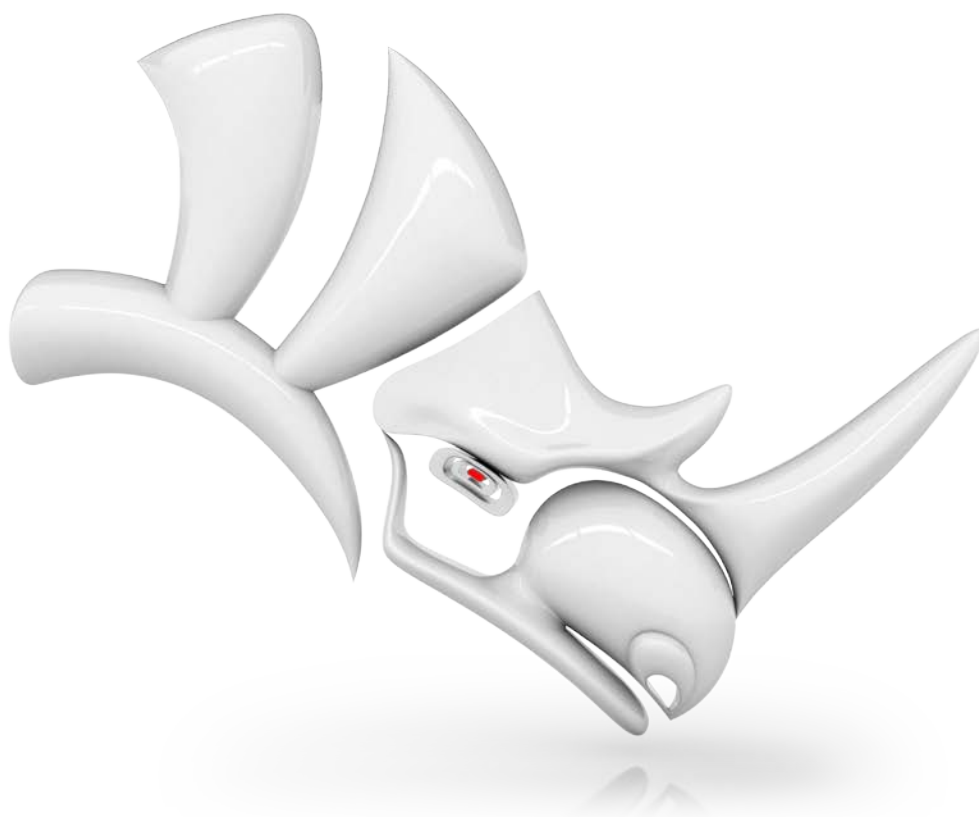


Rhinoceros[®]

modeling tools for designers

Training Manual Level 1



Rhinoceros v5.0, Level 1, Training Manual

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PART ONE

Introduction and Overview

1 Introduction and Overview

Introduction

This course guide accompanies the Level 1 training sessions. Level 1 shows you how to produce 3-D models using NURBS geometry and arrange models for export, annotation and plotting.

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 full days or 6 half day sessions

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 and Gumball
- 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 using Rhino Render
- Dimension and annotate model with text and hatch
- Use Layouts to arrange views of model on paper for printing

Schedule A: Three Classroom Days

Day 1	Topic
8-10AM	Introduction, Rhino interface
10AM-12PM	Rhino interface, panning & zooming
12-1PM	Lunch
1-3PM	Creating geometry
3 -5PM	Creating geometry
Day 2	Topic
8-10AM	Editing
10AM-12PM	Editing
12-1PM	Lunch
1-3PM	Editing
3 -5PM	Control point editing, modeling with solids
Day 3	Topic
8-10AM	Surfacing
10AM-12PM	Surfacing
12-1PM	Lunch
1-3PM	Modeling practice
3 -5PM	Import/export, rendering, dimensioning, printing, customization

Schedule B: Six Half Days (On-line Training)

Session 1	Topic
9-10:45AM	Introduction, Rhino interface
10:45-11AM	Break
11AM-12:45PM	Rhino interface, panning & zooming
Session 2	Topic
9-10:45AM	Creating geometry
10:45-11AM	Break
11AM-12:45PM	Creating geometry
Session 3	Topic
9-10:45AM	Editing
10:45-11AM	Break
11AM-12:45PM	Editing

Session 4 Topic

9-10:45AM Editing

10:45-11AM Break

11AM-12:45PM Control point editing, modeling with solids

Session 5 Topic

9-10:45AM Surfacing

10:45-11AM Break

11AM-12:45PM Surfacing

Session 6 Topic

9-10:45AM Modeling practice

10:45-11AM Break

11AM-12:45PM Import/export, rendering, dimensioning, printing, customization

Overview - What is Rhino?

Rhinoceros is 3D modeling software that can be used in a variety of ways and for very different purposes. It is primarily a surface modeling tool, but has many related functions as well. Many users use only a small portion of what Rhino has to offer, while others need and use a lot more; it depends on what the individual user is trying to accomplish with the application. Even very experienced users can find new and useful tools of which they were previously unaware. In this introduction, a broad overview of various elements that you will encounter while using Rhino to get your work done is provided.

Object types

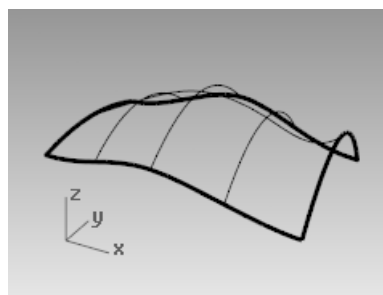
What does *surface modeler* mean?

Rhino has tools to create and edit several different types of objects, of which surfaces are one. A surface in Rhino refers to an infinitely thin, infinitely flexible, mathematically defined digital membrane.

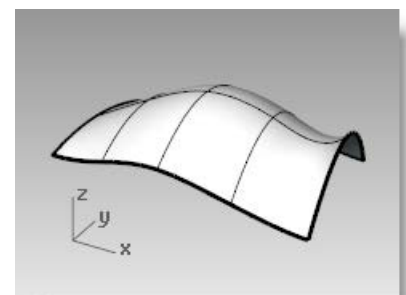
Surface

Surfaces are represented on screen by either some outline curves plus some interior curves, called *isocurves*, or by a shaded picture which makes a surface appear to have some substance and to show light and shading. How surfaces are painted on the screen is dependent on the display mode in the viewport, and does not affect the surface in any way.

The important thing to remember about surfaces is that they are defined with great precision at every point by complex math formulas, they are not approximations.



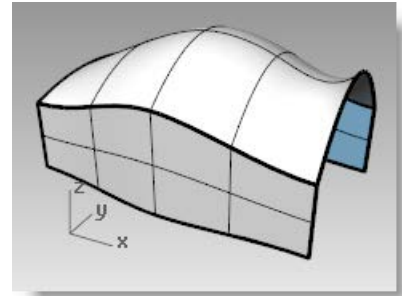
A surface, shown in wireframe



...and shaded views.

Polysurface

Rhino also has objects which consist of two or more surfaces joined together. This is possible when the edge of one surface is located very close to the edge (within tolerance set in the file) of another surface. We call joined surfaces **polysurfaces**. Since there are some restrictions on editing polysurfaces, Rhino makes it very easy to extract the individual surfaces from polysurfaces and join them back together.



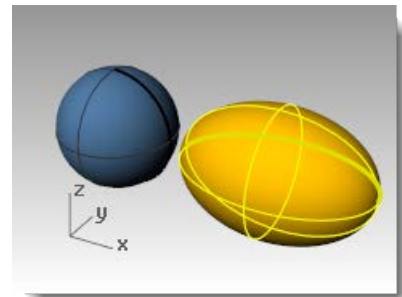
A polysurface in shaded view.

Solid

We have already said that surfaces are infinitely thin; they have zero thickness.

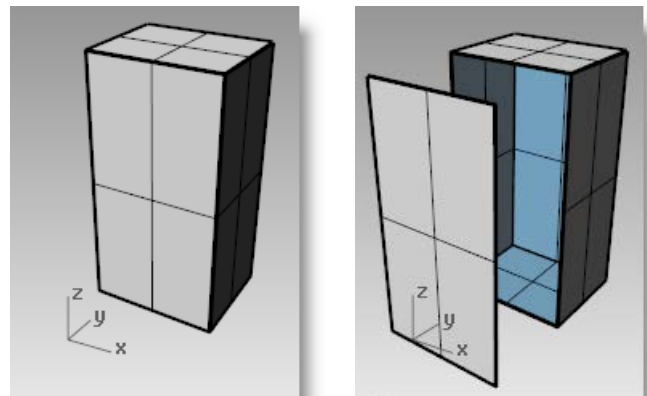
However, any physical object will have some thickness—not zero.

If you need your objects to have thickness, or volume, there are two ways to accomplish this. The first is to make sure the surface closes on itself and has no openings anywhere. Spheres and ellipsoids are examples of this type of surface.



A Sphere and Ellipsoid object.

The second method is to join enough single surfaces together to enclose a space, again, with no openings, not even very small ones. A box is an example of this type of object. We call these objects **solid**, but it is important to remember that there is nothing inside them- they are volumes in space enclosed by the infinitely thin surfaces. If you remove one side of a box, and look inside you will see the backsides of the five surfaces.



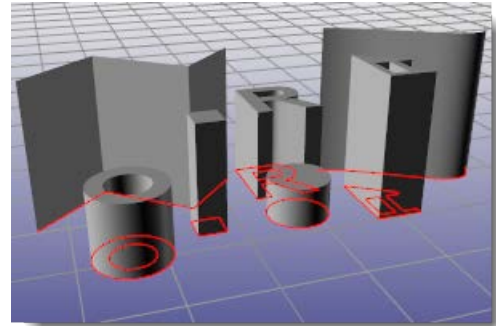
Closed (solid) and open Polysurface.

Lightweight Extrusion Objects

Another object type that is related to a polysurface and a solid is the lightweight extrusion object. Lightweight extrusion objects use less memory, mesh faster, and save smaller than the traditional polysurfaces.

In models containing large numbers of extrusions represented by traditional polysurfaces, performance can be sluggish due to the relatively high demand on resources. If the same objects made in Rhino as lightweight extrusion objects, are more responsive and leave plenty of memory available.

In Rhino 5 commands like Box, Cylinder, Pipe, and ExtrudeCrv create lightweight extrusion objects by default

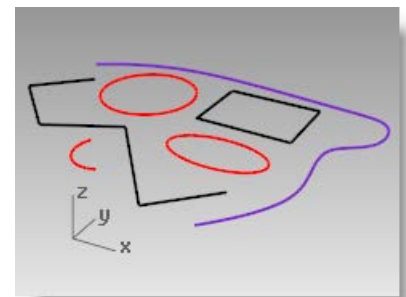


Lightweight extrusion objects

Curve

Another object type that is related to a surface is a **curve**. In Rhino terminology, the word curve includes lines, polylines (think of these as series of straight line segments strung together end to end and joined), arcs, ellipses, circles, or free-form curves that are generally smooth. **Polycurves** are curves made up of two or more curves joined end to end. Separate polycurves into curves by using the **Explode** command.

You can use Curves as inputs for creating and editing surfaces. For example, you can trim a surface or polysurface with a curve, but they may be useful in themselves as well, as in making a 2D drawing of a 3D model, or as reference or construction geometry. You can derive and extract from surfaces. For example, all surfaces have edges, and it is possible to extract an edge curve if needed. You can also extract surface isocurves.



Curves

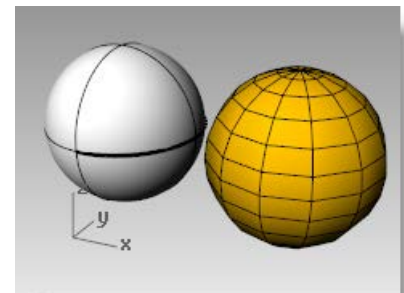
Polygon Mesh

Rhino also creates, edits, and otherwise uses polygon meshes.

Polygon meshes are sometimes used to depict the same type of objects as surfaces but there are important differences. Polygon meshes consist of a number, sometimes a very large number, of points in space connected by straight lines. These straight lines form closed loops of three or four sides, that is, polygons.

One important thing to know about polygon meshes is that the 3D data only exists for these points, or mesh vertices; the space between these points is not considered. Dense meshes are more accurate than very loose ones, but not as accurate as surfaces. Although meshes are important in 3D modeling, initially we will not discuss them.

For instance, if you look at a surface in a shaded viewport, what you see is actually a polygon mesh derived from the surface for the purposes of making a nice image on screen. Typically, you will export mesh data for rapid-prototyped parts. Deriving accurate meshes from surface models is important. Rhino has a number of tools to help accomplish this.



A Sphere surface and mesh sphere object.

2 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, panels 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:

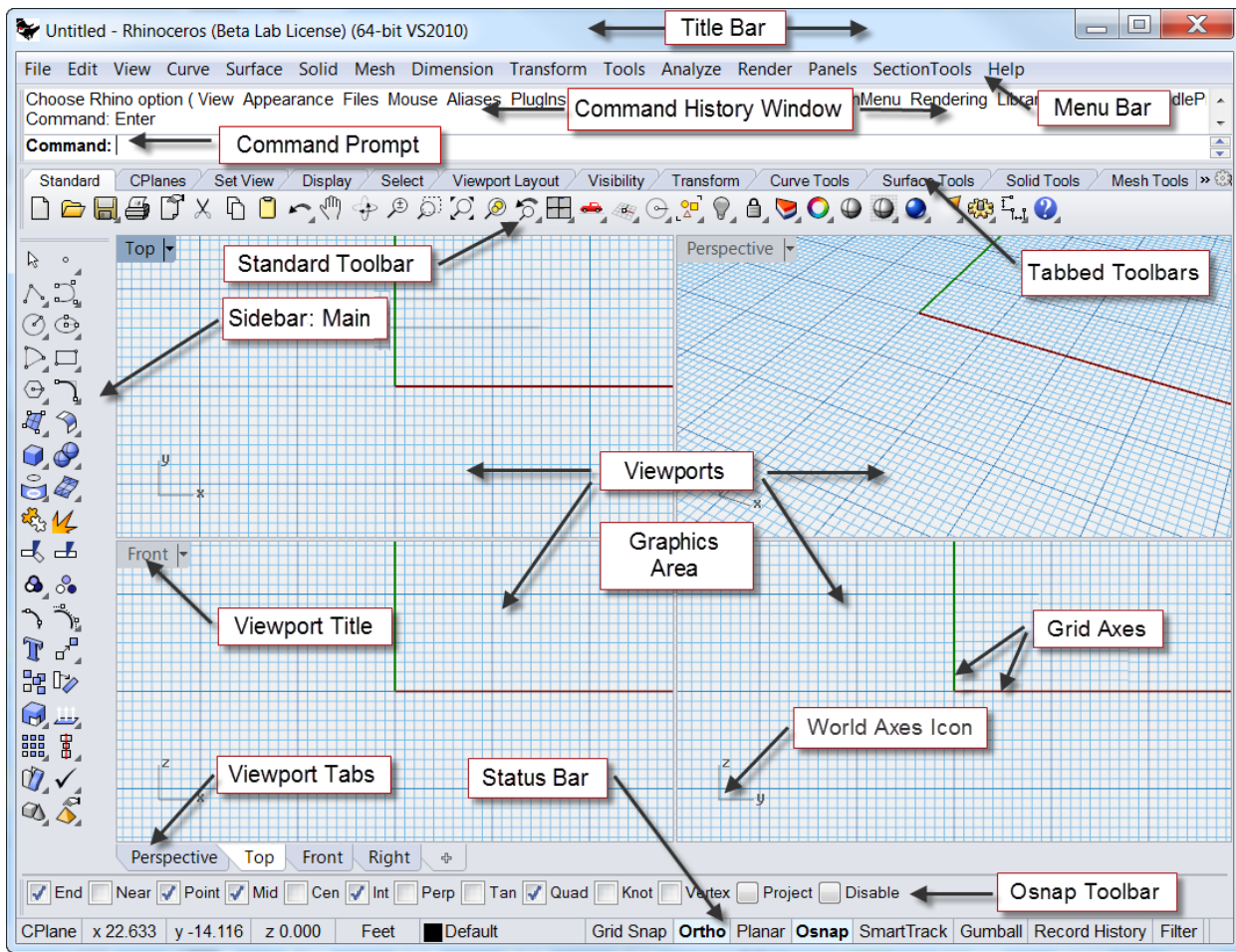
- ▶ Double-click the Rhino icon from the Windows desktop.

The Rhino Screen

Rhino divides its several 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.
Command history window	Displays 500 lines of the most recently used commands. Activate this window with F2.
Tabbed toolbars	Groups are containers with one or more toolbars, with a tab at the top for each toolbar. Floating a toolbar makes single-toolbar group.
Sidebars	Access shortcuts to commands and options. Updates when selecting tabbed toolbar.
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. Viewports can show a grid, grid axes and world axes icon.
Status bar	Displays the coordinates of the pointer, the units and current layer of the model, toggles and other options.
Panels	Rhino controls like layers, properties, materials, lights, display mode and more are displayed in panels with tabs.
Osnap bar	Access running object snap settings

The Rhino Screen



Rhino Main Screen Configuration

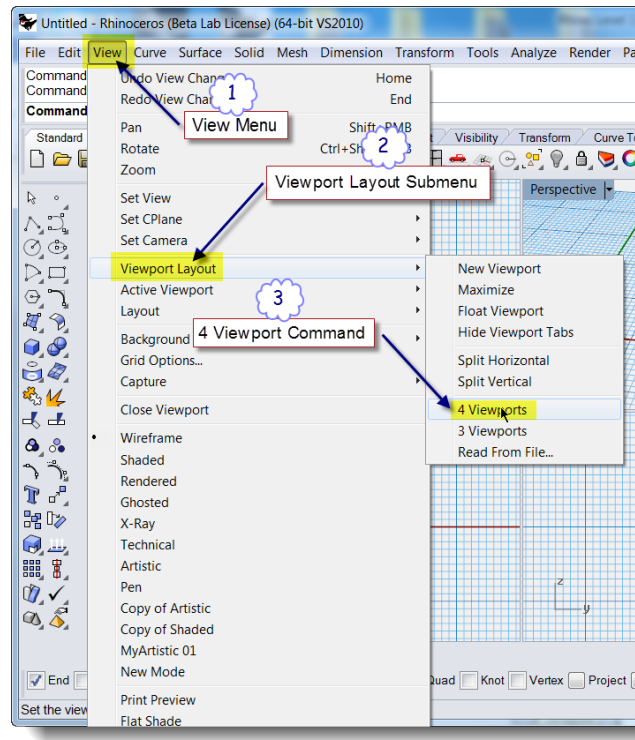
Menus

You can find most of the Rhino commands in the menus.

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 group docked above the graphics area and the Main toolbar as the sidebar on the left.

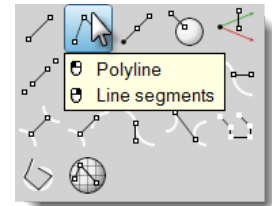


Rhino View menu

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 tooltips indicate which buttons have dual functions.

For example, to start a Polyline, click the Left Mouse Button (LMB), to start the Line Segments command click the Right Mouse Button (RMB).

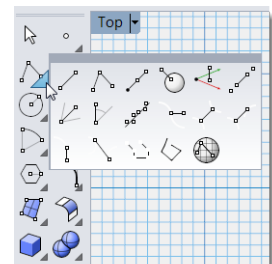
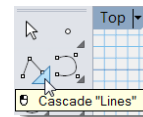


Cascading toolbars

A button on a toolbar may include other command buttons in a cascading toolbar. Usually the cascading toolbar contains variations on the base command. After you select a button on the cascading toolbar, the toolbar disappears.

Buttons with cascading toolbars are marked with a small black triangle in the lower right corner. To open the cascading toolbar, hover over the black triangle and LMB click when the large blue triangle with the tooltip [Cascade "xxx"] appears.

For example, the Lines toolbar is linked to the Main sidebar. After the cascading toolbar is open, you can pick any of the buttons on the toolbar to start a command.



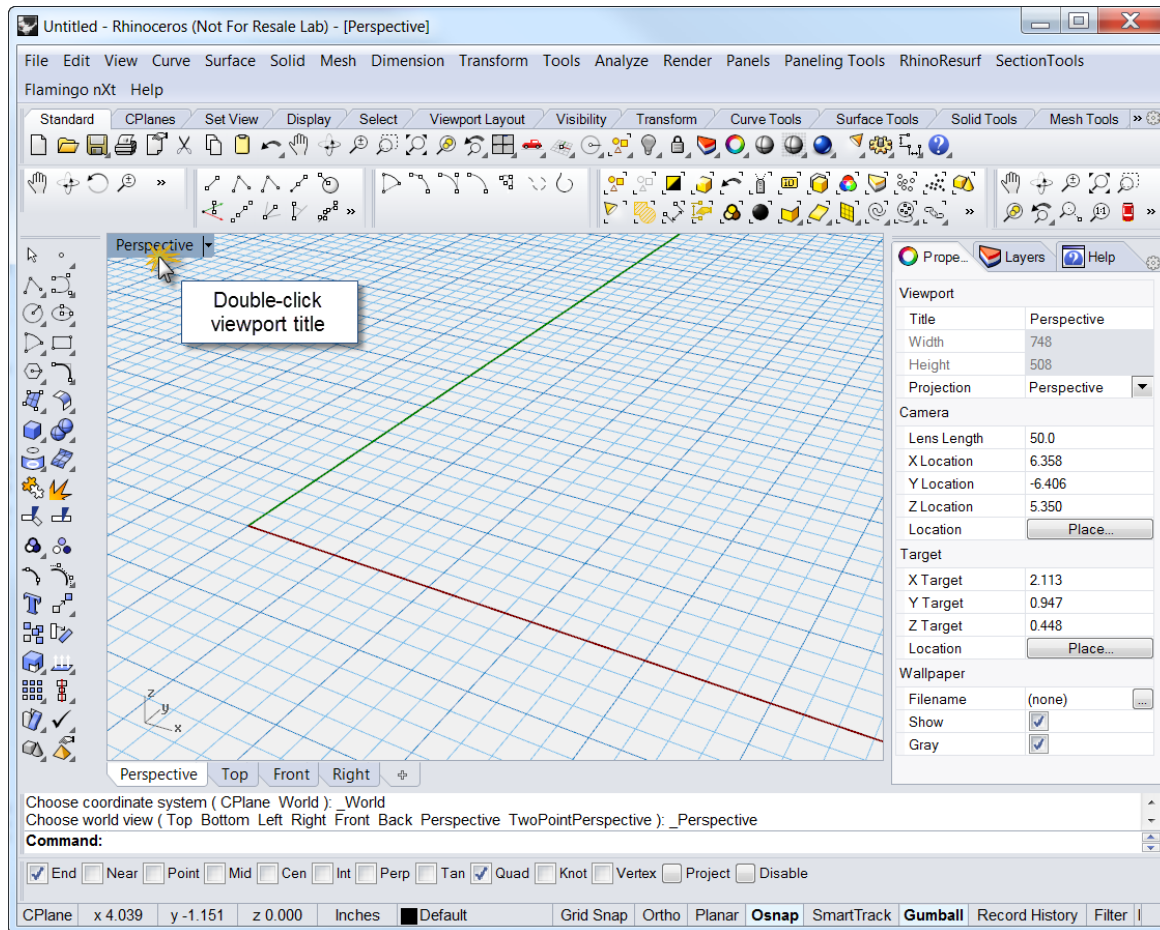
Graphics Area

The Rhino graphics area includes the viewports. You can customize the viewports and their position to suit your preferences. The position of viewports is adjustable.

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.

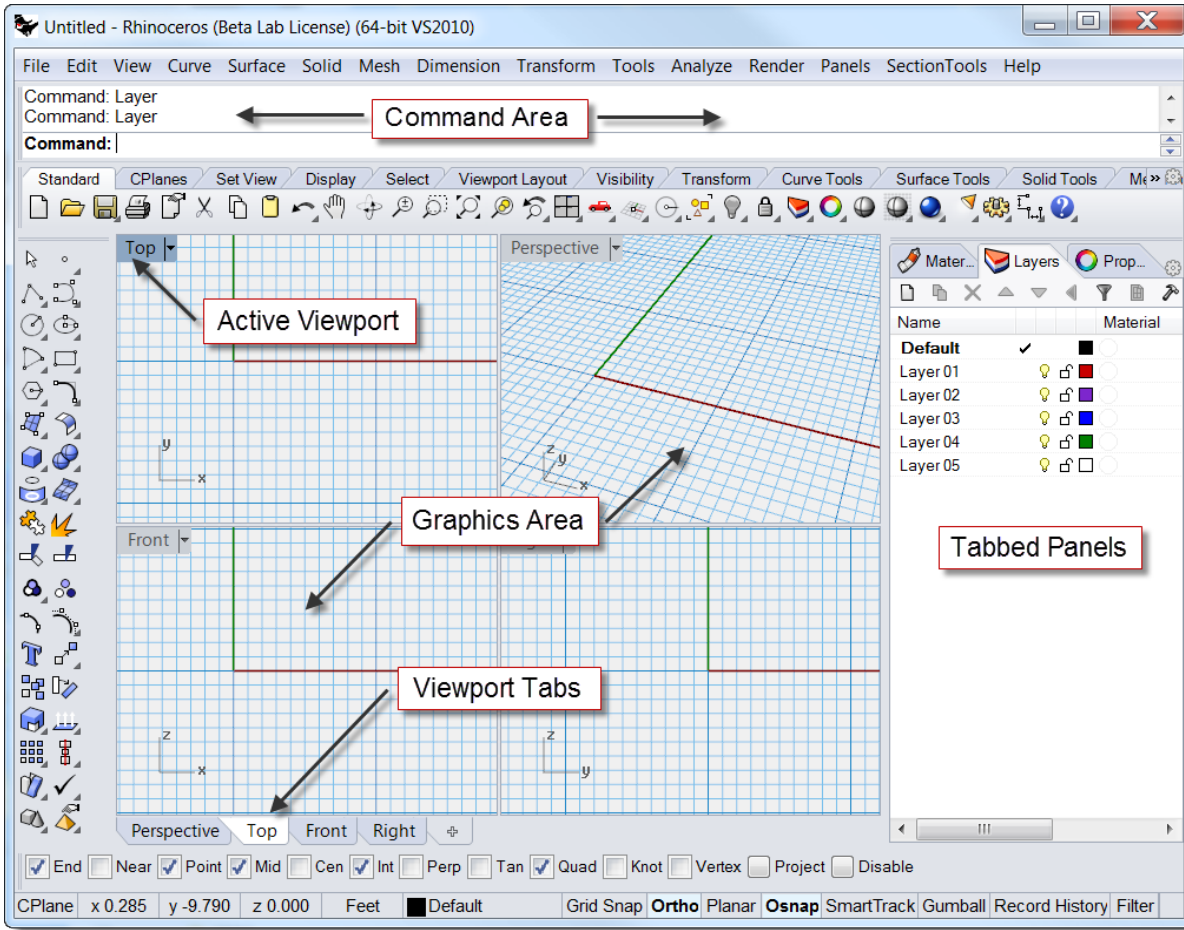
Each viewport has its own construction plane and grid that the cursor normally moves on and a projection mode. To toggle between a small viewport and one that fills the graphics area, double-click the viewport title.



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

Viewport tabs

You can display the viewport titles in tabs if you prefer. The highlighted tab designates the active viewport. Tabs make it easy to switch between viewports when using maximized or floating viewports.



The tabs are located below the graphics area.

Command Area

The command area displays commands and command prompts. You can dock the command line 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. You can select and copy the text in the Command History window 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. Use the right mouse button 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.

Note: Enter and Spacebar perform the same function.

Shortcuts and Aliases

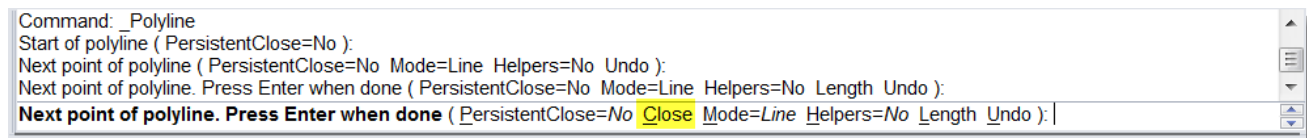
Shortcuts are customizable key combinations. You can program the function keys and **Ctrl** key combinations to perform Rhino commands.

Command aliases run individual Rhino commands or scripts. You can type command aliases just like normal Rhino commands or use them to create abbreviations for often used commands or command scripts.

Rhinoceros Level 2 training discusses Shortcuts and Aliases in detail.

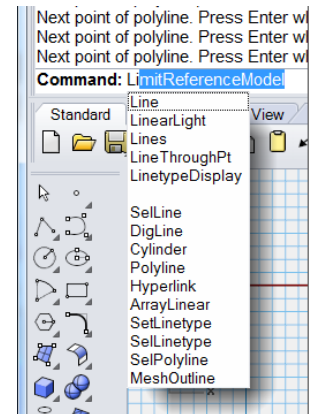
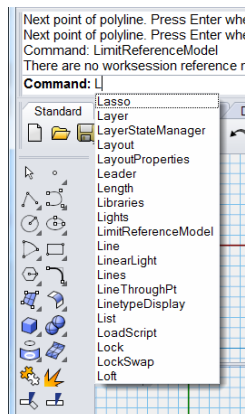
Clickable options

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

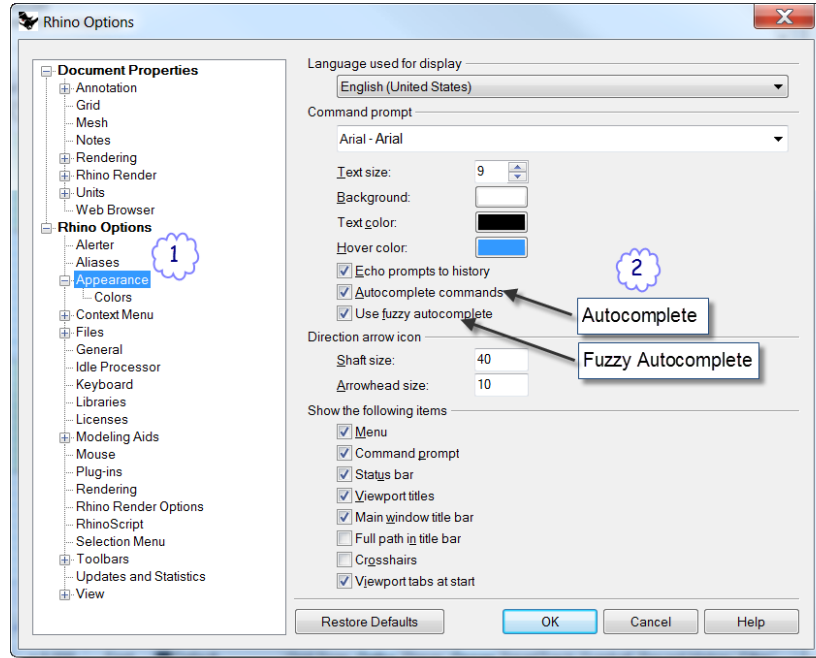
Type the first letter of a command name to activate the autocomplete command list. As you type letters on the command line, the command name that includes those letters will autocomplete and display in a drop down menu. Press *Enter* to activate the command once the full command name appears or left click on the command in the list to start it.



When enabling the "Use fuzzy autocomplete" feature in Options and Appearance, the in-line autocomplete suggests the most used candidate.

For example, LI most likely autocompletes to Line rather than something like LimitReferenceModel and the autocomplete menu contains best partial and inexact matches.

When disabled, autocomplete uses the first letters of the command name in an alphabetical list.



Options -> Appearance


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

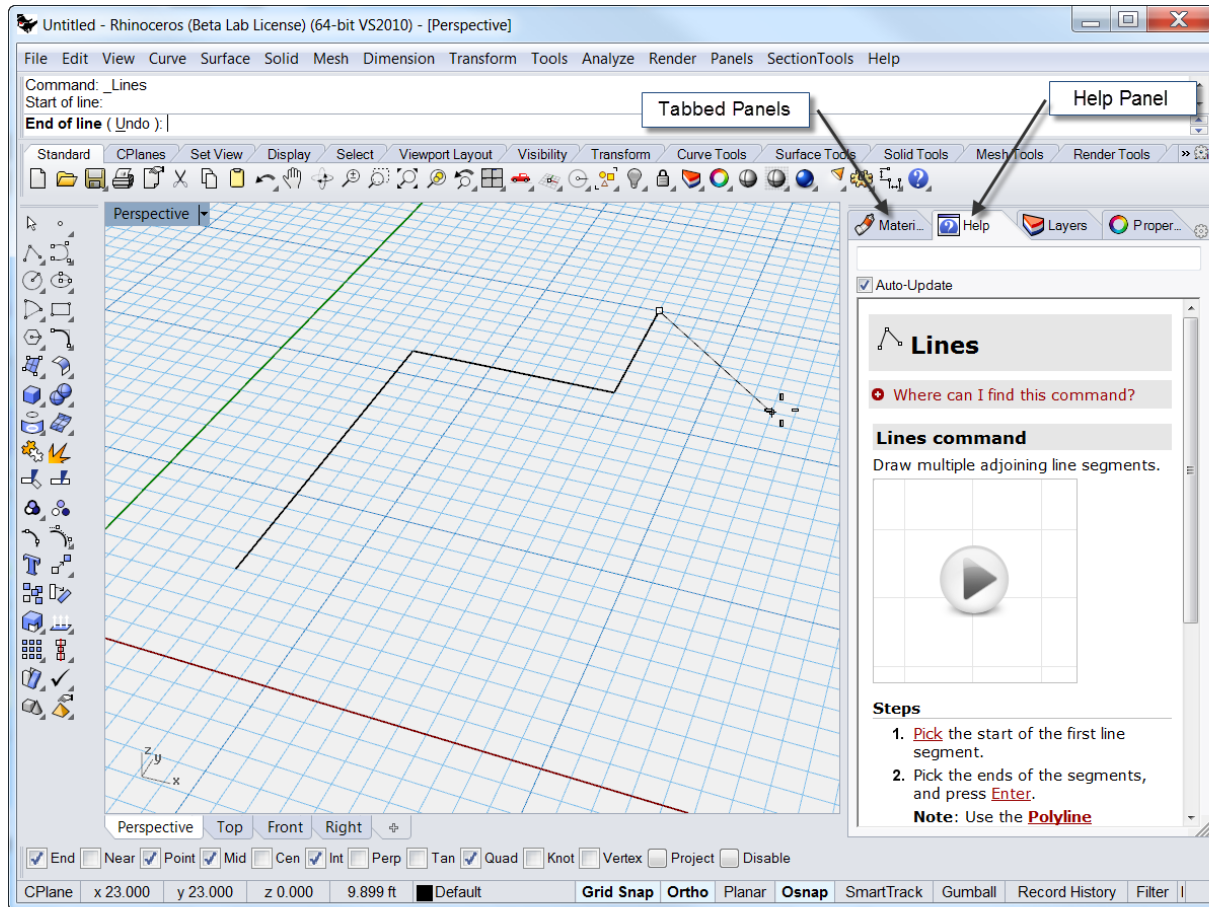
To cancel a command, press **Esc** or enter a new command from a button or a menu.

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.

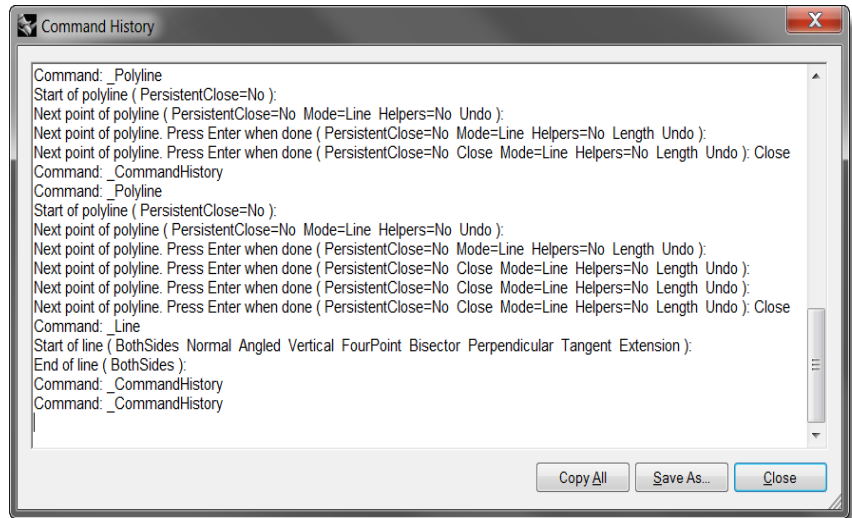
Most of the commands include short video clips that show how the command and the options work.



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.

View the Command Line History

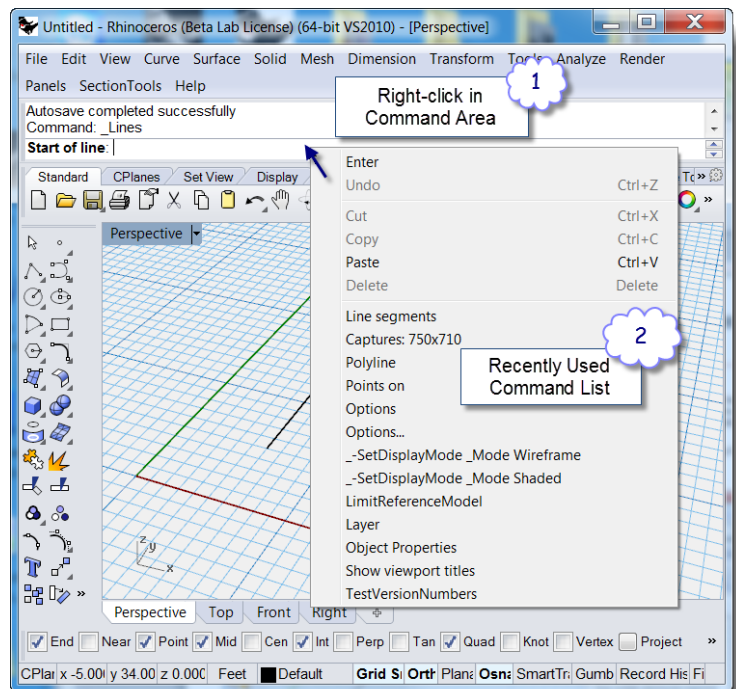
The command history window lists the last 500 command lines from the current Rhino session. Press **F2** to view the command history.



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.



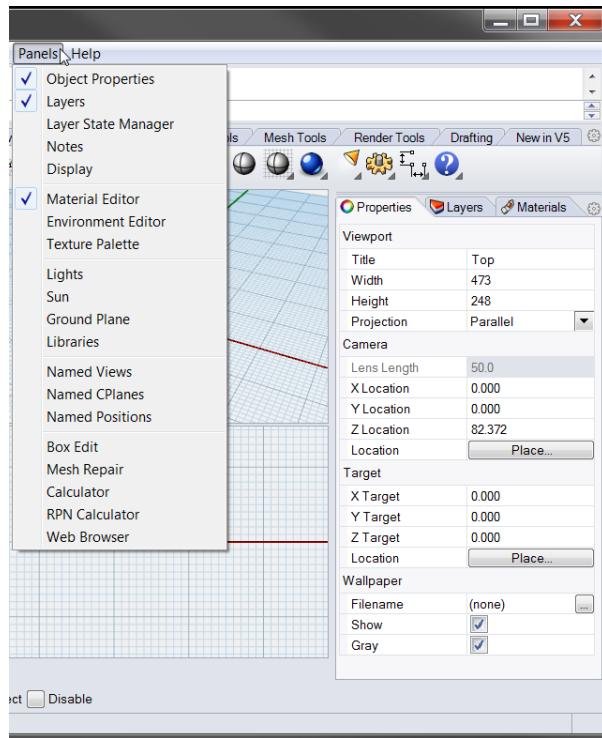
Panels

Many Rhino controls are contained in tabbed panels.

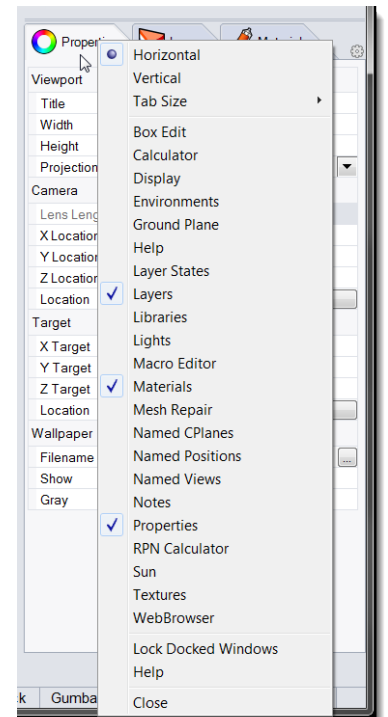
Open the panels from the Panels menu or from the tabbed Panels right-click menu.

Panels that you will work with in Level 1 training are:

- Display
- Layers
- Help
- Named CPlanes
- Notes
- Properties
- Web browser



The Panels menu shows the docked panels on the right side of graphics windows.



The tabbed panels right-click menu.

Note: Positioning the mouse over the tabs allows the mouse wheel to scroll through the tabs.

Exercise 1—Rhino basics

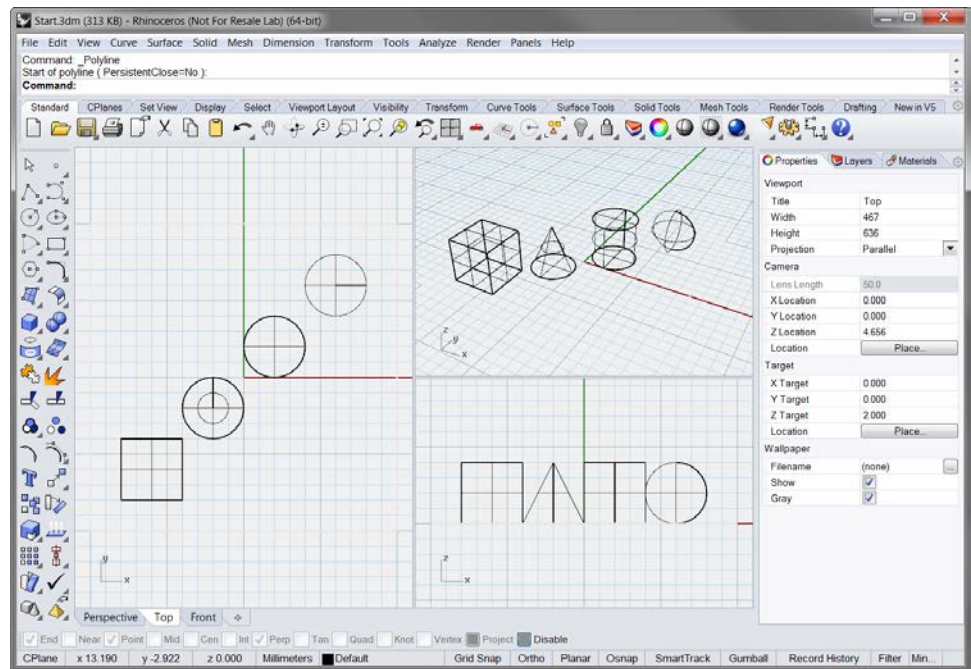
To get started:

- 1 Download the Level 1 Training models.
- 2 Unzip the downloaded file into a **Level 1** folder on your computer.

After downloading the Level 1 Training.zip file to your computer, unzip files into a Level 1 folder located in Desktop or My Documents, or another location in which you have full rights.

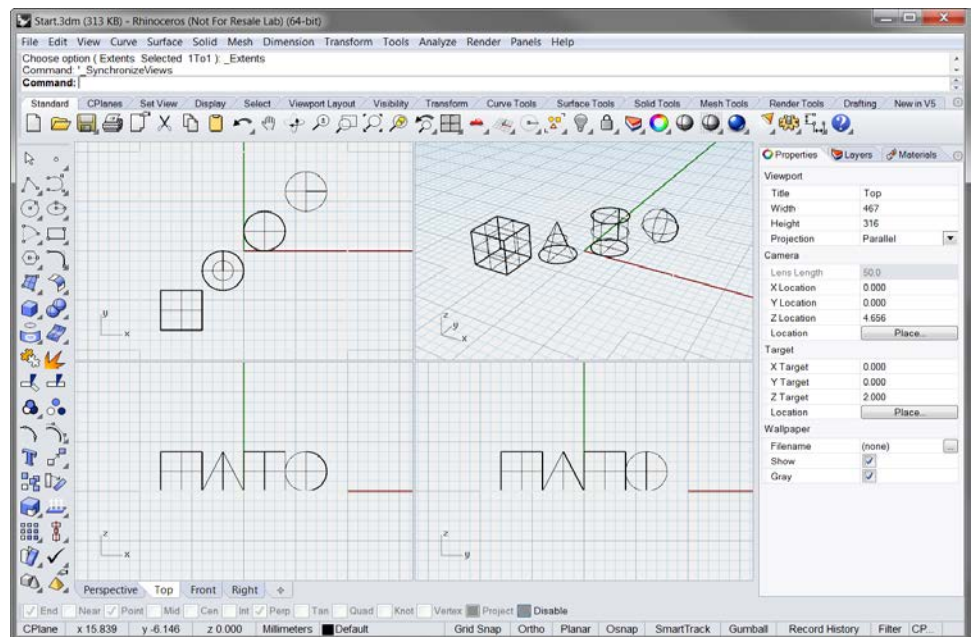
- 3 From the **File** menu, click **Open**.
- 4 In the **Open** dialog box, navigate to the **Level 1** folder and **Open** the model **Start.3dm**.

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



Two parallel viewports and one perspective viewport.

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



Three parallel viewports and one perspective viewport

- 6 In the **Status Bar**, click **Grid 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, it will be bold and black in the status bar. If it is off, the word it will be gray.



Note: This is an important step. Grid Snap only lets your cursor move in certain intervals. In this model, the setting for Grid Snap is one-half of a grid line. Grid Snap helps you line up your objects as if you were building with LEGO® blocks.

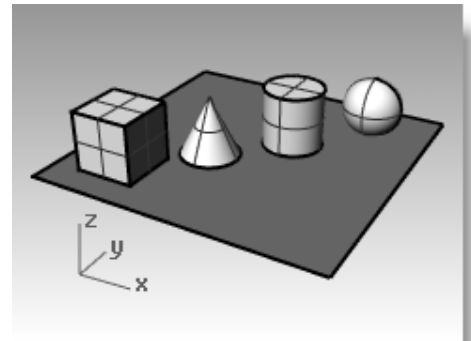
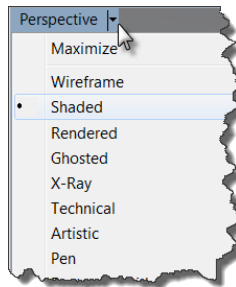
- 7 Click the mouse in the Perspective viewport to make it active.

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

- 8 Click the arrow icon to the right of the **Perspective** title or Right Mouse click on the title to open the drop down menu, 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.



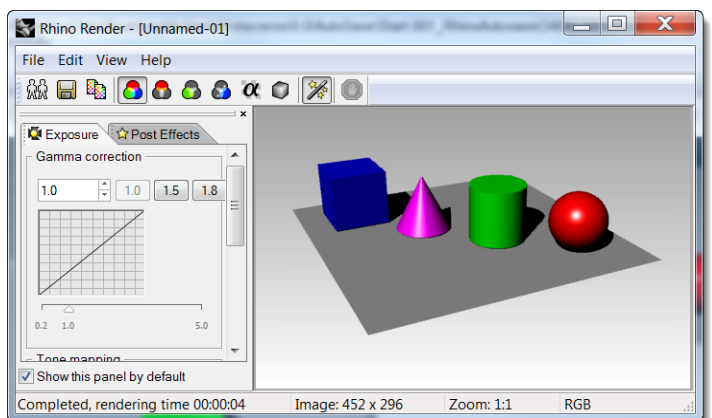
The viewport shows a Shaded display.

- 9 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 you can save image to a file.

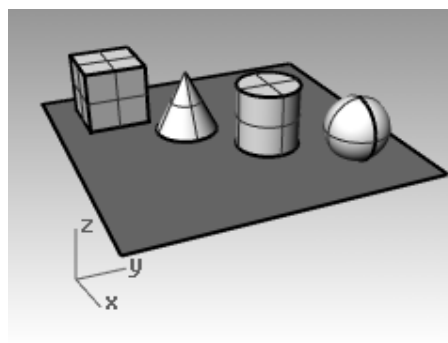
- 10 Close the render window.



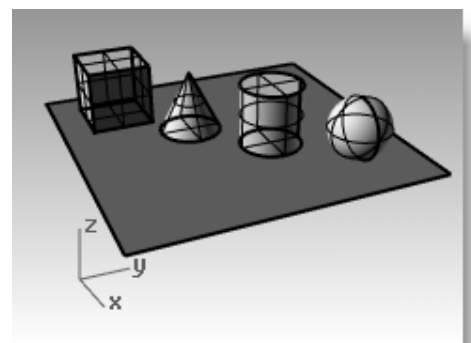
Render

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

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

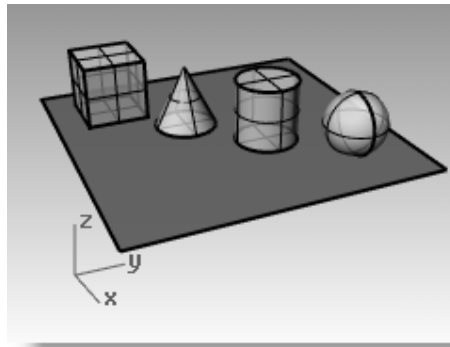


Rotate the view in shaded display.

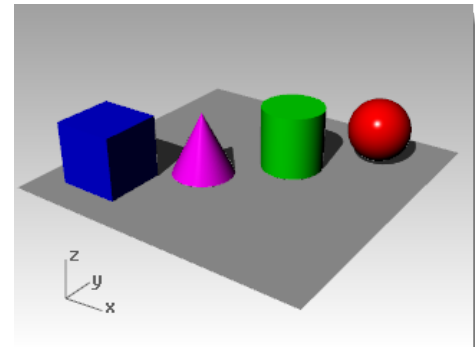


X-Ray shaded display.

- 12 Click the arrow icon to the right of the Perspective viewport title, and then click **X-ray**.
- 13 Click the arrow icon to the right of the Perspective viewport title, and then click **Ghosted**.
- 14 Click the arrow icon to the right of the Perspective viewport title, and then click **Rendered**.



Ghosted shade display.

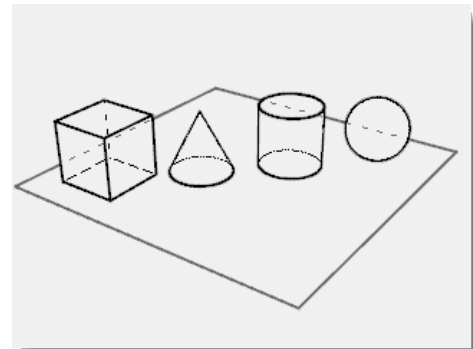


Rendered display

- 15 Click the arrow icon to the right of the Perspective viewport title, and then click **Technical**, **Artistic**, and **Pen**.

Technical mode displays lines as though the drawing were a 2-D, flat drawing on paper.

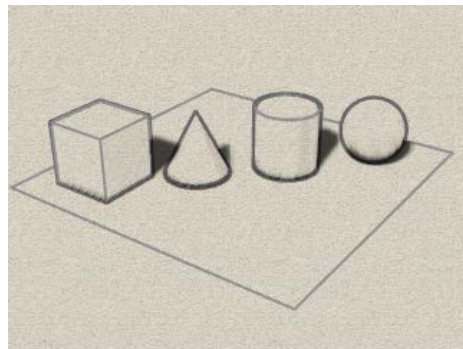
This mode uses real-time silhouettes and intersections, creases, borders, blended shaded and rendered display. Objects behind other objects appear occluded.



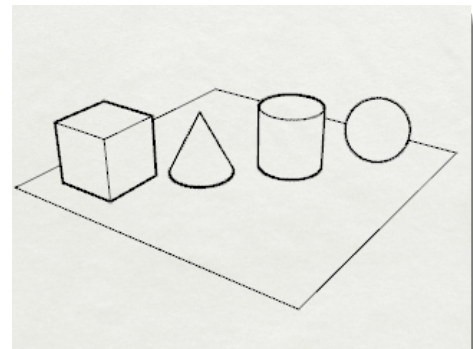
Technical display

Artistic mode displays lines as though the drawing were a 2-D, flat, pencil drawing on textured paper.

Pen mode displays lines as though the drawing were a 2-D, flat, pen drawing on paper.



Artistic display



Pen display

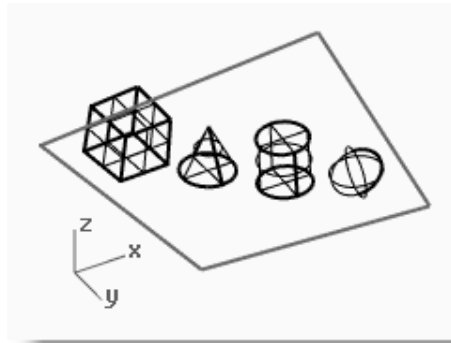
16 Change to **Wireframe** mode.

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

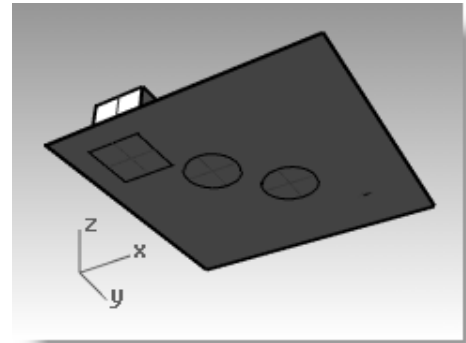
You are now under the objects looking up.

18 Change to **Shaded** mode.

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



This view looks at the objects from the bottom in wireframe mode.



This view looks at the objects from the bottom in shaded mode.

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

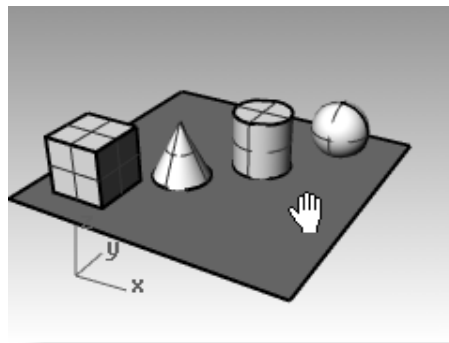
You have used the right mouse button to rotate in the Perspective viewport. You can hold **Shift** and drag with the right mouse button to pan. Dragging the right mouse button to move around does not interrupt any commands in progress.

To pan in a viewport:

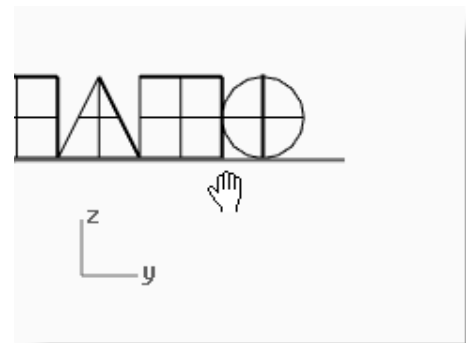
- 1 In the **Perspective** viewport, hold the **Shift** and drag with the right mouse button to pan the view.
- 2 Pan the view in the parallel viewports by dragging with the right mouse button.

The Top, Front, and Right viewports use a parallel projection

In the parallel viewports, it is not necessary to press the **Shift** key.



*Pan the viewport using **Shift** and the right mouse button.*



Pan the viewport in a parallel view with the right mouse button.

Zooming in and out

Sometimes you want to get closer to your objects or move back so you can see more. This is 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 do not have a wheel mouse, hold down the **Ctrl** key and drag up and down in a viewport with the right mouse button.

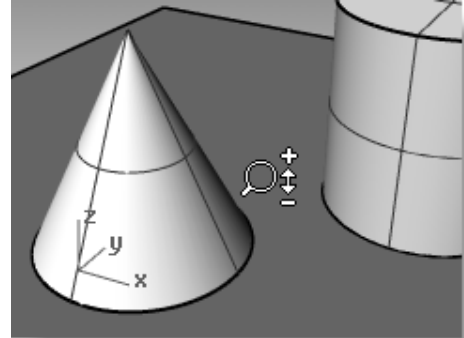
To zoom in and out:

- 1 In the Perspective viewport, roll the wheel on your mouse forward to zoom in, roll it backward to zoom out.
- 2 In the Perspective viewport, hold the **Ctrl** key, click and hold the right mouse button, and drag the mouse up and down.

The camera zooms at cursor position.

Drag up to zoom in.

Drag down to zoom out.



*Zoom the viewport using the **Ctrl** key 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**.

Move Objects

If **Gumball** is bold face in the **Status bar**, click to turn Gumball off. We will discuss Gumball later in this training.

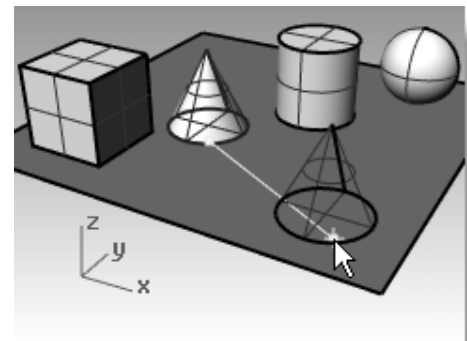
Dragging follows the construction plane of the current viewport.

Now drag the objects around. You can drag in any viewport. In this model, the Grid Snap setting is 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 selected cone highlights.

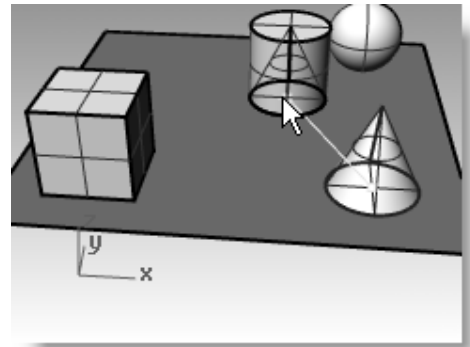


The selected cone highlights.

- 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 represented by the grid. This base is 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.

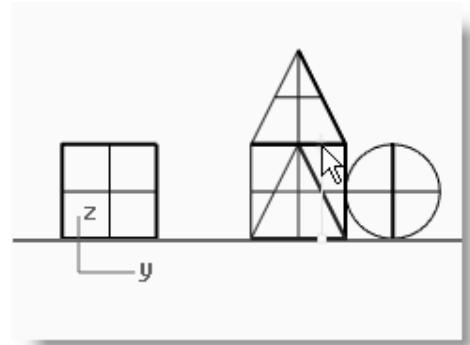


Drag the cone to move it.

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

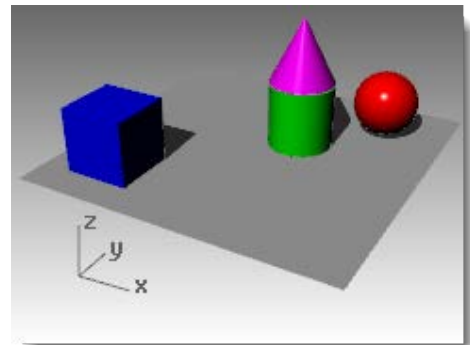
Watch what happens in the Perspective viewport.

Watch the other viewports to 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 with Shadows.

Copy Objects

To create more objects, copy the shapes.

To open the same model again:

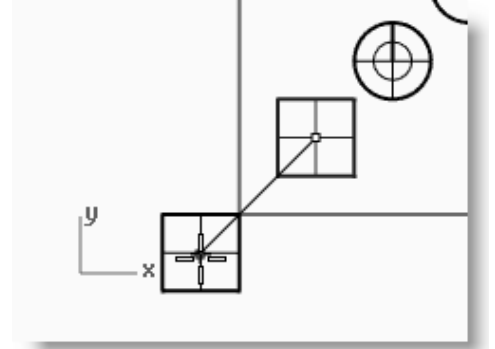
- 1 From the **File** menu, click **Open**.
- 2 Do not save the changes.
- 3 In the **Open** dialog box, select **Start.3dm**.

To copy objects:

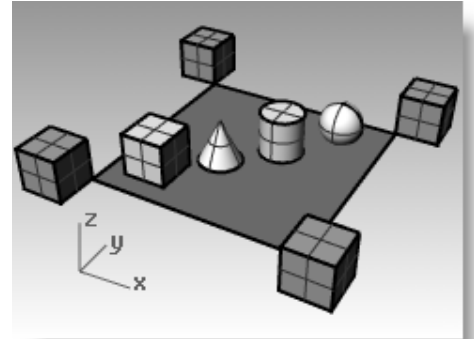
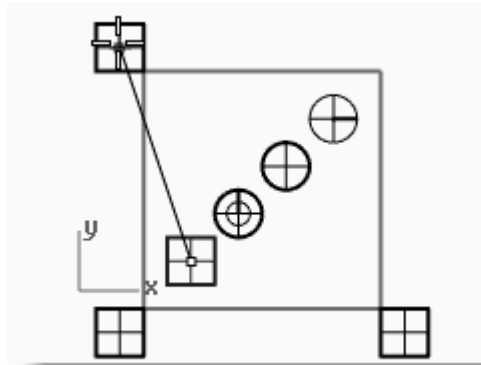
- 1 Click the box to select it.
- 2 From the **Transform** menu, click **Copy**.
- 3 Click somewhere in the Top viewport.

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

Select and copy the box.

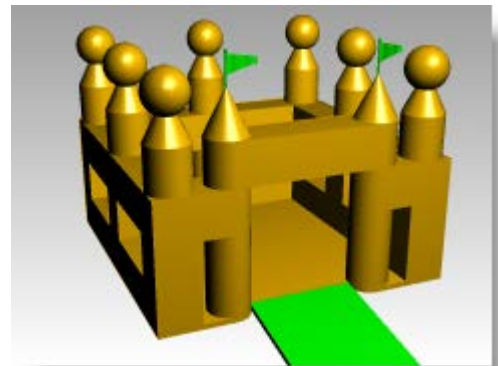


- 4 Click where you want the first copy. Zoom in closer if you like.
- 5 Click other places to make more copies of the box.
- 6 When you have enough copies, press **Enter**.



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.

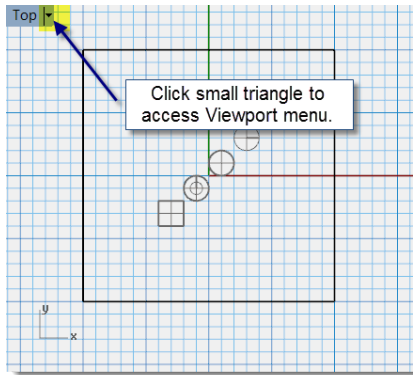
Each view corresponds to the view through a camera lens. The invisible target of the camera is located in the middle of the viewport.

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. Or Click the small triangle on 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, for **Projection** 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:

Perspective Projection			Parallel Projection
Key	Action	Action + Ctrl	Action
Left Arrow	Rotate left	Pan left	Pan left
Right Arrow	Rotate right	Pan right	Pan right
Up Arrow	Rotate up	Pan up	Pan up
Down Arrow	Rotate down	Pan down	Pan down
Page Up	Zoom in		Zoom in
Page Down	Zoom out		Zoom out
Home	Undo View Change		Undo View Change
End	Redo View Change		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.

Reset 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**.

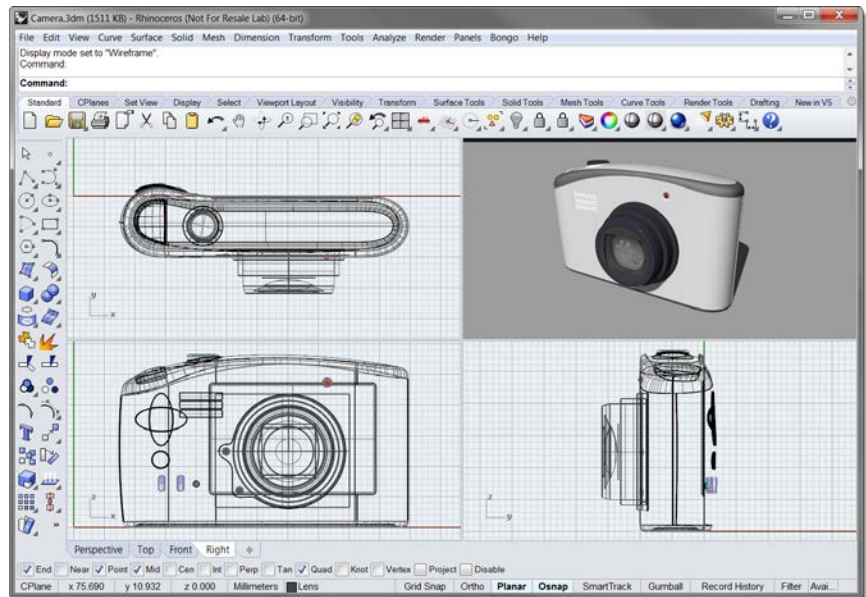
To bring all your objects into view in all viewports:

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

Exercise 2—Display options

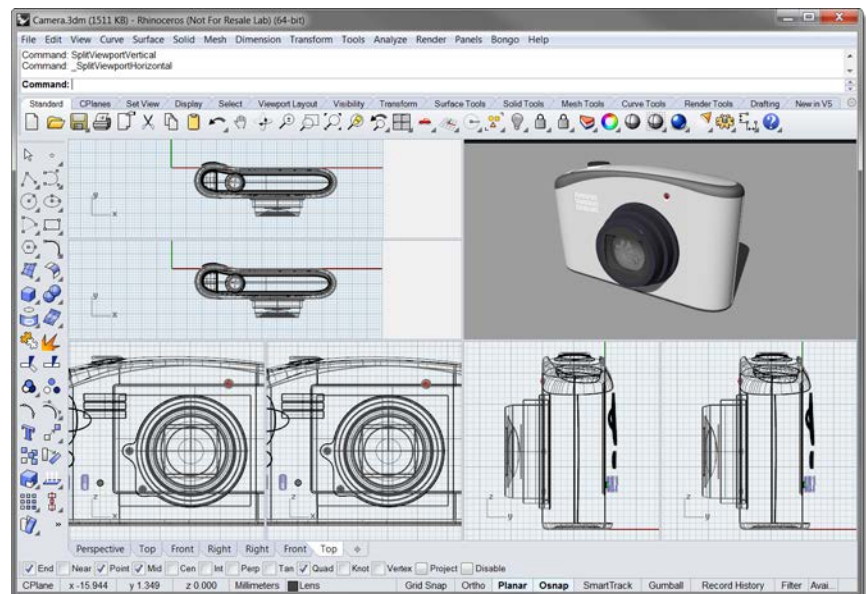
- Open the model **Camera.3dm**.

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






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**, and then click **Bottom**.



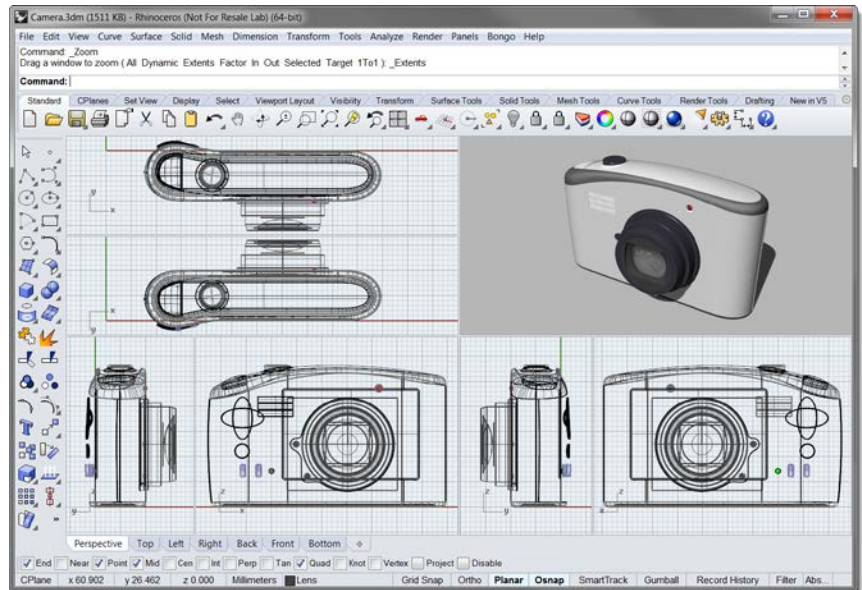
Three viewports are split down the middle either horizontally or vertically.

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 button, and drag the intersection in any direction. If several viewports touch at that corner, all resize.

To synchronize the viewports:

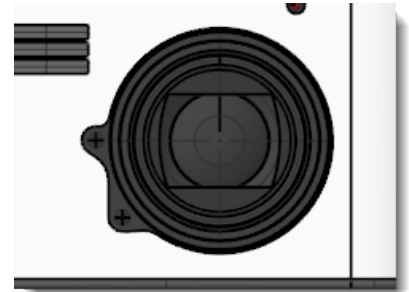
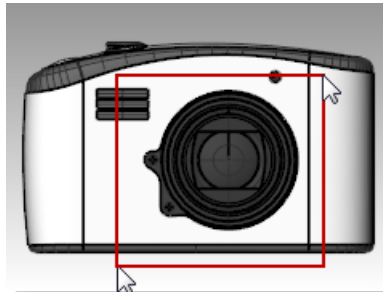
- 1 Adjust the size or shape of the viewports.
- 2 Make one of the Front viewports active.
- 3 From the **View** menu click **Zoom**, and then click **Zoom Extents**.
- 4 Right click on the Front viewport title, click **Set Camera**, and then click **Synchronize Views**.
- 5 Change the viewport displays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



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

To zoom to a window:

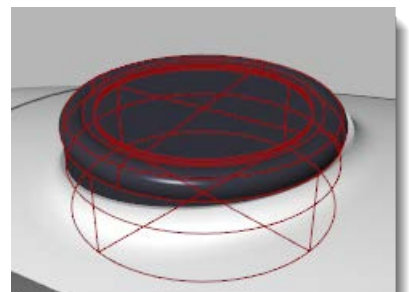
- 1 From the **View** menu click **Zoom**, and then click **Zoom Window**.
- 2 Click and drag a window around a portion of the model.



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.



To rotate the view:

- 1 In a perspective viewport, drag with right mouse button.
- 2 In a parallel viewport, from the View menu, click Rotate or use **Ctrl** + Shift and drag with the right mouse button.

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 and Editing Geometry

3 Creating Geometry

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 or arc joined segments (a single linear curve with multiple segments).

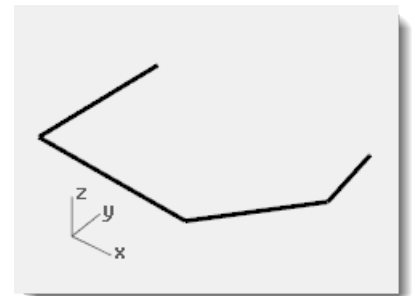
Option	Description
Close	Closes the shape by drawing a segment from the last point picked to the first point picked. This ends the command.
Undo	Deletes the last point picked.

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**.

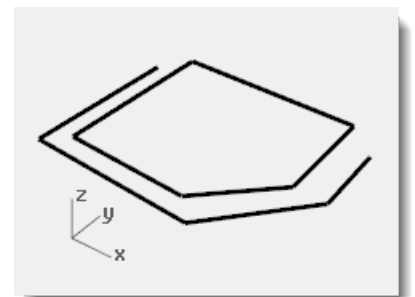
To draw line segments:

- 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 Continue to pick points.
Additional segments appear.
- 5 Press **Enter** to end the command.
The segments meet at a common point, but are not joined.



To use the Close option:

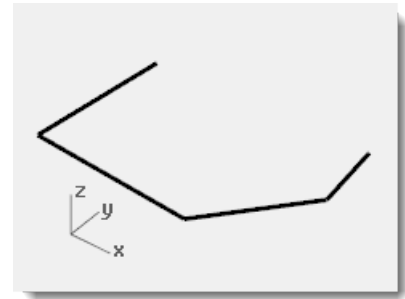
- 1 Repeat the **Lines** command.
- 2 Pick a **Start point** and 3 or 4 more points.
- 3 Click **Close** on the command line.
This is one of the clickable options discussed in the previous chapter.
The last line segment will end at the original start.



To draw a polyline:

- 1 Use the **Undo** command to undo the segments you just created.
- 2 From the **Curve** menu click **Polyline**, and then click **Polyline** to begin the Polyline command.
- 3 Pick a **Start point** and 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.

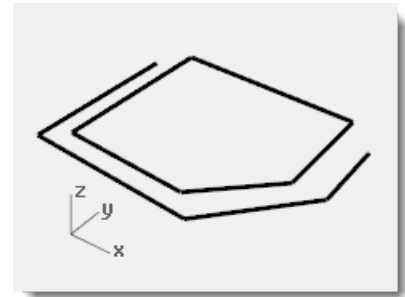


To use the Undo option:

- 1 Repeat the **Polyline** command.
- 2 Pick a **Start point** and 3 or 4 more points.
- 3 Click **Undo** on the command line.
- 4 Continue to pick points.
- 5 Click **Close** to end the command.

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

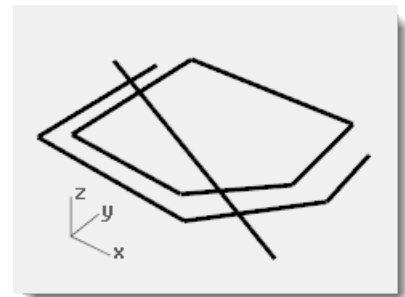
This makes a closed polyline.



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**.

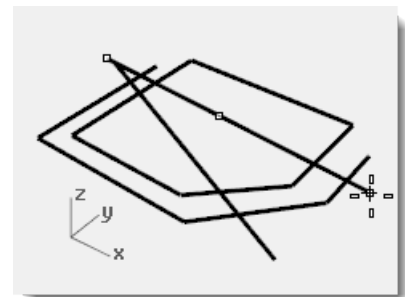
The command ends after one segment is drawn.



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.



Drawing free-form curves

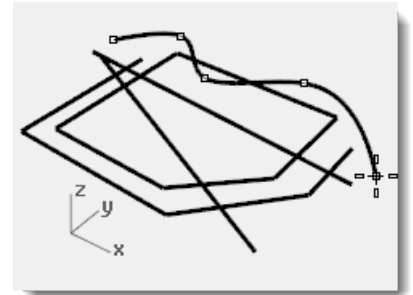
The InterpCrv and Curve commands draw free-form curves. The InterpCrv command draws a curve through the points you pick. The Curve command uses control points to create a curve.

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.

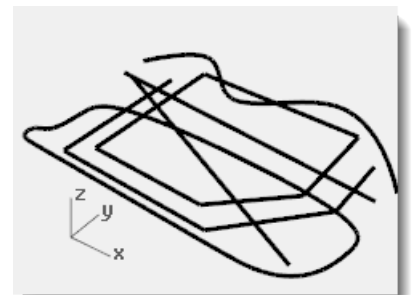
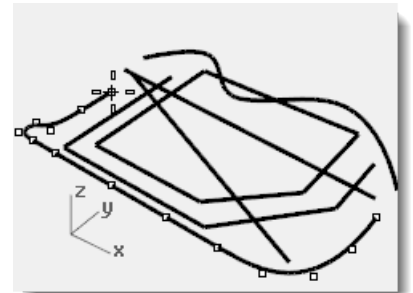
Exercise 4—Drawing interpolated curves

- 1 From the **Curve** menu click **Free-form**, and then click **Interpolate Points**.
- 2 Pick a **Start point**.
- 3 Continue picking points.
Notice that with this command the curve goes through each point that you pick.
- 4 Press **Enter** to end the command.
This makes an open curve.



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.
Notice that most of the points you pick are off the curve as control points.
- 4 Click **Close** to make a closed curve.



Modeling aids

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

Grid Snap	Ortho	Planar	Osnap	SmartTrack	Gumball	Record History	Filter	,
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Click the **Grid Snap**, **Ortho**, **Planar**, **Gumball**, **SmartTrack** or **Record History** panes on the status bar to toggle these modeling aids on and off.

Grid Snap

This modeling aid constrains the marker to snap on grid intersections.

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

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

Ortho

This modeling aid constrains the marker 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.

Osnap

Object snaps constrain the marker to an exact location on an object such as the end of a line or the center of a circle.

SmartTrack

This modeling aid uses temporary reference lines and points that are drawn in the Rhino viewport using implicit relationships among various 3-D points, other geometry in space, and the coordinate axes' directions.

Planar

This is a modeling aid similar to Ortho. This helps you model planar objects by constraining the 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 and off by typing the letter *P* and pressing **Enter**.

Gumball

Displays a display widget, on a selected object, facilitating move, scale, and rotate transformations around the gumball origin.

Record History

Records history and updates history-aware objects. With History recording and Update turned on, a lofted surface, for example, 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.

Filter

This modeling aid restricts any selection mode to specified object types. Objects types that can be restricted are: Annotations, Blocks, Control points, Curves, Lights, Meshes, Point clouds, Points, Polysurfaces, Surfaces, Hatches, and others.

Exercise 6—Drawing lines and curves using mode functions

- 1 Toggle **Grid Snap** on and draw some lines.

In this model the marker snaps to each grid intersection. This is because in the template file we're using the *SnapSize* and the *Grid-MinorLineSpacing* is set to the same value (1).

- 2 Toggle **Grid 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 Grid Snap and Ortho toggles you can draw with precision. We will discuss other ways to get precision in a later session.

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.

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.

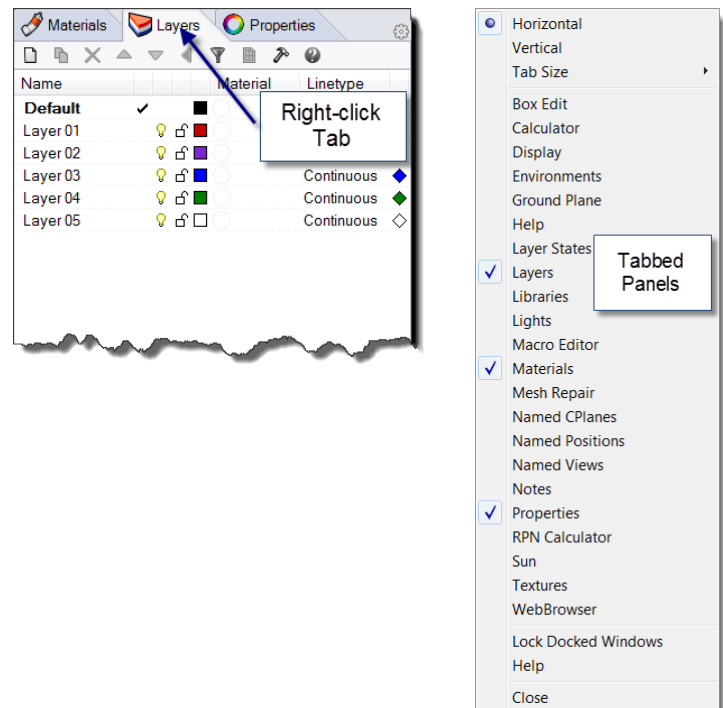
Command	Description
Save	Saves your model and keeps it open.
SaveSmall	Save your model without render or analysis meshes and preview image to minimize file size.
IncrementalSave	Save sequentially numbered versions of your model.
SaveAs	Saves your model to a specified file name, location, and format.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

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 panel manages layers. Use it to set up layers for your model. The Layers panel can be floating or docked with the other tabbed panels like Properties, Command Help, Lights, Notes, and more.



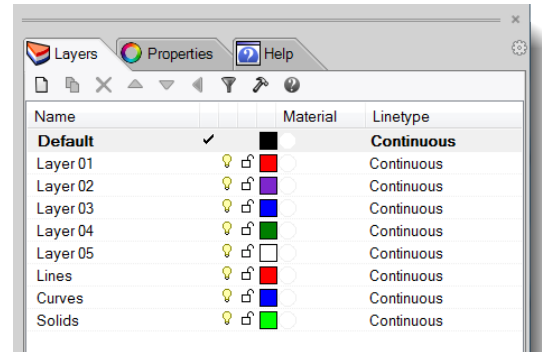
Exercise 7—Layers

To create a new layer:

- 1 From the **Edit** menu click **Layers**, and then click **Edit Layers**.
- 2 In the **Layers** panel, click the **New icon**.
- 3 The new layer, **Layer 06**, appears in the list. Type **Lines** and press **Tab**.

Use the **Tab** key to quickly add layers.

- 4 The new layer, **Layer 06**, appears again. Type **Curves** and press **Tab**.
- 5 The new layer, **Layer 06**, appears. Type **Solids** 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 square **Color** patch on the **Lines** 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.

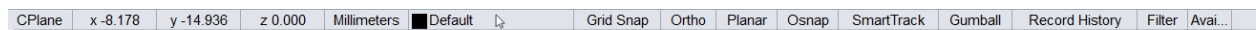
R, G, and B are the red, green and blue components of the color.

- 3 Click **OK**.
- 4 In the **Layers** panel, the new color appears in the color patch on the **Lines** row of the layer list.
- 5 Repeat steps 1–3 to make the **Curves** layer Blue.
- 6 Click **OK** to close the dialog box,



To make a layer current:

- 1 In the **Status Bar**, click the **Layer** pane.



The Status Bar-Layer pane displays the current layer (Default).

- 2 In the **Layer** popup, click **Lines**.

The current layer is displayed in the Layer pane.

- 3 Draw some lines.

The lines are on the Lines layer and they are colored red.

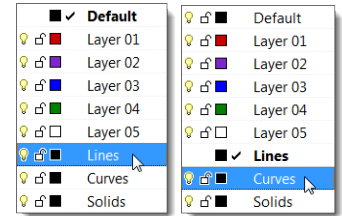
- 4 To make a different layer current, click the **Layer** pane again.

- 5 Click **Curves**.

- 6 Draw some curves.

They are on the Curves layer and are colored blue.

- 7 Draw more lines and curves on each layer.



Clicking the name or the checkbox sets the current layer.

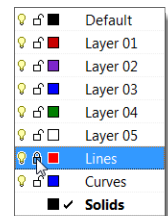
Note: If objects are highlighted, and a layer is selected from the Status bar, the highlighted objects will be changed to the selected layer and the current layer does not change.

To lock a layer:

- 1 In the **Status Bar**, click the **Layer** pane, click **Solids** to make the **Solids** layer current.

- 2 In the **Layer** popup, click the **Lock icon** on the row for the **Lines** 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.

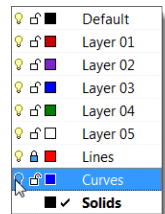


To turn a layer off:

- 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 the **Curves** layer.

Turning a layer off makes all objects on it invisible.



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 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.

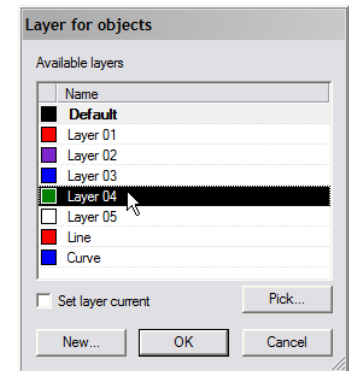
To unlock locked objects:

- From the **Edit** menu click **Visibility**, and then click **Unlock**.

The **Unlock** command unlocks 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**.



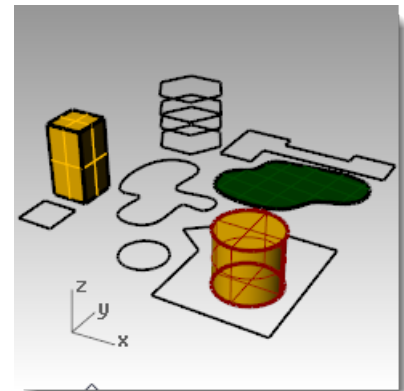
Selecting objects

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

Exercise 8—Practice using selection options

- 1 **Open** the model **Delete.3dm**.
- 2 Select the square and the circle.
- 3 From the **Edit** menu click **Delete** or press the **Delete** key.

The objects disappear.



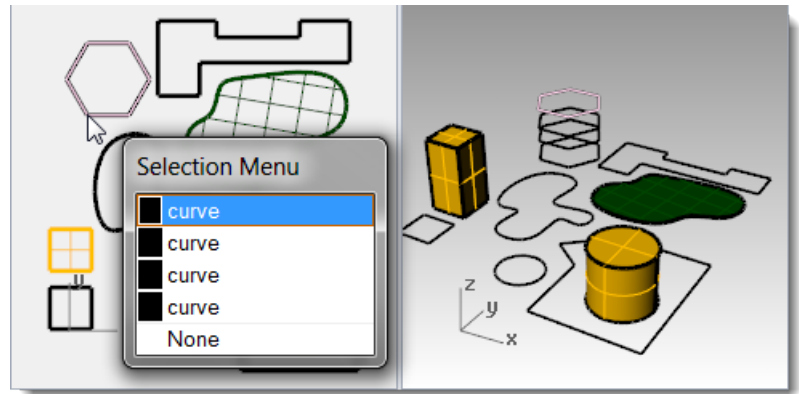
To select objects for deletion:

- 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.

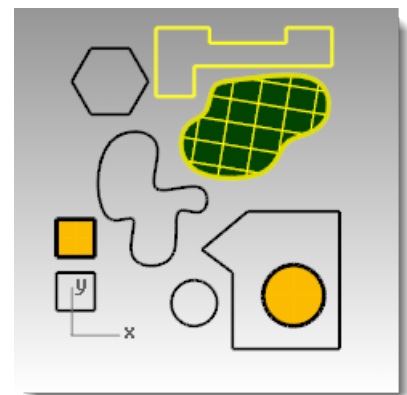
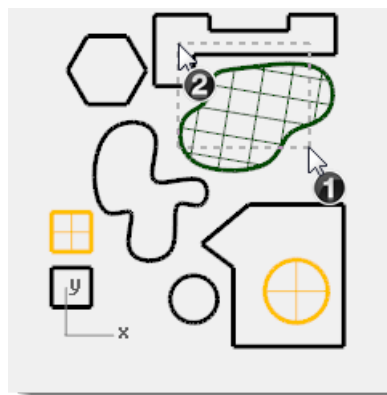


- 4 In the Top viewport, use a crossing box to select the surface and the polyline in the top right part of the drawing.

A crossing box is used when selecting in an open area and then dragging the window from right to left.

Both objects are selected.

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

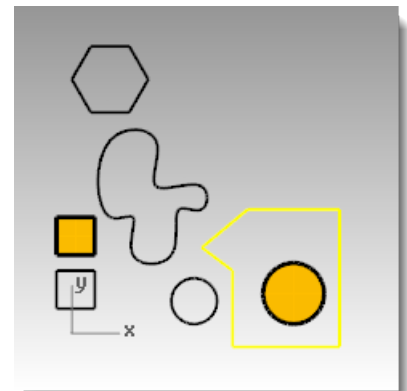
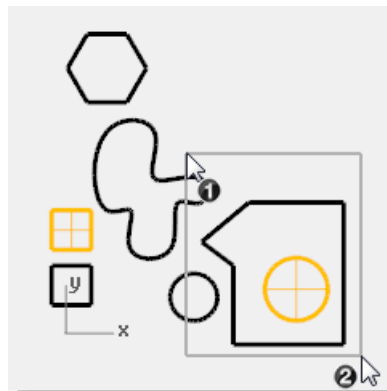


- 6 Make a window to select the polyline and the cylinder in the lower right part of the drawing.

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.

To undo and redo deletions:

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

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

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

Each time you click, the previous Undo is reinstated.

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

Dedicated selection commands

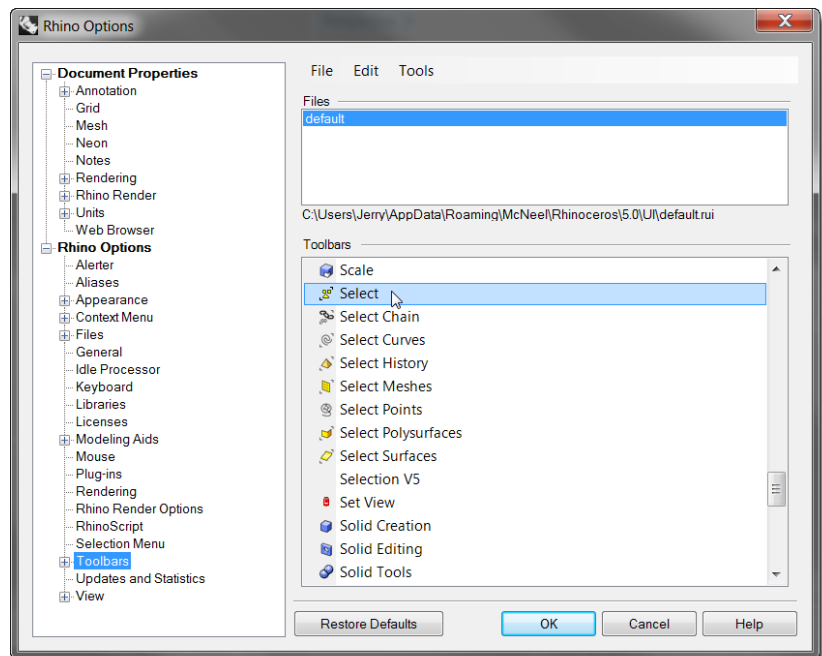
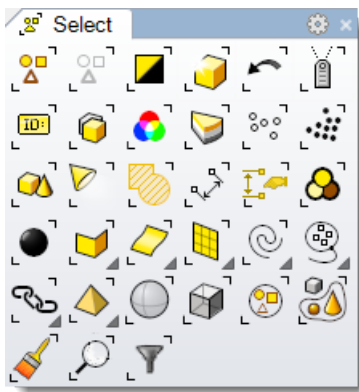
In addition to the selection options we just practiced, there are several other useful tools for selecting objects. In the next exercise we will use some of these tools.

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 access select tools from the Select Toolbar:

- From the **Tools** menu, click **Toolbar Layout**.
- Select **Default** under Files.
- Choose Select as the toolbar to show.

The Select toolbar appears.



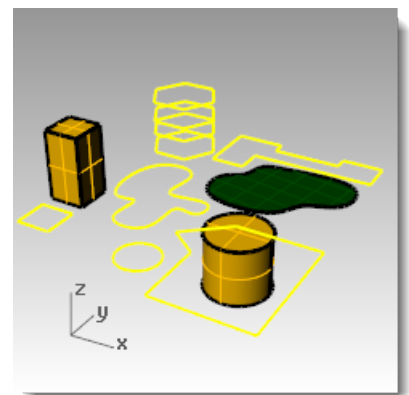
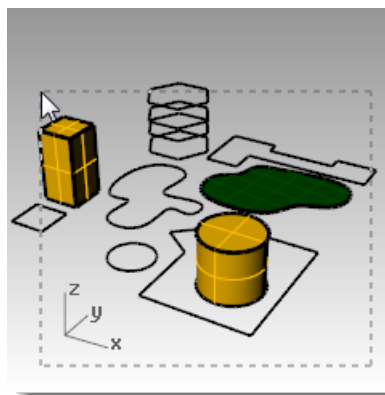
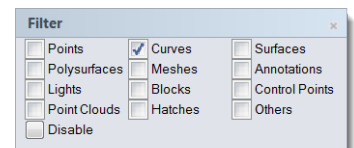
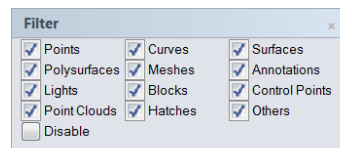
To select objects using select tools:

- 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 last object you created is selected.

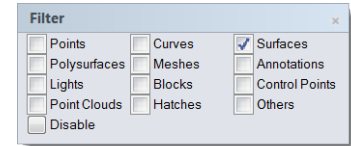
Filter Selection

A filter restricts any selection mode to specified object types. Objects types that can be restricted are: Annotations, Blocks, Control points, Curves, Lights, Meshes, Point clouds, Points, Polysurfaces, Surfaces, Hatches, and Others.

- 1 From the **Status bar**, click on **Filter**. The selection **Filter** dialog will appear.
- 2 Right-click **Curves** and all other selection options are unchecked.
- 3 Use a crossing box to select the entire the drawing.
Only the curves are selected.
- 4 Press **Esc** to deselect.



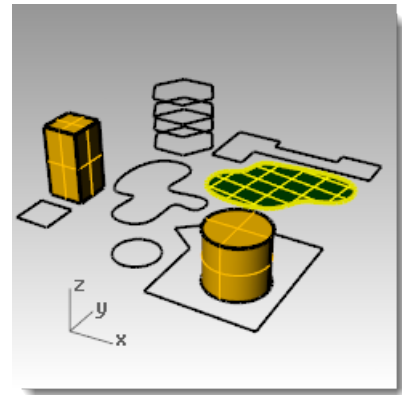
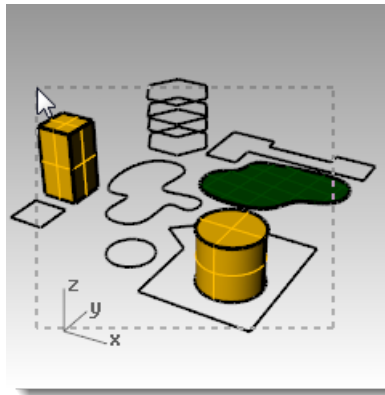
- 5 Right-click **Surfaces** and all other selection options are unchecked.



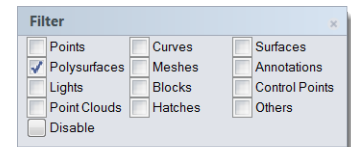
- 6 Use a crossing box to select the entire the drawing.

Only the surfaces are selected.

- 7 Press **Esc** to deselect.



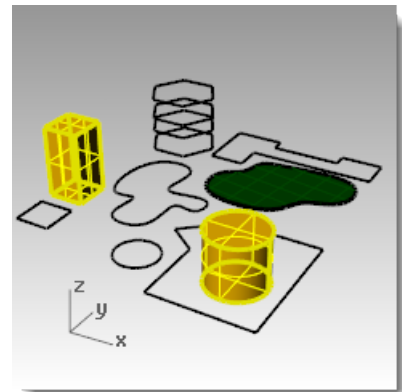
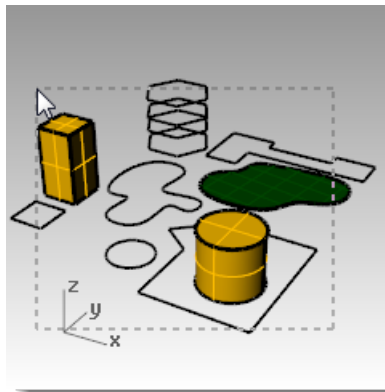
- 8 Right-click **Polysurfaces** and all other selection options are unchecked.



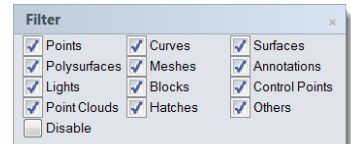
- 9 Use a crossing box to select the entire the drawing.

Only the Polysurface objects are selected.

- 10 Press **Esc** to deselect.



- 11 Reset all filters to on.



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.

You will select the units and tolerance of the model before you begin. You can do this in the Options dialog under the Units page or by picking a template that has units and Tolerance already set.

You can change the tolerance after you start, but objects that are edited before the change still have the old tolerance value.

Coordinate entry

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

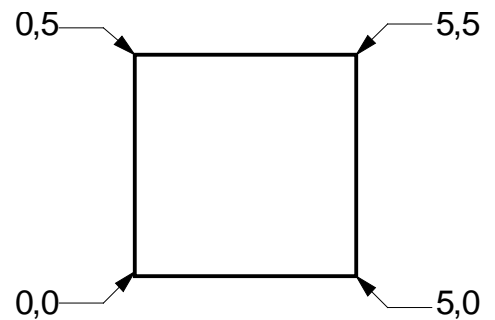
The first forms of coordinates you will use are called absolute coordinates. Absolute coordinates are exact locations relative to the x-, y-, and z-axes.

Exercise 9—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.

Exercise 10—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.



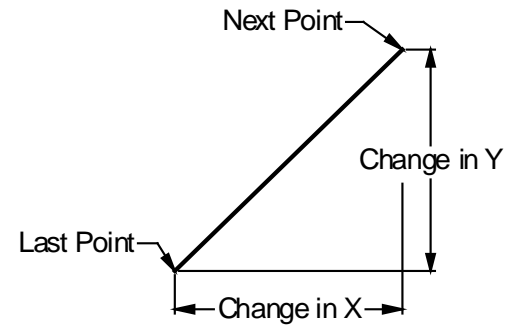
Relative coordinates

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.

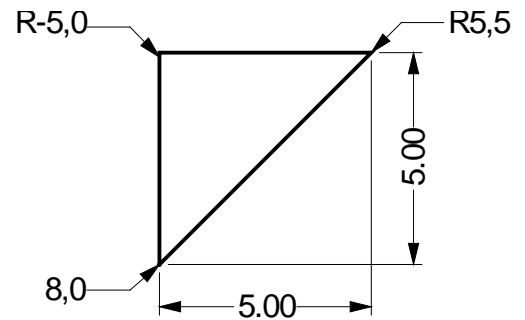
Relative coordinates are based on the last point entered, instead of on the origin (0,0,0) of the construction plane.

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 11—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.



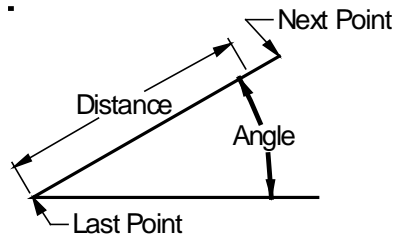
Polar coordinates

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

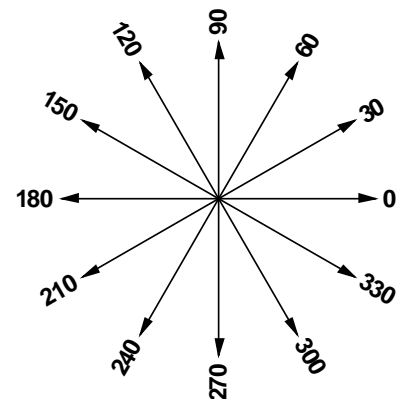
Vector directions in Rhino start with Zero degrees at 3 o'clock on a standard clock. They change in an anti-clockwise direction as illustrated below.

For example, if you want a point four units away from the construction plane origin, at a 45° angle counterclockwise 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.

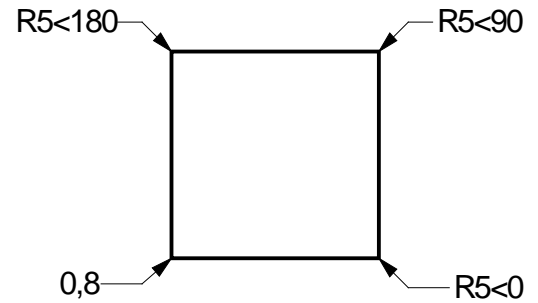


Instead of using x, y, and z coordinates, enter relative polar coordinates like this: Rdistance<angle.



Exercise 12—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.



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.

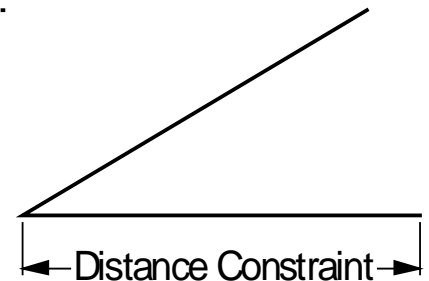
Using angle constraint entry, you can specify an angle by typing **<** followed by a value and pressing **Enter**. The next point is constrained to lines at multiples of the angle relative to the x-axis you specified.

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 13—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**.

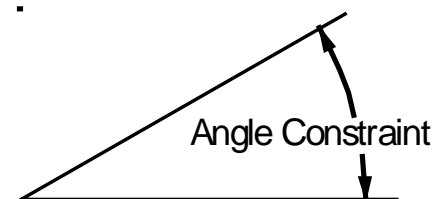


- 6 Hold the **Shift** key down and pick a point up.
Ortho constrains the cursor to 90 degrees
- 7 For the **Next point** type **5** and press **Enter**.
- 8 Hold the **Shift** key down and pick a point to the left.
Ortho constrains the cursor to 180 degrees.
- 9 Click **Close** to close the polyline.

Exercise 14—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.

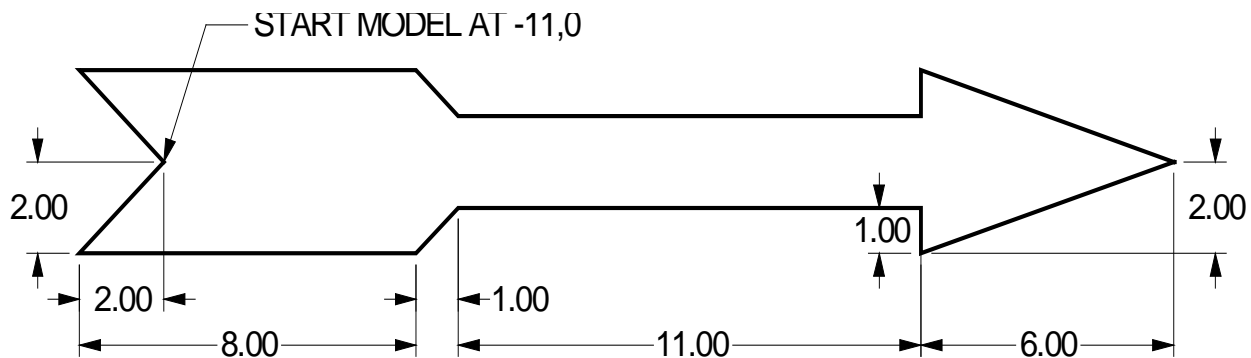


- 4 Pick a point down and to the right.
The angle constraint sets the angle.
- 5 For the **Next point** type **5** and press **Enter**, then type **<45** and press **Enter**.
- 6 Pick a point up and to the right.
The angle constraint sets the angle.
- 7 For the **Next point** type **5** and press **Enter**, 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.

Exercise 15—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**.

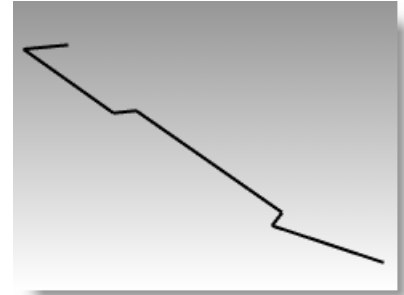
Distance constraint

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

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:

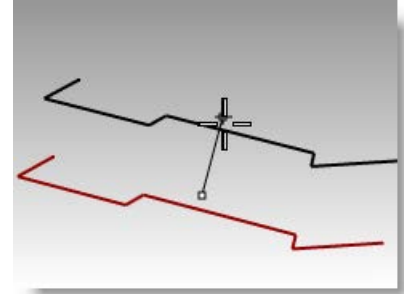
1 Select the polyline.

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.

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



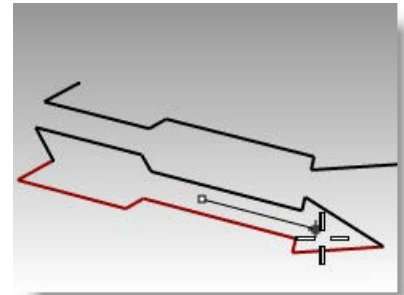
To make a mirror image of the polyline:

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.



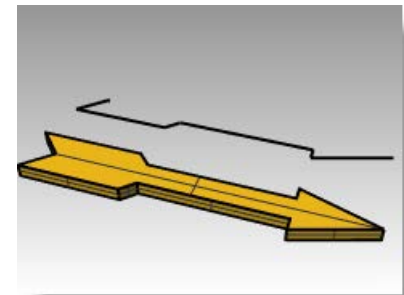
To make it 3-D:

1 Right-click on the **Perspective title** and set the display mode to **Shaded**.

2 Select the original polyline and the mirror copy.


3 From the **Solid** menu and click **Extrude Planar Curve**, click **Straight**.

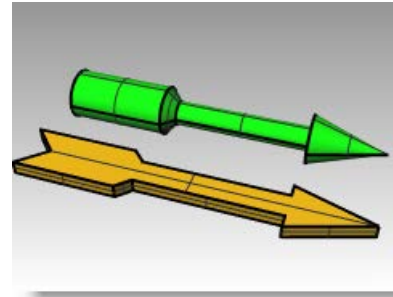
4 For the **Extrusion distance**, type **1** and press **Enter**.



Your arrow is now a three-dimensional model.

To make it 3-D (alternate):

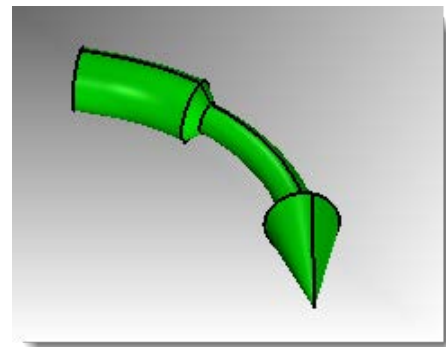
- 1 Select the copy of the polyline.
- 2 On the **Status bar**, click **Osnap**.
- 3 On the **Osnap** toolbar check **End**.
- 4 From the **Surface** menu click **Revolve**. 
- 5 For the **Start of revolve axis**, select the end of the polyline.
- 6 For the **End of revolve axis**, select the other end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



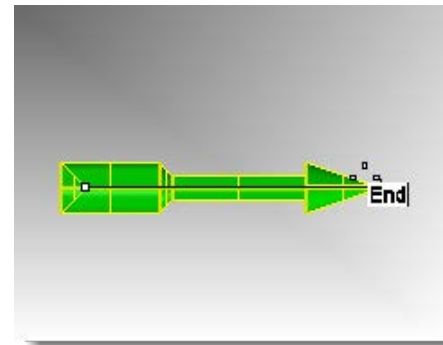
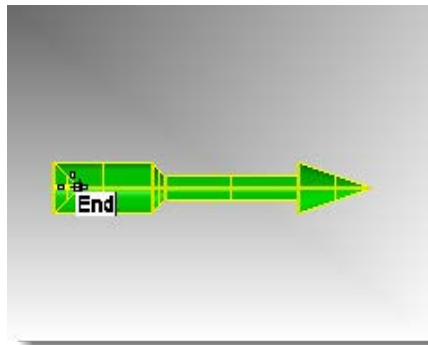
Your arrow is now a three-dimensional model.

To deform the shape:

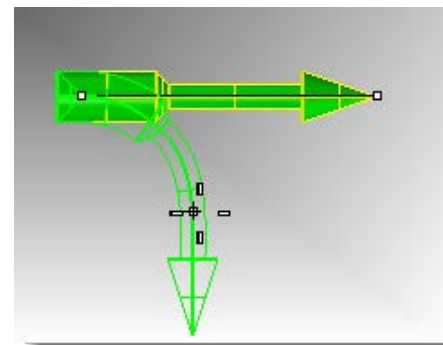
- 1 Select the 3-D arrow.
- 2 From the **Transform** menu click **Bend**.



- 3 For the **Start of spine**, click the end point on the left end of the arrow.
- 4 For the **End of Spine**, click the end point on the right end of arrow.

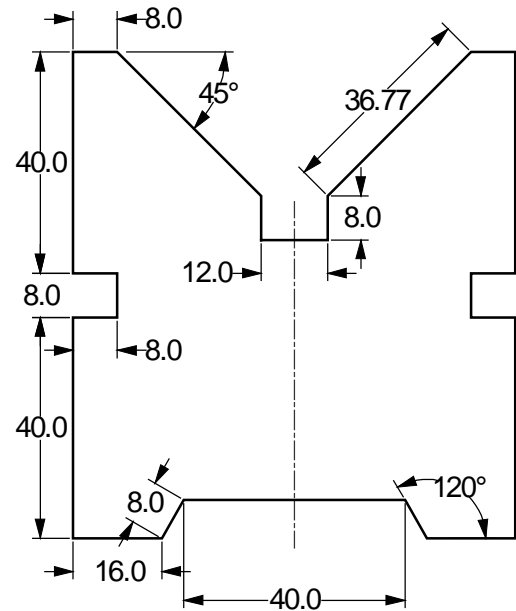



- 5 For the **Point to bend through**, drag the cursor down and pick a point.

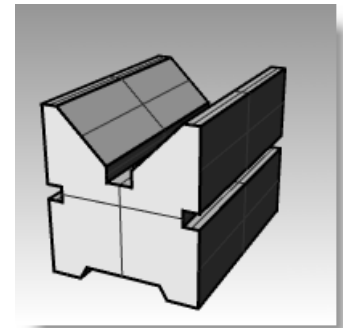


Exercise 16—Practice using distance and angle constraints

- 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 in a clockwise direction.
- 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 150 and press **Enter**.
You can view the model as a three-dimensional object in the Perspective viewport.
- 9 **Save** your model.



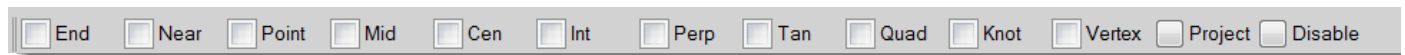
Object Snaps

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.



The display of the Osnap toolbar is controlled with the Osnap pane in the Status bar.

Click on it to toggle the display on and off.



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.

Left click on a check box to turn on the object snap.

Right click on a check box to turn on the object snap and clear all other osnaps.

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.
Vertex		Mesh vertex is the location where the edges of the mesh faces meet.
Disable		Temporarily turns off persistent object snaps, retaining settings

Exercise 17—Using object snaps

In this model we will practice using most of the osnaps listed on the toolbar.

- 1 **Open** the model **Osnap.3dm**.
- 2 Toggle Grid 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.



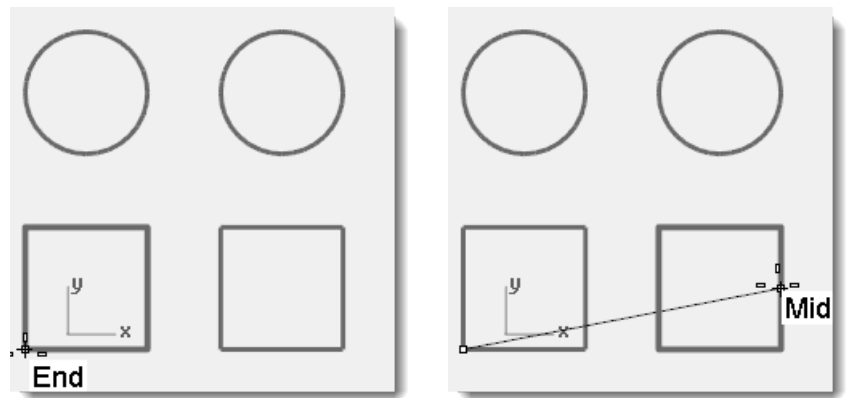
Osnap bar with the End and Mid osnaps turned on.

- 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.



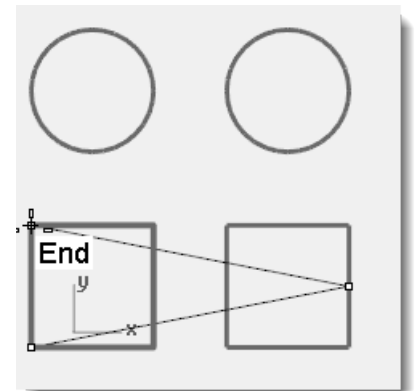
- 5 For the **Next point**, move your cursor close to the midpoint of the vertical line at the right side of the square on the right, pick when the marker snaps to the midpoint.

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.



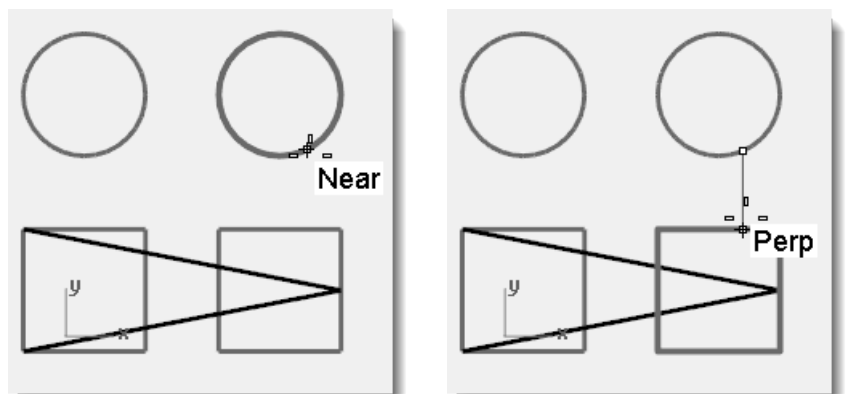
Using Near and Perpendicular to object snaps:

- 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 when the **Perp** tooltip appears.

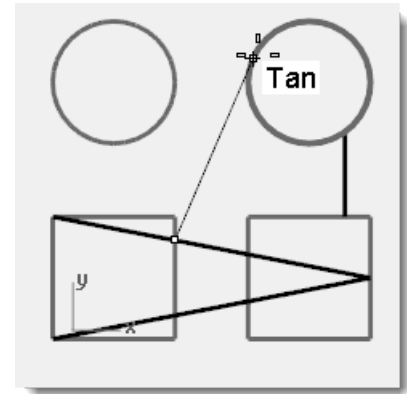
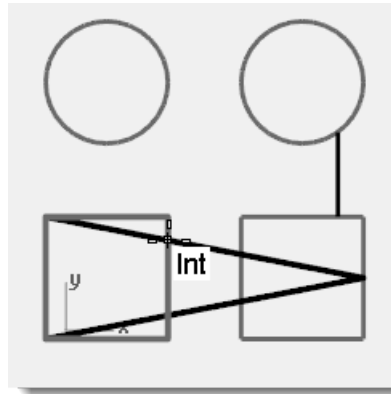
The marker snaps to a point making a perpendicular to the previous point.



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.

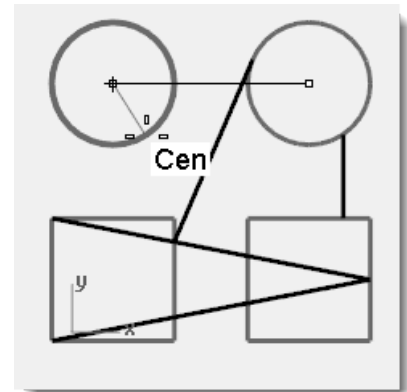
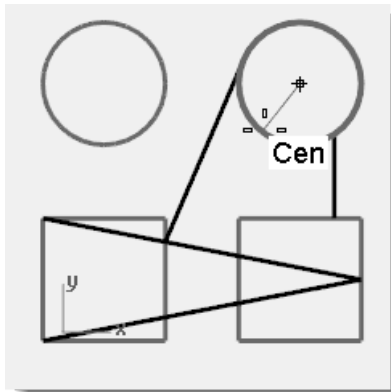
Using Center of object snap:

- 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.



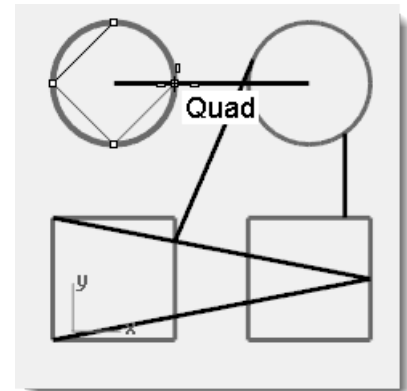
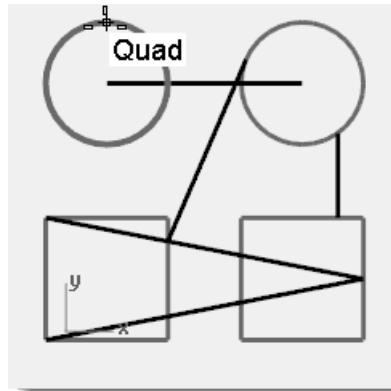
Using Quadrant of object snap:

- 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.

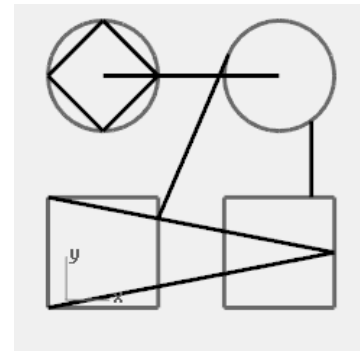
The marker snaps to the quadrant point of the circle.

- 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.



Additional 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.

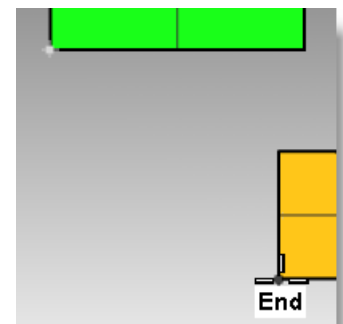
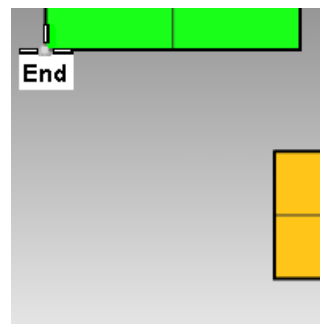
SmartTrack

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

Exercise 18—SmartTrack

To use the SmartTrack tool:

- 1 **Open** the model **Constraints.3dm** 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 **SmartTrack** on the **Status Bar**.
- 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.

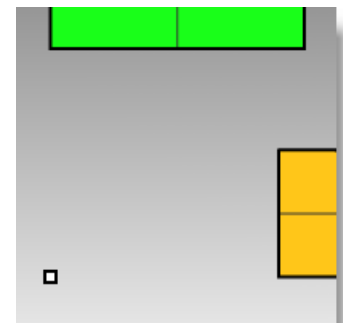
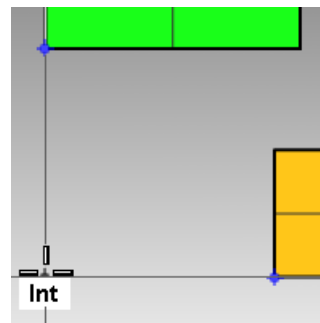


- 7 Now move the cursor towards the apparent intersection point of these two corners. Two temporary construction lines are displayed.

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

- 8 Click to place the Point Object.

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



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.

Exercise 19—Tab Constraint

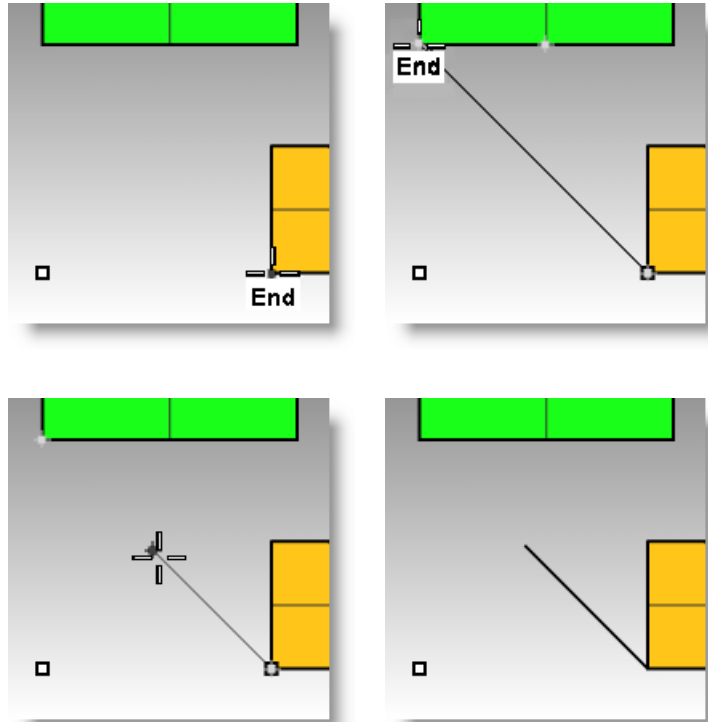
To use the Tab Constraint:

- 1 We will work with the **Constraints.3dm** file as previous and again maximize Top view.
- 2 **Turn off SmartTrack.**
- 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.



Project constraint

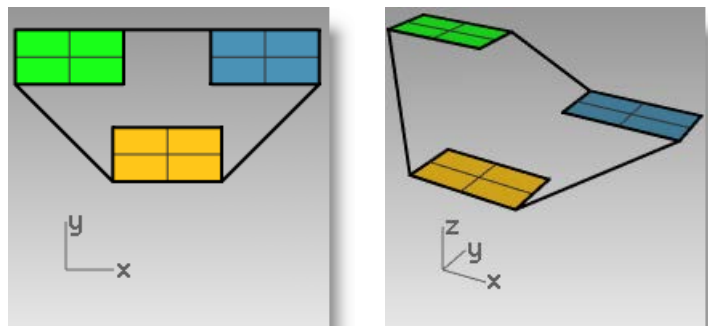
By default 2-D geometry is created on the active construction plane. Object snaps override this behavior and snapping to objects that are not on the construction plane will cause the geometry to be non-planar. The Project constraint ignores objects snaps and pushes all the geometry onto the active construction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

Exercise 20—Project Constraint

To use the Project Constraint:

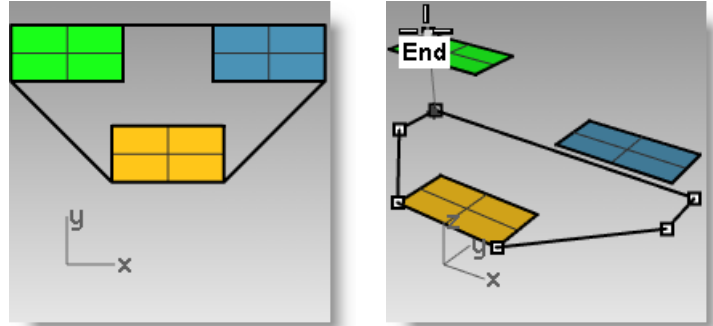
- 1 We will again work with the **Constraints.3dm** file as previous.
- 2 Make sure that the **Ortho** constraint is **On**.
- 3 Turn **Layer 01 off** and turn **Layer 02 on**.
The surfaces on Layer 02 are located at different elevations.
- 4 From the **View** menu, click **Zoom**, then click **Zoom Extents**.
- 5 Double-click on the Top viewport title to restore the 4 View layout.
- 6 Working in Top viewport, draw a Polyline around the perimeter of the three rectangles.

In the Perspective view, notice how the how the planar nature of the polyline is overridden by the object snaps.



- 7 Delete the Polyline.
- 8 **Turn on** the **Project** constraint on the **Osnap** toolbar.
- 9 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.

- 10 Delete the Polyline.

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.

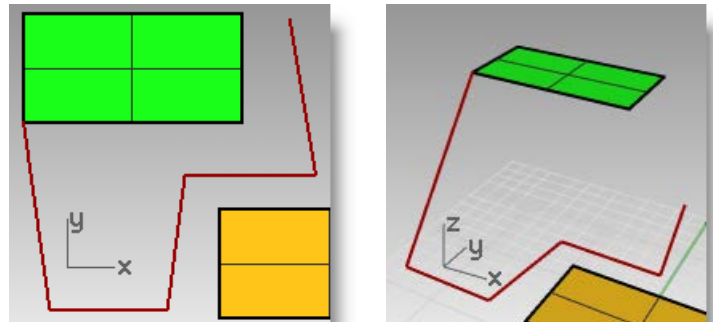
Exercise 21—Planar Constraint

To use the Planar Constraint:

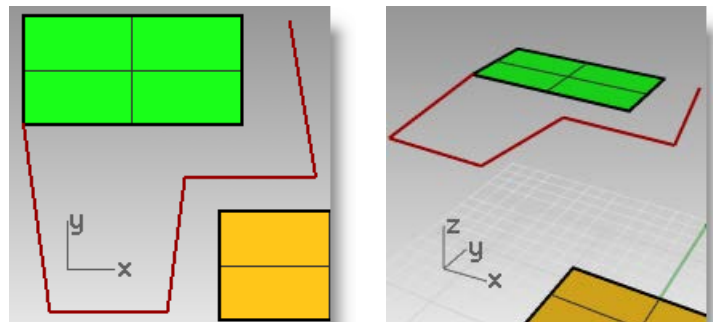
We will first look at what happens with the Planar constraint Off. Then, we will turn the Planar constraint On to see what changes.

- 1 We will again work with the **Constraints.3dm** file as previous.
- 2 **Turn off** the **Ortho** and **Planar** constraints, and **Project** on the Osnap toolbar.
- 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.



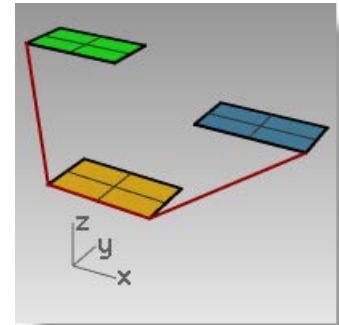
To create a planar curve above the active construction plane:

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.

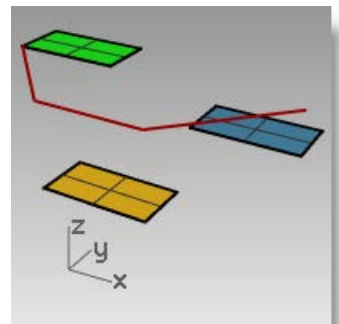
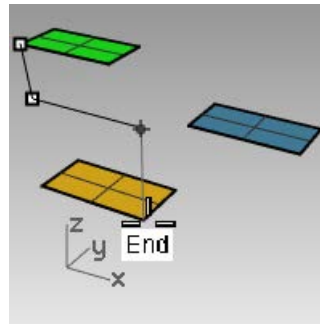
Look at the Perspective view and note how the object snaps override the Planar constraint.

- 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.

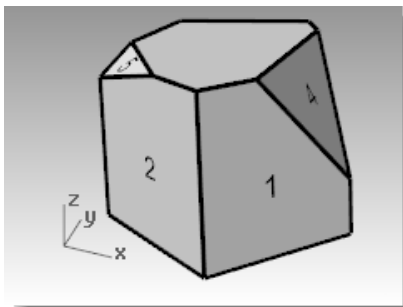


Viewports and construction planes

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

Exercise 22—Intro to construction planes

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



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. 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.

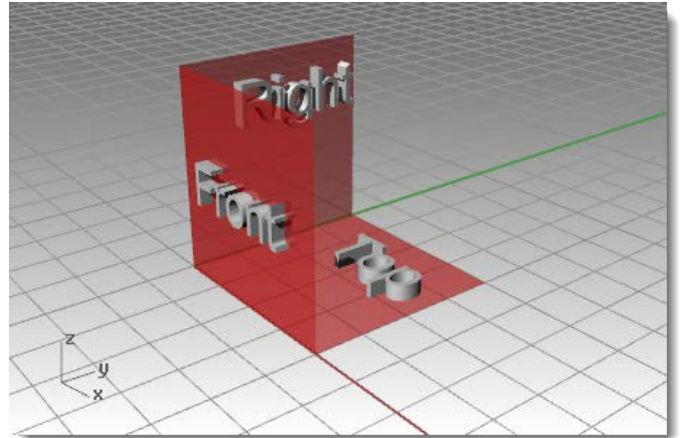
Each construction plane has its own axes, a grid, and an orientation relative to the world coordinate system.

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.



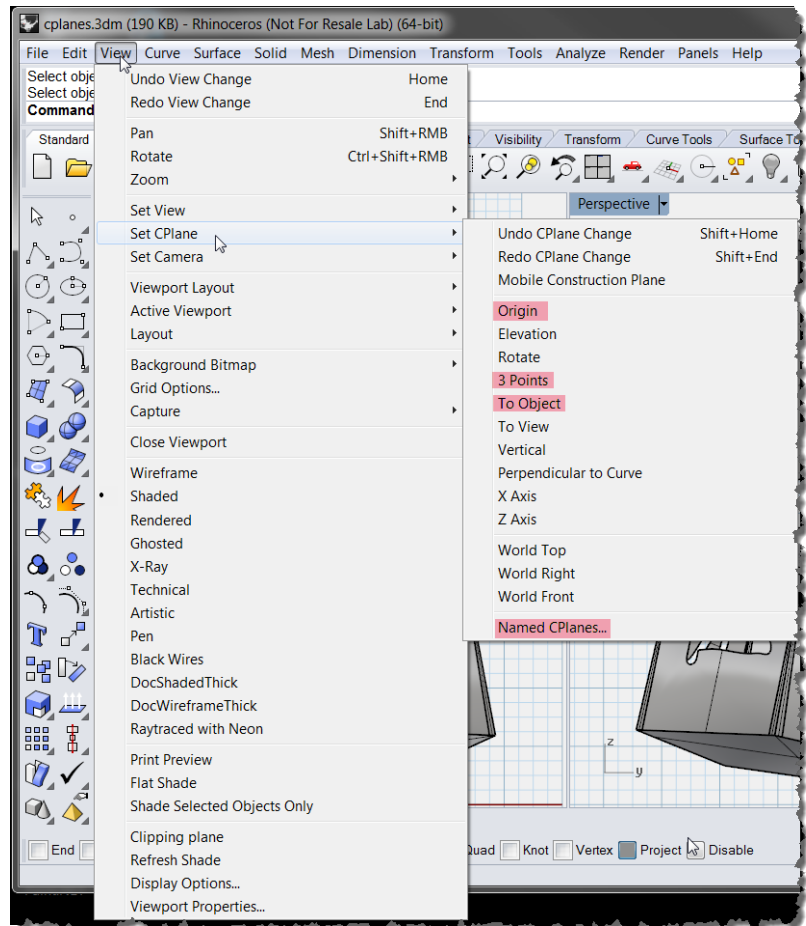
- The Top construction plane x- and y-axes align with the world x- and y-axes.
- The Right construction plane (x- and y-axes) align with the world y- and z-axes.
- The Front construction plane (x- and y-axes) align with the world x- and z-axes.
- The Perspective viewport uses the Top construction plane.

You can set construction planes by:

- Typing Cplane
- From the View Menu
- Right click on the viewport title
- Pick the Arrow on the viewport title.

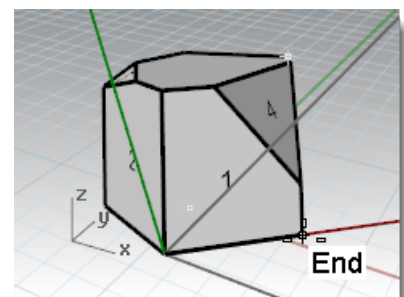
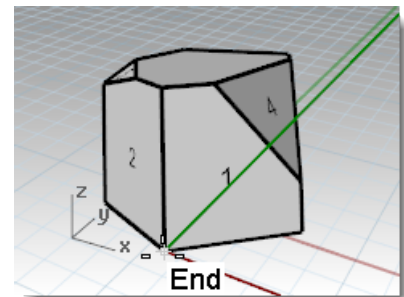
There are many options to the Cplane command. In this exercise, you will work with:

- Origin
- 3 Point
- To Object
- Named Cplanes



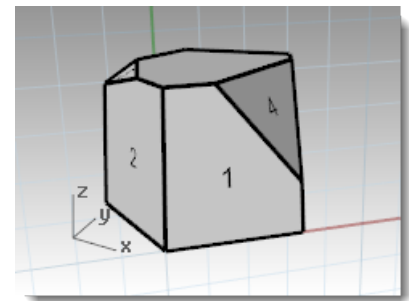
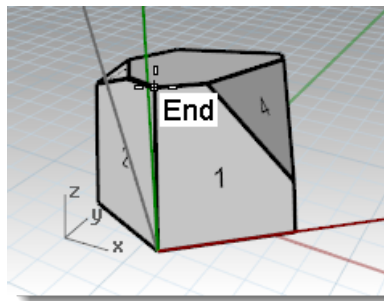
To change a Cplane using the 3 Point option:

- 1 From the **View** menu click **Set Cplane**, and then click **3 Points**.
- 2 For the **Cplane origin**, snap to the lower left corner of the surface labeled as 1.
- 3 For the **X axis direction**, snap to the lower right corner of the surface labeled as 1.

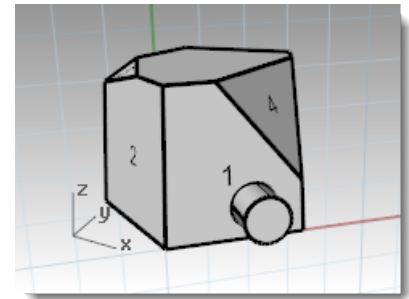


- 4 For the **Cplane orientation**, snap to the upper left corner of the surface labeled as 1.

Now the Cplane is set



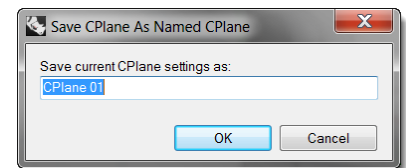
- 5 From the **Solid** menu click **Cylinder**.
- 6 For the **Base of cylinder** and **Radius** pick points anywhere on the new cplane.
- 7 Drag and click to get the **End of cylinder**.



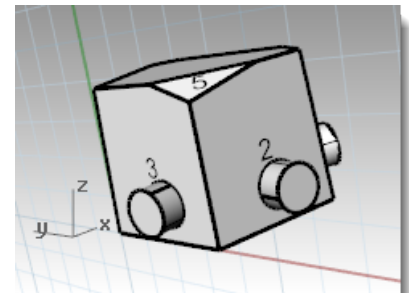
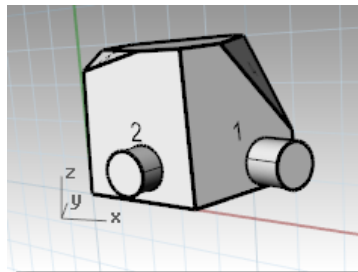
To save the new Cplane:

- 1 From the **View** menu click **Set Cplane**, and then click **Named Cplanes**.
- 2 A **Panel** will open with **Named Cplanes** controls.
- 3 Click the **Save as** button in the toolbar area.
- 4 **Enter a name** or use the default name **Cplane 01**, click **OK**.

You have a named Cplane that can be restored at any time.

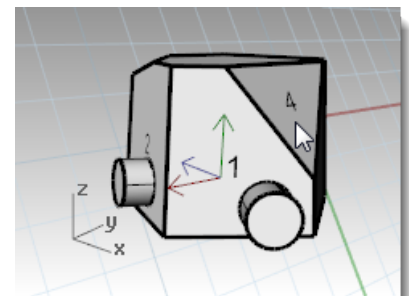


- 5 Repeat setting and saving named Cplanes for surfaces labeled 2 and 3.



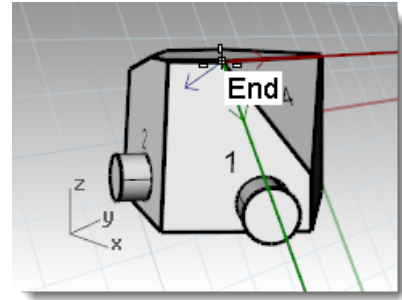
To set a Cplane to an object:

- 1 From the **View** menu click **Set Cplane**, and then click **To Object**.
 - 2 Select the surface labeled as 4.
- The Cplane is set to the surface. The origin of the new Cplane is the center of the underlying untrimmed surface.
- 3 Using **Named Cplane**, name this cplane **Cplane 04**.

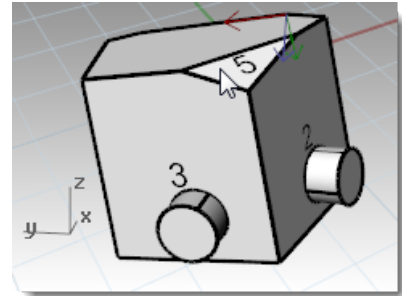


To change the origin of a Cplane:

- 1 From the **View** menu click **Set Cplane**, and then click **Origin**.
- 2 For the **Cplane origin**, snap to the upper left corner of the surface labeled 4.



- 3 Set a **Cplane by Object** for the surface labeled 5.
- 4 Set a new **Cplane Origin** for the surface labeled 5.
- 5 Using **Named Cplane**, name this cplane, **Cplane 05**.



Exercise 23—Viewports and construction planes

- Open the model Chair.3dm.

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.

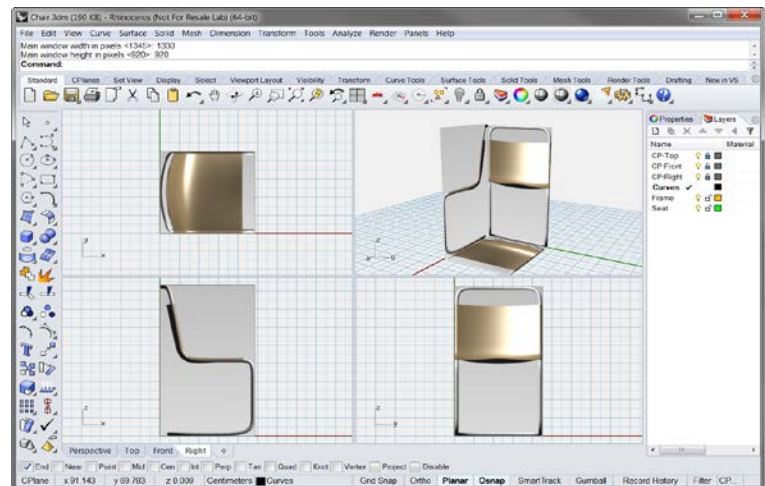
- Turn the following layers on:

- CP-Top
- CP-Front
- CP-Right

Each of the layers contains 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.



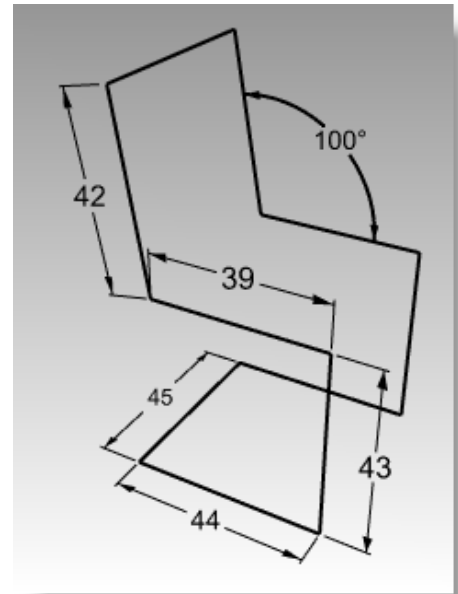
Exercise 24—Modeling in 3-D space

Chair: Technical Method (*Recommended*)

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.

Using the technical drawing on the right and the precision drawing and coordinate input to create the chair curves.

When you have completed the curves, go to section “*Finishing the chair.*”



Chair: Elevator Mode Method (*Alternate*)

Another method for modeling in 3-D space is modeling using elevator mode.

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

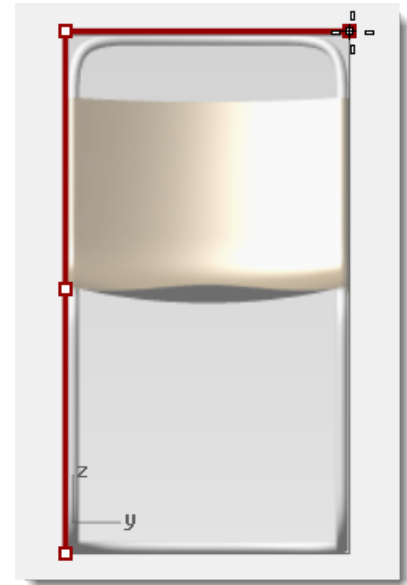
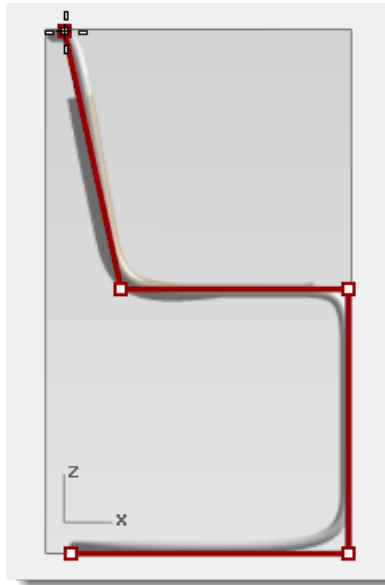
Elevator mode lets you pick points that are off the construction plane. Elevator mode requires two point picks to completely define the point. The first specifies the base point. The second specifies how far the final point is above or below the base point.

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.

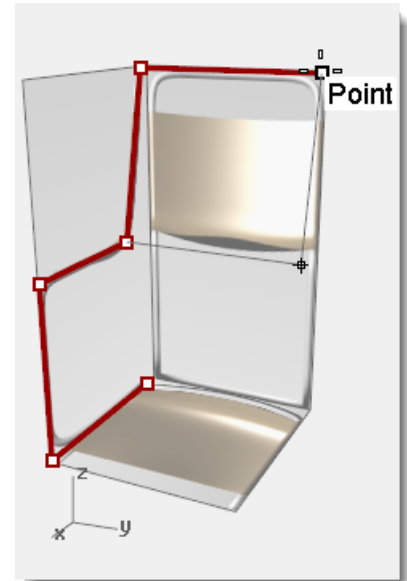
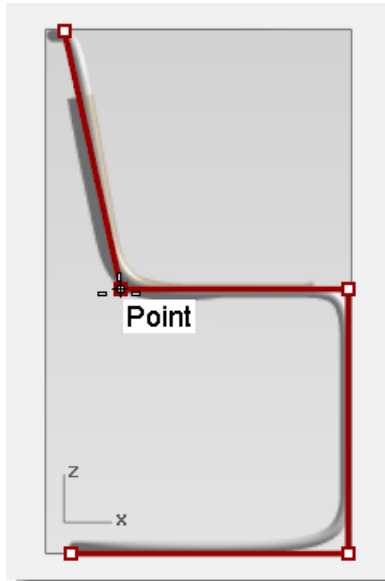
- 1 Toggle **Planar off** and **Grid Snap on**. Toggle **Ortho on** as needed.
- 2 Turn on the **Point** osnap.
- 3 From the **Curve** menu click **Polyline**, and then click **Polyline**.
- 4 Move your cursor into the Right viewport.
- 5 To start click at the lower left side of the tubular chair image.
- 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 **Front** viewport to draw a horizontal line.



- 8 Move your cursor to the **Right** 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.

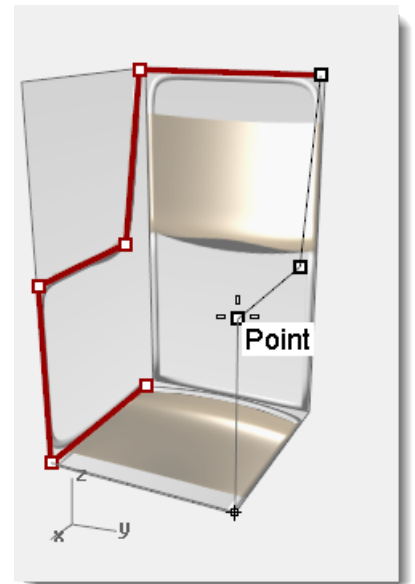
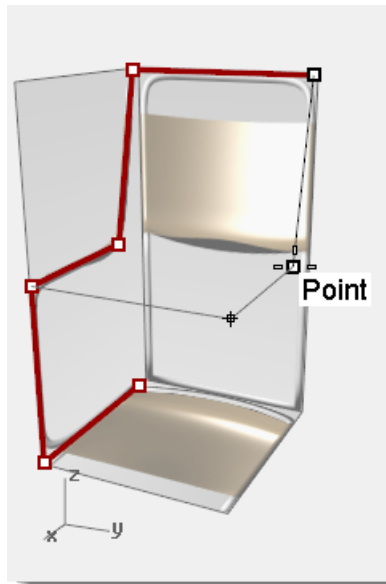
Holding the **Ctrl** key while clicking with the left mouse button activates elevator mode.

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

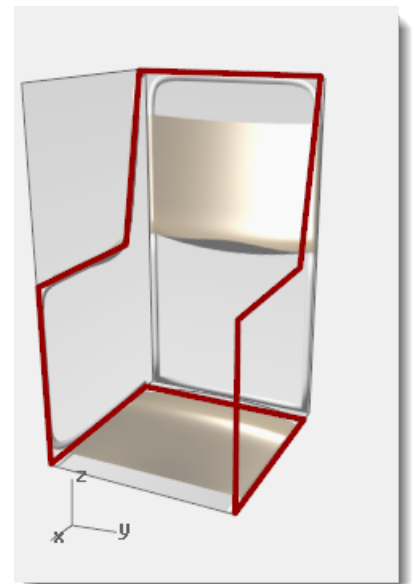
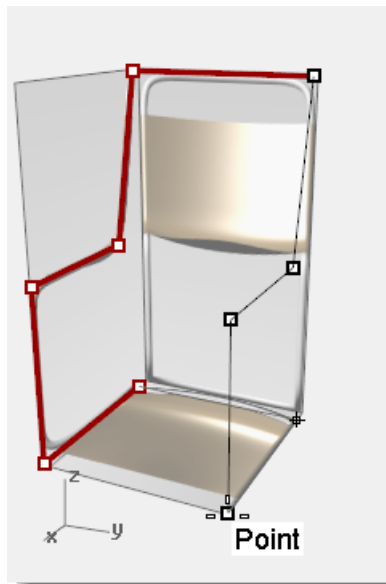


- 11 Continue drawing the rest of the chair frame using the same technique.

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.



Finishing the chair

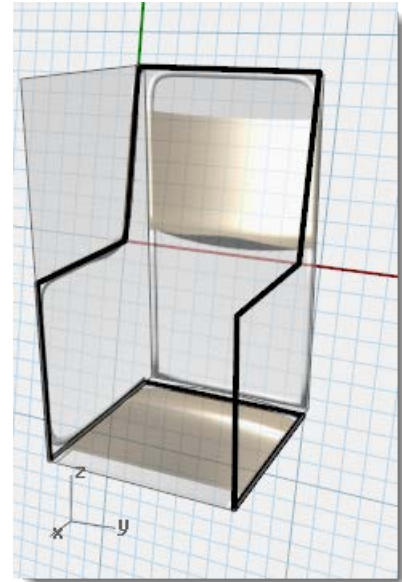
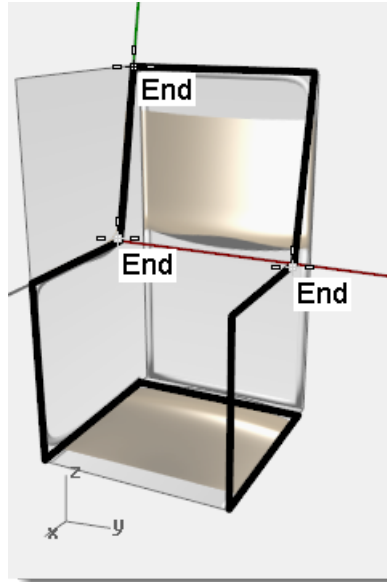
After creating the chair curve either with the technical modeling method or by using the elevator mode, you now need to construct the chair surfaces. It will help to first create a custom cplane aligned to the back of the chair.

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**, and click **Named CPlanes**.
- 2 In the **Named CPlanes** panel, 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.

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.

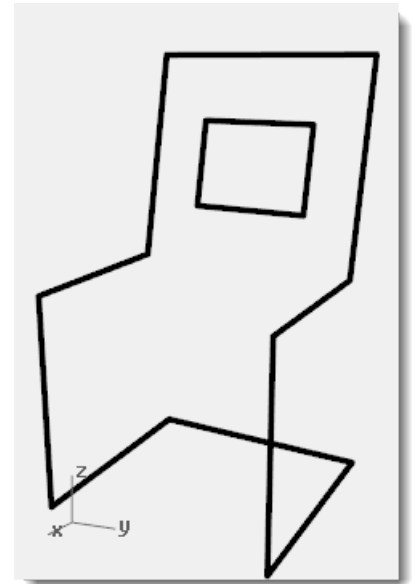
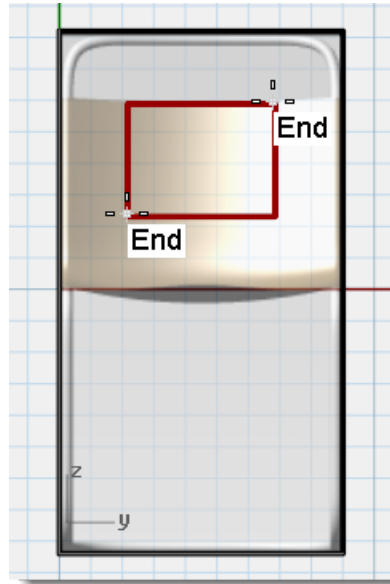
- 2 From the **View** menu click **Set View**, click **Named Views**. 

- 3 In the **Named Views** panel, click **SaveAs**.


- 4 In the **Save Viewport As Named View** dialog, type **ChairBack**, then click **OK**.

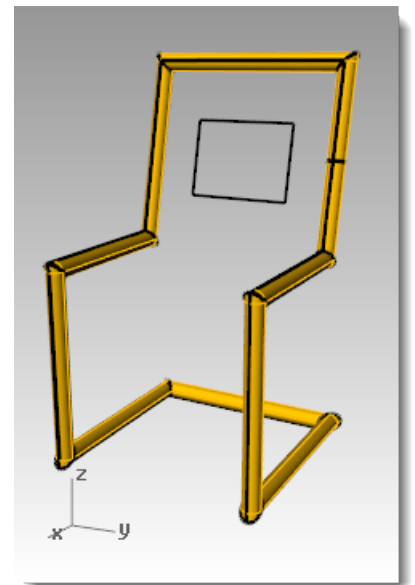
This creates a custom view that can be restored as needed.

- 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:

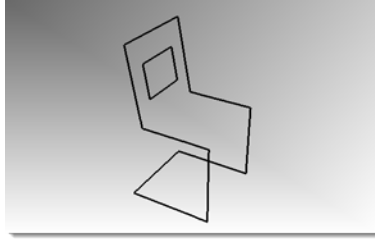
- 1 Change to the Frame layer.
- 2 Select the chair frame.
- 3 From the **Solid** menu, click **Pipe**. 
- 4 For the **Start** and **End Radius**, type **3** and press **Enter**.
The chair has a solid frame.
- 5 **Save** the model.



To Fillet the corners:

- 1 **Undo** the Pipe.
- 2 Select the chair frame.
- 3 From the **Curve** menu click **Fillet corners**.
- 4 Click **Radius** and type **10** and press **Enter**.

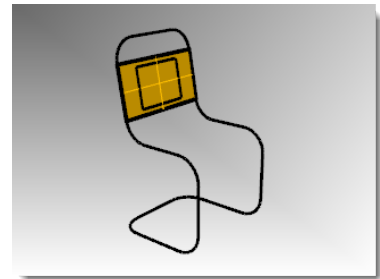
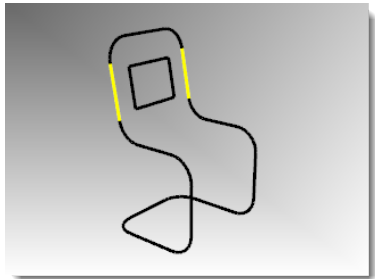
The fillet will be applied to all the chair corners.



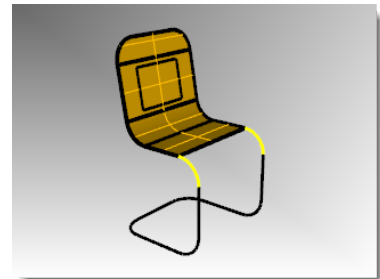
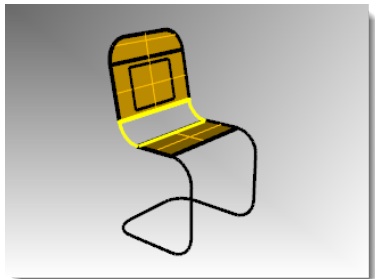
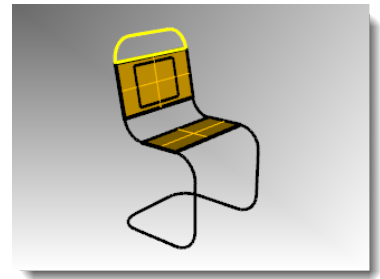
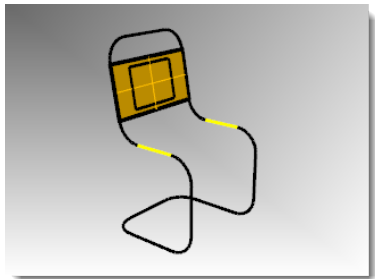
To make the seat surface:

- 1 Select the curve you just filleted. From the **Edit** menu click **Explode**.
- 2 From the **Surface** menu click **Edge Curves**.
- 3 Select the two side edges of the back of the chair, then press **Enter**.

A back surface is created.

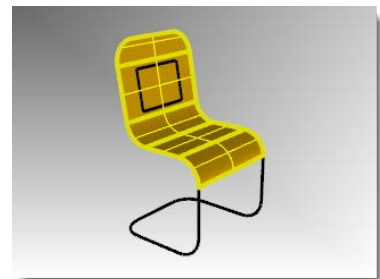
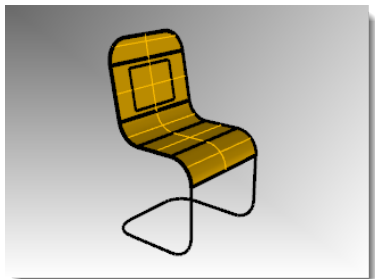


- 4 Repeat this procedure for additional seat surfaces.

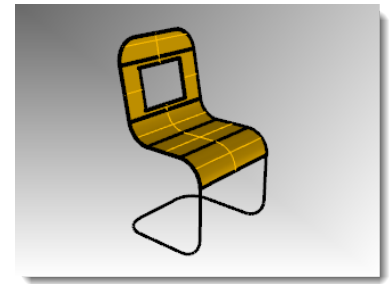
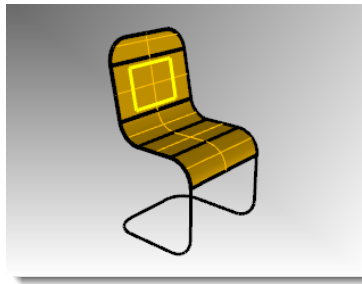


- 5 Select all the seat surfaces. From the **Edit** menu click **Join**.

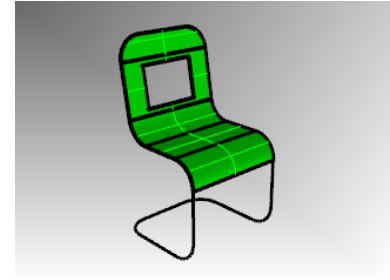
The seat will be joined into one polysurface. Next trim out the opening in the back.



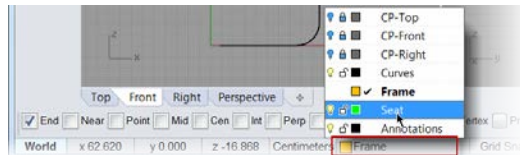
- 6 Select the rectangular curve you created earlier.
- 7 From the **Edit** menu click **Trim**.
- 8 Click on the part the surface inside the rectangle to trim the chair back.



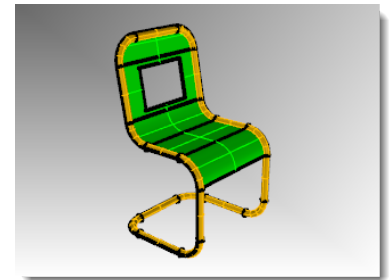
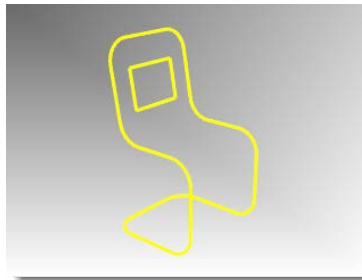
- 9 Highlight the seat surfaces. Using the Layer pane in the Status bar, select the Seat layer.



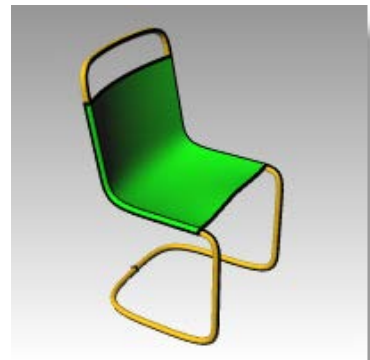
- 10 Turn off the Seat layer.



- 11 Select the curves for the frame.
- 12 From the **Edit** menu click **Join**.
- 13 Use the **Pipe** command to make the new curves into a tubular frame.
- 14 Turn Seat layer back on.



- 15 **Render** your model.



Completed rendering of chair.

On your own:

- Try some variations and add some features.

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.


Distance

Measures the distance between two picked points.

To find the distance between two points:

- 1 **Open** the model **Analyze.3dm** that you saved from an earlier exercise.

If you didn't get a chance to save the model, open the model 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.

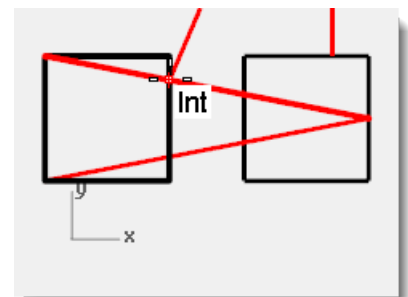
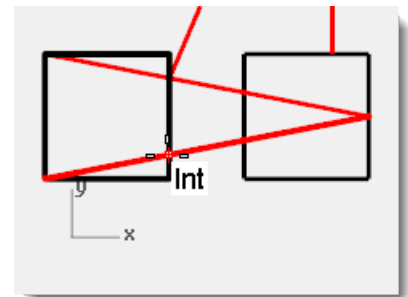
Use object snaps.

- 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



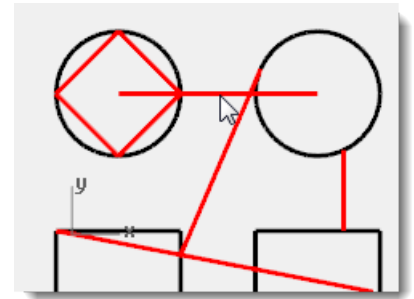
Length

Displays the length of a selected line.

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



To measure the angle between two lines:

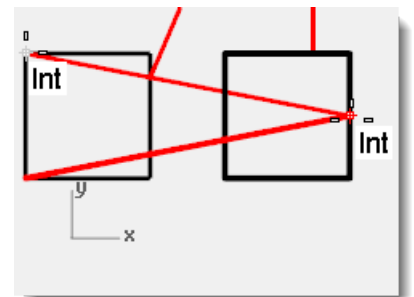
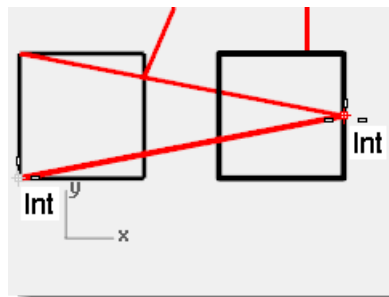
- 1 From the **Analyze** menu click **Angle**.



- 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

Radius

Displays the radius of a circle, arc, or curve segment.

To measure the radius of a circle:

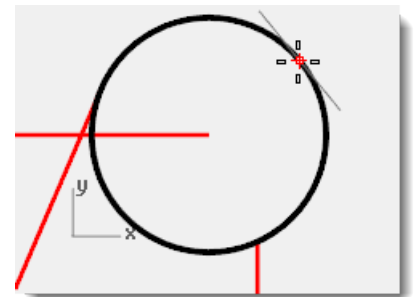
- 1 From the **Analyze** menu click **Radius**.



- 2 Select one of the circles.

This also measures the radius of curvature of a curve at a point.


The radius displays on the command line in the following format:
Radius = 2.5

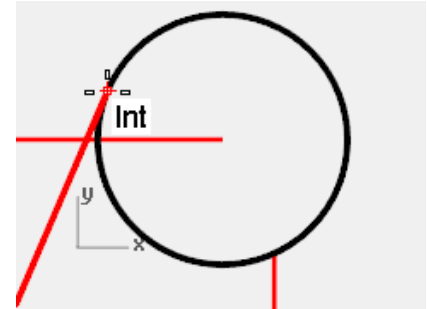


Point evaluation

Displays the coordinates of a picked location.

To display the coordinates of 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



Drawing with precision

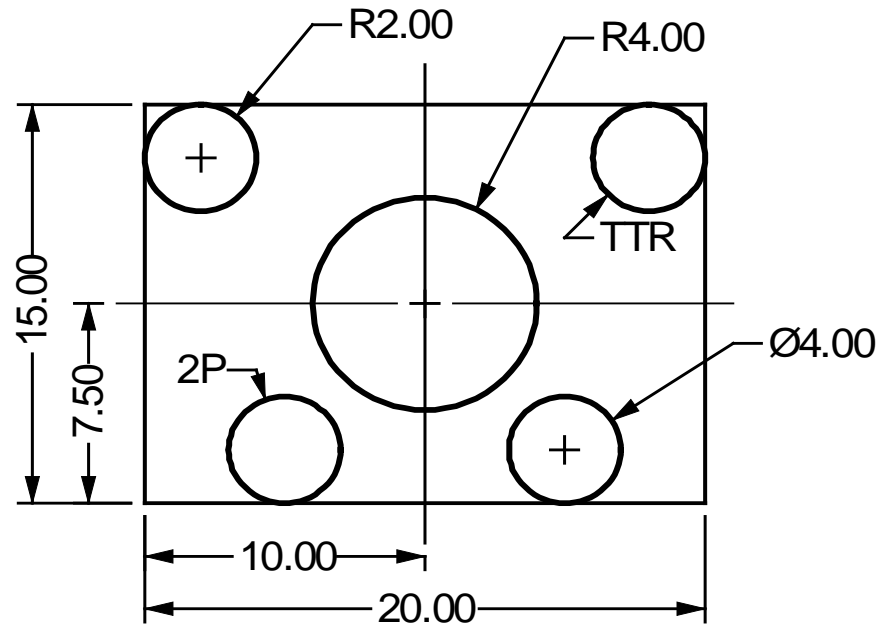
Circles

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

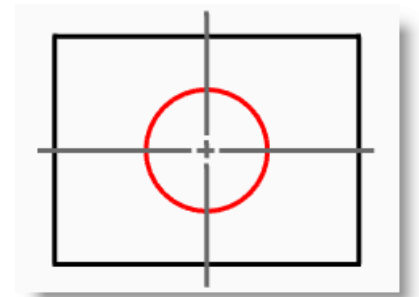
- Open the model **Circles.3dm**.



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.

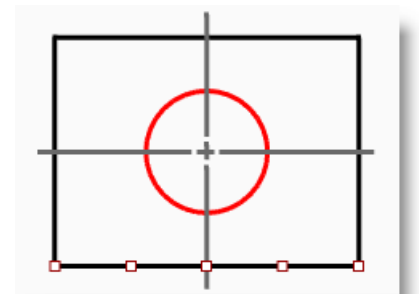


To divide a curve by the number of segments:


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

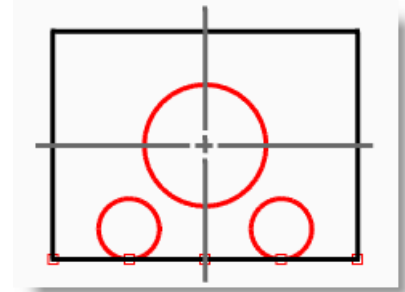
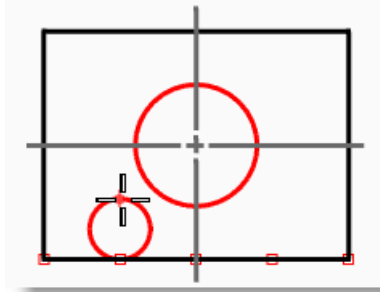
- 1 From the **Curve** menu click **Point Object**, then click **Divide Curve by**, and then click **Number of segments**.
- 2 Select the bottom line as the curve to divide, press **Enter**.
- 3 For the **Number of segments** type **4** and press **Enter**.

The line is marked off 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 in the straight up direction.



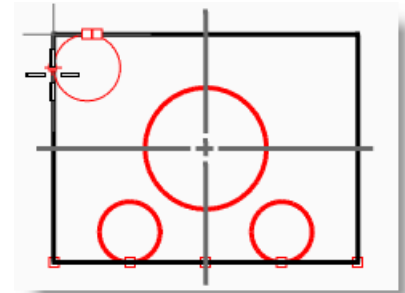
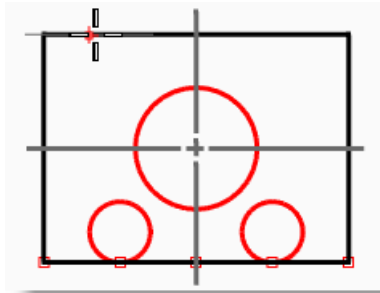
A circle is created with the two points you picked as the diameter, and the diameter was constrained to a value of 4.

- 5 Repeat these steps for the second diameter circle.

To draw a tangent, tangent, radius circle:

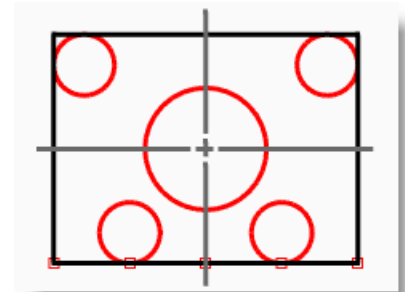


- 1 From the **Curve** menu click **Circle**, and then click **Tangent, Tangent, Radius**.
- 2 For the **First tangent curve**, select the top line.
- 3 For the **Second tangent curve**, select the left line.
- 4 For the **Radius**, type **2** and press **Enter**.



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

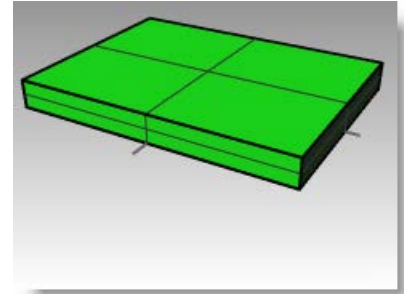
- 5 Repeat these steps for the second TTR circle.



To make it 3-D:

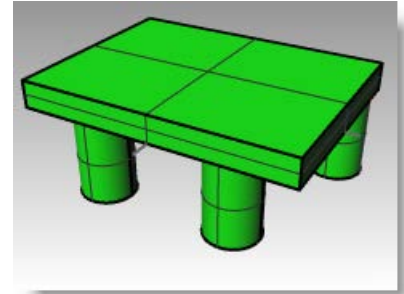
- 1 Select the lines that form the rectangle.
- 2 From the **Solid** menu click **Extrude Planar Curve**, and then click **Straight**.
- 3 For the **Extrusion Distance**, type **2** and press **Enter**.

The rectangle generates a box.



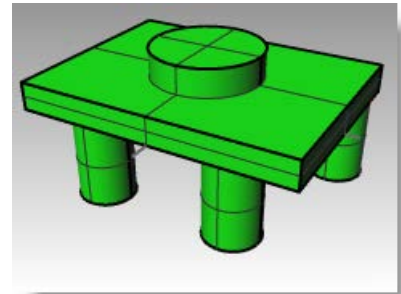
- 4 Select the four smaller circles.
- 5 From the **Solid** menu click **Extrude Planar Curve**, and then click **Straight**.
- 6 For the **Extrusion Distance** type, **-6** and press **Enter**.

The circles generate cylinders.



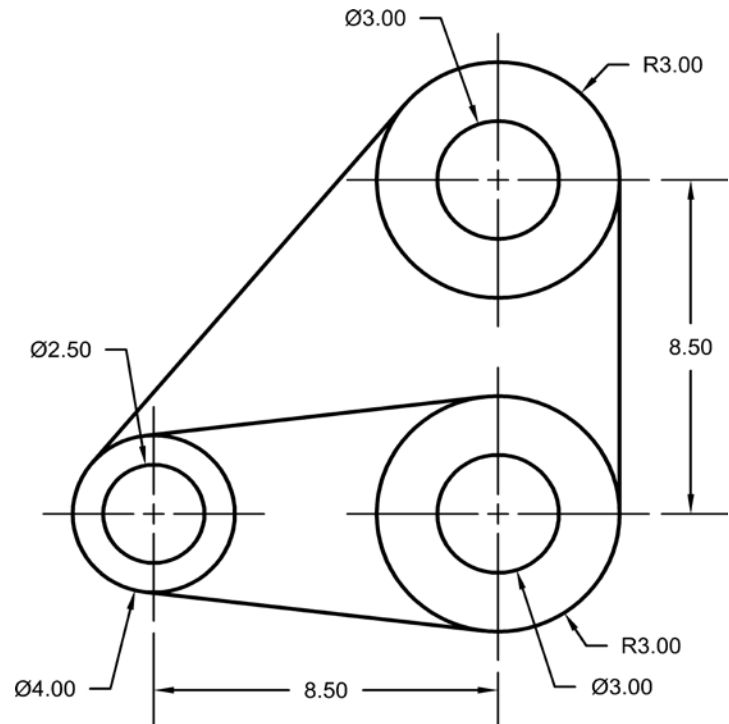
- 7 Select the large circle in the center.
- 8 From the **Solid** menu click **Extrude Planar Curve**, and then click **Straight**.
- 9 For the **Extrusion Distance**, type **4** and press **Enter**.

The circle generates a cylinder.



Exercise 26—Using circle-related object snaps

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

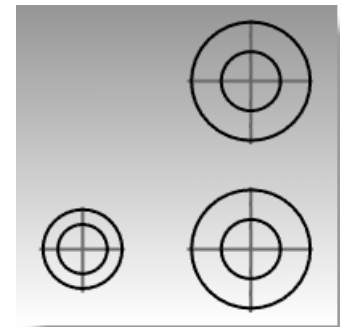
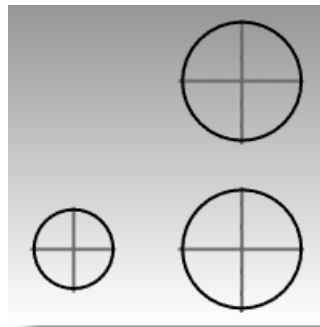


- 3 Draw the three large circles first.


Snap to the centerline intersections to place the circles.

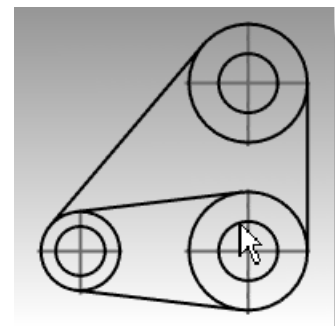
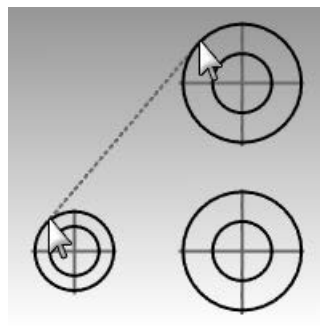
- 4 Draw the small holes next.

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



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.



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.

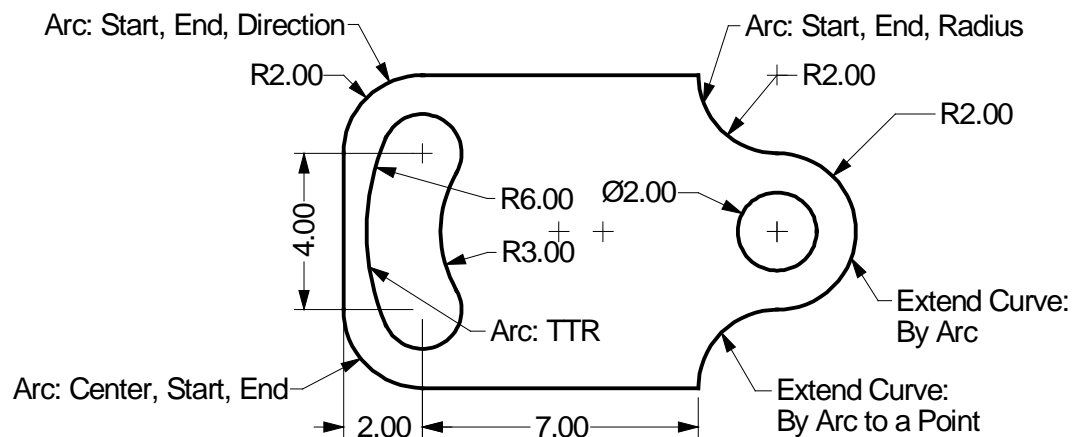
Button	Command	Description
	Arc	Draw an arc from the center, start, and angle.
	Arc : Start, End, Point on Arc	Draw an arc from three points.
	Arc: Start, End, Direction	Draw an arc from start point, end point, and the direction from the start point. The direction can be entered after the start point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

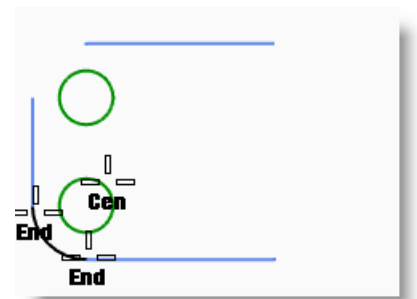
Exercise 27—Practice drawing arcs (1)

- **Open** the model
Arc1.3dm.



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

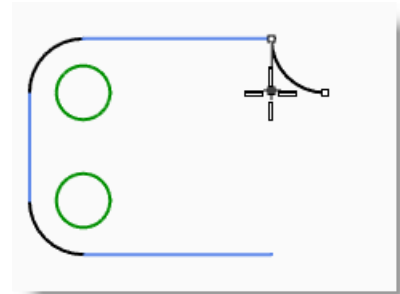
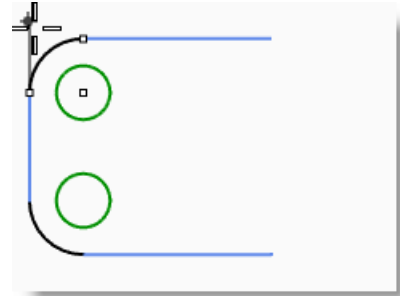
- 1 From the **Curve** menu click **Arc**, and then click **Center, Start, Angle**.
- 2 For the **Center of arc** snap to the center of the circle at the lower left.
- 3 For the **Start of arc** snap to the end of the line.
- 4 For the **End point or angle** snap to the end of the other line.



To draw a start, end, direction arc:



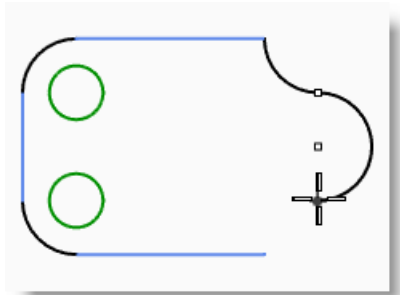
- 1 From the **Curve** menu click **Arc**, and then click **Start, End, Direction**.
- 2 For the **Start of arc** snap to the upper end of the vertical line.
- 3 For an **End of arc** snap to the adjacent end of the horizontal line at the top.
- 4 For the **Direction at start**, toggle Ortho on and drag in the straight up direction and pick.
- 5 Make another **Direction Arc** at the upper right.
- 6 For the **Start of arc**, snap to the right end of the horizontal line.
- 7 For an **End of arc** type **R2,-2** and press **Enter**.
- 8 For the **Direction at start**, toggle Ortho on and drag in the straight down direction and pick..



To add more arc segments using extension option:

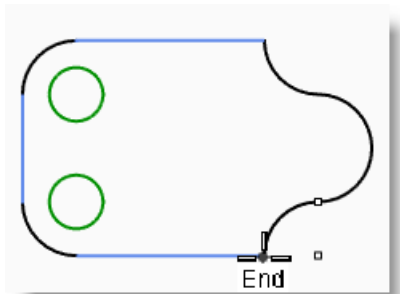
- 1 From the **Curve** menu click **Arc**, and then click **Center, Start, Angle**.
- 2 Click **Extension** on the command line.
- 3 For the Select near curve end pick near the end of the arc you just made.
- 4 For the **End of arc**, type **4** and press **Enter**.
- 5 With Ortho on pick a point below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

The arc will be tangent to the curve you chose.




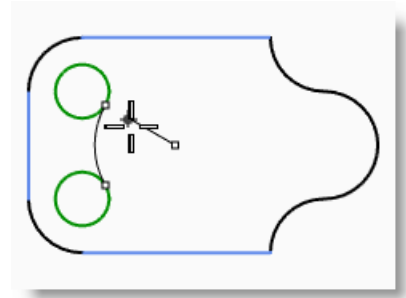
To add more arc segments using the extension option:

- 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.

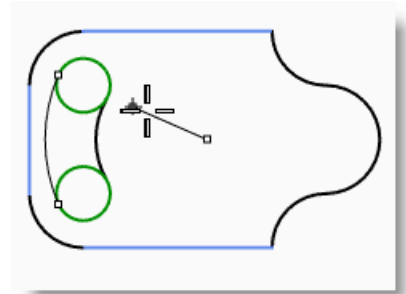


To draw a tangent, tangent, radius arc:

- 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.



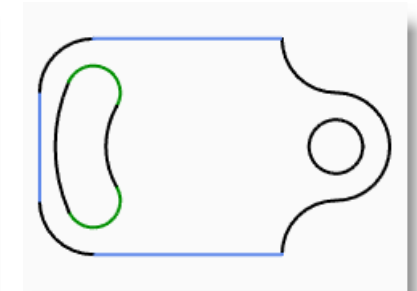
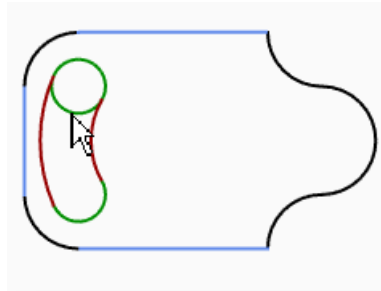
- 6 From the **Curve** menu click **Arc**, and then click **Tangent, Tangent, Radius**.
- 7 For the **First tangent curve** pick the upper left side of the top circle.
- 8 For the **Radius** type **6** and press **Enter**.
- 9 For the **Second tangent curve** pick the lower left side of the lower circle.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



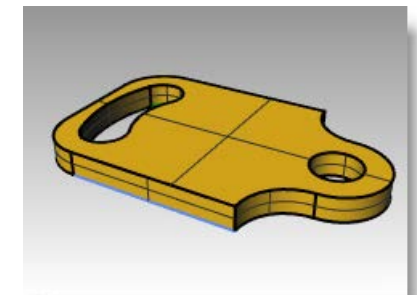
To make it solid:

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.



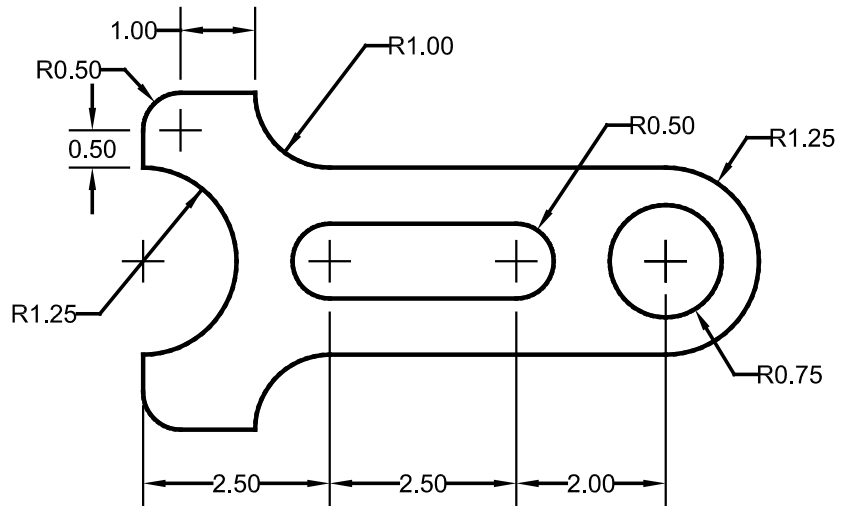
- 5 Select all the curves.
- 6 From the **Solid** menu click **Extrude Planar Curve**, and then click **Straight**.
- 7 For the **Extrusion distance** type **1** and press **Enter**.
The curves have been extruded and capped.



Exercise 28—Practice drawing arcs (2)

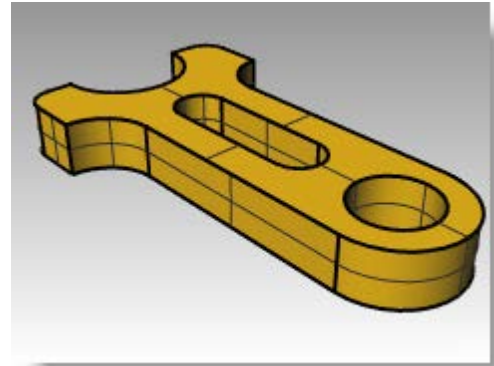
It might be helpful to start this model by drawing centerlines first and then lock them or put them on a locked layer. Use the Line command to construct the centerlines. When you have the centerlines drawn, you can use their intersection points to draw your arcs and circles.

- 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.






Ellipses and polygons

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

Ellipses

Button	Command	Description
	Ellipse	Draw an ellipse by specifying the center point and axis ends.
	Ellipse Diameter	Draw an ellipse by specifying the axis ends.
	Ellipse FromFoci	Draw the ellipse from the focus points.
	Ellipse AroundCurve	Draw an ellipse whose axes are 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 polygonal star.

Polygon options

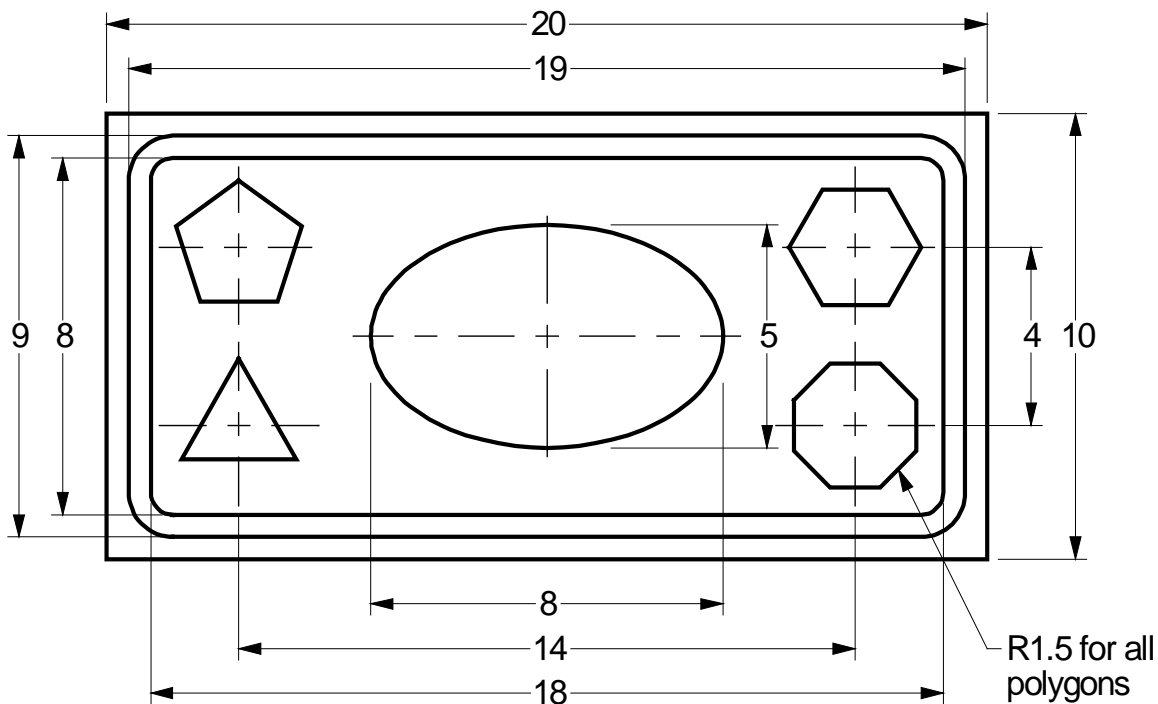
Option	Description
NumSides	Specify the number of sides for the polygon.
Circumscribed	Draw a polygon that is circumscribed about a radius. The default is to draw a polygon that is inscribed in a specified radius.

Rectangles


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

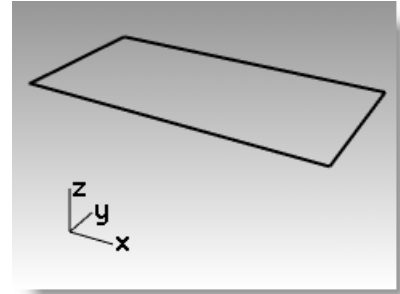
Exercise 29—Practice drawing ellipses and polygons

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




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**.



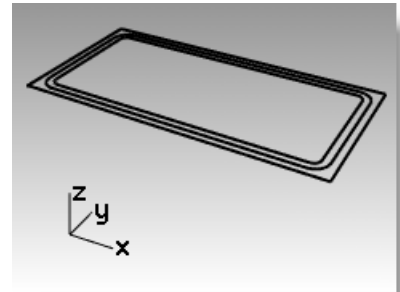
Draw rectangles from a center point with a length and width, and rounded corners:

- 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.

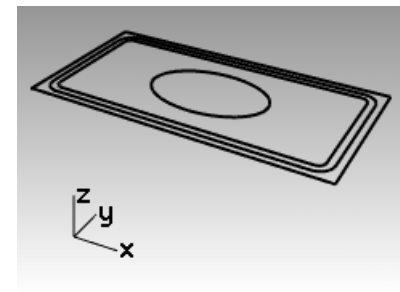
To change the type of corner, click Corner on the command line to toggle from circular rounded corners to conic rounded corners.

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




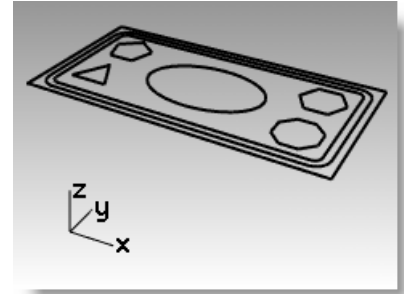
To draw an ellipse from its center and axis ends:

- 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.




To draw a polygon from its center and radius:

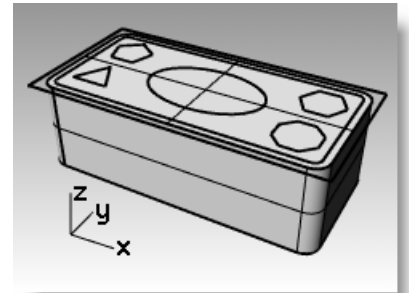
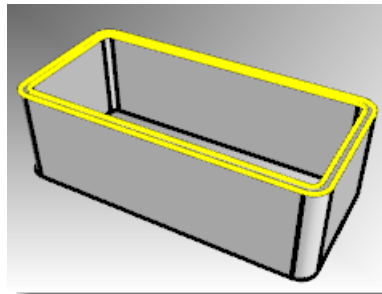
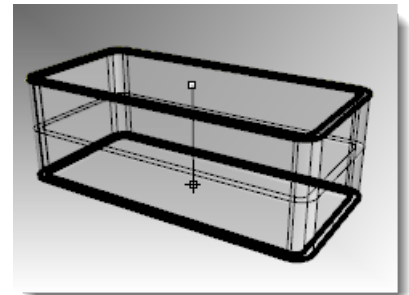
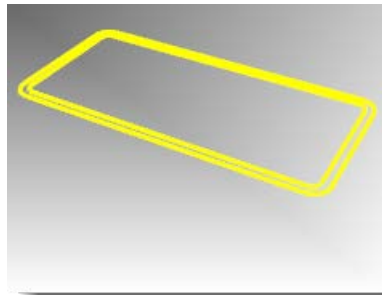
- 1 From the **Curve** menu click **Polygon**, and then click **Center, Radius**.

- 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.



To make the rounded rectangles solid:

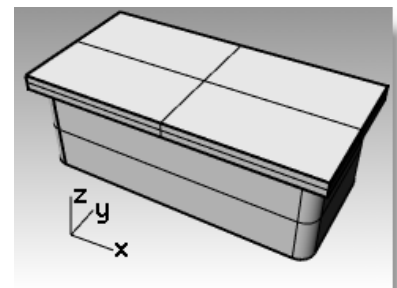
- 1 Select the larger and the smaller rounded rectangle.
- 2 From the **Solid** menu click **Extrude Planar Curve**, and then click **Straight**.

- 3 Drag both down to set the thickness and click or type a value for the thickness. Use a negative number to extrude down.

The two rounded rectangles are coplanar or in the same plane. Because of this the smaller rounded rectangle is removed from the larger one, and the difference is extruded.



To make the rectangle solid:

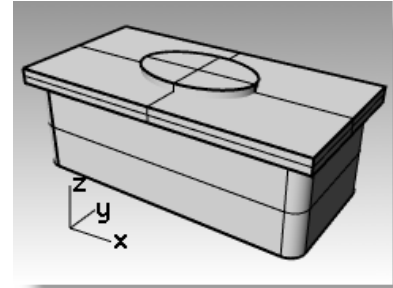
- 1 Select the rectangle.
- 2 From the **Solid** menu click **Extrude Planar Curve**, and then click **Straight**.
- 3 Set the **Bothsides** option to **No**.
- 4 Drag it up to set the thickness and click.



To make the ellipse solid:

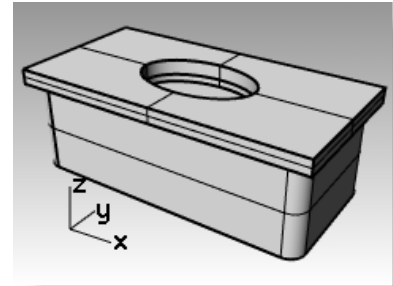
- 1 Select the ellipse.
- 2 From the **Solid** menu click **Extrude Planar Curve**, and then click **Straight**.
- 3 Set the **Bothsides** option to **Yes**.
- 4 Pick to set the thickness.

Make sure the thickness is enough to protrude through both sides of the rectangular solid.



To cut the ellipse solid from the rectangle:

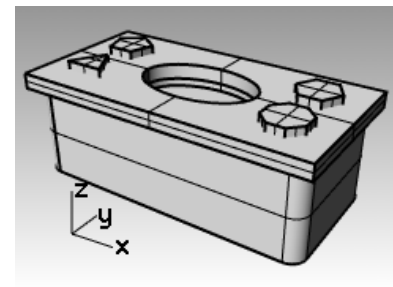
- 1 Select the solid rectangle.
- 2 From the **Solid** menu click **Difference**.
- 3 Select the solid ellipse, and press **Enter**.



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.

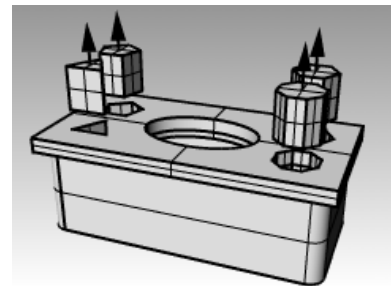
Make sure the thickness is enough to protrude through both sides of the rectangular solid.



To cut holes with the solid polygons:




- 1 Select the rectangular solid.
- 2 From the **Solid** menu click **Difference**.
- 3 On the command line click **DeleteInput**.
Make sure it is set to: **DeleteInput=No**.
- 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.



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 shape.
	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 <i>Alt</i> 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.
PersistentClose	Closes the curve as soon as there are two points placed. You can continue to pick points. The curve updates the shape while remaining closed.

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 for creating free-form curves is to draw lines that are accurately measured. Use these lines as guidelines. Another method for creating free-form curves is to use 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 **Grid Snap** off.

To draw a curve by control points:

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

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



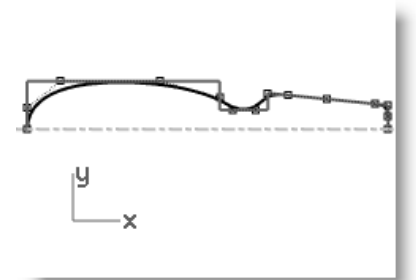
2 For the **Start of Curve**, snap to the end point of the polyline guideline.

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**.

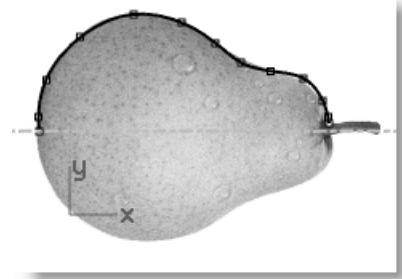
A free-form curve is drawn. The control points, while on the guideline, were not on the curve itself except at the two ends.




To draw a curve interpolate through points:

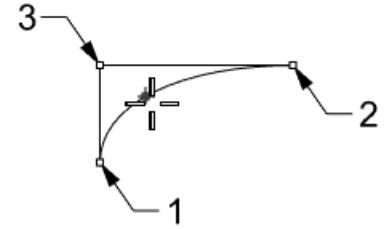
- 1 Change to the Interpolated Curve layer.
- 2 From the **Curve** menu click **Free-form**, and then click **Interpolate Points**. 
- 3 For the **Start of Curve**, snap to the intersection of the centerline and the image using the **Near** object snap.
- 4 For the **Next point**, continue to pick points 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.




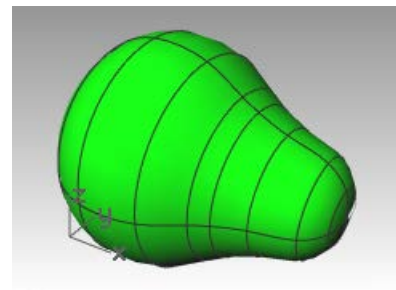
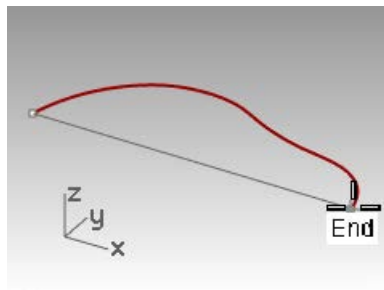
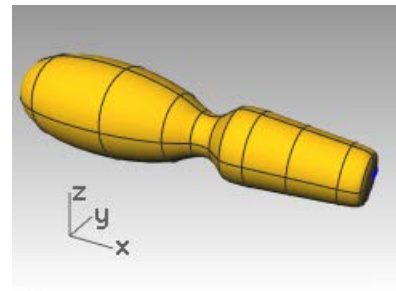
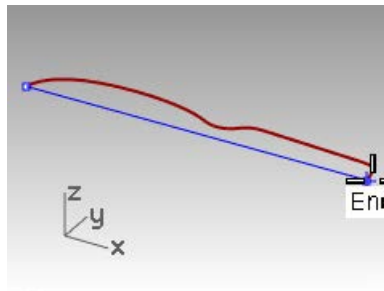
To draw a conic curve:

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





To make the 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**.
- 6 Repeat steps 2-5 for the interpolated curve.



Helix and Spiral


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.

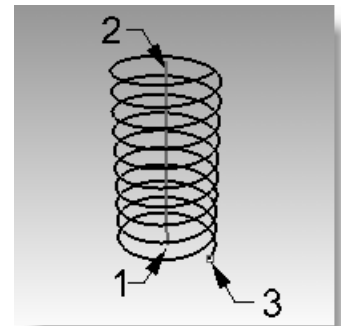
Option	Description
Vertical	The helix or spiral axis will be perpendicular to the construction plane in the active viewport.
AroundCurve	Select a curve for the helix or spiral to wind around to create "phone cord" shapes. This option supports History.
Flat	Allows you to draw a planar spiral.
Mode	Determines 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**.
- 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 Grid Snap off.

To draw a helix

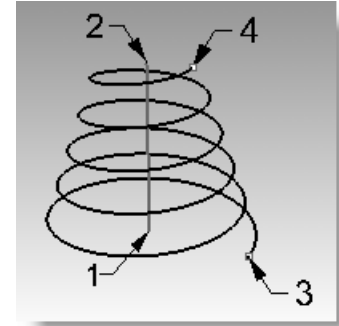
- 1 Change to the Helix layer.
- 2 From the **Curve** menu click **Helix**. 
- 3 For the **Start of axis**, snap to the end of the vertical line (1) in the Perspective viewport.
- 4 For the **End of axis**, snap to the end of the vertical line (2) in the Perspective viewport.
- 5 Click **Mode** on the 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.




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**.

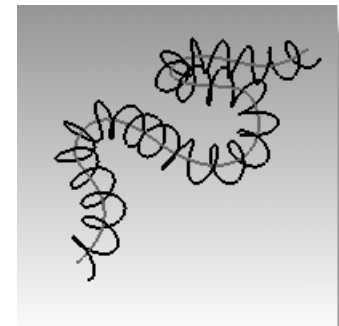
A spiral is created with a reverse twist and the distance between each turn is 15.




To draw a helix around a curve:

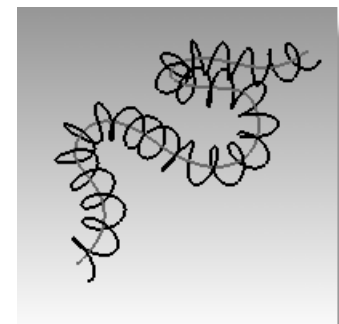
- 1 Change to the HelixAlongCurve layer.
- 2 From the **Curve** menu click **Helix**. 
- 3 Click **AroundCurve**.
- 4 Select the free-form curve.
- 5 Click **Mode=Turns**.
- 6 Click **Turns**
- 7 Type **25** and press **Enter**.
- 8 Click **ReverseTwist=No**.
- 9 For the **Radius** type **5** and press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

A helix is created around the curve. Undo this helix.

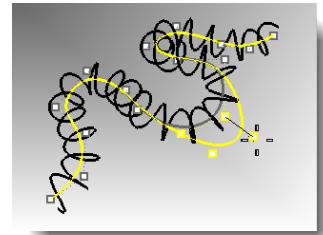
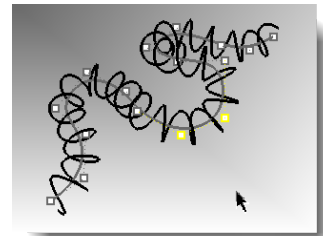


To draw a helix around a curve with History:

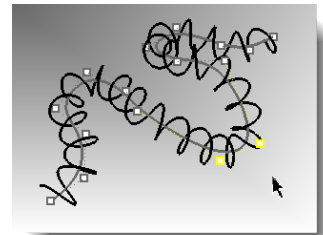
- 1 In the **Status bar**, click **RecordHistory** pane.
- 2 From the **Curve** menu click **Helix**. 
- 3 Click **AroundCurve**.
- 4 Select the free-form curve.
- 5 Click **Mode=Turns**.
- 6 Click **Turns**
- 7 Type **25** and press **Enter**.
- 8 Click **ReverseTwist=No**.



- 9 For the **Radius** type **5** and press **Enter**.
- 10 Pick a point for the Start point.
A helix is created around the curve.
- 11 Highlight the free-form curve. Type **PointsOn (F10)**.
- 12 Pick a few control points and pull the curve into a new shape.



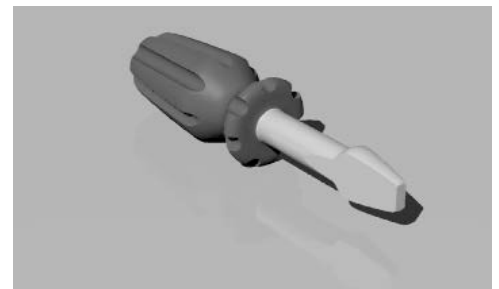
- 13 Helix will update automatically to the new shape for the free-form curve.



Exercise 32—Drawing free-form curves

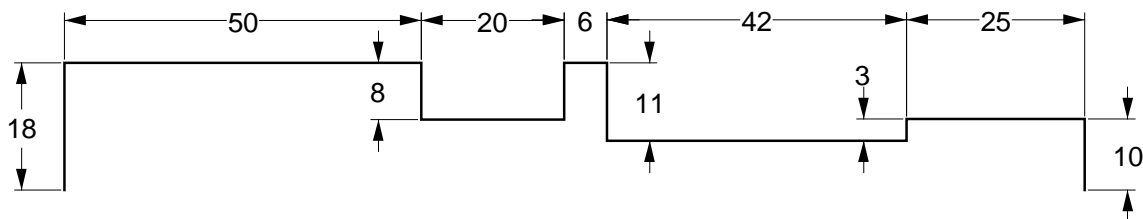
The use of free-form curves allows more flexibility to create complex shapes. In the following exercise we will draw the guidelines and free-form curves for a toy screwdriver.

- 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.



To create the construction lines:

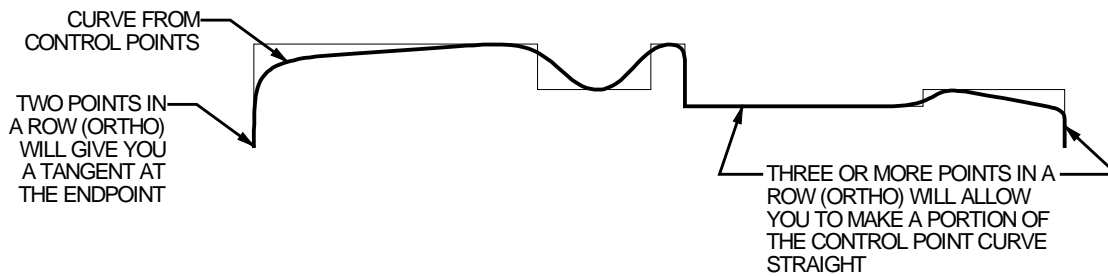
- 1 Change to the Construction layer.
- 2 Draw a **Polyline**, in the Top viewport, using these dimensions for the guideline.




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

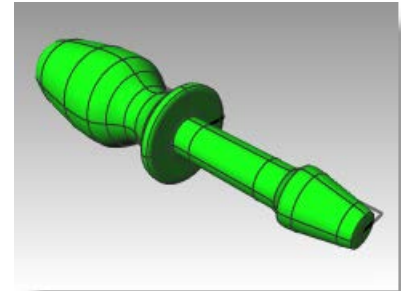
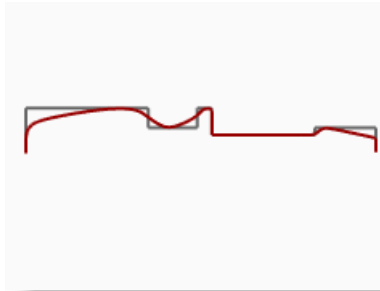
To create the curve by control points:

- 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 Grid 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**.

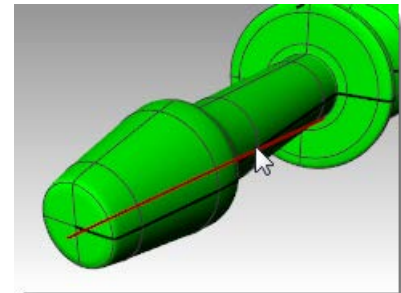
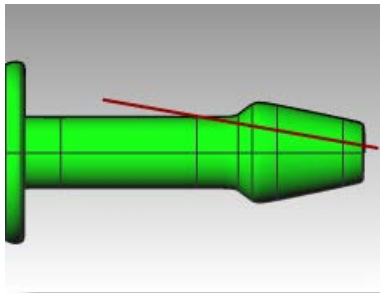


Your model is now a three-dimensional surface.

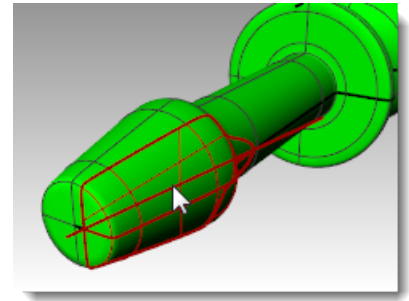
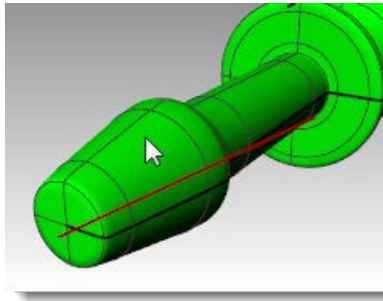
Add details on your own:

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.

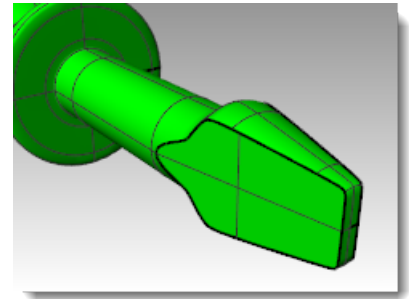
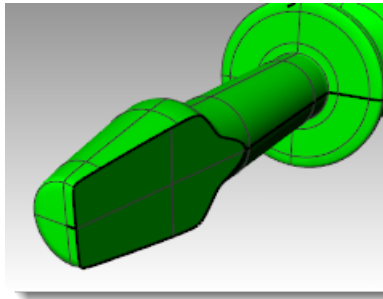
- 1 In the Top view draw a **Single Line** that will be used to slice the flat part from the screwdriver blade.
- 2 Select the line.
- 3 From the **Solid** menu, click **Solid Editing Tools**, and then click **Wirecut**.



- 4 For the **Select objects to cut**, choose the screwdriver, and then press **Enter**.
- 5 For the **First cut depth point**. Press **Enter to cut through object**, press **Enter**.
- 6 For the **Part to cut away**, choose the part you want to remove, and then press **Enter**.



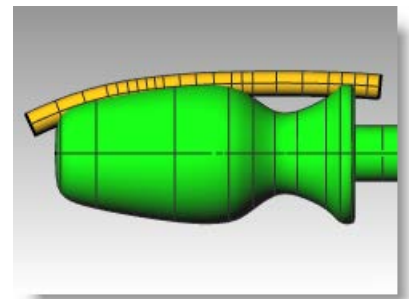
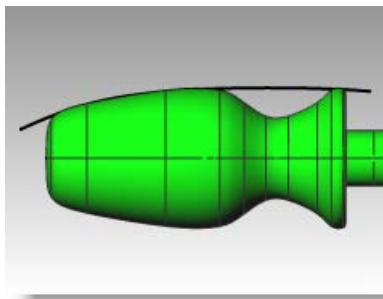
- 7 **Mirror** the line and repeat steps 3-6 to complete the other side of the blade.



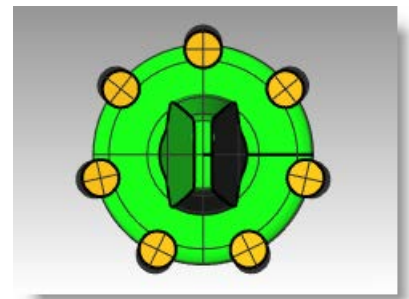
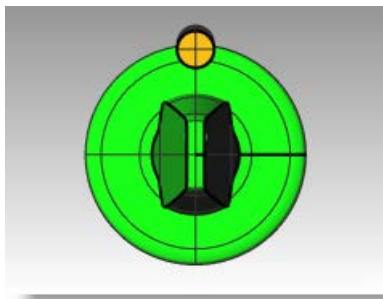
- 8 Draw a **Curve** along the top of the handle to be used for cutting grooves.

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.

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

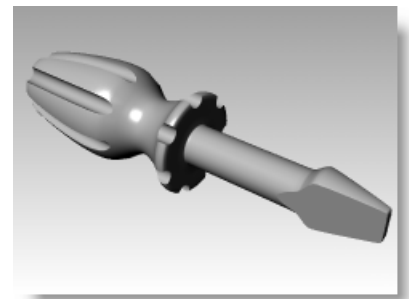
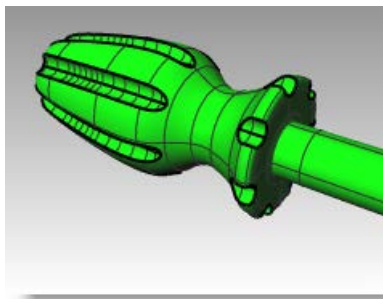


- 10 Use the **ArrayPolar** command to make copies of the pipe around the handle.



- 11 Use the **BooleanDifference** command to remove the pipes from the handle.

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



5 Editing Geometry

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.

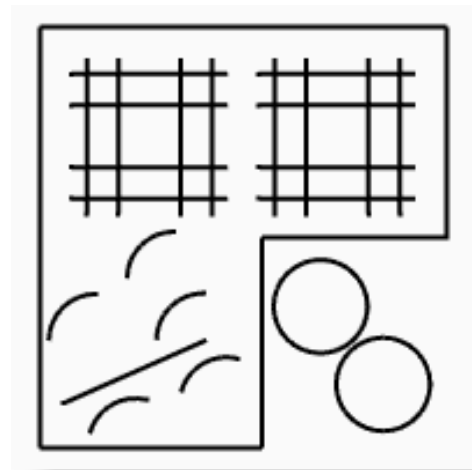
Rules to follow when filleting curves:

- 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.

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	Yes , joins the fillet to the curves. No , does not join the fillet to the curves.
Trim	Yes , trims the curves to the fillet arc. No , does not trim the curves.

Exercise 33—Fillet

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



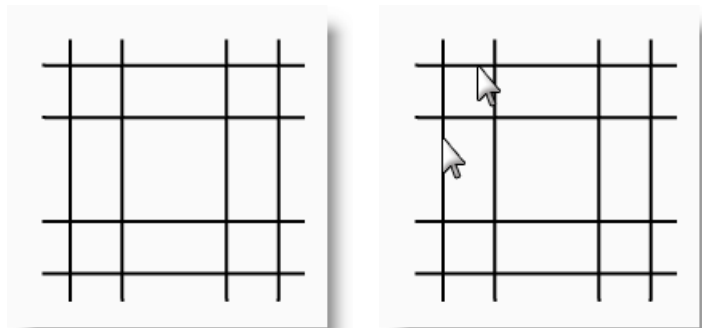
To connect lines that intersect:

- 1 From the **Curve** menu click **Connect Curves**.



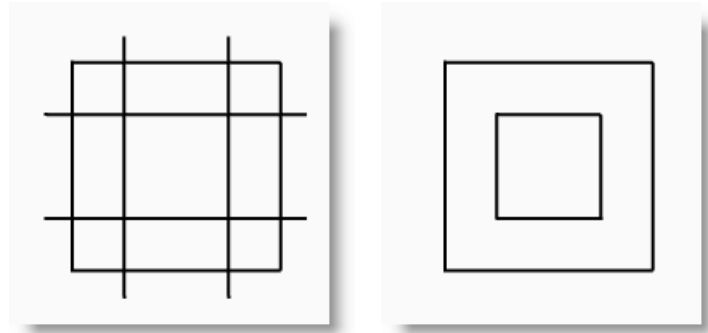
- 2 Select an outer vertical line.
3 Select an adjacent horizontal line.

The ends of the lines are trimmed to a corner.



- 4 Press **Enter** to repeat the command.
- 5 **Connect** the other corners, as shown.

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



To join the connected objects:

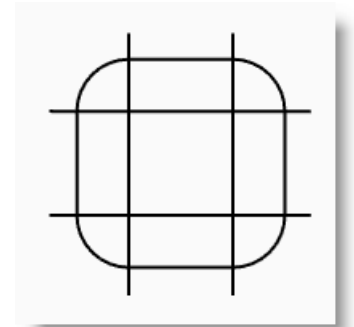
- 1 Select lines you just connected.

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

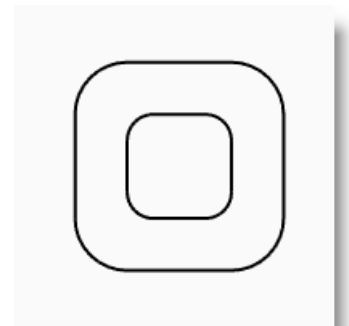
The objects are joined together. Curves join only if they touch.

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.

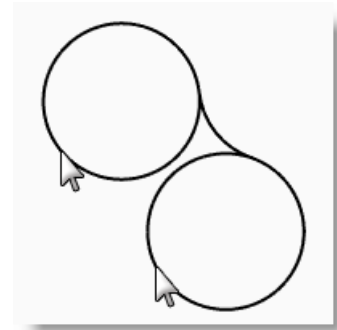
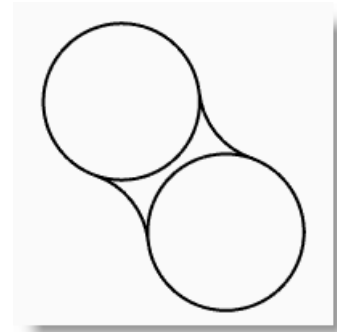
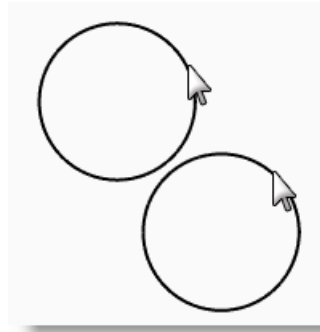


- 8 Press **Enter** to repeat the command.
- 9 Type **1** and press **Enter**.
This radius will be used for the smaller object.
- 10 Select an inner vertical line.
- 11 Select an adjacent horizontal line.
- 12 Fillet the other corners as shown.

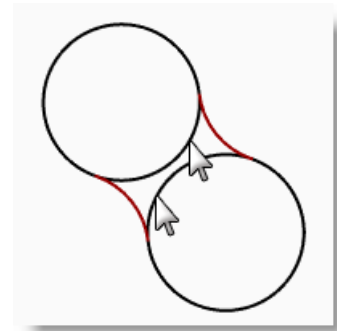


To fillet circles:

- 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.

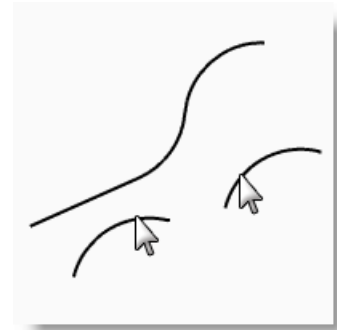
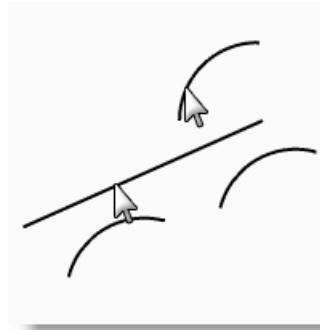


- 5 Repeat the command for the left side of the circles.
- 6 Select the two fillets you just created.
- 7 From the **Edit** menu click **Trim**.
- 8 For the objects to trim pick on the inner edge of each circle.




To fillet and join arcs and lines:

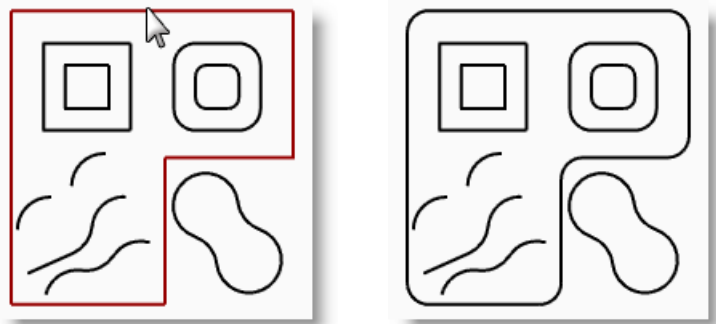
- 1 From the **Curve** menu click **Fillet Curves**. Set **Join=Yes** and **Trim=Yes**.
- 2 Select the line in the lower left part of the viewport.
Make sure you pick on the left half of the line.
- 3 Select the adjacent arc right above the selected line.
- 4 Repeat this procedure for the two arcs below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



Blend

Blending is another method to connect lines, arcs, or curves. There are three blend commands that work on curves, *BlendCrv (default)*, *Blend (Quick Blend)* and *ArcBlend*.

BlendCrv allows you to adjust the continuity with the input curves and has adjustable endpoints.

Selection Option	Description
Curves	Restricts the selection to curves.
Edges	Restricts selection to surface edges.
Option	Description
Continuity	Continuity describes the relationship between curves and surfaces. Types of continuity: Position (G0), Tangent (G1), and Curvature (G2), G3, and G4
Flip 1 & Flip 2	Reverses direction of specified curve.
Trim	Trims the input curves to the resulting curve.
Join	Joins the resulting curves.
CurvatureGraph	Displays the curvature graph for the blended curve.


ArcBlend consists of two arcs with adjustable endpoints and bulge.

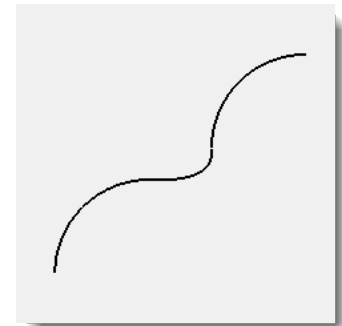
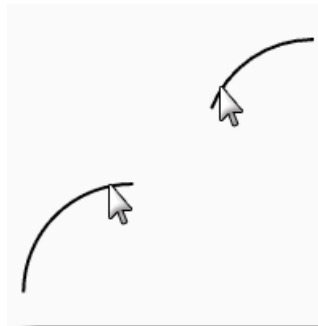
Option	Description
AlternateSolution	Offers the longer solution when more than one arc fills the requirements.
RadiusDifference	Makes it possible to draw an S-shaped blend where the difference of the two radii is specified. If the difference is positive the first radius will be bigger than second, and if negative the first radius will be smaller.
Trim	Trims the input curves to the resulting curve.
Join	Joins the resulting curves.

Blend (*Quick Blend*) creates a between two curves maintaining continuity with the input curves. It is similar to the adjustable blend curve without any adjustments.


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).

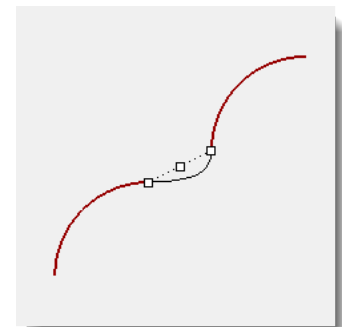
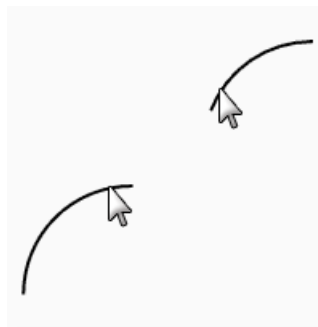
To blend two curves with Quick Blend:

- 1 From the **Curve** menu, **Blend Curves**, then click **Quick Curve Blend**. 
- 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 **Undo** to remove the blend.



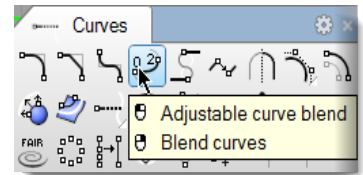
To blend two curves with ArcBlend:

- 1 From the **Curve** menu, **Blend Curves**, then click **Arc Blend**. 
- 2 Select the curves to blend near the ends that you want to connect.
The two input curves are connected with two arcs.
- 3 **Undo** to remove the blend.



To blend two curves with Adjustable curve blend:

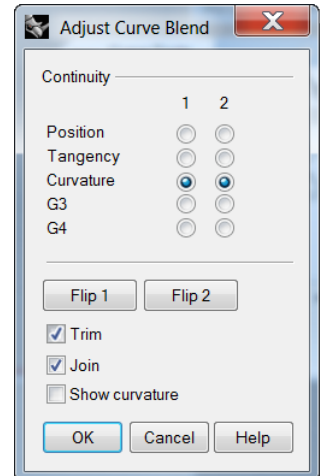
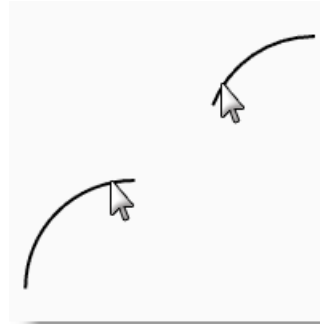
- 1 From the **Curve** menu, **Blend Curves**, then click **Adjustable Curve Blend**.



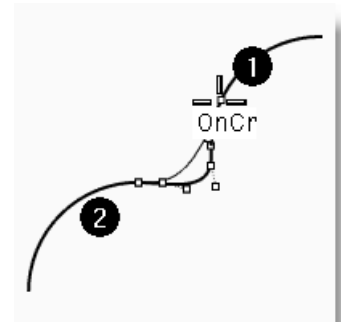
- 2 Select the upper right curve near the left end, then select the lower left curve near the right end to blend the curves.

You will see a preview of the default blend with control points displayed and a dialog.

- 3 Check the **Join** and the **Trim** option.

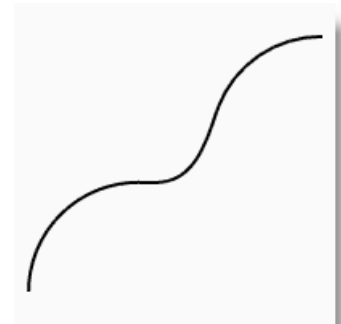
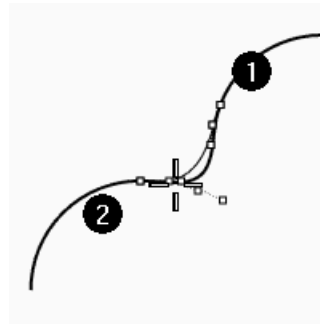


- 4 For the **Select control point to adjust**, select the point (1) and drag it further up the curve and click.



- 5 For the **Select control point to adjust**, select the point to the right of (2), drag it closer to (2) and click.


- 6 After making your adjustments, pick the **OK** button to make the blend.



To make a lofted surface with closed curves:

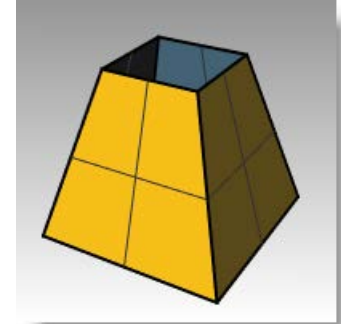
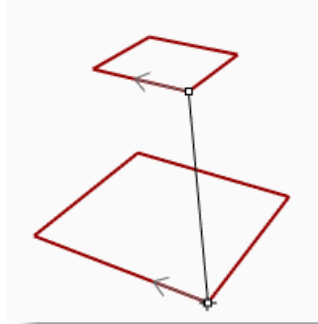
The curves in this model are at two different elevations. You are going to connect the curves at different elevations with a surface.

- 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.

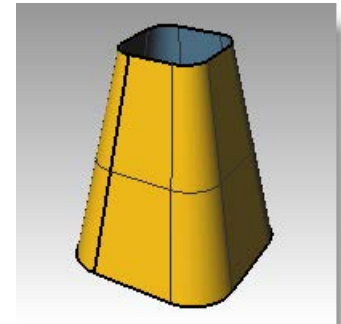
- 4 From the **Surface** menu click **Loft**. 

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.

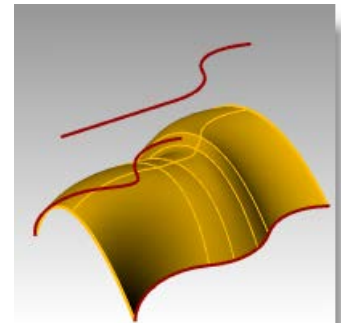
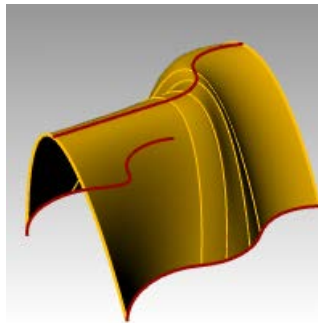


- 5 Press **Enter**.
- 6 In the **Loft Options** dialog box, click **OK**.
A surface is generated between the two closed polylines.
- 7 Repeat the procedure for the rounded squares.
- 8 In the **Loft Options** dialog box, click **OK**.

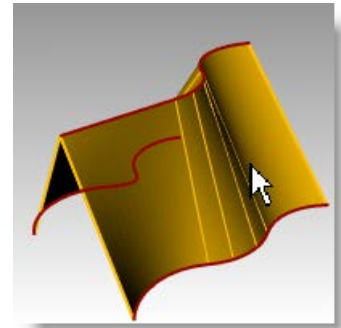


To make a lofted surface with open curves:

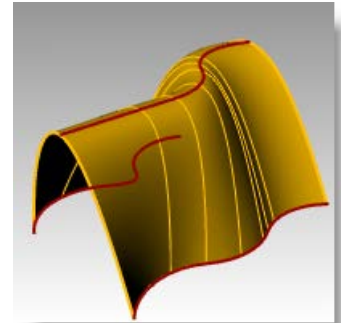
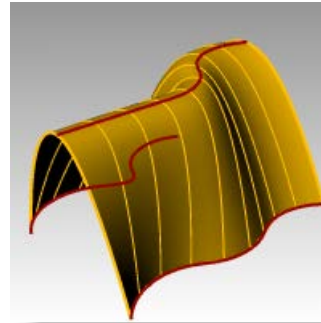
- 1 Repeat the **Loft** command for the three open curves.
- 2 In the **Loft Options** dialog box, change the **Style** to **Loose**, click **Preview**.



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



- 4 Change the **Style** to **Normal**, click **Preview**.
 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**.
 7 Change back to **Do not simplify**, click **OK**.



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.

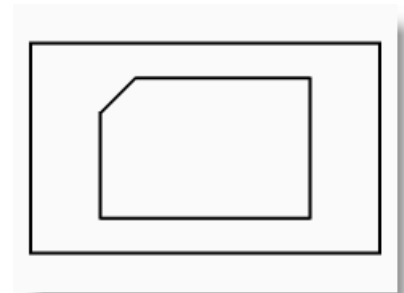
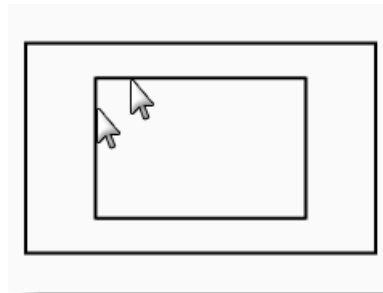
Option	Description
Distances	<p>The first chamfer distance specifies the distance from the chamfer end on the first curve to the point where the two curves would intersect. The second chamfer distance specifies the distance of the chamfer end on the second curve to the intersection point.</p> <p>A chamfer 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.

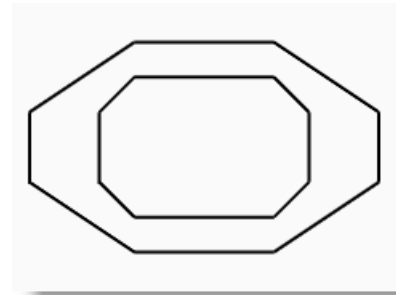
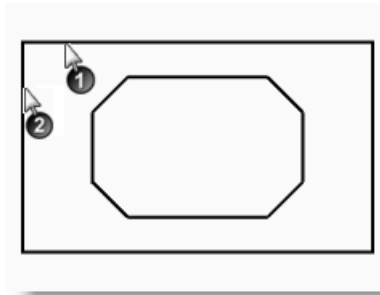
To chamfer lines:

- From the Curve menu click **Chamfer Curves** .
- For the **Select first curve to chamfer**, type **1,1** and press **Enter**, to set the distances.
- Set **Join=Yes**.
- Select one of the inner vertical lines.
- Select an adjacent horizontal line.
- Continue creating chamfers on all of the corners.



- 7 Press **Enter** to repeat the command.
- 8 For the **Select first curve to chamfer**, 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 from the intersection of the two curves on first curve selected, the second value is the distance from the intersection of the two curves on the second line selected.

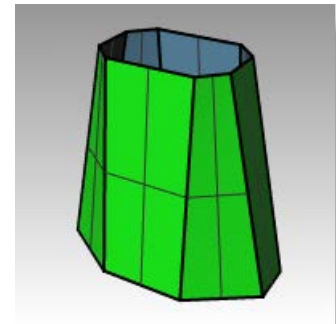
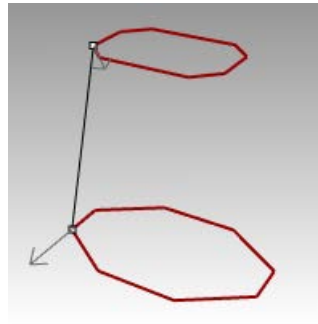


To turn the curves into surfaces:

- 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**.

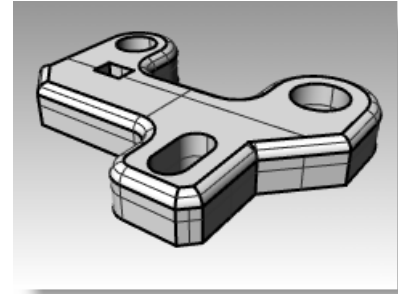
A surface is generated between the two chamfered rectangles.

- 6 **Save** your model.



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**.



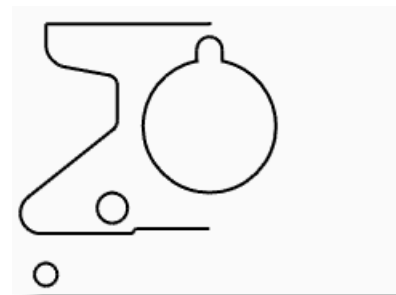
Move

Use Move to move objects without changing orientation or size.


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

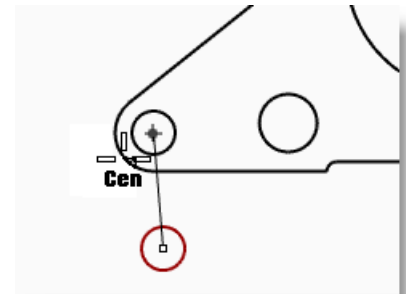
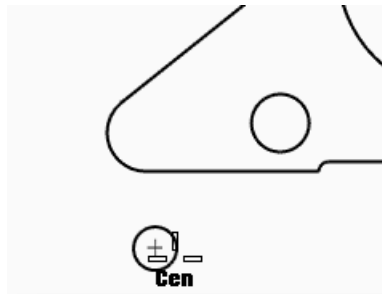
Exercise 36—Move

- 1 **Open** the model **Move.3dm**.
- 2 Turn off both **Ortho** and **Grid Snap** so you can move objects freely.
- 3 Turn **Cen** object snap on.



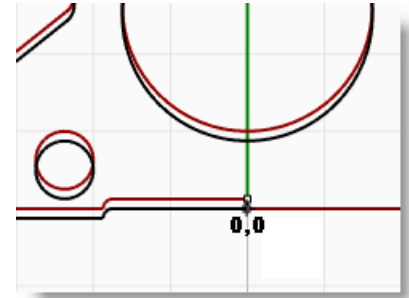
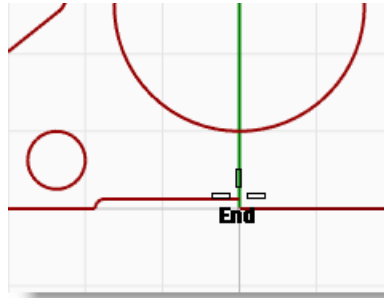
To move using object snaps for placement:

- 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.



To move using absolute coordinates:

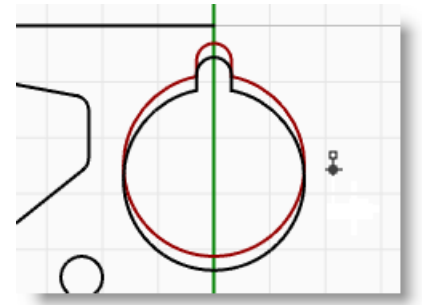
- 1 From the **Edit** menu click **Select Objects**, and then click **Curves**.
- 2 From the **Transform** menu click **Move**.
- 3 For the **Point to move from**, snap to the end of the line at the lower part of the object.
- 4 For the **Point to move to**, type **0,0** and press **Enter**.



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

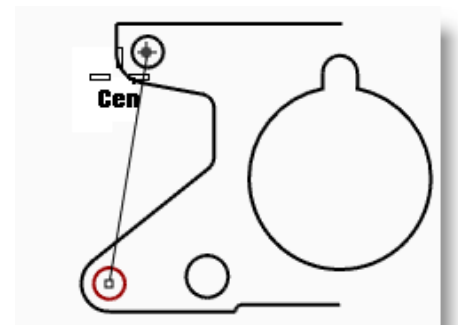
Copy duplicates selected objects and places them in a new location. The command can repeat to create more than one copy in the same command sequence.

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

Exercise 37—Copy

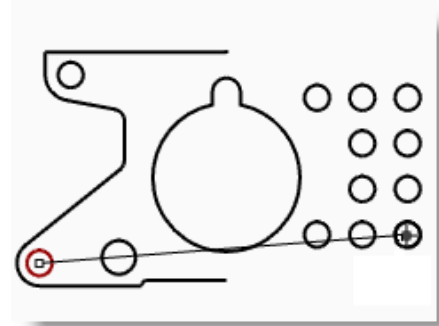
To copy using object snaps for placement:

- 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.



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 <i>Ctrl+Z</i>	Undo	Cancels the effects of the last command.
	Right-click or <i>Ctrl+Y</i>	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.

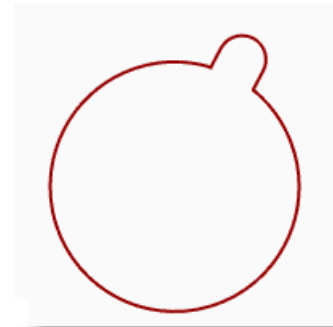
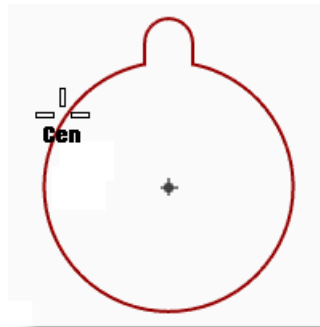
Rotate

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**.



Group

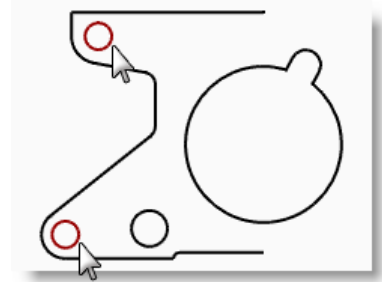
Grouping objects allows all members of the group to be selected as one. You can then apply commands to the entire group.

Rhino Button	Mouse Button	Command	Description
	Left-click or <i>Ctrl+G</i>	Group	Makes a group from selected objects.
	Left-click or <i>Ctrl+Shift+G</i>	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:

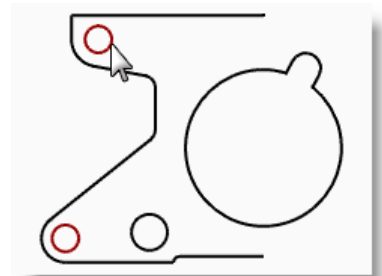
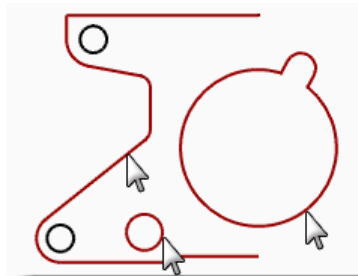
- 1 Select the two circles that you placed.
- 2 From the **Edit** menu click **Groups**, and then click **Group**.



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 For the **Select Group**, select one of the circles in the previous group.

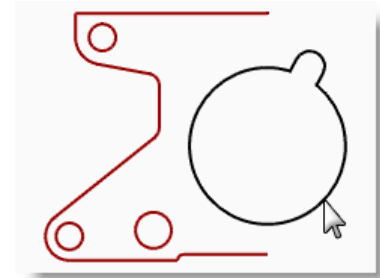
The objects are part of the group.



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.



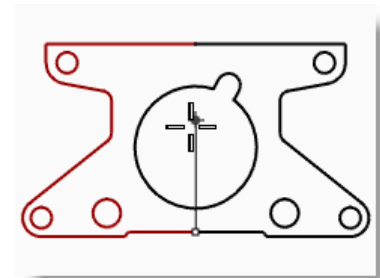
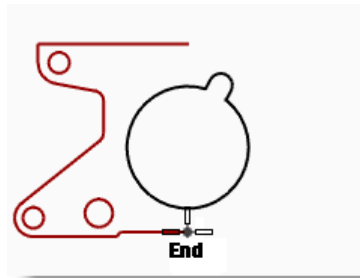
Mirror

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 For the **Start of Mirror plane** 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.

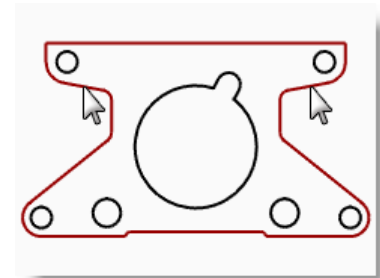


Join

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**.




Scale

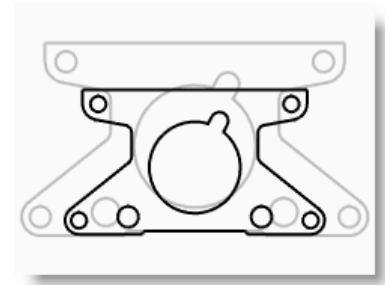
Scale changes the size of existing objects without changing their shape. This command scales three-dimensional objects equally along all three axes. There are also 2-dimensional, one-dimensional, and non-uniform scale commands.

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—Scaling objects

- 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**.

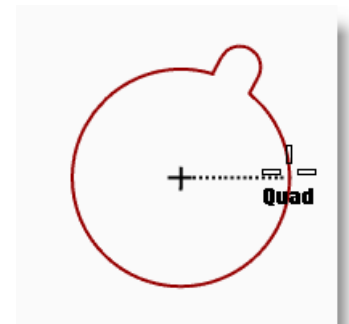
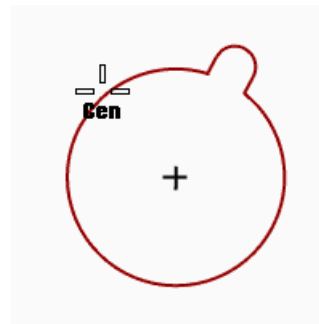
The whole part scaled to 75 percent of its original size.



To Scale2-D using the reference point option:

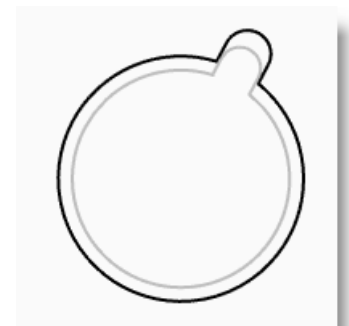
- 1 Select the slotted circle.
- 2 From the **Transform** menu click **Scale**, and then click **Scale 2-D**.
- 3 For the **Origin point**, snap to the center of the slotted circle.
- 4 For the **First reference point**, snap to the quadrant of the slotted circle.

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



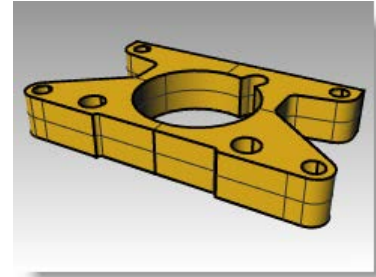
- 5 For the **Second reference point**, type **1.375** and press **Enter**.

The slotted circle now has a larger radius of 1.375.




To make it solid:

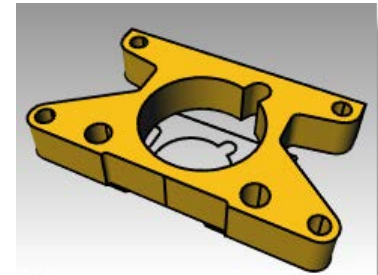
- 1 From the **Edit** menu click **Select Objects**, and then click **Curves**.
- 2 From the **Solid** menu click **Extrude Planar Curves**, and then click **Straight**.
- 3 For the **Extrusion distance**, type **1** and press **Enter**.




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**.

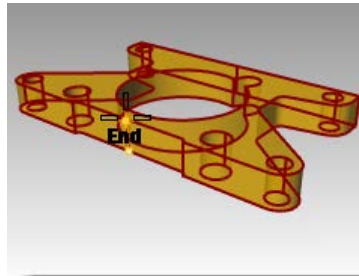
The solid is larger in every dimension.



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**.

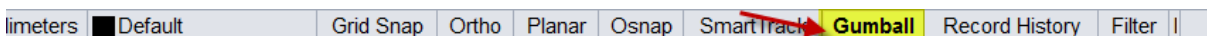
The object is half as thick.



Editing with the gumball

Gumball displays a widget on a selected object which is used to facilitate the direct editing. The gumball provides move, scale, and rotate transformations around the gumball origin.

Click the Gumball pane in the status bar.



Gumball actions:

- Drag gumball arrows to Move the object.
- Drag scale handles (*squares*) to Scale the object in one direction.
- Drag arcs to rotate the object.
- Tap the **Alt** key after starting to drag to toggle copy mode.

- Click a control handle to enter a numeric value.
- Hold the **Shift** key during Scale to force 3-D scale.

Gumball Controls

1. Axis plane indicator
2. Free move origin
3. Menu ball

Move arrows,

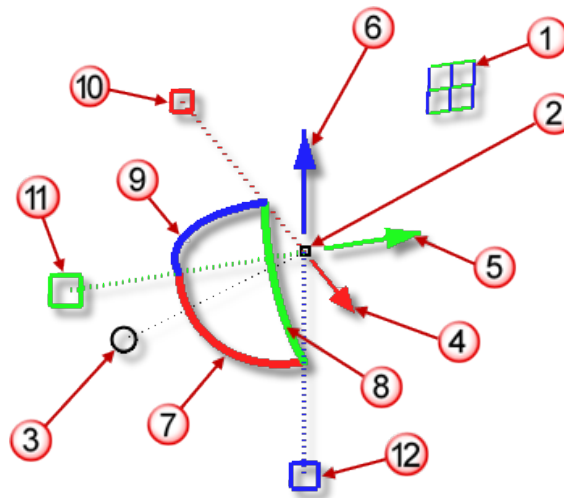
4. Move X,
5. Move Y
6. Move Z

Rotation arcs

7. Rotate X
8. Rotate Y
9. Rotate Z

Scale handles

10. Scale X
11. Scale Y
12. Scale Z

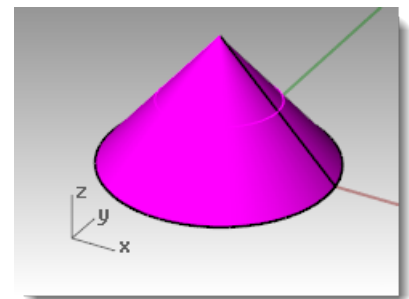


Exercise 43—Gumball basics

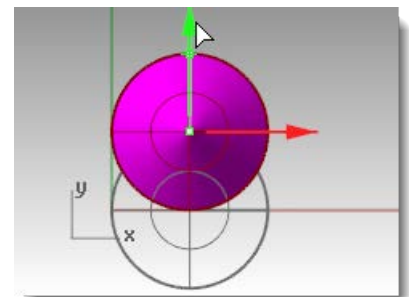
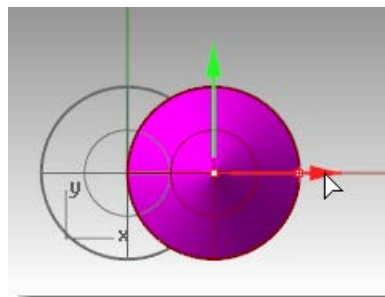
To move geometry with the Gumball:

In this exercise we will drag gumball arrows to Move an object. There are 3 arrows: x (red), y (green) and z (blue).

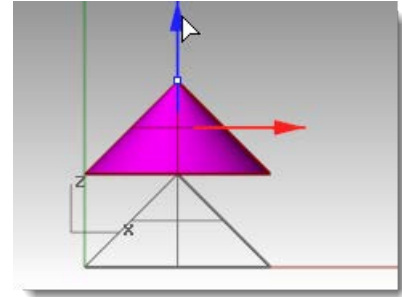
- 1 **Open** the model **Gumball.3dm**.



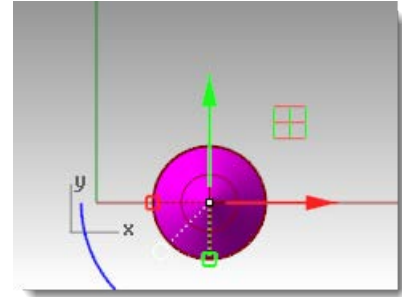
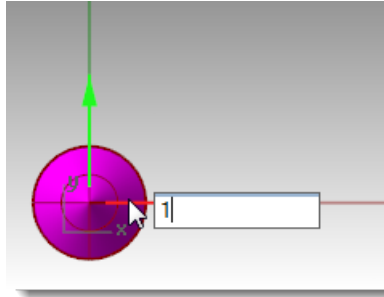
- 2 Highlight the cone in the Top viewport.
- 3 **Drag** the **red arrow** to move the object in the positive **x** or negative **x** direction.
- 4 **Drag** the **green arrow** to move the object in the positive **y** or negative **y** direction.



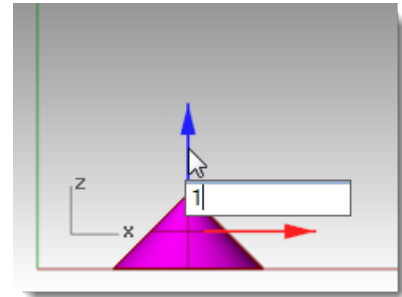
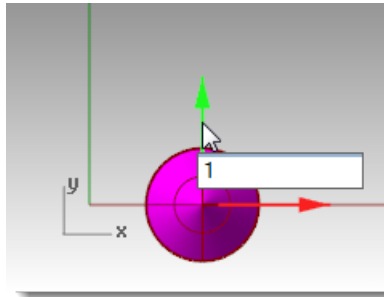
- 5 Highlight the cone in the Front viewport.
- 6 **Drag** the blue arrow to move the object in the positive **z** or negative **z** direction.
- 7 **Undo** enough times to get back to the original model.



- 8 Highlight the cone in the Front viewport.
- 9 Click **x Move Arrow** (red) to enter numeric value of **1**.
Cone moves a distance of 1 units to the right.



- 10 Repeat for **y Move Arrow** and **z Move Arrow**.



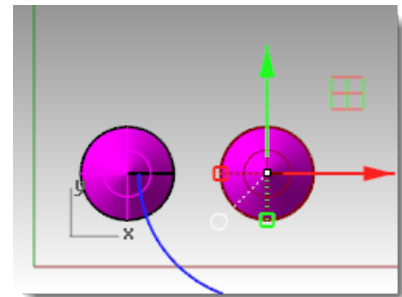
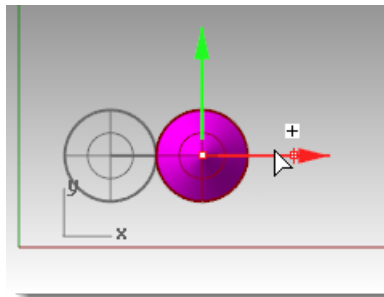
To copy objects with Gumball:

In this exercise you will drag objects with the gumball and tap the **Alt** key after starting to drag to toggle copy mode.

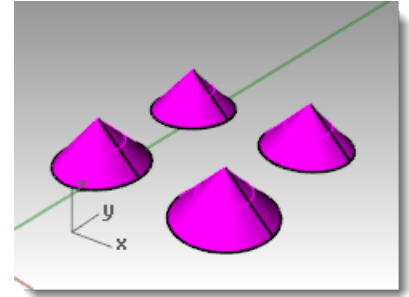
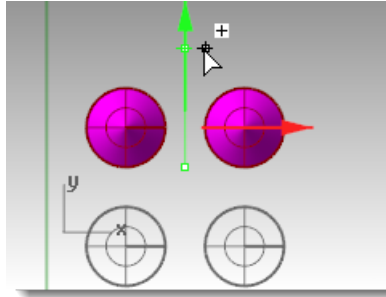
- 1 Highlight the cone in the Top viewport.
- 2 **Drag** the **red arrow** to move the object in the positive x or negative x direction.
- 3 While still dragging, tap the **Alt** key.

A plus appear to the right of the red arrow.

When you release the mouse button, a copy of the object is created.



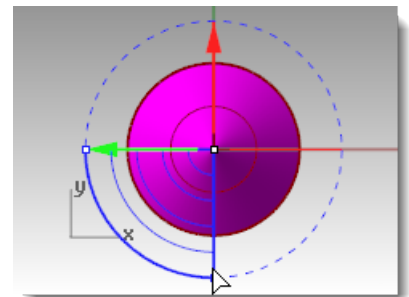
- 4 Highlight the two cones in the Top viewport.
- 5 **Drag** the **green arrow** to move the objects in the positive y direction.
- 6 While still dragging, tap the **Alt** key.
A plus appear to the right of the green arrow.
When you release the mouse button, a copy of the object is created.
- 7 **Undo** enough times to get back to the original model.



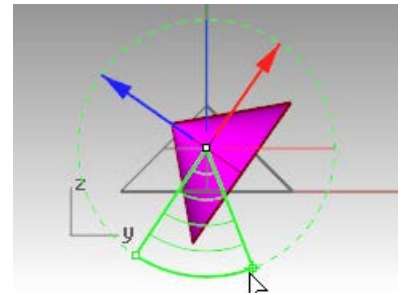
To rotate objects with Gumball:

Drag arcs to rotate the object.

- 1 Highlight the cone in the Top viewport.
- 2 **Press** and **drag** along blue arc to rotate cone.



- 3 Make the Right viewport active.
- 4 **Press** and **drag** along green arc to rotate cone.
- 5 **Undo** enough times to get back to the original model.

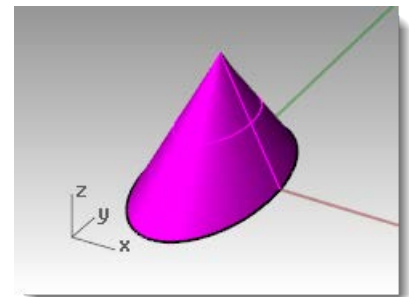
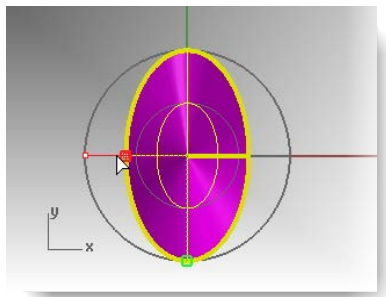


Scaling with Gumball

- Drag scale handles (*squares*) to Scale the object in one direction.
- Click a scale control handles (*squares*) to enter numeric value.
- Hold the **Shift** key during Scale to force 3-D scale.

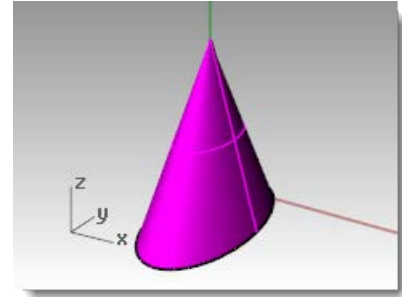
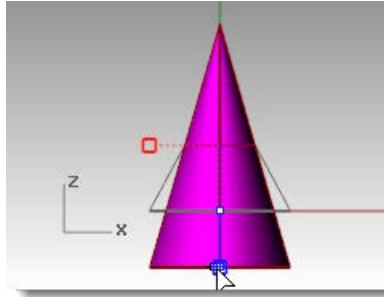
To scale objects with Gumball:

- 1 Highlight the cone in the Top viewport.
- 2 **Drag** the **red scale handle** (square) to scale the object.
Release the mouse button to complete the scale.

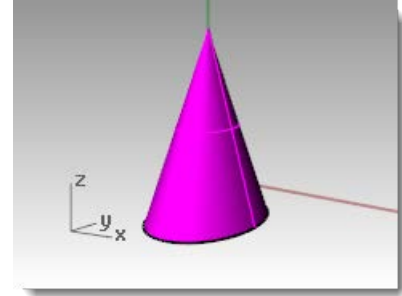
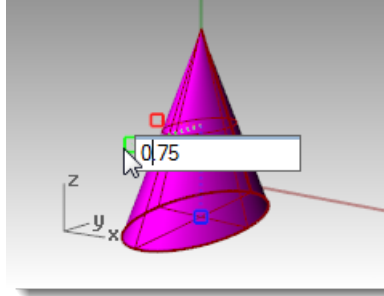


- 3 Highlight a cone in the Front viewport.
- 4 **Drag** the **blue scale handle** (*square*) down to scale the object larger in height.

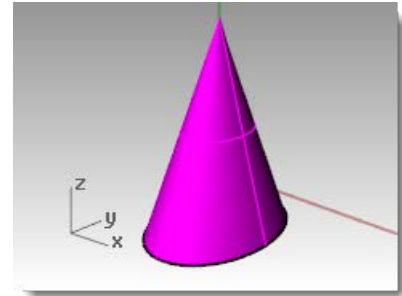
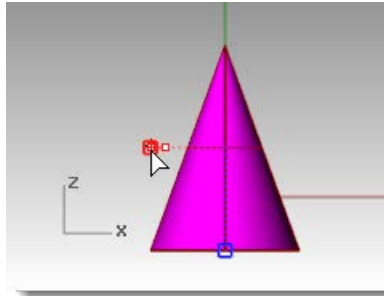
Release the mouse button to complete the scale.



- 5 Try clicking a **scale control handle** (*square*) and enter numeric value or scale factor like **.75**.



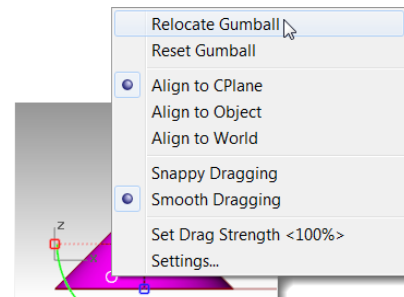
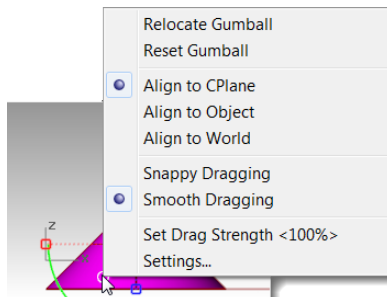
- 6 Highlight the cone in the Front viewport.
- 7 While holding down the **Shift** key, drag the red scale handle (*square*) to scale the object uniformly in the x, y, and z direction. Let go to complete the scale.
- 8 **Undo** enough times to get back to the original model.



Relocating Gumball Widget

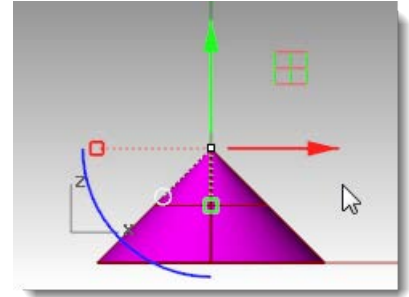
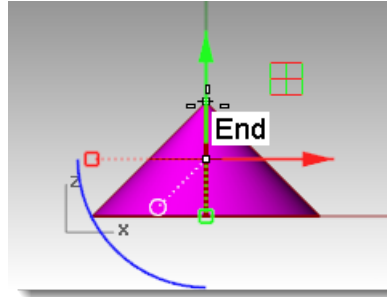
The gumball provides move, scale, and rotate transformations around the gumball origin.

- 1 Highlight the cone in the Front viewport.
- 2 Left click on the Gumball menu marker (*circle*).
- 3 Click **Relocate Gumball**.



- 4 With the **End point** osnap, pick on the top of the cone and press **Enter**.

The Gumball origin is now located at top of cone. All editing is in reference to the new origin.

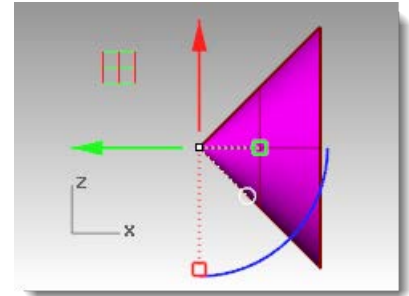
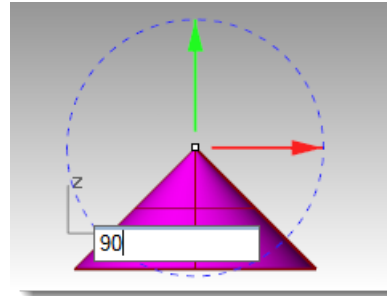


- 5 Click on the **Blue arc**.

A text box will appear. You can type a rotation angle into this box to rotate an object an exact angle.

- 6 Type **90** and press **Enter**.

The cone is rotated exactly 90 degrees in a counter clockwise direction.

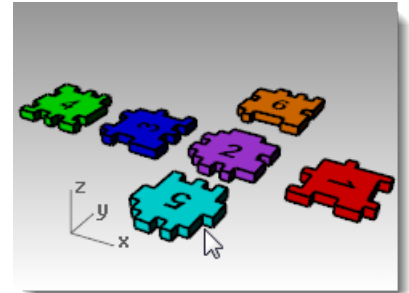
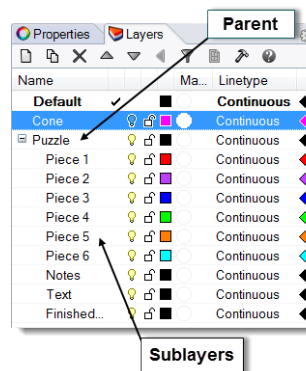


Exercise 44—Practice with Gumball

Gumball Exercise:

- 1 In the **Layer** panel, do the following:
Make Default the current layer.
Turn off the Cone layer.
Turn on the parent Puzzle layer.

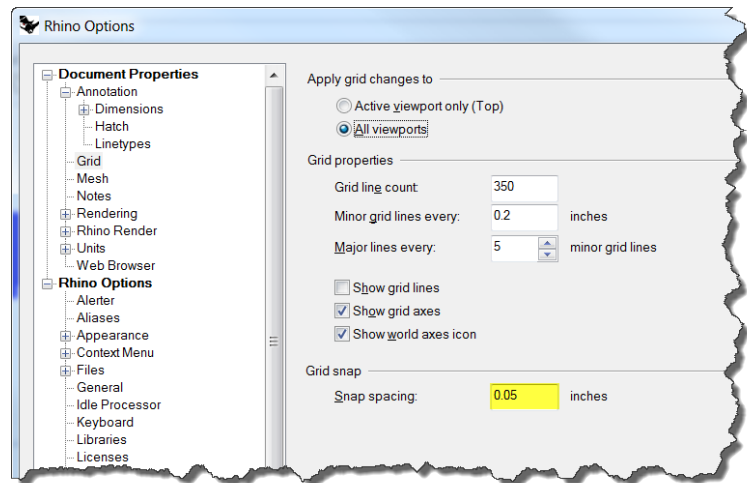
Note: The Puzzle layer contains sublayers. Turning on or off the parent Puzzle layer, affects the sublayers visibility, too.



- 2 From the **View** menu, select **Zoom** and **Zoom Extents All** (**Alt+Ctrl+E**) to see the puzzle pieces.

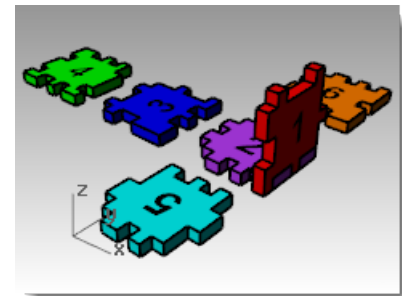
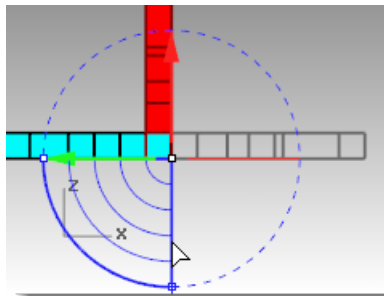
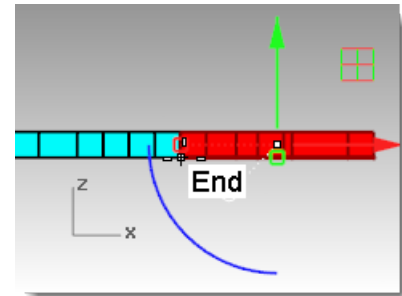
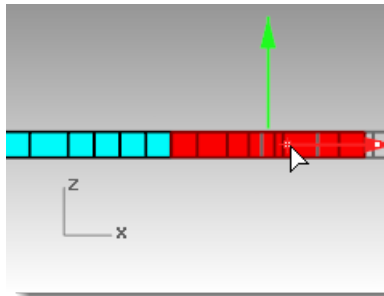
- 3 From the **Status bar**, turn **Ortho** and **Grid Snap** on. Next right click **Grid Snap** and click **Settings**.
- 4 For the **Snap Spacing**, type a value of **0.05**.
- 5 Click **OK**.

Hint: Grid Snap, Ortho, and Gumball should all be on in the Status bar.



To reposition the puzzle pieces:

- 1 Highlight the red puzzle **Piece 1** in the Front viewport.
- 2 Left click on the **Gumball menu marker (circle)**.
- 3 Click **Relocate Gumball**.
- 4 With the End point osnap, pick lower left corner of the piece.
- 5 Press and drag along blue arc to rotate pieces to 90 degrees.



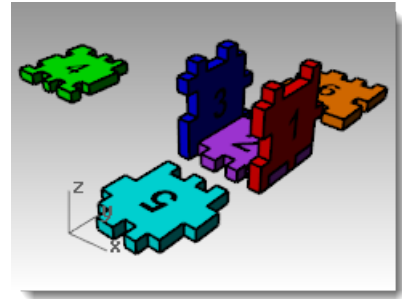
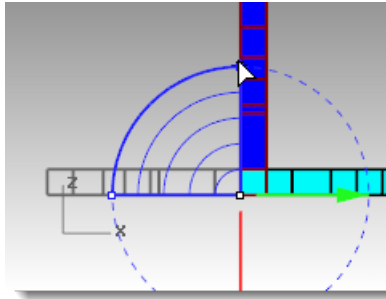
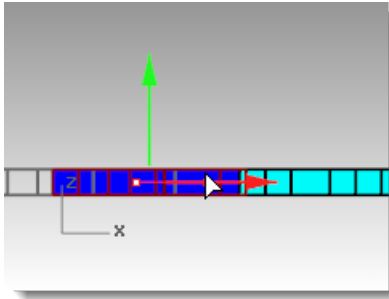
To rotate and move the other Puzzle Pieces:

Repeat these steps for puzzle **Pieces 3, 5, 6**.

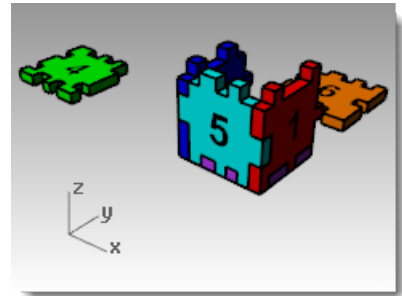
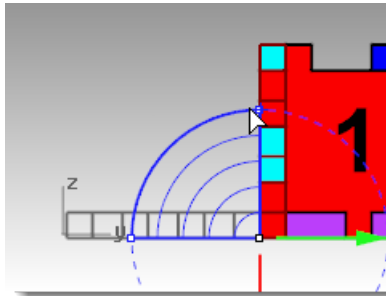
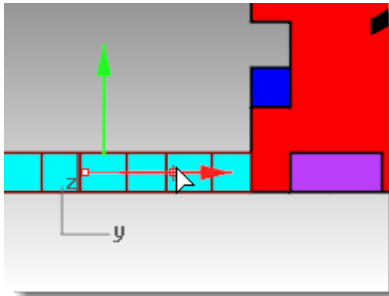
- 1 **Move** with **Gumball**
- 2 **Relocate Gumball** origin.
- 3 **Rotate** with **Gumball**
Use the appropriate viewport for rotation.

Hint: Rotate Piece 3 in the Front viewport, rotate Piece 5 and 6 in the Right viewport.

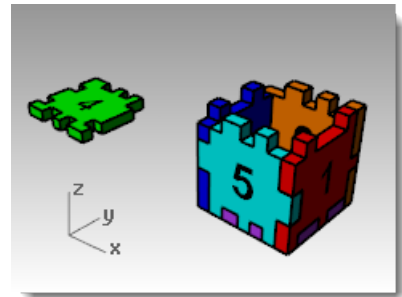
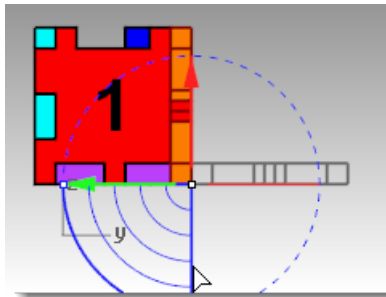
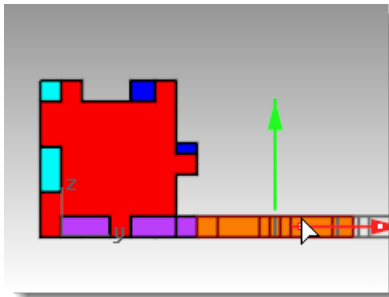
Puzzle Piece 3



Puzzle Piece 5

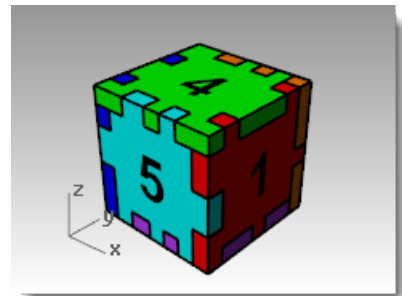
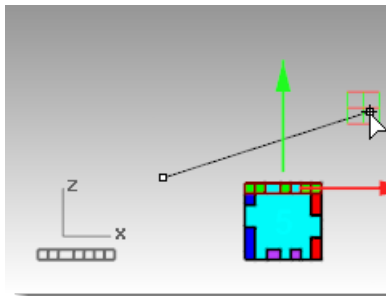
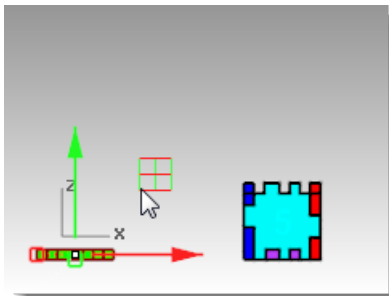


Puzzle Piece 6



- 4 Now use Gumball to move the final **Piece 4** into position on the top of the box using the **Axis plane indicator**.

Drag by the plane icon to constrain movement to that plane.





Trim

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

Exercise 45—Trim

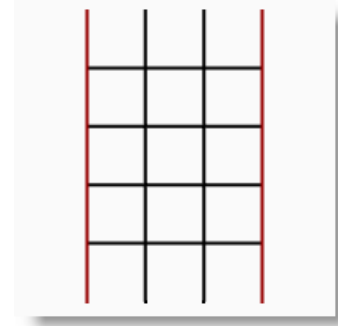
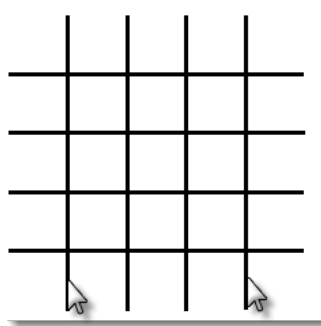
In this exercise, we will to pre-select the cutting objects.

To trim curves:

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

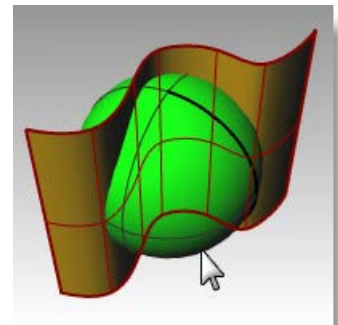
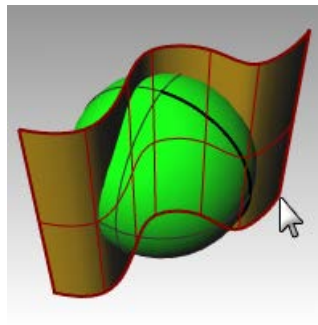
The lines are trimmed to the cutting edges.

- 7 Press **Enter**.



To trim surfaces:

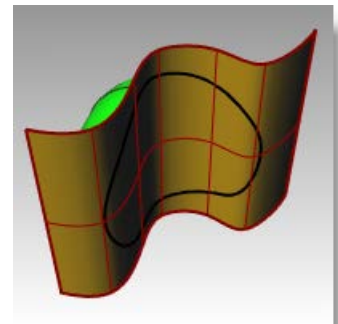
- 1 **Zoom Window** around a sphere and surface in the Perspective viewport.
- 2 Select the surface that intersects the sphere in the Perspective viewport as the cutting object.
- 3 From the **Edit** menu click **Trim**.



- 4 For the **Object to trim**, pick the right side of the sphere.

The sphere is trimmed at the surface.

- 5 Press **Enter**.




Split

Split divides objects into parts using other objects as cutters. The Split command breaks the object where it intersects the cutting object but does not delete anything.

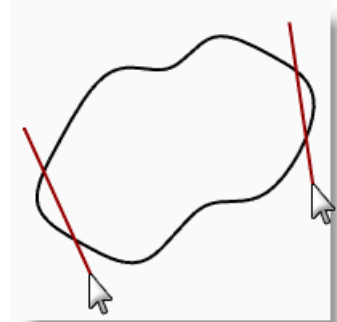
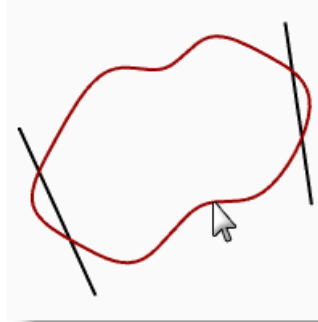
Exercise 46—Split

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

To split a curve:

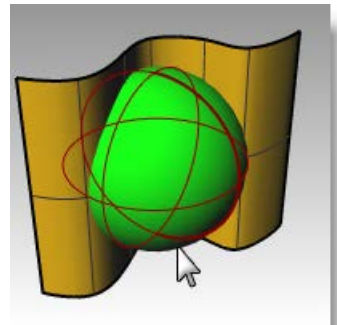
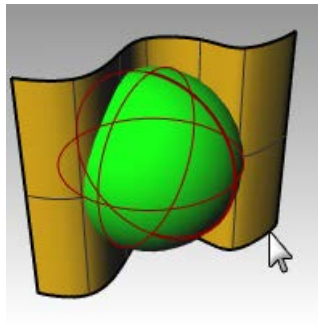
- 1 **Zoom Window** around the closed curve in the lower right corner of the Top viewport.
- 2 Select the closed curve.
- 3 From the **Edit** menu click **Split**. 
- 4 Select the lines and press **Enter**.

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

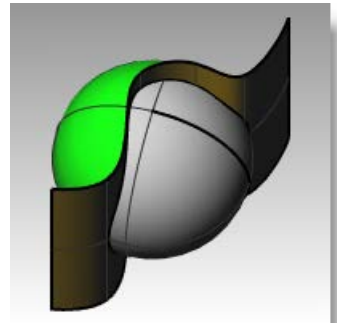


To split a surface:

- 1 From the **View** menu click **Zoom**, and then click **Zoom Extents All**.
- 2 Select the surface that intersects the sphere.
- 3 From the **Edit** menu click **Split**.
- 4 Select 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 47—Extend

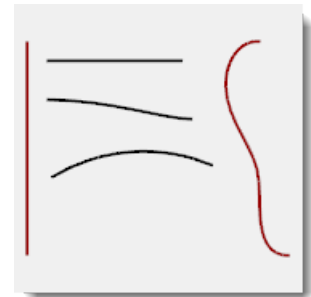
1 Open the model **Extend.3dm**.

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

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

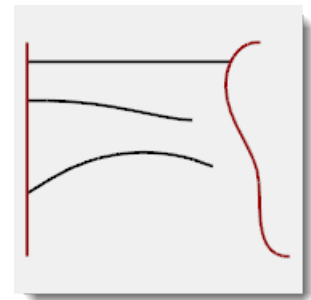
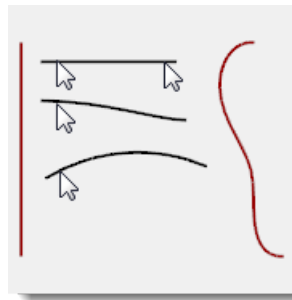
4 For the **Select curve to extend**, click **Type=Natural** on the command line.

5 Change to **Type=Line**.



6 Select both ends of the top line and the left end of two curves.

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

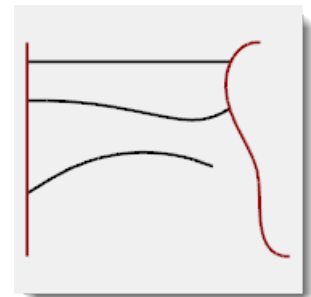
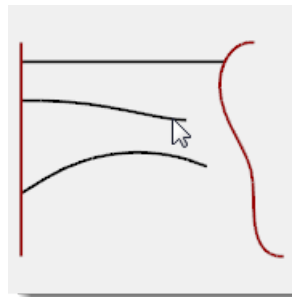


7 For the **Select curve to extend**, click **Type=Line** on the command line.

8 Change to **Type=Arc**.

9 Select the right end of the middle curve.

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

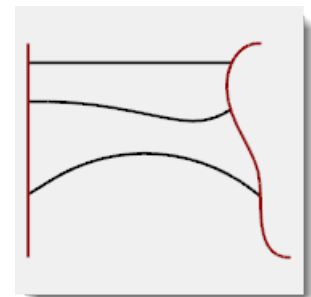
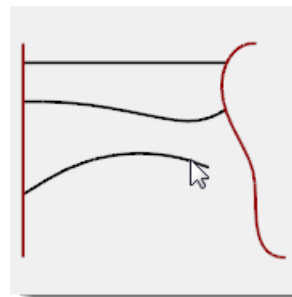


10 For the **Select curve to extend**, click **Type=Arc** on the command line.

11 Change to **Type=Smooth**.

12 Select the right side of the bottom curve.

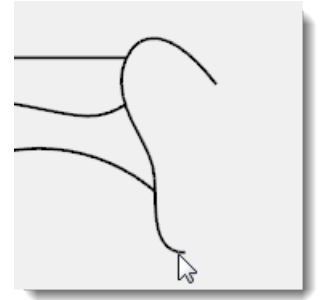
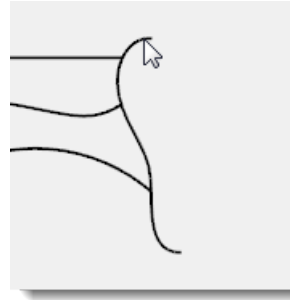
The curve extends with a curvature (G2) continuous extension to touch the boundary.



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

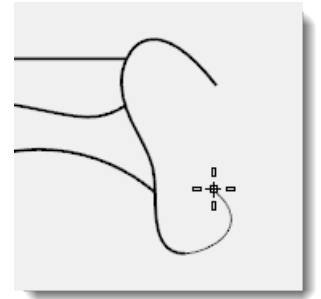
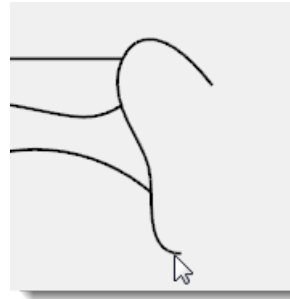
To extend with a set extension length:

- 1 From the **Curve** menu click **Extend Curve**, and then click **Extend Curve**.
- 2 For the **Select boundary objects or enter extension length**. Press **Enter** for dynamic extend, 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.

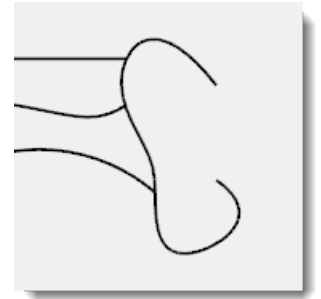


To extend a curve dynamically:

- 1 From the **Curve** menu click **Extend Curve**, and then click **Extend Curve**.
- 2 For the **Select boundary objects or enter extension length**. Press **Enter** for dynamic extend, press **Enter** for a dynamic extension.
- 3 Select the lower end of the curve on the right.
The curve continues with the cursor.

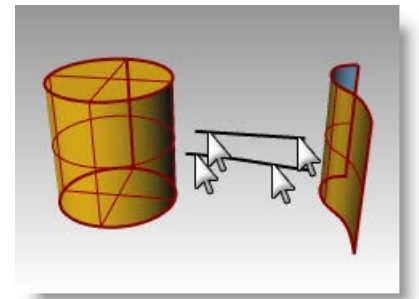
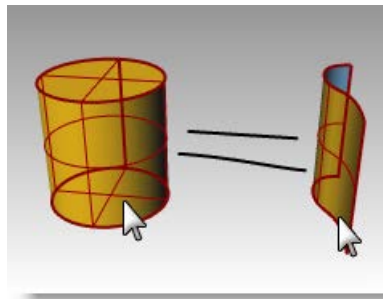


- 4 Click to end the extension.
- 5 Press **Enter** to end the command.



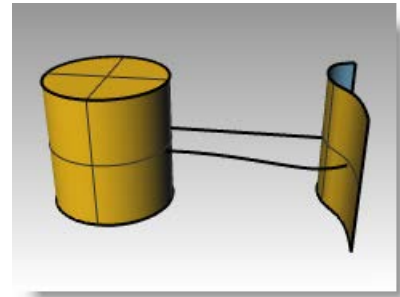
To extend to a surface:

- 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.



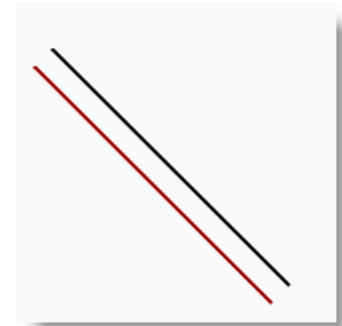
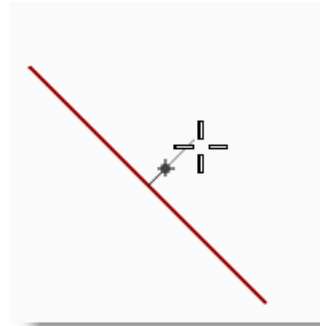
Offset

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 48—Offset

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

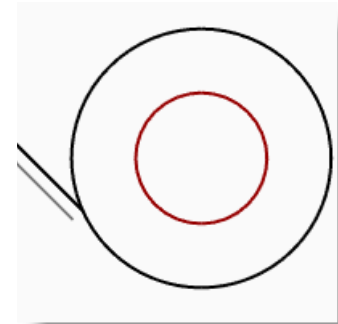
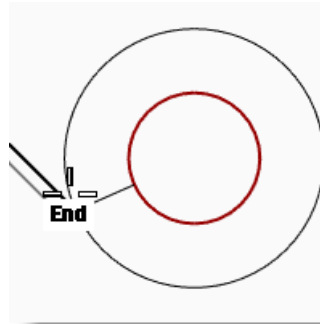
A parallel line is created.



To offset with the through point option:

- 1 Turn on the End osnap.
- 2 Select the circle.
- 3 From the **Curve** menu click **Offset**, then click **Offset Curve**.
- 4 For the **Side to offset**, click **Throughpoint** on the command line.
- 5 For the **Throughpoint**, snap to the lower right end of the line you offset.

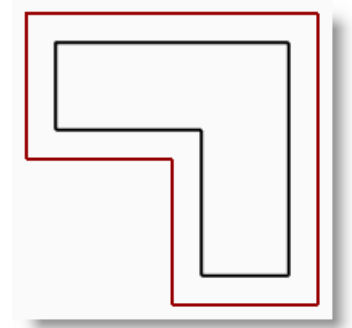
A concentric circle is created that goes through the endpoint of the line.



To offset a polyline with sharp corners:

- 1 Select the polyline.
- 2 From the **Curve** menu click **Offset**, then 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.

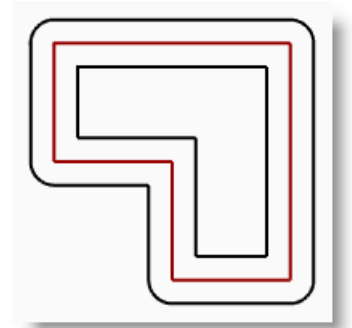


To offset a polyline with round corners:

- 1 Select the polyline.
- 2 From the **Curve** menu click **Offset**, then 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.



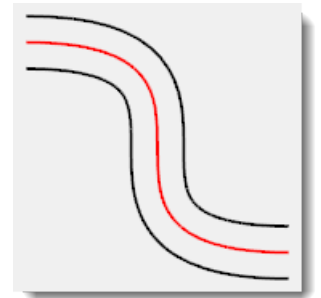
To offset a curve on bothsides:

- 1 Select the free-form curve.
- 2 From the **Curve** menu click **Offset**, then 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.

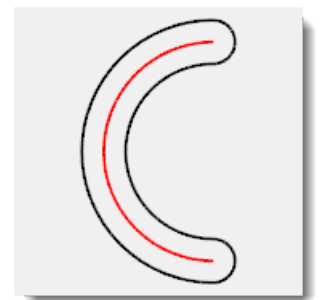
Concentric arcs are created on both sides of the selected arc.




To offset a curve on bothsides with caps:

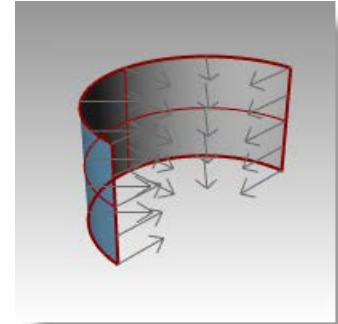
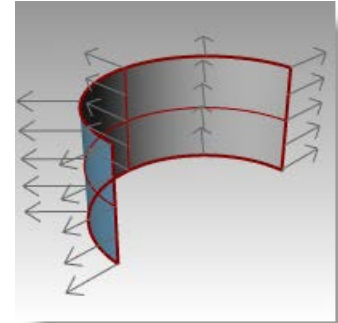
- 1 Select the arc.
- 2 From the **Curve** menu click **Offset**, then click **Offset Curve**.
- 3 Click **Cap** on the command line, and then click **Round**.
- 4 Click **Bothsides** on the command line.
- 5 For **Side to offset**, pick on either side of the curve.

Concentric arcs are created on both sides of the selected arc with a rounded end piece connecting the offset curves.

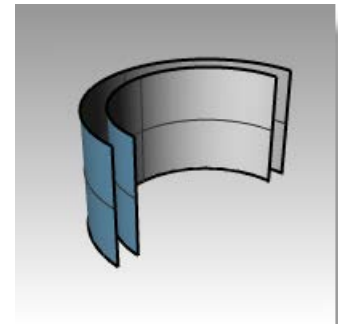


To offset a surface:

- 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 to change the offset direction.

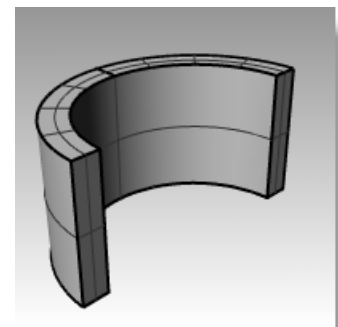


- 4 Press **Enter**.
The surface is offset in the direction of the arrows.



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:

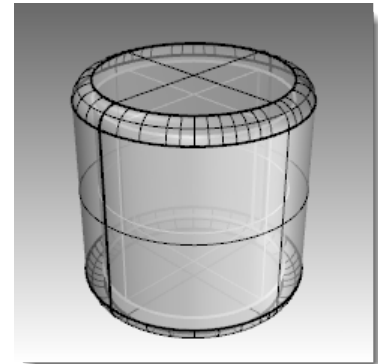
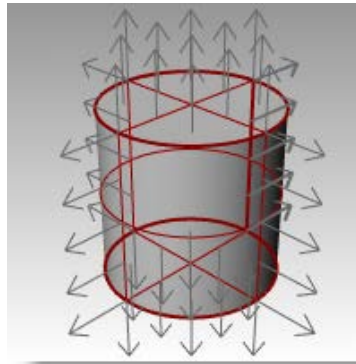
Offsetting polysurfaces may not give the desired results. In this example we will demonstrate some of the problems.

- 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 Pick **Distance** and type **1**.
- 4 Pick **Corners=Round**. Press **Enter**.

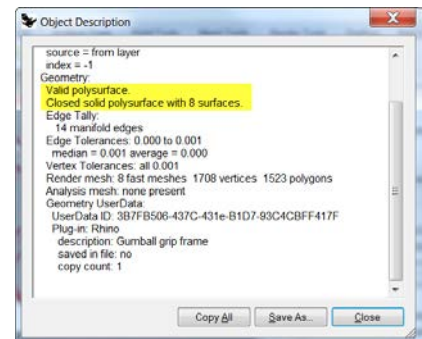
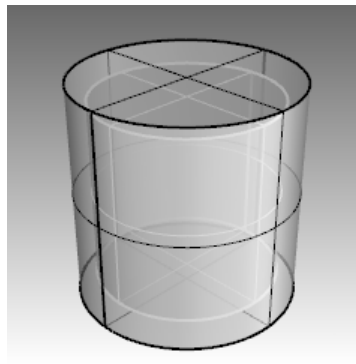
Each surface of the polysurface is offset as a separate surface, then extended or filleted, and joined together as a solid.



- 5 **Undo**. Repeat and pick **Corners=Sharp**.

What you have in both cases is a solid inside of a solid.

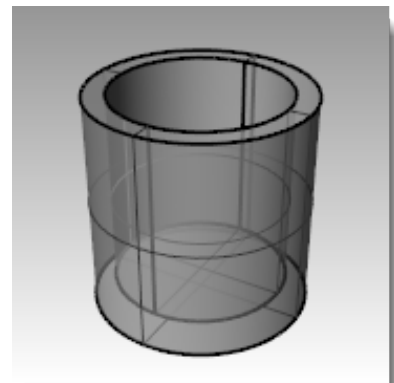
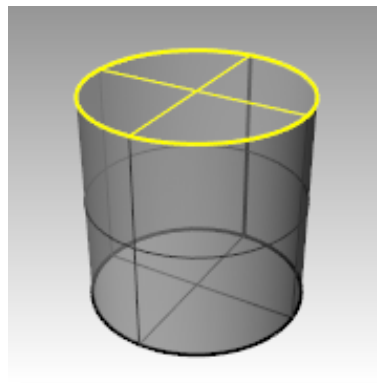
Hint: Use the **What** command to confirm that offset polysurface is a closed solid.



To Shell a polysurface:

- 1 **Undo**.
- 2 Select the Cylinder.
- 3 Type **Shell** on the command line.
- 4 For **Select faces to remove** pick the top of the cylinder and press **Enter**.

The surface is removed and the remainder is offset inward, using the outer parts of the removed surface to join the inner and outer parts.




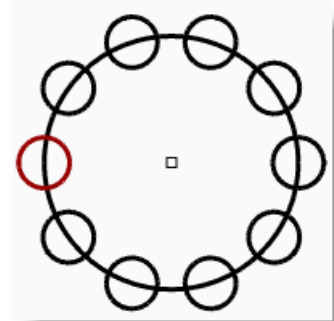
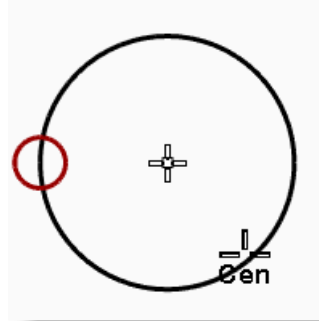
Array

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 49—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**.



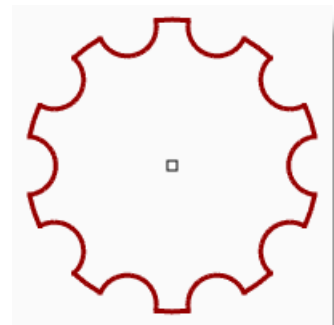
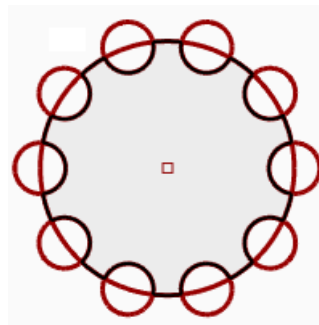
The small circle is arrayed around the larger one.

- 7 Save your model.

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

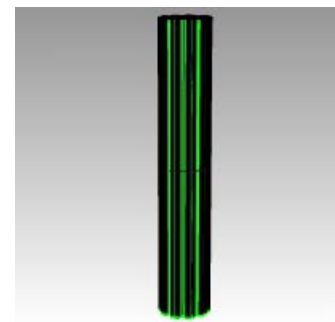
To make the shape for the column:

- 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**, and press **Enter**.



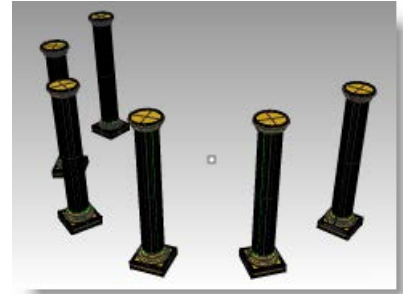
To make the column:

- 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**.




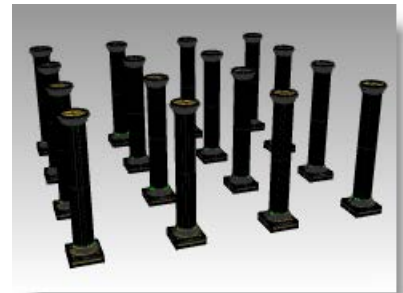
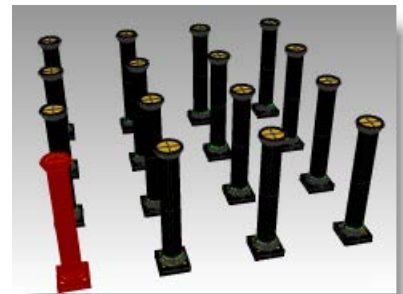
To make a partial polar array:

- 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.

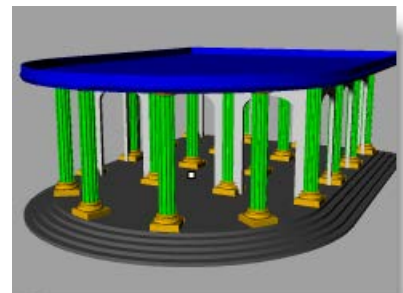


To create a rectangular array:

- 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.

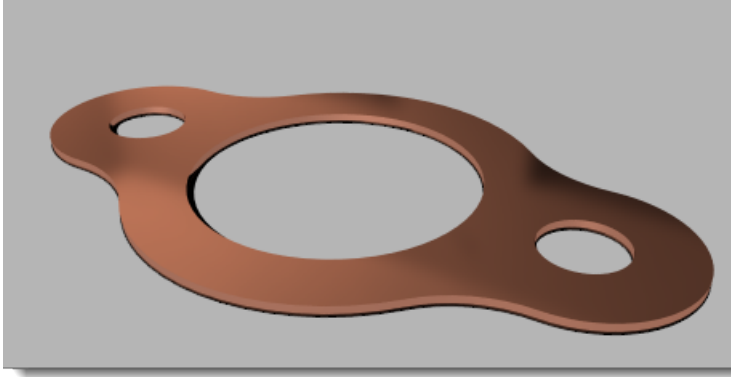


- 9 Press **Enter** to accept.
- 10 Turn on all the layers to see the result.

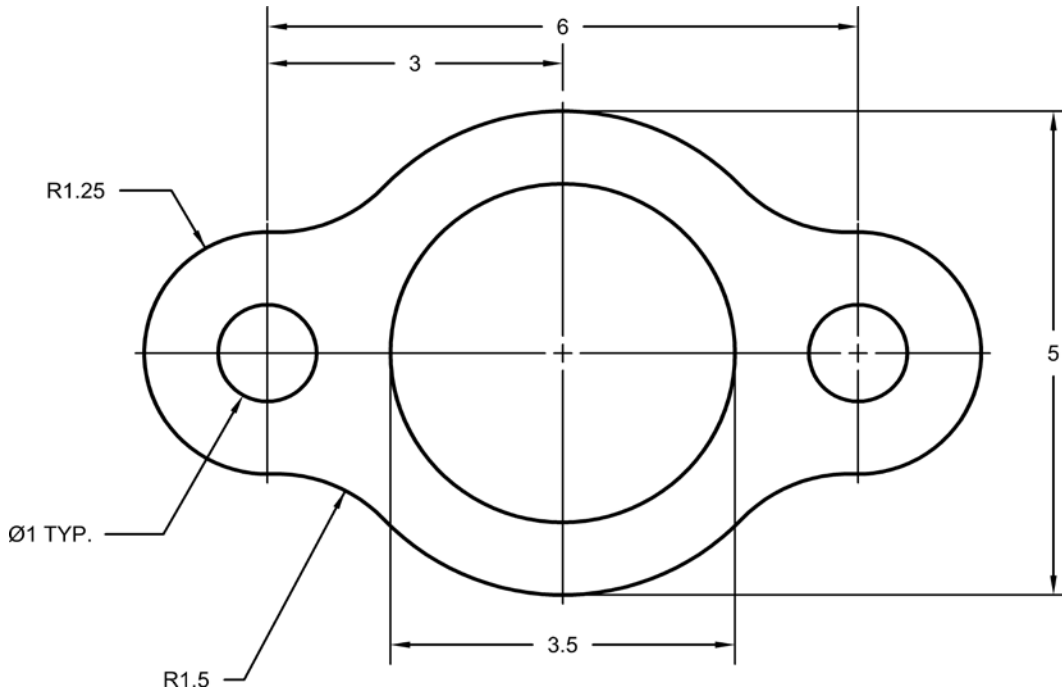


Exercise 50—Practice

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



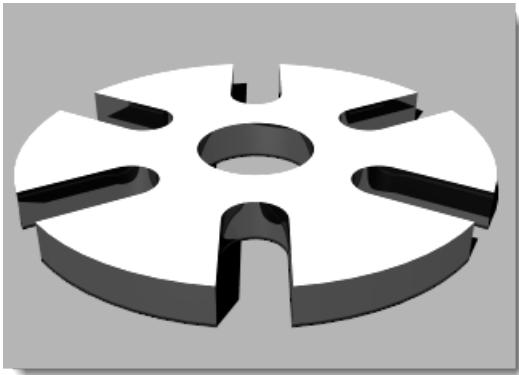
- 2 Use **Circle**, **Arc**, **Trim**, **Fillet**, and **Join** to create the part shown.



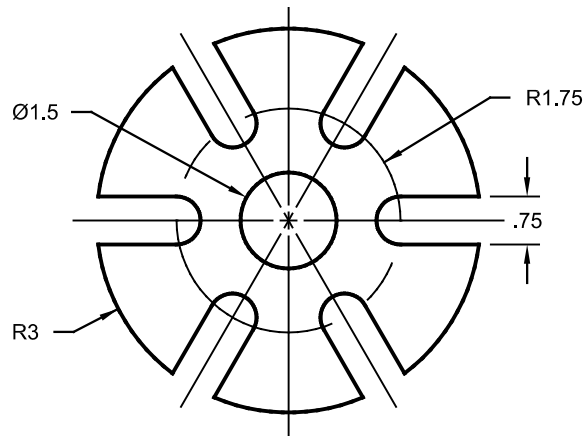
- 3 Use **Extrude Planar Curve > Straight** on the **Solid** menu to create the 3-D part. The extrusion thickness is .125.

Exercise 51—Practice

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



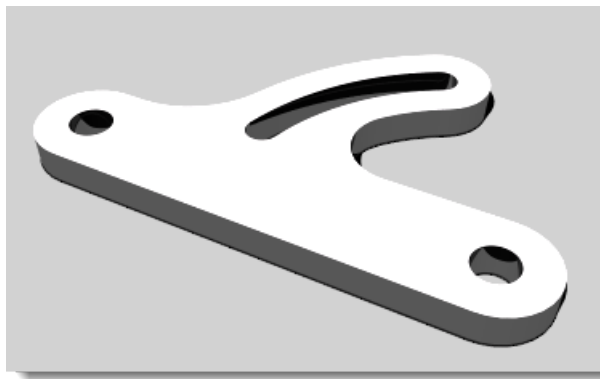
- 2 Use **Circle**, **Arc**, **Line**, **Trim**, **Join**, and **PolarArray** to draw the part shown.



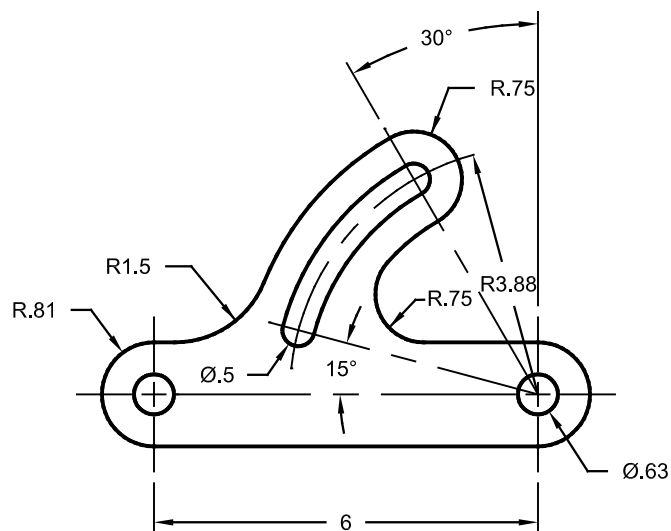
- 3 Use **Extrude Planar Curve > Straight** on the **Solid** menu to create the 3-D part. The extrusion thickness is **0.5**.

Exercise 52—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**.

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.

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 53—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:

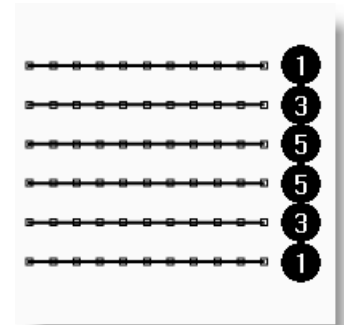
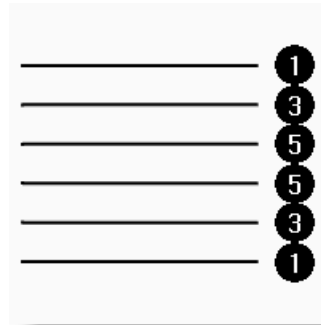
1 Open the model Control Point.3dm.

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

2 Turn Ortho and Grid 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 (F10).

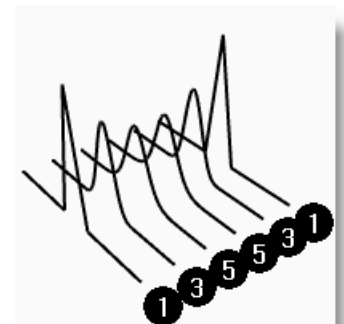
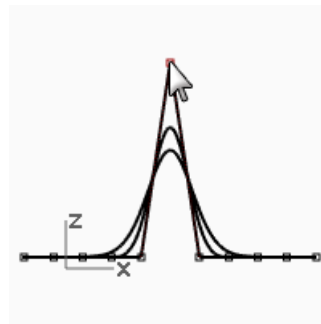


5 In the Front viewport select the middle row of points.

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

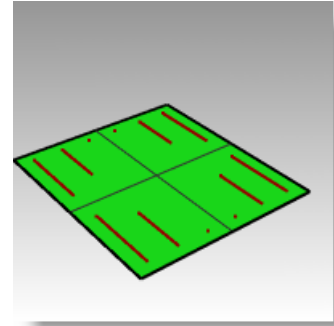
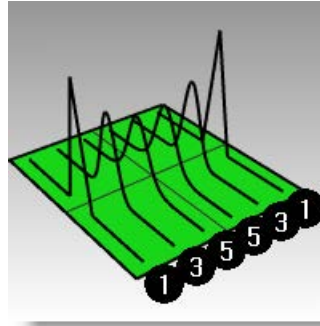


Individual points have more influence on a small area of the curve with degree 3 curves, while points have greater influence over a wider span of the curve with degree 5 curves.

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**.

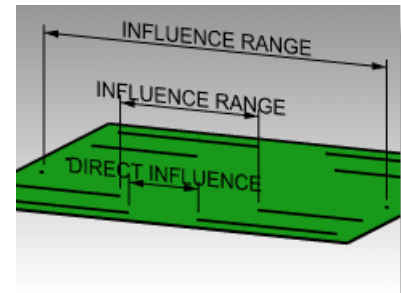
Lines are shown on the surface showing the intersections.



- 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.

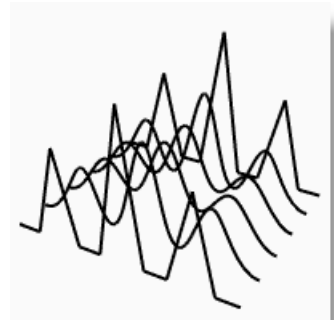
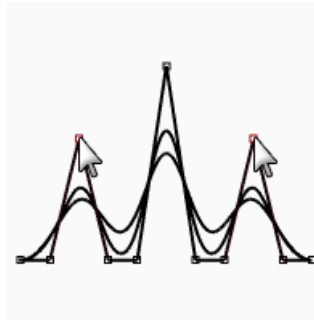
The intersections will disappear, and the Plane layer is off.

To continue editing control points:

- 1 In the Front viewport select the two rows of points that are three 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 sharply 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:

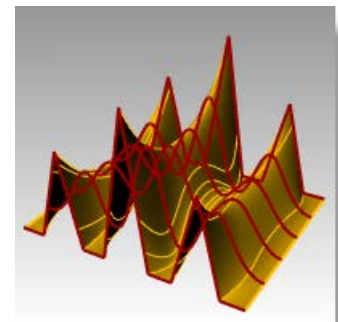
- 1 Select the curves.
- 2 From the **Surface** menu click **Loft**.
- 3 In the **Loft options** dialog box, click **OK**.

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.

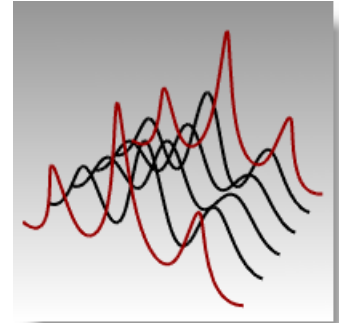
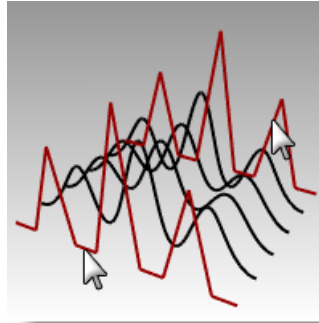
- 6 Undo the loft.



To change the polylines into curves without kinks:

- 1 Select both polylines.
- 2 From the **Edit** menu click **Rebuild**. 
- 3 In the **Rebuild** 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 smoothens and changes shape.



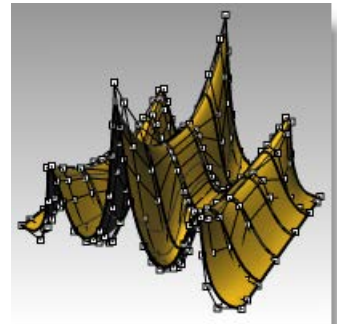
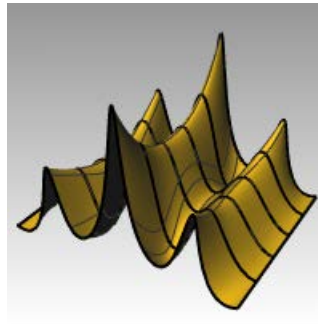
To loft a surface over the curves:

- 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.

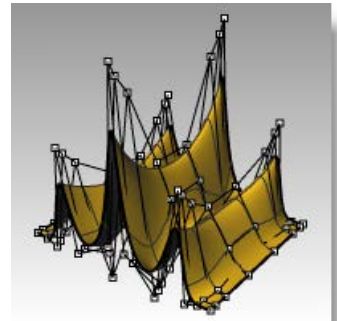
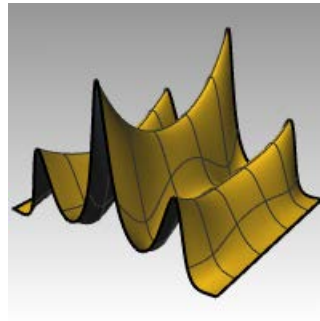
Note that there are more control points than what you started with. Rhino adds control points to make the surface conform the original curves.



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.



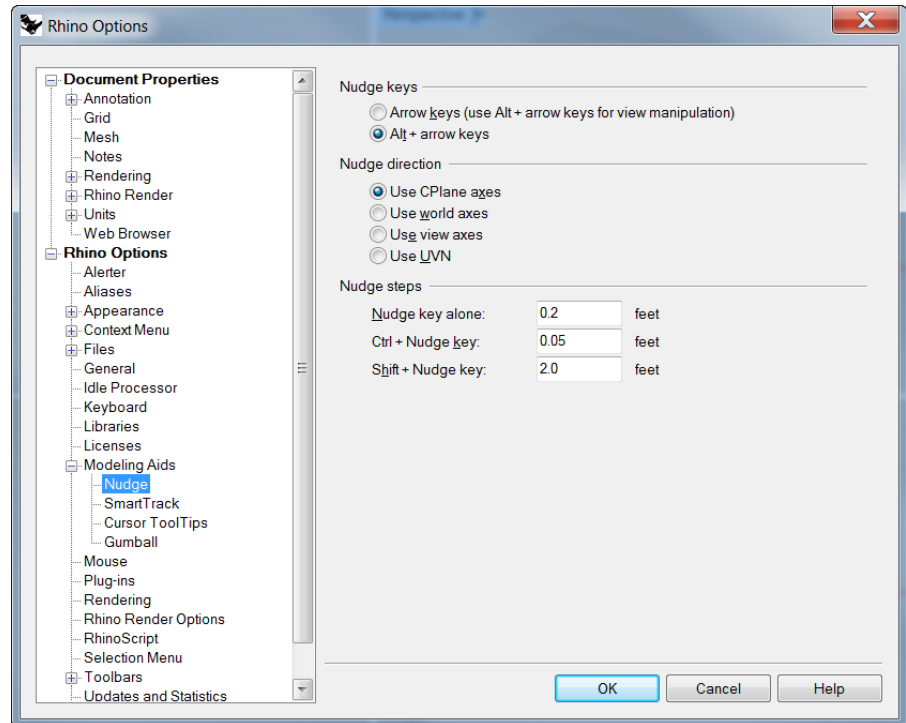
Nudge Controls

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:

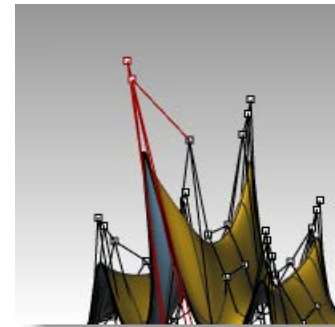
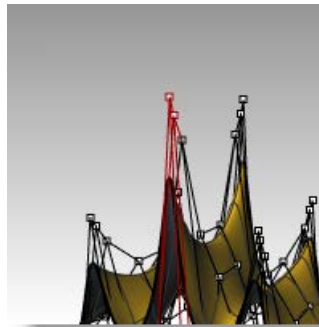
- 1 From the **Tools** menu click **Options**.
- 2 In the **Options dialog box**, on the **Modeling Aids** page, note the **Nudge** settings.

Any of these values can be changed.




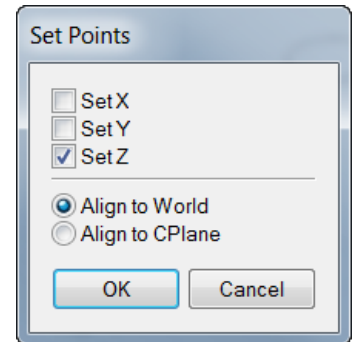
To use Nudge keys to move control points:

- 1 Select one or two control points in the Front viewport.
- 2 Hold the **Alt** key down and press an **Arrow** key.
Notice that it moves (*nudges*) the point a small amount.
- 3 Hold the **Alt** and the **Ctrl** key down and press another **Arrow** key.
The movement is much smaller.
- 4 Hold the **Alt** and the **Shift** key down and press another **Arrow** key.
The movement is magnified.
- 5 Hold the **Alt** and press the **Page Up** or **Page Down** key to nudge in the Cplane Z direction.



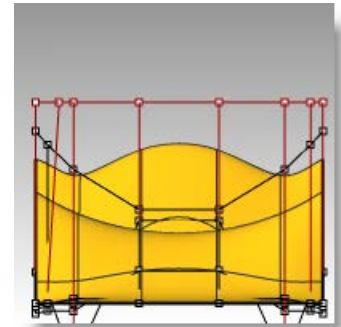
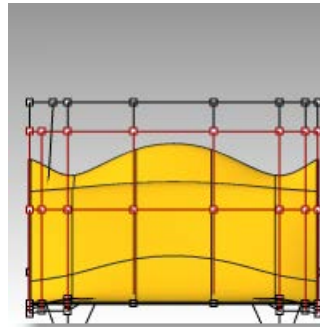
To use Set X, Y, Z coordinates to adjust points:

- 1 Select all the points in one row along the top of the surface.
- 2 From the **Transform** menu click **Set X, Y, Z coordinates**. 
- 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.

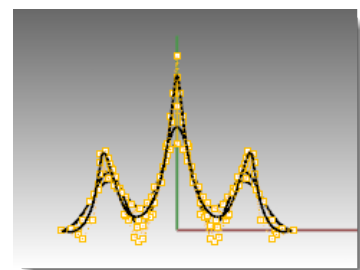
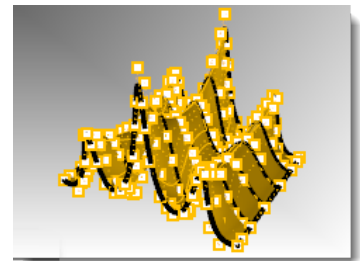
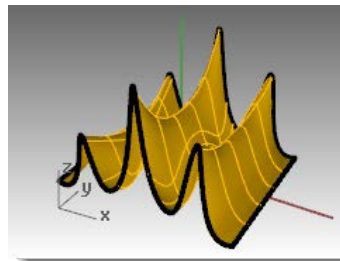
The control points are aligned in the World Z direction.
- 5 Repeat this on some of the other rows of points.



To use Gumball to move Control Points:

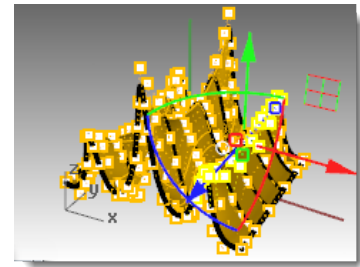
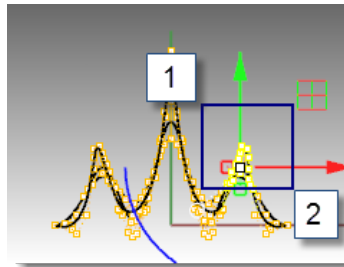
You can use the Gumball to move, rotate and scale Control Points, in a very similar way as you were able to transform objects. Let's look at moving the Control Points with the Gumball.

- 1 Select the surface.
- 2 From the **Edit** menu select **Control Points** and **Control Points On (F10)**.
- 3 Turn on **Gumball** and select the points in one row along the top of the surface.

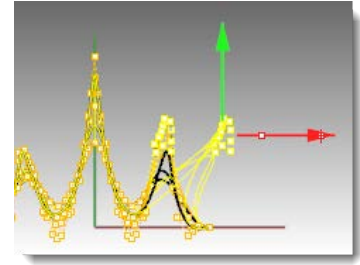
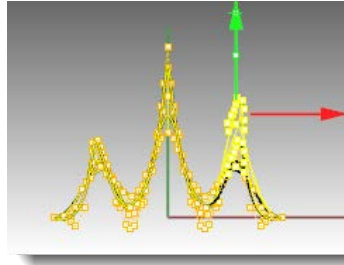


- 4 In the Front viewport, pick to the upper left of the screen.
- 5 Next, drag the cursor to the lower right and pick. A selection window will appear and select all the points within the window.

The points along the top row are selected.

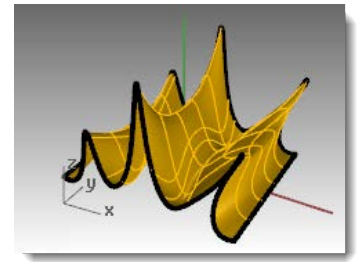
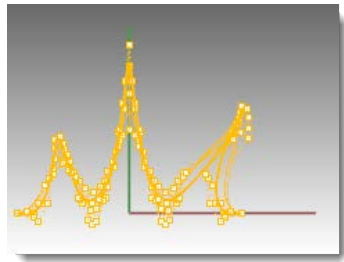


- 6 In the Front viewport, pick the **green Gumball arrow**, drag up and pick. The points move up.
- 7 In the Front viewport, pick the **red Gumball arrow**, drag to the right and pick. The points move to the right. Press **Esc** to unselect points.



- 8 In the Front viewport, pick the **green Gumball arrow**, drag down and pick.

The points move in the top rows move down.



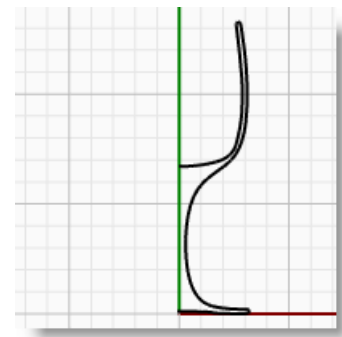
Note: Use the Gumball for Control Point editing whenever possible in the next few exercises.

Exercise 54—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 Select the curve you created.
- 2 From the **Surface** menu click **Revolve**.
- 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 **Save** your model.



PART THREE

3-D Modeling and Editing

7 Creating Deformable Shapes

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

The Rubber Ducky

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 55—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.

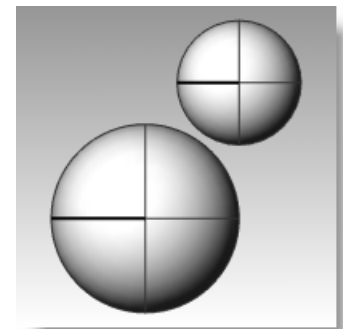


Create the body and head shapes


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.

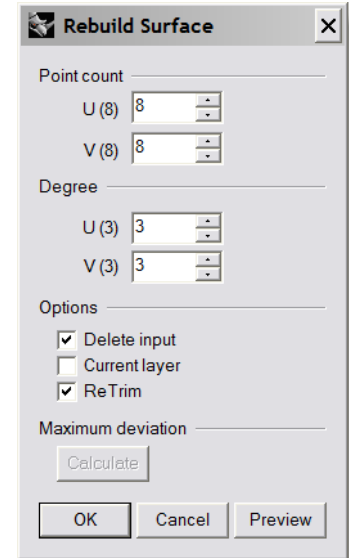
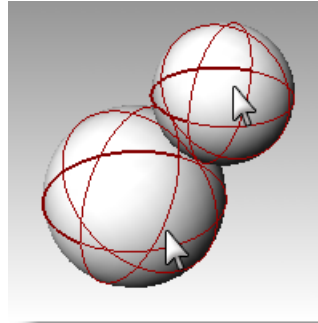
To create the basic shapes:

- 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.

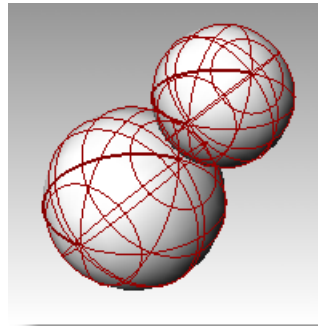


To make the spheres deformable:



- 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**.



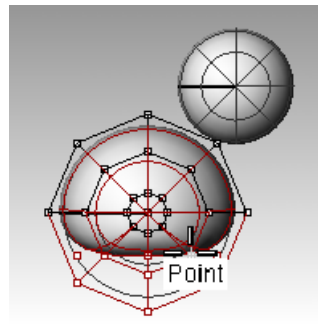
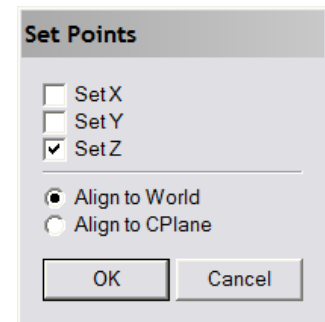
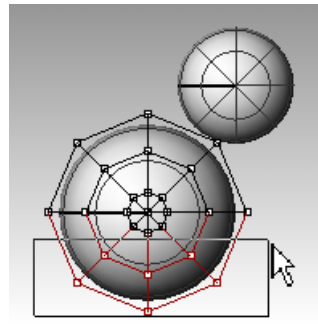
The spheres are now deformable. Having more control points allows more control over smaller parts of the surface. A degree-three surface will have a smoother shape when deformed.




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 X, Y, Z coordinates**. 
- 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 selected points.

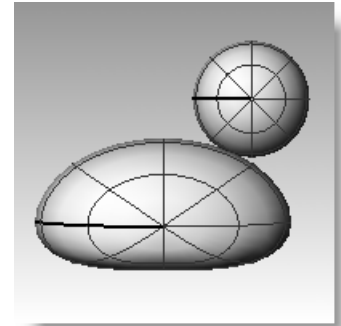
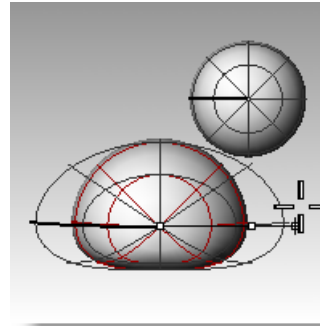
This aligns all of the selected control points to the same world z-value (vertical in Front viewport), flattening the surface.



To scale the sphere shape:

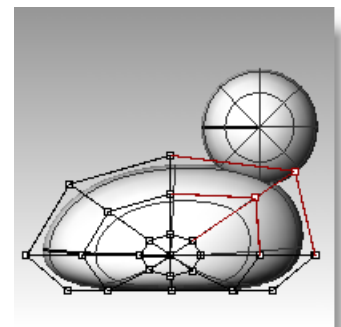
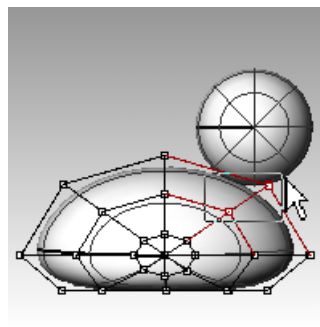
- 1 Turn off control points and select the body shape.
- 2 From the **Transform** menu click **Scale**, and then click **Scale1D**. 
- 3 For the **Origin point**, pick a point near the center of the body sphere.
- 4 For the **First reference point**, with Ortho on, pick a point to the right in the Front 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.



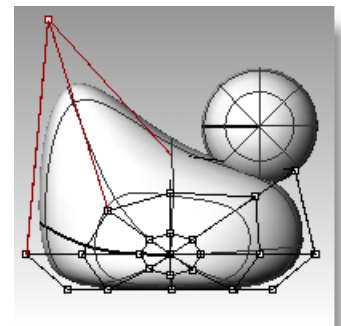
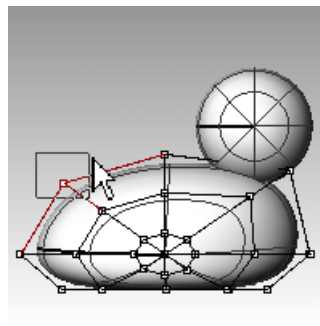
To reshape the chest and tail

- 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.



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.

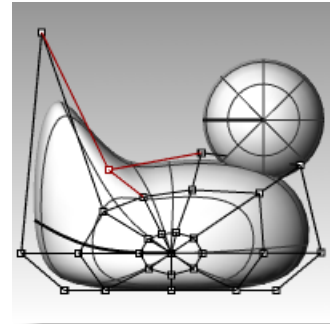
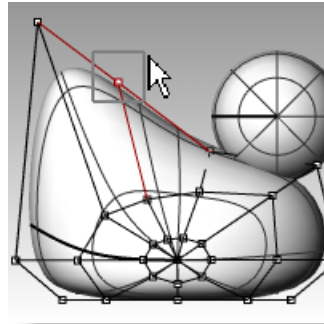
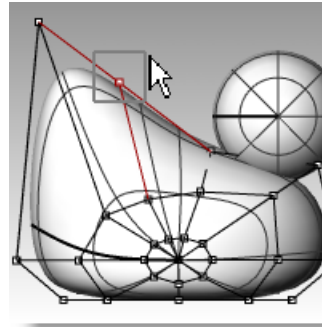
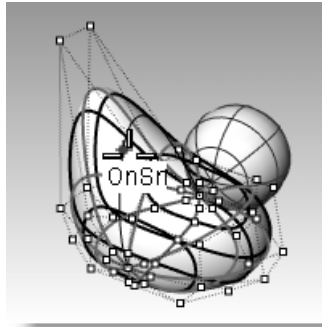
- 3 Change the direction if necessary.

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

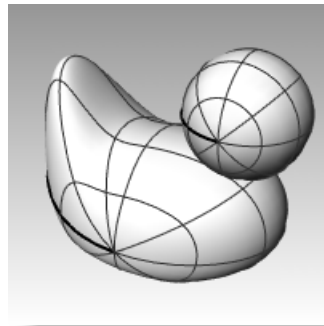
- 5 Press **Enter** to end the command.

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.

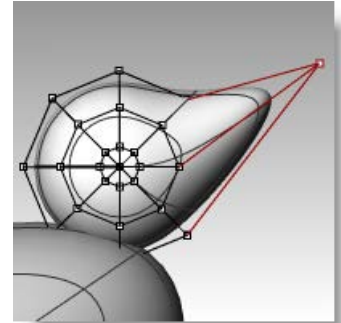
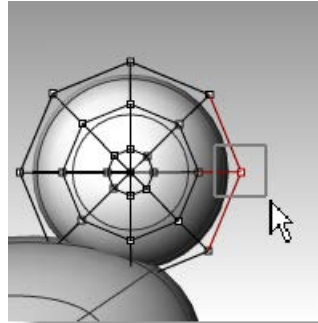


- 8 **Save** your model.



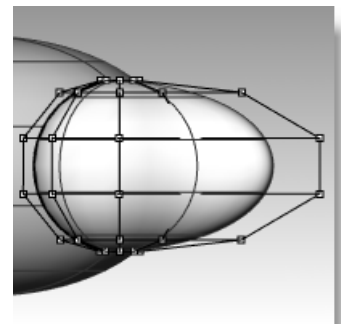
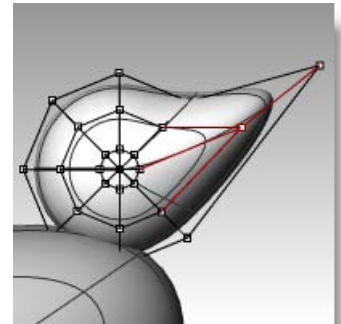
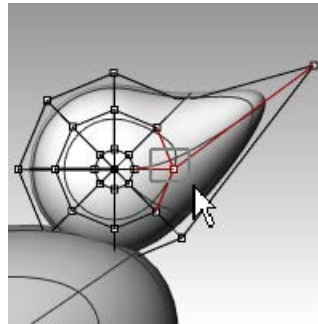
To create the head:

- 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.

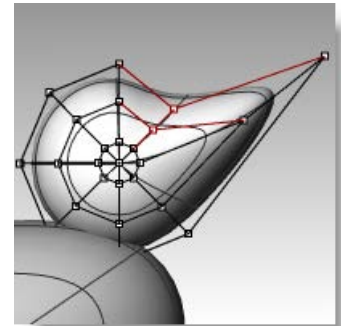
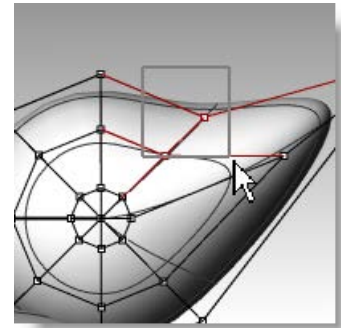
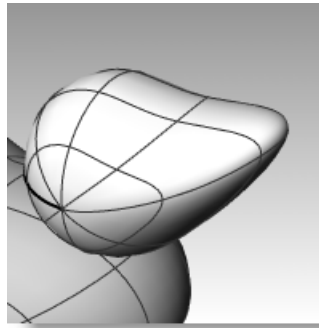


- 4 Window select the control points further back on the same isocurve and drag them forward to widen the bill.

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



- 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.



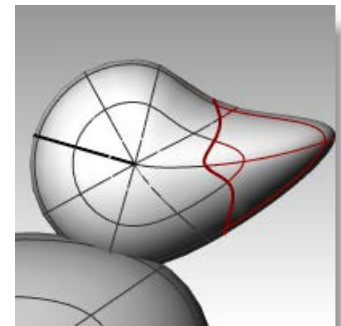
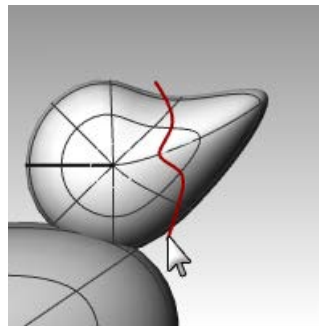
Separate the bill from the head

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.

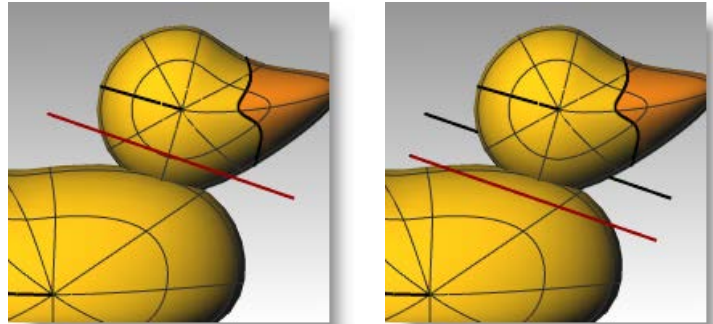


Create the duck's neck

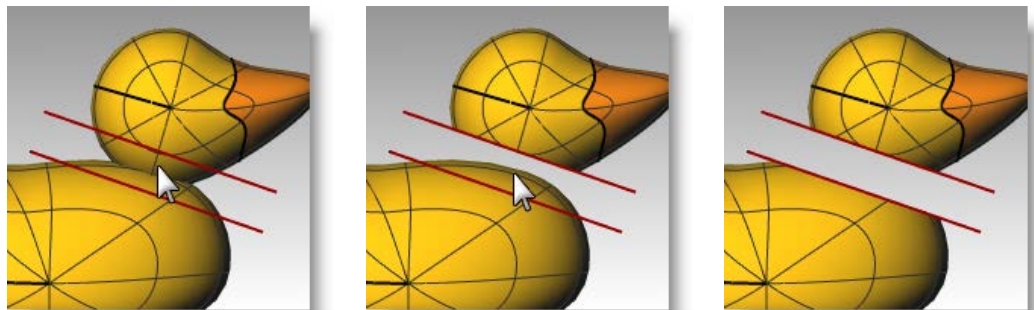
The ducky needs a neck. You will first make an edge on the surface of the head and a corresponding edge on the surface of the body so you can create a blended surface between the two edges.

To trim the head and the body:


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

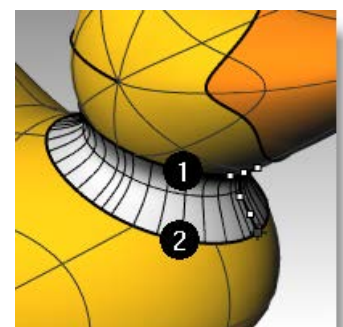
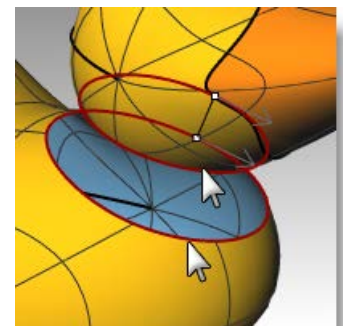
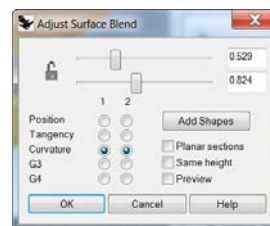


- 5 For the **Object to trim**, pick the bottom of the head and the top of the body.
The bottom of the head and the top of the body is trimmed.
- 6 **Save** your model.




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 **Adjust Surface Blend** 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 click the lock icon at the left of the sliders the surface adjustments are symmetrical.
- 10 **Save** your model.

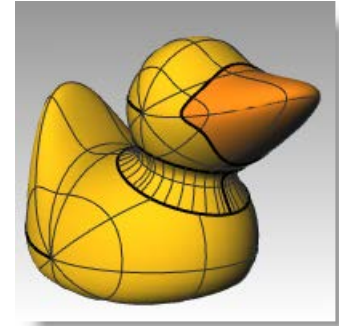


To join the parts:

- 1 Select the body the blend surface and the back of the head.

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

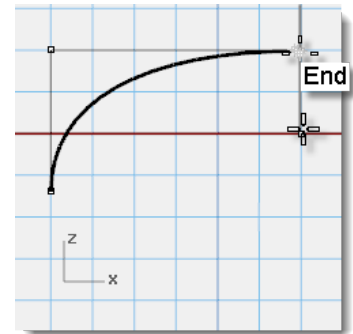
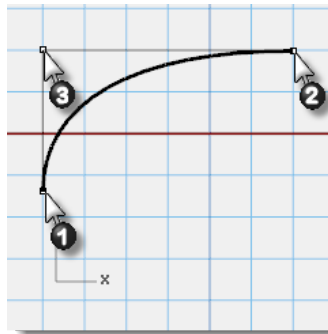
The three surfaces are joined into one. The bill is left separate for rendering purposes.



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 Grid Snap 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.

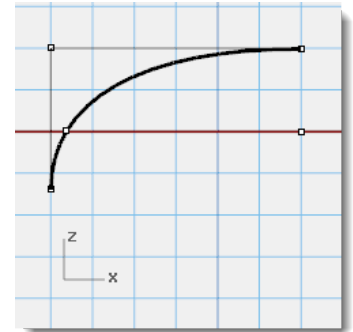
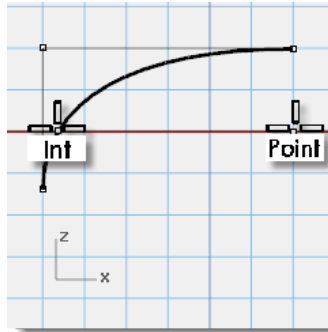


These points will be used for placement of the eye on the head.

- 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.

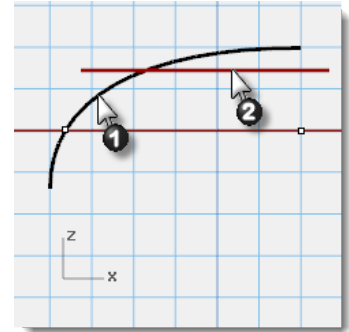
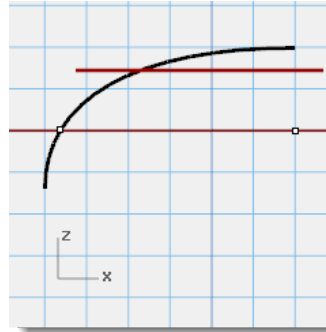
This point is for scaling and rotating the eye.

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**.

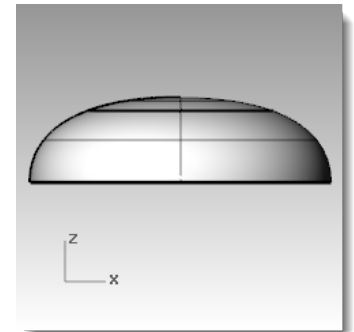
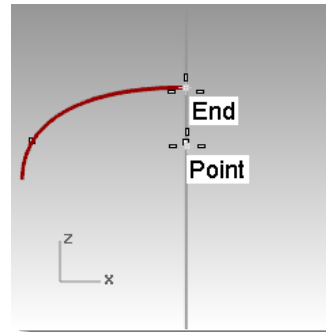
Splitting the curve allows you to assign a different color and material properties for the eye and the pupil.

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




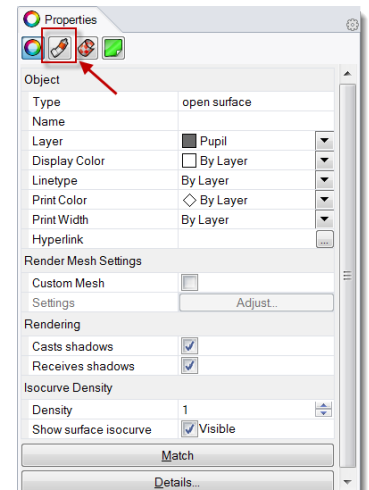
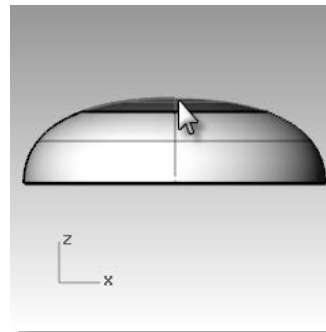
To make the surface:

- 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**.

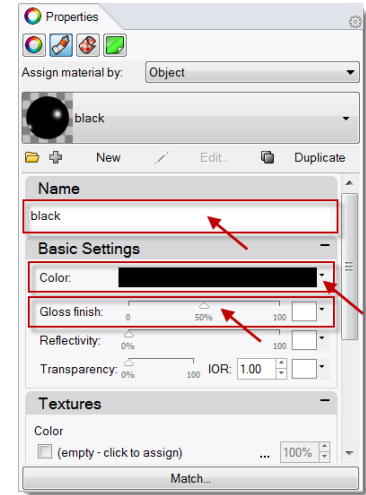
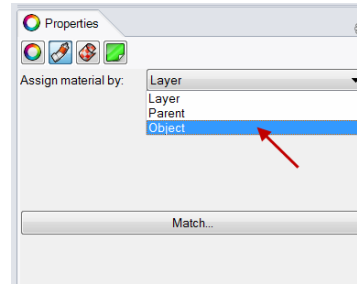


To assign display color and material color:

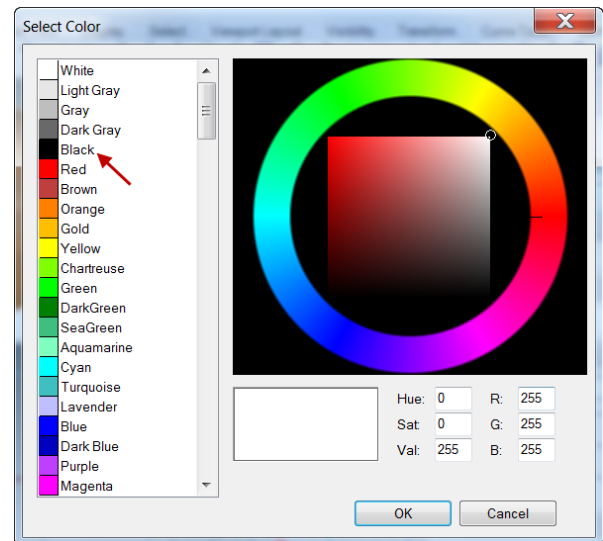
- 1 Select the top of the eye.
 - 2 From the Edit menu click Object Properties.
- 
- 3 For the **Display color**, choose a contrasting color, like red.
 - 4 With surface still selected, in the **Properties** pane, select the **Material** page button



- 5 In **Assign Materials by** select **Object**.
- 6 With **Default Material** selected, type the name of the new material to be used for the pupil of the eye, like **Black**.
- 7 In the **Basic Settings**, select the **Color swatch**. The **Select Color** dialog appears. Pick **Black** and **Ok**. Also set the **Gloss finish** to **50%**.
- 8 Repeat these steps create a white material for the eye.

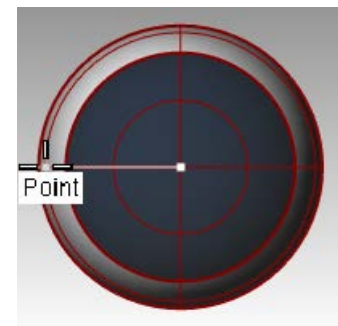
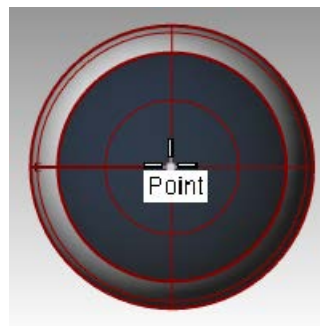


- 9 From the **Render** menu click **Render Preview** to see the material color.

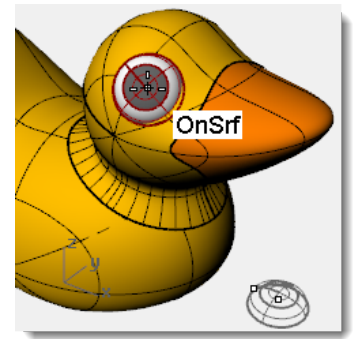
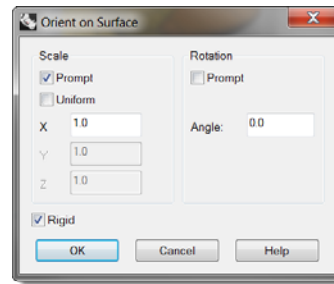


To place the eye on the head:

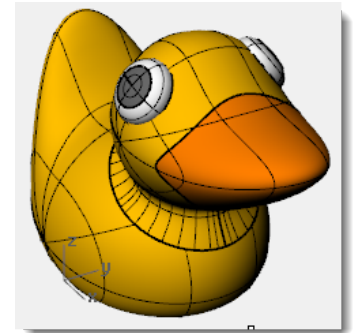
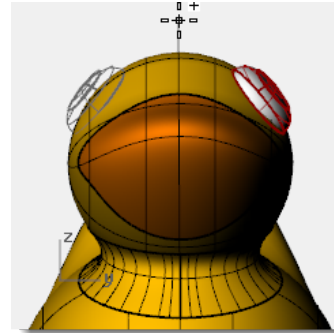
- 1 In the Top viewport select both parts of the eye.
- 2 From the **Edit** menu click **Groups** and then click **Group**.
The eye parts are grouped as one object.
- 3 Select the group.
- 4 From the **Transform** menu click **Orient**, then click **On Surface**.
- 5 For **Base point**, snap to the point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



- 8 In the **Orient on Surface** dialog, for **Scale** check **Prompt** and **Rigid**, click **OK**.
- 9 For the **Point of surface to orient to**, pick a point on the head.
- 10 For the **Scale factor**, drag your mouse and pick to scale the eye.




- 11 **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:

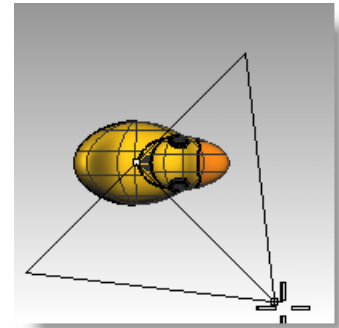
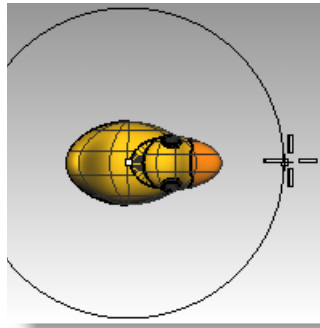
- 1 Select the beak.
- 2 From the **Edit** menu click **Object Properties**.
- 3 In the **Properties** panel, on the **Material** page, click **Basic** and click the **color swatch**.
- 4 In the **Select Color** dialog box, select a color for the beak, like orange.
- 5 Select the body.
- 6 From the **Edit**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

To place lights:

- 1 From the **Render** menu click **Create**

Spotlight. 

- 2 Select a point in the middle of the model.

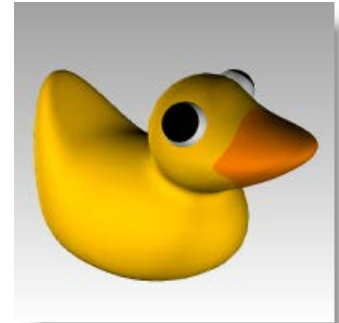
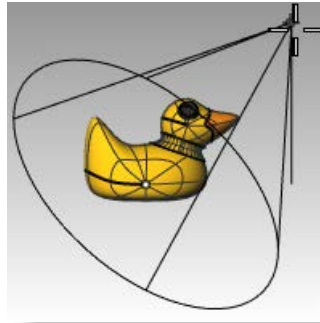


- 3 **Drag** the radius until it is approximately three times as large as the model.

- 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**.






















8 Modeling with Solids

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 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.

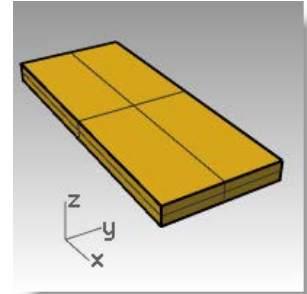
Button	Command	Description
	Box	Draw a rectangular box from two diagonal corners and a height.
	Box 3Point	Draw a box from two adjacent corners, a point on the edge across from them, and a height.
	Sphere	Draw a sphere from a center point and a radius.
	Sphere 2Point	Draw a sphere from the two ends of 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.
	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.

Model a bar with engraved 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.

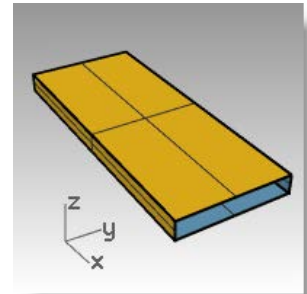
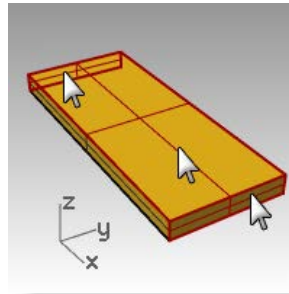
Exercise 56—Model a bar with text



- 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**.

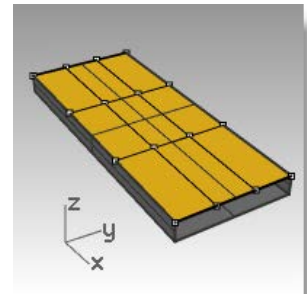
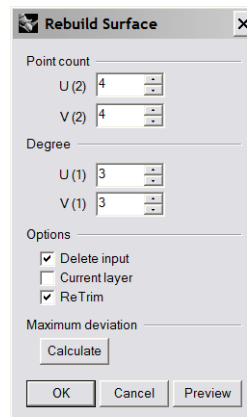


To edit a surface:

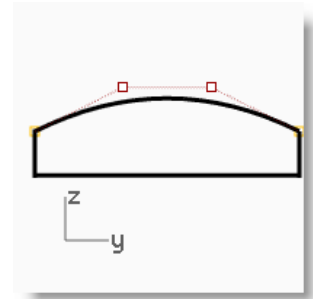
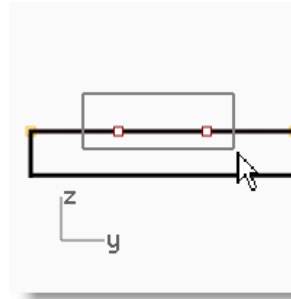
- 1 From the **Solid** menu click **Extract Surface**. 
- 2 For the **Surface to extract**, select the top and both end surfaces, press **Enter**.
- 3 Select the two ends and delete them.



- 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**. 

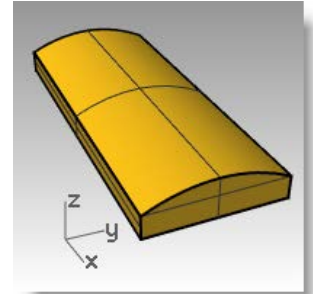
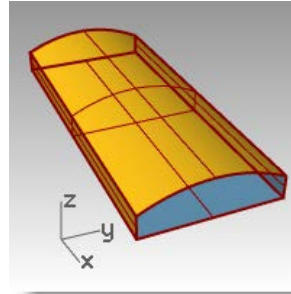


- 8 In the Right viewport, window select the middle points.
- 9 **Drag** the points up approximately one unit.
- 10 Press **F11** to turn Control Points Off.




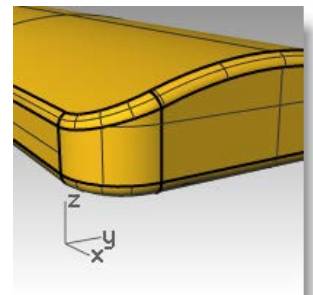
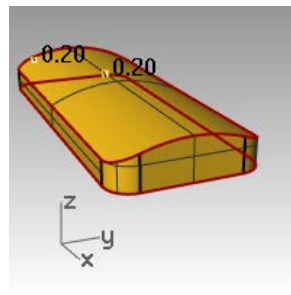
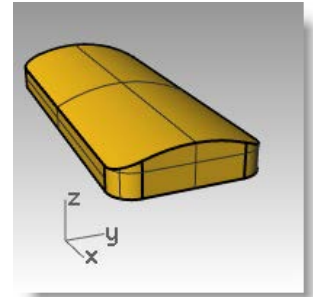
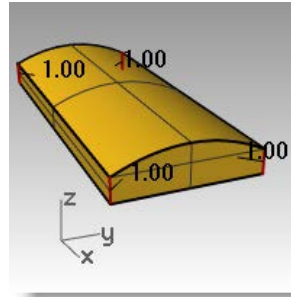
To make the bar solid:

- 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.




To fillet the edges:

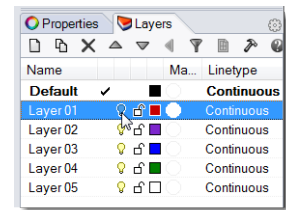
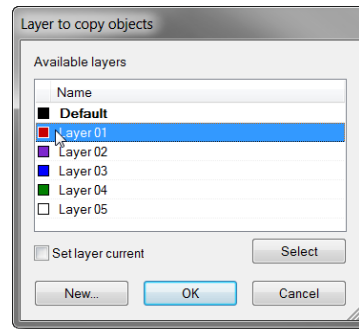
- 1 From the **Solid** menu click **Fillet Edge**, and then click **Fillet Edge**.

- 2 Set the **Next 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 **FilletEdge** command.
- 6 Set the **Next 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**.




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

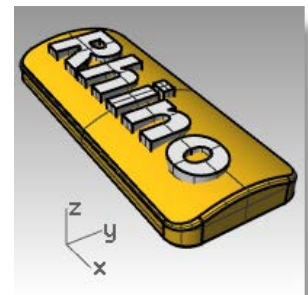
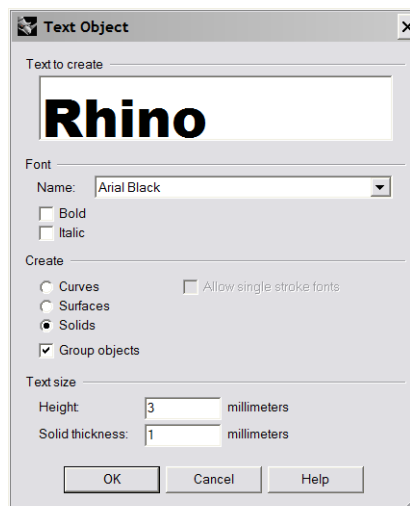
We need to make a copy of the finished bar for the next part of this exercise. On one of the copies we will engrave text, on the other we will emboss text.

- 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 In the **Layer** panel, turn Layer 01 off.



To make solid text:

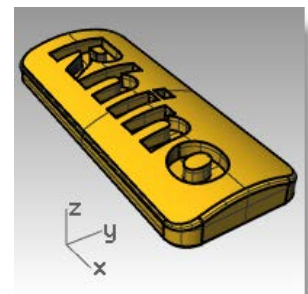
- 1 Change the layer to Layer 02.
- 2 From the **Solid** menu click **Text**. 
- 3 In the **Text Object** dialog box, select a bold **Font**, like **Arial Black**. Under **Create** click **Solids**. Under **Text size**, set the **Height** to **3.00**, the **Solid thickness** to **1.00**, check **Group 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.



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. However, it does not follow the curvature of the bar. Let's look at how to make this text better follow a curved surface.

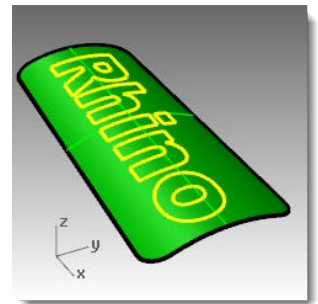
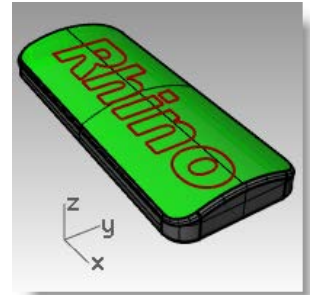
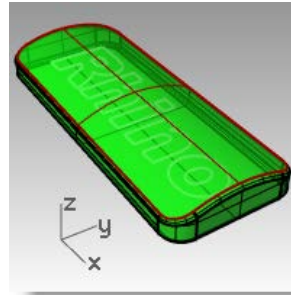


Offset solid text

Occasionally you will want to create text that exactly follows the curvature of a base surface. One method is to split the top surface of the bar with the text curves and offset the surface into solid text objects. The solid text can be used to engrave (difference) or emboss (union) into the original surface or polysurface:

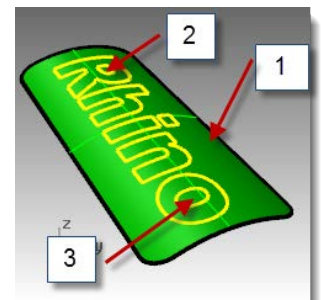
To make a label:

- 1 Turn Layer 01 on and turn the Default layer off.
- 2 From the **Solid** menu click **Extract Surface**.
- 3 Click **Copy=Yes** in command line. Select the top surface, press **Enter**.
- 4 Select the lower part of the bar and Hide 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**, and check **Group Objects**. 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 text curves.
- 2 From the **Edit** menu click **Trim**.
- 3 Select the surface in 3 places: near outer edge, in center of O, and in center of R.
Remember to trim out the center of letters like R and O.
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 Delete the original text curves.



Hint: **Selcrv** will select just the curves.

- 5 Select the text surfaces.
- 6 From the **Edit** menu click **Group**, and then click **Group**.
The text surfaces are now grouped for easier selection.

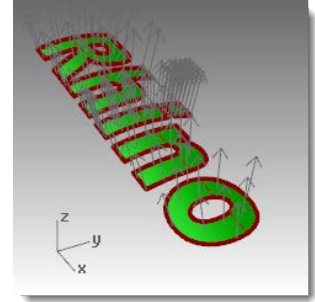


To create solid text:

- 1 From the **Edit** menu click **Select Objects**, and then click **Previous Selection**.

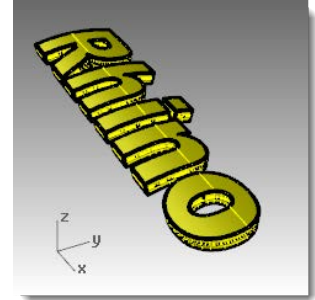
This selects the text surfaces again.

- 2 From the **Surface** menu click **Offset Surface**.
- 3 For the **Offset distance**, click **Bothsides=Yes**, **Solid=Yes**, and **DeleteInput=Yes** on the command line.
- 4 For the **Distance**, type **.1** and press **Enter**.

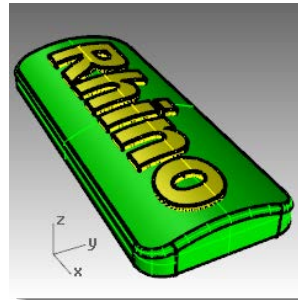


- 5 In the **Properties** panel, on the **Material** page, for **Assign material by choose object**, click the **color swatch** and select another color for the text.
- 6 Right click on the Perspective viewport title.
- 7 Select **Rendered** from the popup menu.

The letters render in a different color.



- 8 Show the bar.
- 9 Use the **Gumball** to **Copy** the bar and solid text.



To make embossed text:

- 1 **Unlock** the bottom part of the bar and Join it with top part.
- 2 Select the bar and the text.


- 3 From the **Solid** menu click **Union**.



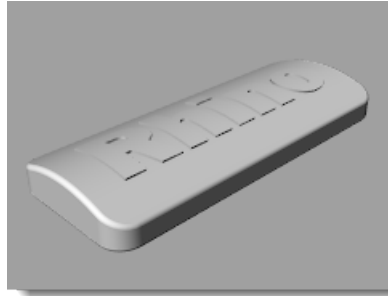
The text and the bar are joined into one closed polysurface with the text embossed on the surface.



To make engraved text:

- 1 Select the bottom polysurface from the copy you made.
- 2 From the **Solid** menu click **Difference**. 
- 3 For the **Select surfaces or polysurfaces to subtract** with **DeleteInput=Yes**, select the text and press **Enter**.

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



Embossed Polysurface




















Engraved Polysurface

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
	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.

Simple surfaces - Phone

Exercise 57— Create simple surfaces

In this exercise, you will model some simple surfaces.

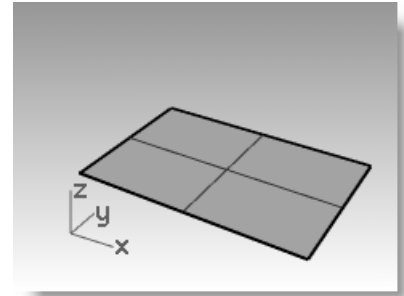
- 1 Start a new model, use the **Small Objects - Millimeters.3dm** template. **Save as Surfaces.**
- 2 Turn on **Grid Snap** and Planar.

Planes

- 1 From the **Surface** menu click **Plane**, and then click **Corner to Corner**.

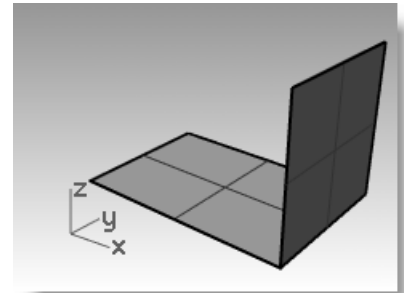


- 2 For the **First corner of plane**, pick a point.
- 3 For the **Other corner**, pick another point to make a rectangular plane.



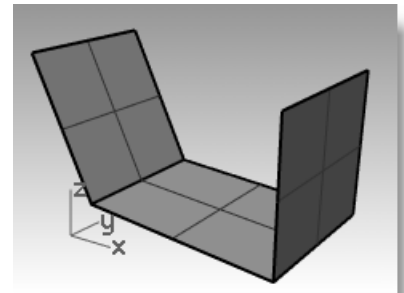
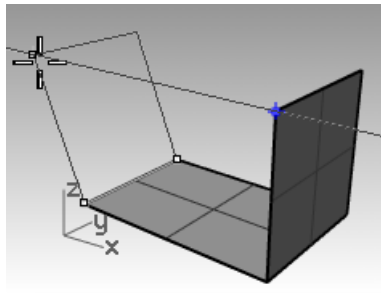
To create a vertical plane:

- 1 From the **Surface** menu click **Plane**, and then click **Vertical**.
- 2 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.



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.

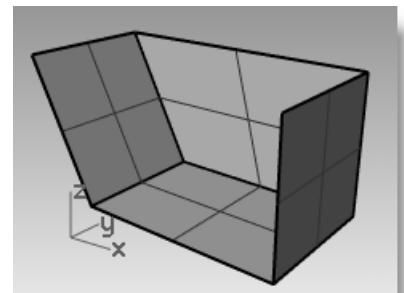
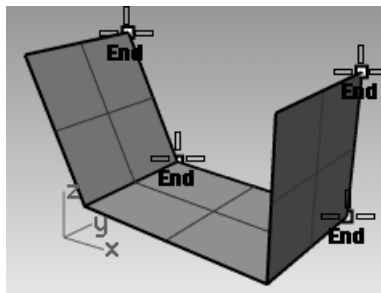
To create a plane from corner points:

- 1 From the **Surface** menu click **Corner Points**.



When you pick the points in the next four steps, pick them in a clockwise direction.

- 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.

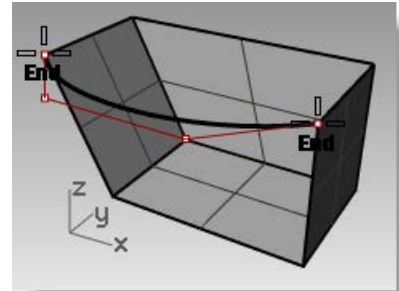
A surface will be created with corners at the points you selected.

Surface from planar curves

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.

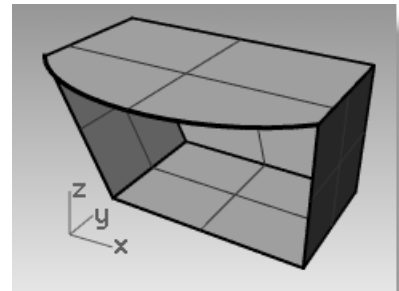
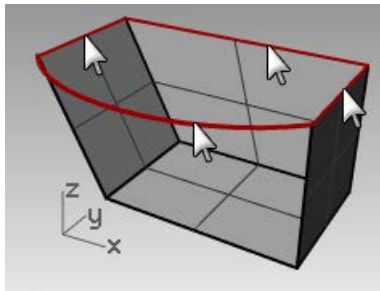
Planar mode keeps this curve on the same plane as the surface corners.



- 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.



Surface from edge curves

To create a surface from edge curves:

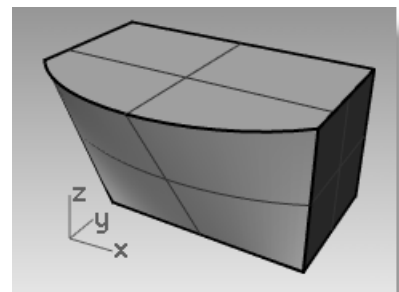
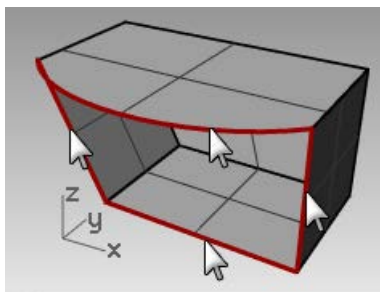
- 1 From the **Surface** menu click **Edge Curves**. 

- 2 Select the four surface edges.

A surface is created.

- 3 Select all the surfaces and from the **Edit** menu click **Join**.

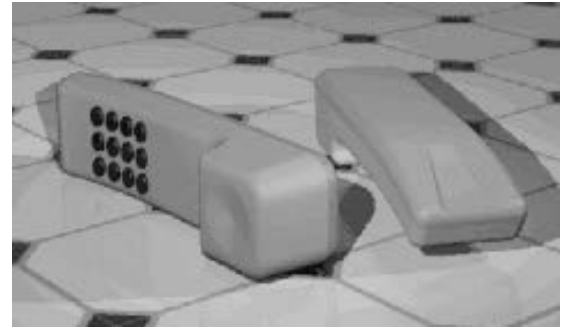
Result should be valid, closed polysurface.




Hint: Use the **What** command to confirm that offset polysurface is valid, closed solid.

Extruding curves

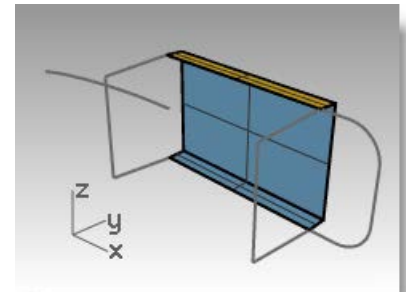
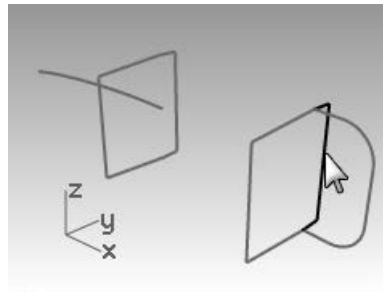
In this exercise, you will be creating a cordless phone using extrusions. To aid in organizing the model, surface and curve layers have been created. Make sure you change layers as you are making the extrusions.




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**.

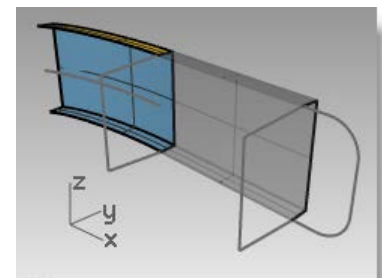
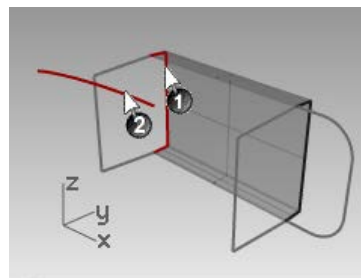
If the object being extruded is a planar curve, the curve is extruded perpendicular to the plane of the curve.



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.

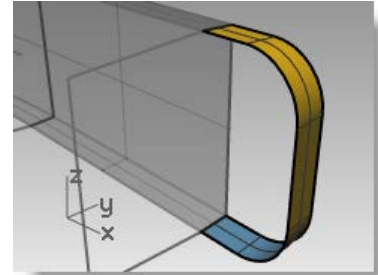
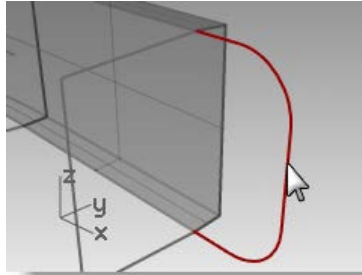
The curve is extruded along the path of the secondary curve.



If you don't get the answer you expected, undo and try picking near the other end of the path curve.

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


- 1 Select the 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**.



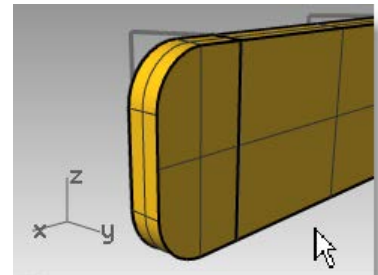
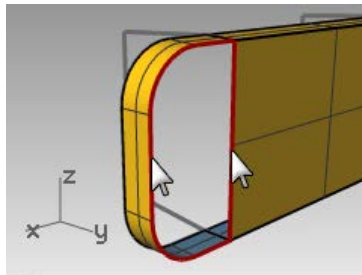
- 5 For the **Extrusion distance**, type **.375** and press **Enter**.

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

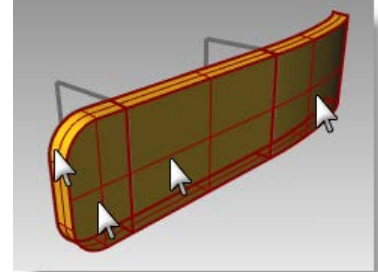
To create a surface from planar curves:

- 1 From the **Surface** menu click **Planar Curves**. 
- 2 Select the edge curves that bound the opening of the tapered extrusion at the top.
- 3 Press **Enter**.

A surface is created at the end.



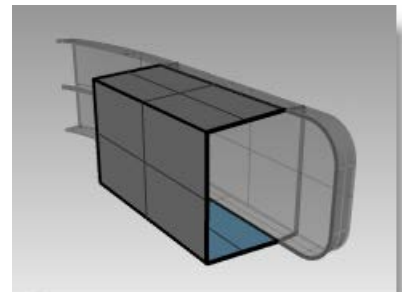
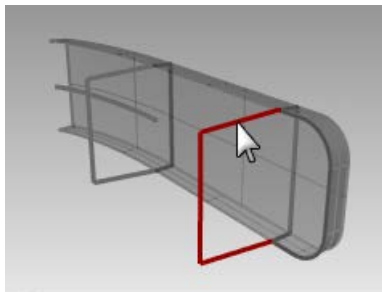
- 4 Select the four surfaces.
- 5 From the **Edit** menu click **Join**.



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.

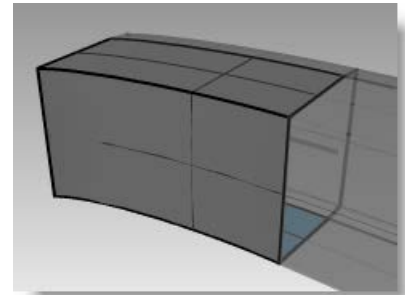
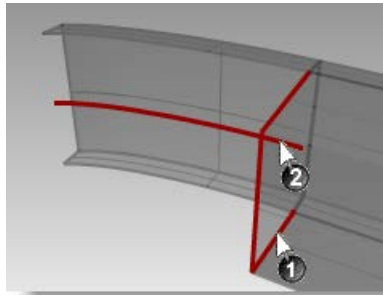
- 1 Make the **Bottom Surface** layer current.
- 2 Select the curve as shown.
- 3 From the **Surface** menu, click **Extrude Curve**, and then click **Straight**.
- 4 For the **Extrusion distance**, type **-3.5** and press **Enter**.



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

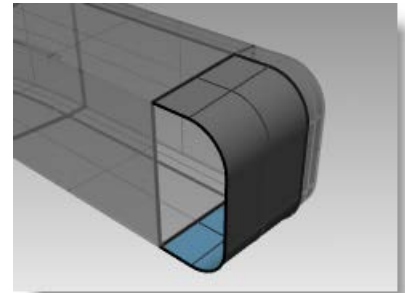
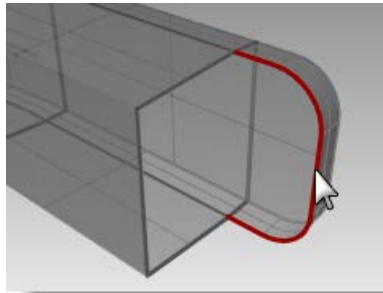
- 5 Select the curve **(1)** on the left of the first extruded surface.
- 6 From the **Surface** menu, click **Extrude Curve**, and then click **Along Curve**.
- 7 Select the path curve **(2)** near its right end.

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



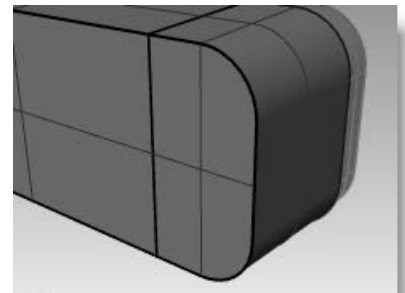
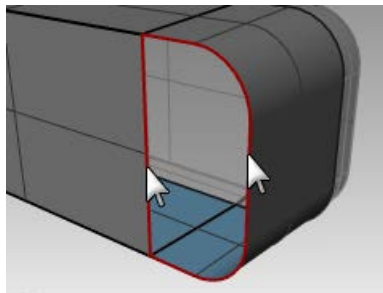
- 8 Select the curve on the right.
- 9 From the **Surface** menu, click **Extrude Curve**, and then click **Tapered**.
- 10 For the **Extrusion distance**, type **-1.375** and press **Enter**.

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



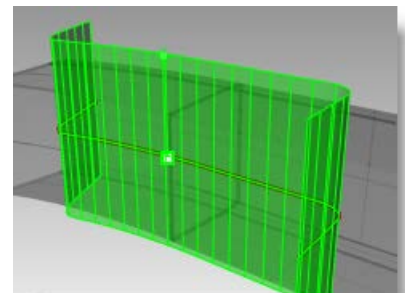
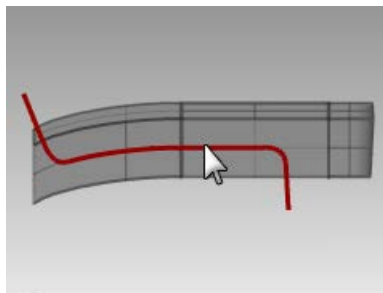
- 11 From the **Surface** menu, click **Planar Curves**.
- 12 Select the edge curves that bound the openings of the tapered extrusion at the top.
- 13 Press **Enter**.
- 14 Select the four surfaces.
- 15 From the **Edit** menu, click **Join**.

A surface is created at the end.



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

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



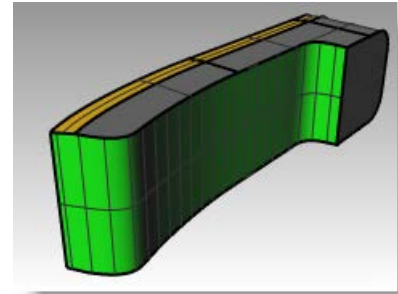
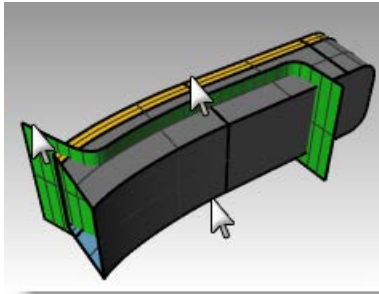
- 5 For the **Extrusion distance**, drag the extrusion and pick.

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

Exercise 58—Basic techniques joining surfaces together

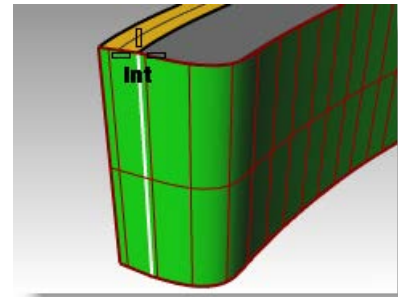
To trim the surfaces:

- 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.



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.

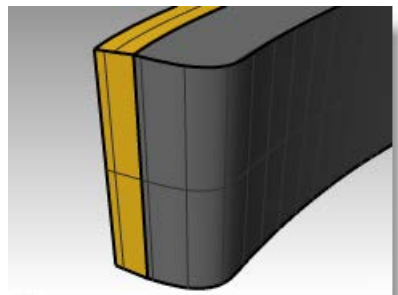


Join


Next you will join the split surface and the polysurface for the Top and the Bottom of the handset. The left hand (smaller) part of the split surface goes with the top of the phone handset, the larger part goes with the bottom part of the phone handset.

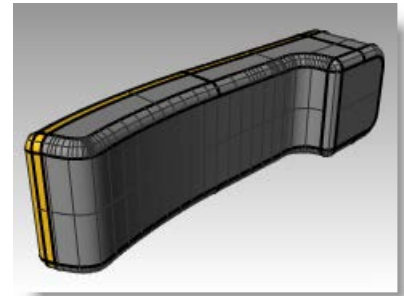
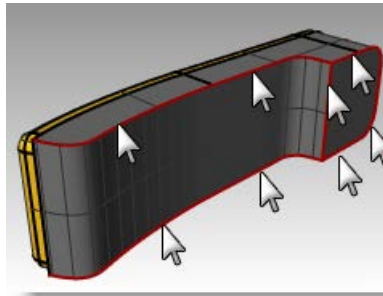
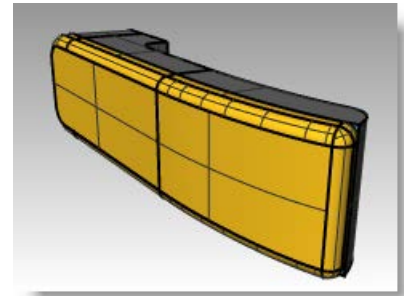
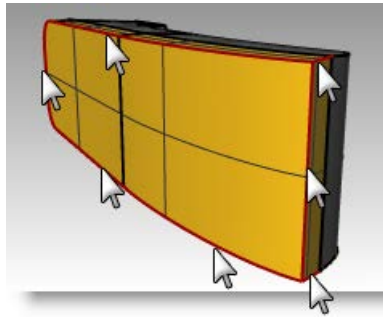
To join the surfaces:

- 1 Select the left (smaller) part of the split surface and the top polysurface
- 2 From the **Edit** menu click **Join**.
- 3 Select the larger part of the split surface and the bottom polysurface
- 4 From the **Edit** menu click **Join**.




To round the edges of the polysurfaces:

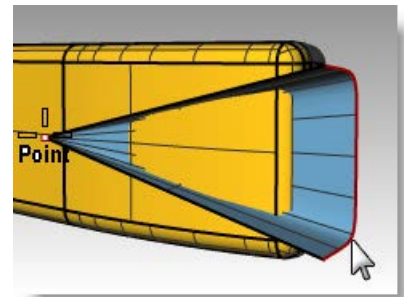
- 1 From the **Solid** menu, **Fillet Edge**, then **Fillet Edge**.  Use a radius of .2 to round the edges.
- 2 For the **Select edges to fillet**, select the edges around the top of the top polysurface and the two horizontal edges at the front, press **Enter**.
- 3 For the **Select fillet handle to 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:

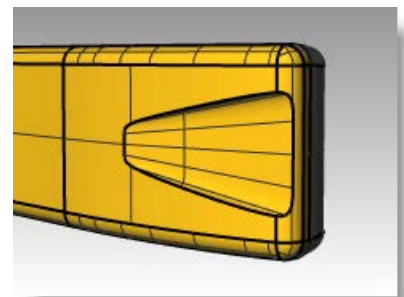
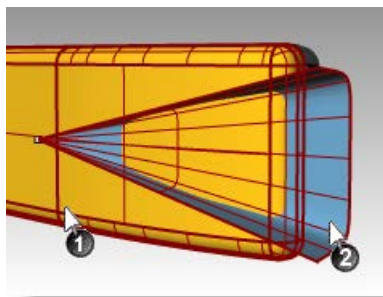
- 1 Turn on the Extrude to a Point layer. Turn off the Extrude Straight and Extrude Along Curve layers.
- 2 Select the U-shaped curve on the Extruded to a point layer.
- 3 From the **Surface** menu click **Extrude Curve**, and then click **To Point**. 
- 4 For the **Point to extrude to**, snap to the point object near the top surface.

The curve is extruded to the point.



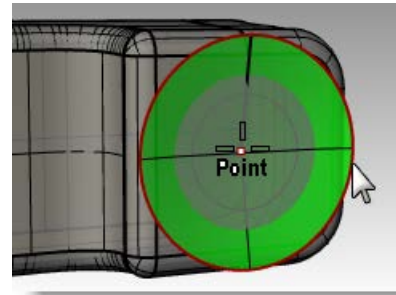
- 5 From the **Solid** menu, click **BooleanDifference** command (*Solid menu > Difference*) to remove the surface from the top of the phone.

If the result is not correct, flip the normals of the top polysurface or the extruded surface with the **Dir** command. The normals 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.

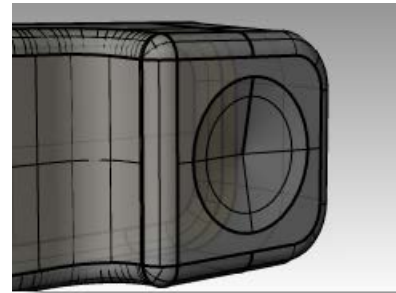
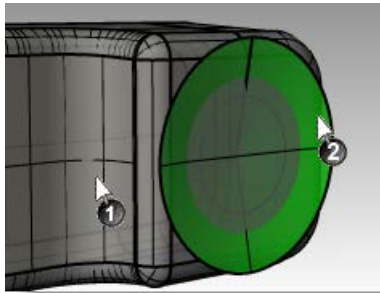
The curve is extruded to the point.



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

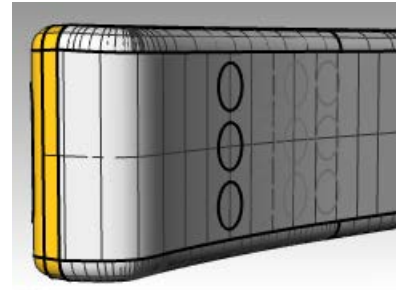
The normals of the bottom polysurface and the extruded surface should point toward each other.

- 10 Save your model as Phone.

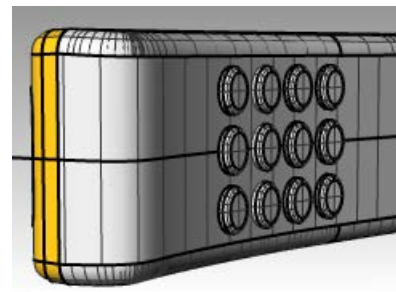
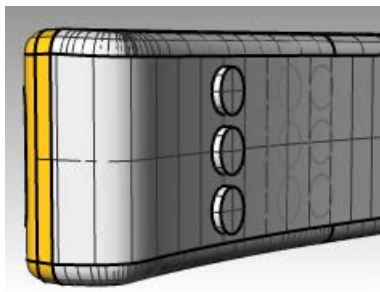


To create the buttons:

- 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 For the **Extrusion Distance**, type **-.2** and press **Enter**.



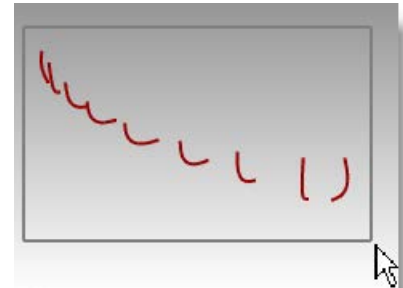
- 5 Repeat these steps for the other columns of buttons.
- 6 Use the **FilletEdge** command (*Solid menu > Fillet Edge > Fillet Edge*) with a radius of **.05** to round the edges.
The button edges are rounded.
- 7 Save your model.



Lofted surfaces - Canoe

Exercise 59—Lofted surfaces

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



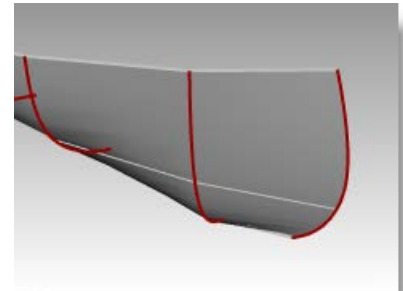
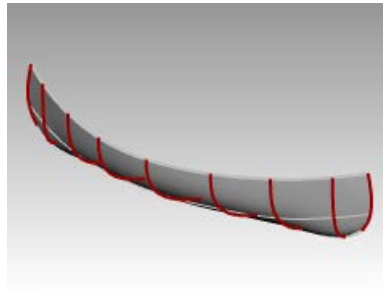
- 3 From the **Surface** menu click **Loft**.



A surface is fitted over the curves.

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

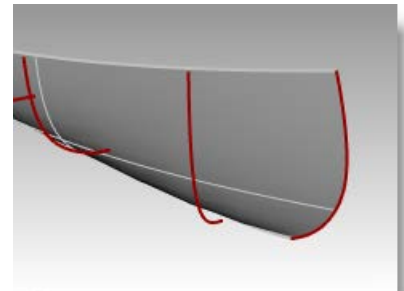
A surface is fitted through the curves, but the sections are straight between the curves.




- 5 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.

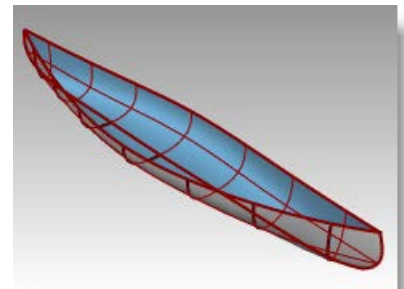
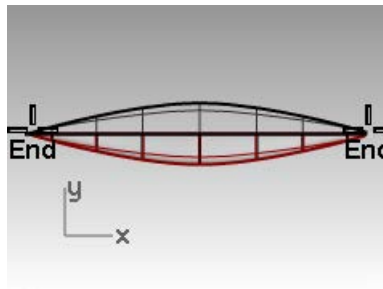
Use this option when you want the surface to conform to the control points of the input curves.




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

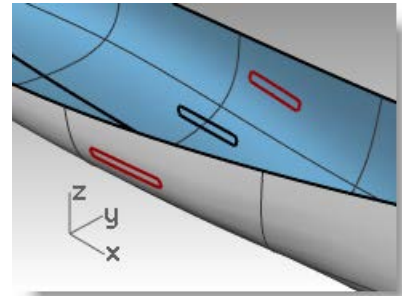
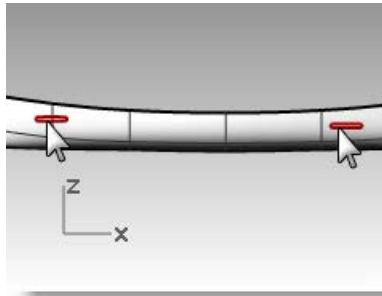
- 7 **Mirror**  the surface to create the other half.

- 8 **Join**  the two halves.



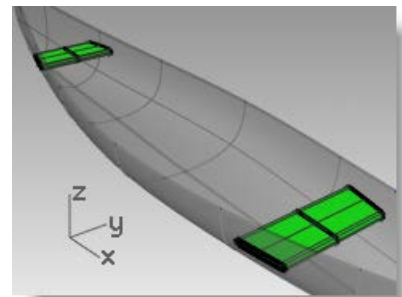
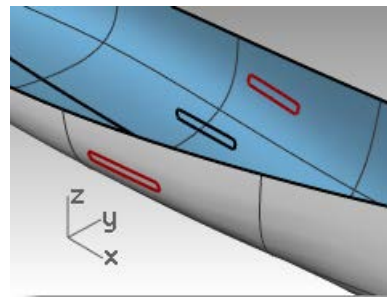
To make a seat:

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




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.

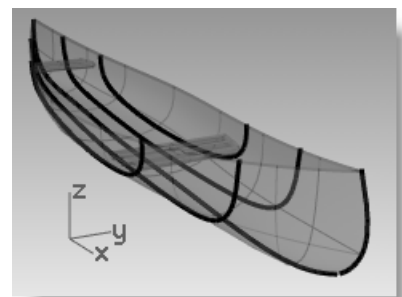
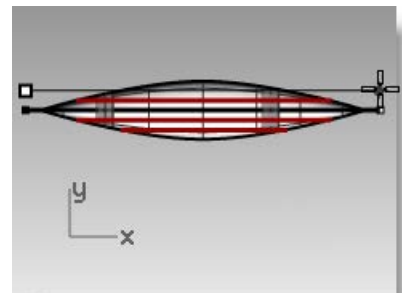
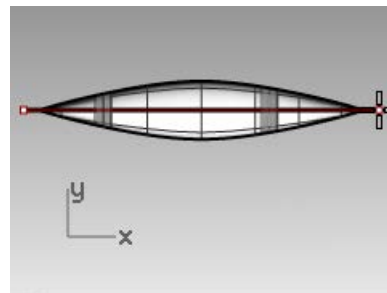
- 5 For the **Surface to project onto**, select the hull.
The curves will be projected to both sides of the hull surface.
- 6 Select one pair of curves on the surface.
- 7 From the **Surface** menu click **Loft**.
- 8 In the **Loft Options** dialog, click **OK**.
- 9 Repeat this for the other seat.




A surface is fitted over the curves that fit exactly with the shape of the hull.

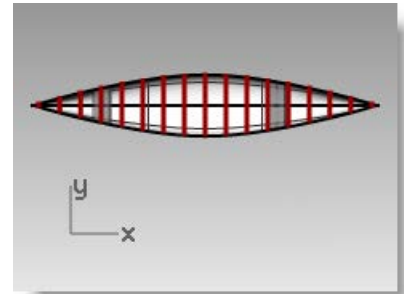
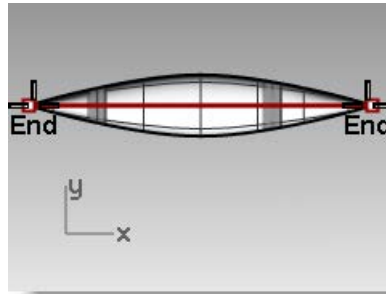
To create section curves from the surfaces:

- 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.
- 6 Press **Esc** to deselect the section curves.

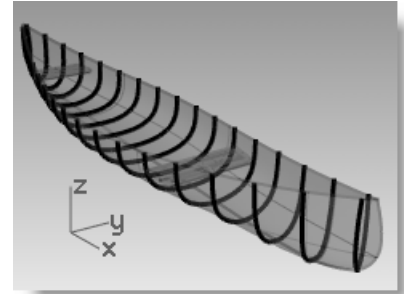


To create contour curves across the hull surfaces:


- 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.



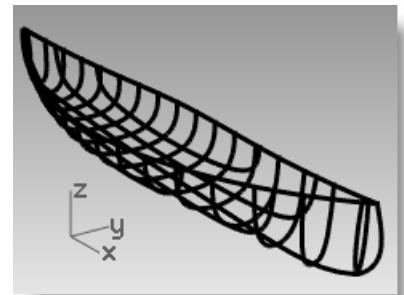
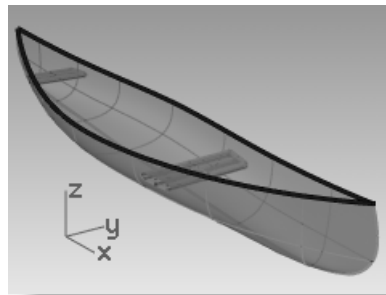
- 5 For the **Direction perpendicular to contour planes**, snap to the other end of the canoe.
- 6 For the **Distance between contours**, type **12** and press **Enter**.
A curve is generated every foot along the hull.
- 7 Press **Esc** to deselect the contour curves.



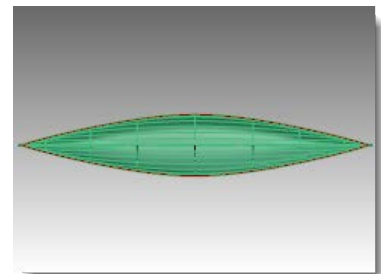
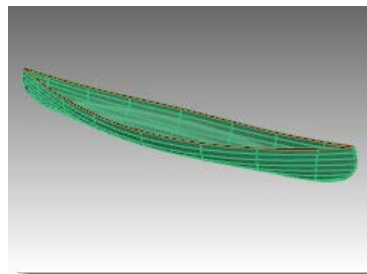
To create an edge curve from the surfaces:

- 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

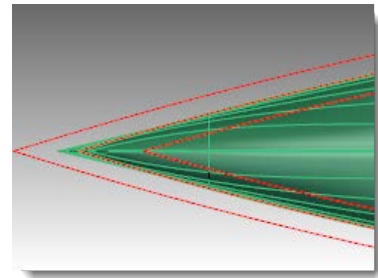
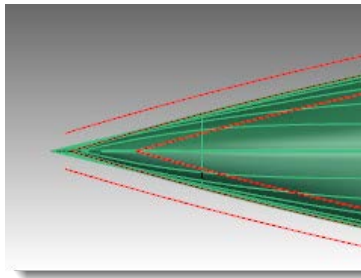


- 5 With the curves still highlighted, from the **Edit** menu click **Join**.

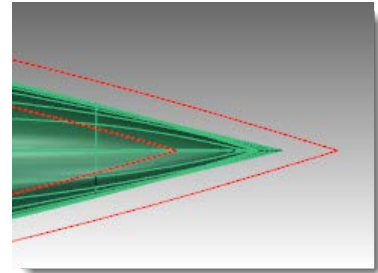
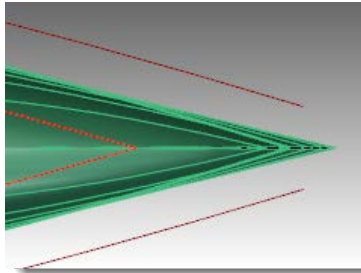


- 6 From the **Curve** menu click **Offset**, then click **Offset Curve** and enter an offset distance of **1**. Also select the **Bothsides** option.
- 7 **Delete** original curve, keeping inside and outside offset curves.

Note: The model unit is inches.

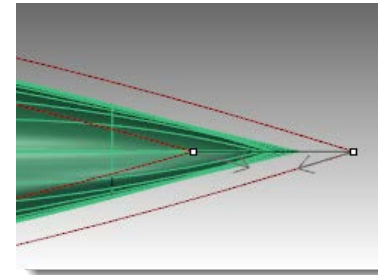


- 8 From the **Curve** menu click **Fillet Curves**. Set **Fillet radius** to **0** and **Join=Yes**. Select the two outside curves. Repeat for other end of canoe.

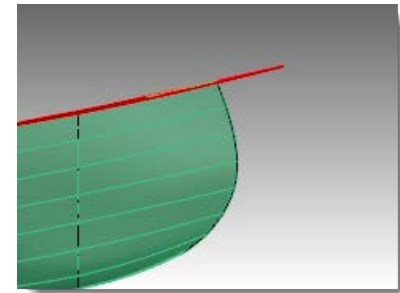
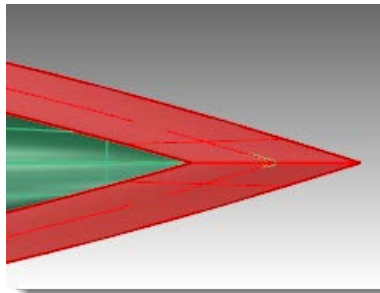


To create surfaces from the curves:

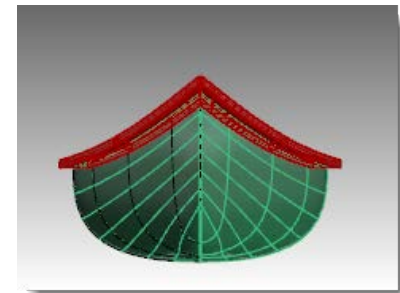
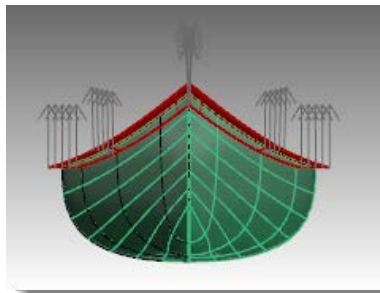
- 1 Select the curves you just made.
- 2 From the **Surface** menu click **Loft**.
- 3 In the **Loft Options** dialog box, set **Style** to **Normal**, and then click **OK**.
- 4 Select the lofted surface.



- 5 From the **Surface** menu click **Offset Surface**. Set the **Distance** to **1** and set the **Solid=Yes** and press **Enter**.



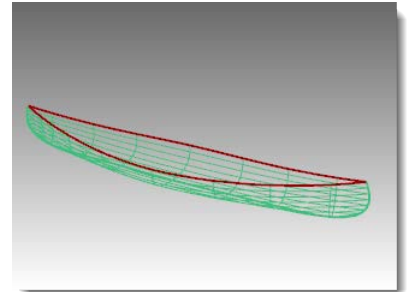
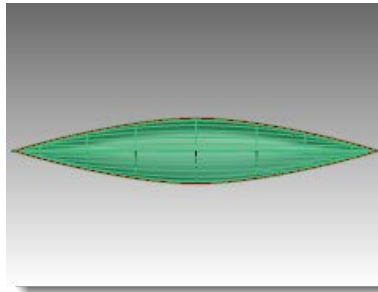
A polysurface rim is created.



To make a solid canoe:

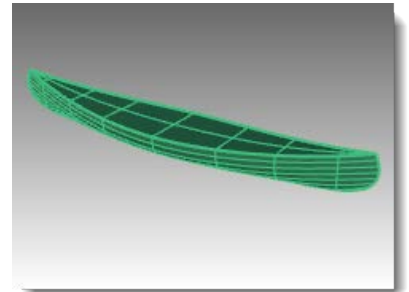
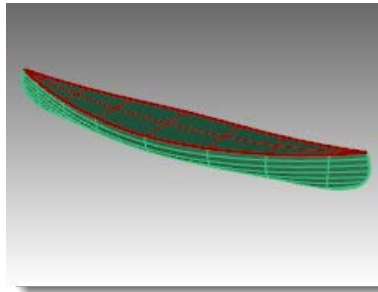
- 1 Make the Hull layer current and turn off Top Rail layer.
- 2 From the **Surface** menu click **Loft**
- 3 Pick the top edge of the hull.
- 4 Pick the other top edge and press **Enter**.

Lofted surface is created.



- 5 **Join** the new surface to the hull.

A closed polysurface is created. You should see the Join command end when the new surface is selected.

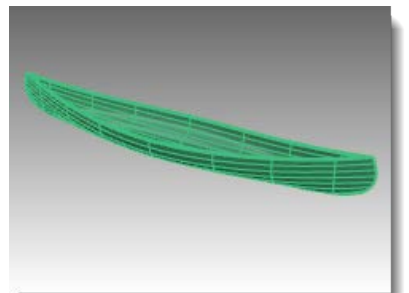
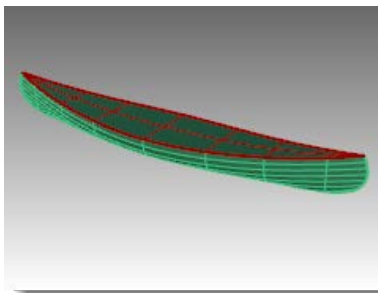


- 6 Use the **What** command to confirm that it is a valid closed polysurface.

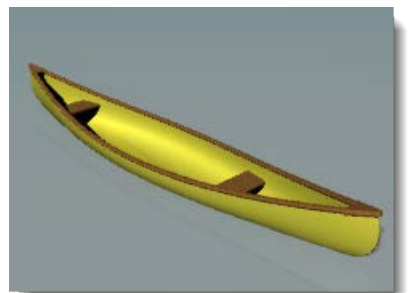
To shell the canoe:

- 1 Type **Shell** command.
- 2 For the **Faces to remove from closed polysurface** select the top surface.
- 3 Click **Thickness** option in command line, and type or **.5** and press **Enter**.

The surface will shell to a thickness of 1/2 inch.




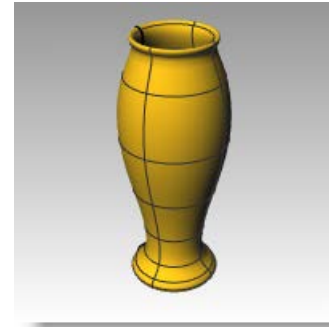
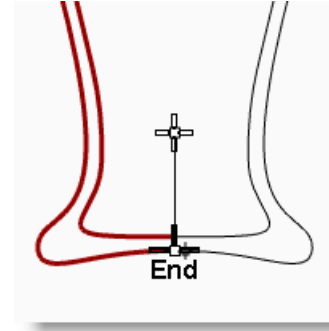
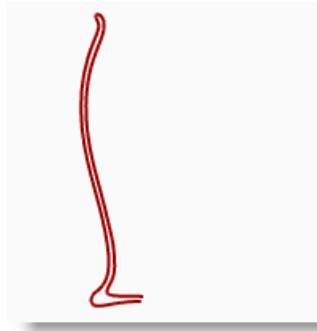
- 4 Turn on the Top Rail layer.
- 5 Assign materials to the hull and rim.
- 6 **Render** the canoe.



Revolved surfaces – Vase

Exercise 60—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.

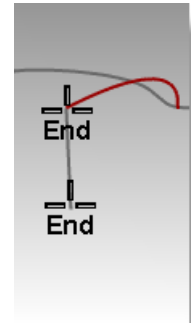
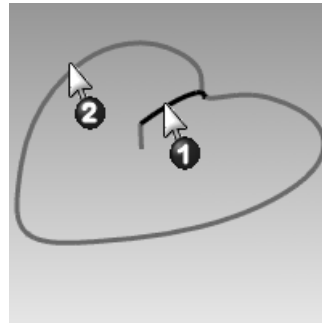
Rail Revolve – Heart, star

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

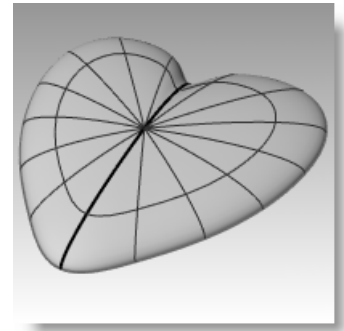
Exercise 61—Using a rail revolve

To create a rail revolve:

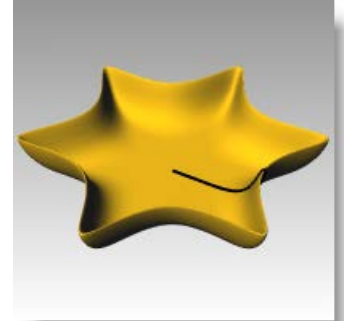
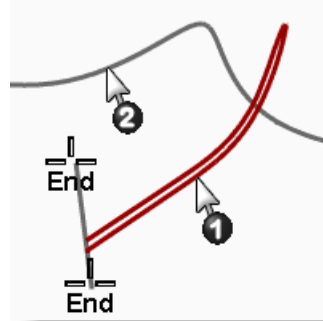
- 1 Open the model **Rail Revolve.3dm**.
- 2 From the **Surface** menu click **Rail Revolve**.

- 3 For the **Profile Curve**, select the conic shaped curve (1).
- 4 For the **Rail curve**, select the heart shaped curve (2).
- 5 For the **Start of revolve axis**, select one end of the vertical line.
- 6 For the **End of revolve axis**, select the other end of the vertical line.



A surface is revolved around the axis while it follows along the curve. The profile curve is scaled in one direction between the axis and the rail.




- 7 Turn the Bowl layer on and other layers off.
- 8 Repeat the previous steps to make a bowl.

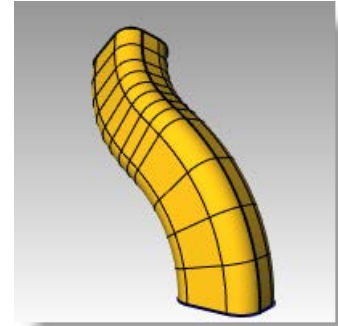
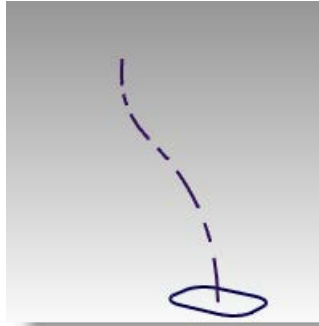


Sweeps and curve networks

Exercise 62—Using 1-rail sweeps to create surfaces

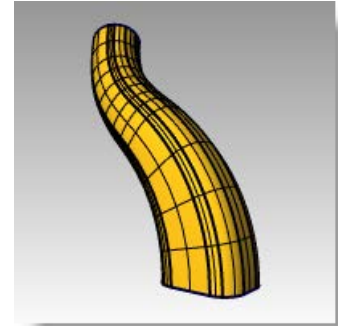
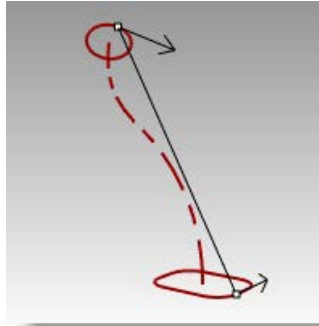
One cross-section:

- 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**.



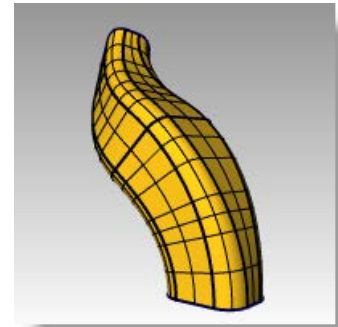
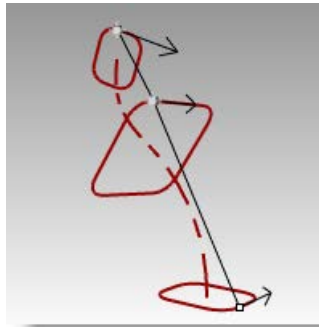
Two cross-sections:

- 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**.



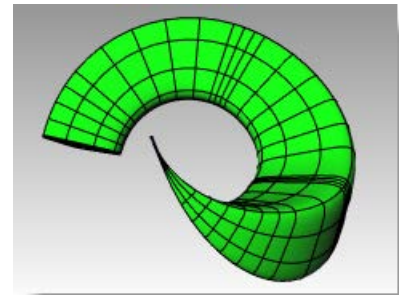
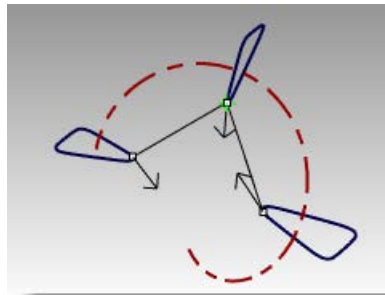
Multiple cross-sections:

- 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**.



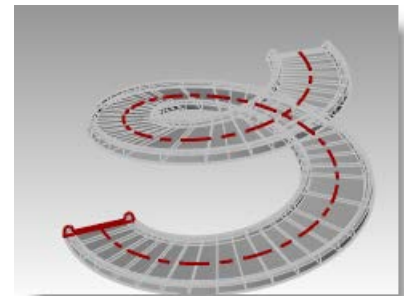
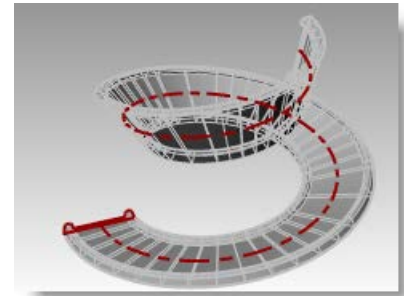
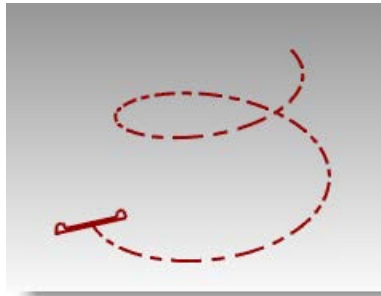
To create a 1-rail sweep to a point:

- 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**.



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 and press **Enter**.
- 5 For the **Drag seam point to adjust**, press **Enter**.
- 6 Change the style to **Roadlike Right**, click **Preview**.
- 7 Change the style to **Roadlike Front**, click **Preview**.
- 8 Change the style to **Roadlike Top**, click **Preview**. If the preview looks correct, click **OK**.



Rear-view mirror

Exercise 63—Using two-rail sweeps to create surfaces

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

To create the base—Part 1:

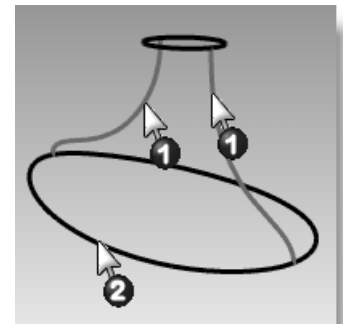
In the first part of this exercise, we will explore one of the Sweep2 options. To illustrate the option we will use one cross-section. In the second part we will use the same rail curves with two cross-sections. Lastly, we will use two rails that converge to a single point.



- 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.



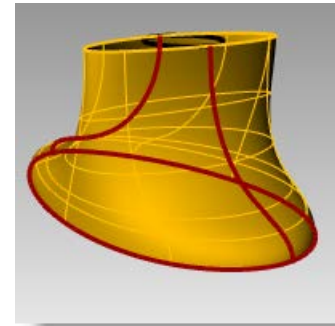
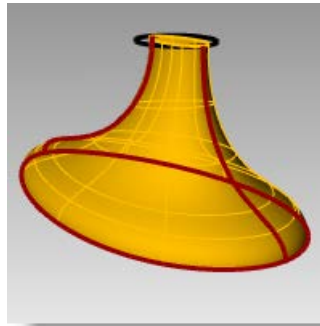
Since we only picked one cross-section, the surface doesn't conform to the circle at the top of the sweep.



- 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.

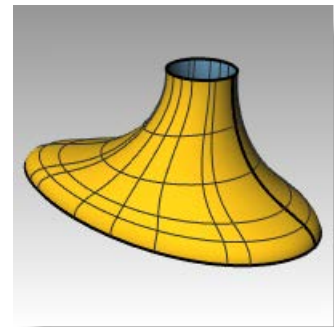
- 7 In the **Sweep 2 Rail Options** dialog, click **Cancel**.



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**.

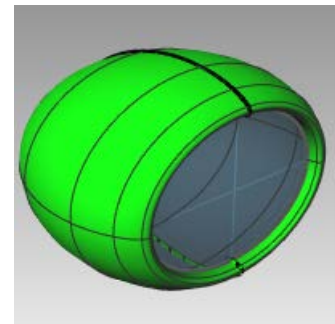
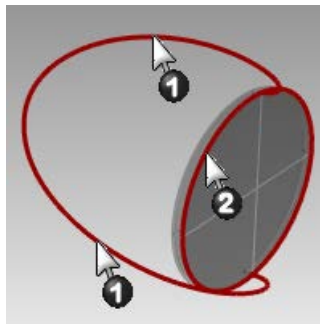
A surface is created whose edges match the rail curves and the cross-section curves.



To create the housing:

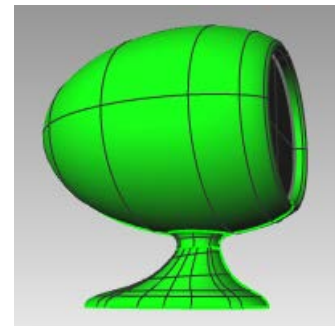
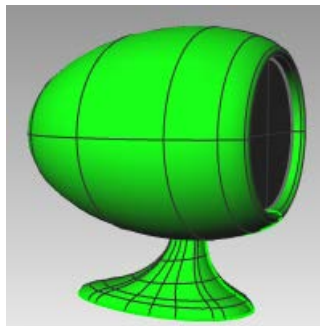
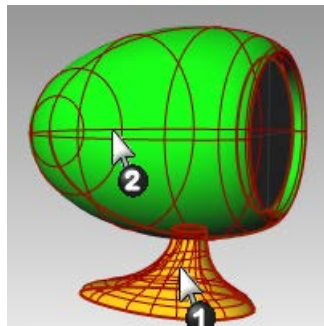
- 1 Turn the Housing Surface, Housing Curves, and Mirror layers on.
- 2 Make the Housing Surface layer current.
- 3 From the **Surface** menu click **Sweep 2 Rails**.
- 4 Select the **two rail curves** (1).
- 5 For the **cross-section**, select the outer edge of the cylinder, press **Enter** twice.
-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

A surface is created.



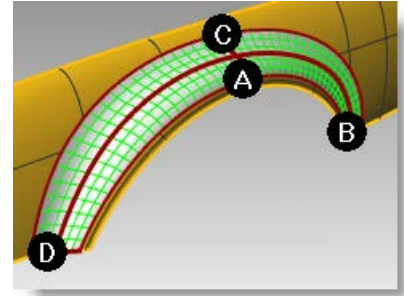
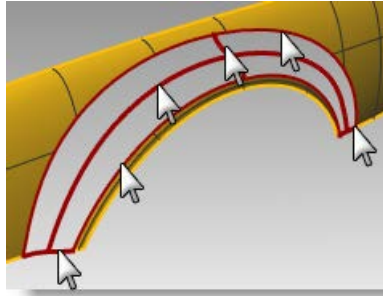
To join the two parts:

- 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 64—Using a network of curves to create surfaces

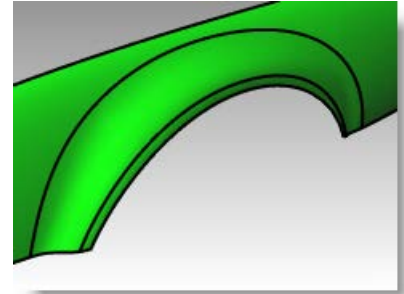
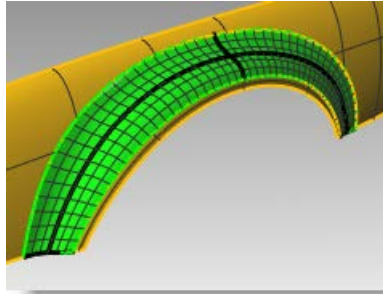
- 1 Open the model **Networksurf.3dm**.
- 2 From the **Surface** menu click **Curve Network**.
- 3 For **Select curves in network**, pick the two edge curves and the cross-section curves, and press **Enter**.



- 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.

Next let's analyze the surface continuity of the joined surface.

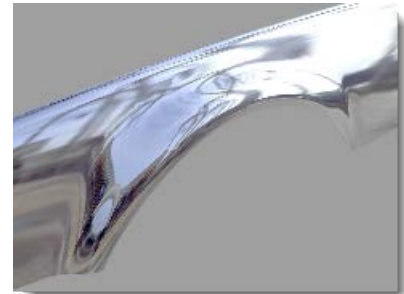
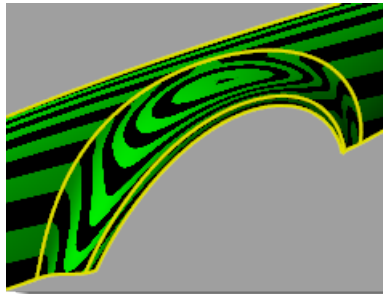


- 5 From the **Analyze** menu, click **Surface**, then click **Zebra**.

View the stripes where they cross the seams. Notice that they seem to be seamless and smooth.

- 6 From the **Analyze** menu, click **Surface**, then click **Environment map**.

Try changing the image by clicking the dropdown menu in the EMap options dialog.

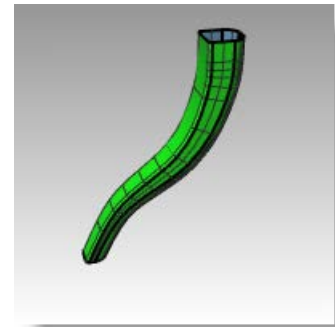
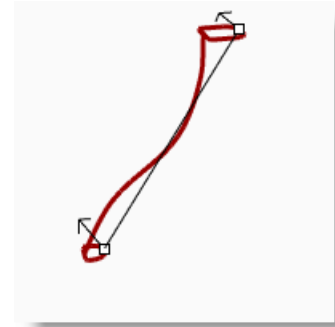


Exercise 65—Practice using one-rail sweeps

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.

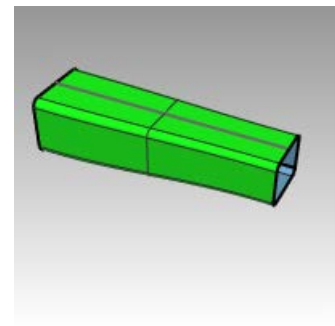
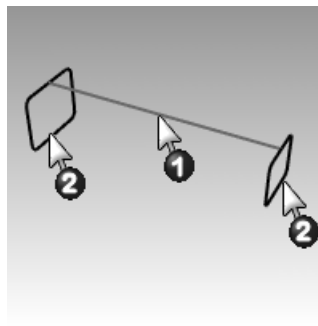


- 5 Press **Enter**.
- 6 Press **Enter**.
- 7 In the **Sweep 1 Rail Options** dialog box, click **OK**.

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

To create the brace:

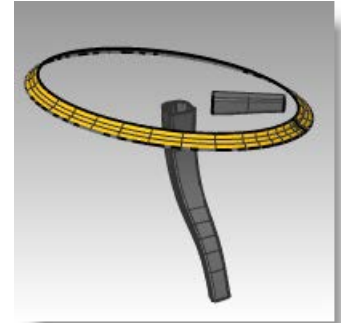
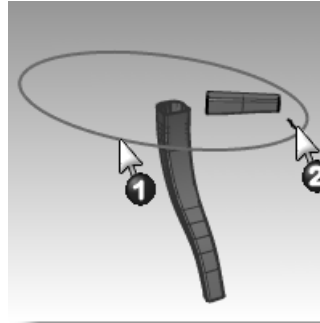
- 1 Change to the Braces layer.
- 2 Repeat the previous process to create the brace.



To create the top and finish the table:

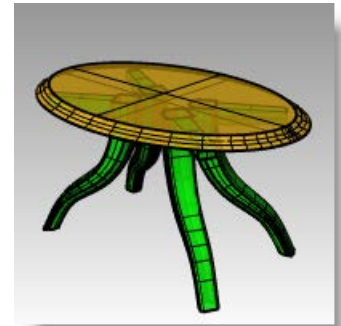
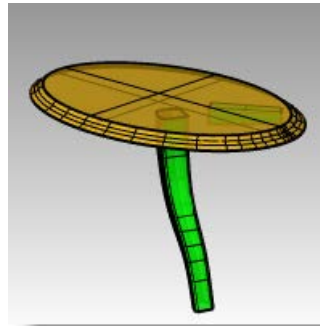
- 1 Change to the Top layer.
- 2 From the **Surface** menu click **Sweep 1 Rail**.
- 3 For the **Rail**, select the ellipse.
- 4 For the **Cross-section**, select shape curve.
- 5 Press **Enter**.
- 6 In the **Sweep 1 Rail Options** dialog box, click **OK**.

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.

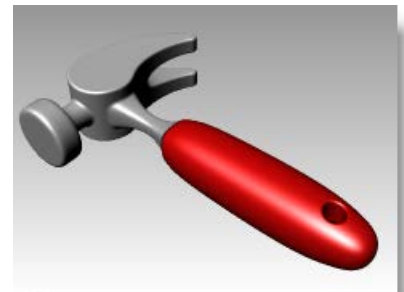


Toy hammer

Exercise 66—Creating a toy hammer

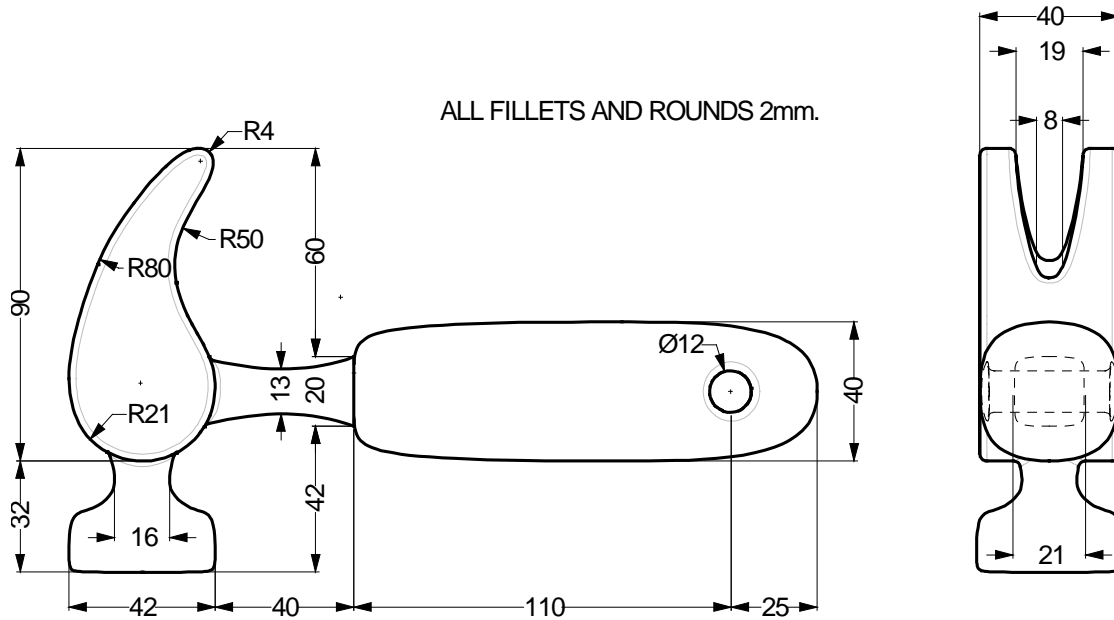
In this exercise you will use most of the techniques that you've learned in the previous sessions.

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



- 1 **Open** the model **Hammer.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.

Note: There is a layer named Construction Lines in this model that has the construction lines drawn. It also has a sublayer with centerlines for the model. Turn on these layers to speed up the exercise, if necessary.

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.

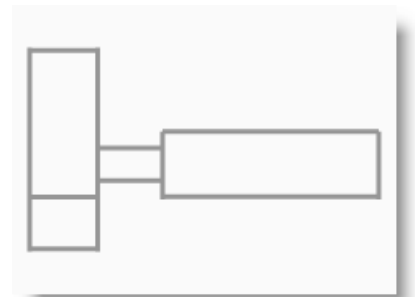
- 1 Change to the Curves layer
- 2 Draw a curve defining the shape of the claw in the Top viewport.

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.

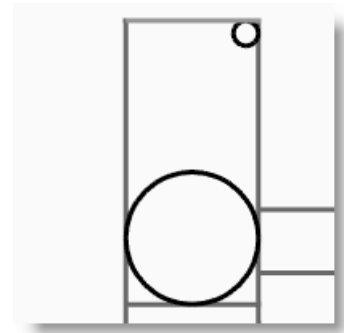
- 3 Use the **Circle** command (*Curve menu > Circle > Tangent to 3 curves*) to create a circle at the lower end of the claw.

Draw the circles tangent to the construction geometry

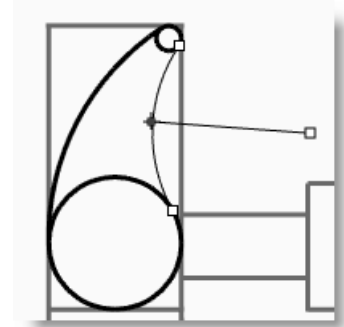
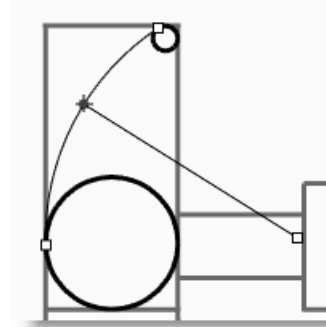


- 4 Use the **Circle** command (*Curve menu > Circle > Tangent, Tangent, Radius*) to create a circle at the upper end of the claw that is tangent to the upper right corner with a **4 mm** radius.

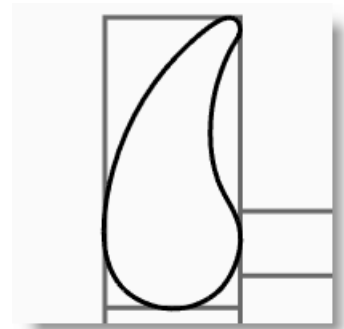
Draw the circles tangent to the construction geometry.



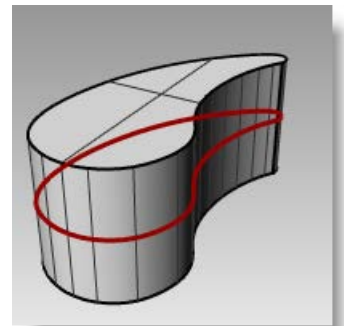
- 5 Use the **Arc** command (*Curve menu > Arc > Tangent, Tangent, Radius*) to create arcs that are tangent to the two circles.



- 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.



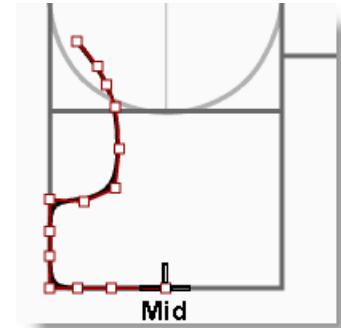
- 8 Change to the Claw layer.
- 9 Select the joined segments.
- 10 Use the **ExtrudeCrv** command (*Solid menu > Extrude Planar Curve > Straight*) to extrude the curve on both sides of the construction plane.



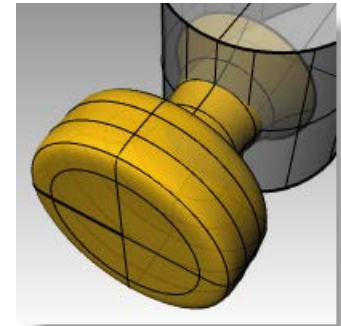
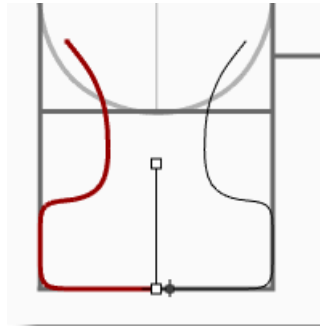
To create the head:

- 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.

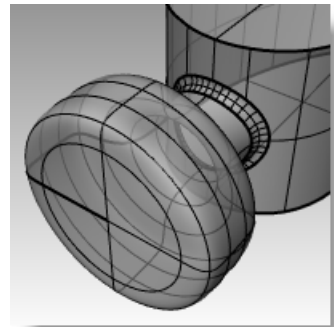
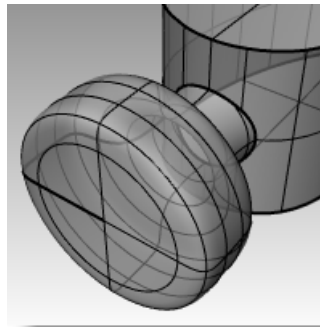


- 3 Change to the Head layer.
 - 4 Use the **Revolve** command (*Surface menu > Revolve*) to revolve the curve.
- Use the midpoint of the construction line for the revolve axis.
- 5 Save your model.



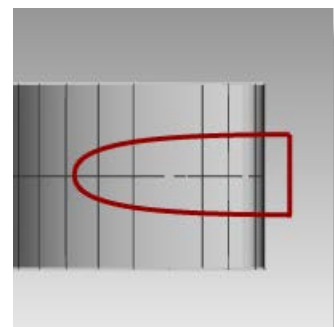
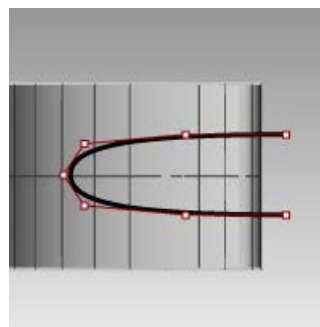
To add the head to the claw part:

- 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 Edge*) fillet the intersection between the head and the claw.
 - 3 Save your model.

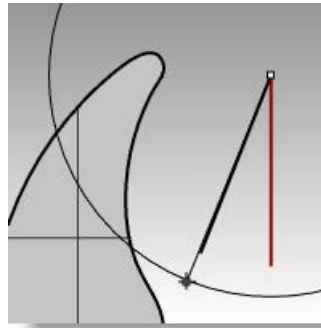


To create slot for the claw part of the hammer:

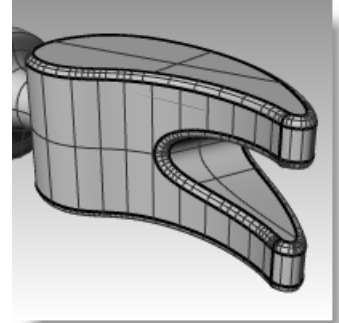
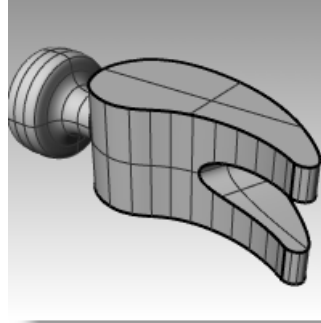
- 1 Use the **Curve** command (*Curve menu > Free-form > Control Points*) to draw a curve for the slotted part of the claw.
- Make sure the curve is symmetrical.
- 2 Use the **Line** command (*Curve menu > Line > Single Line*) to draw a line between the endpoints.
 - 3 Use the **Join** command (*Edit 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*) or the **Gumball** modeling aid 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.

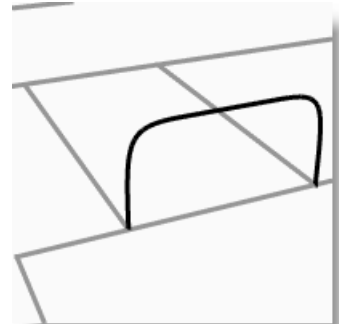
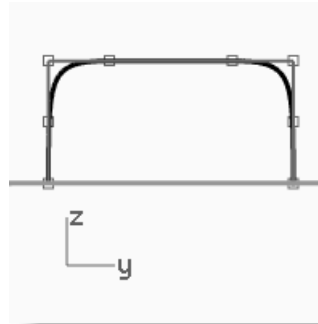


To create the shape curve for the tang and the handle:

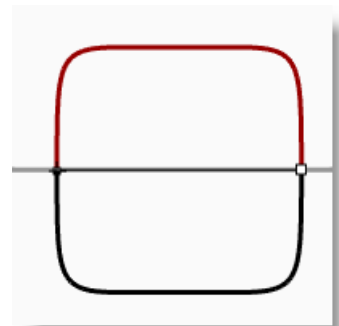
Create the shape curve for the tang in the Right viewport. This curve will also be used for the handle.

- 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.

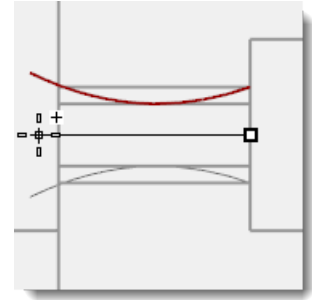
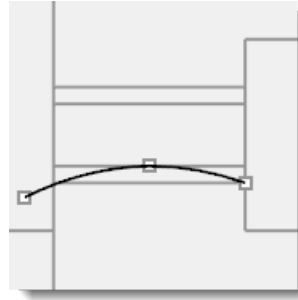


- 3 Use the **Mirror** command (*Transform menu > Mirror*) to create the other curve.
- 4 Use the **Join** command (*Edit > Join*) to join the curves.
- 5 Save your model.

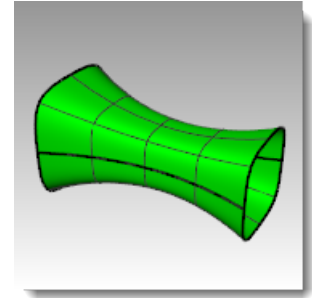


To create the tang:

- 1 Use the **InterpCrv** command (*Curve menu > Free-form > Interpolate Points*) to draw one of the curves for the tang of the hammer.
- 2 Use the **Mirror** command (*Transform menu > Mirror*) to create the other curve.
- 3 Change to the Tang layer.

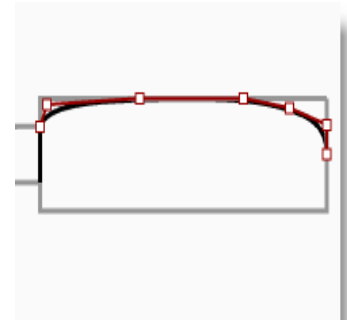
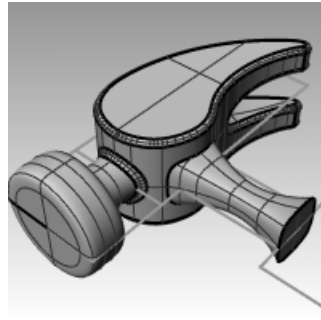


- 4 Use the **Sweep2** command (*Surface menu > Sweep 2 Rails*) to make the surface.
- 5 Use the **Cap** command (*Solid menu > Cap Planar Holes*) to make the tang a closed polysurface.
- 6 Save your model.



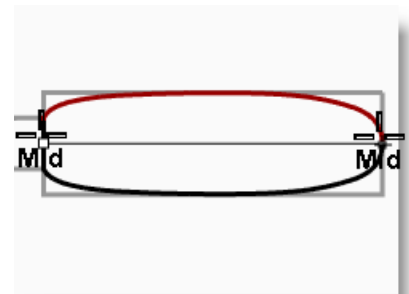
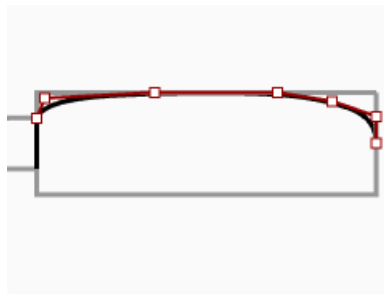
To finish the hammer head:

- 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.
- 4 Save your model.



To create the handle:

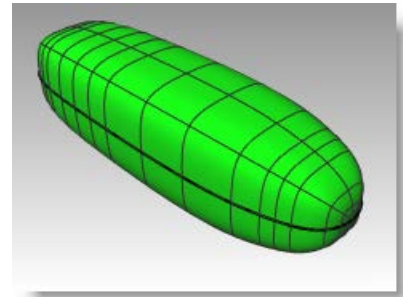
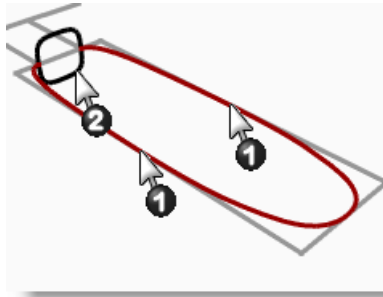
- 1 Change to the Curves layer.
- 2 Use the **Curve** command (*Curve menu > Free-form > Control Points*) to draw a curve for the top edge of the handle.
- 3 Use the **Mirror** command (*Transform menu > Mirror*) to make the other half.



- 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.



To create the hole for the handle:

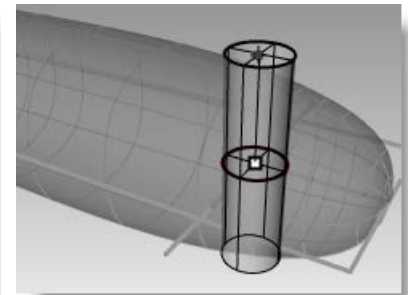
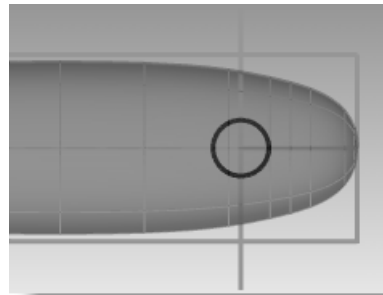
- 1 Use the **Circle** command (*Curve menu > Circle > Center, Radius*) to make a circle 25mm from the end of the handle.

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.

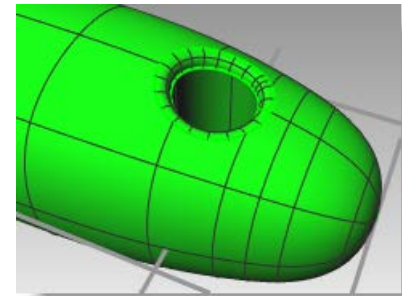
- 3 Save your model.



- 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.

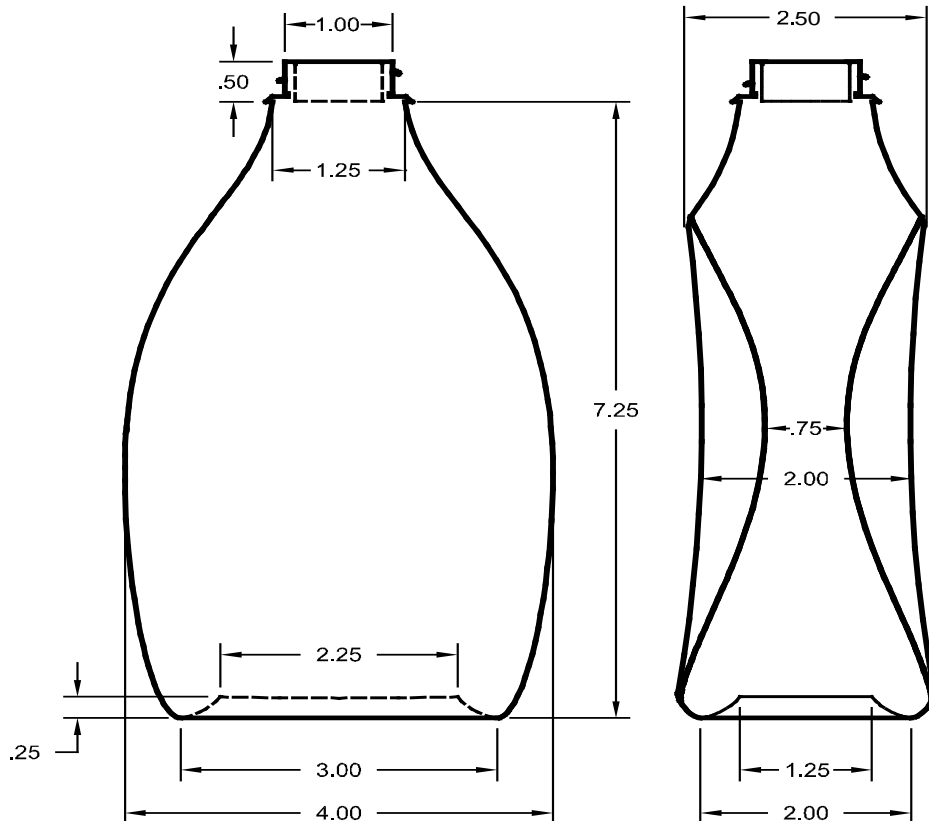


Squeeze bottle

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.

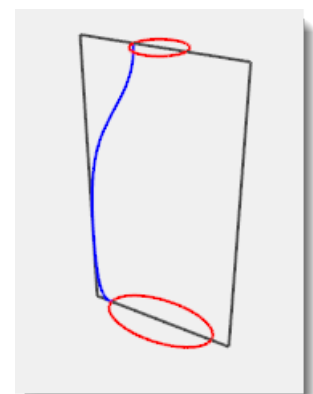
Exercise 67—Creating a squeeze bottle

The technical drawing is included to help you create a very precise model.

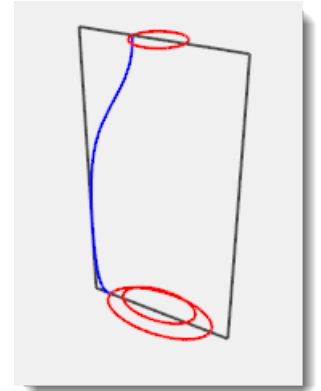
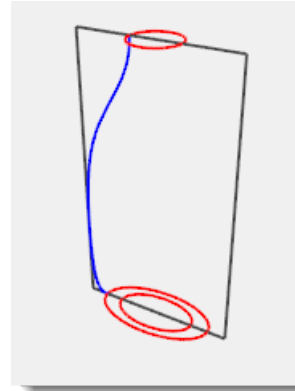


- 1 Open model **Squeeze Bottle.3dm**.
- 2 Use the pre-drawn rectangle to create a **circle**, an **ellipse**, and a profile **curve**.
These curves will be used to generate the surfaces of the bottle.

Note: A set of these curves is already included in the model on layers Rail Curves and Profile Curves. These layers are sublayers of a layer named Bottle Curves.

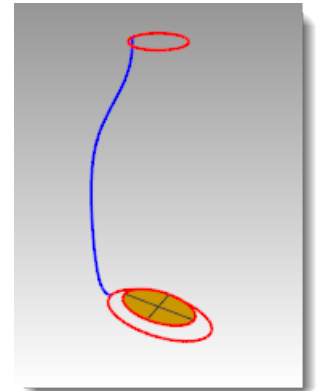


- 3 Make another (smaller) **ellipse** that will be used for the concave part at the bottom of the bottle.
- 4 **Move** this ellipse vertically **.25** units.

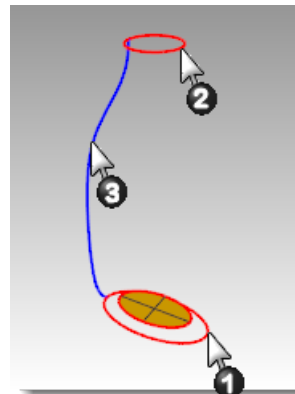


To create the surfaces for the bottle:

- 1 Change to the Bottle Srf layer and turn off the Construction layer.
- 2 Select the small ellipse.
- 3 Use the **PlanarSrf** command (*Surface > Planar Curves*) to make a flat surface.

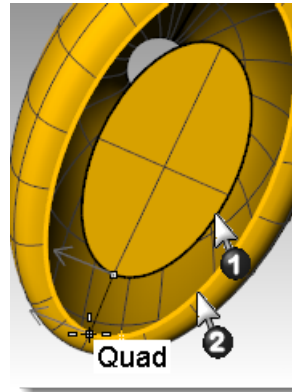


- 4 Select the large ellipse and the circle.
- 5 Start the **Sweep2** command (*Surface > Sweep 2 Rails*).
The pre-selected ellipse and circle will be the rails for the sweep.
- 6 For the **Select cross section curves**, choose the profile curve, and then press **Enter**.
- 7 In the **Sweep 2 Rail Options** dialog box, click **Do not simplify** and check **Closed sweep**, and then click **OK**.



To create a blend surface for the bottom of the bottle:

- 1 **Hide** the rail and profile curves.
- 2 Start the **BlendSrf** command (*Surface > Blend Surface*).
- 3 For the **Segment for first edge**, select the edge of the ellipse surface, and then press **Enter**.
- 4 For the **Segment for second edge**, select the edge of the bottle surface, and then press **Enter**.
- 5 For the **Drag seam point to adjust**, move the seam points so that they align with each other, and then press **Enter**.
- 6 In the **Adjust Surface Blend** dialog, make click **Preview**. Make any adjustments that are necessary and then click **OK**.




- 7 **Join** the three surfaces.

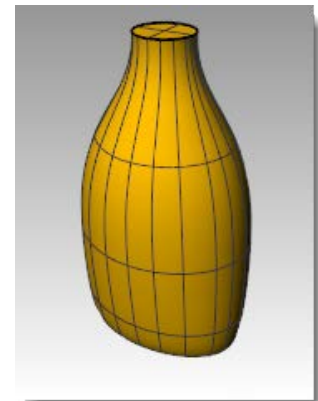
Cap the top

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.

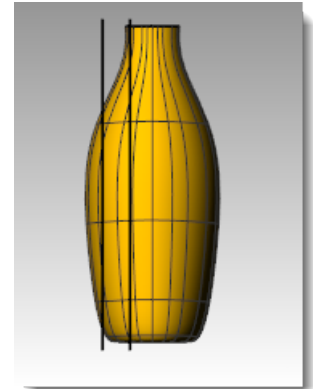
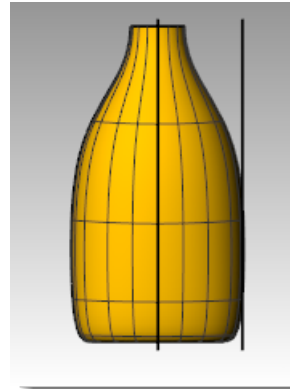


Flatten the sides

In this part of the exercise you will create 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 Default layer.
- 2 In the Front viewport, draw two lines. One line in the middle and one on the 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.

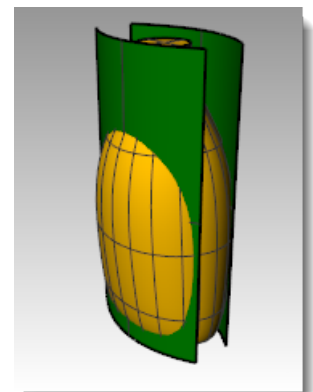
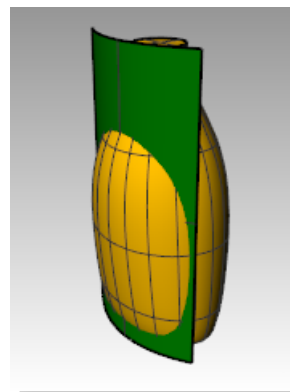


- 4 **Mirror** the line on the side to the other side of the bottle.
These lines will be used to create a cutting surface for the flat side of the bottle.

Note: A set of these curves is already included in the model on layer Cut Curves, a sublayer of the Bottle Curves layer.




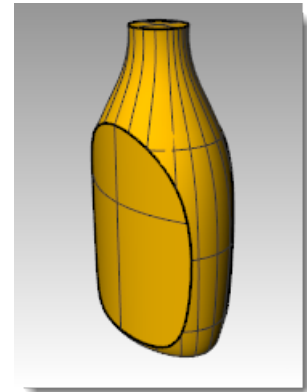
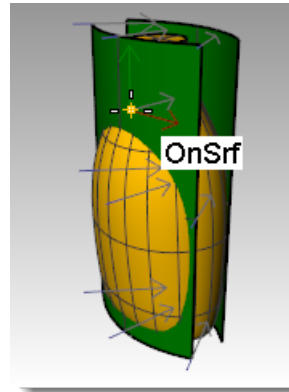
- 5 Change to the Cut Srf layer.
- 6 Select the three curves you just created.
- 7 Use the **Loft** command (*Surface menu > Loft*) to make the cutting surface.
- 8 In the **Loft Options** dialog, uncheck **Closed loft**, click **OK**.
A lofted surface intersects the bottle.
- 9 **Mirror** the surface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



To remove the surface from the bottle:

- 1 Change to the Bottle Srf 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.



Note: It is possible to create a hollowed out shell from a solid polysurface, like this bottle. The command for doing this is **Shell**.

Shell only operates on simple, solid, manifold polysurfaces. For more information on this command view the **Shell Help topic**.

Create the bottle's top

To create the bottle top, you are going to revolve a profile curve to create the surface.

To create the profile curve:

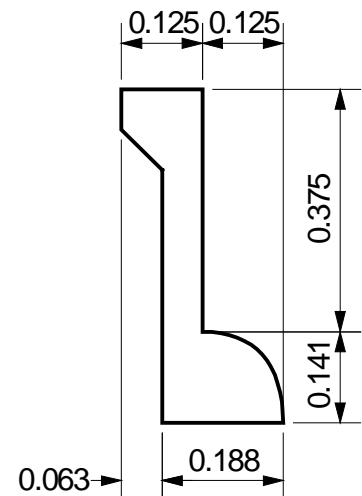
- 1 Change to the Default layer.
- 2 In the Front viewport, use the **Lines** command (*Curve menu > Line > Line Segments*) and the **Arc** command (*Curve menu > Arc > Center, Start, Angle*) to create a profile curve for the outside and the inside of the top.
- 3 Use the drawing to the right to draw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

You will move it to a precise location after it is made.

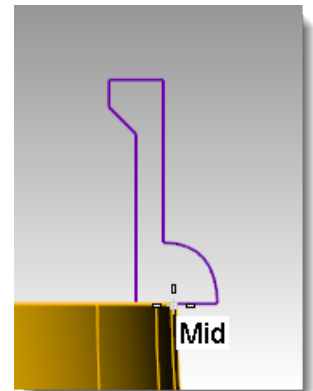
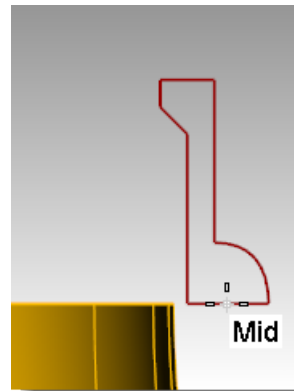
Tip: When drawing line segments, you can use object snaps, the distance constraint, and ortho to draw precisely.

- 5 Use the **Join** command (*Edit menu > Join*) to join the segments together.


Note: A profile curve for the top is already included in the model on layer Bottle Top Curve, a sublayer of the Bottle Curves layer.



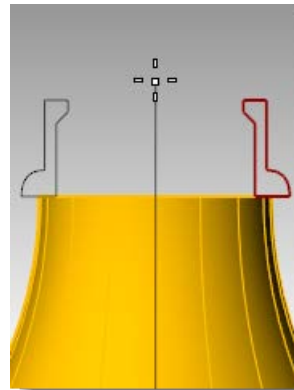
- 6 Use the **Move** command (*Transform menu > Move*) to move the profile from the midpoint of the bottom edge to the midpoint of the top of the bottle.
- 7 Change to the Bottle Top layer.




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.



- 6 Use the **BooleanUnion**  command (*Solid menu > Union*) to join the two polysurfaces together.



Note: Practice additional surface modeling techniques by creating threads on the bottle top model. Here are two web links that include a video on techniques to make threads with Rhino:

[Modeling Screw Threads, Part 1](#)

[Modeling Screw Threads, Part 2](#)

10 Importing and exporting

Rhino supports many different import and export formats, making it possible to model in Rhino and then export your model to downstream processes, or import models from other software applications into Rhino. For a complete list of import and export file types refer to the **Rhino Help > Contents > File I/O > File Formats**.

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 use the **Save As** command to choose a specific export format to export an entire model. Or, you can select some objects and use the **Export Selected** command to 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.

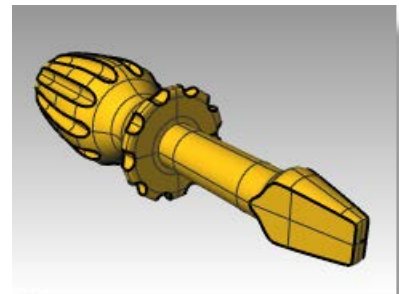
Importing other file formats into Rhino

We will not be importing any models in this class. Importing files from other applications is covered in our Level 2 Training course. If you have any specific questions about importing models into Rhino, consult your instructor.

Exercise 68—Exporting models

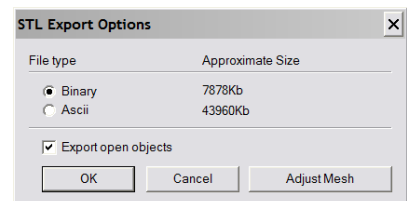
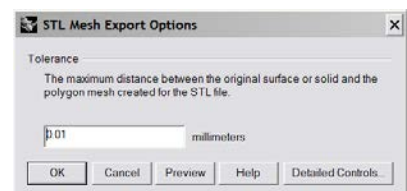
To export a model to a mesh format:

- 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.

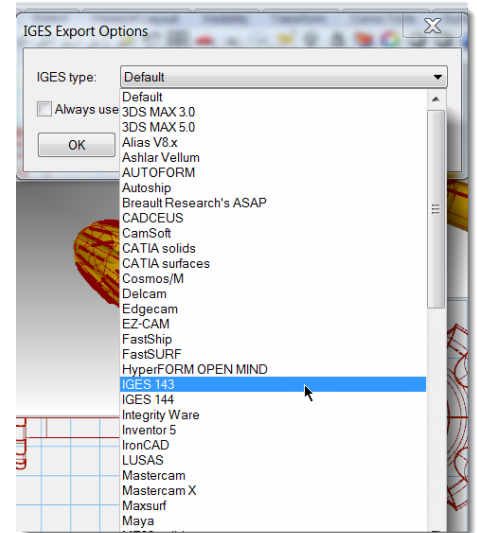


To export a model to IGES:

- 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**.

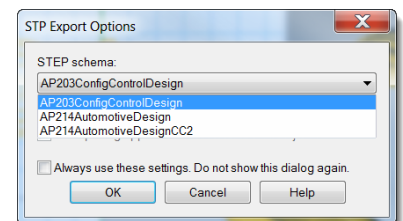
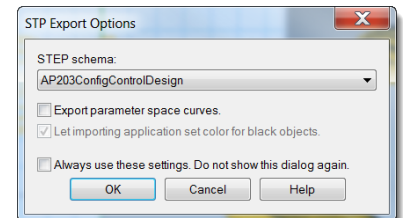
Detailed controls allow the user more input.

- 4 Click **Cancel** to terminate, or **OK** to create the IGES file.



To export a model to STEP:

- 1 From the **File** menu click **Save As**.
- 2 In the **Save** dialog box, change the **Save as type** to **STEP (*.stp, *.step)**.
- 3 In the **STEP Options** dialog box, use the default setting.



11 Rendering

Rendering is available for showing your model as if it was photographed or sketched. If you render to look like a photo, this is called photorealistic rendering. [Flamingo nXt](#) is an example of a photorealistic render plug-in for Rhino. If you render to look hand sketched, this is called non-photorealistic. [Penguin](#) is an example of this type of render.

Both types of renderers are available as plug-ins for Rhino. The built-in Rhino renderer may be good enough for much of your work. If not, use another rendering program such as Flamingo nXt, V-Ray, Maxwell, Brazil or other plug-ins for higher quality results. Plug-ins for Rhino are listed on the Resources page of the [Rhino web site](#).

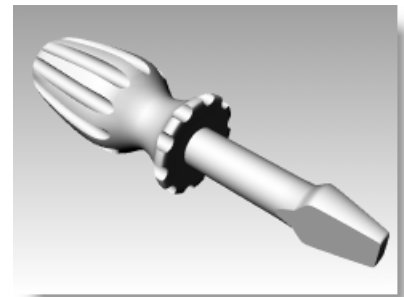
Apply materials

Rhino's built-in renderer uses materials with color, gloss and transparency settings, 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 69—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.

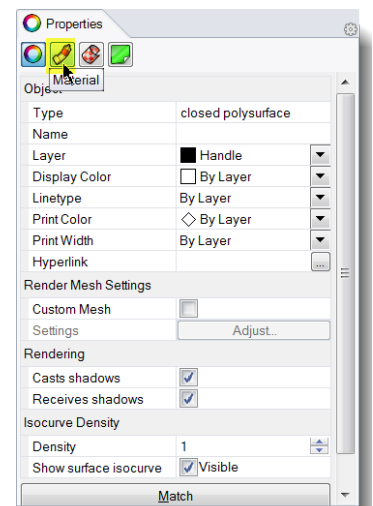
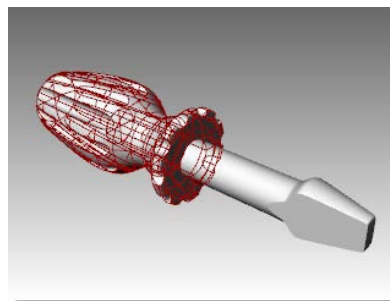
The viewport mimics but does not exactly duplicate what you will get in a Render.



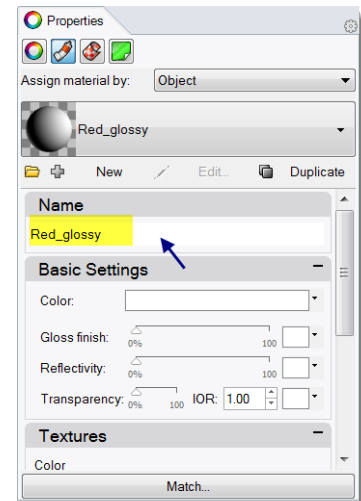
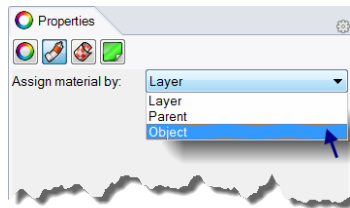
To assign a material to the handle by object:

To render the handle in color, we will first assign a red glossy material to the handle object. This material assigned to the object will override any material that might be assigned to the layer of the object.

- 1 Select the handle.
- 2 In the **Properties** panel, click on the **Material** page.

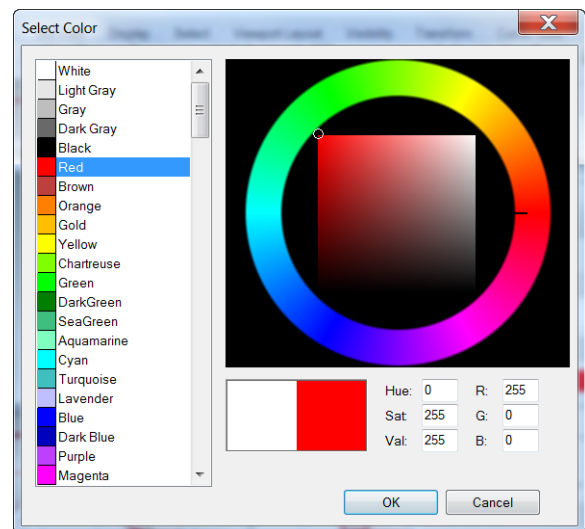
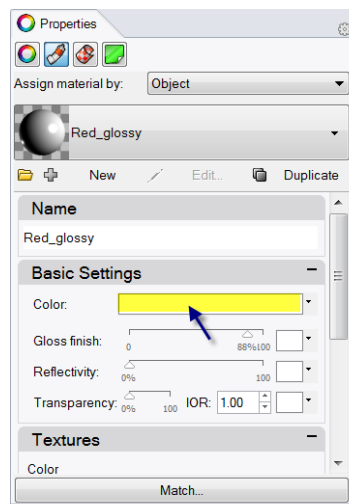


- 3 On the **Material** page box, at **Assigned material by:** choose **Object**.
- 4 From the **Material** page, in the name field type **Red_glossy**.



- 5 Next click the **Color** swatch.
- 6 In the **Select Color** dialog box, select a color, like **Red**, and click **OK**.

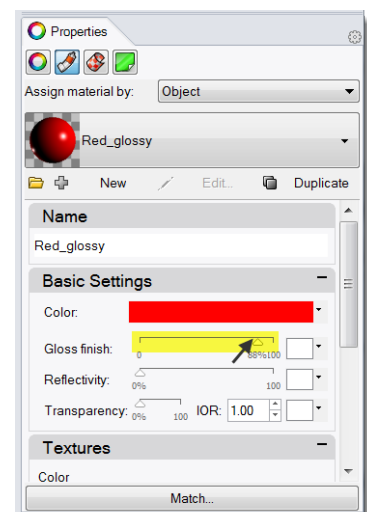
To give the handle a highlight, change the Gloss finish setting.



- 7 Change the **Gloss finish setting** slider to value between **80** and **90**.

A gloss finish of 0 means that the object is not shiny at all and it will not have a specular highlight. A low value of gloss finish makes the shiny spot to be small, and the object will look glossier. As the gloss finish increases in value, the shiny spot gets larger—this makes the object look more like it is made out of a reflective material.

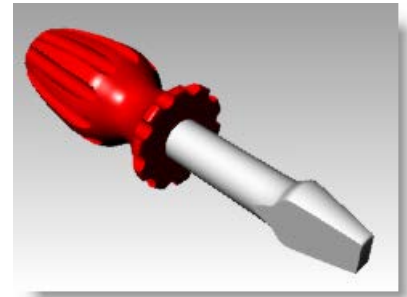
The shiny spot only appears when you are looking at an object at a certain angle relative to the angle of the light.



- 8 From the **Render** menu click **Render**.



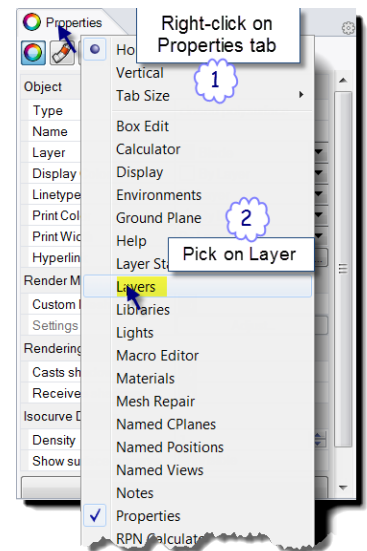
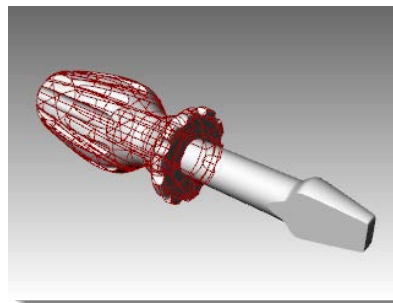
A display window appears with the current viewport rendered in colors, but it will probably lack detail. You can close the Display Window without disturbing your model. Placing lights will add depth and detail to the rendered image.



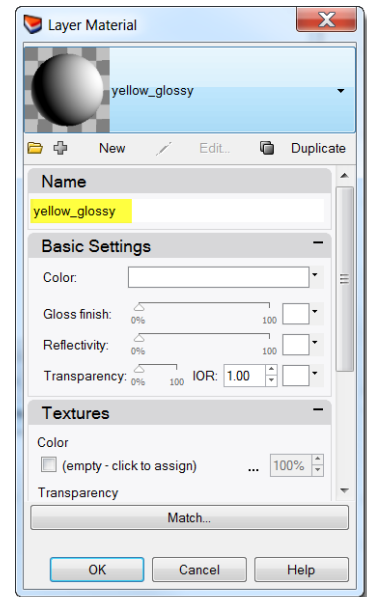
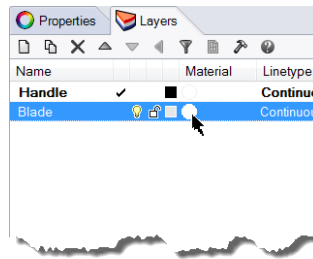
To assign a material to the blade by layer:

To render the blade in color, we will next assign a yellow glossy material to the blade layer. All the objects on the blade layer that do not have an object material override, are rendered in the material. That is an advantage of rendering with material set to by layer. Changing the layer's material will result in all the objects on that layer updating.

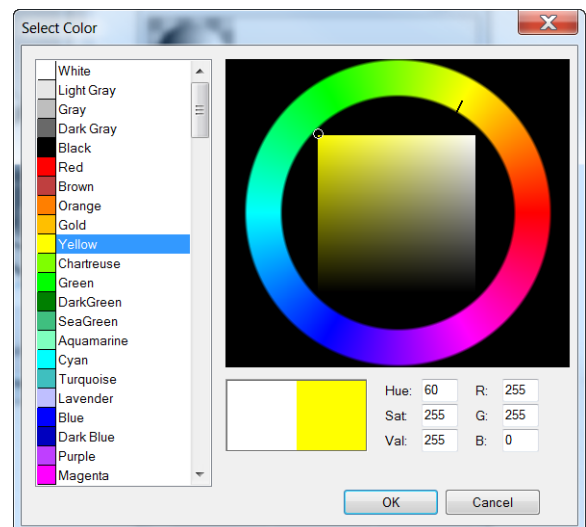
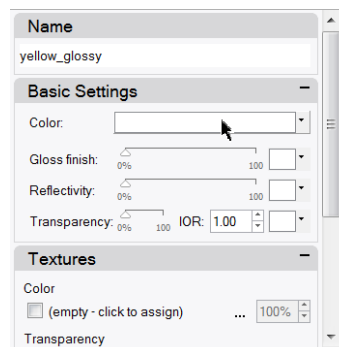
- 1 Select the blade.
- 2 Right-click on the **Properties** panel.
- 3 From the right-click menu pick on Layers.



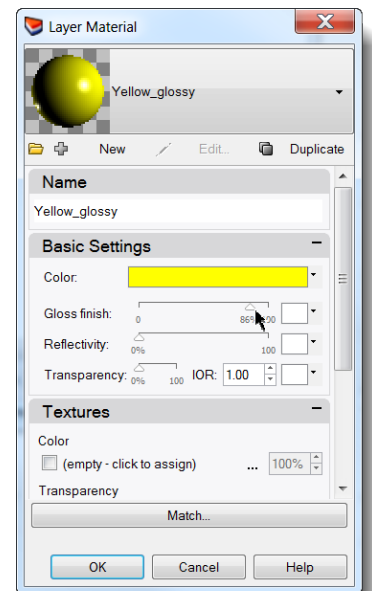
- 4 In the **Layers** panel, click on the **Material** icon.
- 5 In the **Layer Material** dialog, type the name **Yellow_glossy**.



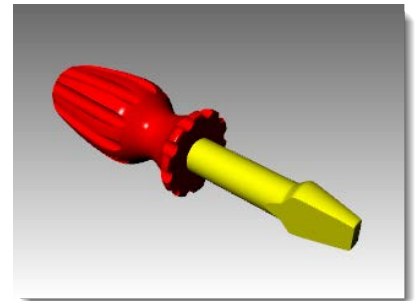
- 6 Under **Basic Settings**, click the **Color** swatch.
- 7 In the **Select Color** dialog box, select a color, like **Yellow**, and click **OK**.



- 8 Change the **Gloss finish setting** slider to value between **80** and **90**.



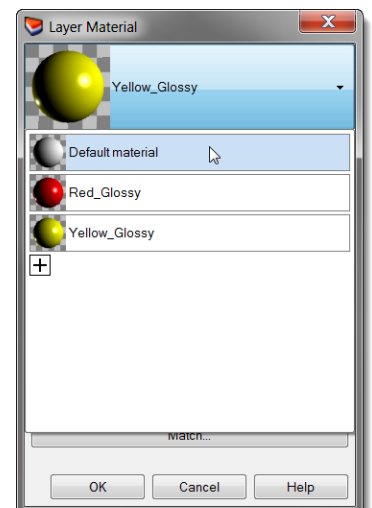
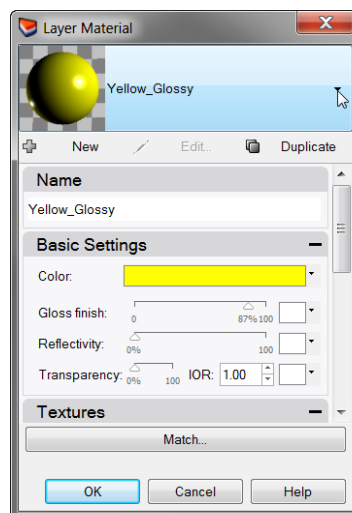
- 9 From the **Render** menu click **Render**.



To add a new material to a layer:

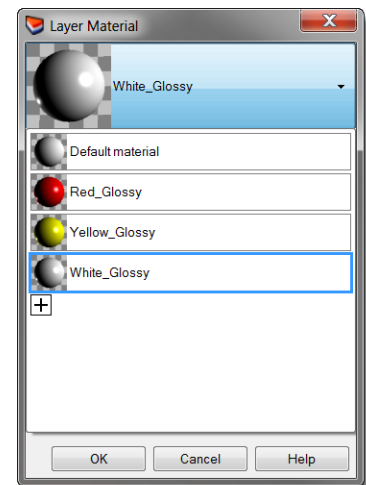
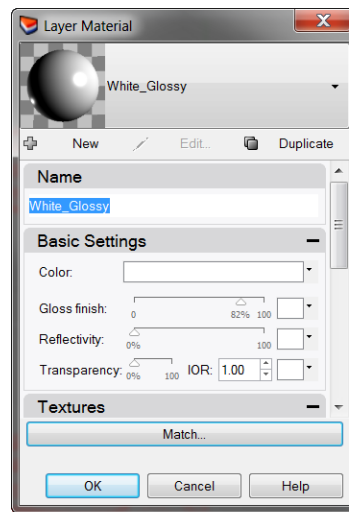
- 1 In the **Layers** panel, click on the **Material** icon.
- 2 In the **Layer Material** dialog, click the down arrow next to the Yellow_Glossy material.
- 3 When the list of materials appears, click **Default** material.

It's best to use the Default material when making new materials.



- 4 In the **Name** box type **White_glossy**, and change the **Gloss finish** setting slider to value between **80** and **90**.
- 5 **Render** the model.
- 6 The next time you click on the down arrow for a material listed in the **Layer Material** dialog, you will see a list of the Default material and the three materials you just created.


You can switch between the any of the materials in the model or make a new one at any time. This works whether you are assigning the material by layer or object.



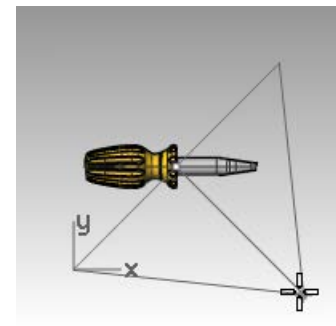
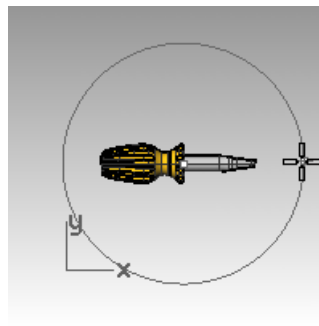
Add lights

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

To place a light:

- 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.

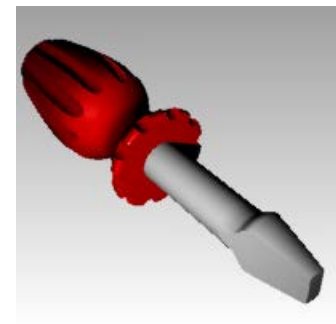
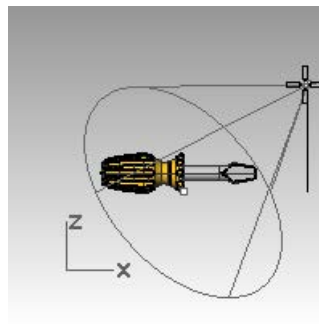


- 7 For the **End of cone**, pick above the object in the Front viewport.

This will be your main light.

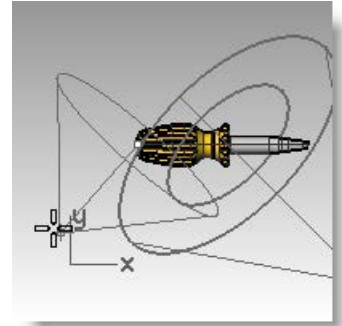
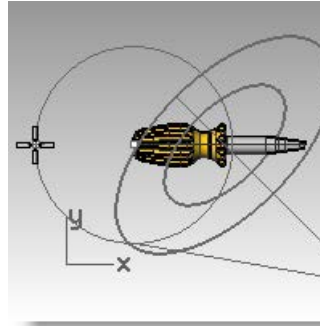
- 8 Click in the Perspective viewport.
- 9 From the **Render** menu click **Render**.

The image has some highlights and shadows.



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.

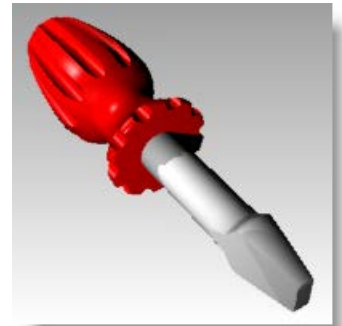
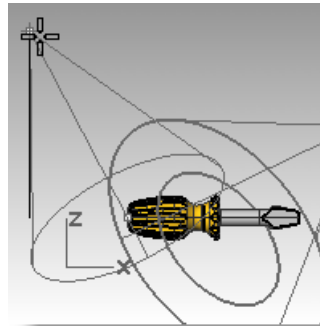


This starts elevator mode.

- 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**.

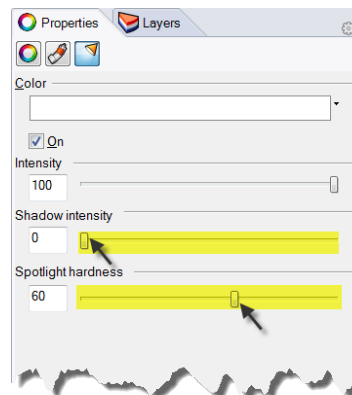
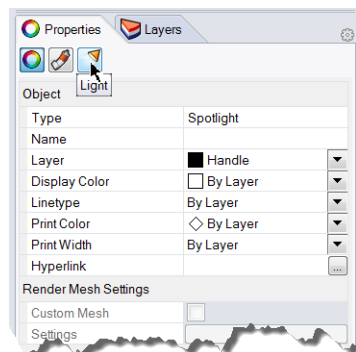


To assign properties to the light:

- 1 Select the new light.
- 2 In the **Properties** panel, click the **Light** page.
- 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.



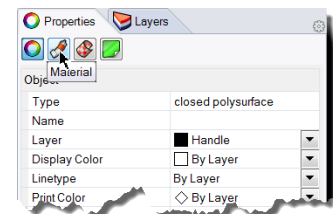
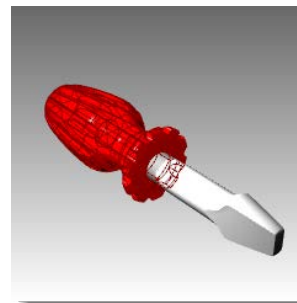
- 5 From the **Render** menu click **Render**.



Add textures

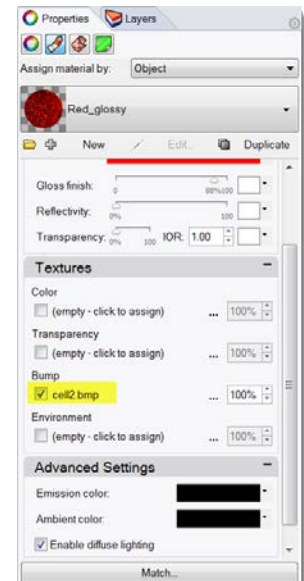
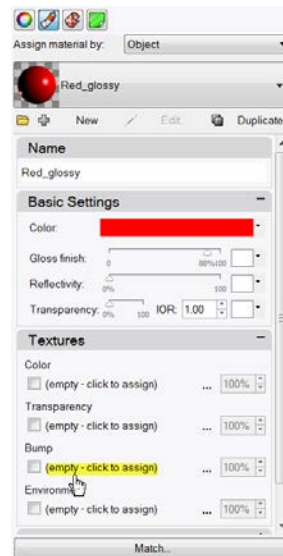
To add a bumpy surface to the handle:

- 1 Select the handle.
- 2 In the **Properties** panel, click on the **Material** page.

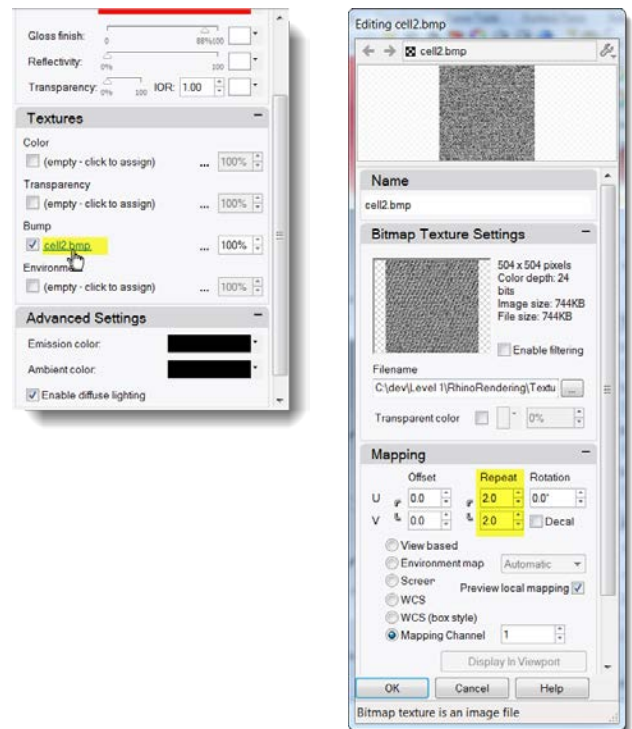


- 3 In the **Material Editor** page, under **Bump**, click (empty-click to assign).
- 4 In the **Open** dialog box, select **cell2.bmp**, and then click **Open**.

Note: You can use any bitmap file for a bump. The bumps come from the pattern of light and dark in the bitmap image.



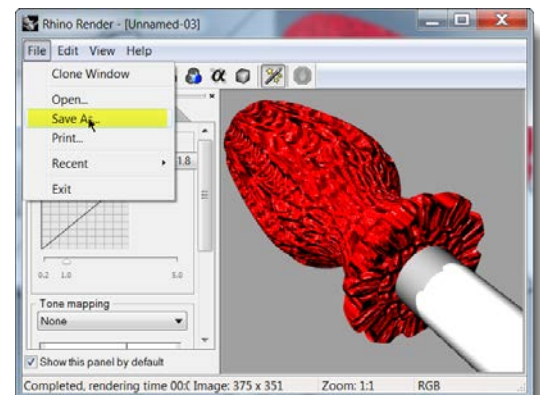
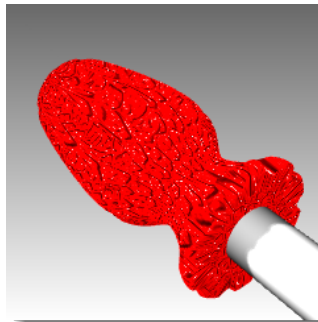
- 5 In the **Material Editor** dialog box, under **Bump**, click **cel2.bmp**.
- 6 In the **Mapping** area, change the **U Repeat** to **2.0** and the **V Repeat** to **2.0**, click **OK**.



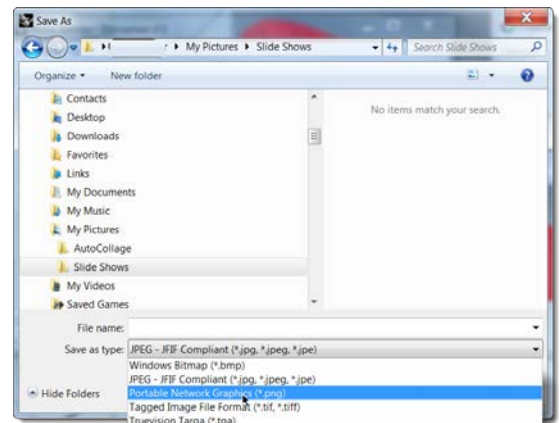
- 7 The Render display will update to display the bump.

The surface of the handle has a bumpy appearance, however the color of the material and gloss setting is still used.

- 8 From the **Render** menu click **Render**.
- 8 From the **File** menu in the **Render** dialog, click **Save As**.

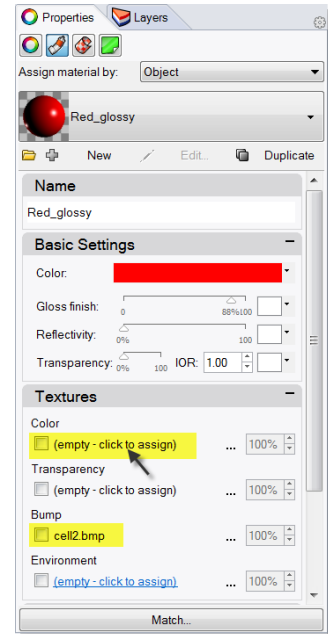
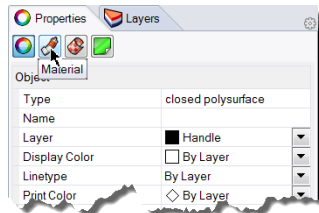
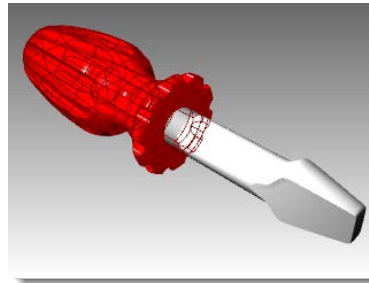


- 9 Under **Save as type**, click **PNG**. enter your file name and location.
- 10 Click **Save** button.



To add a texture to the handle:

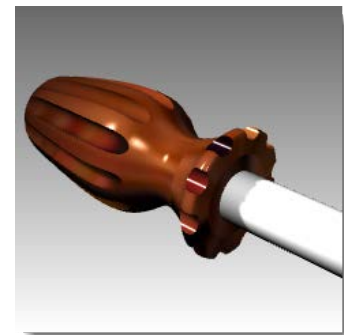
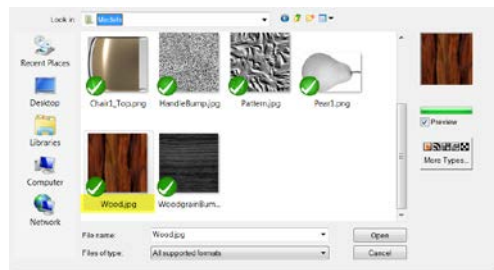
- 1 Highlight the handle. In the **Properties** panel, click on the **Material** page.
- 2 In the **Textures** area, uncheck **Bump**.
- 3 In the **Textures** area, for **Color** click (empty click to assign).



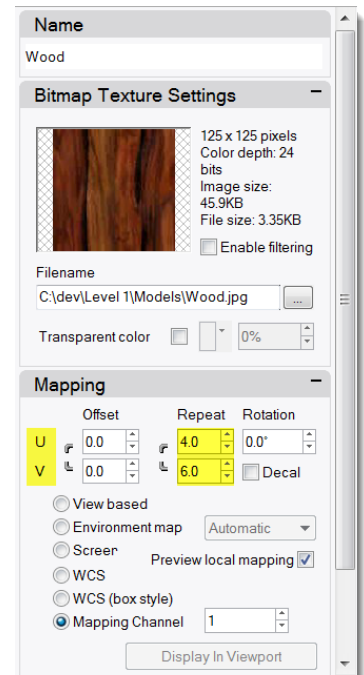
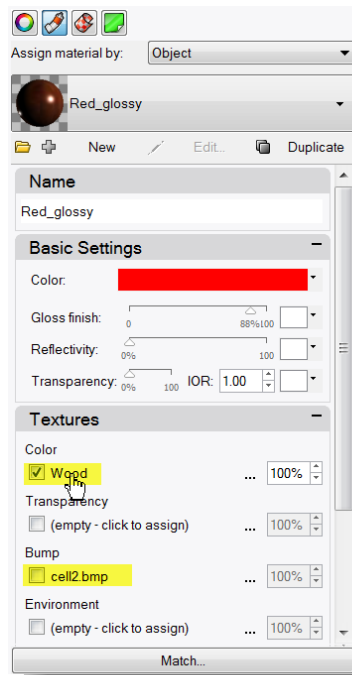
Uncheck Bump and click to assign Color texture.

- 4 In the **Open Bitmap** dialog box, select **Wood.jpg**, and then click **Open**.

Wood color texture is mapped to handle.



- 5 Click **Wood** to open the **Editing Wood** dialog.
- 6 In **Mapping** area, set the **U Repeat** to **4** and the **V Repeat** to **6**.
- 7 Click **OK** to close the **Editing Wood** dialog.



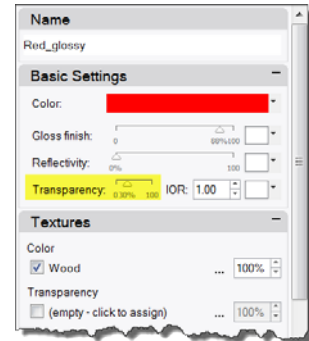
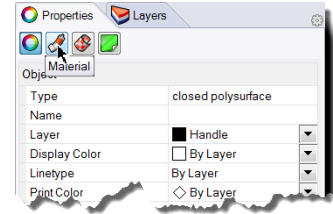
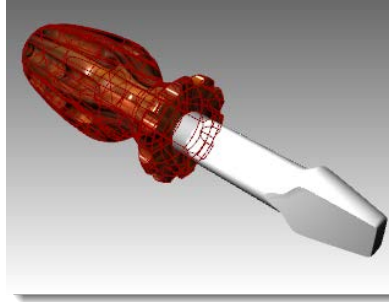
- 8 From the **Render** menu click **Render** or use a Rendered viewport.

The surface of the handle has a wood texture appearance tiled at 4 times in the U and 6 times in the V.



To make the wood handle transparent:

- 1 Highlight the handle. In the **Properties** panel, click on the **Material** page.
- 2 In the **Basic Settings** area of the **Material Editor** dialog box, change the **Transparency** slider to **30**.

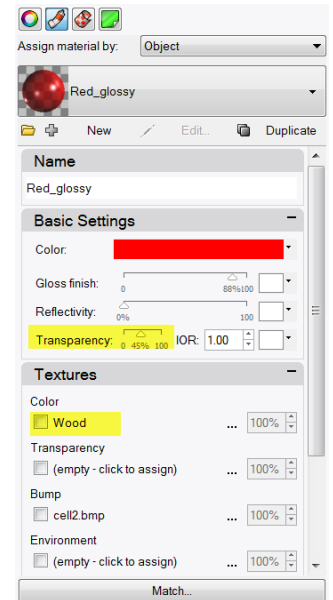
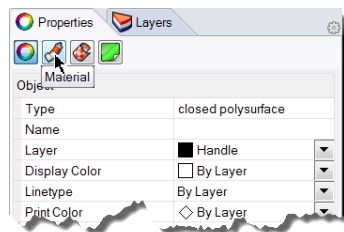
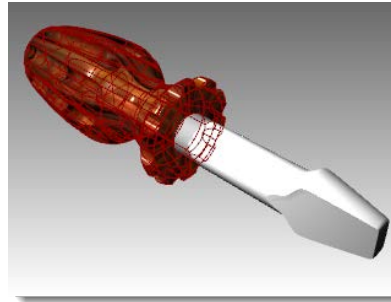


- 3 Click back into the graphics area.
- 4 From the **Render** menu click **Render**.
The handle will look transparent with the Wood texture.

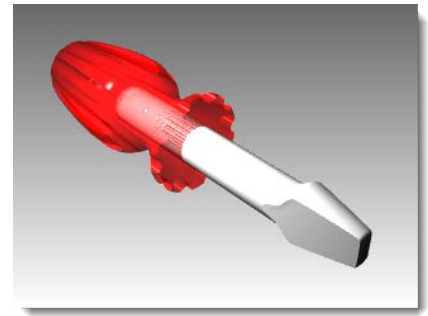


To make the red handle transparent:

- 1 Highlight the handle. In the **Properties** panel, click on the **Material** page.
- 2 In the **Basic Settings** area of the **Material Editor** dialog box, change the **Transparency** slider to **30** and uncheck the **Color** texture.



- 3 Click back into the graphics area.
- 4 From the **Render** menu click **Render**.
The handle will look transparent with the red glossy material.

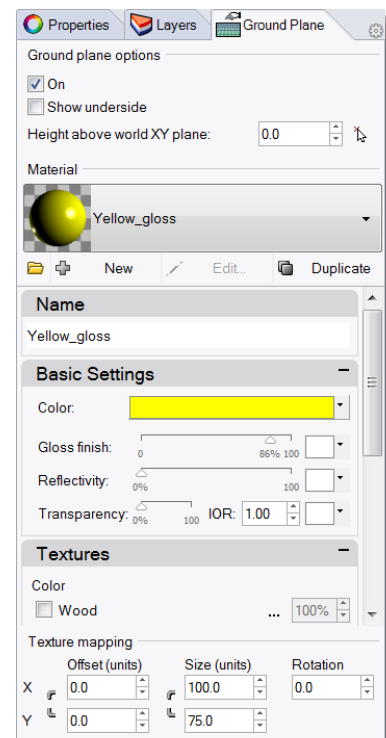
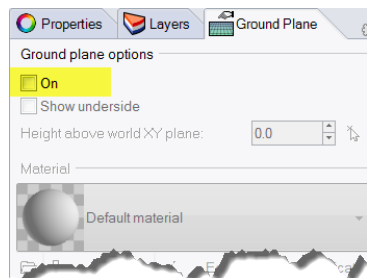
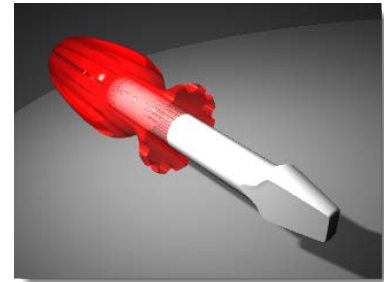
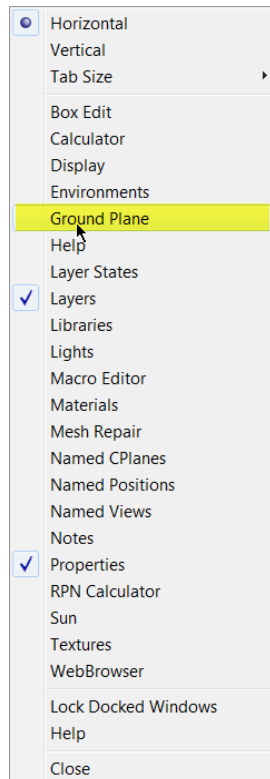


Use a groundplane

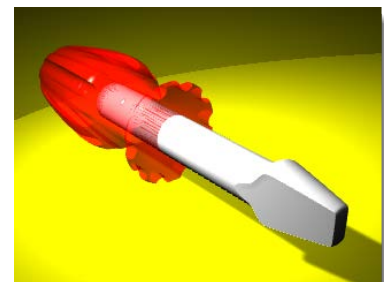
The Rhino Render has a ground plane option. The ground plane provides an infinite horizontal platform for the image that stretches to the horizon in all directions positioned at a defined elevation. A ground plane renders much faster than using a surface as a background. Any material can be assigned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

Adding a ground plane:

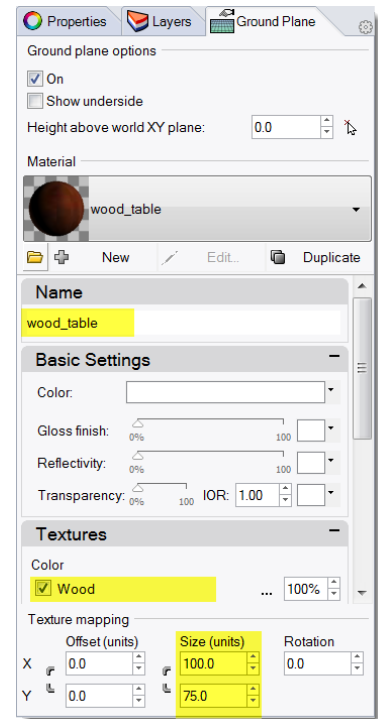
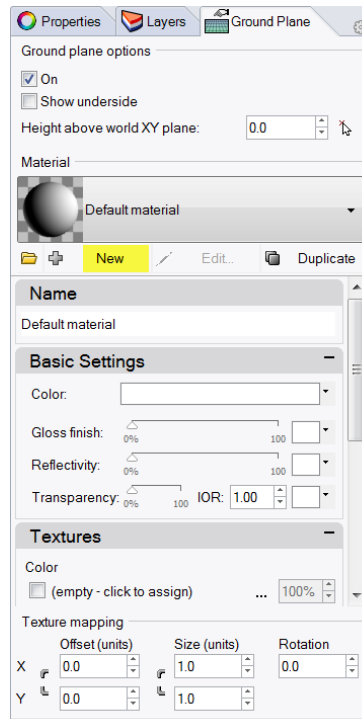
- 1 Right-click Properties panel tab.
- 2 Click the **Ground Plane** panel.
- 3 In **Ground Plane** panel pick **On**.
Viewport will now show a ground plane. Let's assign a material.
- 4 In the **Material** area assign the existing **yellow_glossy**, a material created earlier in this exercise.
Viewport will now show a yellow glossy ground plane.



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



- 6 In **Ground Plane** panel, click the down arrow next to the **Yellow_Glossy** material.
- 7 When the list of materials appears, click **Default** material.
A new default material will be added.
- 8 In the **Material editor** dialog in the **Name** area type **Wood_table**.
- 9 In the **Textures** area, check **Color** texture.
- 10 In the **Open Bitmap** dialog box, select **Wood.jpg**, and then click **Open**.
- 11 In the **Texture mapping** area, type **X** size **100** and **Y** size **75**.



- 12 From the **Render** menu click **Render**.
- 13 From the **File** menu in the **Render** dialog, click **Save As**.
- 14 Under **Save as type**, click **PNG**. enter your file name and location.
- 15 Click **Save** button.

The viewport will now show and render a wood ground plane.



12 Annotating your model

Rhino can also produce 2D drawings of your models. Rhino has these annotation objects:

- Dimensions
- Annotation Text
- Leaders
- Dots
- Hatching

Dimensions

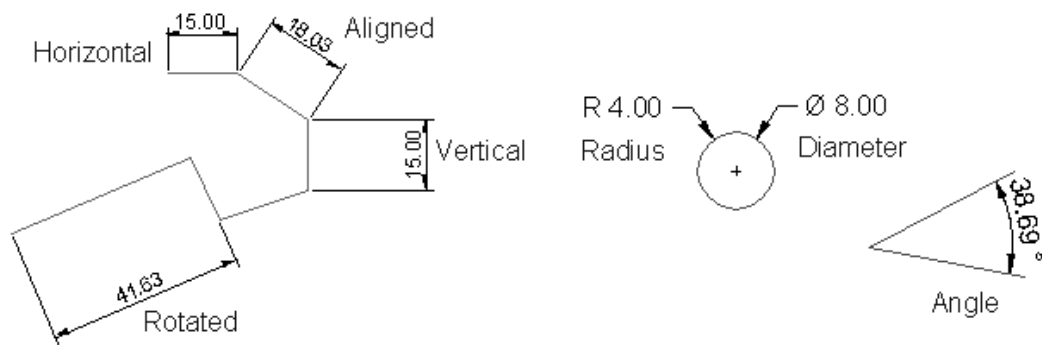
You can create dimensions on all viewports. They will be created parallel to the current viewport's Cplane. Dimension commands are combined with object snaps to give precise values. There are many different types of dimensions that are used to annotate a model. We will look at linear, radial, diameter, and angular. 2-D annotation text, Leaders, and Dots will also be covered.

The dimension style controls the way a dimension displays. For example the dimension text location can be above the dimension line or in the dimension line. The end of a dimension line can display an arrow, a tick, or a dot. The dimension text can show decimal, fractional, or feet and inches numbers. A new model will open with Default as the dimension style.

You can make additional dimension styles, assign existing dimensions to a different style, or update a style to make all the dimensions assigned to that style update. You can also import a dimension style from another model. Or add the dimension style to a template so new models always have it.




Let's look at the dimension types:

Dimension types



Dimension tools

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.
	DimDiameter	Creates a diameter dimension.
	Text	Creates 2-D annotation text.
	Leader	Draw an arrow leader.

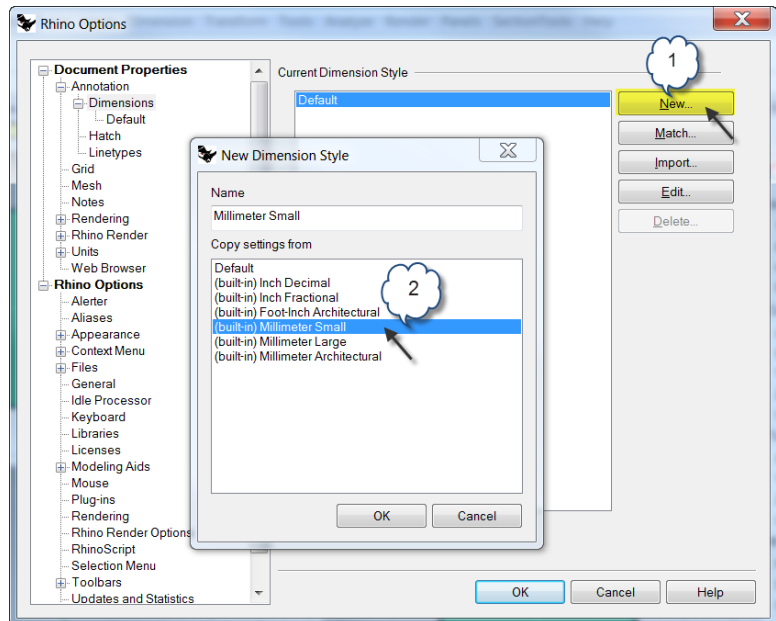
Button	Command	Description
	Properties	Edit dimensions and text.
	DimRecenterText	Returns text that has been moved away from its default position to its original location.
	Make2-D	Creates curves from the selected objects as silhouettes relative to the active construction plane. The silhouette curves are projected flat and then placed on the world x,y-plane.

Exercise 70—Dimensioning the model

To start this exercise we will create a new dimension style.

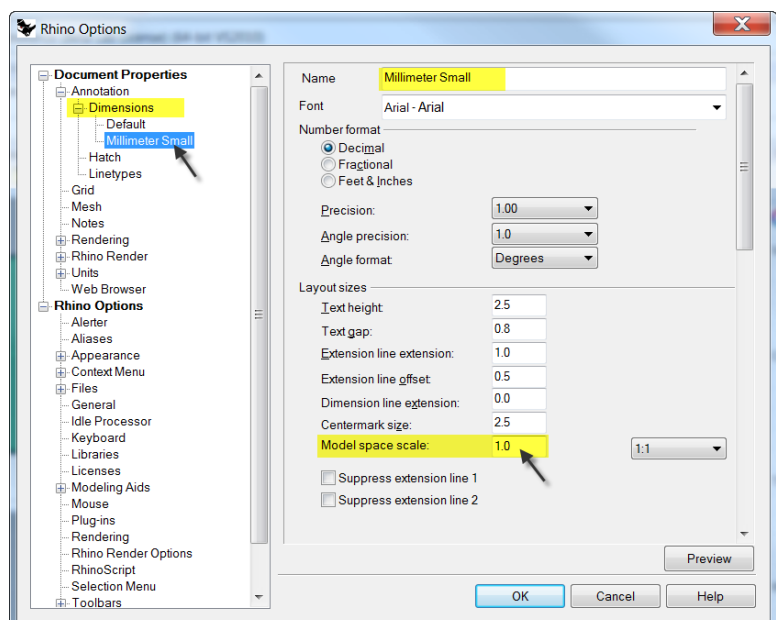
To make a new dimension style:

- 1 Open the model **Dimension.3dm**.
- 2 From the **Tools** menu click **Options**.
- 3 In the **Rhino Options** dialog, click the + next to **Annotations**, click the + next to **Dimensions** to expand the list.
- 4 Click **Dimensions**, and on the **Current Dimension Style** page, click **New**.
- 5 Select **(built-in) Millimeter Small** as the new dimension style template.



- 6 Click the **Edit** button
- 7 Change **Model space scale** to **1.0**. Click **OK**.

Dimensions created will be assigned to the Millimeters Small dimension style.



Linear Dimensions

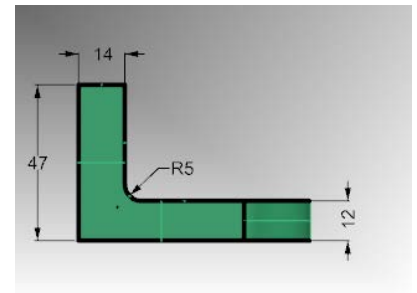
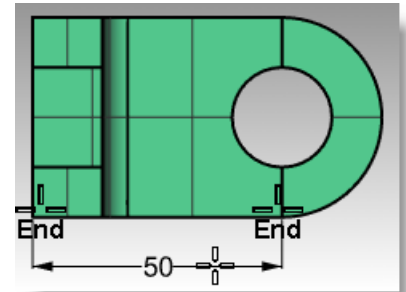
Linear dimension creates a horizontal or a vertical dimension.



To make linear dimensions:

- 1 From the **Dimension** menu click **Linear Dimension**.
- 2 Snap to the lower left end of the part in Top viewport.
- 3 Snap to the lower right end of the part in the Top viewport.
- 4 Pick a point below the part in the Top viewport.
- 5 From the **Dimension** menu, click **Linear Dimension**.
- 6 Snap to the lower left end of the part in Front viewport.
- 7 Snap to the upper left end of the part in Front viewport.
- 8 Pick a point to the left of the part in the Front Viewport.
- 9 Create two additional linear dimensions on the top and right sides of the part in the Front viewport.

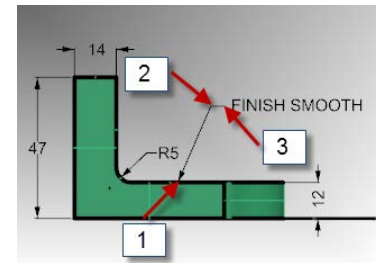
Note: Use object snaps to locate the extension line origins. To move extension line or text locations for dimensions, turn on control points for the dimension and move the control points.



Leaders

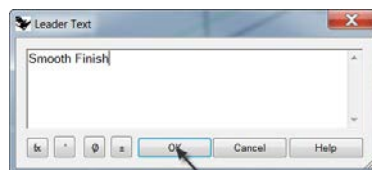
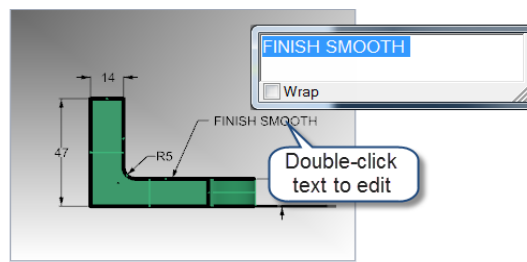
To draw an arrow leader and text:

- 1 From the **Dimension** menu click **Leader**.
- 2 Click the top of the part in the Front viewport. This will be where the arrow will be created
- 3 Turn Grid Snap on and click a point to the right and at an angle from the first point.
- 4 Click to the right of the second point and press Enter.

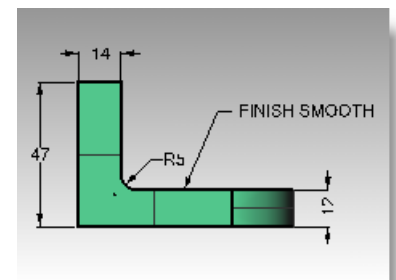


- 5 In the **Leader Text** dialog box, type **Finish Smooth** and click **OK**.

- 6 To edit the text, double-click and make changes in text box. Click in graphics area when editing is complete.



For long notes try **Wrap**. **Wrap** activates the word-wrap feature on the leader object.



Radial and Diameter Dimensions

To make a radius or diameter dimension:

- 1 From the **Dimension** menu click

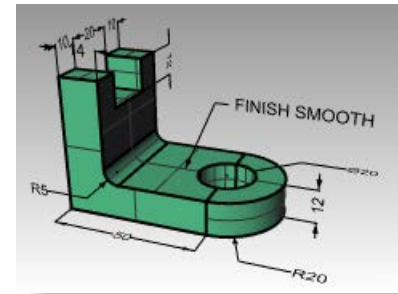
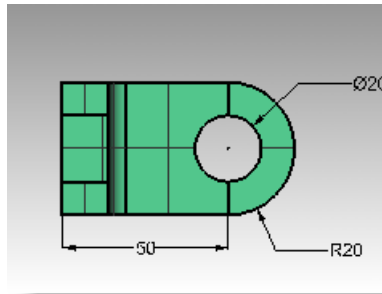
Radial Dimension. 

- 2 Select the lower right quadrant of the arc in the Top viewport.

- 3 Click to place the dimension text.

- 4 From the **Dimension** menu click

Diameter Dimension. 



- 5 Select the upper right quadrant of the hole in the Top viewport.

- 6 Click to place the dimension text.

Continuous Linear Dimensions

The Dim command has a Continue option, that when activated adds chained dimensions along the same dimension line. This option needs to be activated for each Dim command that you begin.

To make continuous (chained) dimensions:

- 1 From the **Dimension** menu click **Linear Dimension.** 

- 2 In command line, click **Continue=No**. It will toggle to **Continue=Yes**.

- 3 Snap to the left end of the part in Top viewport (1).

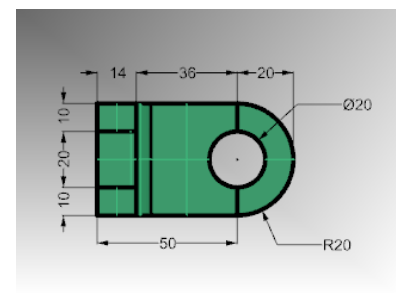
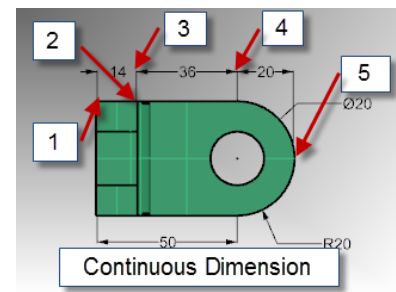
- 4 Snap to the end of the vertical part in the Top viewport (2).

- 5 Pick a point above the part in the Top Viewport (3).

- 6 Continue to snap to the center of the hole (4), and the right end of the object (5).

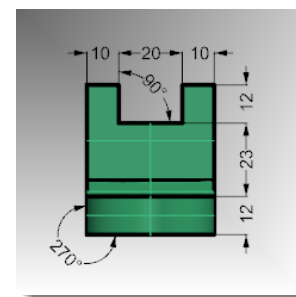
- 7 Press **Enter** to end continuous linear dimension.

- 8 Create one additional linear dimension on the right side of the part in the Top viewport and the top of the part in Right Viewport.



- 9 Dimension the rest of the drawing using leaders, text blocks, horizontal, vertical, radius, and diameter dimensions.

- 10 **Save** your model.




Making a 2-D drawing from a 3-D model

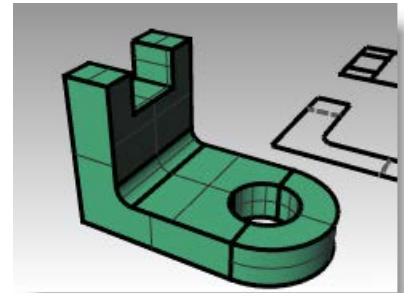
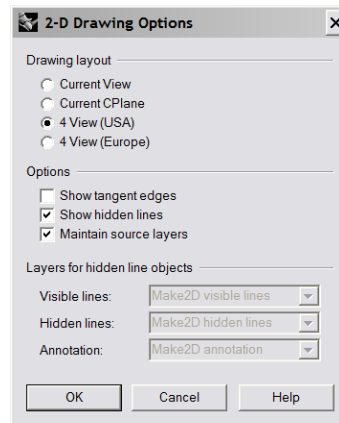
Rhino has the ability to generate a two-dimensional drawing from a three-dimensional model, by projecting the geometry to the world coordinate plane, and aligning the views. Options for first angle projection or third angle projection are available. In addition to the three orthographic views, a two-dimensional perspective drawing is also generated. Hidden lines are removed and placed on a separate layer.

Options for creating four views (three parallel viewports and a perspective viewport), or single views of individual viewports are supported.

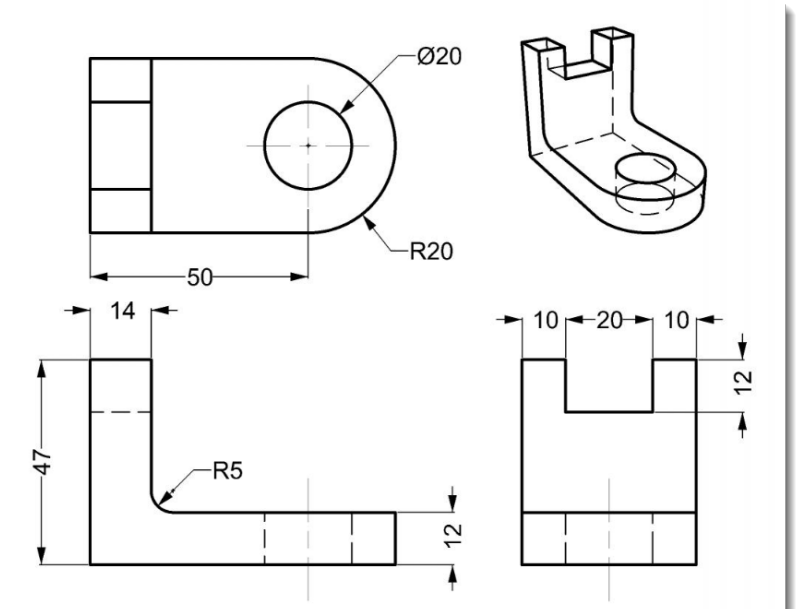
Exercise 71—Practice making a 2-D drawing

- 1 **Open** the model **Make2D.3dm**.
- 2 Select the 3D model.
- 3 From the **Dimension** menu click **Make 2-D Drawing**. 
- 4 In the **2-D Drawing Options** dialog box, click **4-view (USA)** and check **Show hidden lines**, and then click **OK**.

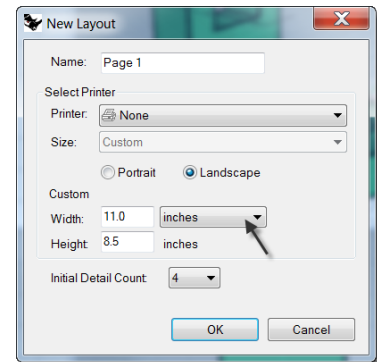
The 2-D drawings are created on the Top CPlane near the origin on the world xy plane. View them in the Top viewport.



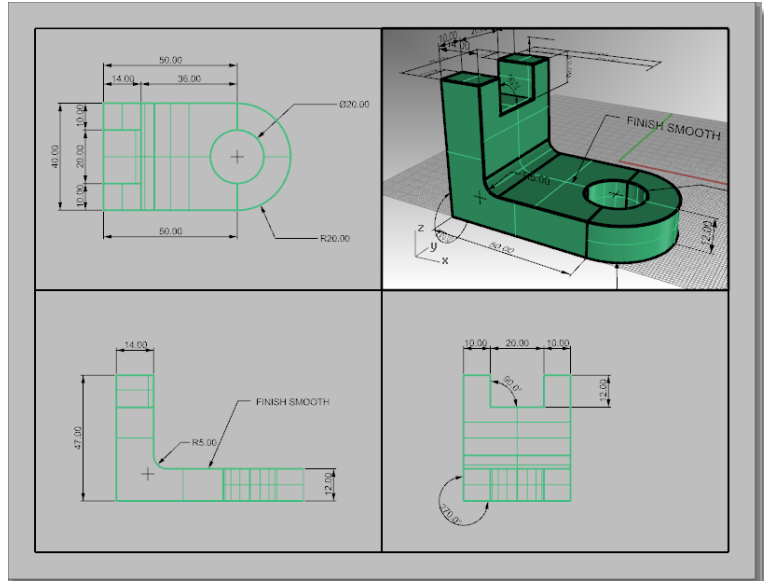
- 5 **Dimension** the 2-D drawing.



- 3 From the **New Layout** dialog, select **Inches**, set **Width** to **11** and **Height** to **8.5**. Set the **Initial Detail Count** to **4**.



- 4 **Double-click** to activate **Perspective** detail. From the **View** menu, pick the **Shaded** display mode.

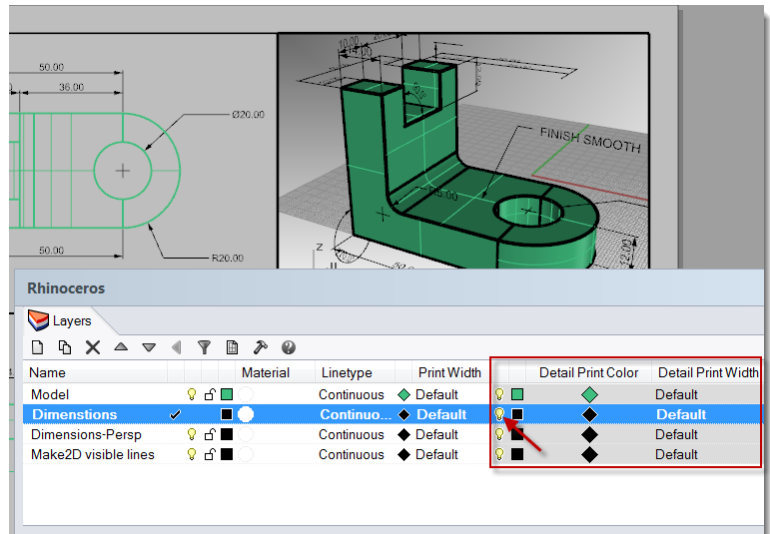


To configure the perspective detail:

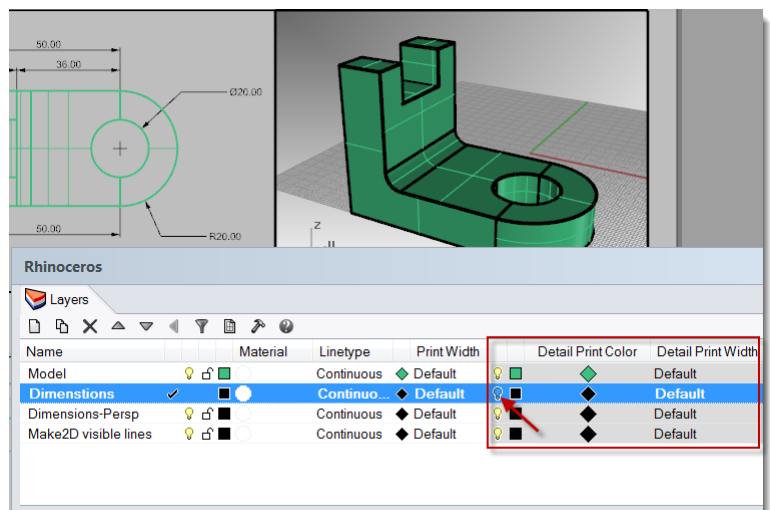
You can suppress the display of geometry in any detail by hiding the object in the detail or by turning off the layer in the details. Objects are hidden in the details with the HideObjectInDetails command and made visible with the ShowObjectsInDetail command

In addition, the Layers panel allows a layer to be turned off only in the detail, but remain visible in other details or viewports.

- 1 **Double-click** to activate **Perspective detail**. In the **Layers** panel, scroll to the right or undock and stretch Layers panel.
- 2 Highlight the **Dimension** layer, and pick on the lightbulb icon in the **Detail On** column.

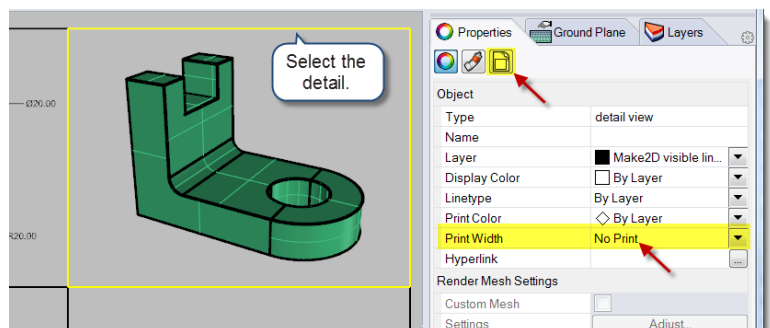


All the objects on the layer that was turned off in the Detail On column are not visible in the Perspective view but remain visible in all other details.



- 3 **Double-click** the **Perspective detail** to deactivate. Highlight the boundary edge of the detail. In **Object Properties** set the **Print Width** to **No Print**

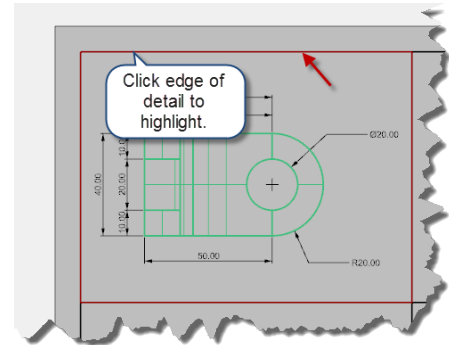
The edge of the detail will not print unless you assign a width.



To assign a scale to details:

Details that are parallel can get a scale assigned to them. The scale will tell Rhino how many model units are in each paper unit. By assigning the scale to the detail, the layout can be plotted 1=1. Also by assigning the scale to the detail, details can have different scales.

- 1 Highlight the Top detail. Do not double-click to activate.
- 2 In the **Properties** panel, click the **Detail** page.

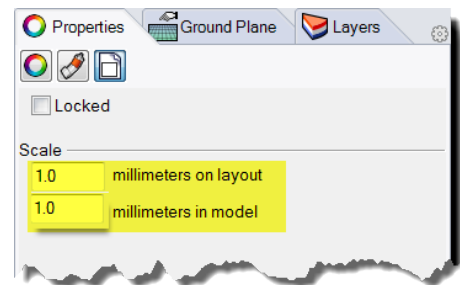


- 3 In the **Scale** area, set **1.0 mm** on layout equal to **1 mm** in model.

The scale is now set to 1=1.

If you set 1 mm in the layout equal to 2 mm in the model detail, it is half size or 1=2.

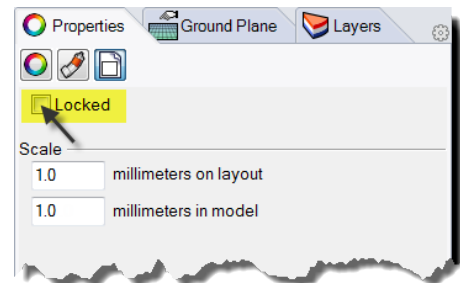
If you set 1 mm in the layout equal to 10 mm in the model detail, the scale is 1=10.



- 4 Make detail active by double clicking it and pan geometry to center in viewport. Deactivate detail by double clicking.
- 5 Select detail and in the **Properties** panel, click the **Detail** page and then click **Locked**.

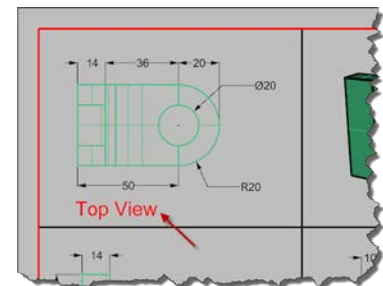
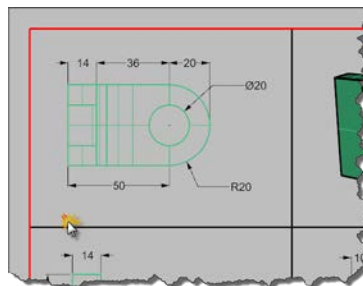
Details that are locked prevent zooming or panning, making it easy to keep them from changing scale.

- 6 Repeat this procedure for the Front and Right details.

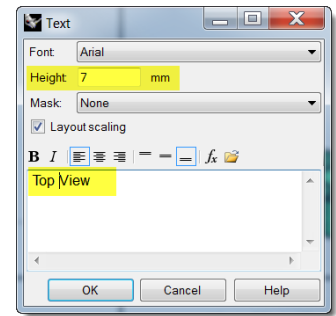


To label details:

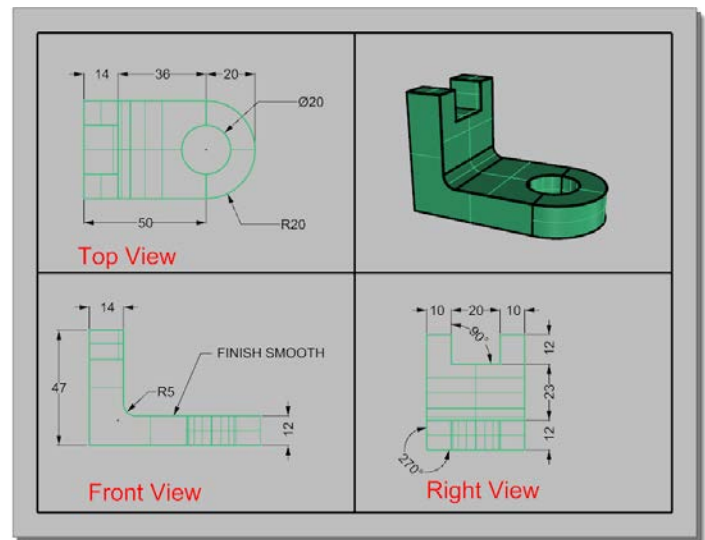
- 1 Make a new layer called Notes. Make it Red and current.
- 2 Turn Ortho off and deactivate any details. You should be on the layout and not in an active detail.
- 3 From the **Dimension** menu click **Text Block**.
- 4 Pick a **Start point**, below the Top view of the part.



- 5 In the **Text** dialog set the **Height** to **7mm** and type **Top View**.

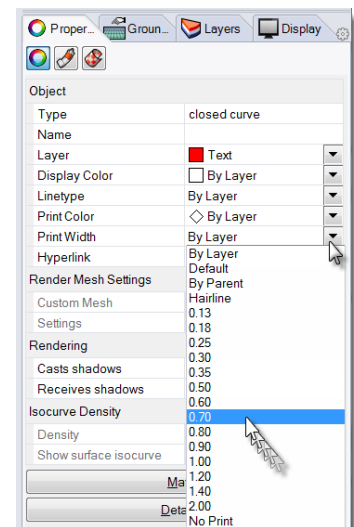
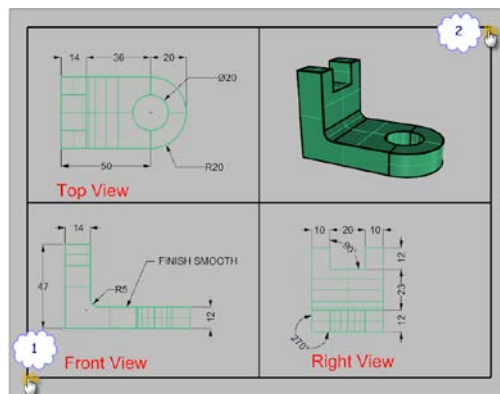


- 6 Repeat to add labels for Front View and Right View.



To add a border:

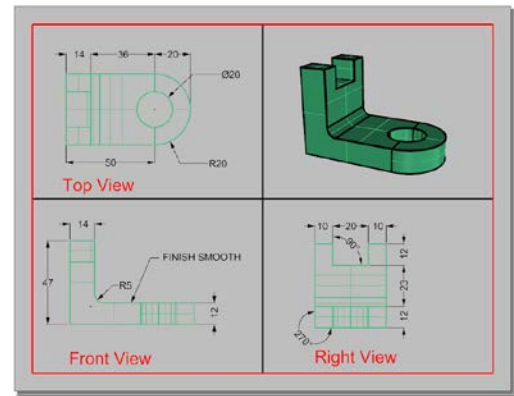
- 1 From the **Curve** menu click **Rectangle, Corner to Corner**. Snap to the lower left corner of the Front detail boundary and the upper right corner of the Perspective viewport boundary.
- 2 Select the border. In the **Properties** panel, on the **Object** page, click **Print Width**, and then click **0.70mm**.



- 3 Right click on the **Layout Title (Page 1)**, and then click **Print Preview**.

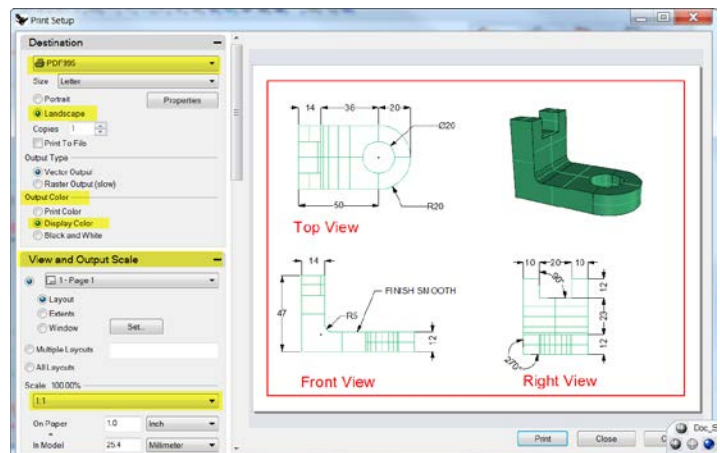
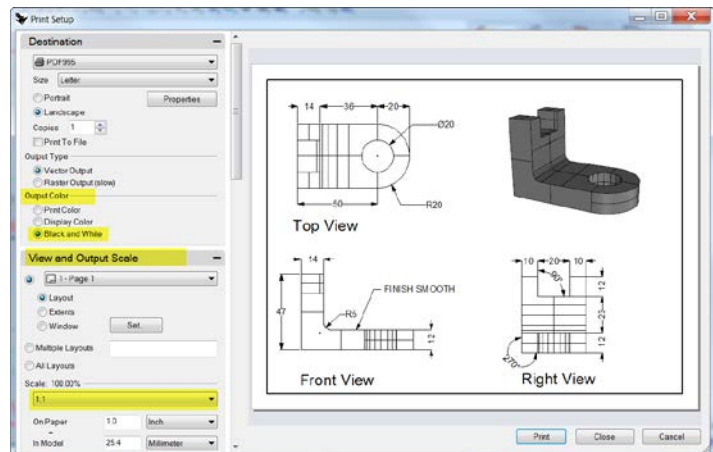
The layout has a thick border around the detailed viewports.

You can also add a title block at this stage if you choose.



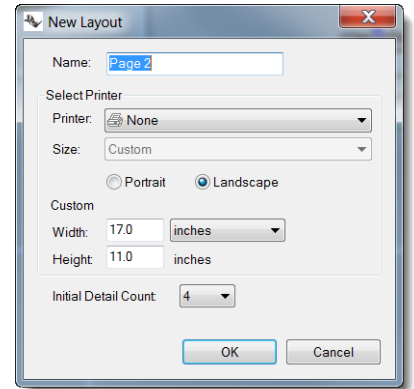
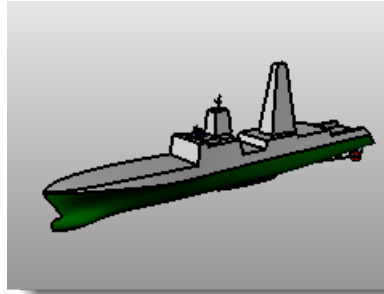
To print the layout:

- 1 From the **File** menu click **Print**.
- 2 Select a physical printer or a virtual printer like **PDF995**. Select sheet **size** as **letter** or **11x8 1/2** landscape.
- 3 In the **View** and **Output Scale** area, set the scale to **1=1**.
- 4 Toggle Output color between **Black & White** and **Display Color**.
- 5 Pick **OK** to print. Cancel if no printer is available.
- 6 **Save** the file.



Exercise 73—Scaling and locking details on a layout.

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

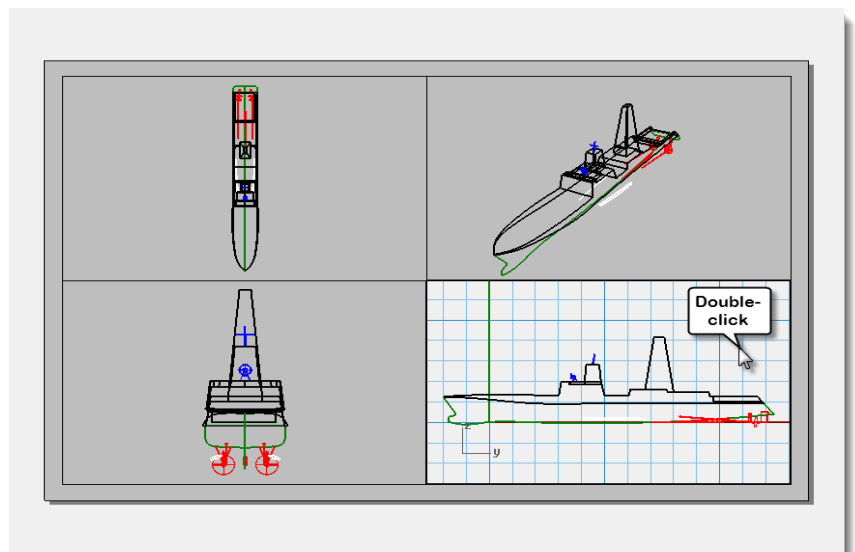


A layout page opens and a new viewport tab labeled Page 1 appears at the lower edge of the Rhino window.

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

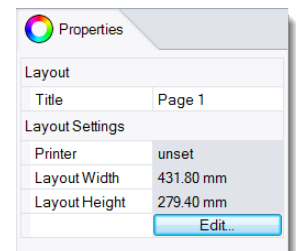
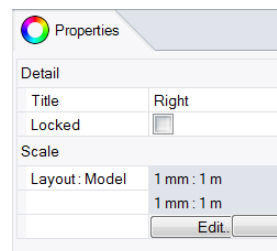
Details are windows into the 3-D model. They can be activated as modeling viewports by double-clicking in the viewport.

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

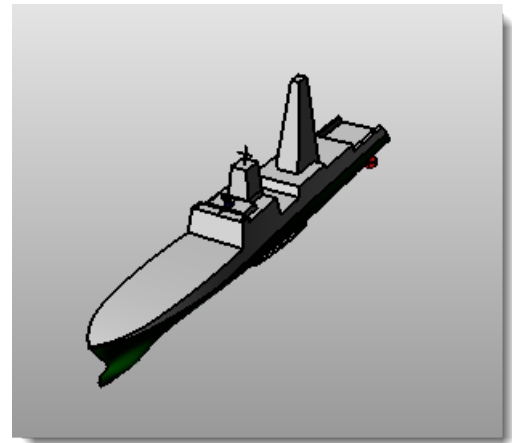
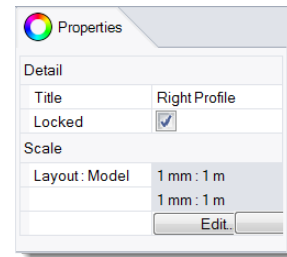


Set scale and lock details:

- 1 If the **Properties** panel is not open, open it now (*Edit>Object Properties*) and dock it to one side.
 With nothing selected, the properties panel shows the viewport properties.
 With no detail active, the properties shown are for the layout as a whole.

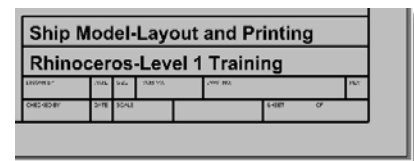
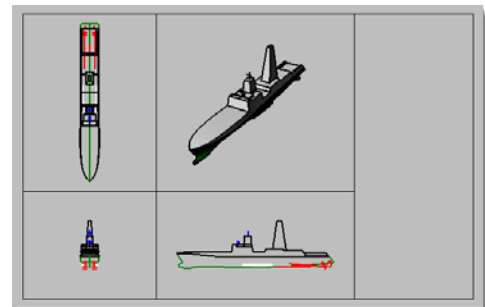


- 2 Click the **Edit** button to edit the title, and size and printer properties.
- 3 On the command line, for the **Distance on layout (mm)**, set it to **1** and press **Enter**.
- 4 On the command line, for the **1.000 millimeter on layout = Distance in model (m)**, set it to **1** and press **Enter**.
- 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**.



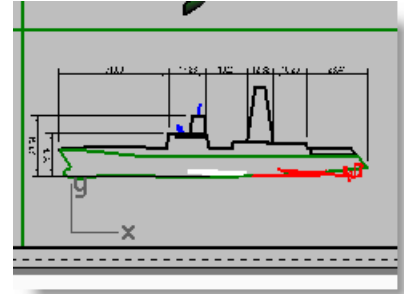
To draw borders and title blocks in the layout space:

- 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 box, check **Prompt for Insertion point**, uncheck **Prompt for Scale** and **Rotation**.
- 5 On the **Insert** dialog box, click **File...**, select **TitleBlock.3dm**, click **Open**, click **OK**, click **OK** to close the **Insert** dialog box and insert the title block.
- 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.



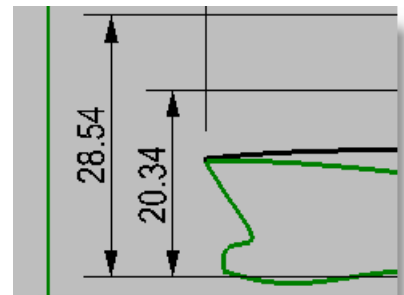
To set linetypes and line weights for curves:

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 panel**, 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 **Properties** panel, change the **Print Width** to a thicker weight.
- 5 Right click on the **Layout Title (Page 1)**, and then click **Print Preview**.

Note the difference in thickness of the curves.

Linetypes can be adjusted the same way.



PART FOUR

Extras

14 Transforming solids

It's much easier to model things on a flat plane than it is on an organic surface or any other 3D shape. Rhino has several tools that allow you to model something in a simple way and then transform it onto a surface or a curve in 3D space. In this chapter we will demonstrate two commands that make this easy to accomplish—Flow and FlowAlongSrf.

Flow along surface

Morphs objects from a source surface to a target surface.

Options	Description
Copy=Yes/No	Specifies whether or not the objects that you are flowing are copied. A plus sign appears at the cursor when copy mode is on.
Rigid=Yes/No	Note: This option does not apply to polysurfaces and will not be displayed if polysurfaces are selected for editing. Yes Individual objects are not deformed as they are transformed. No Individual selected objects are deformed
Plane	Allows you to draw a plane instead of using an existing surface as the base object.

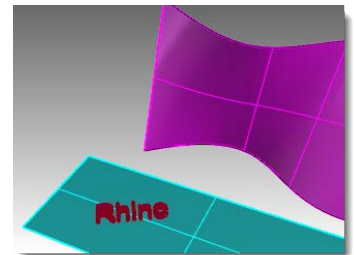
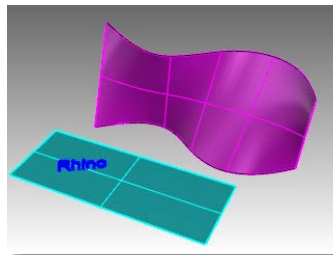
Exercise 74—Flowing Solids to a Surface

Modeling on a curved surface is difficult and hard to achieve accurate results. FlowAlongSrf command simplifies the process by allowing you to model on the flat construction plan first. Then FlowAlongSrf to morphs objects from a source surface to follow target surface.

To Flow Solid Text

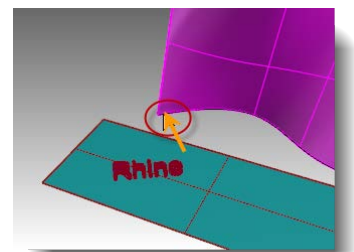
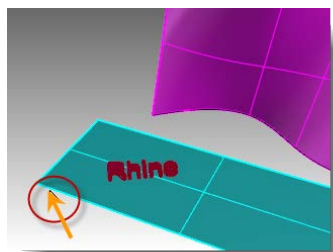
- 1 **Open** model FlowAlongSrf.3dm
- 2 From the **Transform** menu click **Flow Along Surface**.
- 3 For the **Select objects to flow along a surface**, select the solid text and press **Enter**.

The text is a group and will select as one.

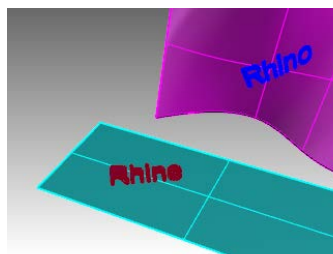


- 4 For the **Base surface**, select the lower left corner of cyan surface.
- 5 For the **Target surface**, select the lower left corner of the magenta target surface.

The text is flowed to the target surface.



- 6 **Undo** the FlowAlongSrf.



Surface direction

The success of the FlowAlongSrf is dependent on the direction of the base and target surface. Each surface has a normal direction, a U direction, and a V direction.

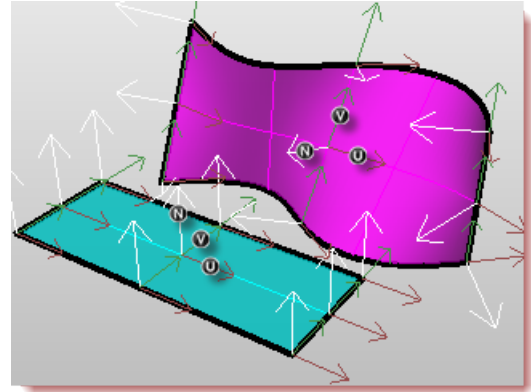
Ideally, the normal, U and V directions on the base surface should correspond to the normal, U and V directions on the target surface.

The direction arrow colors are:

- U=Red
- V=Green
- Normal=White

To check the surface direction:

- 1 Select both the cyan and magenta colored surfaces.
- 2 From the **Analyze** menu click **Direction**.
- 3 At the **Select object to flip direction**, you can click on either surface to change the normal direction, press **Enter** when done.
You can use the NextMode option to toggle through all of the possible changes to U, V, and Normal directions.
- 4 If you only need to change one surface, select it and start the **Dir** command again.
- 5 Make any necessary changes to the direction of the surface so that the U, V, Normal directions match between the two surfaces.



History and Gumball

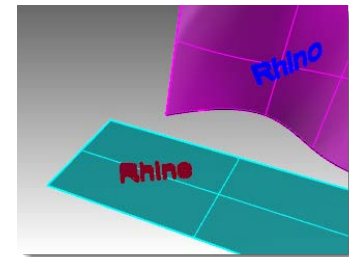
Now let's combine FlowAlongSrf with RecordHistory. RecordHistory keeps the objects to be flowed and the objects that have been flowed linked. If the original objects are moved, scaled, or rotated, the flowed objects will update.

To Flow with History and Gumball:

- 1 In the **Status bar** turn on **Gumball** and **RecordHistory**.

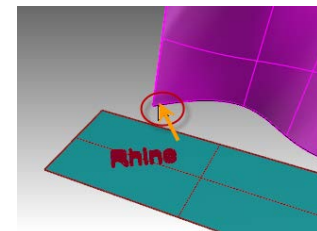
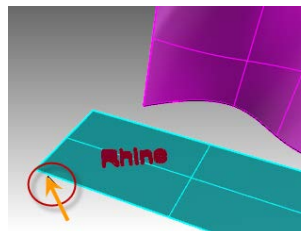


- 2 From the **Transform** menu click **Flow Along Surface**.
- 3 For the **Select objects to flow along a surface**, select the solid text and press **Enter**.



- 4 For the **Base surface**, select the lower left corner of cyan surface.
- 5 For the **Target surface**, select the lower left corner of the magenta target surface.

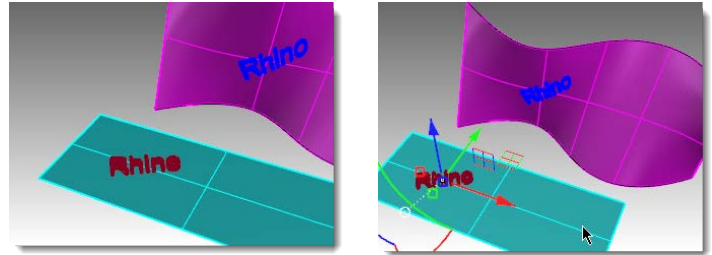
The text is flowed to the target surface.



Note: The **Record History** pane is toggled off as soon as the flow command is complete. By default the **Always Record History** option is not checked. If you want to record history on another command, you will need to select it before you run the next command that supports history.

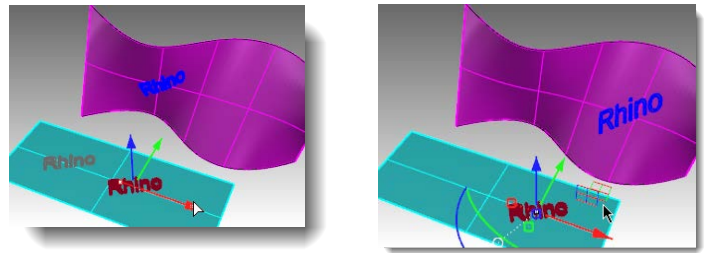
See the **Help** for a complete listing of **History Enabled Commands**.

- 6 Select the original text on the base surface.



- 7 Use the **Gumball** arrow to move the text to the right.

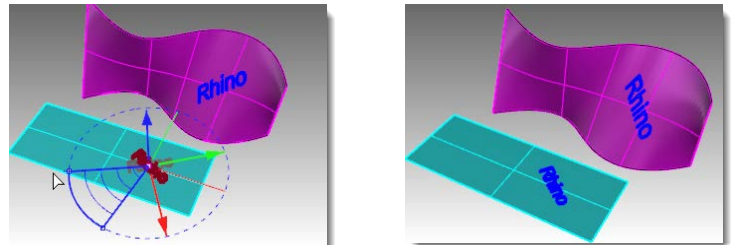
Notice that after you release the text on the base surface, the flowed text is updated on the target surface.



Hint: If the flowed text does not update, return to step 1 and make sure **Record History** is on before starting **FlowAlongSurface**.

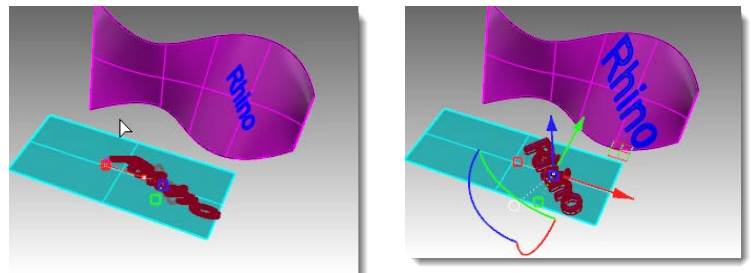
- 8 Select the original text on the base surface.

- 9 Use the **Gumball** arc to rotate the text to the left. Release your mouse button and watch the flowed text update.



- 10 Select the original text on the base surface.

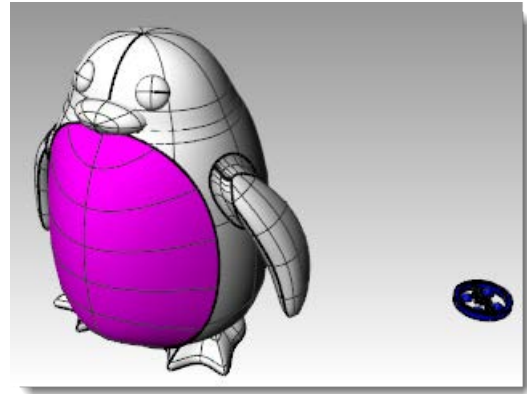
- 11 Using the **Gumball** red scale handle, and the **Shift** key held down, pull to the handle to the left. Release your mouse and watch the flowed text update.




Exercise 75—Flowing a logo onto a freeform surface

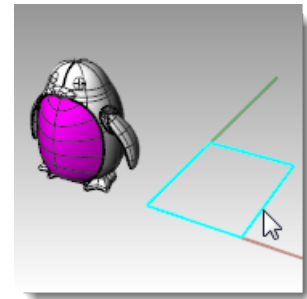
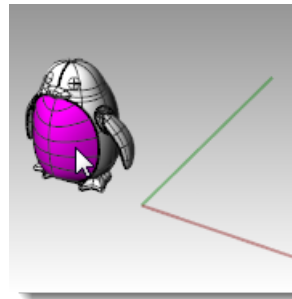
To create the base surface:

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

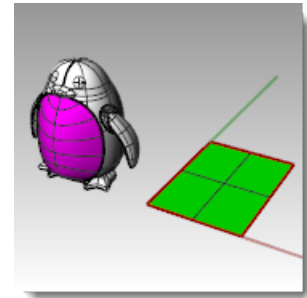
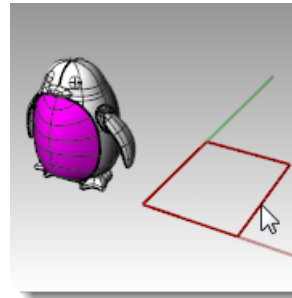


- 2 On the **Layer** panel, make the **Curves** layer current.
- 3 Select the magenta surface (*front panel*) on the penguin.
- 4 From the **Curve** menu click **Curve from Objects**,
then **Create UV Curves**  and press **Enter**.

The UV curves are created at the origin of the World XY plane.



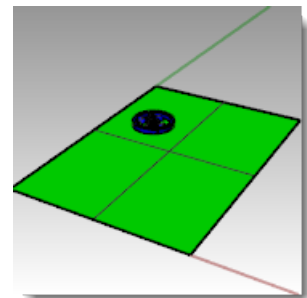
- 5 From the **Layer** panel, make the Surface layer current.
- 6 Highlight the closed curve and from the **Surface** menu click **Planar Curves**.



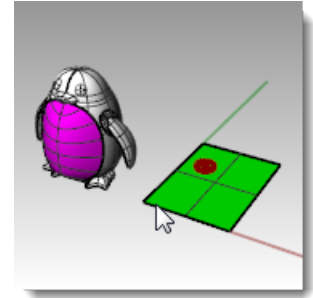
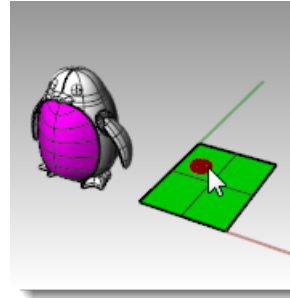
To Flow the Logo:

- 1 Turn on **RecordHistory** in the **Status bar**.
- 2 On the **Layer** panel turn on the logo layer.

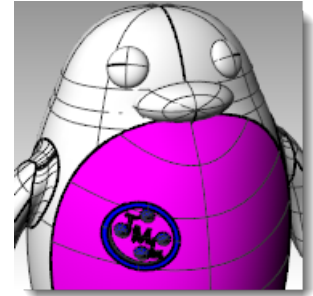
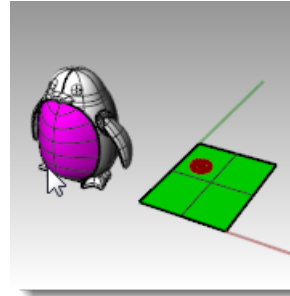
The logo appears.



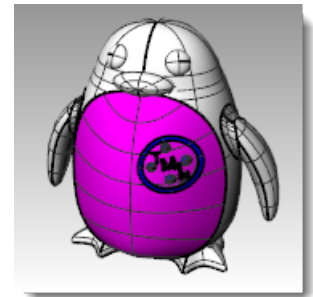
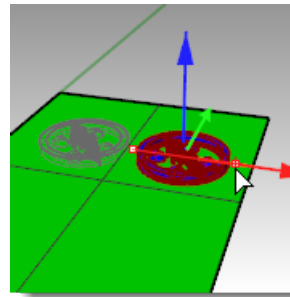
- 3 Select the logo.
- 4 From the **Transform** menu click **Flow Along Surface** and press **Enter**.
- 5 For the **Base surface**, select the lower left corner of green surface.



- 6 For the **Target surface**, select the lower left corner of the magenta surface.



- 7 Turn on **Gumball** in the **Status bar**.
 - 8 Use **Gumball** to **move**, **scale** and **rotate** the original logo on the base surface.
- The flowed logo updates.



- 9 From the **Render** menu, **Render** to render the model.



Flow

The Flow command re-aligns an object or group of objects from a base curve to a target curve.

Steps

- 1 Select objects.
- 2 Select the base curve near one end.
- 3 Select the target curve near the matching end.

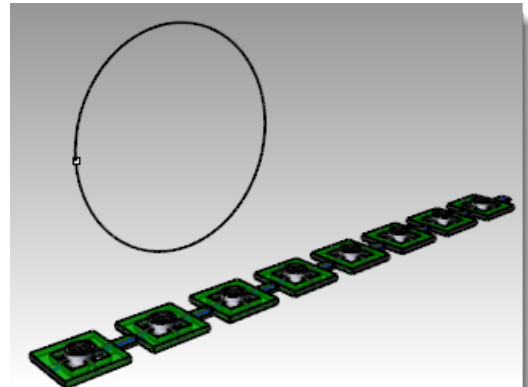
Options	Description
Copy=Yes/No	Specifies whether or not the objects that you are flowing are copied. A plus sign appears at the cursor when copy mode is on.
Rigid=Yes/No	Yes Individual objects are not deformed as they are transformed. No Individual selected objects are deformed
Line	Allows you to draw a line to be used as the base curve.
Stretch=Yes/No	No The length of the objects along the curve directions are not changed. Yes Objects are stretched or compressed in the curve direction so that the relationship to the target curve is the same as it is to the base curve.

Similar to Flow Along Surface, Flow allows you to flow solids along a curve. This makes designing in 3D easier and lets Rhino do all the morphing work. The command is called Flow.

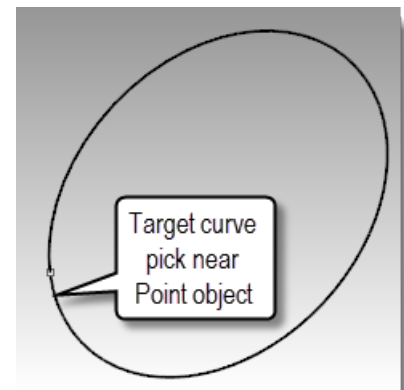
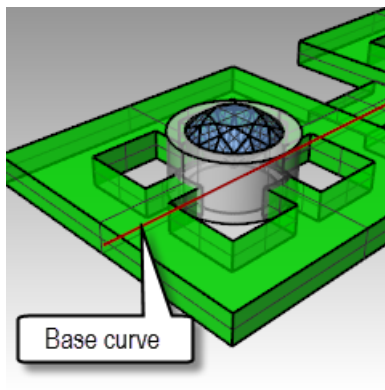
Exercise 76—Making a Ring with Flow

To flow the parts of a ring along the shank curve:

- 1 Open the model **Flow_ring.3dm**.
- 2 Select the green polysurface as the object to flow.
- 3 From the **Transform** menu click **Flow along Curve**.



- 4 Select the red linear curve towards the left end for the **Base curve**.
- 5 Stop at this stage and confirm the following option settings on the command line (**Copy=Yes Rigid=No Stretch=No**).
- 6 Select the circle curve slightly below the point location as the **Target curve**.



The polysurface is morphed or flowed to the shape of the target curve.

Notice that the polysurface doesn't flow completely around the circle.

7 Undo.

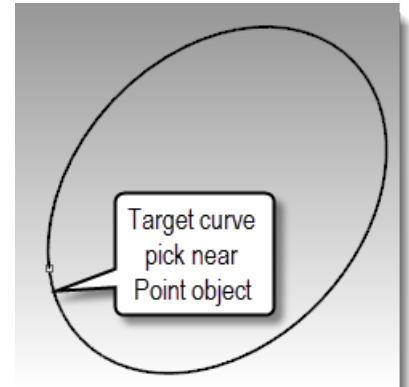
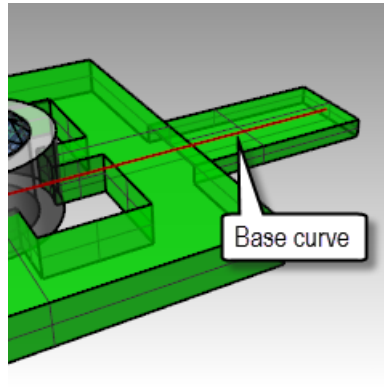


We will flow this polysurface a few more times using different options.

First, we will change the direction of the Flow.

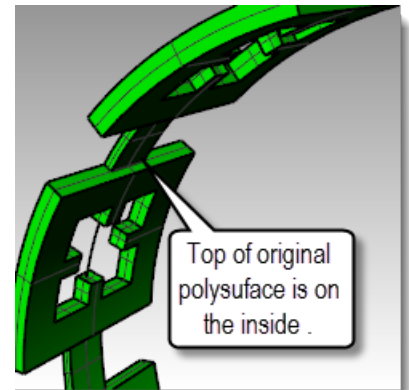
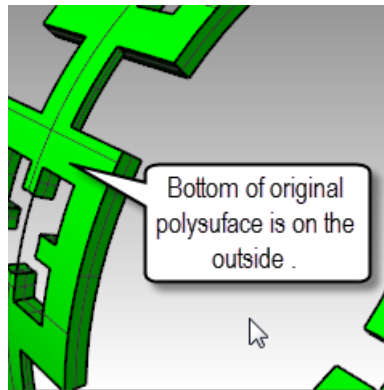
To flow the parts of a ring along the shank curve in a different direction:

- 1 Repeat **Flow along curve** with identical steps, except pick the **Base Curve** near opposite end.
- 2 Select the circle curve slightly below the point location as the **Target curve**.



Notice that the inside and outside of the original polysurface has flipped.

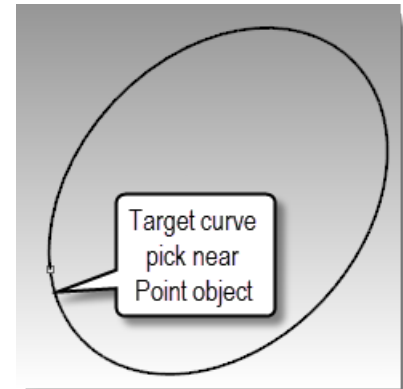
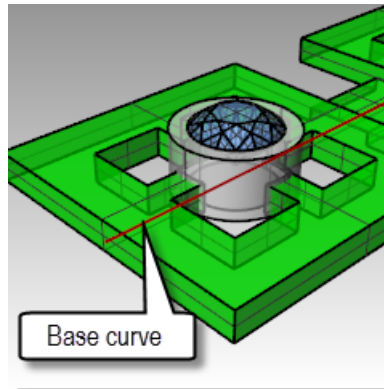
- 3 **Undo** again.



Second we will stretch the original polysurface so that it fits completely around the circle.

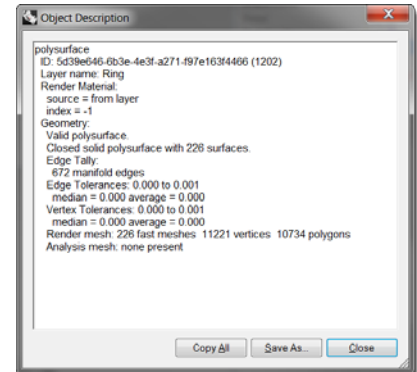
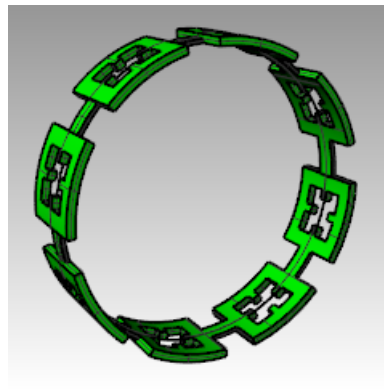
To flow the parts of a ring along the shank curve, stretching it to fit the whole curve:

- 1 Repeat **Flow along curve** the same way you did it the first time, picking the **Base Curve** towards the left end.
- 2 Stop at this stage and confirm the following option settings on the command line (**Copy=Yes Rigid=No Stretch=Yes**).
- 3 Select the circle curve slightly below the point location as the **Target curve**.



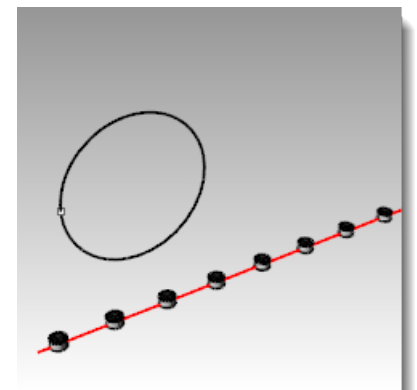
The polysurface is morphed or flowed completely around the circular shape of the target curve.

- 4 Use the **What** command to confirm that it is a closed solid polysurface.



To flow the gems and bezels:

- 1 Hide both the original polysurface and the flowed polysurface.
- 2 From the **Transform** menu click **Flow along Curve**.
- 3 For the **Objects to flow**, select the group of gems and bezels.
They are a group and will select as one.



- 4 Select the **Base Curve** towards the left end.
- 5 Stop at this stage and confirm the following option settings on the command line:
(Copy=Yes Rigid=No Stretch=Yes).
- 6 Select the circle curve slightly below the point location as the **Target curve**.

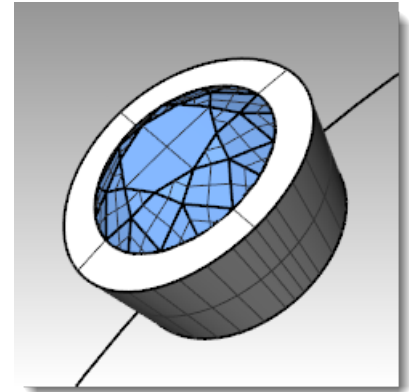
The bezels and gems are morphed to fit around the circle.



- 7 Examine the results.

The sides of the bezels are not perpendicular, the top surface is not flat, and gem is stretched.

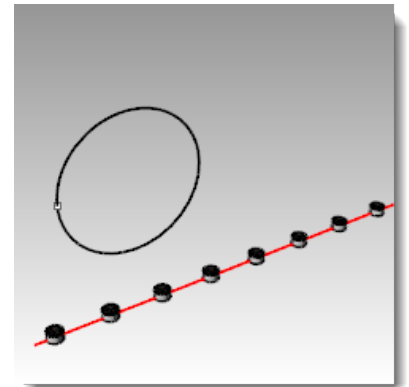
- 8 **Undo**.



To Flow the gems and bezels with Rigid=Yes:

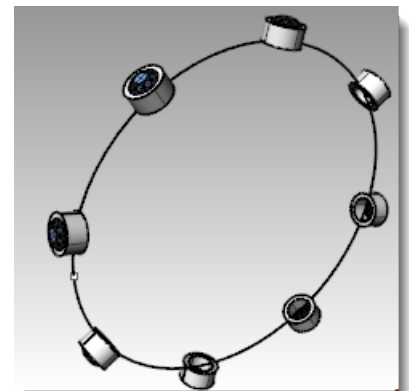
- 1 From the **Transform** menu click **Flow along Curve**.
- 2 For the **Objects to flow**, select the group of gems and bezels.

They are a group and will select as one.



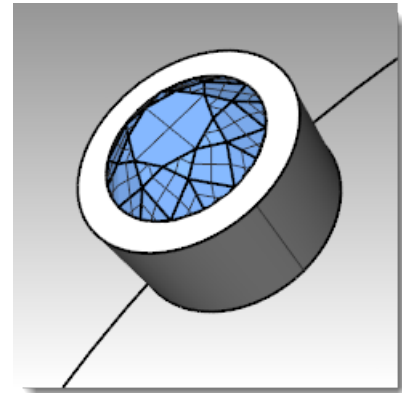
- 3 Select the **Base Curve** towards the left end.
- 4 Stop at this stage and confirm the following option settings on the command line:
(Copy=Yes Rigid=Yes Stretch=Yes).
- 5 Select the circle curve slightly below the point location as the **Target curve**.

The bezels and gems are stretched to fit around the circle. But the objects are not deformed.

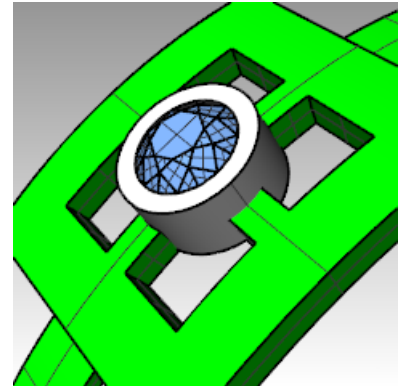


6 Examine the results.

The sides of the bezels are perpendicular, the top surface is flat, and gem is not stretched.



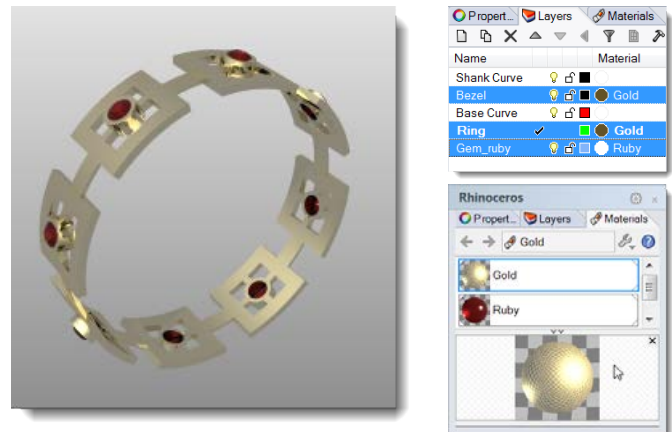
7 Show the green polysurface again.



To view the ring in a rendered viewport:

1 From the **View** menu click **Rendered**.

A Ruby material was assigned to the Gem_ruby layer. A Gold material was assigned to the Bezel and Ring layers.



2 Render the ring.

