
- 3 For the **Number of segments** type **4** and press **Enter**.

A line is divided into four segments with a point at the end of each segment.



### To draw a diameter circle:

- 1 Turn **On** the **Point** osnap.
- 2 From the **Curve** menu, click **Circle**, and then click **2 Points**.
- 3 For the **Start of diameter** pick snap to the second point from the left end of the line you divided.
- 4 For the **End of diameter** type **4** and press **Enter**, then turn **Ortho** on and pick to the straight up.



A circle is created with the two points you picked as the diameter, and the diameter was constrained to a value of 4.

- 5 Repeat these steps for the second diameter circle.

### Notes:



Circle: Center, Radius



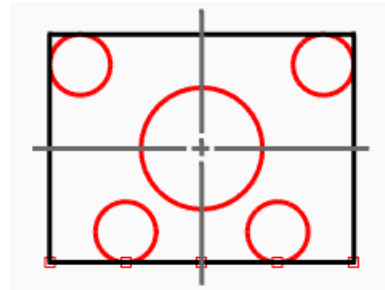
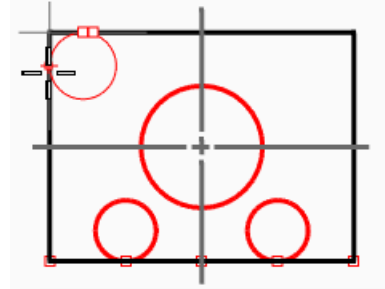
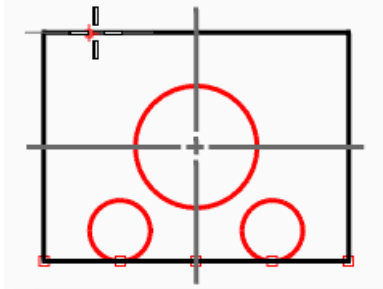
Circle: Diameter

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

- 1** From the **Curve** menu, click **Circle**, and then click **Tangent, Tangent, Radius**.
- 2** For the **First tangent curve**, select the top line.
- 3** For the **Second tangent curve**, select the left line.
- 4** For the **Radius**, type **2** and press **Enter**.

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

- 5** Repeat these steps for the second TTR circle.



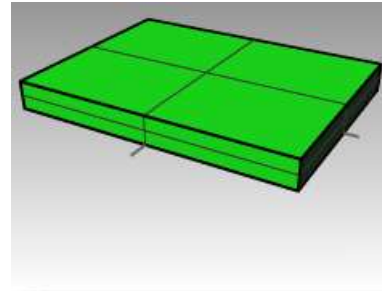
**Notes:**



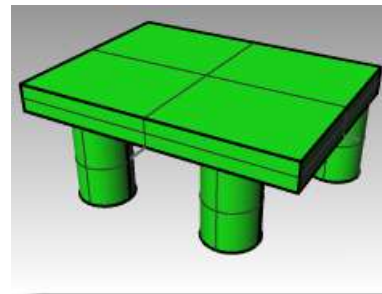
Circle: Tangent, Tangent, Radius

### To make it 3-D:

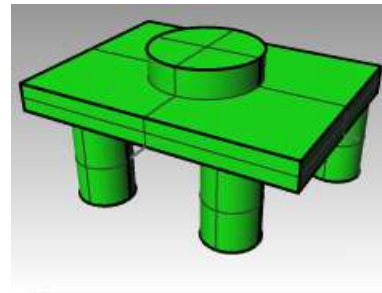
- 1 Select the lines that form the rectangle.
- 2 From the **Solid** menu, click **Extrude Planar Curve**, and then click **Straight**.
- 3 For the **Extrusion Distance**, type **2** and press **Enter**.  
The rectangle generates a box.



- 4 Select the four smaller circles.
- 5 From the **Solid** menu, click **Extrude Planar Curve**, and then click **Straight**.
- 6 For the **Extrusion Distance** type, **-6** and press **Enter**.  
The circles generate cylinders.



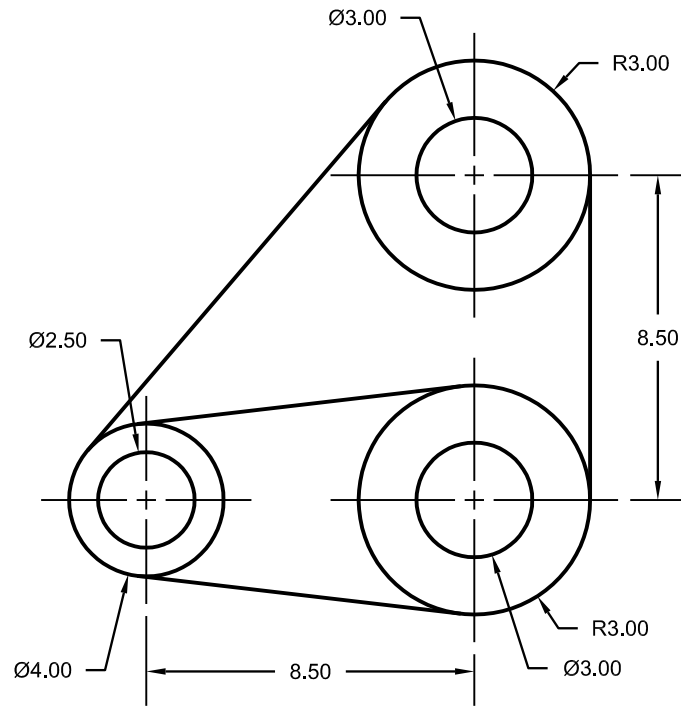
- 7 Select the large circle in the center.
- 8 From the **Solid** menu, click **Extrude Planar Curve**, and then click **Straight**.
- 9 For the **Extrusion Distance**, type **4** and press **Enter**.  
The circle generates a cylinder.



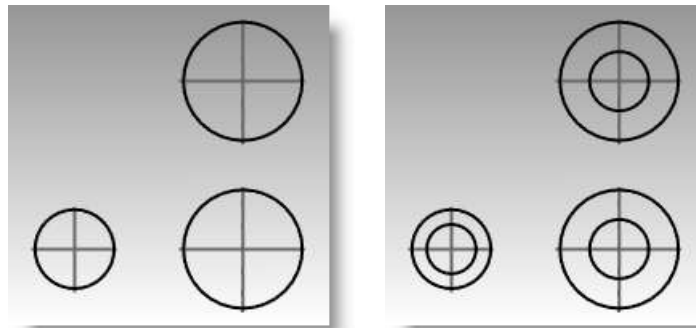
### Notes:

**Exercise 26—Using circle-related object snaps**

- 1 Open the **Link.3dm**.
- 2 Complete the model as shown.



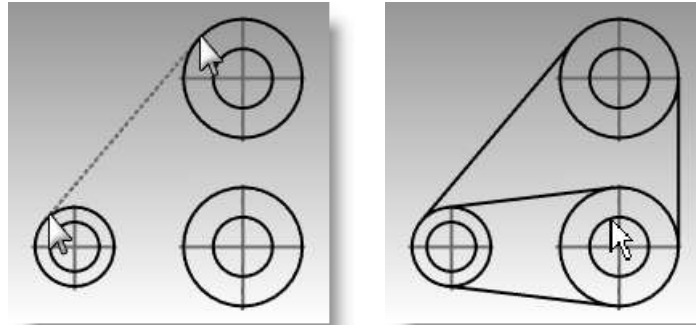
- 3 Draw the three large circles first.  
Snap to the centerline intersections to place the circles.
- 4 Draw the small holes next.  
Use object snaps to snap to the centers of the large circles



**Notes:**

### To draw the tangent lines:

- 1 From the **Curve** menu, click **Line**, and then click **Tangent to 2 Curves**.
- 2 For the **first tangent point** pick on the edge of one of the circles near where you want the tangent line to attach.
- 3 For the **second tangent point** pick on the edge of another circle. Rhino will find the tangent points for you.
- 4 Continue to use this command to complete the model.



### Notes:



Line Tangent to 2 Curves

## 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.

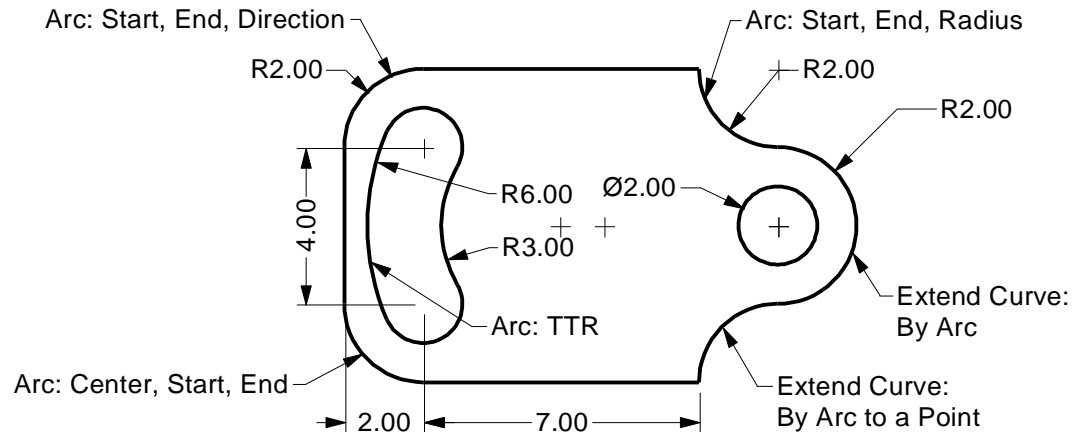
Button	Command	Description
	Arc	Draw an arc from the center, start, and angle.
	Arc : Start, End, Point on Arc	Draw an arc from three points.
	Arc: Start, End, Direction	Draw an arc from start point, end point, and the direction from the start point. The direction can be entered after the start point is entered or after the end point is entered.
	Arc: Tangent, Tangent, Radius	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.

## Arc options

Option	Description
Deformable	Creates an arc shaped NURBS curve
Extension	Extend a curve with an arc.

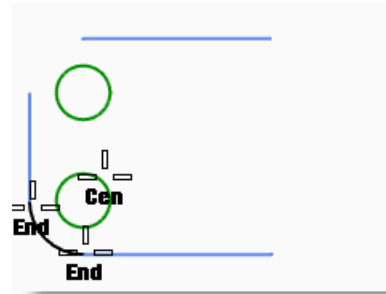
**Exercise 27—Practice drawing arcs (1)**

- ▶ **Open** the model **Arc1.3dm**.



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

- 1 From the **Curve** menu, click **Arc**, and then click **Center, Start, Angle**.
- 2 For the **Center of arc** snap to the center of the circle at the lower left.
- 3 For the **Start of arc** snap to the end of the line.
- 4 For the **End point or angle** snap to the end of the other line.



**Notes:**

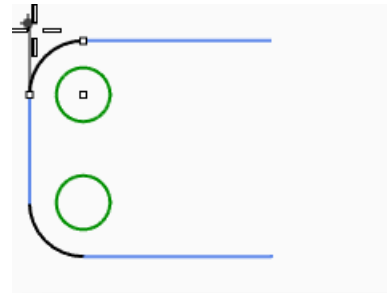


Arc: Center, Start, Angle

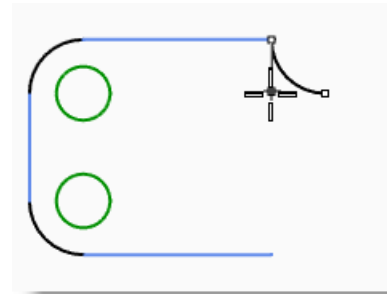


### To draw a start, end, direction arc:

- 1 From the **Curve** menu, click **Arc**, and then click **Start, End, Direction**.
- 2 For the **Start** point snap to the upper end of the vertical line.
- 3 For an **End** point snap to the adjacent end of the horizontal line at the top.
- 4 Toggle **Ortho** on and drag straight up for the tangent at the start point and pick.

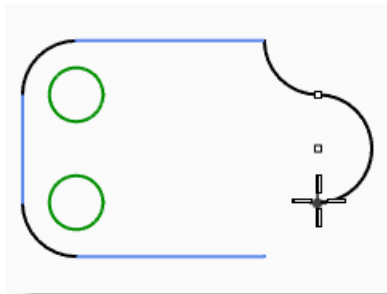


- 5 Make another **Direction Arc** at the upper right.
- 6 For the **Start** point snap to the right end of the horizontal line.
- 7 For an **End** point type **R2,-2** and press **Enter**.
- 8 Toggle **Ortho** on and drag straight down for the tangent at the start point and pick.



### To add more arc segments using extension option:

- 1 From the **Curve** menu, click **Arc**, and then click **Center, Start, Angle**.
- 2 Click **Extension** on the command line.
- 3 For the **Select near curve end** pick near the end of the arc you just made.
- 4 For the **End of arc**, type **4** and press **Enter**.
- 5 With **Ortho** on pick a point below the first point.  
The arc will be tangent to the curve you chose.



### Notes:



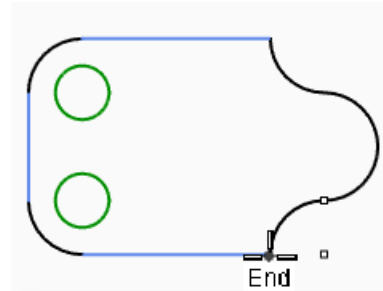
Arc: Start, End, Direction at Start



Arc: Center, Start, Angle

**To add more arc segments using the extension option:**

- 1 From the **Curve** menu, click **Arc**, and then click **Center, Start, Angle**.
- 2 Click **Extension** on the command line.
- 3 For the **Select near curve end** pick near the end of the arc you just made.
- 4 For the **End of arc**, snap to the end of the horizontal line.



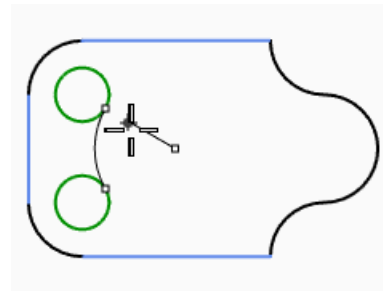
**Notes:**



Extend: By Arc to Point

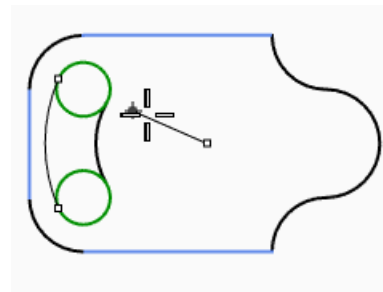
**To draw a tangent, tangent, radius arc:**

- 1 From the **Curve** menu, click **Arc**, and then click **Tangent, Tangent, Radius**.
- 2 For the **First tangent curve** pick the lower right side of the top circle.
- 3 For the **Radius** type **3** and press **Enter**.
- 4 For the **Second tangent curve** pick the upper right side of the lower circle.
- 5 Move your cursor and pick when the correct arc is displayed.



Arc Tangent, Tangent, Radius

- 6 From the **Curve** menu, click **Arc**, and then click **Tangent, Tangent, Radius**.
- 7 For the **First tangent curve** pick the upper left side of the top circle.
- 8 For the **Radius** type **6** and press **Enter**.
- 9 For the **Second tangent curve** pick the lower left side of the lower circle.
- 10 Move your cursor and pick when the correct arc is displayed.



### To make it solid:

First you will have to trim the two small circles, then you can extrude the remaining curves.

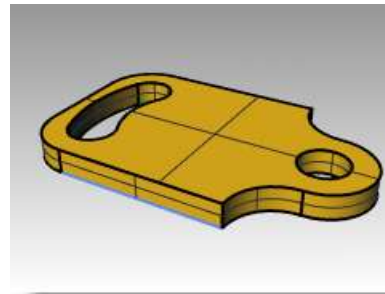
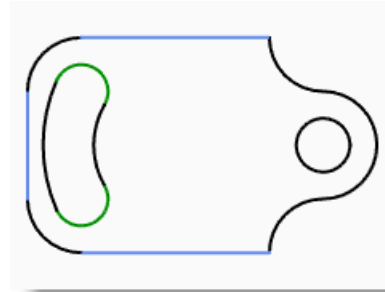
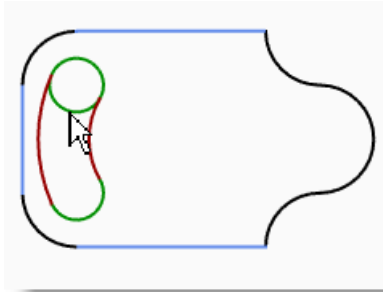
- 1 Select the two arcs you just completed.
- 2 From the **Edit** menu, click **Trim**.
- 3 For the objects to trim pick on the inner edge of each small circle.

You will be left with a slot.

- 4 Use the **Circle** command to draw the circle concentric to the arc on the right.

- 5 Select all the curves.
- 6 From the **Solid** menu, click **Extrude Planar Curve**, and then click **Straight**.
- 7 For the **Extrusion distance** type **1** and press **Enter**.

The curves have been extruded and capped.

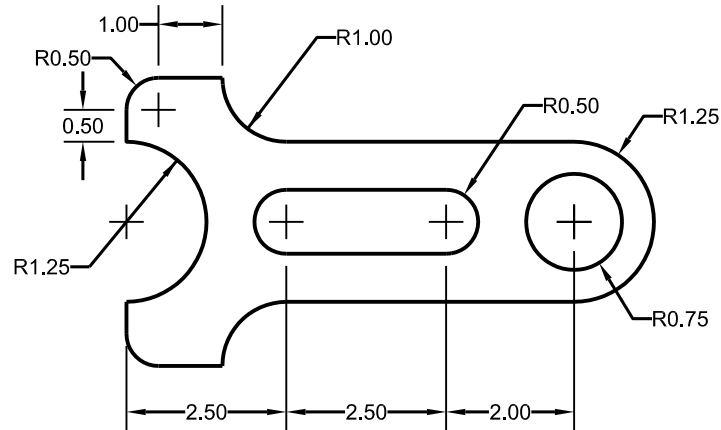


### Notes:

### Exercise 28—Practice drawing arcs (2)

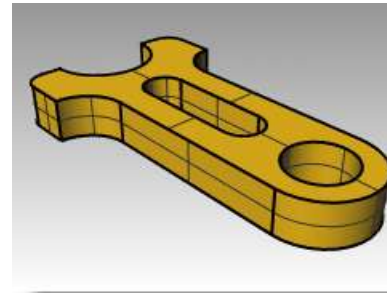
Draw construction centerlines first: use their intersection point to draw your arcs and circles.

- 1 Start a new model, use the **Small Objects - Inches.3dm** template. Save as **Arc2**.
- 2 Create this model using object snaps and the **Line**, **Circle**, and **Arc** commands.



#### To make it solid:

- 1 Select the curves.
- 2 From the **Solid** menu, click **Extrude Planar Curve**, and then click **Straight**.
- 3 For the **Extrusion distance** type **1** and press **Enter**.  
The curves have been extruded and capped.







#### Notes:




## Drawing Ellipses and Polygons

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

### 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 From Foci	Draw the ellipse from the focus points
	AroundCurve	Draw an ellipse whose axis is perpendicular to a curve.






### 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

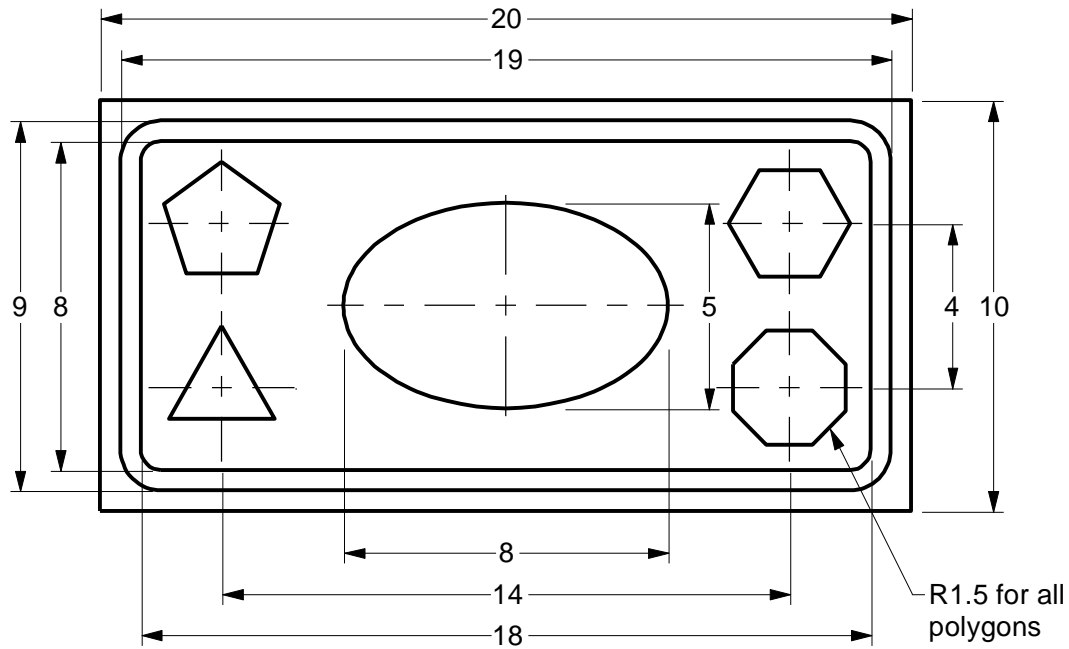
Option	Description
NumSides	Specify the number of sides for the polygon.
Circumscribed	Draw a polygon that is circumscribed about a radius. The default is to draw a polygon that is inscribed in a specified radius.

### 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).

### Exercise 29—Practice drawing ellipses and polygons

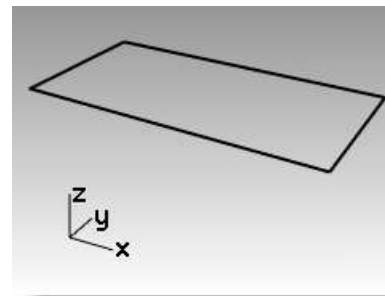
- ▶ Start a new model, use the **Small Objects - Millimeters.3dm** template. Save as **Toy**.



### Notes:

#### To draw a rectangle from diagonal corners:

- 1 From the **Curve** menu, click **Rectangle**, and then click **Corner to Corner**.
- 2 For the **First Corner**, type **-10,-5** and press **Enter**.
- 3 For the **Length**, type **20** and press **Enter**.
- 4 For the **Width**, type **10** and press **Enter**.



Rectangle

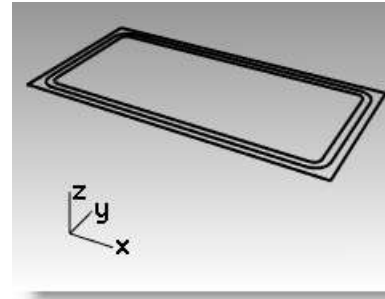
### Draw rectangles from a center point with a length and width, and rounded corners:

- 1 From the **Curve** menu, click **Rectangle**, and then click **Center, Corner**.
- 2 Type **R**, and press **Enter** to give the rectangle rounded corners.
- 3 For the **Center of rectangle**, type **0** and press **Enter**.
- 4 For the **Length**, type **19** and press **Enter**.
- 5 For the **Width**, type **9** and press **Enter**.
- 6 For the **Radius**, type **1**, and press **Enter**.

If you are creating a rounded rectangle, you can choose to select a point in the corner to set its curvature instead of typing a radius value.

To change the type of corner, click **Corner** on the command line 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.



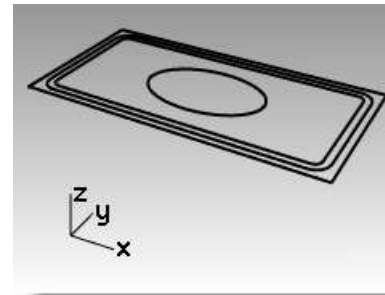
### Notes:



Rectangle: Center, Corner

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

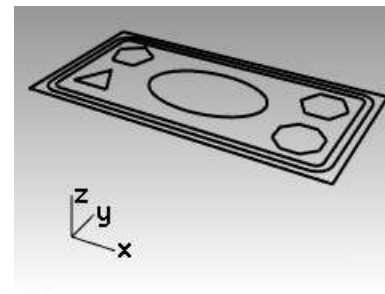
- 1 From the **Curve** menu, click **Ellipse**, and then click **From Center**.
- 2 For the **Ellipse center**, type **0** and press **Enter**.
- 3 For the **End of first axis**, type **4**, and press **Enter**.
- 4 Toggle **Ortho** on and pick to the right.
- 5 For the **End of second axis**, type **2.5**, and press **Enter**.
- 6 Pick a point.



Ellipse

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

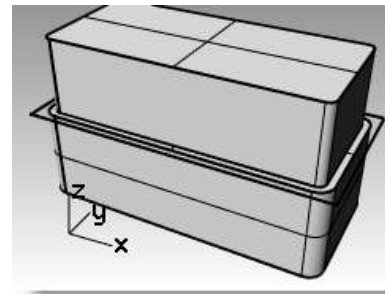
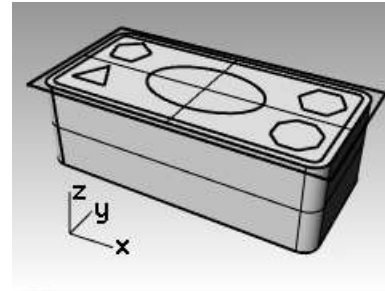
- 1 From the **Curve** menu, click **Polygon**, and then click **Center, Radius**.
- 2 Click **NumSides** on the command line, type **3** and press enter to change the number of sides for the polygon.
- 3 For the **Center**, type **-7,-2** and press enter to locate the center point for the polygon.
- 4 For the **Corner**, type **1.5**, and press **Enter**.
- 5 Pick a point to orient the polygon.
- 6 Continue drawing the rest of the polygons. Use the same radius for each one.



Polygon

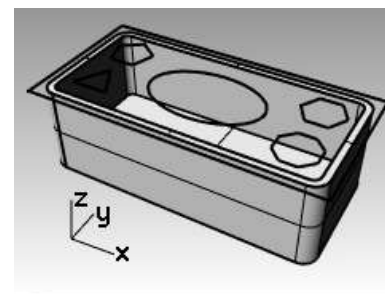
### To make the rounded rectangles solid:

- 1 Select the larger rounded rectangle.
- 2 From the **Solid** menu, click **Extrude Planar Curve**, and then click **Straight**.
- 3 Drag it down to set the thickness and click.
- 4 Select the smaller rounded rectangle.
- 5 From the **Solid** menu, click **Extrude Planar Curve**, and then click **Straight**.
- 6 Click **Bothsides** on the command line.
- 7 Drag it down until it is slightly above the previous solid and click.  
The smaller rounded rectangle is protruding out of the larger one.



### To make the rounded rectangle hollow:

- 1 Select the solid outer rounded rectangle.
- 2 From the **Solid** menu, click **Difference**.
- 3 Select the inner rounded rectangle, and press **Enter**.



### Notes:



Extrude closed planar curve

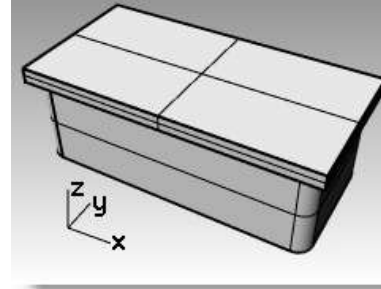


Boolean Difference



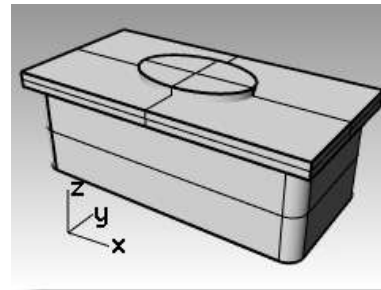
### To make the rectangle solid:

- 1 Select the rectangle.
- 2 From the **Solid** menu, click **Extrude Planar Curve**, and then click **Straight**.
- 3 Set the **Bothsides** option to **No**.
- 4 Drag it up to set the thickness and click.



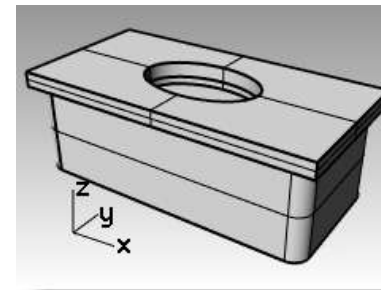
### To make the ellipse solid:

- 1 Select the ellipse.
- 2 From the **Solid** menu, click **Extrude Planar Curve**, and then click **Straight**.
- 3 Set the **Bothsides** option to **Yes**.
- 4 Pick to set the thickness.



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

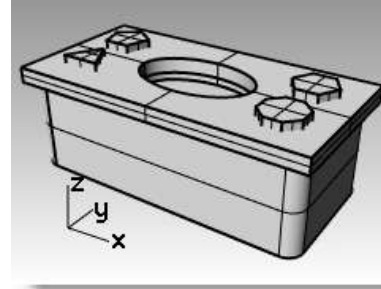
- 1 Select the solid rectangle.
- 2 From the **Solid** menu, click **Difference**.
- 3 Select the solid ellipse, and press **Enter**.



**Notes:**

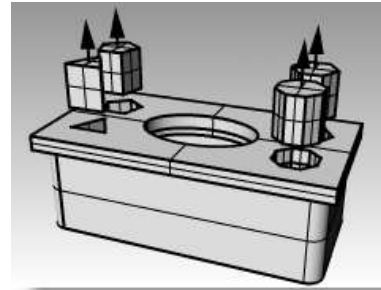
### To extrude the polygons:

- 1 Select the polygons.
- 2 From the **Solid** menu, click **Extrude Planar Curve**, and then click **Straight**.
- 3 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 On the command line click **DeleteInput**.
- 4 For the **second set of surfaces or polysurfaces**, select the solid polygons and press **Enter**.  
Holes will be cut, but the objects will remain.



### Notes:

## 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	Draws 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.
Autoclose	Move the cursor close to the start point of the curve, and pick. The curve will close automatically. Press ALT to suspend autoclose.
Sharp	If Yes, when you make a closed curve, it will have a kink at the start/end point instead of making a smooth (periodic) closure.
Degree	Allows you to set the degree of the curve.

### Exercise 30—Practice drawing curves (1)

#### 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 or a background image made from a sketch or a photograph. In this exercise the guidelines have been created for one of the exercises and an image has been included for the other exercise.

#### 2 In the **Osnap** toolbar, check **End** and **Near**, clear all others.

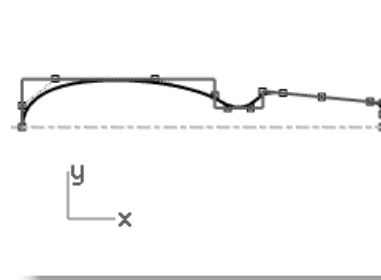
If you click **End** with your right mouse, it will clear all the others.

#### 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 For the **Start of Curve**, snap to the end point of the polyline guideline.
- 3 For the **Next point**, snap on the polyline guideline using the **Near** object snap.
- 4 Continue to snap to the guideline until you get to the end.
- 5 Press **Enter**.

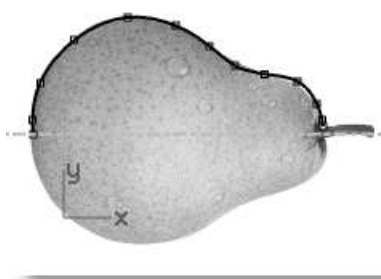
A free-form curve is drawn. The control points, while on the guideline, were not on the curve itself except at the two ends.



### To draw a curve interpolate through points:

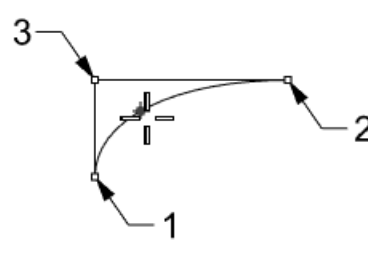
- 1 Change to the **Interpolated Curve** layer.
- 2 From the **Curve** menu, click **Free-form**, and then click **Interpolate Points**.
- 3 For the **Start of Curve**, snap to the intersection of the centerline and the image using the **Near** object snap.
- 4 For the **Next point**, continue to pick points along the edge of the image until you get to the other end, then use the **Near** object snap to snap to the intersection of the centerline and the image.
- 5 Press **Enter**.

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



### To draw a conic curve:

- 1 Change to the **Conic** layer.
- 2 From the **Curve** menu, click **Conic**.
- 3 Snap to the point (1) at the lower left.
- 4 Snap to the point (2) above and to the right from the previous point.
- 5 Snap to the point (3) in between the previous points.
- 6 Pick a point for the desired curvature.



### Notes:



Control Point Curve

*Control points control the curvature of the curve, but the points are usually not on the curve.*



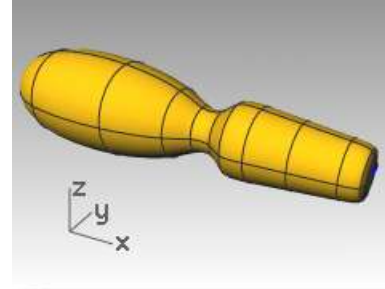
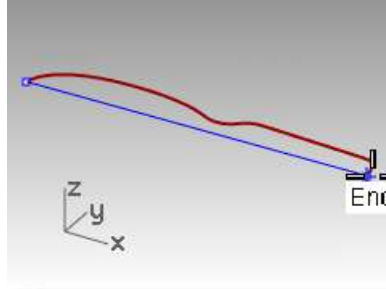
Curve: Interpolate Points



Conic

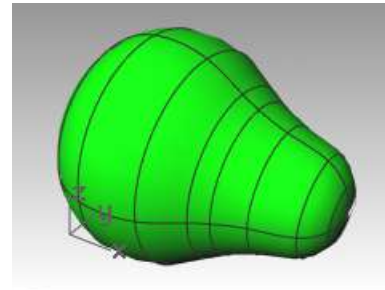
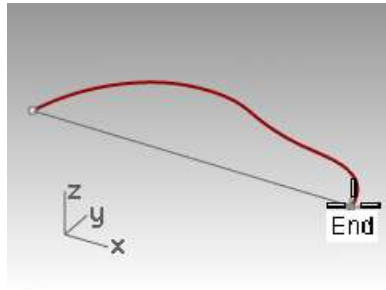
**To make the control point curve into a surface:**

- 1 Select the control point curve.
- 2 From the **Surface** menu, click **Revolve**.
- 3 For the **Start of revolve axis**, snap either end of the curve.
- 4 For the **End of revolve axis**, snap to the other end of the curve.
- 5 For the **Start angle**, click **FullCircle**.



**To make the interpolated curve into a surface:**

- 1 Select the interpolated curve.
- 2 From the **Surface** menu, click **Revolve**.
- 3 For the **Start of revolve axis**, snap either end of the curve.
- 4 For the **End of revolve axis**, snap to the other end of the curve.
- 5 For the **Start angle**, click **FullCircle**.





**Notes:**



Revolve

## 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.

### Exercise 31—Practice drawing curves (2)

#### 1 Open the model **Helix-Spiral.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.

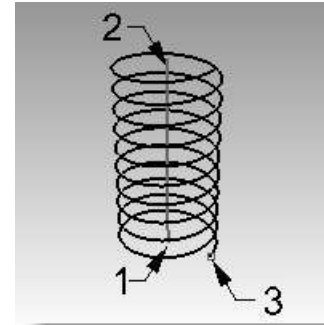
#### 2 In the **Osnap** toolbar, check **End** and **Point**, clear all others.

If you click **End** with your right mouse, it will clear all the others.

#### 3 Toggle **Ortho** and **Snap** off.

### To draw a helix

- 1 Change to the **Helix** layer.
- 2 From the **Curve** menu, click **Helix**.
- 3 For the **Start of axis**, snap to the end of the vertical line **(1)** in the **Perspective** viewport.
- 4 For the **End of axis**, snap to the end of the vertical line **(2)** in the **Perspective** viewport.
- 5 Click **Mode** on the command line until it is set to **Mode=Turns**.
- 6 Click **Turns** on the command line.
- 7 For the **Number of turns**, type **10** and press **Enter**.
- 8 Pick the point **(3)** to the right of the axis line.  
A helix with 10 turns and a radius of 20 is created.

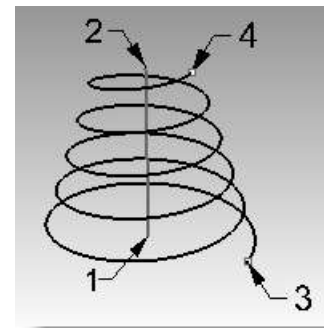


### Notes:



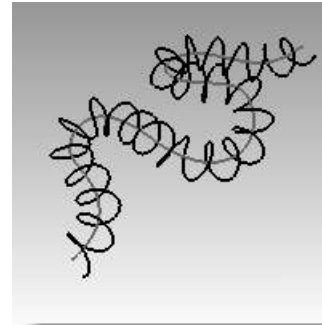
### To draw a spiral:

- 1 Change to the **Spiral** layer.
- 2 From the **Curve** menu, click **Spiral**.
- 3 For the **Start of axis**, snap to the end of the other vertical line **(1)** in the **Perspective** viewport.
- 4 For the **End of axis**, snap to the other end of the same line **(2)**.
- 5 Click **Mode** on the command line until it is set to **Mode=Pitch**.
- 6 Click **Pitch** on the command line.
- 7 For the **Pitch**, type **15** and press **Enter**.
- 8 Click **ReverseTwist** on the command line until it is set to **Reverse Twist=Yes**.
- 9 Snap the point **(3)** for the radius of the base of the spiral.
- 10 Snap to the other point **(4)** for the ending radius.  
A spiral is created with a reverse twist and a pitch of 15.



**To draw a helix around a curve:**

- 1** Change to the **HelixAlongCurve** layer.
  - 2** From the **Curve** menu, click **Helix**.
  - 3** Click **AroundCurve**.
  - 4** Select the free-form curve.
  - 5** Click **Mode=Turns**.
  - 6** Click **Turns**
  - 7** Type **25** and press **Enter**.
  - 8** Click **ReverseTwist=No**.
  - 9** For the **Radius** type **5** and press **Enter**.
  - 10** Pick a point for the **Start point**.
- A helix is created around the curve.



**Notes:**



Helix



### Exercise 32—Drawing free-form curves

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

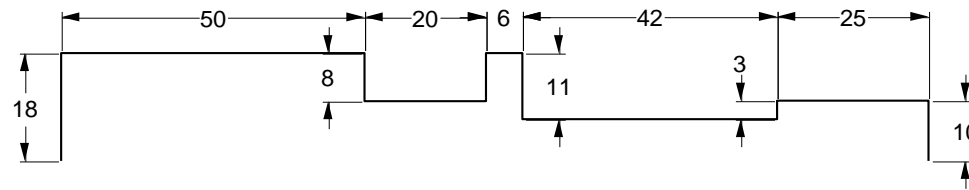
- 1 Start a new model, use the **Small Objects - Millimeters.3dm** template. Save as **Screwdriver**.
- 2 Create **Construction** and **Curve** layers.  
Make them different colors.



### Notes:

#### To create the construction lines:

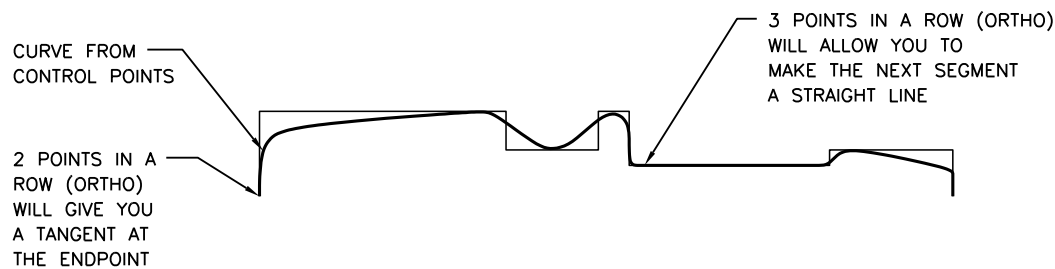
- 1 Change to the **Construction** layer.
- 2 Draw a **Polyline**, in the **Top** viewport, using these dimensions for the guideline.



A good starting point for the Polyline would be **-70,0**.

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

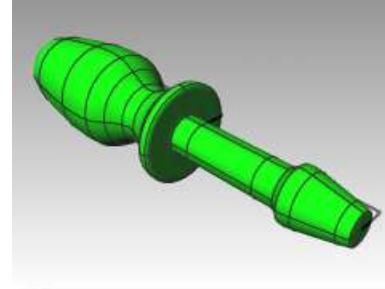
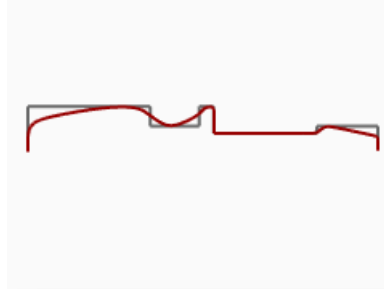
- 1 Change to the **Curve** layer.
- 2 Use the **Curve** command to draw the shape for the toy screwdriver.
- 3 **Save** your model.



### To make it solid:

- 1 Toggle **Snap** and **Ortho** on.
- 2 Select the curve.
- 3 From the **Surface** menu and click **Revolve**.
- 4 For the **Start of revolve axis**, snap to the end of the curve.
- 5 For the **End of revolve axis**, snap to the other end of the curve.
- 6 For the **Start angle**, click **FullCircle**.

Your model is now a three-dimensional surface.



### Notes:

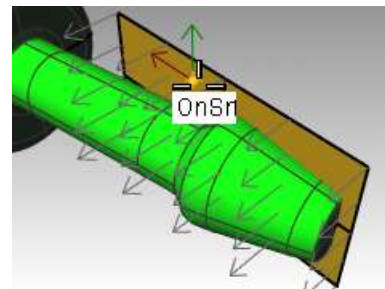
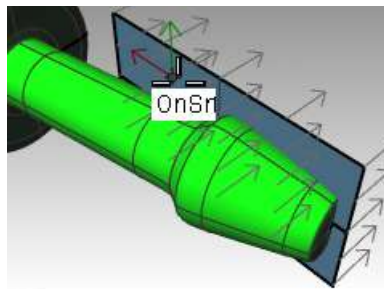
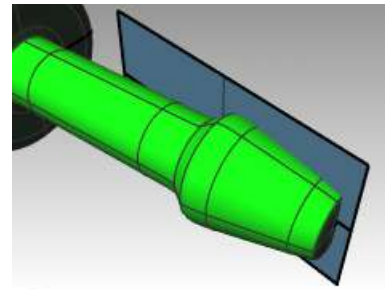
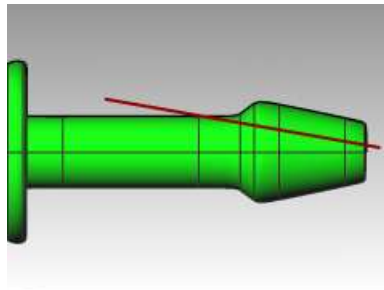


Revolve

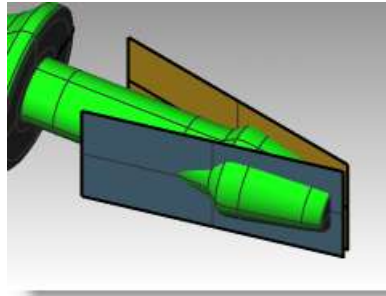
### Add details on your own:

Since we haven't discussed many of the commands that you need to complete this model, use **Command Help** for assistance. Following is a list of procedures for one method to finish the model.

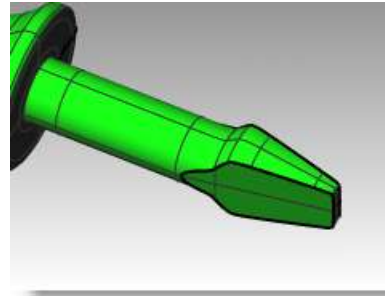
- 1 In the Top view draw a **Single Line** that will be used to slice the flat part from the screwdriver blade.
- 2 **Extrude** the line to make a surface that intersects the blade end.
- 3 Use the **Dir** command to check the normal direction of the surface.
- 4 Make sure the direction arrows are pointing toward the blade. Use the **Flip** option if needed.



5 **Mirror** the surface around the origin to get a surface for cutting both sides of the blade.

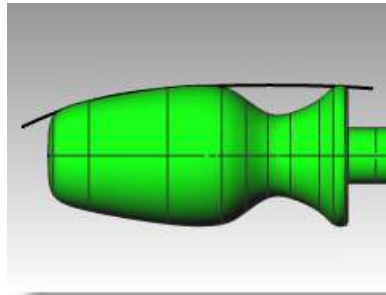


6 Use the **BooleanDifference** command to remove the surfaces from the blade.

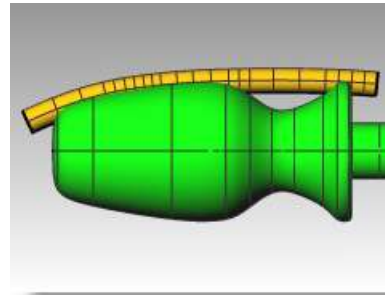


7 Draw a **Curve** along the top of the handle to be used for cutting grooves.

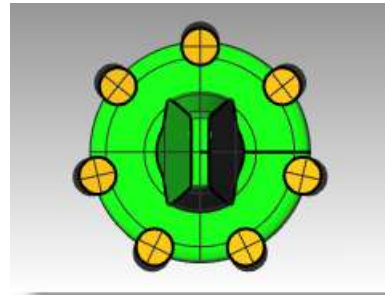
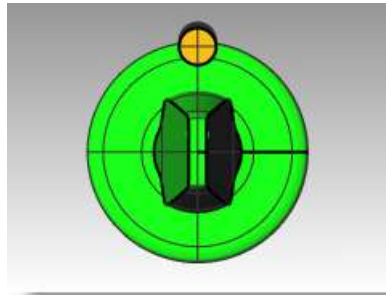
If you do this in the Front viewport, you will eliminate any problems that might arise from having one of the slots along the seam of the surface.



8 Use the **Pipe** command to make a surface on the curve.



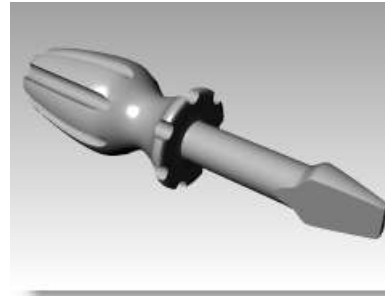
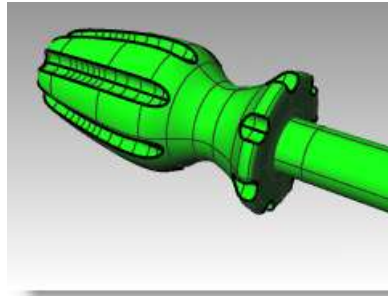
9 Use the **ArrayPolar** command to make copies of the pipe around the handle.



**Notes:**

**10** Use the **BooleanDifference** command to remove the pipes from the handle.

**11** Try using the **Render** command to make a rendering of the finished screwdriver.



**Notes:**

# 5

## Editing Objects

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

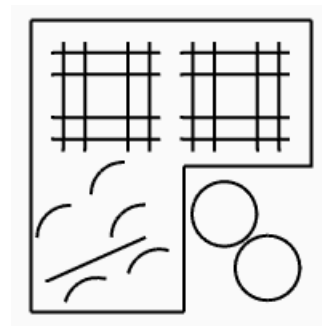
### Fillet

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 33—Fillet

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



### Notes:

*Rules to follow when filleting curves:*

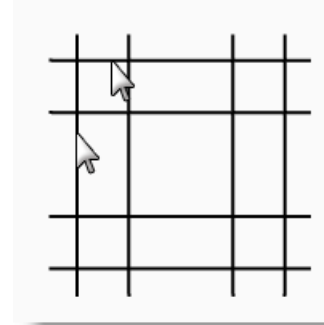
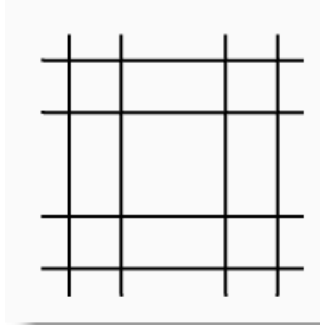
*The curves must be coplanar.*

*The created fillet is determined by picking on the curve portion to keep.*

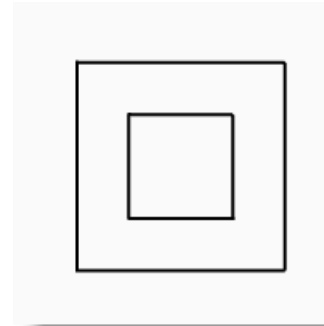
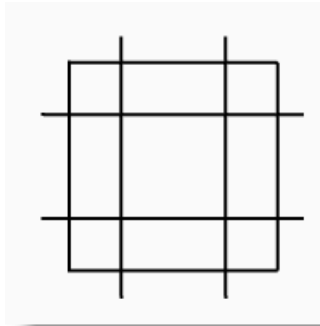
*The radius cannot be so large that it runs past the end of the curve.*

### To fillet lines at a zero radius:

- 1 From the **Curve** menu, click **Fillet Curves**.
- 2 Click **Radius** to change the radius.
- 3 Type **0**, and press **Enter**.  
This Radius option remains the default until you change it.
- 4 Select an outer vertical line.
- 5 Select an adjacent horizontal line.  
The ends of the lines are trimmed to a corner.



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



### To join the filleted objects:

- 1 Select lines you just filleted.
- 2 From the **Edit** menu, click **Join**.  
The objects are joined together. Curves join only if they touch.

### Notes:



Fillet Curves

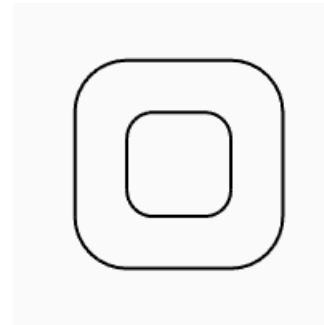
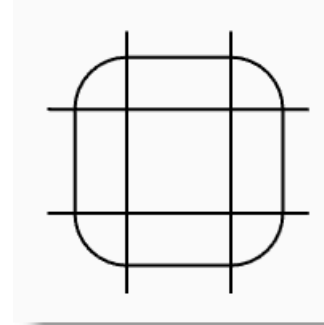
*Remember to pick on the part of the line that you want to keep.*



Join

### To fillet lines using an arc:

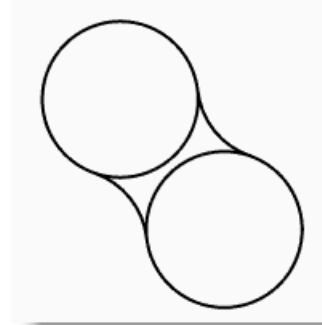
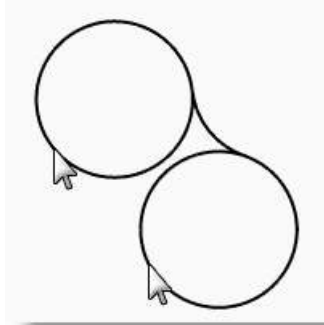
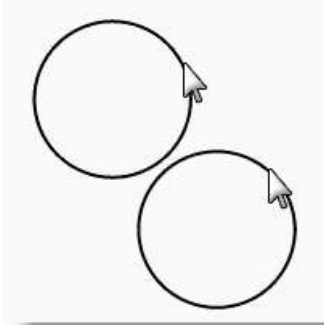
- 1** From the **Curve** menu, click **Fillet Curves**.
- 2** Type **2** and press **Enter** to change the radius.  
This is another method to change the radius.
- 3** Click **Join** to change the Join option to Yes.  
This option joins the curves as they are filleted.
- 4** Select an outer vertical line.
- 5** Select an adjacent horizontal line.  
The ends of the lines are trimmed to a radius.
- 6** Press **Enter** to repeat the command.
- 7** Fillet the other corners as shown.
- 8** Press **Enter** to repeat the command.
- 9** Type **1** and press **Enter**.  
This radius will be used for the smaller object.
- 10** Select an inner vertical line.
- 11** Select an adjacent horizontal line.
- 12** **Fillet** the other corners as shown.



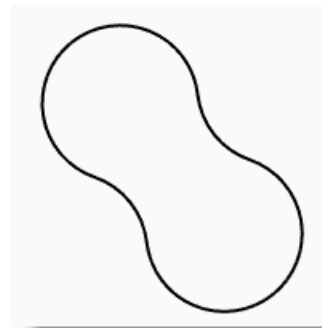
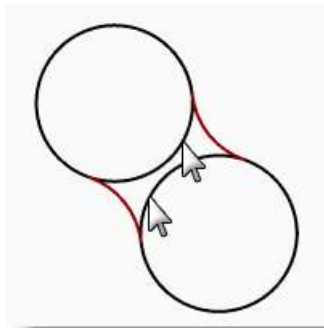
### Notes:

**To fillet circles:**

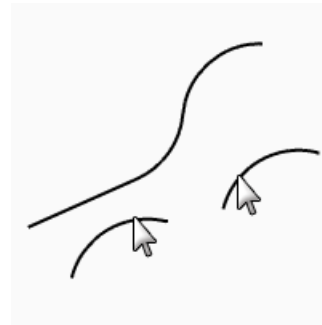
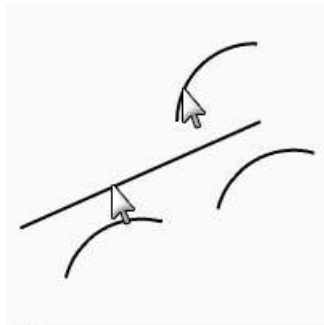
- 1 From the **Curve** menu, click **Fillet Curves**.
- 2 Type **3** and press **Enter**.
- 3 Select the right edge of a circle.
- 4 Select the right edge of the other circle.



- 5 Repeat the command for the left side of the circles.
- 6 Select the two fillets you just created.
- 7 From the **Edit** menu, click **Trim**.
- 8 For the objects to trim pick on the inner edge of each circle.

**To fillet and join arcs and lines:**

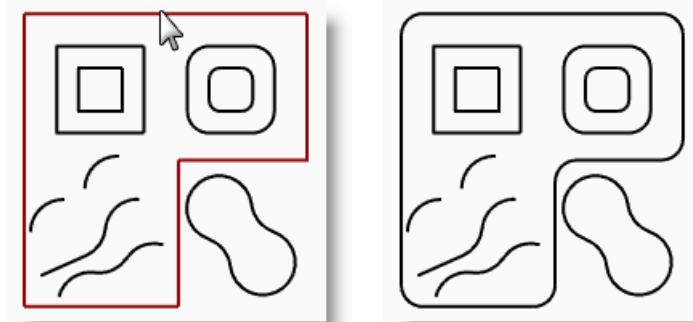
- 1 From the **Curve** menu, click **Fillet Curves**. Set Join=Yes and Trim=Yes.
- 2 Select the line in the lower left part of the viewport.  
Make sure you pick on the left half of the line.
- 3 Select the adjacent arc right above the selected line.
- 4 Repeat this procedure for the two arcs below the line and the arc you just filleted.





### To fillet the corners of a closed polyline:

- 1 Select the closed polyline.
- 2 From the **Curve** menu, click **Fillet Corners**.
- 3 For the **Fillet** radius, type **2** and press **Enter**.  
All corners are filleted at once.



### Notes:



Fillet corners

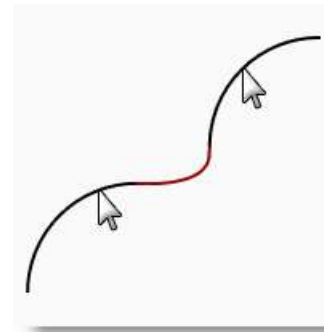
## Blend

Blending is another method to connect lines, arcs, or curves. There are two blend commands that work on curves, **Blend** (*default blend*) and **BlendCrv** (*adjustable blend*). Blend offers many additional options for how the curves connect.

Option	Description
Perpendicular	When Continuity=Tangent or Curvature, allows you to blend a curve to a surface edge perpendicular to the surface with continuity.
AtAngle	When Continuity=Tangent or Curvature, orients the direction of the blend curve other than perpendicular to the edge.
Continuity	Continuity describes the relationship between curves and surfaces. Types of continuity: Position (G0), Tangent (G1), and Curvature (G2).

### To blend two curves:

- 1 From the **Curve** menu, click **Blend Curves**.
- 2 Select the **Curves to blend** near the ends that you want to connect.  
The arcs are connected with a smooth curve. The default continuity is Curvature.
- 3 **Delete** the blend curve.



Blend Curves

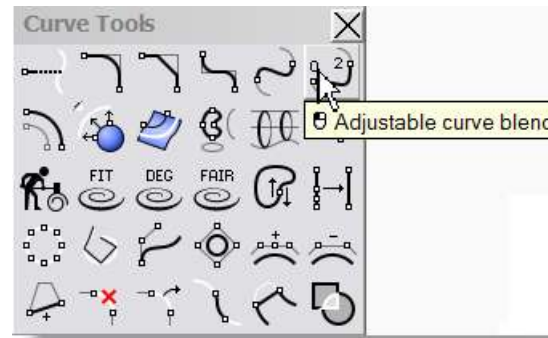
## Notes:



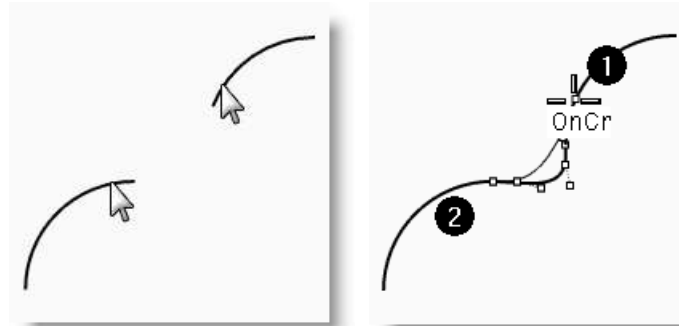
Adjustable curve blend

### To blend two curves with Adjustable curve blend:

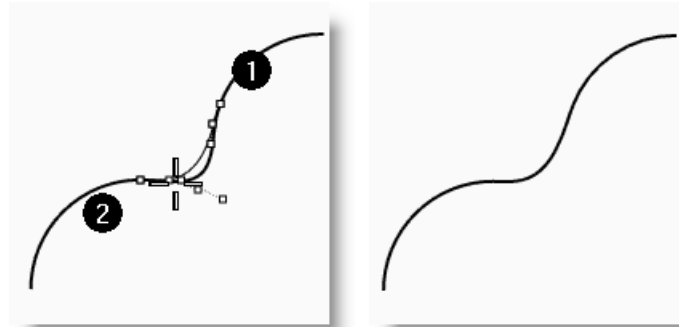
- 1 Since there is no menu pick for this command, type **BlendCrv** on the command line or select the **Adjustable curve blend** icon from the **Curve Tools** toolbar.



- 2 Select the **Curves to blend** near the ends that you want to connect.  
You will see a preview of the default blend with control points displayed.
- 3 For the **control point to adjust**, select the point **(1)** and drag it further up the curve and click.



- 4 For the next **control point to adjust**, select the point to the right of **(2)**, drag it closer to **(2)** and click.
- 5 After making your adjustments, press **Enter** to make the blend.



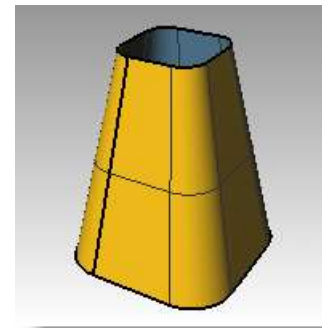
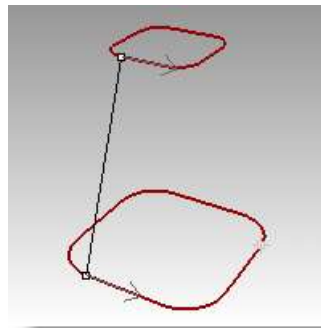
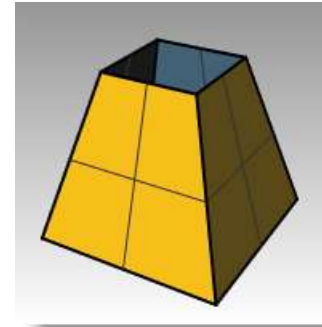
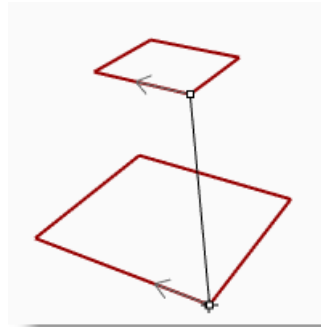
### To make a lofted surface with closed curves:

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

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

If the seams don't line up at corresponding points on the two curves, drag the seam point until it does.

- 5 Press **Enter**.
- 6 In the **Loft Options** dialog box, click **OK**.  
A surface is generated between the two closed polylines.
- 7 Repeat the procedure for the rounded squares.
- 8 In the **Loft Options** dialog box, click **OK**.
- 9 Save your model.



### Notes:

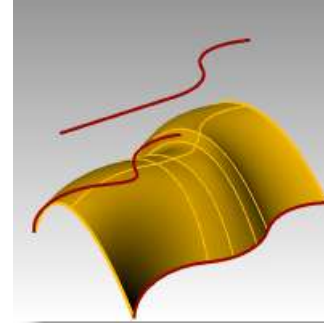
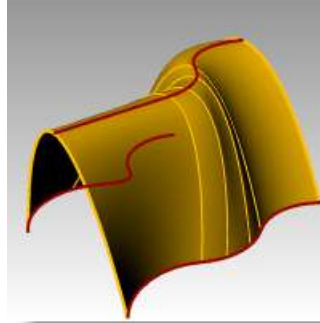
*The curves in this model are at two different elevations. You are going to connect the curves at different elevations with a surface.*



Loft

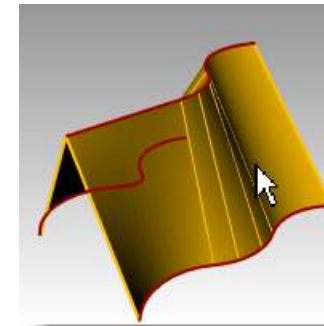
**To make a lofted surface with open curves:**

- 1** Repeat the **Loft** command for the three open curves.
- 2** In the **Loft Options** dialog box, change the **Style** to **Loose**, click **Preview**.

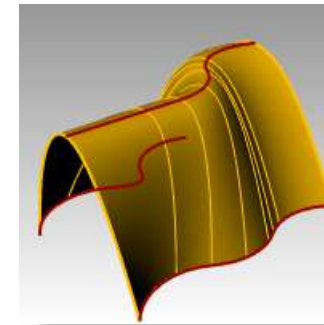
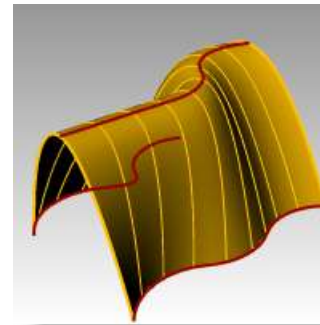


- 3** Change the **Style** to **Straight sections**, click **Preview**.

- 4** Change the **Style** to **Normal**.



- 5** For the **Cross-section curve options**, change to **Rebuild with**, change the number of **control points** to **12**, click **Preview**.



- 6** Change to **Refit within**, click **Preview**.

**Notes:**

## 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>

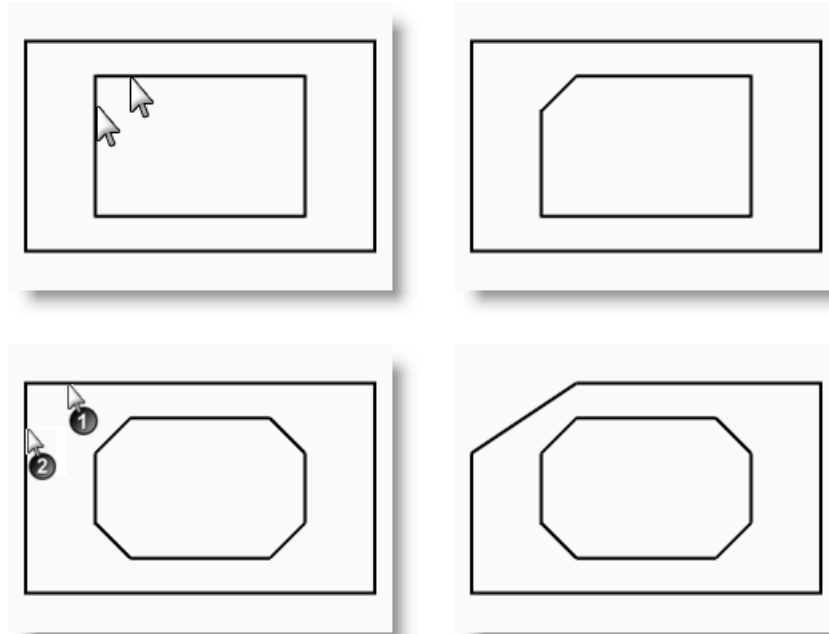
### Exercise 34—Chamfer

► **Open** the model **Chamfer.3dm**.

#### To chamfer lines:

- 1 From the **Curve** menu, click **Chamfer**.
- 2 For the **Distance**, type **1,1**, and press **Enter**, to set the distances.
- 3 Set **Join=Yes**.
- 4 Select one of the inner vertical lines.
- 5 Select an adjacent horizontal line.
- 6 Continue creating chamfers on all of the corners.
- 7 Press **Enter** to repeat the command.
- 8 For the **Distance**, type **3,2** and press **Enter**.
- 9 Select one of the outer horizontal lines.
- 10 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.

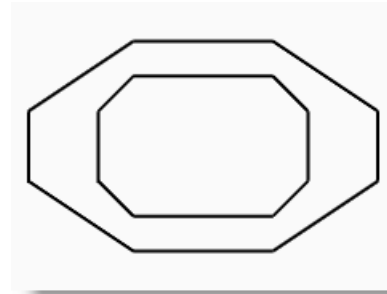


## Notes:



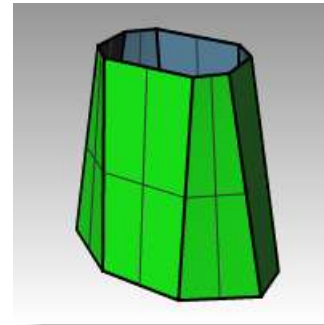
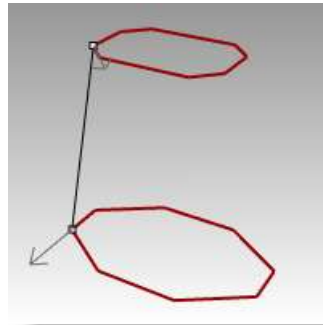
Chamfer Curves

**11** Continue creating chamfers on all of the corners as shown.



**To turn the curves into surfaces:**

- 1 Change to the **Surfaces** layer.
- 2 From the **Edit** menu, click **Select Objects**, and then click **Curves**.
- 3 From the **Surface** menu, click **Loft**.
- 4 **Adjust the seam** line if necessary, press **Enter**.
- 5 In the **Loft Options** dialog box, click **OK**.  
A surface is generated between the two chamfered rectangles.
- 6 **Save** your model.



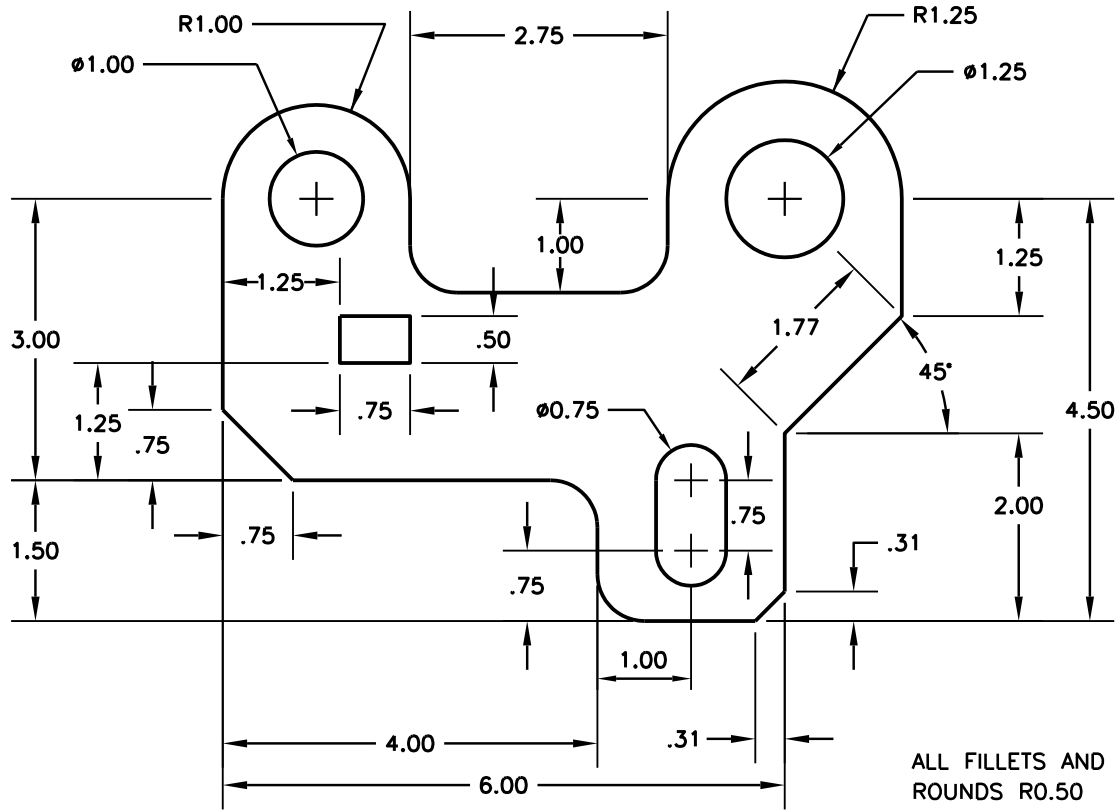
**Notes:**



Select Curves

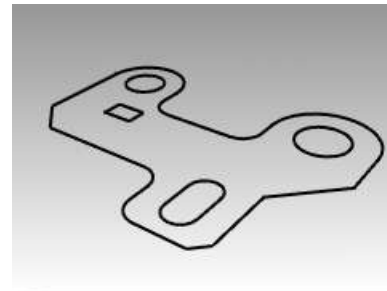
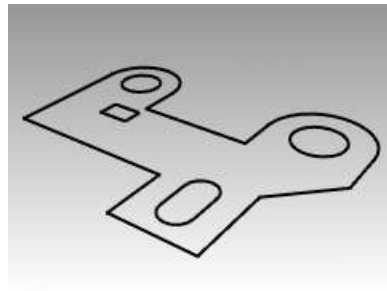
**Exercise 35—Practice with Fillet and Chamfer**

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



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

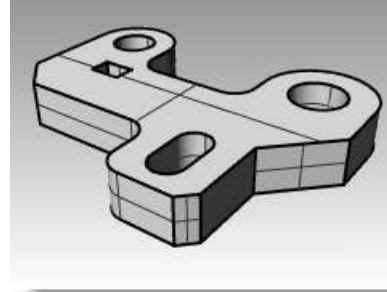
All fillets and rounds use a radius of 0.5 units.



**Notes:**

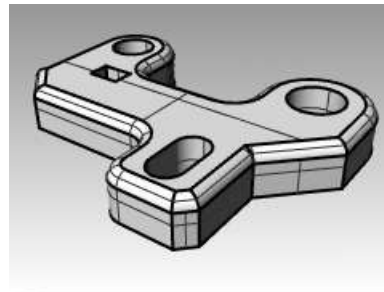
**To make it solid:**

- 1 From the **Edit** menu, click **Select Objects**, and then click **Curves**.
- 2 From the **Solid** menu, click **Extrude Planar Curve**, and then click **Straight**.
- 3 For the **Extrusion distance**, type **1** and press **Enter**.



**To fillet the top edge of the solid:**

- 1 From the **Solid** menu, click **Fillet Edge**, and then click **Fillet Edge**.
- 2 Change the **CurrentRadius** to **0.25**.
- 3 Select the edges around the top of the solid, press **Enter**.



**Notes:**



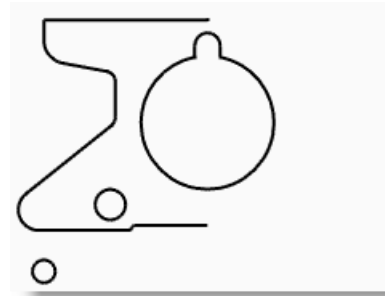
## Move

Use Move to move objects without changing orientation or size.

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

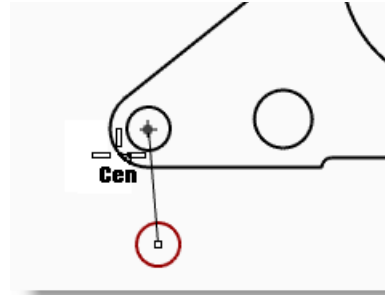
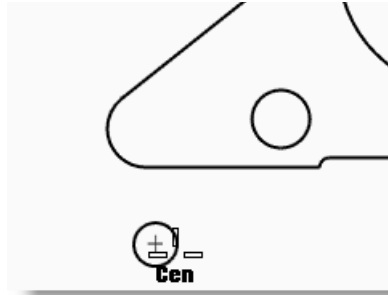
### Exercise 36—Move

- 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 For the **Point to move from**, snap to the center of the small circle.
- 4 For the **Point to move to**, snap to the center of the arc at the lower left of the object.

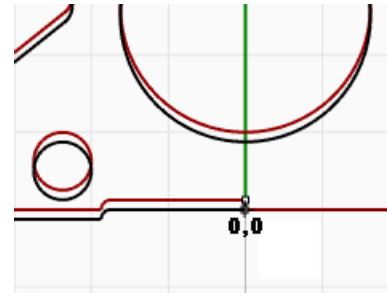
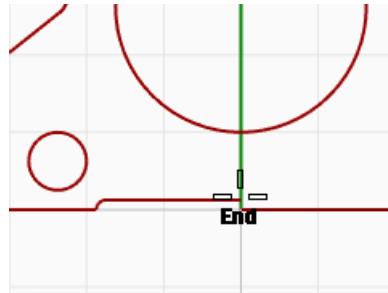


### Notes:



**To move using absolute coordinates:**

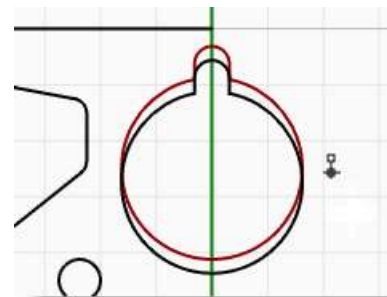
- 1 From the **Edit** menu, click **Select Objects**, and then click **Curves**.
- 2 From the **Transform** menu, click **Move**.
- 3 For the **Point to move from**, snap to the end of the line at the lower part of the object.
- 4 For the **Point to move to**, type **0,0** and press **Enter**.



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 Pick any point in the **Top** viewport.  
It is usually better if you pick near the object you are going to move.
- 4 For the **Point to move to**, type **r0,-.25** and press **Enter**.  
The circle moved down .25 units.



## Copy

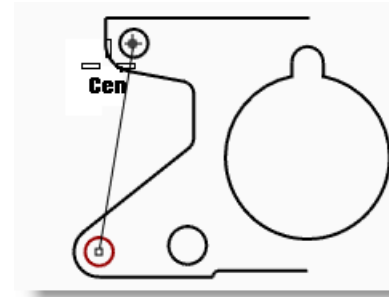
Copy duplicates selected objects and places them in a new location. The command can repeat to create more than one copy in the same command sequence.

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

### Exercise 37—Copy

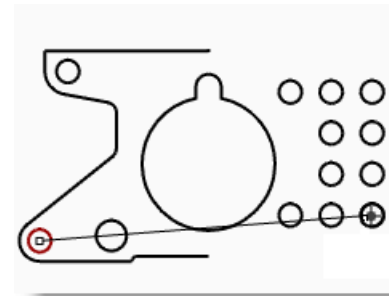
#### To copy using object snaps for placement:

- 1 Select the small circle at the lower left of the object.
- 2 From the **Transform** menu, click **Copy**.
- 3 For the **Point to copy from**, snap to the center of the small circle.
- 4 For the **Point to copy to**, 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 For the **Point to copy from**, snap to the center of the small circle.
- 4 For the **Point to copy to**, begin picking points on the screen.  
Each time you pick, a circle will be copied to that location.
- 5 Press **Enter** to end the command.
- 6 **Undo** the multiple copies.





## Notes:



## 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, or click **Undo** on the command prompt.

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

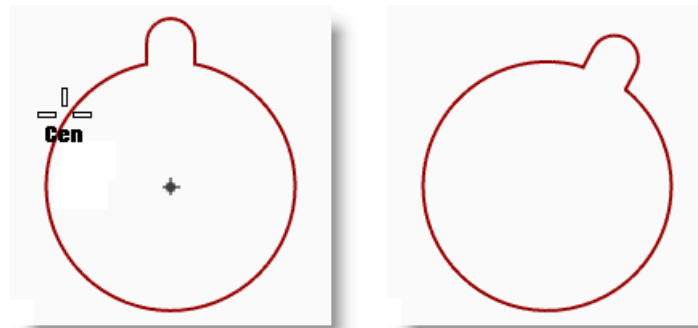
## Rotate

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 38—Rotate

- 1 Select the large slotted circle in the middle of the object.
- 2 From the **Transform** menu, click **Rotate**.
- 3 For the **Center of rotation**, snap to the center of the slotted circle.
- 4 For the **Angle**, type **-28** and press **Enter**.








Rotate 2-D

**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 39—Grouping

#### To group selected objects:

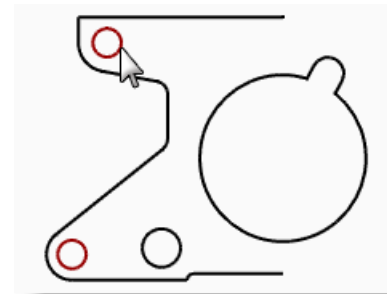
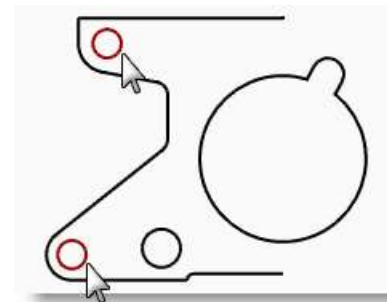
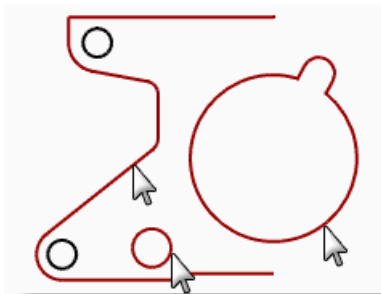
- 1 Select the two circles that you placed.
- 2 From the **Edit** menu, click **Groups**, and then click **Group**.

#### To add objects to a group:

- 1 Select the polyline on the left, the original circle, and the slotted circle in the center.
- 2 On the command line type **AddToGroup** or pick the **AddToGroup** icon from the **Grouping** toolbar, press **Enter**.

- 3 Select one of the circles in the previous group.

The objects are part of the group.



### Notes:



Group

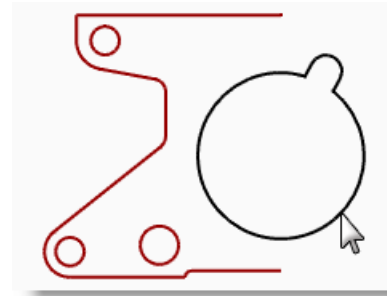


Add to group

### To remove an object from a group:

- 1 On the command line type **RemoveFromGroup** or pick the **RemoveFromGroup** icon from the **Grouping** toolbar.
- 2 For the **Select objects to remove from group**, select the slotted circle, press **Enter**.

The slotted circle is removed from the group.



### Notes:



Remove from group

## Mirror

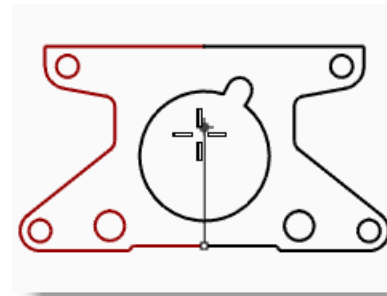
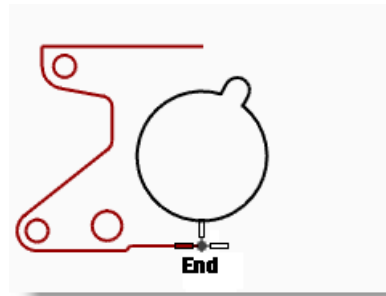
Mirror creates a copy of the objects flipped over a specified axis on the construction plane.

### Exercise 40—Mirror

- 1 Select the group.
- 2 From the **Transform** menu, click **Mirror**.
- 3 Type **0,0** or snap to the end of the line at the lower right of the part.
- 4 Toggle **Ortho** on and pick directly above the previous point.

Since you mirrored a group, in addition to having a mirror image copy, you also have two groups.

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



Mirror



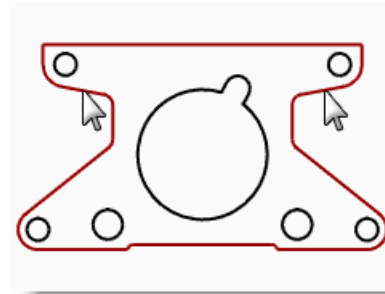
Ungroup

## 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 41—Join

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



## Notes:



Join

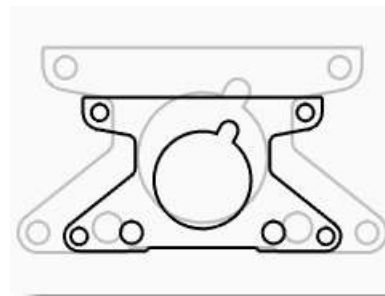
## Scale

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 42—Scale

- 1 From the **Edit** menu, click **Select Objects**, and then click **Curves**.
- 2 From the **Transform** menu, click **Scale**, and then click **Scale 2-D**.
- 3 For the **Origin point**, type **0** and press **Enter**.
- 4 For the **Scale factor**, type **.75** and press **Enter**.  
The whole part scaled to 75 percent of its original size.



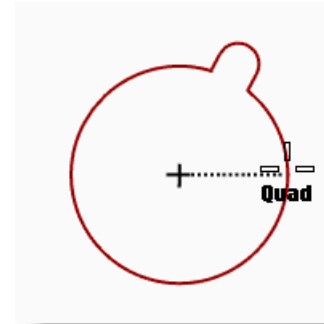
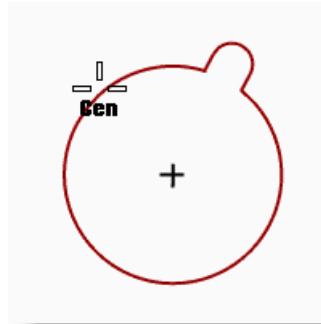
Scale 2-D

## Notes:

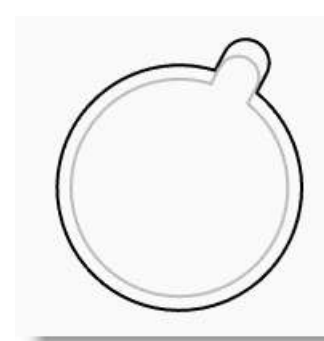
### To scale using the reference point option:

- 1 Select the slotted circle.
- 2 From the **Transform** menu, click **Scale**, and then click **Scale 2-D**.
- 3 For the **Origin point**, snap to the center of the slotted circle.
- 4 For the **First reference point**, snap to the quadrant of the slotted circle.

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

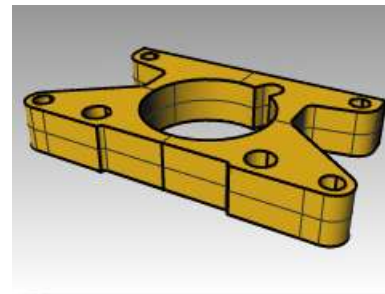


- 5 For the **Second reference point**, type **1.375** and press **Enter**.  
The slotted circle now has a larger radius of 1.375. Everything else was scaled proportionally.



### To make it solid:

- 1 From the **Edit** menu, click **Select Objects**, and then click **Curves**.
- 2 From the **Solid** menu, click **Extrude Planar Curves**, and then click **Straight**.
- 3 For the **Extrusion distance**, type **1** and press **Enter**.





### To scale in 3-D:

- 1 From the **Edit** menu, click **Select Objects**, and then click **Polysurfaces**.
- 2 From the **Transform** menu, click **Scale**, and then click **Scale 3-D**.
- 3 For the **Origin point**, type **0** and press **Enter**.
- 4 For the **Scale factor**, type **1.5** and press **Enter**.  
The solid is larger in every dimension.



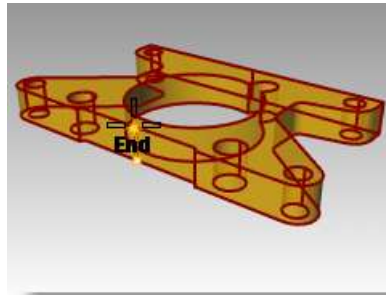
### Notes:



Scale 3-D

### To scale in one dimension:

- 1 From the **Edit** menu, click **Select Objects**, and then click **Polysurfaces**.
- 2 From the **Transform** menu, click **Scale**, and then click **Scale 1-D**.
- 3 For the **Origin point**, type **0** and press **Enter**.
- 4 For the **First reference point**, snap to the **Perpendicular** point at the top of the part.
- 5 For the **Second reference point**, type **.5** and press **Enter**.  
The object is half as thick.



Scale 1-D

## 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

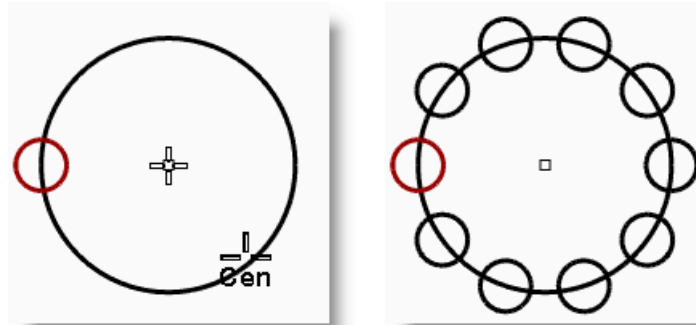
### Exercise 43—Polar Array

#### To create a polar array:

- 1 **Open** the model **Array.3dm**.
- 2 Select the small circle in the **Top** viewport.
- 3 From the **Transform** menu, click **Array**, and then click **Polar**.
- 4 For the **Center of polar array**, snap to the center of the large circle.
- 5 For the **Number of items**, type **10** and press **Enter**.
- 6 For the **Angle to fill**, check to see if it is set to **360**, press **Enter**.

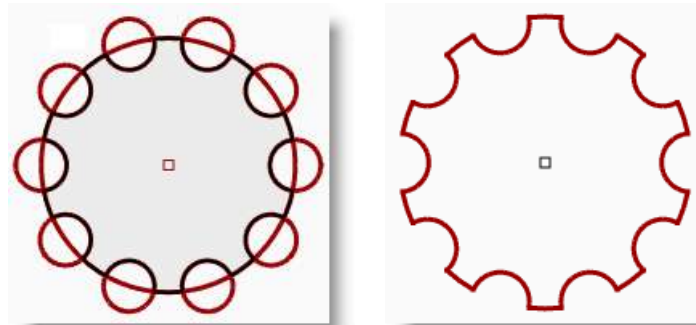
The small circle is arrayed around the larger one.

- 7 **Save** your model.



#### To make the shape for the column:

- 1 Window select the circles.
- 2 From the **Curve** menu, click **Curve Edit Tools**, then click **Curve Boolean**.
- 3 For the **Regions to keep**, click inside the large curve.  
The inside of the circle, excluding the smaller circles is shaded.
- 4 On the command line, click **DeleteInput**, click **All**, press **Enter**.



#### Notes:



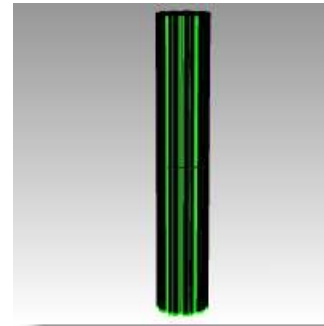
Polar Array

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

**Notes:**

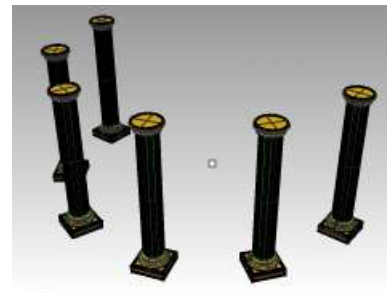
**To make the column:**

- 1 Select the new polycurve.
- 2 From the **Solid** menu, click **Extrude Planar Curve**, then click **Straight**.
- 3 For the **Extrusion distance**, type **14**, press **Enter**.



**To make a partial polar array:**

- 1 Turn on the **Base** layer.
- 2 Select the column base, the column, and the column cap.
- 3 From the **Edit** menu, click **Groups**, then click **Group**.  
The three parts are grouped into one.
- 4 Select the group.
- 5 From the **Transform** menu, click **Array**, and then click **Polar**.
- 6 For the **Center of polar array**, type **0**, press **Enter**.
- 7 For the **Number of items**, type **6** and press **Enter**.
- 8 For the **Angle to fill**, type **-180**, press **Enter**.  
Six columns are arrayed to fill 180 degrees in the negative direction.



**To create a rectangular array:**

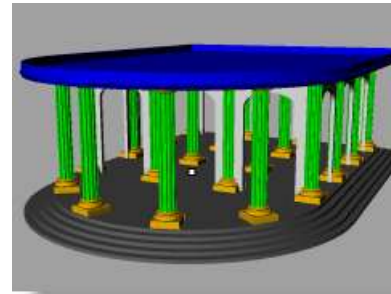
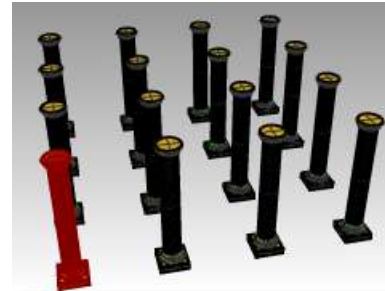
- 1** Select the same column group as the last exercise.
- 2** From the **Transform** menu, click **Array**, then click **Rectangular**.
- 3** For the **Number in X direction**, type **4**, press **Enter**.
- 4** For the **Number in Y direction**, type **4**, press **Enter**.
- 5** For the **Number in Z direction**, type **1**, press **Enter**.
- 6** For the **X spacing**, type **12** and press **Enter**.
- 7** For the **Y spacing**, type **12** and press **Enter**.

The columns are visible.

- 8** At this stage you can change the number in each direction or the spacing.

If you need to make a change, click the option in the command line and then make the adjustment.

- 9** Press **Enter** to accept.
- 10** Turn on all the layers to see the result.



**Notes:**



Rectangular Array

## Trim

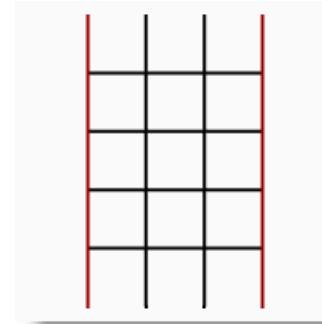
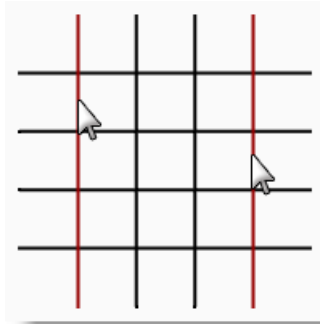
Trim cuts and deletes portions of an object to make it end precisely at its intersection with another object.

### Exercise 44—Trim

In this exercise, we will to pre-select the cutting objects.

#### To trim curves:

- 1 **Open** the model **Trim-Split.3dm**.
- 2 **Zoom Window** around the grid in the lower left corner of the **Top** viewport.
- 3 Pre-select the cutting objects by selecting the two outside vertical lines in the grid.
- 4 From the **Edit** menu, click **Trim**.
- 5 Select each of the horizontal lines at their left and right ends.  
The lines are trimmed to the cutting edges.
- 6 Press **Enter**.



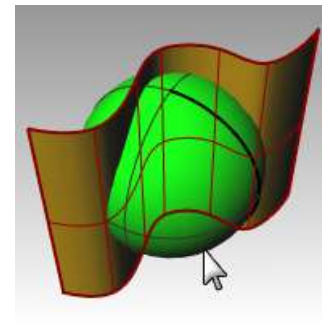
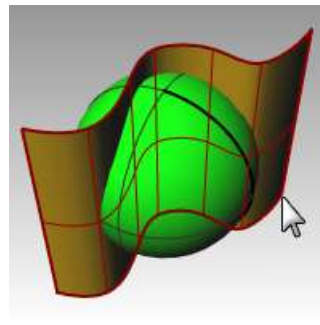
#### Notes:



Zoom Window

#### To trim surfaces:

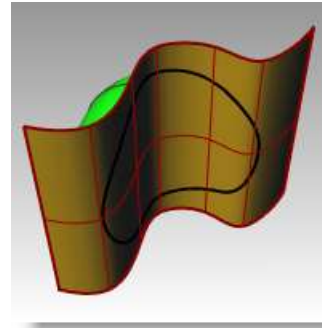
- 1 **Zoom Window** around a sphere and surface in the **Perspective** viewport.
- 2 Select the surface that intersects the sphere in the **Perspective** viewport as the cutting object.
- 3 From the **Edit** menu, click **Trim**.



Trim

4 For the **Object to trim**, pick the right side of the sphere.  
The sphere is trimmed at the surface.

5 Press **Enter**.



**Notes:**

## Split

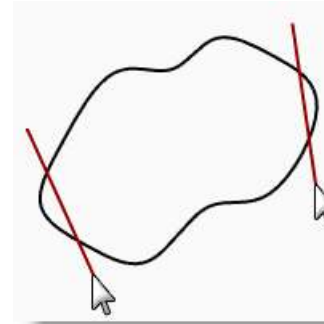
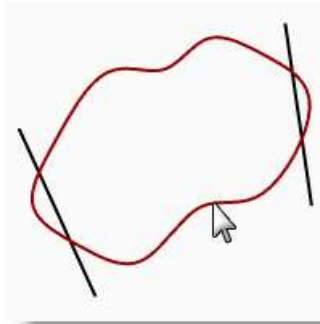
Splits one object with another into objects, splits a curve at a point you specify. The Split command breaks the object where it intersects the cutting object but does not delete anything.

### Exercise 45—Split

In this exercise, we will pre-select the object or objects we are going to split.

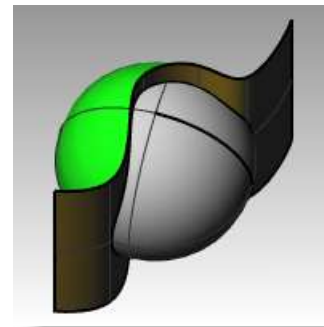
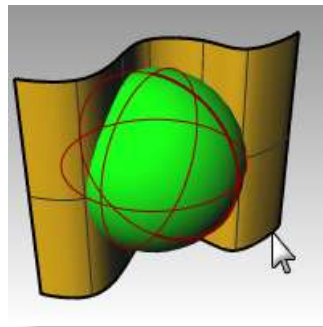
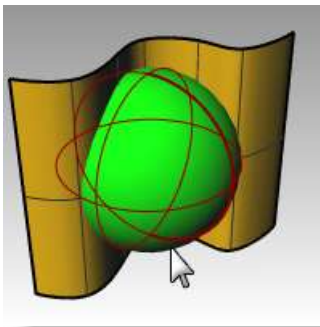
#### To split a curve:

- 1 **Zoom Window** around the closed curve in the lower right corner of the **Top** viewport.
- 2 Select the closed curve.
- 3 From the **Edit** menu, click **Split**.
- 4 Select the lines and press **Enter**.  
The curve is separated into four curves precisely where the lines intersect it.



#### To split a surface:

- 5 From the **View** menu, click **Zoom**, and then click **Extents All**.
- 6 From the **Edit** menu, click **Split**.
- 7 Select the sphere, and press **Enter**.
- 8 Select the surface that intersects the sphere and press **Enter**.  
The sphere is separated into two pieces precisely where the surface intersects it.



## Extend

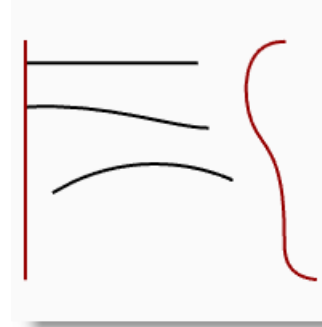
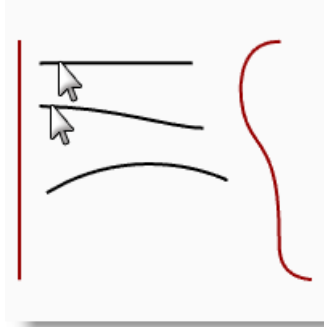
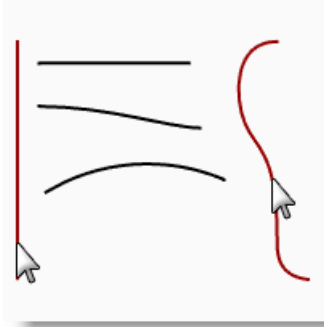
Extend lengthens an object to make it end precisely at its intersection with another object or you can lengthen an object when there is no intersection.

### Exercise 46—Extend

1 **Open** the model **Extend.3dm**.

2 From the **Curve** menu, click **Extend Curve**, and then click **Extend Curve**.

3 For the **boundary objects**, select the line at the left and the curve at the right.



4 Press **Enter**.

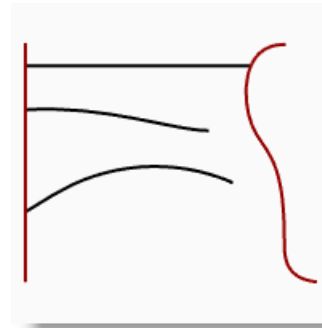
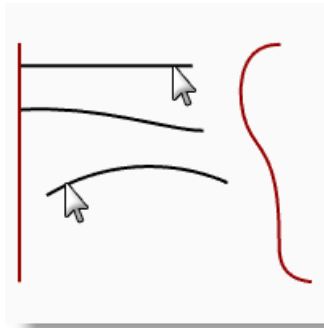
5 For the **curve to extend**, select the left end of the line and the middle curve. The line and curve extend to touch the boundary edge on the left.

6 For the next **curve to extend**, click **Type=Natural** on the command line.

7 Change to **Type=Line**.

8 Select left end of the bottom curve and the right end of the line.

The curve and line extend to touch the boundary. The extension is a straight segment.



### Notes:



Extend Curve

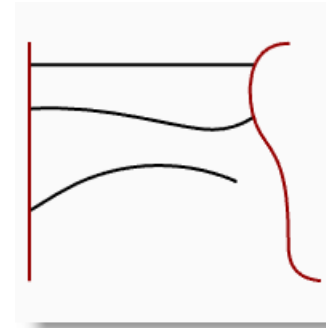
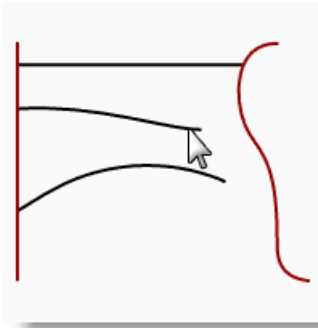


**9** For the next **curve to extend**, click **Type=Line** on the command line.

**10** Change to **Type=Arc**.

**11** Select the right end of the middle curve.

The curve extends with a tangent arc to touch the boundary.

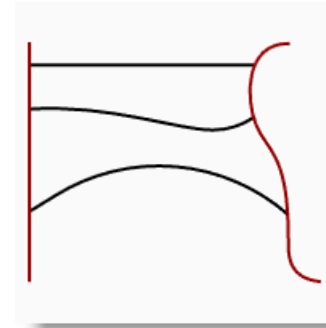
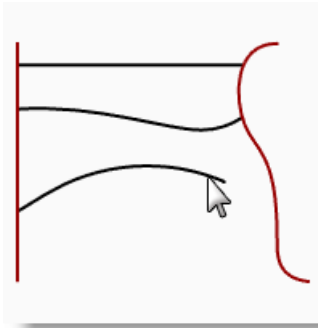


**12** For the next **curve to extend**, click **Type=Arc** on the command line.

**13** Change to **Type=Smooth**.

**14** Select the right side of the bottom curve.

The curve extends with a curvature (G2) continuous extension to touch the boundary.



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

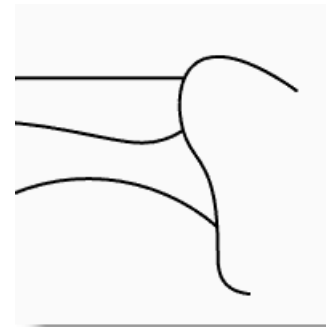
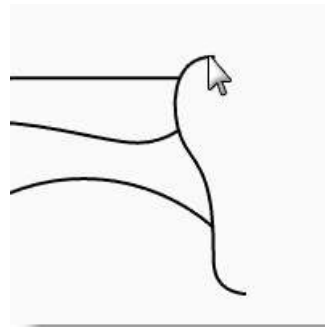
#### To extend with a set extension length:

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

**2** **Enter extension length**, type **4** and press **Enter**.

**3** Select the upper end of the curve on the right.

The curve extends exactly 4 units.

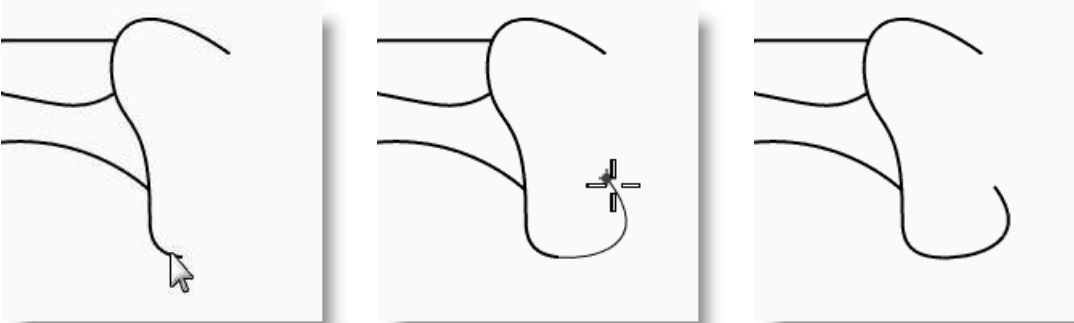


**4** Press **Enter** to end the command.

#### Notes:

**To extend a curve dynamically:**

- 1 From the **Curve** menu, click **Extend Curve**, and then click **Extend Curve**.
- 2 Press **Enter** for a dynamic extension.
- 3 Select the lower end of the curve on the right.

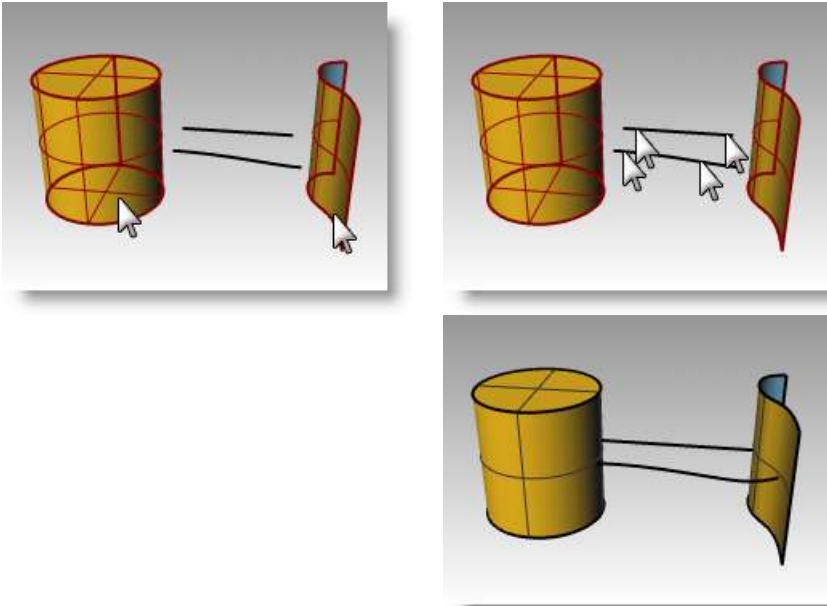


The curve continues with the cursor.

- 4 Click to end the extension.
- 5 Press **Enter** to end the command.

**To extend to a surface:**

- 1 From the **Curve** menu, click **Extend Curve**, and then click **Extend Curve**.
- 2 For the **boundary objects**, select the cylinder on the left and the surface on the right.
- 3 Press **Enter**.
- 4 Change the type to **Type=Arc**.
- 5 Select both ends of the line and the curve.



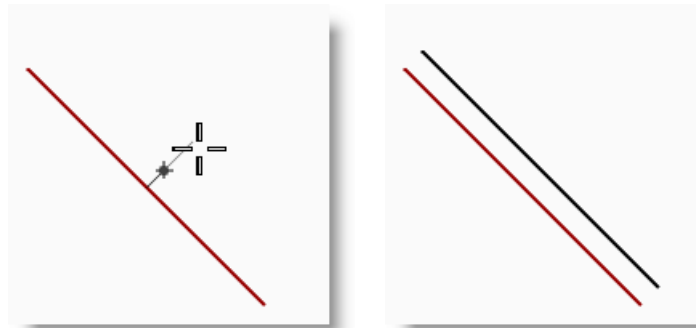
The curves extend to the surface of the cylinder and to the surface.

## 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.

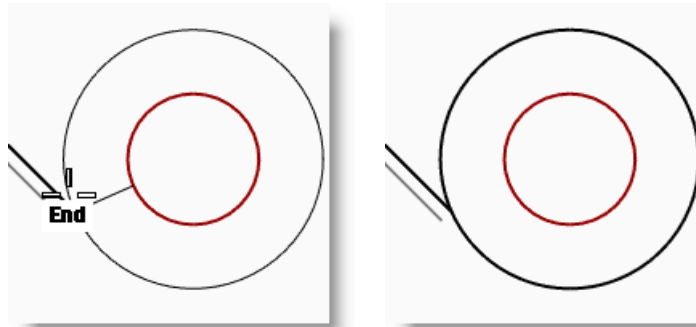
### Exercise 47—Offset

- 1 **Open** the model **Offset.3dm**.
- 2 Maximize the **Top** viewport.
- 3 Select the line.
- 4 From the **Curve** menu, click **Offset Curve**.
- 5 For the **Side to offset**, pick on the upper right side of the line.  
A parallel line is created.



### To offset with the through point option:

- 1 Turn on the **End** osnap.
- 2 Select the circle.
- 3 From the **Curve** menu, click **Offset Curve**.
- 4 For the **Side to offset**, pick click **Throughpoint** on the command line.
- 5 For the **Throughpoint**, snap to the lower right end of the line you offset.  
A concentric circle is created that goes through the endpoint of the line.



### Notes:



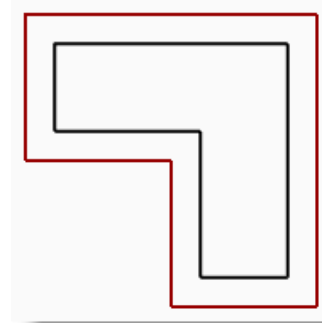
Offset Curve

## Notes:

### To offset a polyline with sharp corners:

- 1 Select the polyline.
- 2 From the **Curve** menu, click **Offset Curve**.
- 3 Type **1** and press **Enter** to change the offset distance.
- 4 For the **Side to offset**, pick inside the polyline.

The polyline is offset with sharp corners.

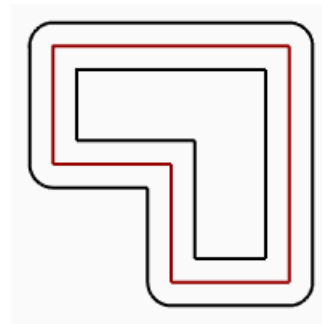


### To offset a polyline with round corners:

- 1 Select the polyline.
- 2 From the **Curve** menu, click **Offset Curve**.
- 3 Click **Corner=Sharp** on the command line.
- 4 Change the type to **Corner=Round**.
- 5 Pick outside the polyline.

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

Other corner options include **Smooth** and **Chamfer**. Smooth makes a tangent curve that is smoother than an arc at each vertex, while Chamfer makes a bevel at each vertex.



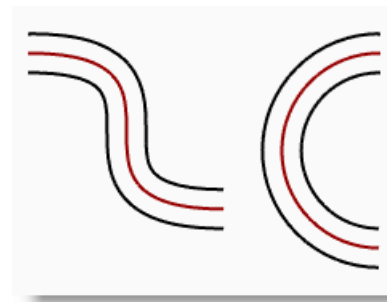
### To offset a curve with the bothsides option:

- 1 Select the free-form curve.
- 2 From the **Curve** menu, click **Offset Curve**.
- 3 Click **Bothsides** on the command line.
- 4 For **Side to offset**, pick on either side of the curve.

Free-form curves are created on both sides of the selected curve.

- 5 Repeat this process on the arc.

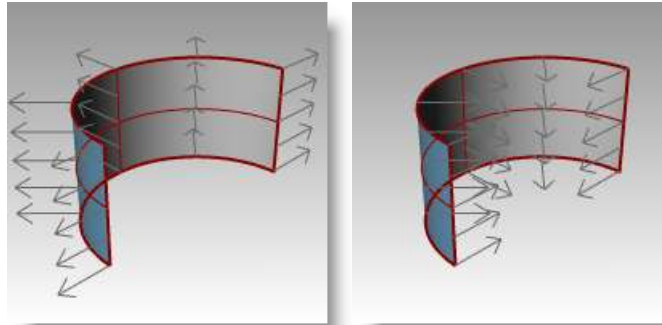
Concentric arcs are created on both sides of the selected arc.



### To offset a surface:

- 1 Select one of the open surfaces.
- 2 From the **Surface** menu, click **Offset Surface**.
- 3 Place your cursor over the surface and click your left mouse button.

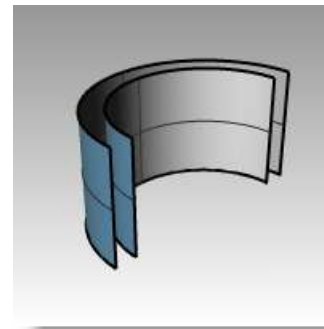
Notice that the direction of the arrows changed. The normal direction of the surface was flipped by the mouse click. If you have a positive number for offset distance, the surface will be offset in the direction of the normal. If you have a negative number it will offset the opposite direction.



*The normals should point toward the concave side of the surface.*

- 4 Press **Enter**.

The surface is offset in the direction of the normals.



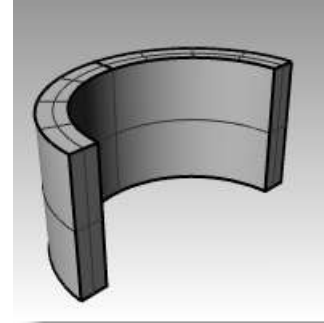
### Notes:



Offset Surface

### To offset a surface to a solid:

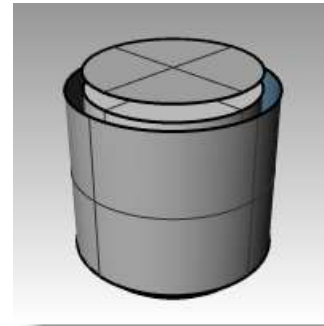
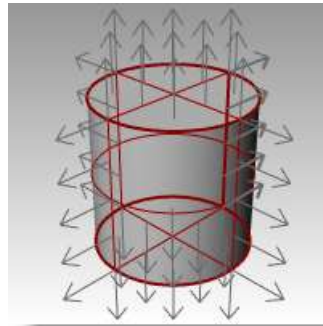
- 1 Select the other open surface.
- 2 From the **Surface** menu, click **Offset Surface**.
- 3 Click on the surface to change the normal direction if necessary.
- 4 Choose the **Solid** option.
- 5 Press **Enter** to create the offset surface and the surfaces needed to make the solid.



### To offset a polysurface:

Offsetting polysurfaces usually doesn't give the desired results. In this example we will demonstrate some of the problems.

- 1 Select the cylinder.
- 2 From the **Surface** menu, click **Offset Surface**.  
The normals on a closed polysurface will always point to the outside.
- 3 Press **Enter**.  
Each surface of the polysurface is offset as a separate piece.



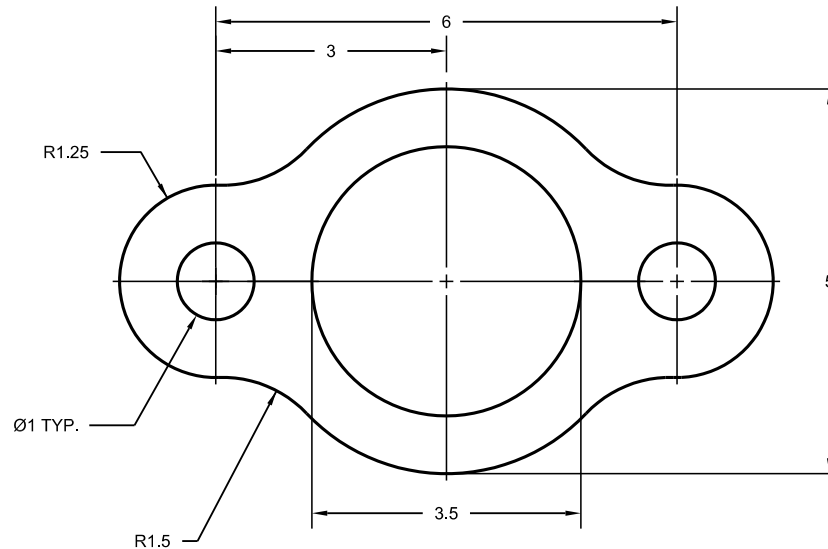
### Notes:

### Exercise 48—Practice

- 1 Start a new model, use the **Small Objects - Inches.3dm** template. Save as **Gasket1**.



- 2 Use **Circle**, **Arc**, **Trim**, **Fillet**, and **Join** to create the part shown.
- 3 Use **Extrude Planar Curve > Straight** on the **Solid** menu to create the 3-D part. The extrusion thickness is **.125**.



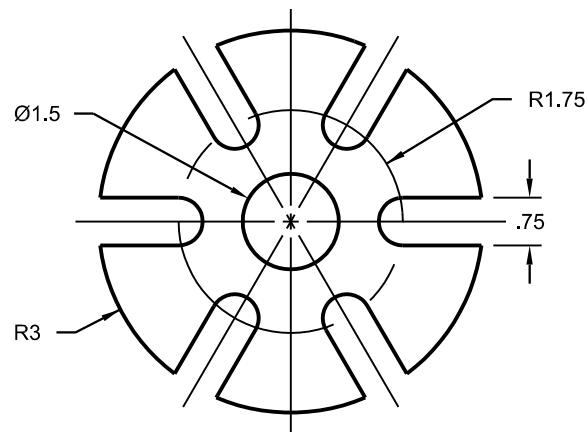
Notes:

### Exercise 49—Practice

- 1 Start a new model, use the **Small Objects - Inches.3dm** template. Save as **Cam**.



- 2 Use **Circle**, **Arc**, **Line**, **Trim**, **Join**, and **Array** to draw the part shown.
- 3 Use **Extrude Planar Curve > Straight** on the **Solid** menu to create the 3-D part. The extrusion thickness is **0.5**.

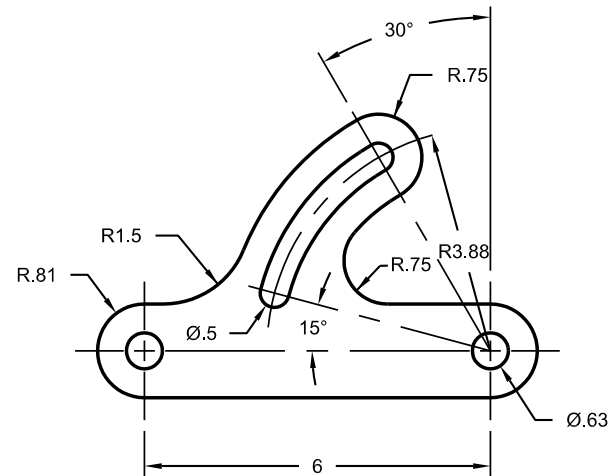


### Exercise 50—Practice

- 1 Start a new model, use the **Small Objects - Inches.3dm** template. Save as **Link**.



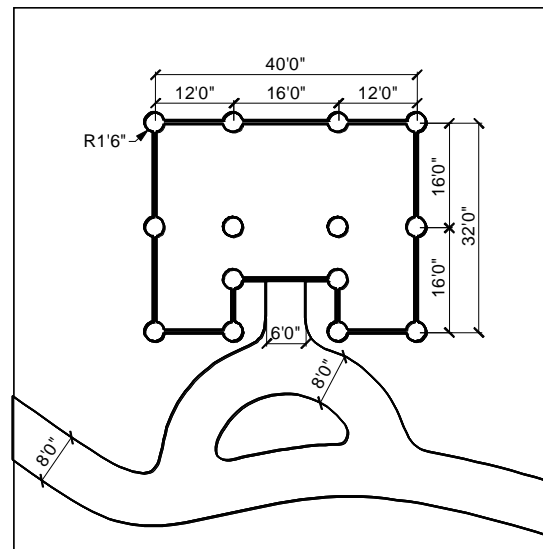
- 2 Use **Line, Arc, Trim, Offset, Join, Fillet** and **Circle** to draw the part shown.
- 3 Use **Extrude Planar Curve > Straight** on the **Solid** menu to create the 3-D part. The extrusion thickness is **0.5**.



### Notes:

### Exercise 51—Practice

- 1 Start a new model, use the **Large Objects - Inches.3dm** template. 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.

## Exercise 52—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 Objects**, and then click **Curves**.

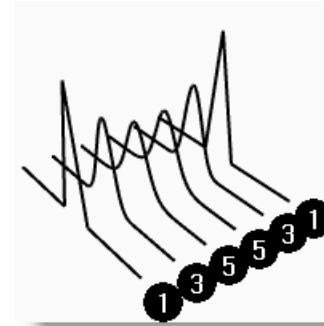
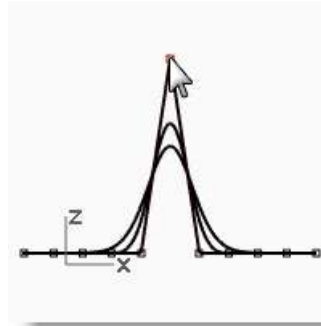
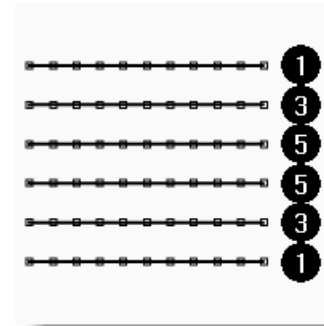
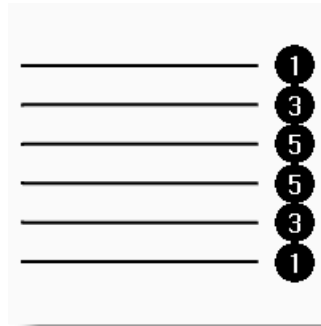
**4** From the **Edit** menu, click **Control Points**, and then click **Control Points On**. (Press **F10**.)

**5** In the **Front** viewport, select the middle row of points.

**6** Drag the points vertically, **8** units.

Notice that the degree 1 curves (polylines) come to a point at each moved control point and the control points are exactly on the curve.

The degree 3 and 5 curves are smooth. The degree 3 curves have more curvature than the degree 5 curves



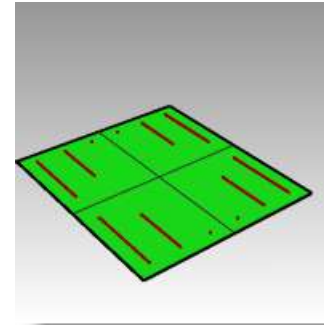
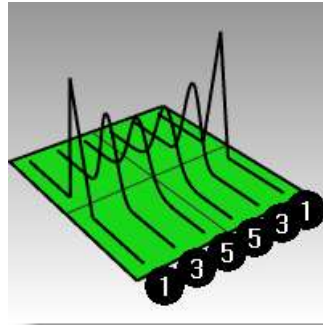
Control Points On

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.

### To check the differences:

- 1 Turn **off** the control points by pressing the **F11** key or use the **Esc** key.
- 2 Turn the **Plane** layer on.
- 3 Select the curves and the plane.
- 4 From the **Curve** menu, click **Curve from objects**, then click **Intersection**.

Lines are shown on the surface showing the intersections.



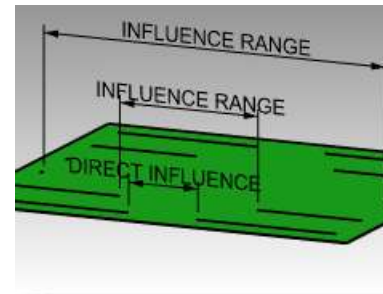
- 5 Note the difference between the degree 3 and 5 curves.

The higher the degree, the greater the influence over the span of a curve, when you edit control points.

As you can see by the illustration, editing one point out of 11, on the degree 5 curve, influenced most of the curve. The degree 3 curve has a sharper curvature, because the influence of one point takes place over a shorter span.

- 6 Undo twice, until you get back to seeing just the curves.

The intersections will disappear, and the Plane layer is off.

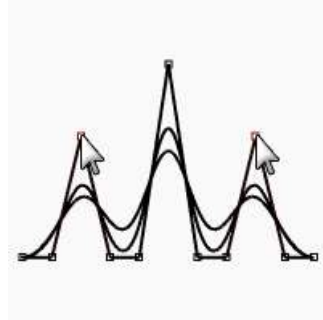


### To continue editing control points:

- 1 In the **Front** viewport, select the third two rows of points on either side of the center.
- 2 Drag the points vertically, **5** units.
- 3 Turn **off** the control points by pressing the **F11** key or use the **Esc** key.

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.



### To make a lofted surface:

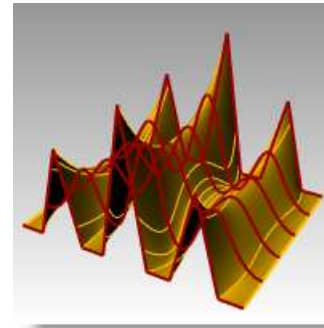
- 1 Select the curves.
- 2 From the **Surface** menu, click **Loft**.
- 3 In the **Loft options** dialog box, click **OK**.

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

- 4 Select the surface.
- 5 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.**

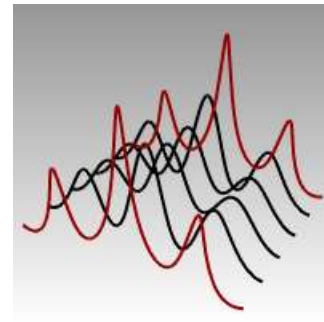
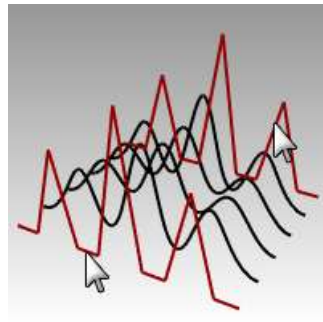
- 6 **Undo** the loft.



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

- 1 Select both polylines.
- 2 From the **Edit** menu, click **Rebuild**.
- 3 In the **Rebuild Curve** dialog box change the point count to **11** and the degree to **3**, and click **OK**.

A degree 3 curve cannot have kinks. The curve smoothes and changes shape.



Rebuild

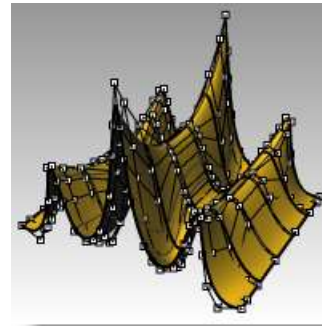
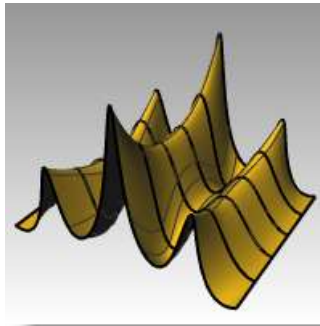
### To loft a surface over the curves:

- 1 Select all of the curves.
- 2 From the **Surface** menu, click **Loft**.
- 3 In the **Loft Options** dialog box, click **OK**.

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

- 4 Select the surface.
- 5 Turn on the control points.

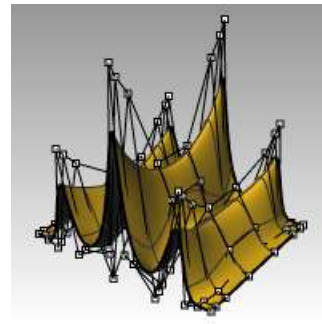
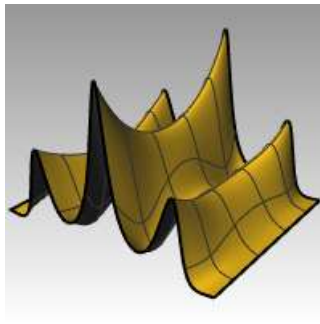
Note that there are more control points than what you started with. Rhino adds control points to make the surface conform the original curves.



### To rebuild a surface:

- 1 Turn off the control points.
- 2 Select the surface.
- 3 From the **Edit** menu, click **Rebuild**.
- 4 In the **Rebuild Surface** dialog, change the point count to **8** in U direction and **13** in the V direction. Change the degree to **3** for both U and V. Check **Delete input**.

The surface is smoother with fewer control points.



Rebuild Surface

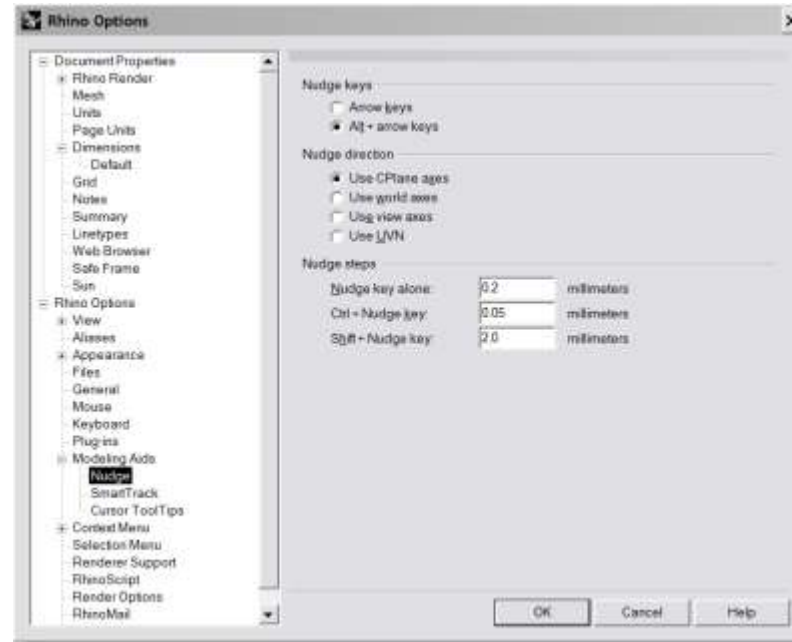
## Nudge Controls

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, Alt+Ctrl, and Alt+Shift keys.

### To change the nudge settings:

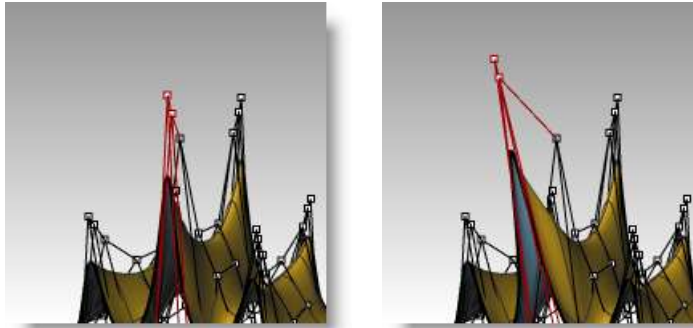
- 1 From the **Tools** menu, click **Options**.
- 2 In the **Options** dialog box, on the **Modeling Aids** page, note the **Nudge settings**.

Any of these values can be changed.



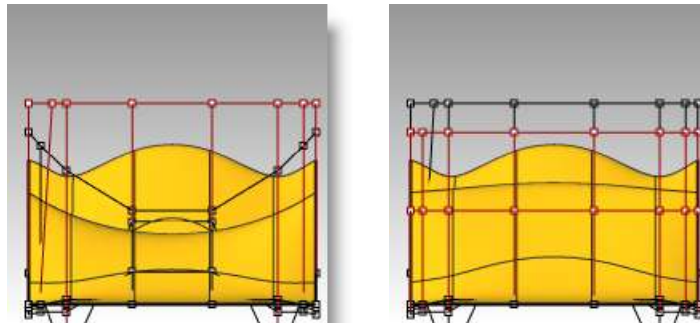
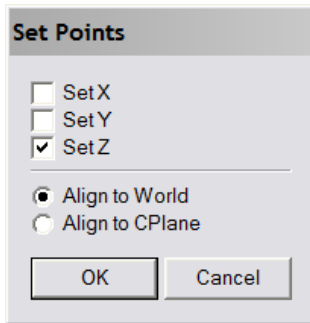
### To use Nudge keys to move control points:

- 1 Select one or two control points in the **Front** viewport.  
Notice that it moves (nudges) the point a small amount.
- 2 Hold down the **Alt** key down and press an arrow key.  
The movement is much smaller.
- 3 Hold the **Alt** and the **Ctrl** key down and press another arrow key.  
The movement is magnified.
- 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 in one row along the top of the surface.
- 2 From the **Transform** menu, click **Set Points**.
- 3 In the **Set Points** dialog box, check **Set Z**, and uncheck **Set X** and **Set Y**.
- 4 In the Right viewport move the points and click.  
The control points are aligned in the World Z direction.
- 5 Repeat this on some of the other rows of points.



Set XYZ Coordinates

### Exercise 53—Practice with curves and control point editing

- 1 Start a new model, use the **Small Objects - Millimeters.3dm** template. Save as **Glass**.
- 2 Use the **Curve** command to create a half cross-section of the glass.
- 3 Use control point editing to adjust the curve until you get the desired shape.

#### To make it 3-D:

- 1 From the **Surface** menu, click **Revolve**.
- 2 Select the curve you created, and press **Enter**.
- 3 For the **Start of revolve axis**, pick one end of the curve.

- 4 For the **End of revolve axis**, pick the other end of the curve.

- 5 For the **Start Angle**, click **Deformable=Yes**.

This changes the structure of the revolved surface to make it easier to deform without creating kinks.

- 6 For the **Start Angle**, click **FullCircle**.

- 7 **Save** your model.

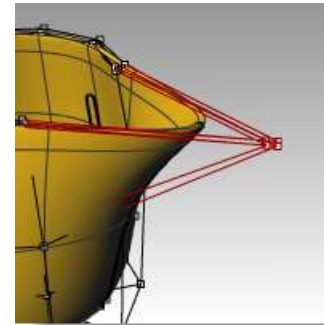
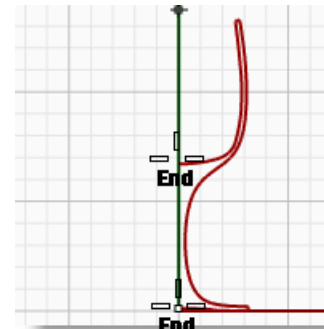
- 8 Experiment with adjusting the control points to see what happens.

- 9 **Incremental Save** your model.

- 10 Delete the surface.

- 11 Change the original curve and make another revolved surface.

- 12 **Incremental Save** your model.





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

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# 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.”

### Exercise 54—Creating a rubber duck

- 1 Start a new model, use the **Small Objects - Millimeters.3dm** template. 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” in the *Rhino Help* index.

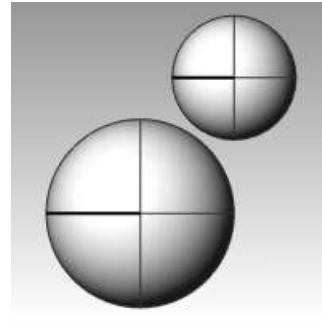


### Create the body and head shapes

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 Pick a point in the **Front** viewport.
- 3 Pick another point in the same viewport to create a sphere.
- 4 Repeat this procedure for the second sphere.

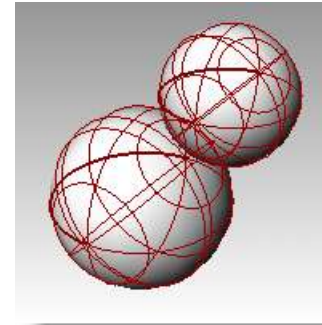
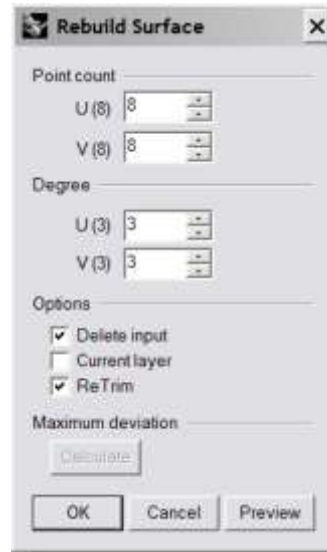
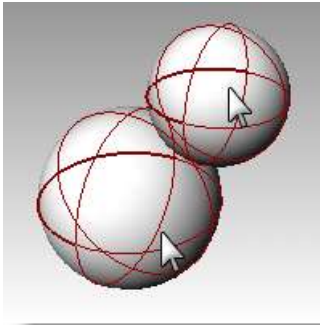


### Notes:



**To make the spheres deformable:**

- 1 Select both spheres.
- 2 From the **Edit** menu, click **Rebuild**.
- 3 In the **Rebuild Surface** dialog box, change the **Point Count** to **8** for both **U** and **V**.
- 4 Change the **Degree** to **3** for both **U** and **V**.
- 5 Check **Delete Input**, clear **Current Layer**, and click **OK**.



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

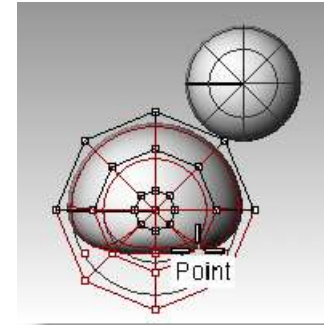
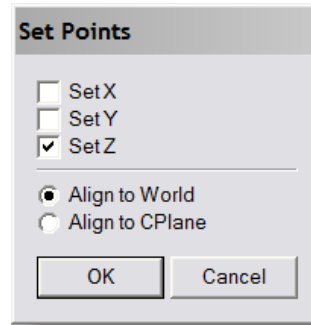
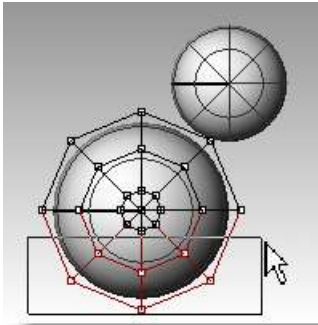
**Notes:**



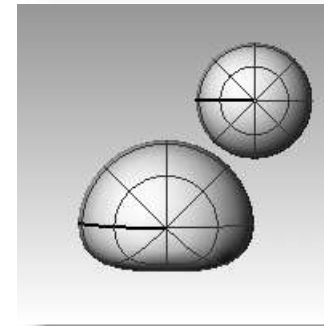
Rebuild Surface

### 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.

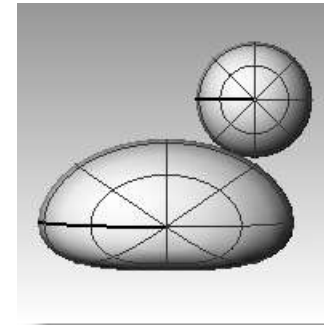
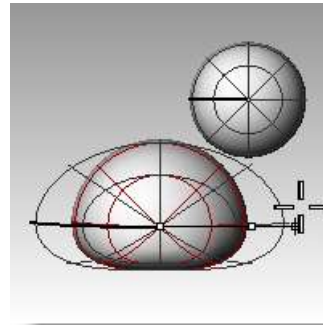


- To window select, drag a box left to right around the control points.
- 4 From the **Transform** menu, click **Set Points**.
  - 5 In the **Set Points** dialog box check **Set Z**, and **Align to World** as illustrated above.
  - 6 Drag the selected control points up, snap to one of the upper most points. 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**, and then click **Scale1D**.
- 3 For the **Origin point**, pick a point near the center of the body sphere.
- 4 For the **First reference point**, with **Ortho** on, pick a point to the right in the **Front** viewport.
- 5 For the **Second reference point**, pick a point further to the right in the **Front** viewport.



The body is shaped like an ellipsoid.

### Notes:



Control Points On



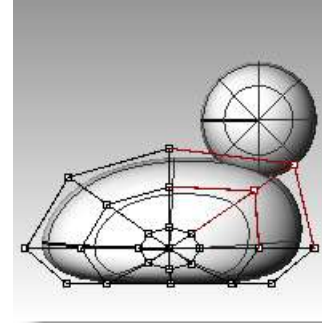
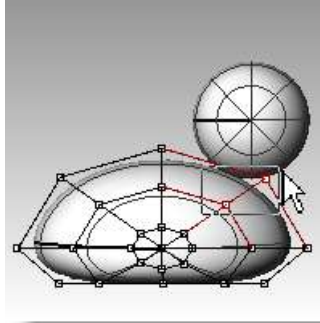
Set XYZ Coordinates



Scale 1D

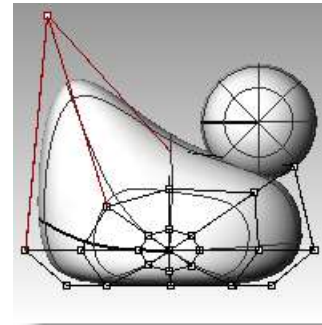
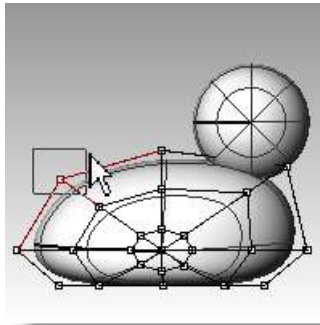
## To reshape the chest and tail

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



- 3 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.



**Notes:**

### 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 For the **Surface for knot insertion**, pick the body surface.

A surface isocurve will be displayed. It will either be in the U or the V direction.

- 3 Change the direction if necessary.

- 4 For the **Point on surface to add knot**, pick a point midway between the tail and the center of the body.

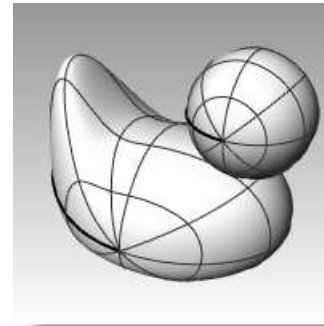
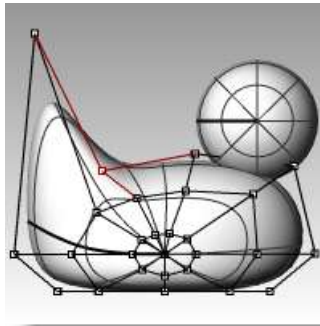
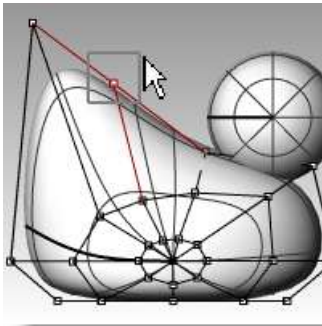
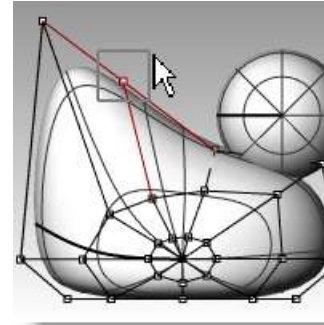
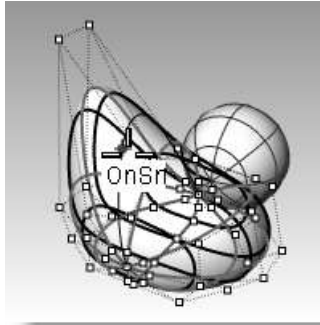
- 5 Press **Enter** to end the command.

A new set of isocurves and a new row of points 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:



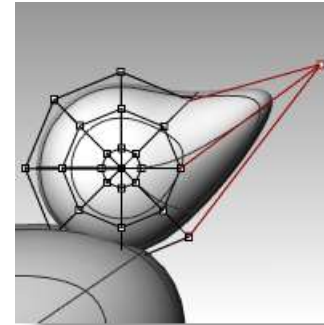
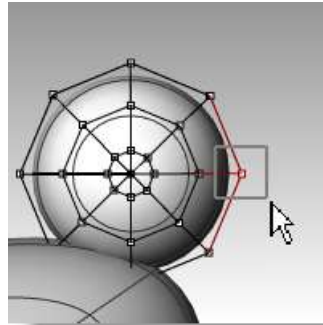
Insert Knot

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

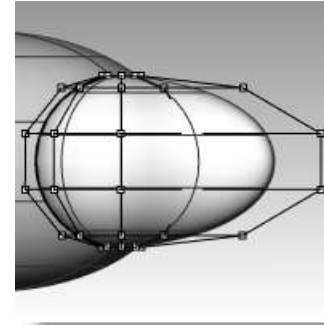
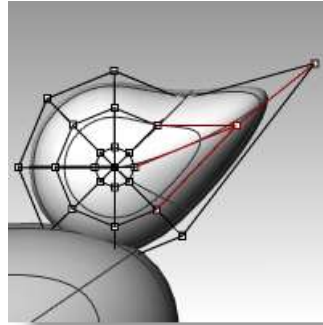
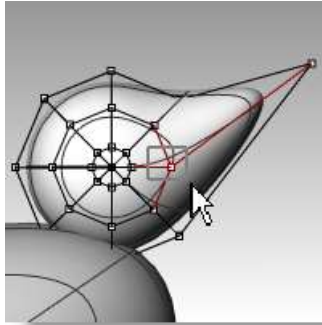


**To create the head:**

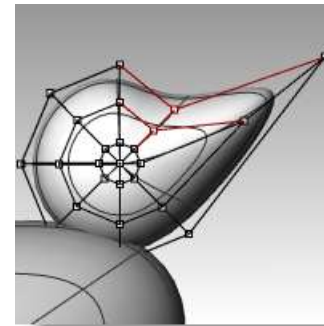
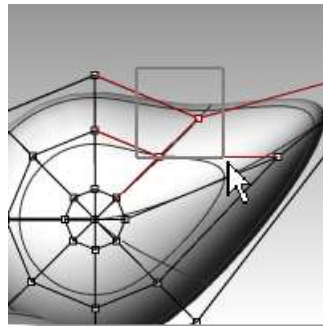
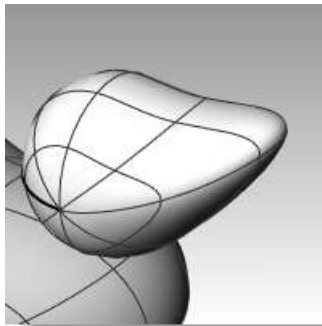
- 1 In the **Front** viewport, select the small sphere.
- 2 From the **Edit** menu, click **Control Points**, and then click **Control Points On**.
- 3 Select the control points on the right side and drag them to begin forming the bill.



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



**Notes:**

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

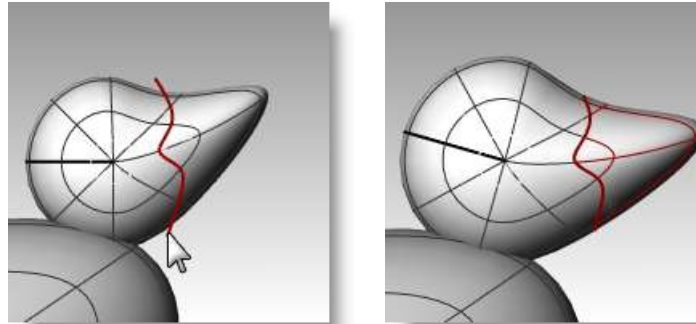
## Separate the bill from the head

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 on the right.
- 2 Select the head.
- 3 From the **Edit** menu, click **Split**.
- 4 For the **Cutting object**, pick the curve you just created.
- 5 Press **Enter**.

The bill and head are now separate surfaces. This will allow you to render the surfaces with different colors.



### Notes:

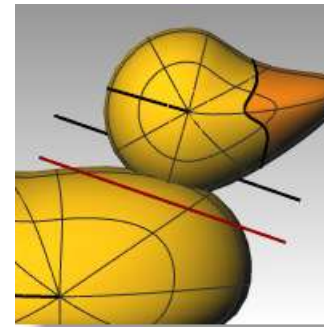
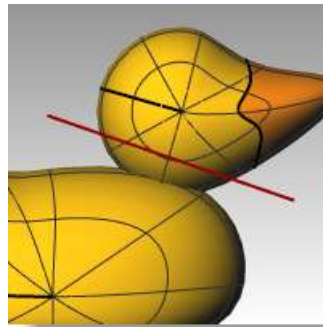
## Notes:

### 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 and the body:

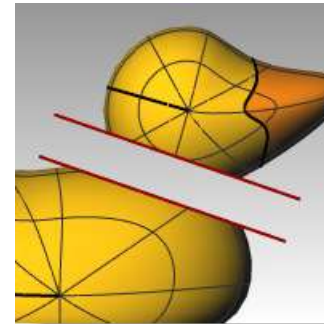
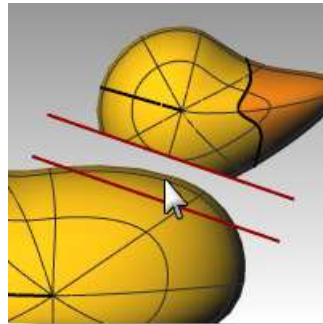
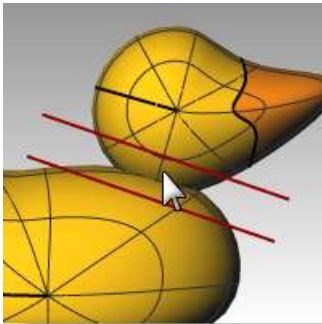
- 1 Draw a **Line** across the bottom of the head.
- 2 Make a copy of the line and adjust it so that it intersects the top of the body as illustrated on the right.  
It is important that the lines intersect the bottom of the head and the body completely.
- 3 Select the lines.
- 4 From the **Edit** menu, click **Trim**.



- 5 For the **Object to trim**, pick the bottom of the head and the top of the body.

The bottom of the head and the top of the body is trimmed.

- 6 **Save** your model.



Trim

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

- 1 From the **Surface** menu, click **Blend Surface**.
- 2 For the **Segment for first edge**, pick the edge curve at the bottom of the head.
- 3 If the whole edge is not selected, click **All** on the command line.
- 4 Press **Enter** to go to the next stage of the command.
- 5 For the **Segment for second edge**, pick the edge curve at the top of the hole in the body.
- 6 If the whole edge is not selected, click **All** on the command line.
- 7 Press **Enter** to go to the next stage of the command.

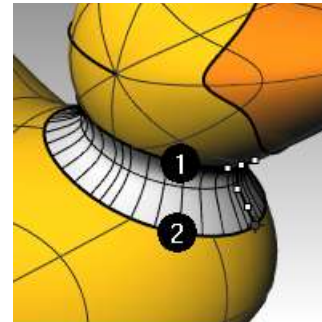
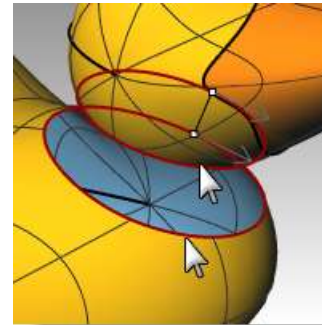
- 8 In the **Blend Bulge** dialog box, check the **Preview** check box.

The blend surface between the body and the head is displayed.

- 9 Make any adjustments you desire by moving the slider bars in the dialog box, click **OK** when finished.

If you check the box at the left of the sliders the surface adjustments are symmetrical.

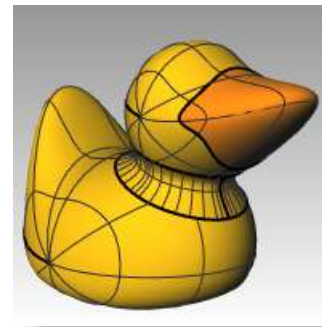
- 10 **Save** your model.



### 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.



### Notes:



Blend Surface



Join

## Notes:

### To make an eye:

For this part of the exercise we will make a curve and revolve it to get the surfaces for the eye.

- 1 Toggle **Ortho** and **Snap** on to help.
- 2 From the **Curve** menu, click **Conic**.
- 3 In the **Front** or **Right** viewport make a conic curve as illustrated.  
**Start of Conic (1), End of Conic (2), Apex (3), Curvature point.**
- 4 Use **SmartTrack**, with **Point**, **End** and **Int** osnaps on, to help place the two points as illustrated.

These points will be used for placement of the eye on the head.

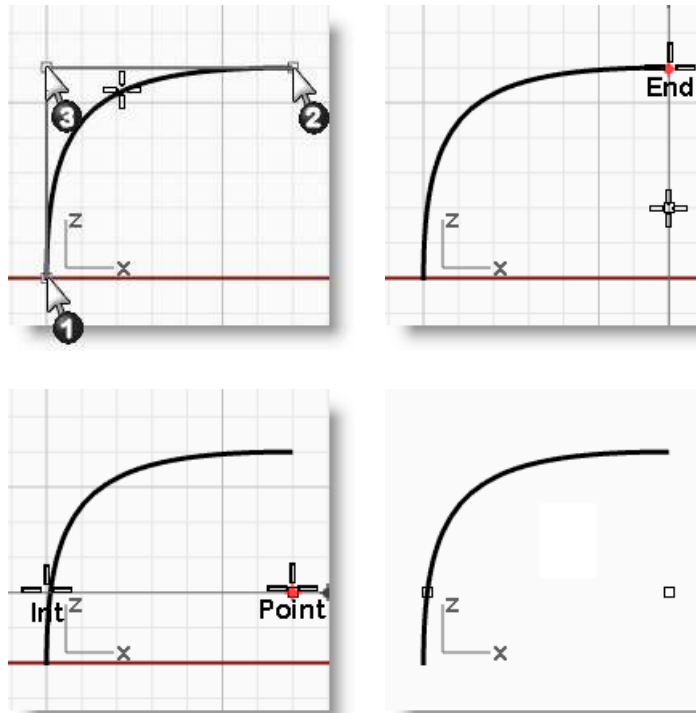
- 5 From the **Curve** menu, click **Point Object**, then click **Multiple Points**.
- 6 For the **Location of point object**, hover over the end of the conic until the point activates, drag your cursor down and click to place the point.

The point should be placed above the lower end of the conic.

This will be the insertion point for eye.

- 7 For the **Location of point object**, hover over the point you just made until the point activates, drag your cursor left and click when you reach the intersection point.

This point is for scaling and rotating the eye.

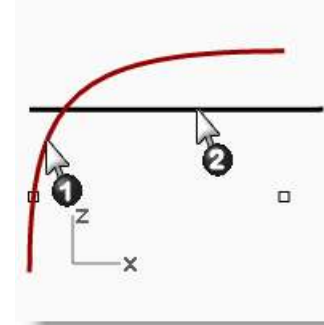
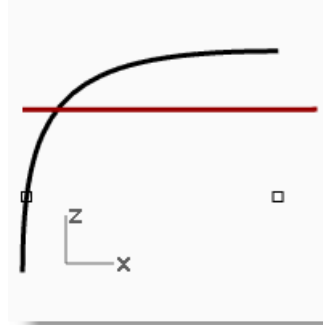


### To split the curve:

- 1 Draw a **line** that intersects the conic
- 2 Select the conic curve,
- 3 From the **Edit** menu, click **Split**.
- 4 For the **Cutting object**, select the line.

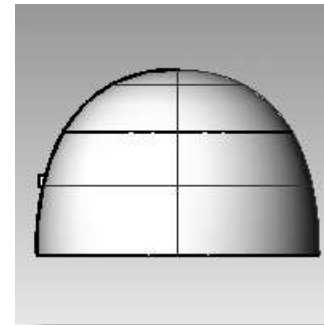
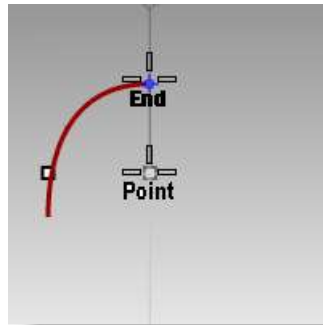
Splitting the curve allows you to assign a different color and material properties for the eye and the pupil.

This step can be done at this stage or after you make the surface.



### To make the surface:

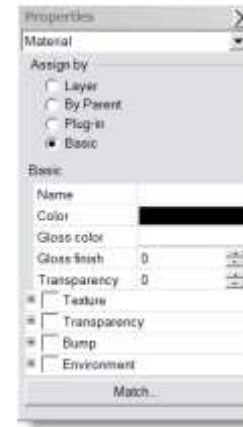
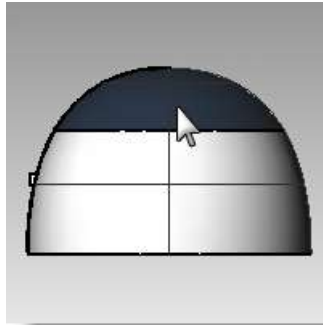
- 1 Select both parts of the conic curve.
- 2 From the **Surface** menu, click **Revolve**.
- 3 For the **Start of revolve axis**, snap to the point.
- 4 For the **End of revolve axis**, snap to the end of the conic.
- 5 For the **Start Angle**, click **FullCircle**.



**Notes:**

### To assign display color and material color:

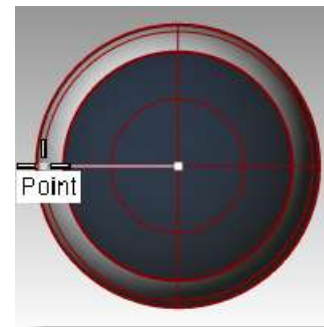
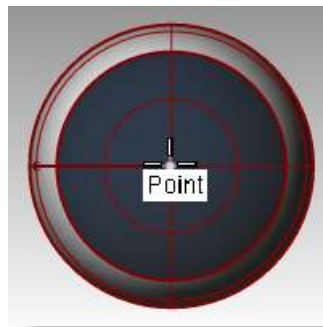
- 1 Select the top of the eye.
- 2 From the **Edit** menu, click **Object Properties**.
- 3 For the **Display** color, choose a contrasting color.
- 4 In the **Properties** window, on the **Material** page, click **Basic**, click on the **Color** button, and select a color for the pupil of the eye, like black.
- 5 Repeat these steps for the lower part of the surface. Set the color for both object display and material to white.



- 6 From the **Render** menu, click **Render Preview** to see the material color.

### To place the eye on the head:

- 1 In the **Top** viewport, select both parts of the eye.
- 2 From the **Edit** menu, click **Groups**, then click **Group**.  
The eye parts are grouped as one object.
- 3 Select the group.
- 4 From the **Transform** menu, click **Orient**, then click **On Surface**.
- 5 For **Reference point 1**, snap to the point at the center of the eye.
- 6 For **Reference point 2**, snap to the point at the edge of the eye.
- 7 For **Surface to Orient on**, pick the head.

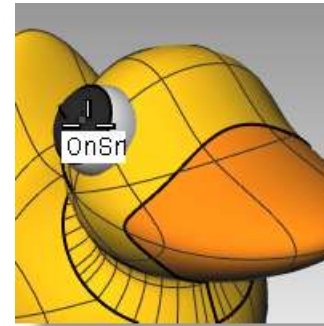


### Notes:

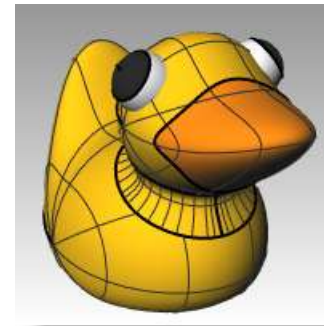
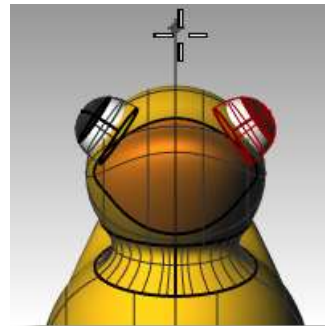


Object Properties

- 8 In the **Orient on Surface** dialog, check **Uniform** and **Rigid**, click **OK**.
- 9 For the **Point of surface to orient to**, pick a point on the head.



- 10 **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**.

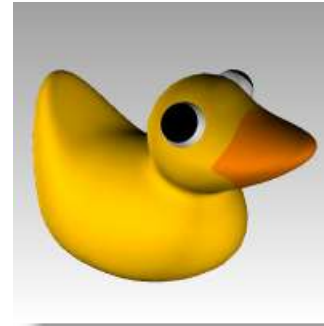
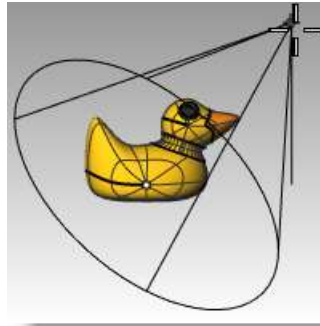
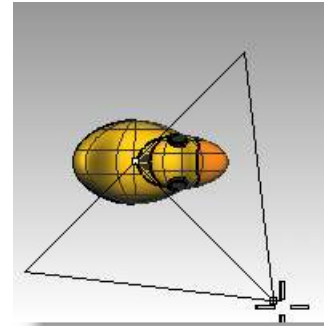
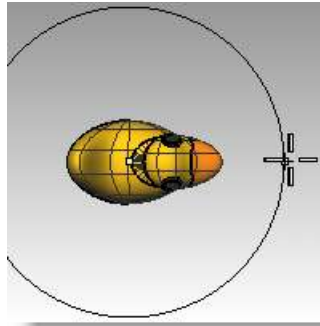
### Notes:





### To place lights:

- 1 From the **Render** menu, click **Create Spotlight**.
- 2 Select a point in the middle of the model.
- 3 Drag the radius until it is approximately three times as large as the model.
- 4 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:



Create Spotlight

---

**Notes:**

## 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 deformable by using the new UDT (Uniform Deformation Technology) tools. You can also extract surfaces and deform the surfaces with control point editing like the last exercise.

In this part of the class we will focus on making some solids, separating the parts, making changes and then joining the parts back together to make a solid.

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 2Point	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.
	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.

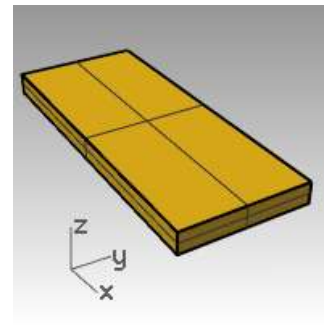
## Notes:

Button	Command	Description
	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.
	ExtrudeCrv	Draw a solid by extruding a closed planar curve vertically.
	ExtrudeSrf	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 55— 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.

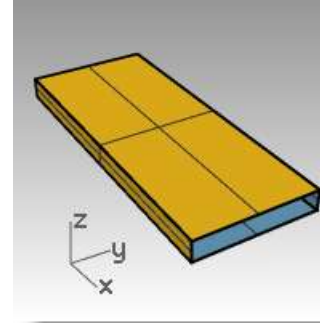
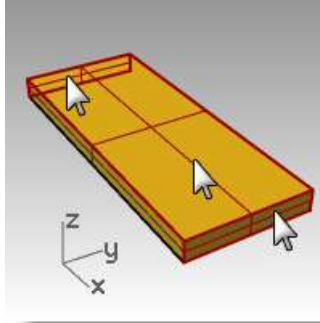
- 1 Start a new model using the **Small Objects - Millimeters** template. **Save** as **Bar**.
- 2 From the **Solid** menu, click **Box**, and then click **Corner to Corner, Height**.
- 3 For the **First corner** type **0,0** and press **Enter**.
- 4 For the **Length** type **15** and press **Enter**.
- 5 For the **Width** type **6** and press **Enter**.
- 6 For the **Height** type **1** and press **Enter**.



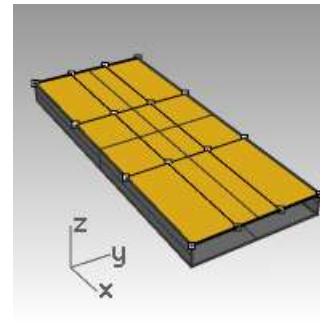
Box

**To edit a surface:**

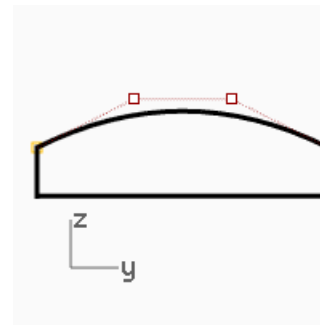
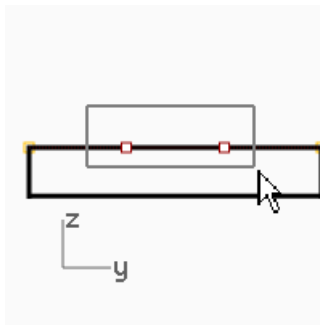
- 1 From the **Solid** menu, click **Extract Surface**.
- 2 For the **Surface to extract**, select the top and both end surfaces, press **Enter**.
- 3 Select the two ends and delete them.



- 4 Select the top surface.
- 5 From the **Edit** menu, click **Rebuild**.
- 6 In the **Rebuild Surface** dialog box, set the **Point count** to **4** and the **Degree** to **3** for both **U** and **V**, click **OK**.
- 7 Press **F10** to turn **Control Points On**.



- 8 In the **Right** viewport, window select the middle points.
- 9 Drag the points up approximately one unit.
- 10 Press **F11** to turn **Control Points Off**.



**Notes:**



Extract Surface



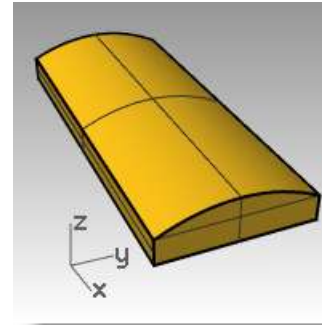
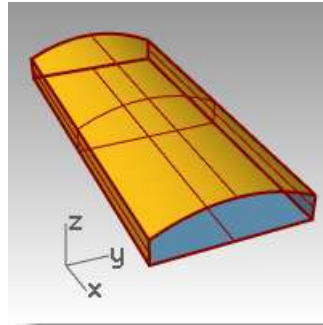
Rebuild Surface



Control Points On

### To make the bar solid:

- 1 Select all of the surfaces.
- 2 From the **Edit** menu, click **Join**.  
The surfaces are joined making an open polysurface.
- 3 Select the polysurface.
- 4 From the **Solid** menu, click **Cap Planar Holes**.  
Two end caps are created.



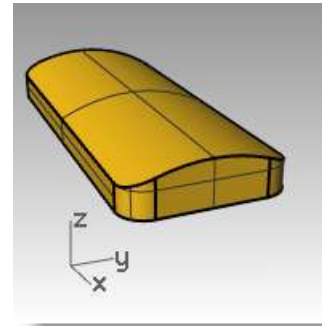
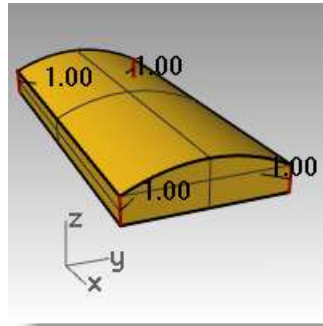
### Notes:



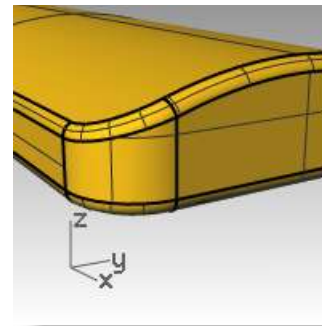
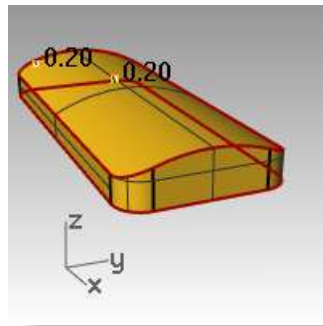
Cap Planar Holes

### To fillet the edges:

- 1 From the **Solid** menu, click **Fillet Edge**, and then click **Fillet Edge**.
  - 2 Set the **Current Radius=1.0**.
  - 3 For **Select edges to fillet**, pick the four vertical edges, press **Enter**.
  - 4 For **Select fillet handle to edit**, press **Enter**.
- 
- 5 Repeat the **Fillet Edge** command.
  - 6 Set the **Current Radius=0.2**.
  - 7 For **Select edges to fillet**, window select the entire bar to get the horizontal edges, press **Enter**.
  - 8 For **Select fillet handle to edit**, press **Enter**.



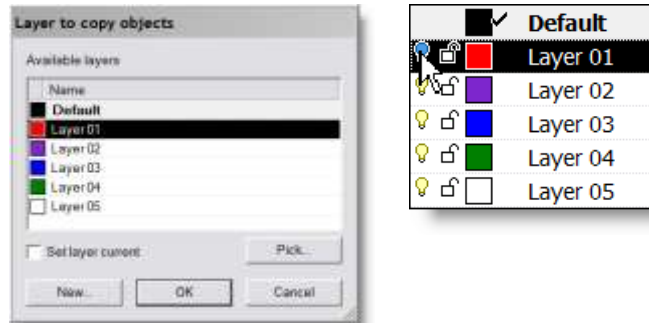
Fillet Edge



### To make a copy of the bar on a different layer:

We need to make a copy of the finished bar for the next part of this exercise. On one of the copies we will engrave text, on the other we will emboss text.

- 1 Select the finished bar.
- 2 From the **Edit** menu, click **Layers**, then **Copy objects to layer**.
- 3 In the **Select layer to copy objects** dialog, pick **Layer 01** and click **OK**.
- 4 Turn **Layer 01** off.



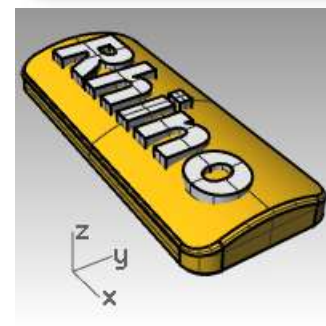
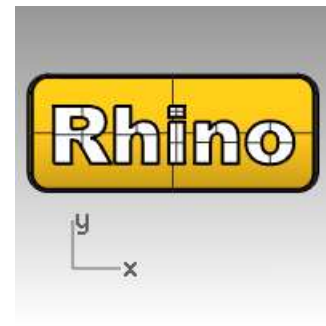
### Notes:



Copy Objects to Layer

### To make solid text:

- 1 Change the layer to **Layer 02**.
- 2 From the **Solid** menu, click **Text**.
- 3 In the **Text Object** dialog box, select a bold **Font**, like Arial Black.  
Under **Create** click **Solids**.  
Under **Text size**, set the **Height** to **3.00**, the **Solid thickness** to **1.00**, check **Group objects**, click **OK**.
- 4 For the **Insertion point** position the text in the center of the bar in the **Top** viewport and click.
- 5 In the **Front** or **Right** viewport, drag the text until it protrudes through the top surface.

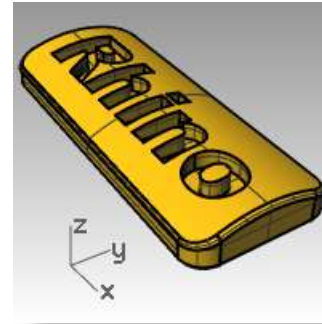


Text Object

### To engrave the text in the bar:

- 1 Select the bar.
- 2 From the **Solid** menu, click **Difference**.
- 3 For **Select second set of surfaces or polysurfaces**, set **DeleteInput=Yes**, select the text, press **Enter**.

The text is engraved into the bar.



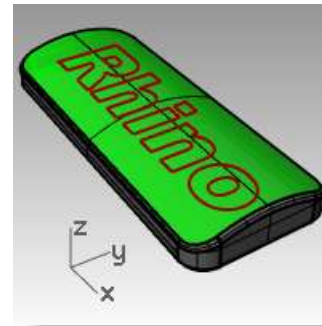
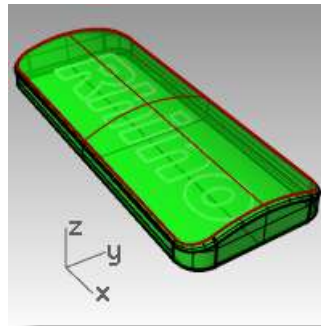
### Notes:

### Using a model for illustration

Occasionally you will want to use your 3-D model for illustration purposes. In this case we're going to divide the top surface into pieces. Each piece can then be assigned a material property. When the part is rendered or displayed in a rendered viewport, it will appear like a label.

### To make a label:

- 1 Turn **Layer 01** on and turn the **Default** layer off.
- 2 From the **Solid** menu, click **Extract Surface**.
- 3 Select the top surface, press **Enter**.
- 4 Select the lower part of the bar and **Lock** it.
- 5 Make the **Top** viewport active.
- 6 From the **Solid** menu, click **Text**.
- 7 In the **Text Object** dialog box, under **Create**, click **Curves**, click **OK**.
- 8 For the **Insertion point** position the text in the center of the bar in the **Top** viewport and click.





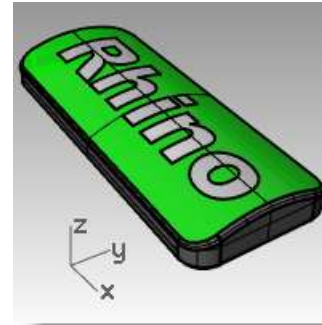
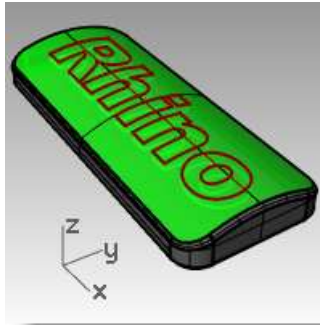
### To split the top surface of the bar with the text:

- 1 In the **Top** or **Perspective** viewport, select the extracted top surface.
- 2 From the **Edit** menu, click **Split**.
- 3 Select the text curves, press **Enter**.

Since the Group objects box was checked when you created the text, you can pick all of the text by clicking on one element.

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

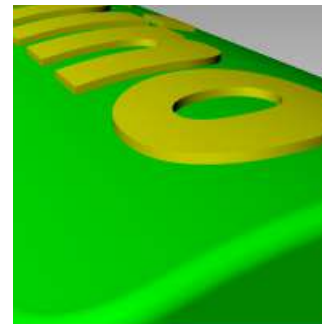
- 4 **Hide** the original text curves.
- 5 Select the text surfaces.  
Remember to leave the center of letters like R and O out of the selection set.
- 6 From the **Edit** menu, click **Group**, and then click **Group**.  
The text surfaces are now grouped for easier selection.
- 7 Select the group.
- 8 In the **Properties** window, on the **Material** page, click **Basic** and select another color for the text.
- 9 Right click on the **Perspective** viewport title.
- 10 Select **Rendered** from the popup menu.  
The letters render in a different color.



### To emboss the text in the bar:

- 1 From the **Edit** menu, click **Select Objects**, and then click **Previous Selection**.
- 2 From the **Solid** menu, click **Extrude Surface**, and then click **Straight**.
- 3 For the **Extrusion distance**, click **Bothsides=Yes**, **Cap=Yes**, and **DeleteInput=Yes** on the command line.
- 4 For the **Extrusion distance**, type **.1** and press **Enter**.

The surfaces are extruded perpendicular to the active construction plane by .1 in both directions. Notice that the top of the text follows the curvature of the original surface and the bottom of the text penetrates into the bar.



### Notes:



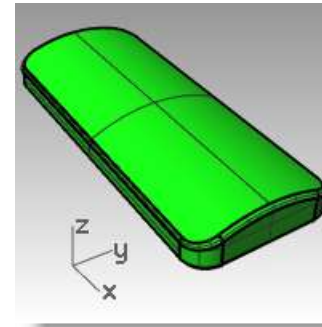
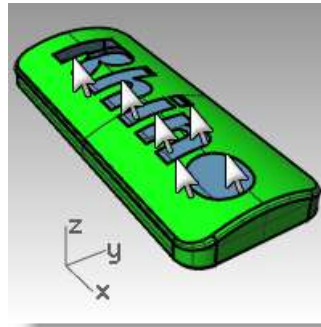
Extrude Surface

**5 Hide** the extruded text.

Don't forget to delete the center of letters.

**6** From the **Surface** menu, click **Surface Edit Tools**, click **Untrim**.

**7** For **Select an edge to untrim**, pick on the edge of the text openings.



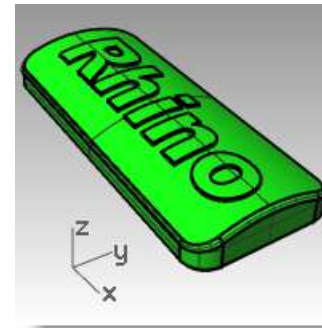
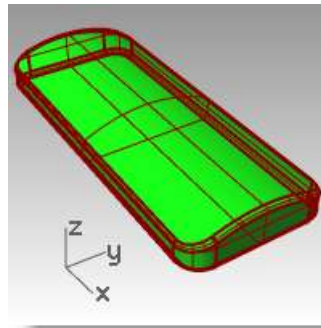
**8 Unlock** the bottom part of the bar and **Join** it with top part.

**9 Show** the text.

**10** Select the bar and the text.

**11** From the **Solid** menu, click **Union**.

The text and the bar are joined into one closed polysurface.



### Notes:



Untrim



Boolean Union











## 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.

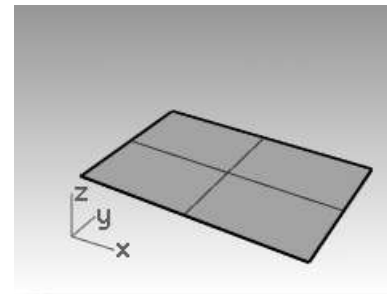
Button	Command	Description
	SrfPt	Creates a surface by picking three or four points in space for corners.
	EdgeSrf	Creates a surface by selecting two, three, or four existing curves whose ends meet exactly.
	PlanarSrf	Creates a surface from planar curves that enclose an area.
	Patch	Creates a surface 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.
	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

Button	Command	Description
	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.
	ExtrudeCrv	Extrudes a curve perpendicular to the construction plane with option to taper the surface with a draft angle.
	ExtrudeCrvAlongCurve	Extrudes a curve following along a second curve.
	ExtrudeCrvToPoint	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 56—Basic techniques for making surfaces

In this exercise, you will model some simple surfaces.

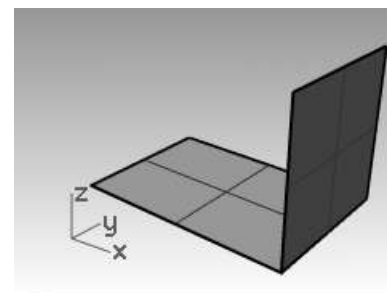
- 1 Start a new model, use the **Small Objects - Millimeters.3dm** template. Save as **Surfaces**.
- 2 Turn on **Snap** and **Planar**.
- 3 From the **Surface** menu, click **Plane**, and then click **Corner to Corner**.
- 4 For the **First corner of plane**, pick a point.
- 5 For the **Other corner**, pick another point to make a rectangular plane.



Plane: Corner to Corner

#### To create a vertical plane:

- 1 From the **Surface** menu, click **Plane**, and then click **Vertical**.
- 2 For the **Start of edge**, snap to the endpoint at the right side of the surface.
- 3 For the **End of edge**, snap to the other endpoint at the right side of the surface.
- 4 Drag your cursor up and pick.

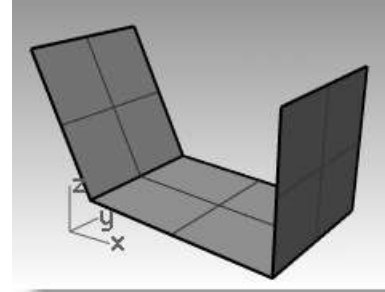
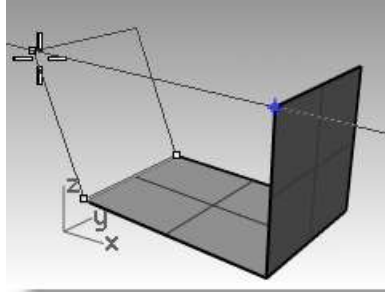


Vertical Plane

### Notes:

### To create a plane from 3 points:

- 1 From the **Surface** menu, click **Plane**, and then click **3 Points**.
- 2 For the **Start of edge**, snap to the endpoint at the left of the first surface.
- 3 For the **End of edge**, snap to the other endpoint at the left side of the first surface.
- 4 For the **Height**, use **SmartTrack** to track a point from the top of the vertical plane. Drag the tracking point until the surface is tilted slightly and click.



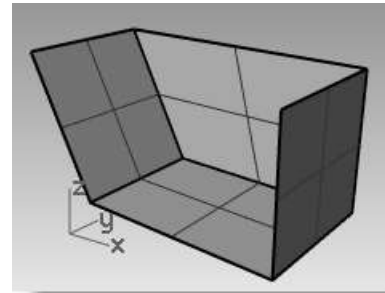
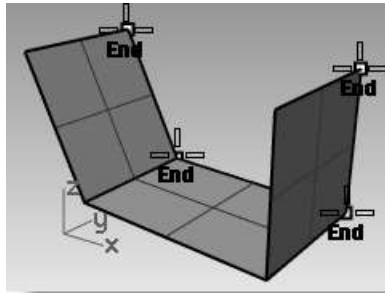
### Notes:



Rectangular Plane: 3 Points

### To create a plane from corner points:

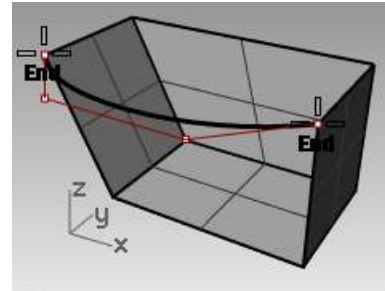
- 1 From the **Surface** menu, click **Corner Points**.
- 2 For the **First corner**, snap to an endpoint at the edge of the first surface.
- 3 For the **Second corner**, snap to the endpoint at the edge of the second surface.
- 4 For the **Third corner**, snap to an endpoint at the edge of the third surface.
- 5 For the **Fourth corner**, snap to the other endpoint at the edge of the third surface.  
A surface will be created with corners at the points you selected.



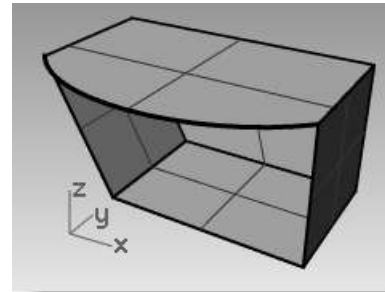
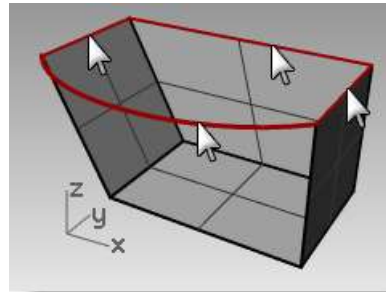
Surface from 3 or 4 Corner Points

### To create a surface from planar curves:

- 1 Turn **Planar** mode on.
- 2 Draw a curve 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.

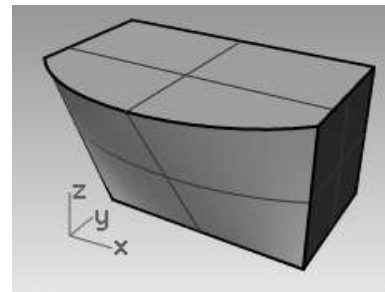
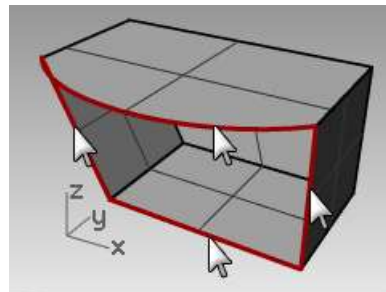


- 3 From the **Surface** menu, click **Planar Curves**.
- 4 Select the curve you just created.
- 5 Select the top edge of the three surfaces and press **Enter**.  
A surface is created.



### To create a surface from edge curves:

- 1 From the **Surface** menu, click **Edge Curves**.
- 2 Select the four surface edges.  
A surface is created.



### Notes:



Surface from Planar Curves



Surface from 2, 3, or 4 Edge Curves

## Exercise 57—Extruding surfaces

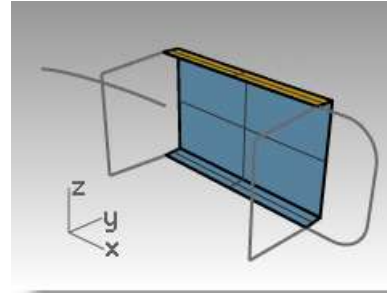
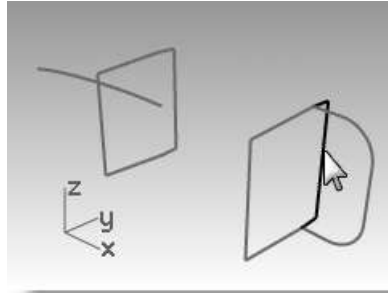
In this exercise, you will be creating a cordless phone using extrusions. To aid in organizing the model, surface and curve layers have been created. Make sure you change layers as you are making the extrusions.



### To extrude a curve:

- 1 **Open** the model **Extrude.3dm**.
- 2 Make the **Top Surface** layer current.
- 3 Select the curve as shown.
- 4 From the **Surface** menu, click **Extrude Curve**, and then click **Straight**.
- 5 For the **Extrusion distance**, 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.



### Notes:

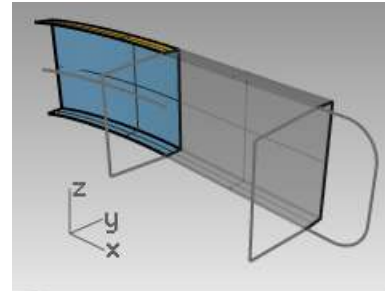
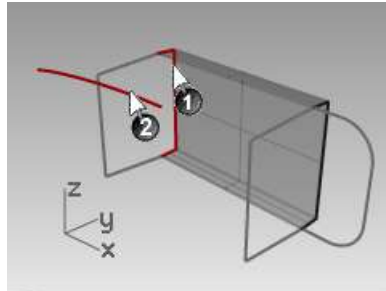


Extrude Straight

### To extrude a curve along another curve:

- 1 Select the curve **(1)** on the left of the first extruded surface.
- 2 From the **Surface** menu, click **Extrude Curve**, and then click **Along Curve**.
- 3 Select the path curve **(2)** near its right end.

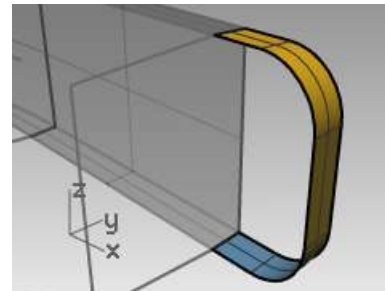
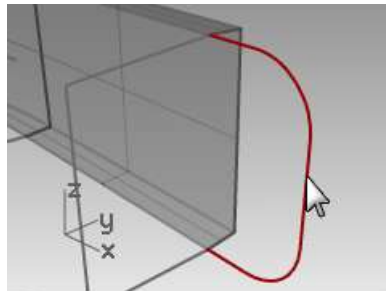
The curve is extruded along the path of the secondary curve.



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

- 1 Select the curve on the right.
- 2 From the **Surface** menu, click **Extrude Curve**, and then click **Tapered**.
- 3 For the **Extrusion distance**, click **DraftAngle** on the command line.
- 4 For the **Draft angle**, type **-3** and press **Enter**.
- 5 For the **Extrusion distance**, type **.375** and press **Enter**.

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



### Notes:



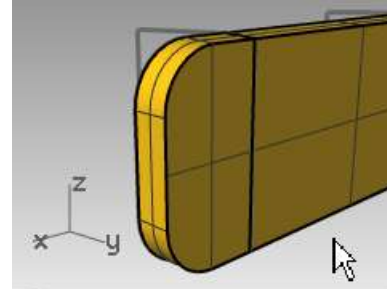
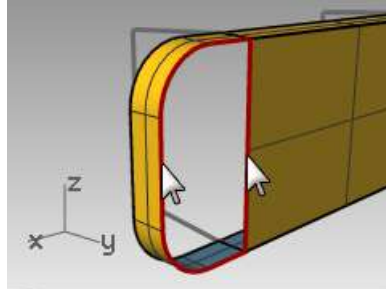
Extrude Along Curve

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

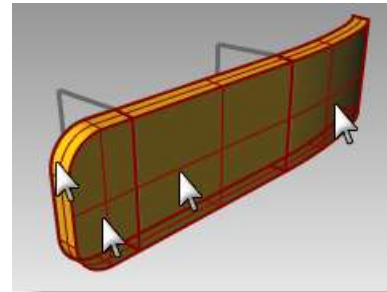


### To create a surface from planar curves:

- 1 From the **Surface** menu, click **Planar Curves**.
- 2 Select the edge curves that bound the openings of the tapered extrusion at the top.
- 3 Press **Enter**.  
A surface is created at the end.



- 4 Select the four surfaces.
- 5 From the **Edit** menu, click **Join**.

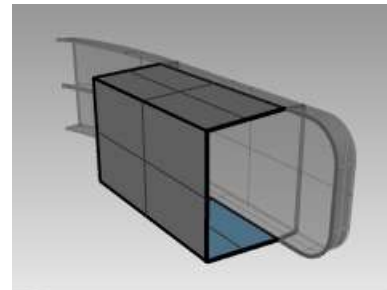
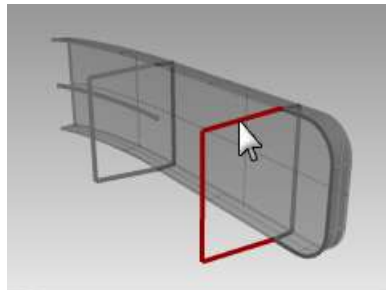


### To make the extruded surfaces for the other half of the phone:

Next, we will repeat the previous steps for the other half of the phone.

- 1 Make the **Bottom Surface** layer current.
- 2 Select the curve as shown.
- 3 From the **Surface** menu, click **Extrude Curve**, and then click **Straight**.
- 4 For the **Extrusion distance**, 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.



### Notes:

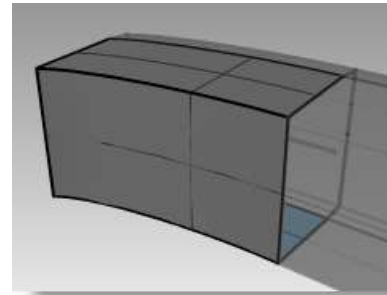
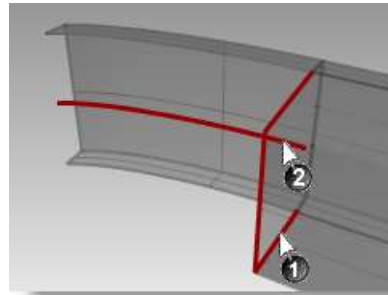


Surface from Planar Curves

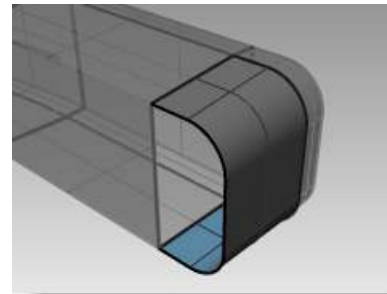
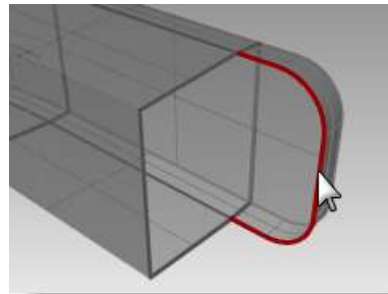


Extrude Straight

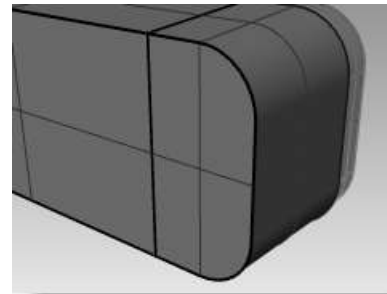
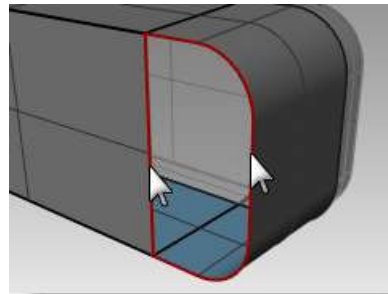
- 5 Select the curve **(1)** on the left of the first extruded surface.
- 6 From the **Surface** menu, click **Extrude Curve**, and then click **Along Curve**.
- 7 Select the path curve **(2)** near its right end.  
The curve is extruded along the path of the secondary curve.



- 8 Select the curve on the right.
- 9 From the **Surface** menu, click **Extrude Curve**, and then click **Tapered**.
- 10 For the **Extrusion distance**, type **-1.375** and press **Enter**.  
The curve is extruded with a three-degree draft angle in the negative direction on the y-axis.



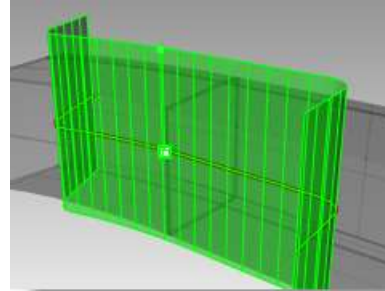
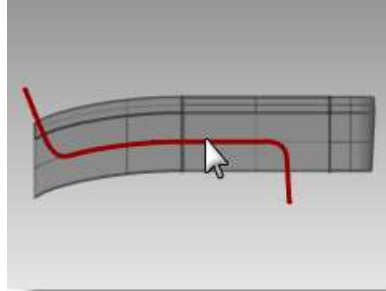
- 11 From the **Surface** menu, click **Planar Curves**.
- 12 Select the edge curves that bound the openings of the tapered extrusion at the top.
- 13 Press **Enter**.  
A surface is created at the end.
- 14 Select the four surfaces.
- 15 From the **Edit** menu, click **Join**.



**Notes:**

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

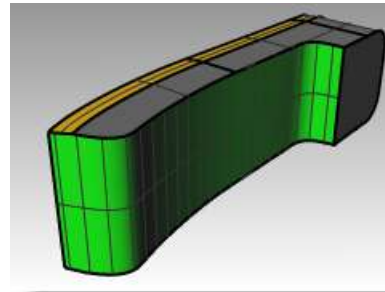
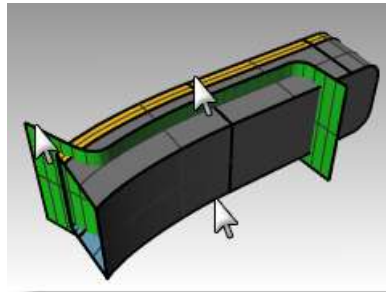
- 1 Turn on the **Extrude Straight-both sides** layer.
- 2 Select the freeform curve as shown.
- 3 From the **Surface** menu, click **Extrude Curve**, and then click **Straight**.
- 4 For the **Extrusion distance**, click **Bothsides** on the command line.
- 5 For the **Extrusion distance**, drag the extrusion and pick.



Make sure the surface extends past the other surfaces in both directions.  
The curve is extruded symmetrically from the curve.

**To trim the surfaces:**

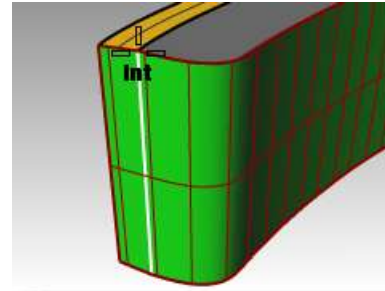
- 1 Select the joined Top and Bottom polysurfaces, and the surface you just extruded.
- 2 From the **Edit** menu, click **Trim**.
- 3 For the **Object to trim**, pick the outer edge of each surface.



**Notes:**

### To split the trimmed surface:

- 1 Select the trimmed extruded surface.
- 2 From the **Edit** menu, click **Split**.
- 3 For **Select cutting objects**, click **Isocurve** on the command line.
- 4 Drag your cursor along the surface to determine which isocurve direction is selected. Click **Toggle** on the command line if it's not the correct direction.
- 5 For the **Split point**, snap to the intersection where all three surfaces intersect.

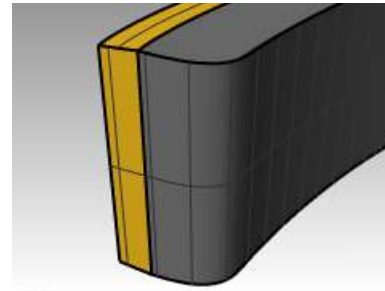


### Notes:

### To join the surfaces:

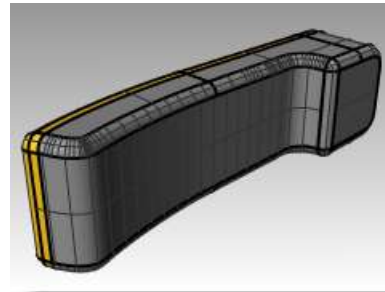
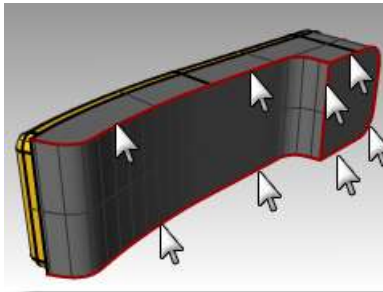
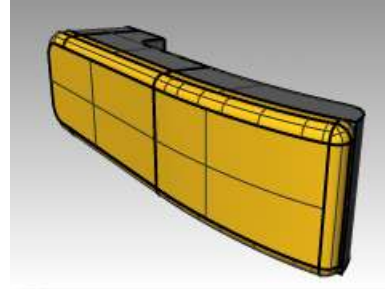
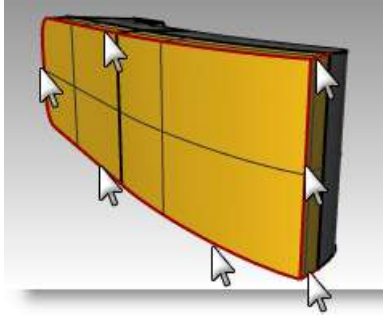
Next you will join the surface and the polysurface for the Top and the Bottom of the handset.

- 1 Select the upper part of the split surface and the top polysurface
- 2 From the **Edit** menu, click **Join**.
- 3 Select the lower part of the split surface and the bottom polysurface
- 4 From the **Edit** menu, click **Join**.



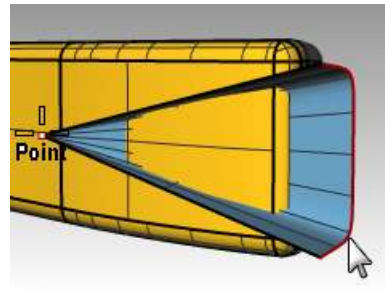
### To round the edges of the polysurfaces:

- 1 From the **Solid** menu, **Fillet Edges**, then **Fillet Edges**. Use a radius of **.2** to round the edges.
- 2 For the **Select edges to fillet**, select the edges around the top of the top polysurface and the two horizontal edges at the front, press **Enter**.
- 3 For the **Select fillet handle to edit**, click **Preview**.
- 4 Check to make sure the fillet is doing what you expect, press **Enter**.
- 5 Repeat these steps for the Bottom polysurface.



### 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 Curve**, and then click **To Point**.
- 4 For the **Point to extrude to**, snap to the point object near the top surface.  
The curve is extruded to the point.



### Notes:



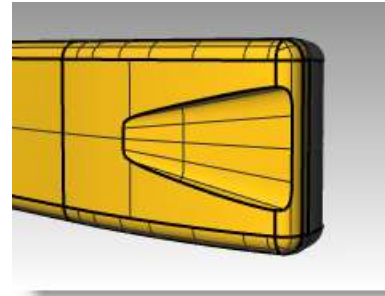
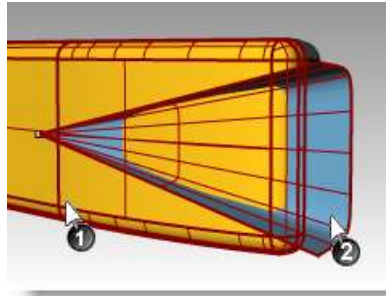
Variable Radius Fillet



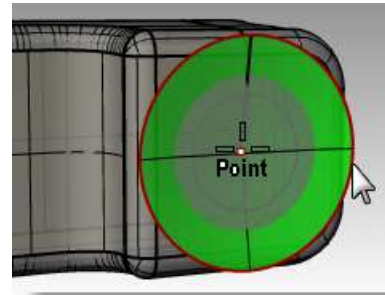
Extrude To Point

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

If the result is not correct, flip the normal of the top polysurface and the extruded surface with the **Dir** command. The normal of the top polysurface and the extruded surface should point toward each other.



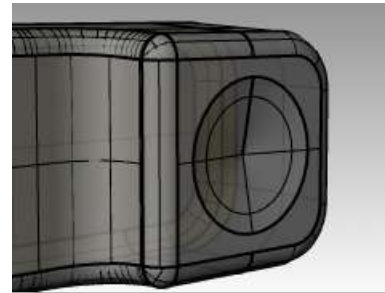
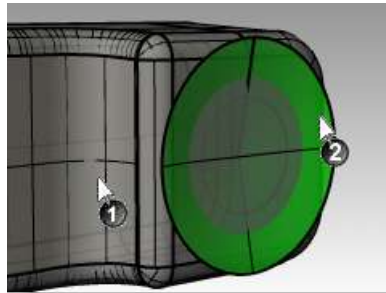
- 6** Select the circle.
- 7** From the **Surface** menu, click **Extrude Curve**, and then click **To Point**.
- 8** For the **Point to extrude to**, snap to the point object inside the bottom surface.
- The curve is extruded to the point.



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

The normal of the bottom polysurface and the extruded surface should point toward each other.

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

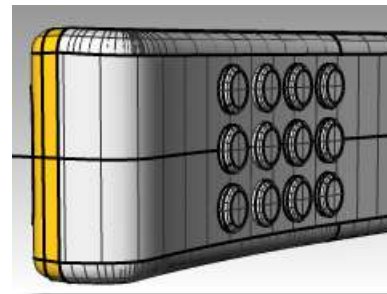
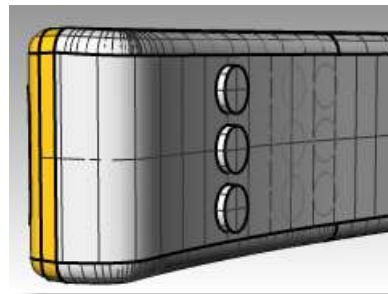
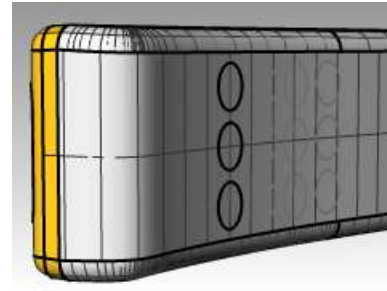


**Notes:**

**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 Planar Curve**, and then click **Straight**.
- 4 Type **-.2** and press **Enter**.

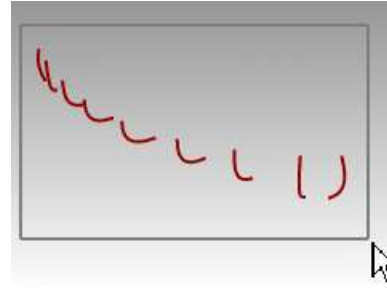
- 5 Repeat these steps for the other columns of buttons.
- 6 Use the **FilletEdge** command (*Solid menu > Fillet Edge > Fillet Edge*) with a radius of **.05** to round the edges.  
The button edges are rounded.
- 7 **Save** your model.



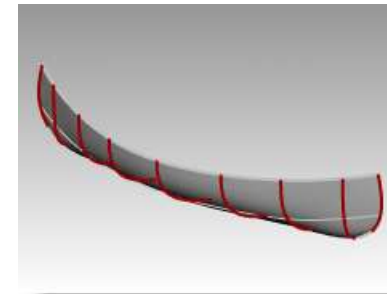
**Notes:**

### Exercise 58—Lofted surfaces

- 1 **Open** the model **Loft.3dm**.
- 2 Window select all of the curves.

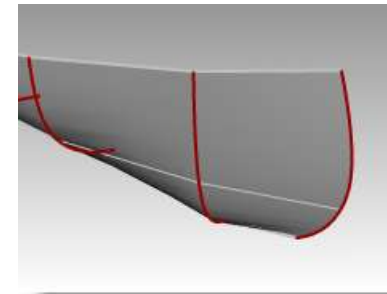


- 3 From the **Surface** menu, click **Loft**.  
A surface is fitted over the curves.



- 4 In the **Loft Options** dialog box, switch **Style** to **Straight sections**, and then click **Preview**.

A surface is fitted through the curves, but the sections are straight between the curves.



**Notes:**



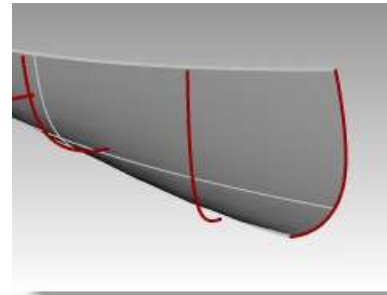
Loft



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

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

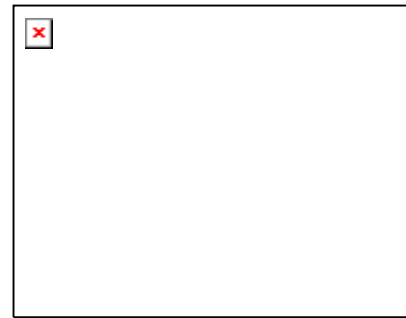
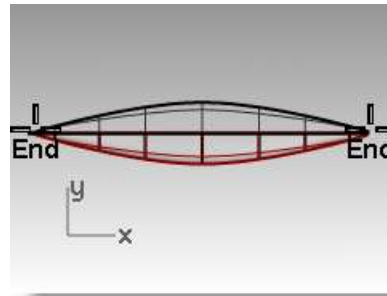
Use this option when you want the surface to conform to the control points of the input curve.



- In the **Loft Options** dialog box, switch **Style** to **Normal**, and then click **OK**.

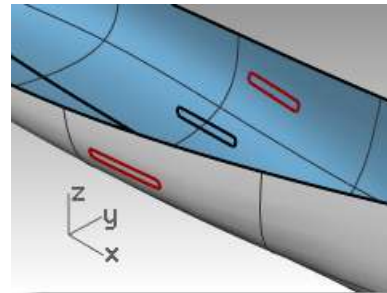
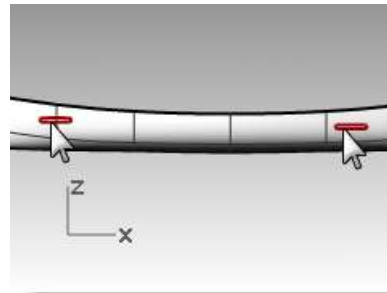
- Mirror** the surface to create the other half.

- Join** the two halves.



#### To make a seat:

- Turn the **Hull Curves** layer off and turn the **Seat Curves**, and **Seat** layers on.
- Make the **Seat Curves** layer current.
- In the **Front** view, select the rounded rectangles.
- From the **Curve** menu, click **Curve From Objects**, and then click **Project**.



#### Notes:



Mirror



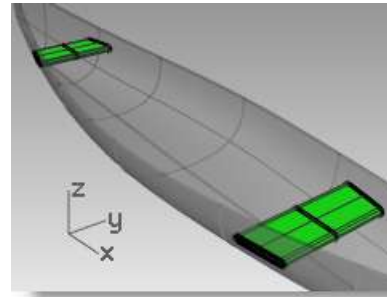
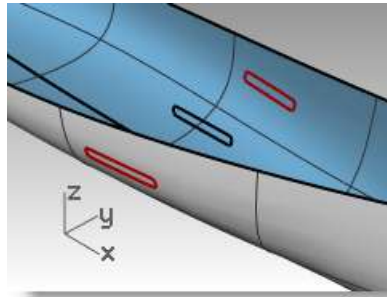
Join



Project to Surface

Project uses the current construction plane to determine the direction of projection. Make sure you select the curves and surface to project onto in the Front viewport.

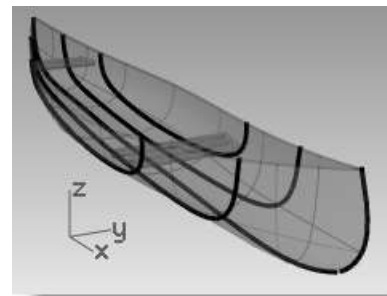
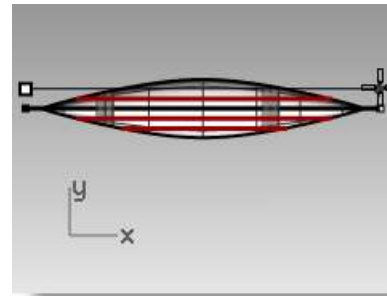
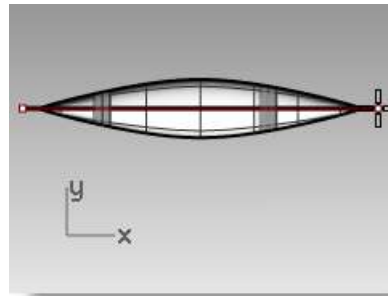
- 5 For the **Surface to project onto**, select the hull.  
The curves will be projected to both sides of the hull surface.
- 6 Select the curves on the surface.
- 7 From the **Surface** menu, click **Loft**.
- 8 In the **Loft** dialog, click **OK**.



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 **Sections** layer.
- 3 From the **Curve** menu, click **Curve From Objects**, and then click **Section**.
- 4 For the **Start of section**, in the **Top** viewport, pick a point to the left at the center of the hull.
- 5 For the **End of section**, with **Ortho** on, drag a line to the right and pick. A curve is generated on the surface. Repeat this at various locations.



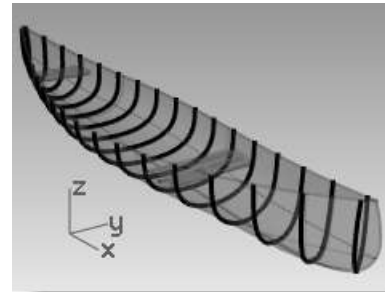
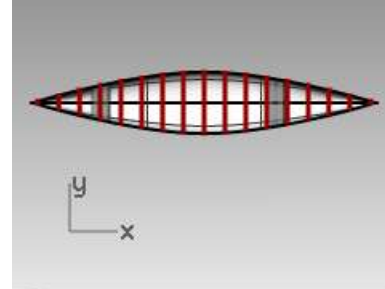
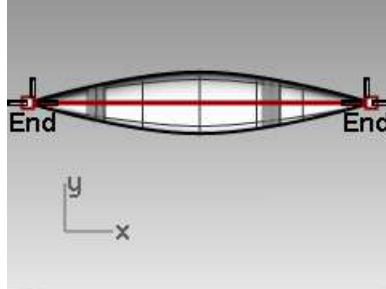
#### Notes:



Section

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

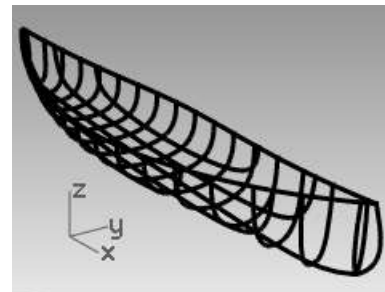
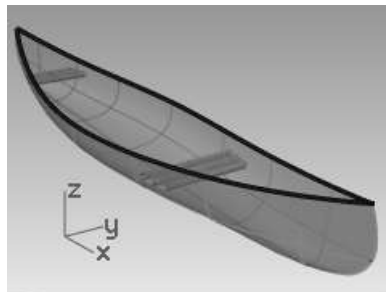
- 1 Select the hull.
- 2 Change to the **Contours** layer.
- 3 From the **Curve** menu, click **Curve From Objects**, and then click **Contour**.
- 4 For the **Contour base point**, snap to the left end of the canoe.
- 5 For the **Direction perpendicular to contour planes**, snap to the other end of the canoe.
- 6 For the **Distance between contours**, type **12** and press **Enter**.  
A curve is generated every foot along the hull.



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

- 1 Change to the **Top Rail** layer.
- 2 From the **Curve** menu, click **Curve From Objects**, and then click **Duplicate Edge**.
- 3 Pick the top edge of the hull.
- 4 Pick the other top edge and press **Enter**.

Two curves are generated at the edges of the hull



### Notes:



Contour

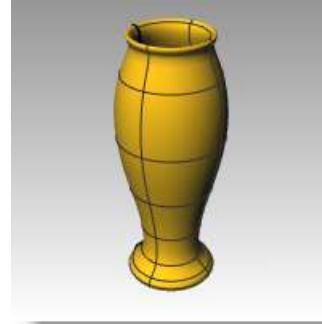
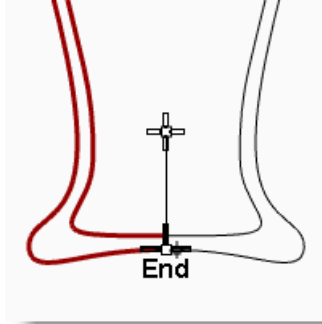
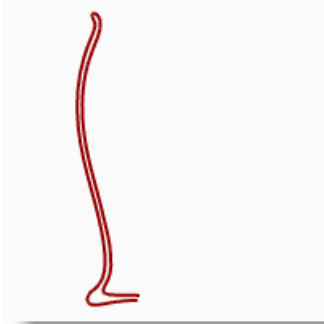


Duplicate Edge

### Exercise 59—Revolved surfaces

- 1 **Open** the model **Revolve.3dm**.
- 2 **Select** the free-form curve
- 3 From the **Surface** menu, click **Revolve**.
- 4 **Select** one end of the curve for the **Start of revolve axis**.
- 5 **Select** the other end of the curve for the **End of revolve axis**.
- 6 Press **Enter** to use the default **Start Angle**.
- 7 Press **Enter** to use the default **Revolution Angle**.

A surface is revolved around the axis line.



### Notes:



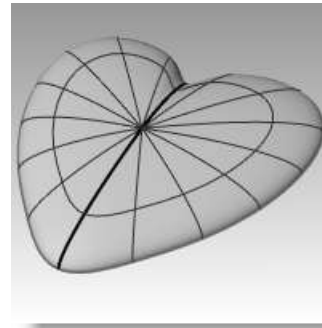
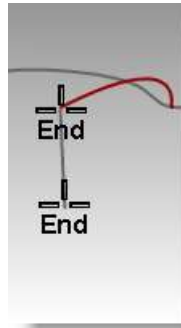
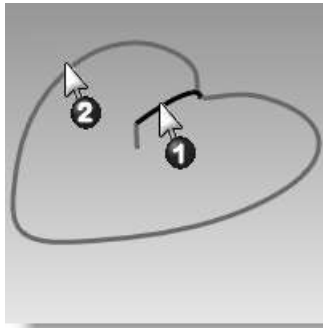
Revolve

### Exercise 60—Using a rail revolve

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

#### To create a rail revolve:

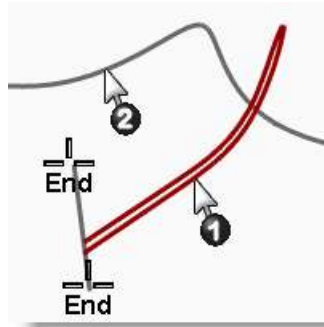
- 1 **Open** the model **Rail Revolve.3dm**.
- 2 For the **Profile Curve**, select the conic shaped curve.
- 3 From the **Surface** menu, click **Rail Revolve**.
- 4 For the **Rail curve**, select the heart shaped curve.
- 5 Select one end of the **RailRevolve axis** line.



- 6 Select the other end of the **RailRevolve 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 Repeat the previous steps to make a bowl.



#### Notes:

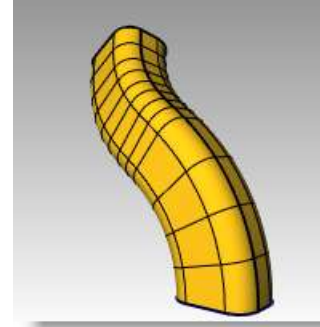
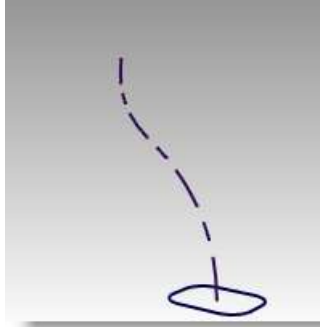


Rail Revolve  
Right click this button

## Exercise 61—Using 1-rail sweeps to create surfaces

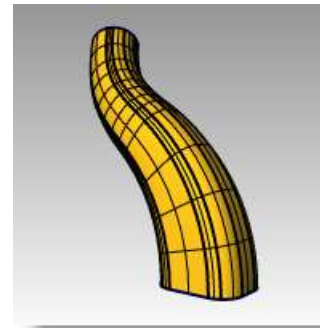
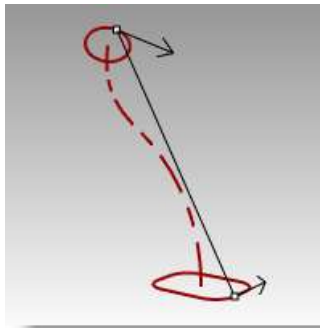
### One cross-section:

- 1 Open the model **1 Rail Sweep.3dm**.
- 2 Select the two curves on the left.
- 3 From the **Surface** menu, click **Sweep 1 Rail**.
- 4 In the **Sweep 1 Rail Options** dialog, click **OK**.



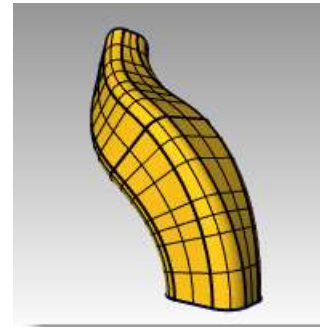
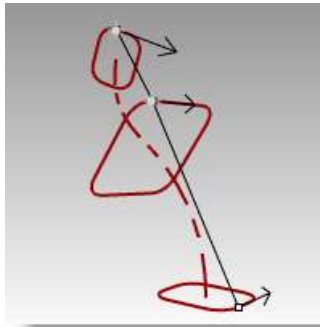
### Two cross-sections:

- 1 Select the three curves in middle.
- 2 From the **Surface** menu, click **Sweep 1 Rail**.
- 3 In the **Sweep 1 Rail Options** dialog, check **Global shape blending**.
- 4 In the **Sweep 1 Rail Options** dialog, click **OK**.



### Multiple cross-sections:

- 1 Select the four curves on the right.
- 2 From the **Surface** menu, click **Sweep 1 Rail**.
- 3 In the **Sweep 1 Rail Options** dialog, uncheck **Global shape blending**.
- 4 In the **Sweep 1 Rail Options** dialog, click **OK**.



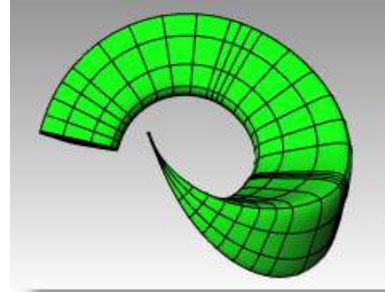
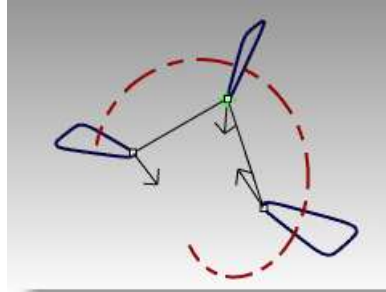
### Notes:



Sweep 1 Rails

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

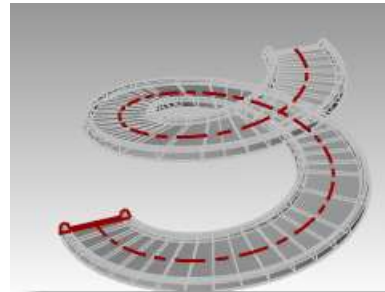
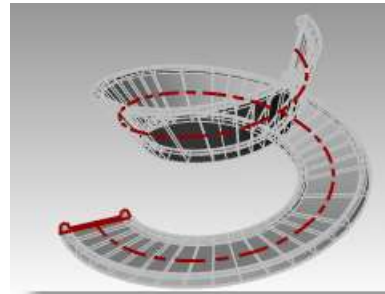
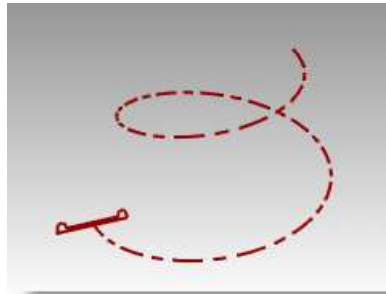
- 1 Make **Surface 02** layer group current and turn off **Surface 01** layer group.
- 2 From the **Surface** menu, click **Sweep 1 Rail**.
- 3 For the **Rail**, select the open free-form curve.
- 4 For the **Select cross section curves**, select the three closed curves, then click **Point** on the command line.
- 5 For the **Pick end point**, snap to the **End** of the free-form curve.
- 6 In the **Sweep 1 Rail Options** dialog, click **OK**.



**Notes:**

**To create a roadlike 1-rail sweep:**

- 1 Make **Surface 03** layer group current and turn off **Surface 02** layer group.
- 2 Select the helix.
- 3 From the **Surface** menu, click **Sweep 1 Rail**.
- 4 For the **Select cross section curves**, select the closed curve, press **Enter**.
- 5 Change the style to **Roadlike Right**, click **Preview**.
- 6 Change the style to **Roadlike Front**, click **Preview**.
- 7 Change the style to **Roadlike Top**, click **Preview**, click **OK**.



## Exercise 62—Using 2-rail sweeps to create surfaces

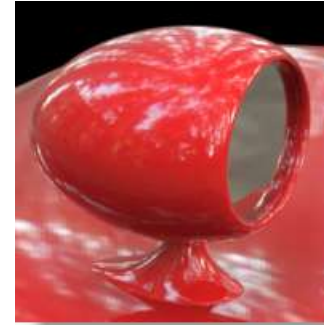
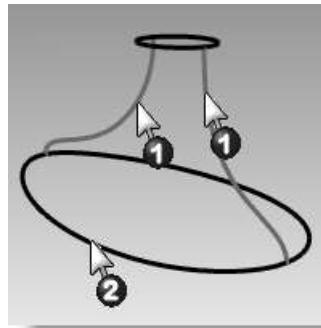
► **Open** the model **2 Rail Sweep.3dm**.

### To create the base —Part 1:

In the first part of this exercise, we will explore one of the Sweep2 options. To illustrate the option we will use one cross-section. In the second part we will use the same rail curves with two cross-sections. Lastly, we will use two rails that converge to a single point.

- 1 Change to the **Base Surface** layer.
- 2 From the **Surface** menu, click **Sweep 2 Rails**.
- 3 Select the two **rail curves (1)**.
- 4 Select the **cross-section curve (2)**.
- 5 Press **Enter** twice.

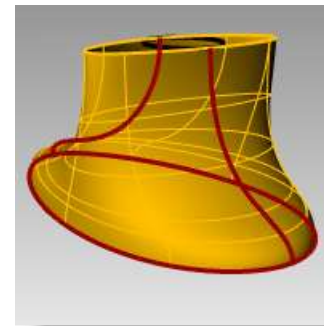
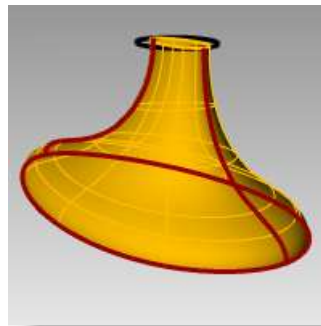
Since we only picked one cross-section, the surface doesn't conform to the circle at the top of the sweep.



- 6 In the **Sweep 2 Rail Options** dialog, check **Maintain height**, click **Preview**.

Notice that the cross-section maintains the same height throughout the sweep.

- 7 In the **Sweep 2 Rail Options** dialog, click **Cancel**.



## Notes:



Sweep 2 Rails

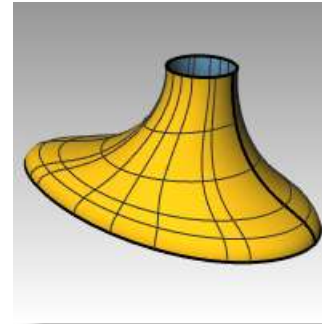


**Notes:**

**To create the base—Part 2:**

- 1 Select the two **rail curves** (1).
- 2 From the **Surface** menu, click **Sweep 2 Rails**.
- 3 Select both **cross-section curves** (2).
- 4 Press **Enter** twice.
- 5 In the **Sweep 2 Rail Options** dialog, click **OK**.

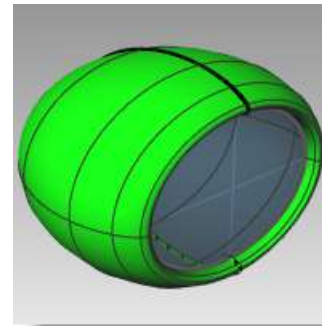
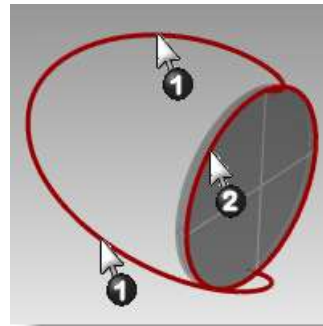
A surface is created whose edges match the rail curves and the cross-section curves.



**To create the housing:**

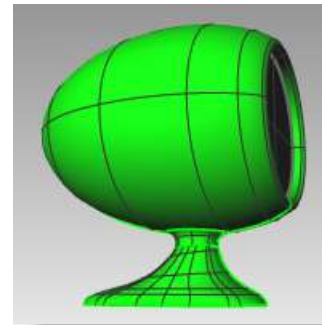
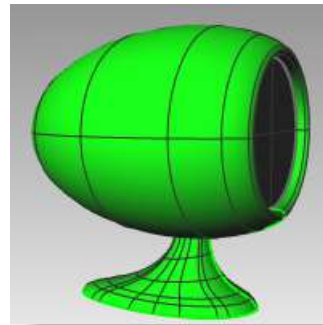
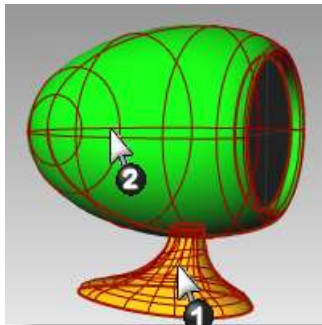
- 1 Turn the **Housing Surface**, **Housing Curves**, and **Mirror** layers on.
- 2 Make the **Housing Surface** layer current.
- 3 From the **Surface** menu, click **Sweep 2 Rails**.
- 4 Select the two **rail curves** (1).
- 5 For the **cross-section**, select the outer edge of the cylinder, press **Enter** twice.
- 6 In the **Sweep 2 Rail Options** dialog, click **OK**.

A surface is created.



**To join the two parts:**

- 1 **Select the base and the housing surfaces.**
- 2 From the **Solid** menu, click **Union**.
- 3 Use **FilletEdge** with a radius of **.25** to round the intersecting edge.



**Exercise 63—Using a network of curves to create surfaces**

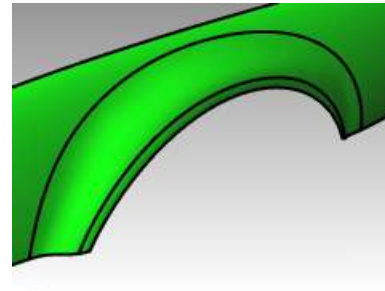
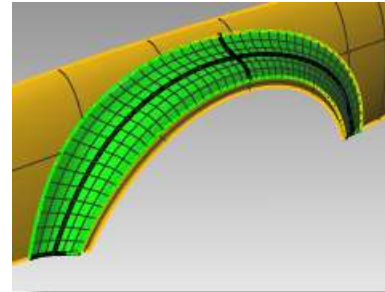
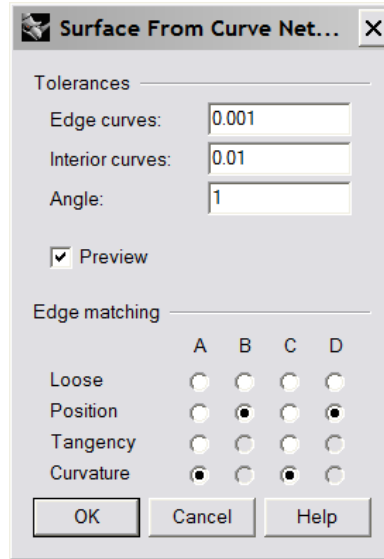
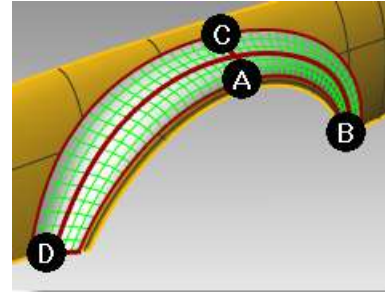
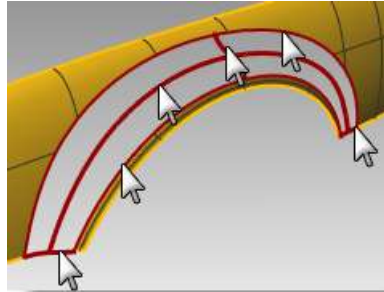
**1** Open the model **Networksurf.3dm**.

**2** From the **Surface** menu, click **Curve Network**.

**3** For **Select curves in network**, pick 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:**

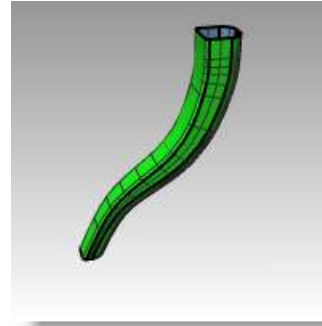
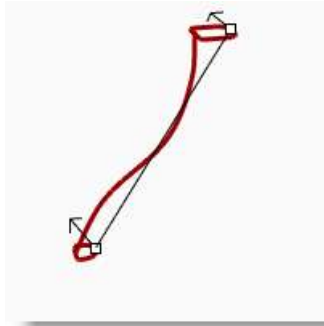
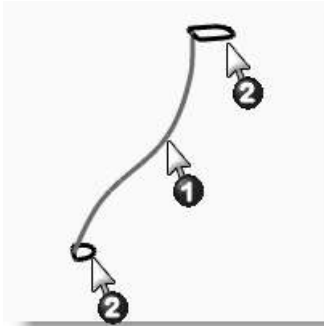
**Notes:**

**Exercise 64— 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:**

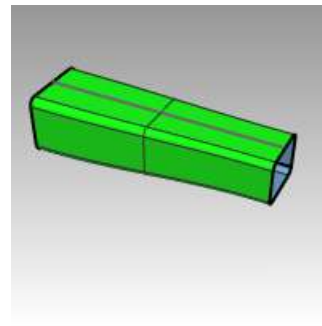
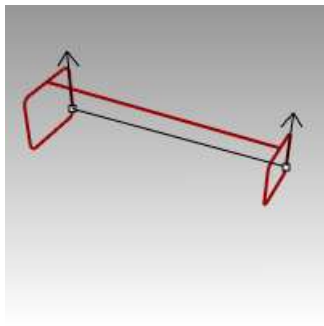
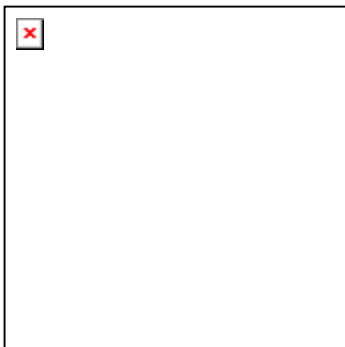
- 1 **Open** the model **Table.3dm**.
- 2 From the **Surface** menu, click **Sweep 1 Rail**.
- 3 Select the path curve for the leg.
- 4 Select the shape curve for both ends of the leg.
- 5 Press **Enter**.
- 6 Press **Enter**.
- 7 In the **Sweep 1 Rail Options** dialog box, click **OK**.



The table leg is created. Note the nice transition from one cross-section curve to the other.

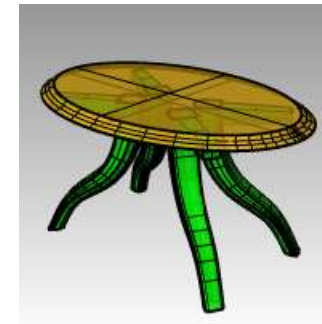
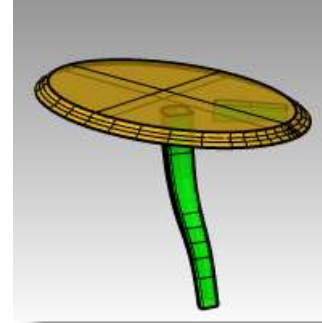
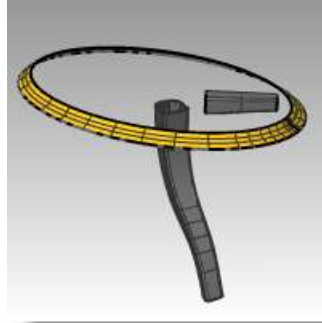
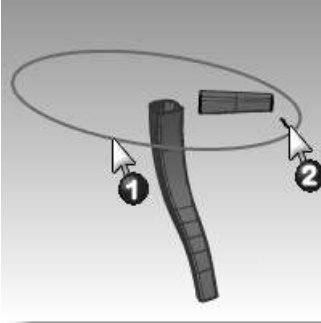
**To create the brace:**

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



**To create the top and finish the table:**

- 1** Change to the **Top** layer.
- 2** From the **Surface** menu, click **Sweep 1 Rail**.
- 3** Select the ellipse.
- 4** Select the shape curve.
- 5** Press **Enter**.
- 6** In the **Sweep 1 Rail Options** dialog box, click **OK**.  
The surface for the rim of top is created.
- 7** Select all of the surfaces you created.
- 8** From the **Solid** menu, click **Cap Planar Holes**.  
Six caps were created.
- 9** Use **Mirror** to copy the brace and the leg to finish the model.  
Mirror them around **0,0** in the **Top** viewport.



**Notes:**

### Exercise 65— Creating a toy hammer:

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

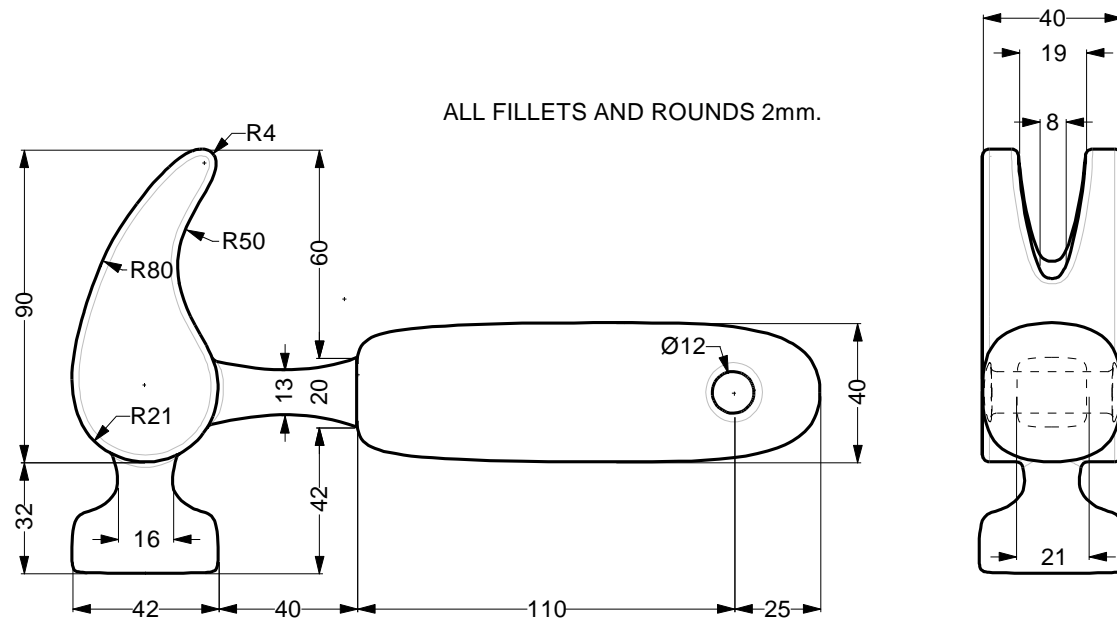
Some models require more attention to detail. This is an example of a model that requires precise modeling techniques. This exercise also requires a number of different surface creation techniques. The technical drawing is included to help you create a very precise model.



### Notes:

#### 1 Open the model **Hammer.3dm**.

In addition the following layers have been created: **Construction Lines**, **Curves**, **Handle**, **Tang**, **Head**, **Hole**, **Cutout**, and **Claw**. Use the appropriate layer when constructing the model.



#### 2 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.

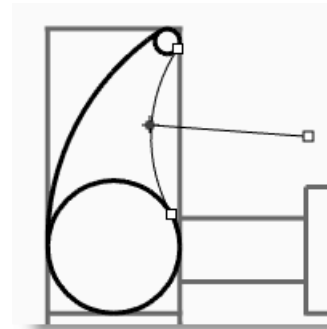
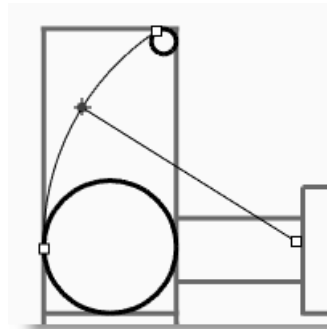
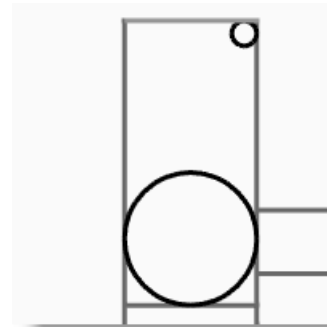
- 1 Change to the **Curves** layer
- 2 Draw a curve defining the shape of the claw in the **Top** viewport.  
You can use a free-form curve or use a combination of arcs and circles that are trimmed and joined to create the curve. Following is a systematic approach to creating the curve for the claw part 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.

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



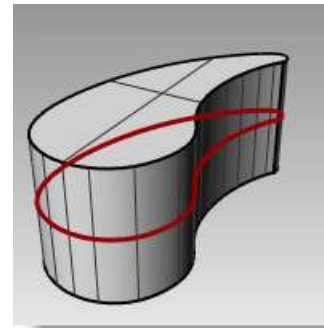
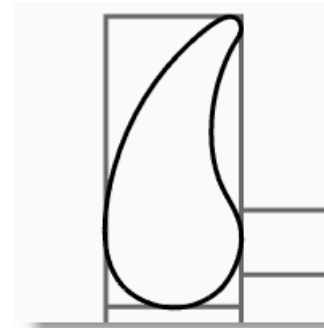
6 Use the **Trim** command (*Edit menu > Trim*) to trim the inside part of the circles.

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 **ExtrudeCrv** command (*Solid menu > Extrude Planar Curve > Straight*) to extrude the curve on both sides of the construction plane.



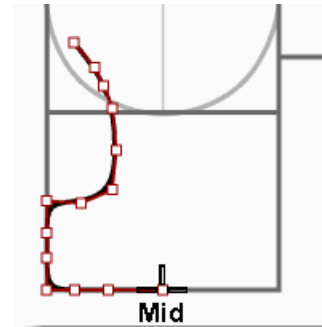
**Notes:**

## Notes:

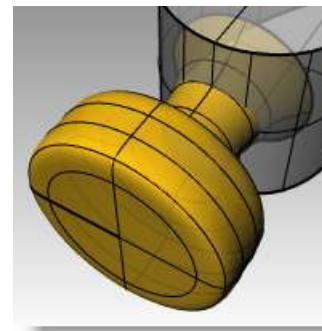
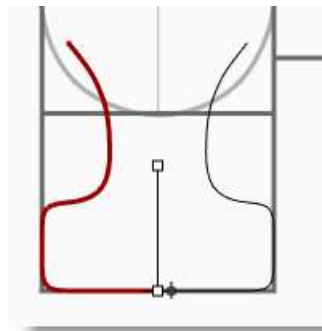
### To create the head:

- 1 Change to the **Curves** 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.

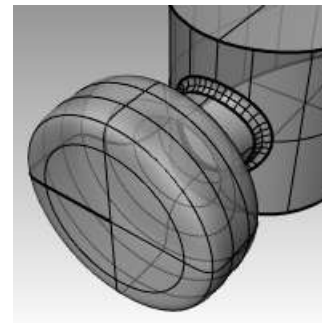
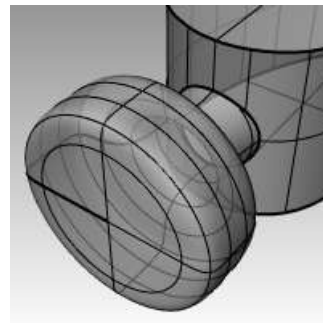


- 3 Change to the **Head** layer.
  - 4 Use the **Revolve** command (*Surface menu > Revolve*) to revolve the curve.
- Use the midpoint of the construction line for the revolve axis.
- 5 Save your model.



### To add the head to the claw part:

- 1 Use the **BooleanUnion** command (*Solid menu > Union*) to join the head with the claw.
- If the result is not correct, flip the normal of the head surface with the Dir command. The normal of the head surface should be towards out.
- 2 Use the **FilletEdge** command (*Solid menu > Fillet Edge*) fillet the intersection between the head and the claw.
  - 3 **Save** your model.





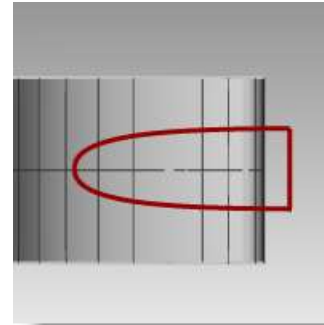
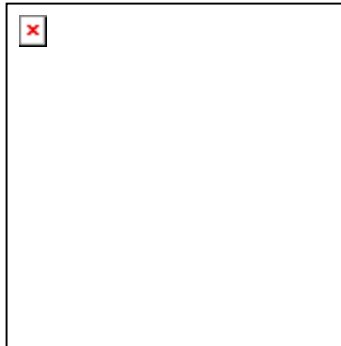
### To create slot for the claw part of the hammer:

- 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.

- 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.

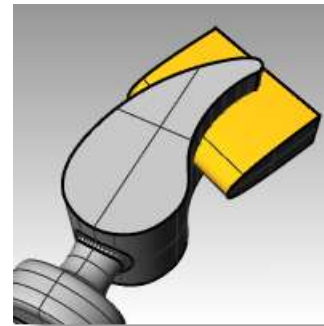
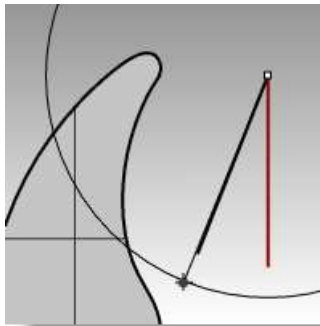


- 4 Drag the closed curve closer to the claw.

- 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.

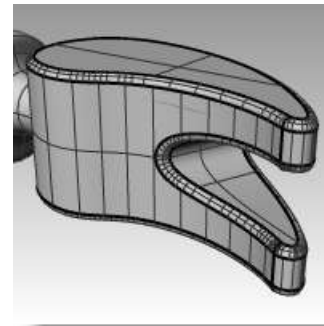
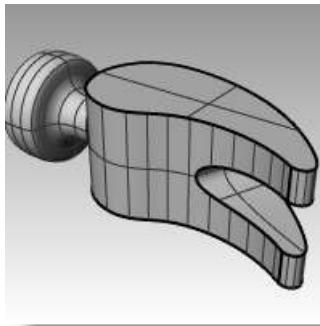
- 7 Use the **ExtrudeCrv** command (*Solid menu > Extrude Planar Curve > Straight*) to extrude the curve through the claw.



- 8 **Save** your model.

- 9 Use the **BooleanDifference** command (*Solid menu > Difference*) to subtract the slot from the claw.

- 10 Use the **FilletEdge** command (*Solid menu > Fillet Edge*) to make the fillets around the top and bottom of the claw, and the slot.



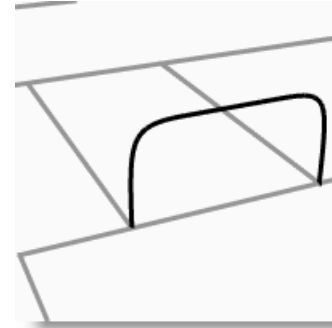
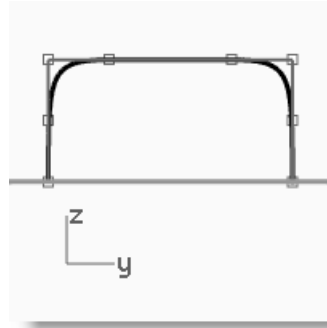
**Notes:**

### To create the shape curve for the tang and the handle:

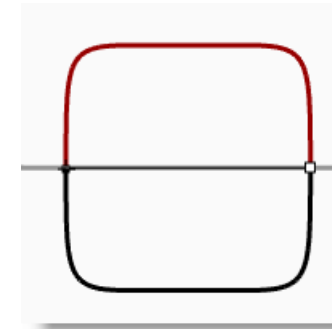
Create the shape curve for the tang in the **Right** viewport. This curve will also be used for the handle.

- 1 Change to the **Curves** 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 Use the **Mirror** command (*Transform menu > Mirror*) to create the other curve.
- 4 Use the **Join** command (*Edit > Join*) to join the curves.
- 5 **Save** your model.



### Notes:

## Notes:

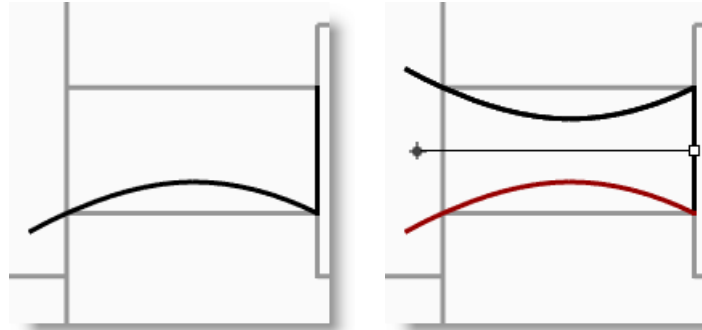
### To create the tang:

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

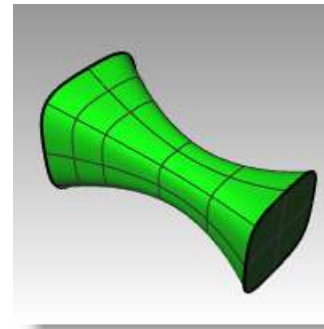
Make sure that it intersects the claw.

- 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.
- 5 Use the **Cap** command (*Solid menu > Cap Planar Holes*) to make the tang a closed polysurface.
- 6 **Save** your model.



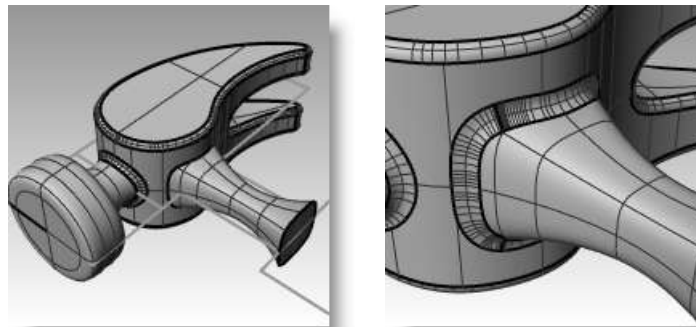
### To finish the hammer head:

- 1 Select the tang and the claw.
- 2 Use the **BooleanUnion** command (*Solid menu > Union*) 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.

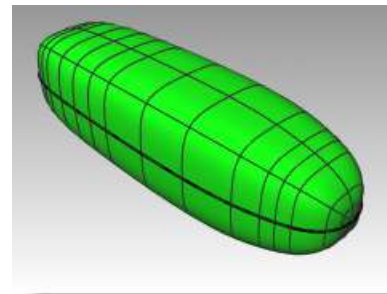
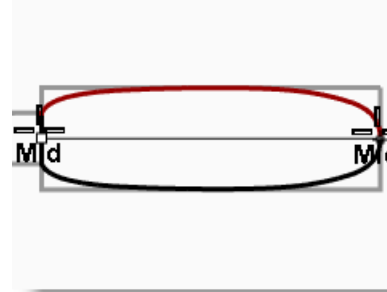
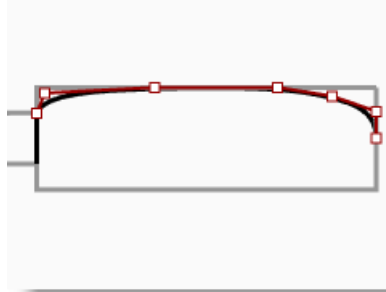
The edge has a round on it.

- 4 **Save** your model.



### To create the handle:

- 1 Change to the **Curves** 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 Use the **Cap** command (*Solid menu > Cap Planar Holes*) to cap the open end.
- 7 **Save** your model.



### Notes:

**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.

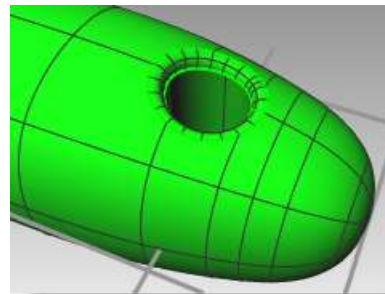
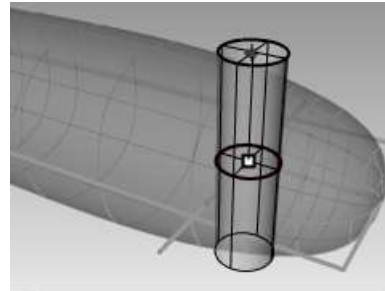
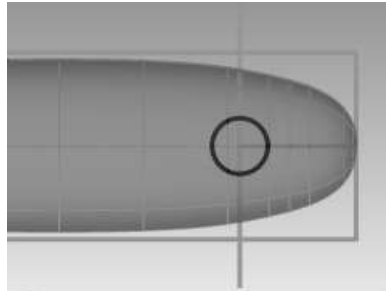
- 2 Use the **ExtrudeCrv** command (*Solid menu > Extrude Planar Curve > Straight*) to extrude the curve on both sides of the construction plane.

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.



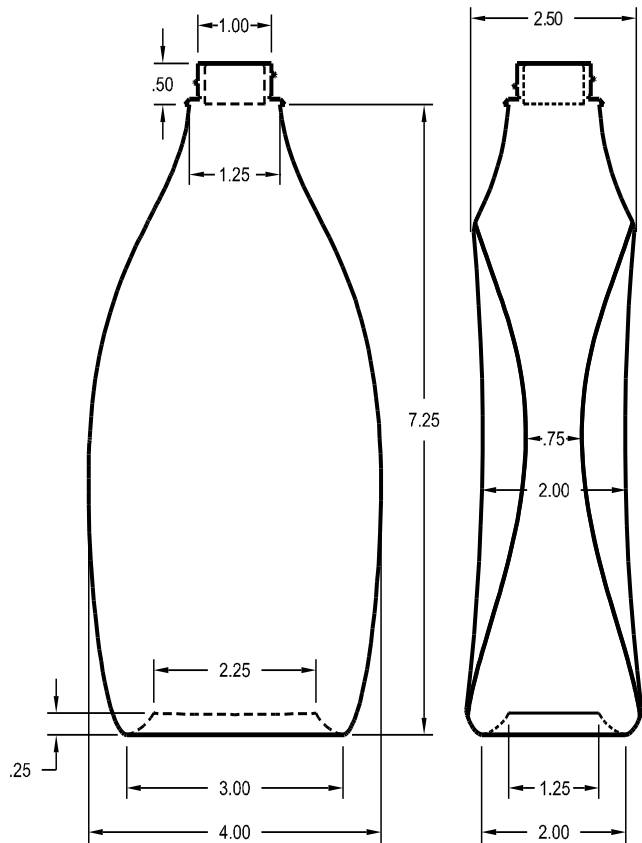
**Notes:**

## Notes:

### Exercise 66— 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.



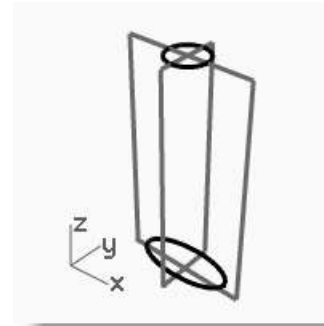
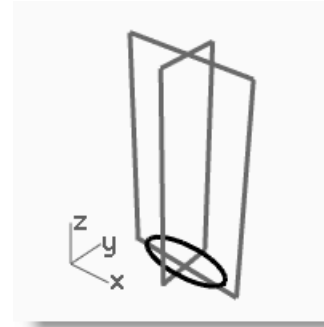
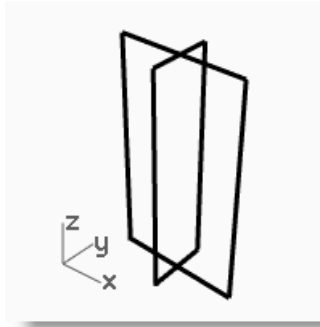
- 1 Start a new model, use the **Small Objects - Inches.3dm** template.
- 2 Save as **Bottle**.
- 3 Make the following layers: **Construction**, **Curve**, **Bottle**, **BottleTop**, **Threads**, **Cap1**, **Cap2**, and **CapTop**.
- 4 Change to the **Construction** layer.

## Create the bottle shape

To create the bottle shape, you will draw a curve 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.
- 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.



### Notes:



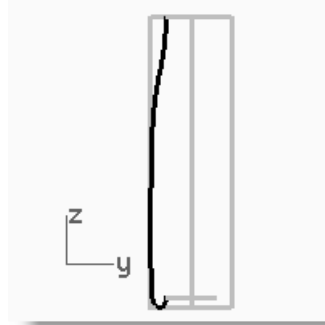
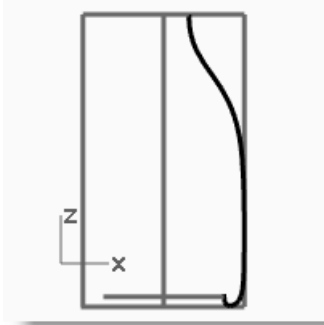
Rectangle



Ellipse

### To draw the profile curves:

- 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** and **Right** viewport.

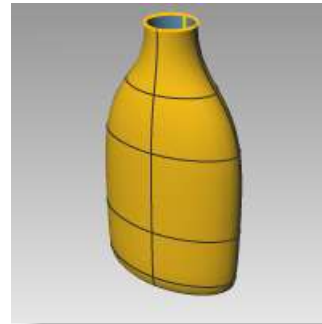


Use the rectangle and object snaps to help you establish the correct size. The curve will be used to create the surface of the bottle.

- 3 Use the **Mirror** command (*Transform menu > Mirror*) to create the other curves.

### To create the surface for the bottle:

- 1 Change to the **Bottle** layer.
- 2 Use the **Loft** command (*Surface menu > Loft*) to make the bottle surface.
- 3 For **Select curves to loft**, select the profile curves in a clockwise direction, press **Enter**.
- 4 In the **Loft Options** dialog, check **Closed loft**, set the **Style=Normal**, click **OK**.



### Notes:



Loft



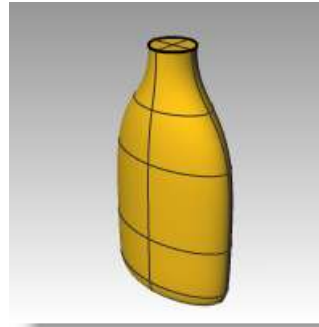
## Cap the top and bottom

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 circular shape and the bottom elliptical shape, 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:



Cap Planer Holes

## Flatten the sides

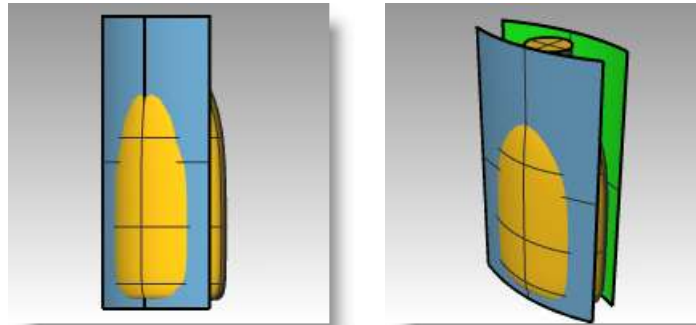
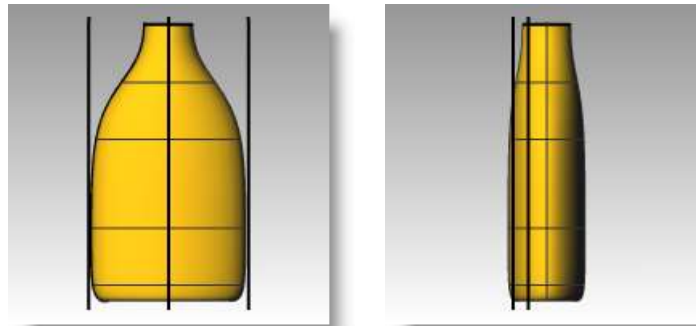
In this part of the exercise you will create a custom surfaces to trim an area on each side of the bottle for a label. The new surface will have curvature in only one direction.

### To create the trimming surface:

- 1 Change to the **Curve** layer.
- 2 In the **Front** viewport, draw three lines. One line in the middle and one on each side.  
Make sure the lines extend a little below and above the height of the bottle.
- 3 In the **Right** viewport, **Move** the lines so that they will intersect the bottle, like the illustration on the right.

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 Select the three curves you just created.
- 5 Use the **Loft** command (*Surface menu > Loft*) to make the cutting surface.
- 6 In the **Loft Options** dialog, uncheck Closed loft, click **OK**.  
A lofted surface intersects the bottle.
- 7 **Mirror** the surface to the other side of the bottle.
- 8 **Save** your model.



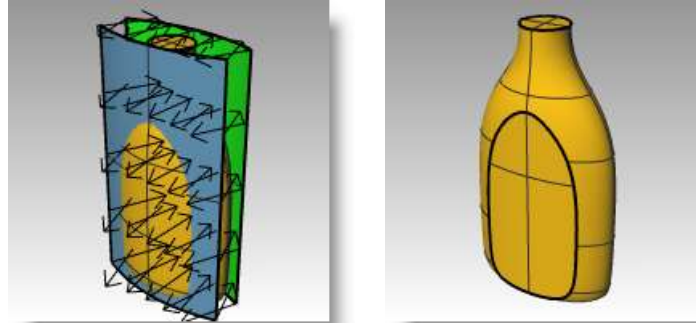
## Notes:



Loft

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



### Notes:



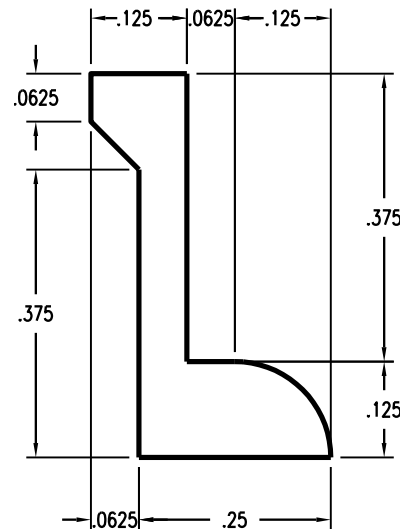
Boolean Difference

### Create the bottle's top

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

### To create the profile curve:

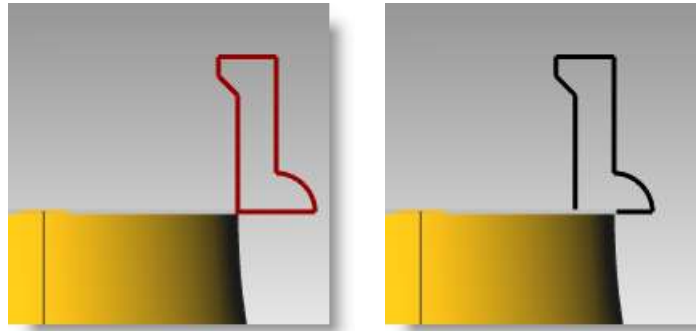
- 1 Change to the **Curve** layer.
- 2 In the **Front** viewport, use the **Lines** command (*Curve menu > Line > Line Segments*) and the **Arc** command (*Curve menu > Arc > Center, Start, Angle*) to create a profile curve for the outside and the inside of the top.
- 3 Use the drawing to the right to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

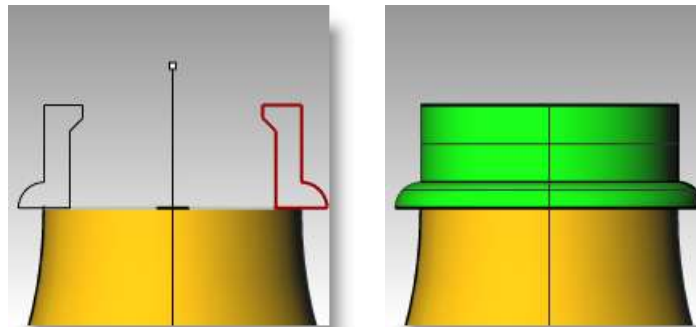
**Tip** When drawing line segments, you can use **object snaps**, the **distance constraint**, and **ortho** to draw precisely. For example, for the first line type .5,7.25 for the starting point, then simply type .25 to constrain the line to .25 units. Turn Ortho on and drag the line to the right and 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 in an upward direction.

- 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 Type **0** and press **Enter** for the first axis point
- 4 Turn **Ortho** on and pick another point up or down from the first point for the other axis point.
- 5 For the **Start Angle**, click **FullCircle**.  
The top is created.



Revolve

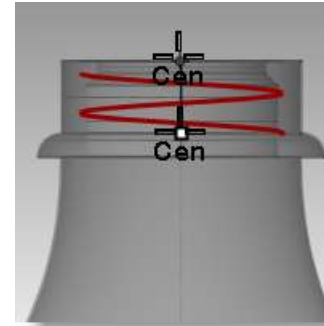
**Note:** Practice additional surface modeling techniques by creating threads on the bottle top and the caps for the model. Use the technical drawing to help.

## 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:

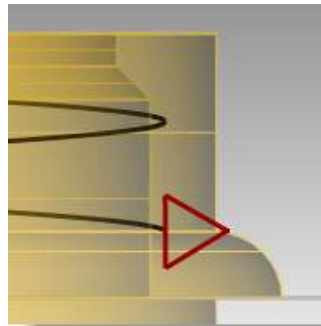
- 1 Change to the **Curve** layer.
- 2 From the **Curve** menu, click **Helix**.
- 3 Snap to the center point at the bottom of the area to be threaded.
- 4 Snap to the center at the top of the area to be threaded.
- 5 Click **Turns**.
- 6 Type **1.5** and press Enter.
- 7 Pick a point near the edge that is slightly smaller than the outside edge of the area to be threaded.



  
Helix

### 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 Select the helix near the bottom.
- 5 In the **Array Along Curve Options** dialog, change the **Number of items** to **5**, click **Roadlike** then click **OK**.

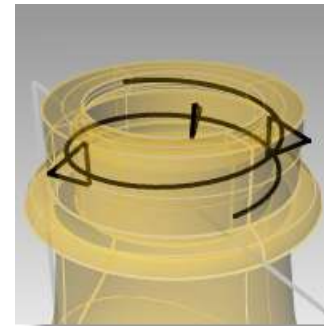
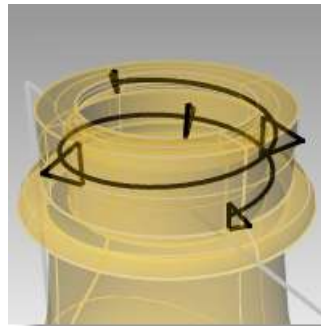


- 6 Click in the **Top** viewport.

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.



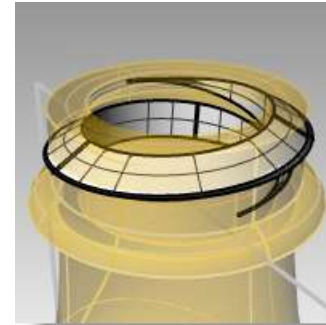
## To create the thread surface:

- 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 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 Use the **End** object snap to pick one end of the helix.
- 6 Select the profile curves in order.
- 7 To end the surface at a point, at type **P** and press **Enter**.
- 8 Use the **End** object snap to pick the other end of the helix.
- 9 Press **Enter**.
- 10 In the **Sweep 1 Rail Options** dialog box, change the **Style** to **Roadlike Top**, and click **OK**.



- 11 **Show** the bottle.
- 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.

## Notes:



Sweep 1 Rail

# 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 many 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.

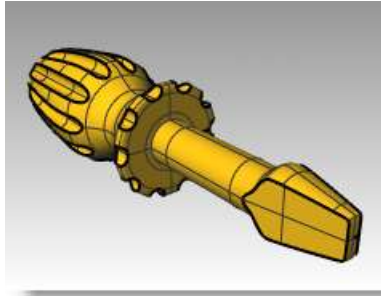
There are two methods to export models to other formats. You can "Save As" and choose a specific export format to export an entire model. You can select some objects and "Export Selected" then choose a specific export format to export a portion of the model. In the following exercise you will use the "Save As" method to export three of the most common file formats.

## Exercise 67— Exporting models

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

- 1 **Open** the model **Export.3dm**.
- 2 From the **File** menu, click **Save As**.
- 3 In the **Save** dialog box, change the **Save as type** to **Stereo lithography (\*.stl)**.
- 4 In the file name box type **Export** and click **Save**.
- 5 In the **STL Mesh Export Options** dialog box, set the Tolerance to **0.01**, and click **Preview**.
- 6 Set the Tolerance to **0.1**, click **Preview**, and then click **OK**.
- 7 In the **STL Export Options** dialog box, select **Binary**, check **Export open objects**, and click **OK**.

Detailed mesh controls are discussed in more depth in the Level 2 training class.

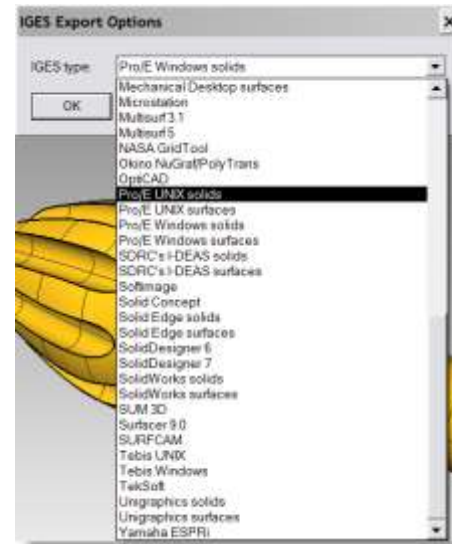


## Notes:



### To export a model to IGES:

- 1 From the **File** menu, click **Save As**.
- 2 In the **Save** 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 allow the user more input.
- 4 Click **Cancel** to terminate, or **OK** to create the IGES file.



### Notes:

### 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.



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**Notes:**

# 11

## Rendering

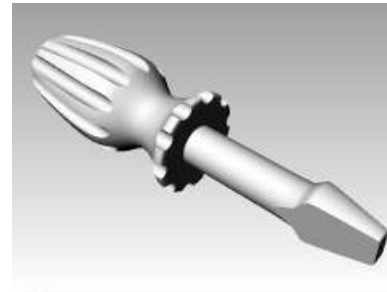
Rendering is available for showing your model as if it was photographed. While the Rhino renderer may be good enough for much of your work, use another rendering program such as Rhino's Flamingo plug-in for higher quality results. Flamingo is available as a plug-in to Rhino 2.0. Visit [www.flamingo3d.com](http://www.flamingo3d.com) 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 68— Practice rendering a model**

- 1** Open the model **Render.3dm**.
- 2** From the **Render** menu, click **Current Renderer**, and then click **Rhino Render**.
- 3** Right click on the **Perspective** title bar, and then click **Rendered display**.

The viewport mimics but does not exactly duplicate what you will get in a Render.



### **Notes:**



Rendered Viewport

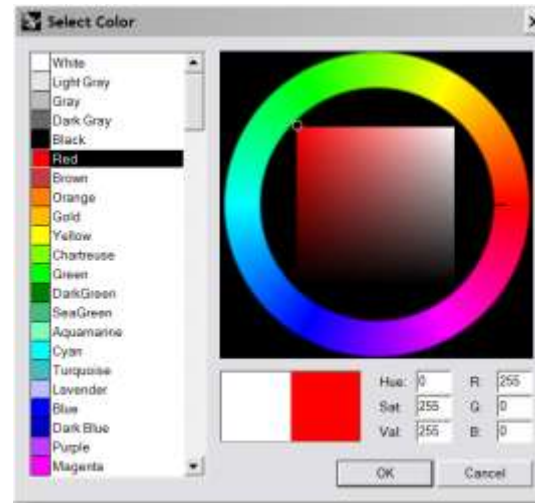
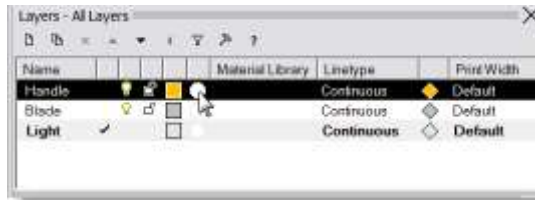
## To assign a color to the handle:

- 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 **Gloss finish** setting.

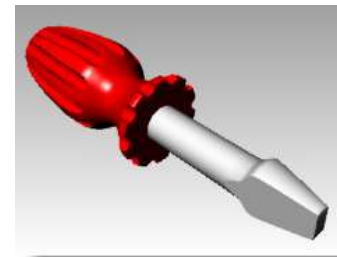
A highlight of 0 means that the object is not shiny at all and it will not have a specular highlight. A low value of highlight makes the shiny spot to be small, and the object will look glossier. As the highlight increases in value, the shiny spot gets large—this makes the object look more like it is 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 **Gloss finish** setting to **90**, and then click the **Color** switch.
- 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.



## Notes:



## To place a light:

Start with a standard lighting scheme. You can experiment to develop your own lighting schemes later.

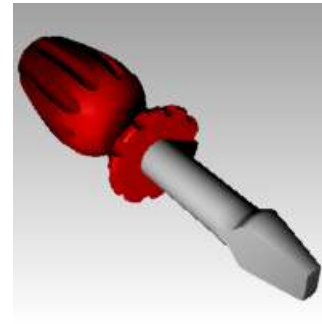
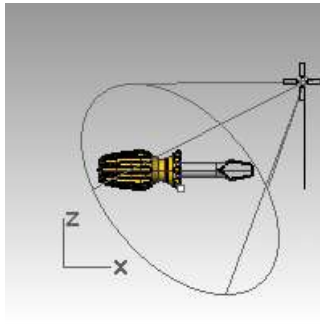
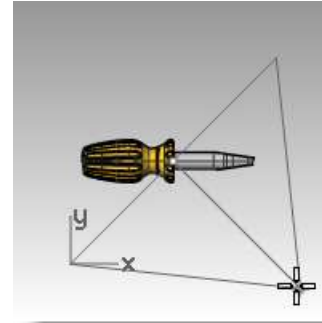
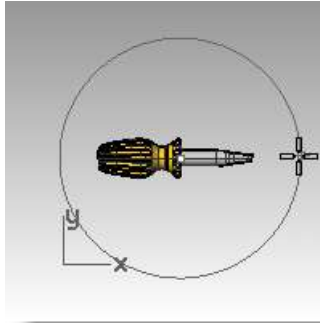
- 1 Zoom out in the **Top** and the **Front** viewport.
- 2 Change to the **Lights** layer.
- 3 From the **Render** menu, click **Create Spotlight**.
- 4 For the **Base of cone**, type **0** and press **Enter**.
- 5 For the **Radius**, pick a point so that the circle is larger than the entire screwdriver in the **Top** viewport.
- 6 For the **End of cone**, hold down the control key, and pick a point below and to the right in the **Top** viewport.

This starts elevator mode.

- 7 For the **End of cone**, pick 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.



## Notes:



Create Spotlight

## Notes:

### To place a second light:

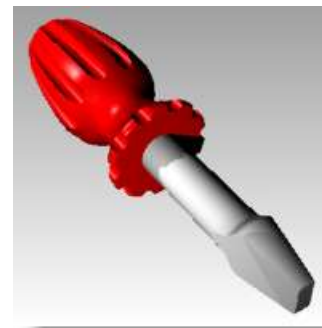
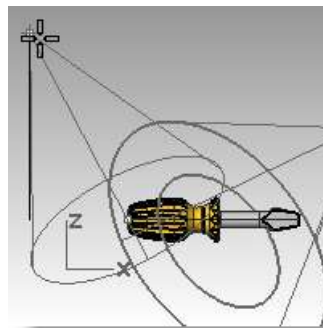
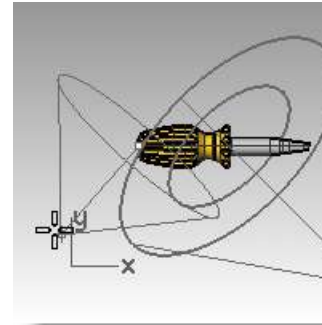
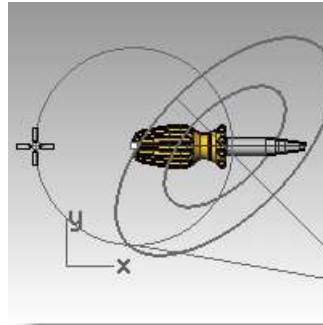
- 1 Zoom out in the **Top** and the **Front** viewport.
- 2 From the **Render** menu, click **Create Spotlight**.
- 3 For the **Base of cone**, type **0** and press **Enter**.
- 4 For the **Radius**, pick a point so that the circle is larger than the handle of the screwdriver in the **Top** viewport.
- 5 For the **End of cone**, hold down the control key, and pick a point below and to the left in the **Top** viewport.

This starts elevator mode.

- 6 For the **End of cone**, pick above the object in the **Front** viewport.

This will be your secondary (fill) light.

- 7 Click in the **Perspective** viewport.
- 8 From the **Render** menu, click **Render**.

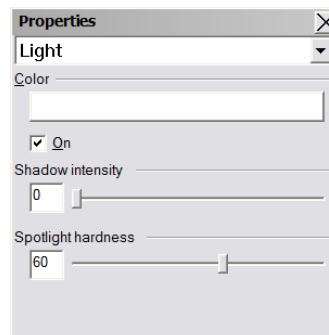


### To assign properties to the light:

- 1 Select the new light.
- 2 From the **Edit** menu, click **Object Properties**.
- 3 On the **Light** page, change the **Shadow intensity** to **0** 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**.



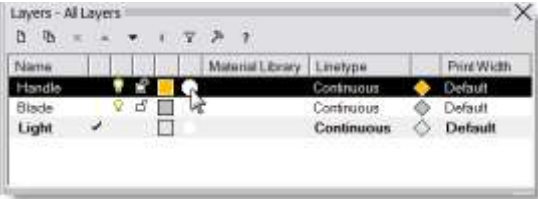
**To add a bumpy surface to the handle:**

- 1 In the **Layers** dialog box, click on the **Materials** column for the **Handle** layer.
- 2 In the **Material Editor** dialog box, check **Bump**.

You can use any bitmap file for a bump.

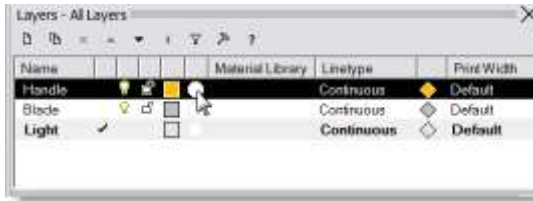
The bumps come from the pattern of light and dark in the bitmap image.

- 3 In the **Open Bitmap** dialog box, select **HandleBump.jpg**, and then click **Open**.
  - 4 In the **Material Editor** dialog box, for **Bump Tiling**, click **Modify**.
  - 5 In the **Offset and Tiling** dialog box, change the **U** tiling to 4.0 and the **V** tiling to 6.0, click **OK**.
  - 6 Click **OK** to close the **Material Editor** dialog box.
  - 7 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 Editor** dialog box, uncheck **Bump**.
- 3 In the **Material Editor** dialog box, check **Texture**.
- 4 In the **Open Bitmap** dialog box, select **Wood.jpg**, and then click **Open**.
- 5 In the **Offset and Tiling** dialog box, change the **U** tiling to 4.0 and the **V** tiling to 6.0, click **OK**.
- 6 Click **OK** to close the **Material Editor** dialog box.
- 7 From the **Render** menu, click **Render**.  
The surface of the handle has a wood texture appearance.

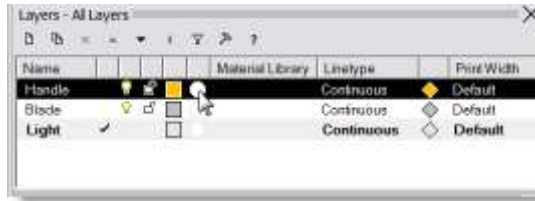


**Notes:**

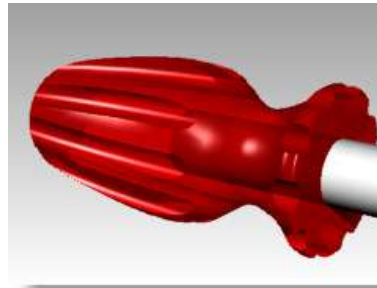


**To make the handle transparent:**

- 1 In the **Layers** dialog box, click on the **Materials** column for the **Handle** layer.
- 2 In the **Material Editor** dialog box, uncheck **Texture**.



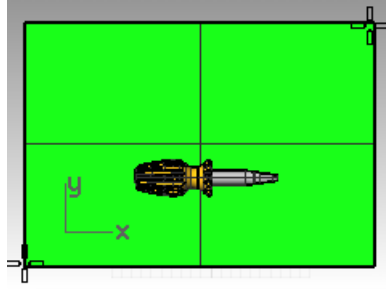
- 3 In the **Material Editor** dialog box, change the **Transparency** setting to **50**.
- 4 Click **OK** to close the **Material Editor** dialog box.
- 5 From the **Render** menu, click **Render**.  
The handle will look transparent.



**Notes:**

**To add a surface for a ground plane:**

- 1 Use the **Plane** command (*Surface menu > Plane > Corner to Corner*) to draw a flat surface in the **Top** viewport.
- 2 Select the surface.
- 3 Use the **Properties** command (*Edit menu > Object Properties...*) on the **Material** page, click **Basic**.
- 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**, and then click **Flamingo Raytrace**.
- 4 In the **Layers** dialog box, click on the **Materials** column for the **Handle** layer.
- 5 In the **Material Editor** dialog box, click **Plug-in**, and then click **Browse**.
- 6 In the **Material Library** dialog box, in the **Plastics** library, in the **Transparent** folder, select a material, press **OK**.
- 7 In the **Material Editor** dialog box, press **OK**.
- 8 Repeat this process for the **Blade** layer.
- 9 Choose a shiny metal material, like **Metal\Steel\Polished\Plain**.
- 10 **Render** the model.



## Notes:

### 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.



---

**A more complete discussion of Flamingo rendering is included in Level 2 training.**

---

For extra practice use the canoe, the hammer, or the bottle you made in the earlier exercises.

---

**Notes:**

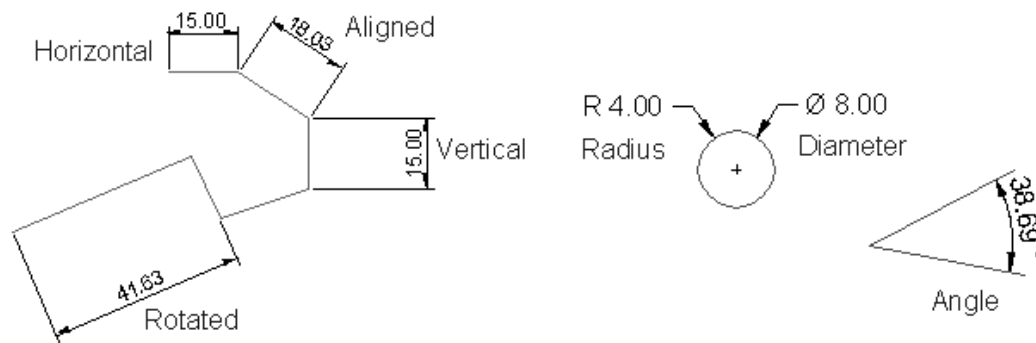
# 12 Dimensions






Notes:







## Dimensions

You can create simple dimensions on all viewports.

### Dimension Types



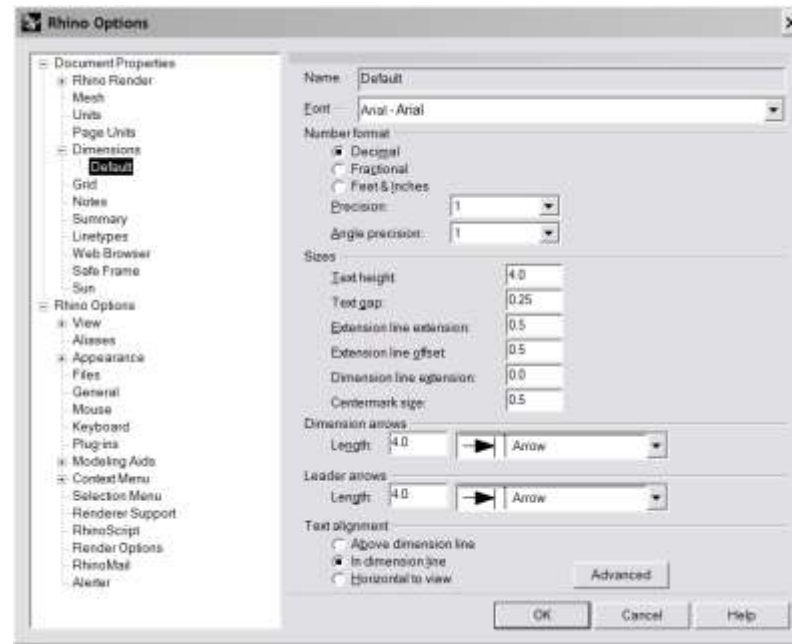
Button	Command	Description
	Dim	Linear Dimension creates a horizontal or a vertical dimension.
	DimAligned	Creates an aligned dimension.
	DimRotated	Creates a rotated dimension.
	DimAngle	Creates an angular dimension.
	DimRadius	Creates a radius dimension.

Button	Command	Description
	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.

## Notes:

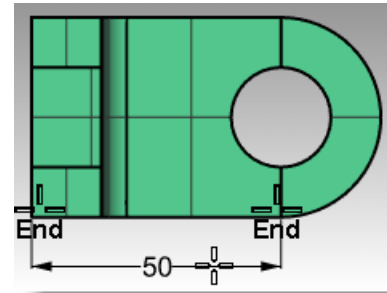
### Exercise 69— Practice dimensioning

- 1 **Open** the model **Dimension.3dm**.
- 2 From the **Tools** menu, click **Options**.
- 3 In the **Rhino Options** dialog box, on the **Dimensions** page, make the following changes.

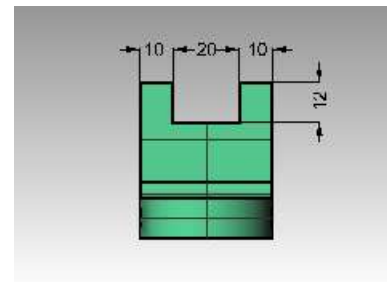
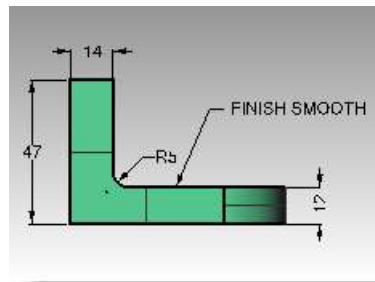
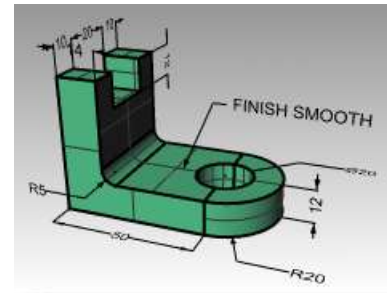
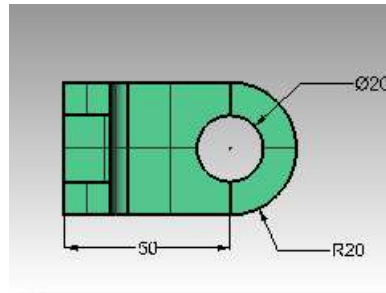


Options

- 4 From the **Dimension** menu click **Linear Dimension**.
- 5 Snap to the lower left end of the part in **Top** viewport.
- 6 Snap to the lower right end of the part in the **Top** viewport.
- 7 Pick a point below the part in the **Top** viewport.



- 8 From the **Dimension** menu, click **Radial Dimension**.
- 9 Select the lower right quadrant of the arc in the **Top** viewport.
- 10 Click to place the dimension text.
- 11 From the **Dimension** menu, click **Diameter Dimension**.
- 12 Select the upper right quadrant of the hole in the **Top** viewport.
- 13 Click to place the dimension text.
- 14 Dimension the rest of the drawing using leaders, text blocks, horizontal, vertical, radius, and diameter dimensions.
- 15 **Save** your model.



### Notes:



Linear dimension

Use object snaps to locate the extension line origins.

To move extension line or text locations for dimensions, turn on control points for the dimension and move the control points.



Radial dimension



Diameter dimension

## 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.

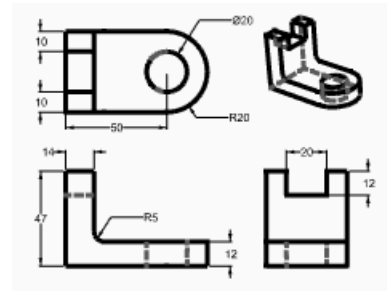
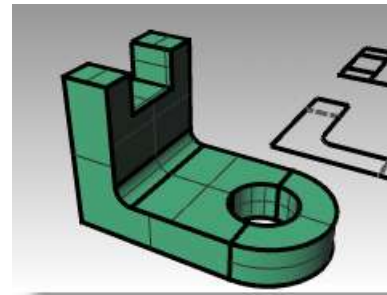
The Make2-D command makes a two-dimensional drawing for all four views.

### Exercise 70— 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 **2-D Drawing 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 2-D drawing.



### Notes:



Make 2-D Drawing



# 13

## Layout and Printing

Notes:

### Layouts and Details

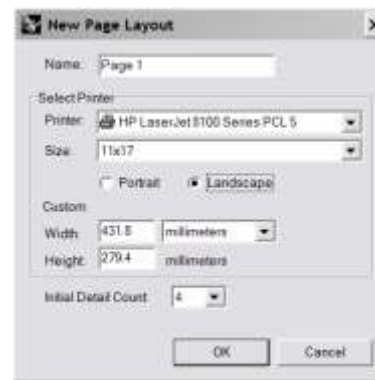
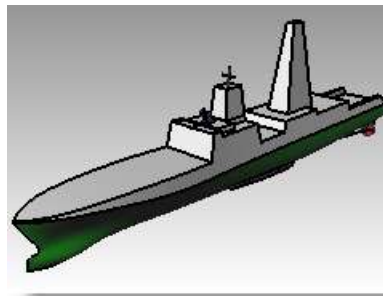
Layout viewports can include various views of the model and annotations like title blocks and notes. When these viewports are tabbed with the model viewports, they can be easily accessed from the tabs.

Geometry in a layout viewport does not appear in the model viewports. Layout viewports include a background with one or more detail views of the model. The detail edge appearance is controlled with the Properties command and has exactly the same object properties as any curve, including Layer, Display Color, Print Color, and Print Width.

The **Print Width—No Print** property is set by default. Detail properties include Locked status and Scale.

### Exercise 71— Practice making layouts

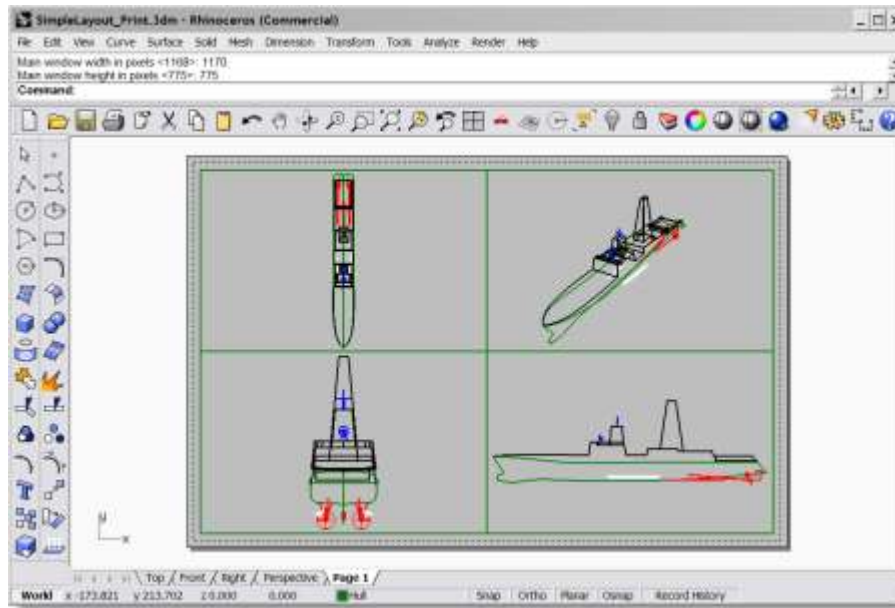
- 1 **Open** the model **SimpleLayout\_Print.3dm**.
- 2 Click in the **Top** viewport.
- 3 From the **View** menu, click **Page Layout**, click **New Page Layout**.
- 4 In the **New Page Layout** dialog, click **Landscape**. Set the **Initial Detail Count** to **4**. Adjust any other options appropriate for your printer or plotter, click **OK**.



A layout page opens and tabs appear at the lower edge of the Rhino window labeling the modeling windows and the layout page.

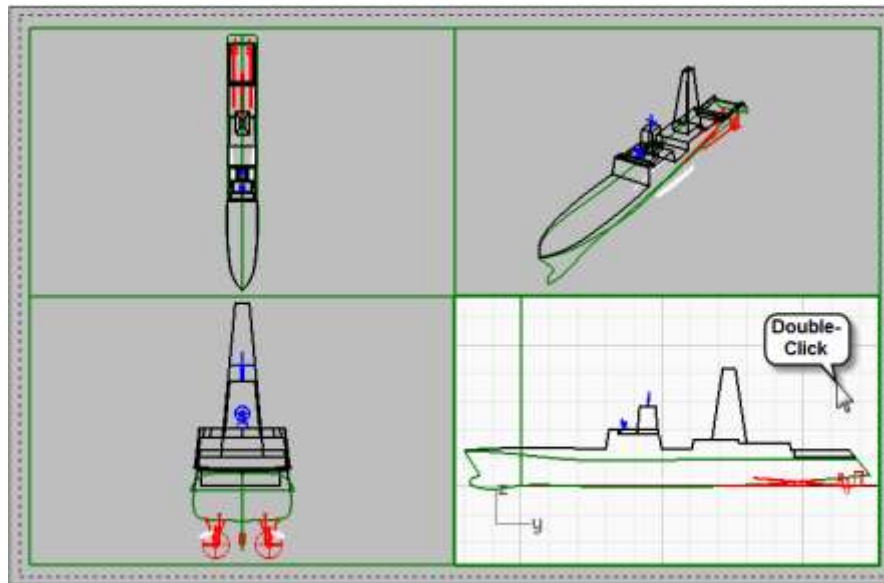
The new layout has four detail views, showing the object from the same direction as the four default Rhino viewports.

Rhino turns on the tabbed viewports to make it easier to navigate among multiple layouts and modeling viewports.



Details are windows into the 3-D model. They can be activated as modeling viewports by double-clicking in the viewport.

- 5 **Double-click** in the **Right** viewport.



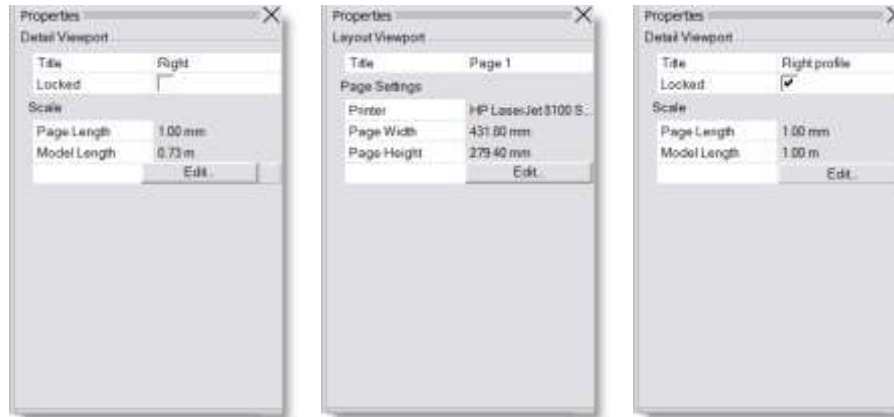
**Notes:**

## Set scale and lock details:

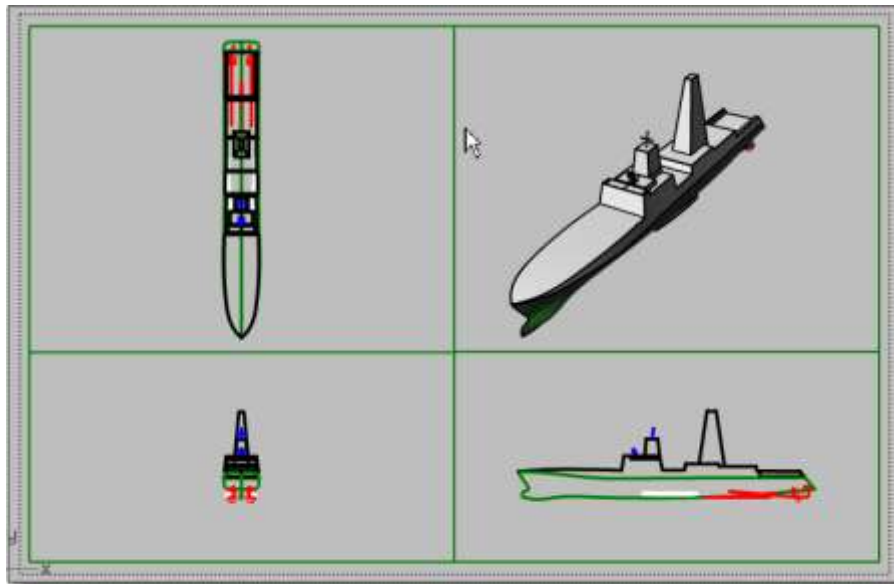
- 1 If the **Properties** window is not open, open it now (*Edit>Object Properties*) and dock it to one side.

With nothing selected, the properties window shows the viewport properties.

With no detail active, the properties shown are for the layout as a whole.



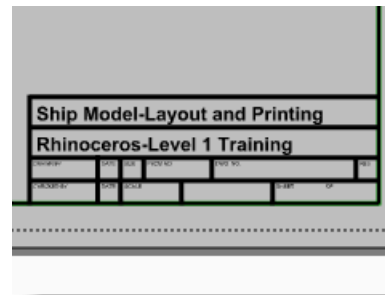
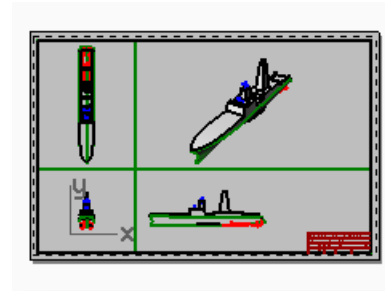
- 2 Click the **Edit** button to edit the title, and size and printer properties.
- 3 On the command line, for the **Distance on page (mm)**, set it to **1**, press **Enter**.
- 4 On the command line, for the **Distance in model (m)**, set it to **1**, press **Enter**.
- 5 Change the title to **Right Profile**, and **Lock** the detail.
- 6 Repeat this in the **Top** and **Front** details to set all of these details to the same scale.
- 7 Activate the **Perspective** detail.
- 8 From the **View** menu, click **Shaded**.



## Notes:

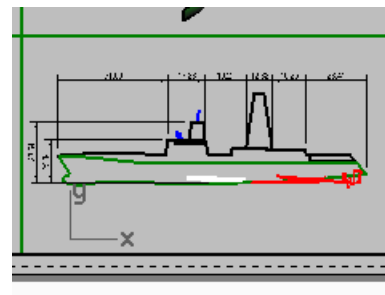
### To draw borders and title blocks in the layout space:

- 1 **Double-click** in the **Perspective detail** to deactivate it and make the layout space active.
- 2 Draw a **Rectangle** (*Curve>Rectangle>Corner to corner*) around the viewports on the Layout.
- 3 From the **File** menu, click **Insert**.
- 4 On the **Insert** dialog, check **Prompt** for **Insertion point**, uncheck **Prompt** for **Scale** and **Rotation**.
- 5 On the **Insert** dialog, click **File...**, select **TitleBlock.3dm**, click **Open**, click **OK**.
- 6 For the **Insertion Point**, snap to the lower right corner of the rectangle. A title block is inserted on the layout.
- 7 You can add information to the title block with the text command. To edit the text on the title block, explode it first.



### To add dimensions to the layout space:

- 1 **On the layout use Linear dimensions to measure some features.** The dimensions are located on the layout. They will not display in the model views.
- 2 Add more dimensions as needed to the other layout viewports.



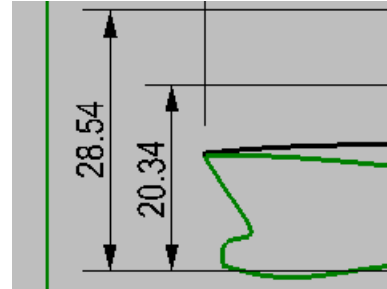
### Notes:

### To set linetypes and line weights for curves:

These are used in printing and can be displayed in the Rhino views using the PrintDisplay and LinetypeDisplay commands.

- 1 From the **Edit** menu, click **Layers**, then click **Edit Layers**.
- 2 On the **Layers** dialog, in the **Print Width** column, change the width of some of the layers.
- 3 Select the **Border rectangle** and the **Title Block**.
- 4 On the **Object Properties** dialog, change the **Print Width** to a thicker weight.
- 5 Type **PrintDisplay** on the command line, click **State=Off** on the command line, change it to **On**.

Note the difference in thickness of the curves.  
Linetypes can be adjusted the same way.



## Printing

### Exercise 72— Practice printing

#### To print the layout:

- 1 From the **File** menu, click **Print**.
- 2 On the **Print** dialog, set the print **Destination** to the desired printer.

In addition to your Windows printers, the list of printers includes pdf print drivers, if available, as well as Image File which allows printing to an image file like \*.png, \*.jpg, \*.bmp, \*.tif, or \*.tga.

The detailed settings below the list of destinations will change, depending on the selected printer or driver. Generally, the settings control print quality, orientation (landscape or portrait), paper size, number of copies to print, and a button to access the printer properties.

- 3 For **Printer output**, set it to raster.

To print layouts or viewports that contain a shaded display, the raster setting is required.

To print layouts or viewports as curves or lines only, use the vector setting.



### Notes:



Print

**4** For the **Output color**, set it to **Print Color**.

Output color can be set on any one of the following:

**Print Color** uses object color for the print color. This can be set to any color per object or per layer. An object that is displayed with a blue wireframe in the viewport can print to any other color.

**Display Color** is the default setting in the print dialog. Objects print the color of the wireframe in the viewport.

**Black and White** forces black and white printing only, ignoring the display or print colors of objects.

In the **SimpleLayout** file, the wireframes of the various objects are of varying color. Some of them correspond to the layer color of the layer they are on, like the hull, which displays green. Others have their display color set by object to override the layer color.

**5** Set the **View and Output Scale**. Set the **View** to **Extents**, and the **Scale** to **1:1**.

The reason we can set this to 1:1 is because we set up our layout to the paper size and we scaled the viewports to the correct scale.

**6** **Cancel** the **Print**, unless you have a printer available.



**Notes:**

# **Part Four: Customizing Workspaces and Toolbars**

---





# 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.

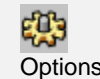
### Exercise 73— Practice with options

#### 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.
- 3 **Modeling Aids** also contains pages for **SmartTrack** options, **Nudge** controls, and **Cursor ToolTips**.



### Notes:



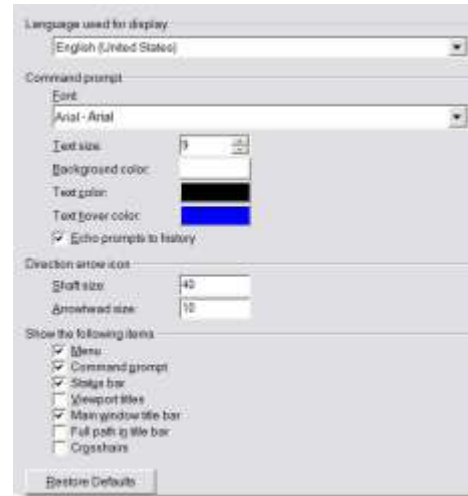
Options

### To change Rhino's screen appearance:

- 1 Click the **Appearance** page.

The Appearance page controls the appearance of the Rhino window.

- 2 Click the white rectangle next to **Background color** and change it to a different color.
- 3 Check the **Crosshairs** box.



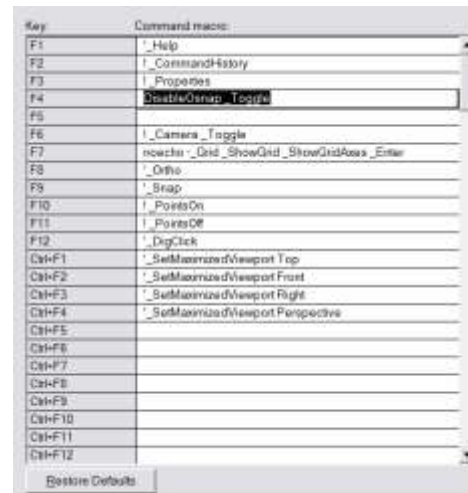
### To change or create shortcuts:

- 1 Open the **Keyboard** page.

The Keyboard page sets up shortcut keys for Rhino commands.

- 2 Type **DisableOsnap \_Toggle** in the box next to the **F4** key.

This gives you a key that toggles your persistent object snaps between on and disabled.



**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**, 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.

**To change general options:**

- ▶ Click the **General** page.

The General page controls the number of undo's 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 74— 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.

### **Notes:**



Document Properties

# 15

## Custom Toolbar Layouts

Notes:

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 75— Customizing a toolbar layout

- 1 Begin a new model.
- 2 From the **Tools** menu, click **Toolbar Layout**.
- 3 In the **Toolbars** dialog box, from the **File** menu, click **Save As**.
- 4 In the **Save Toolbars Collection** dialog box, in the **File name** box type **Level 1**, and click **Save**.

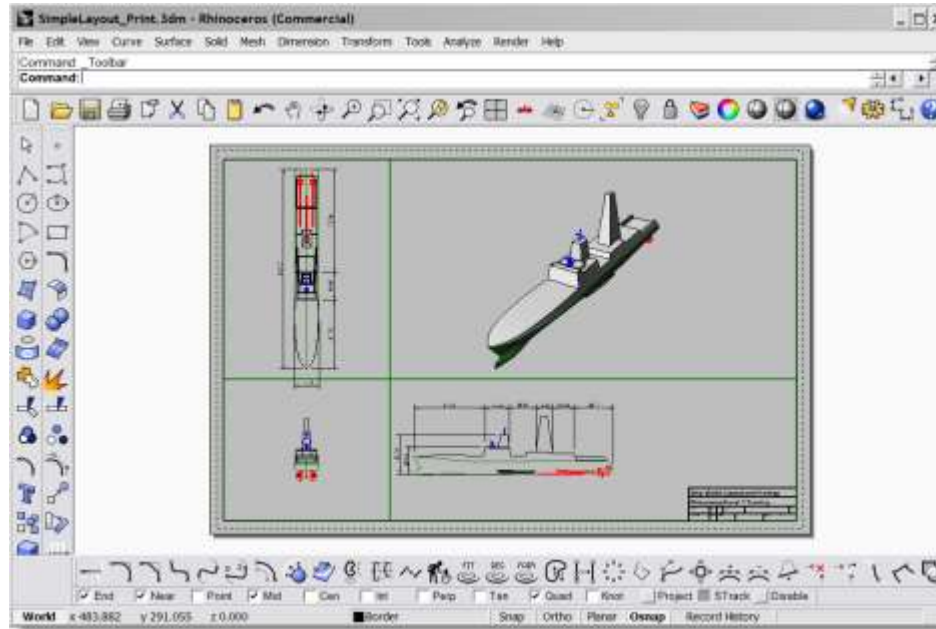
A new toolbar collection file has been created. Toolbar collection files are saved with a .tb extension. You will use this new toolbar collection to do some customization.

### 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, and then click **Close**.



- 5 To dock the **Curve Tools** toolbar at the bottom, drag down until it changes to a horizontal shape, then releasing your mouse button.



**Notes:**

**To save your toolbar layout:**

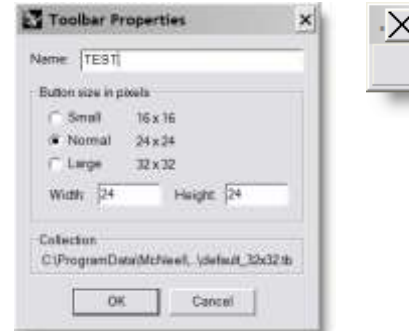
- ▶ In the **Toolbars** dialog box, from the **File** menu, click **Save**.

**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 Toolbar 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**.
- 3 In the **Left mouse button command** box, type **! SelLast ChangeLayer**.
- 4 In the **Right mouse button command** box, type **! SelCrv ChangeLayer**.

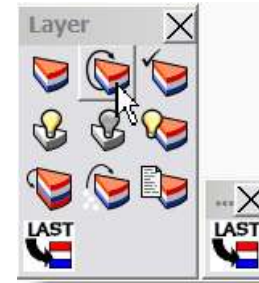


- 5 Click **Edit Bitmap**.
- 6 In the **Edit Bitmap** dialog box, make a picture for the button face image, and click **OK**.
- 7 In the **Edit Toolbar Button** dialog box, click **OK**.



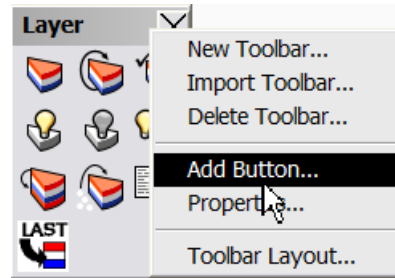
### To copy a button from one toolbar to another:

- 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 Right click on the **Test** toolbar in the list, and select **Add 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 title bar.



### Notes: