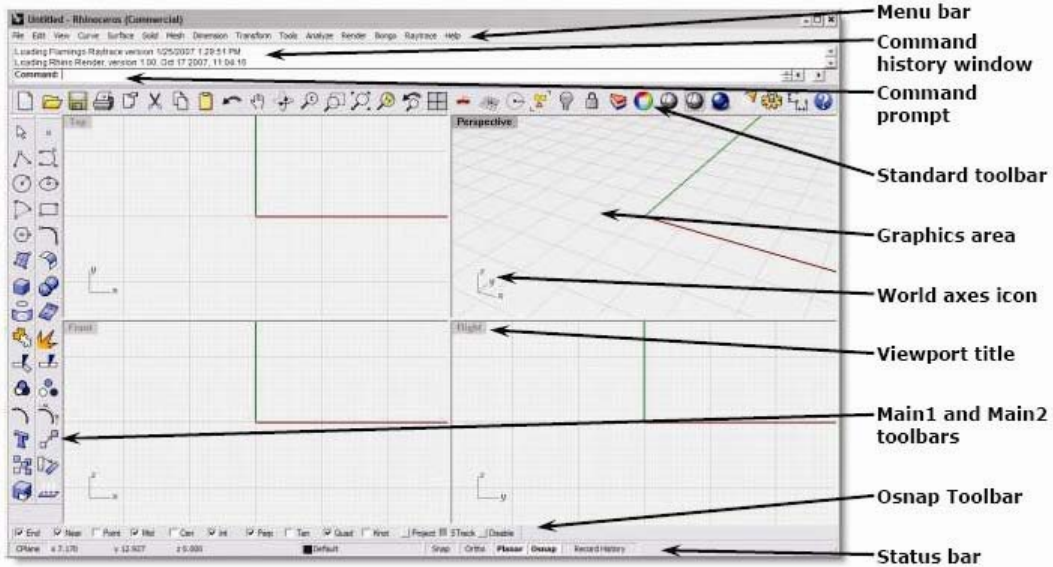
1. Basic Interface

The Rhino Screen

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

Screen Area	Description
Menu Bar	Access commands, options, and help.
Command area	Lists prompts, commands you enter, and information displayed by the command.
Toolbars	Access shortcuts to commands and options.
Graphics area	Displays the open model. Several viewports can be displayed. The default viewport layout displays four viewports (Top, Front, Right, and Perspective).
Viewports	Displays different views of the model within the graphics area.
Status bar	Displays the coordinates of the pointer, the status of the model, options, and toggles.



Rhino screen

Notes:

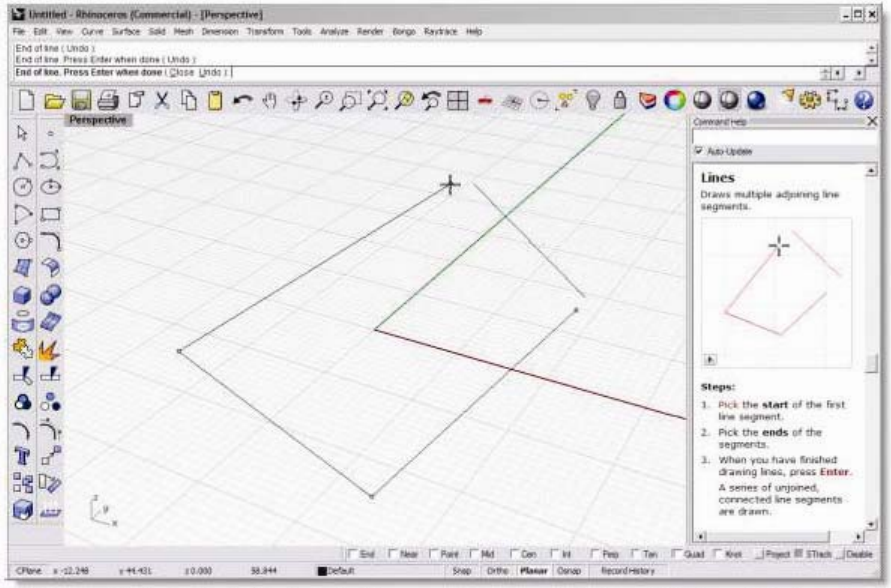
Watch the command line to find out what is happening.

Help

Press **F1** at any time to access Rhino Help. In addition to finding information about each command, Rhino help has conceptual information as well as many examples and graphics to help you complete your model. When you are confused or unsure about what to do, the first place you should look is the help file. You can also access help for a specific command by starting the command and press **F1**.

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

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



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

Notes:

2 Customizing Rhino

The toolbar layout

The toolbar layout is the arrangement of toolbars containing command buttons on the screen. The toolbar layout is stored in a file with the .tb extension that you can open and save. Rhino comes with a default toolbar collection and automatically saves the active toolbar layout before closing unless the .tb file is read-only. You can create your own custom toolbar collections and save them for later use.

You can have more than one toolbar collection open at a time. This allows greater flexibility to display toolbars for particular tasks.

Rhino's customization tools make it easy to create and modify toolbars and buttons. Adding to the flexibility is the ability to combine commands into macros to accomplish more complex tasks. In addition to toolbar customization, it is possible to set up command aliases and shortcut keys to accomplish tasks in Rhino.

Notes:

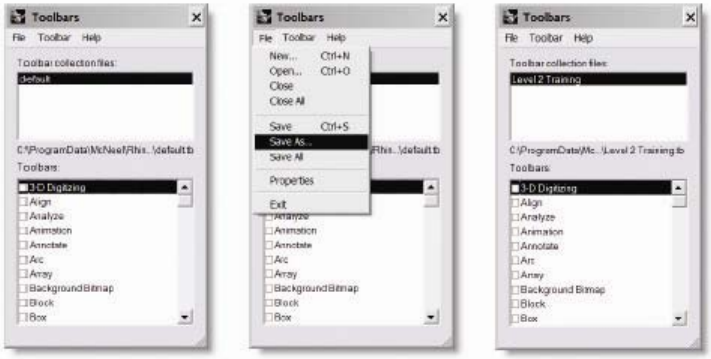
Exercise 2—Customizing Rhino's interface

In this exercise we will create buttons, toolbars, macros, aliases, and shortcut keys that will be available to use throughout the class.

To create a custom toolbar collection:

1. **Open** the model **ZoomLights.3dm**.
2. From the **Tools** menu, click **Toolbar Layout**.
3. Highlight the Default toolbar collection.
4. From the **Toolbars** dialog **File** menu, click **Save As**.
5. Type **Level 2 Training** in the **File name** box and click **Save**.

   A copy of the current default toolbar collection has been saved under the new name.



Toolbar collections are saved with a .tb extension. You will use this new toolbar collection to do some customization.

In the Toolbars dialog all the open toolbar collections are listed along with a list of all the individual toolbars for the selected toolbar collection. Check boxes show the current state of the toolbars. A checked box indicates that the toolbar is displayed.

Notes:

Edit Toolbar Layout

4 In the **Import Options** dialog, select the file you just saved.

5 For the **Options to import**, click **Aliases**, **Appearance**, or any other options you wish to import.

The saved options are imported.

Shortcut keys

The same commands, command strings, and macros that you can use for buttons are also available for keyboard shortcuts. Shortcuts are commands and macros that are activated by a function key, Ctrl, Alt, and Shift combinations, and an alphanumeric key on the keyboard.

To make a shortcut key:

1 From the **Tools** menu, click **Options**.

2 In the **Rhino Options** dialog, on the **Keyboard** page, you can add command strings or macros.

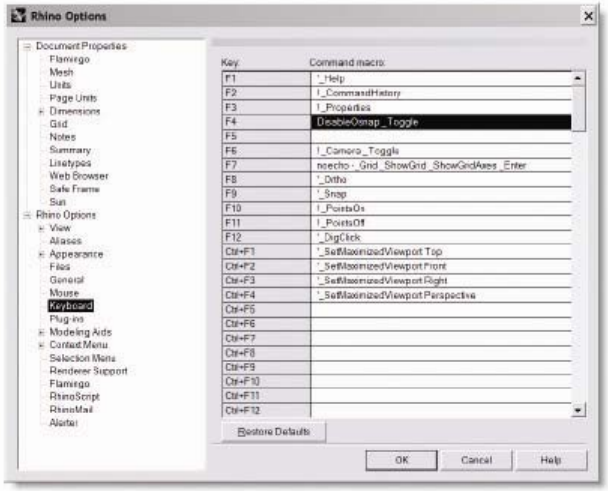
3 Click in the column next to the **F4** to make a new shortcut.

4 Type **DisableOsnap Toggle** for the shortcut.

This shortcut will make it easy to toggle the state of running object snaps.

5 Close the dialog and try it out.

There are several shortcut keys that already have commands assigned. The same rules apply here as with the buttons.



11 To make the Curves layer the only visible layer, from the **Edit** menu, click **Layers** then click **One Layer On**. Select the **Curves** layer.

One Layer ON

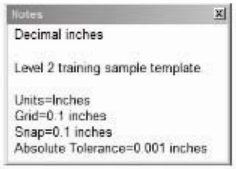
12 From the **File** menu, click **Notes**.

Type the details about this template.

13 From the **File** menu, click **Save As Template** and navigate to the templates directory.

Name the template **Small Objects –Decimal Inches - 0.001.3dm**.

This file with all of its settings is now available any time you start a new model.



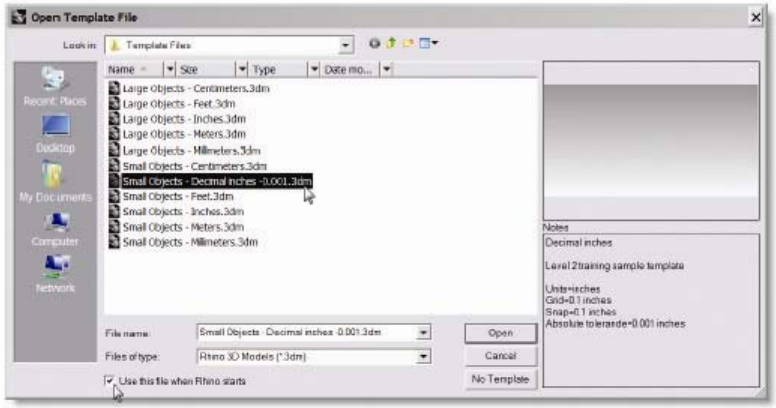
To set a default template:

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

2 Select the template you want to use as the default template.

3 In the **Open Template File** dialog, check the **Use this file when Rhino starts** checkbox.

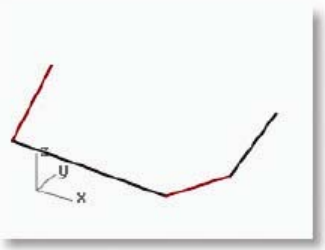
You should make custom templates for the kind of modeling that you do regularly to save set up time.



2. Lines

To draw line segments:

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

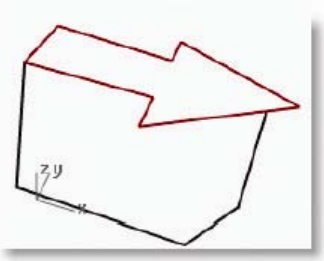


To use the Close option:

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

To draw a polyline:

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



Notes:

Line Segments
Right Click

Polyline
Left Click

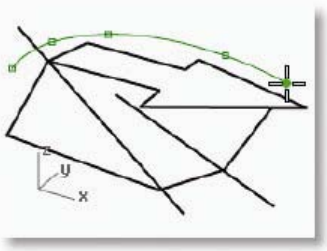
Drawing Free-form Curves

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
Close	Closes the shape by drawing from the last point picked to the first point picked. This ends the command.
EndTangent	After choosing a point on another curve, the next segment will be tangent to the point you picked and end the command.
Undo	Deletes the last point picked.
Degree	You can set the degree of the curve.
Knots	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.
Sharp	When you make a closed curve, it will come to a point instead of making a smooth closure as it normally does.

Exercise 4—Drawing interpolated curves

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

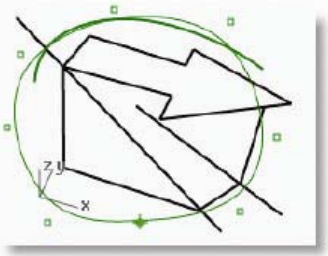


Notes:

Curve: Interpolate Points

Exercise 5—Drawing curves from control points

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



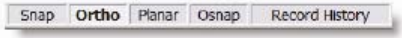
Notes:


Control Point Curve

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

Modeling Aids

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



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

Snap

Forces the marker to snap on grid intersections.

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

Ortho

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

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

If Ortho is set to on, hold down the Shift key to toggle Ortho off. If Ortho is off, hold down the Shift key to toggle Ortho on. F8 or Shift.

Planar

This is a modeling aid similar to Ortho. This helps you model planar objects by forcing input to be on a plane parallel to the construction plane that passes through the last point that you picked.

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

History

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

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

Grid

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

Notes:

Exercise 6—Drawing lines and curves using mode functions

1. Toggle **Snap** on and draw some lines.
   The marker snaps to each grid intersection.
2. Toggle **Snap** off, toggle **Ortho** on and draw some lines and curves.
   You can only input points that are at 90 degree intervals from the last point. Using Snap and Ortho toggles you can draw with precision. We will discuss other ways to get precision in a later session.

Model Setup

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

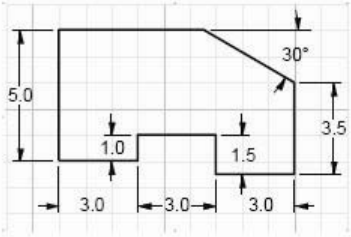
To change the options:

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

   Modeling Aids lets you control **Ortho**, **Object Snap**, **Grid Snap**, and other mode options.
3. Change the **Ortho** option to snap every **30** degrees.
4. In the **Document Properties** dialog box, click **Grid**.
5. In **Grid properties**, change the following settings.

   You can change the appearance of the modeling environment by changing the grid elements. The grid spacing, the frequency of the major lines, and the number of grid elements can be changed. The Grid dialog box lets you configure grid settings.
6. Change the **Grid Extents** setting to **10**.
7. Change the **Minor grid lines every** setting to **1**.
8. Change the **Major lines every** setting to **4**.
9. Change the **Snap Spacing** setting to **.25**, and click **OK**.
10. Draw some more lines and curves with **Snap** and **Ortho** on.

    Notice that the marker now snaps between the grid intersections and that **Ortho** snaps at every 30 degrees.
11. Try to draw the closed polyline to the right with **Snap** and **Ortho** turned on.



To reset the modeling aids options:

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

Notes:

Document Properties

The value for Grid extents is for each quadrant.

Saving Your Work

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

To save your model:

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

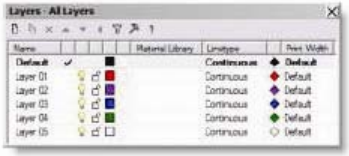
Command	Description
Save	Saves your model and keeps it open.
SaveSmall	Save your model without render or analysis meshes and preview image to minimize file size.
IncrementalSave	Save sequentially numbered versions of your model.
SaveAs	Saves your model to a specified file name, location, and format.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

Layers

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

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

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



Notes:

Save

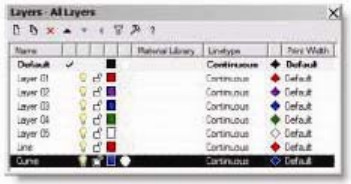
It is good practice to save your model in stages under different names, using the Save As command. This lets you go back to an earlier version of your model for modifications if necessary.

Edit Layers

Exercise 7—Layers

To create a new layer:

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



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

To assign a color to a layer:

1. Click the **Color** patch on the **Line** row in the list.
2. In the **Select Color** dialog box, click **Red** from the list.
   The right half of the sample rectangle turns red.
   Hue, Sat, Val are the hue, saturation and value components of the color.
   R, G, and B are the red, green and blue components of the color.
3. Click **OK.**
4. In the **Layers** window, the new color appears in the color bar on the Line row of the layer list.
5. Repeat steps 1–3 to make the **Curve** layer **Blue.**
6. Click **OK** to close the dialog box,

Notes:

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

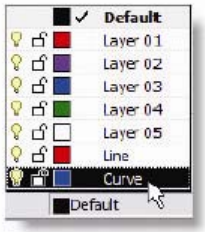
Hue is the color that is referred to as a scale ranging from red through yellow, green and blue and then circularly back to red.

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

Saturation is the vividness of hue. Value is the relative lightness or darkness of a color.

To make a layer current:

1. In the **Status Bar**, click the **Layer** pane.
2. In the **Layer popup**, click **Line.**
3. Draw some lines.
   The lines are on the Line layer and they are colored red.
4. To make a different layer current, click the **Layer** pane of the status bar.
5. Click **Curve.**
6. Draw some curves.
   They are on the Curve layer and are colored blue.
7. Draw more lines and curves on each layer.



Clicking the name or the checkbox sets the current layer.

To lock a layer:

1. In the **Status Bar**, click the **Layer** pane.
2. In the **Layer popup**, click the **Lock** icon on the row for the **Line** layer.
   Locking a layer turns it into a reference only layer. You can see and snap to objects on locked layers. You cannot select any objects on locked layers. You cannot make a locked layer current without unlocking it.

To turn a layer off:

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

Notes:

3. Precision Modeling

4 Precision Modeling

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

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

You can enter a point two ways: pick a point in a viewport with the mouse, or type coordinates at the command line.

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

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

Absolute Coordinates

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

Exercise 10—Setting up a model

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

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

Notes:

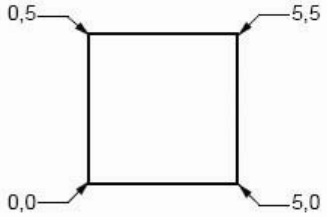
Set the units and tolerance of the model before you begin.

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

Cross-shaped cursor

Exercise 11—Entering absolute coordinates

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



Notes:

Polyline
Left Click

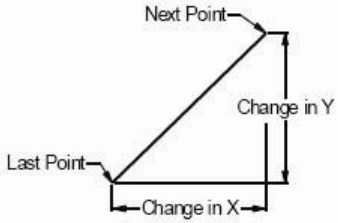
Relative Coordinates

Absolute coordinates can be slow and cumbersome, but they do work. Most of the time, ***relative*** coordinates are easier to use.

Every time you select a point, Rhino saves that point as the ***last point***.

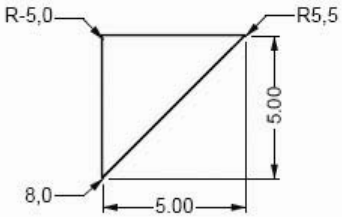
Relative coordinates are based on the last point entered, instead of on the origin (0,0,0) of the construction plane.

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



Exercise 12—Entering relative coordinates

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



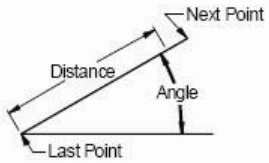
Polar Coordinates

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

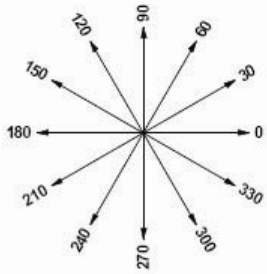
Vector directions in Rhino start with Zero degrees at 3 o'clock on a standard clock. They change in an anti-clockwise direction as illustrated below.

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

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

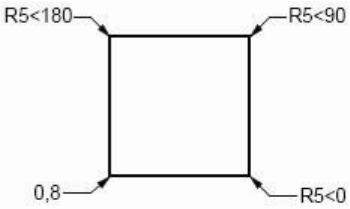


Instead of using x-, y-, and z-coordinates, enter relative polar coordinates like this: Rdistance<angle.



Exercise 13—Entering polar coordinates

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



Notes:

Distance and Angle Constraint Entry

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

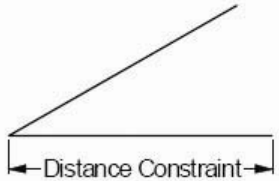
Using angle constraint entry, you can specify an angle by typing < followed by a value and pressing **Enter**. The next point is constrained to lines at multiples of the angle relative to the x-axis you specified.

Using the Shift key to toggle Ortho on and off:

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

Exercise 14—Distance constraint entry

1. From the **Curve** menu, click **Polyline**, and then click **Polyline**.
2. To **Start** type **8,8** and press **Enter**.
3. For the **Next point** type **5** and press **Enter**.
4. Hold the **Shift** key down and pick a point to the right.
   Ortho constrains the marker to 0 degrees.
5. For the **Next point** type **5** and press **Enter**.
6. Hold the **Shift** key down and pick a point up.
   Ortho constrains the cursor to 90 degrees
7. For the **Next point** type **5** and press **Enter**.
8. Hold the **Shift** key down and pick a point to the left.
   Ortho constrains the cursor to 180 degrees.
9. Click **Close** to close the polyline.



Notes:

Exercise 15—Distance and angle constraint entry

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

2 To **Start** type **16,5** and press **Enter.**

3 For the **Next point** type **5** and press Enter, then type **<45** and press Enter.

As you drag your cursor around, the marker snaps to a distance of 5 and an angle of 45 degrees.

4 Pick a point down and to the right.

The angle constraint sets the angle.

5 For the **Next point** type **5** and press **Enter**, then type **<45** and press **Enter.**

6 Pick a point up and to the right.

The angle constraint sets the angle.

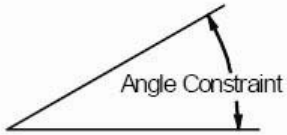
7 For the **Next point** type **5** and press **Enter**, then type **<45** and press **Enter.**

8 Pick a point up and to the left.

The angle constraint sets the angle.

9 Click **Close** to close the polyline.

10 Save your model. You will use this model for another exercise.



Notes:

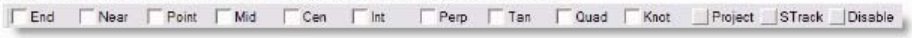
Object Snaps

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

To open the Osnap toolbar

- Click the **Osnap** pane in the status bar.

This toolbar controls persistent object snaps. Use persistent objects snaps to maintain an object snap through choosing several points without having to reactivate the object snap.



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

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

Command	Button	Description
End		End snaps to the end of a curve, surface edge corner or polyline segment end.
Near		Near snaps to the nearest point on an existing curve or surface edge.
Point		Point snaps to a control point or point object.
Mid		Midpoint snaps to the midpoint of a curve or surface edge.
Cen		Center snaps to the center point of a curve. This works best with circles and arcs.
Int		Intersection snaps to the intersection of two curves.
Perp		Perpendicular To snaps to the point on a curve that makes a perpendicular to the last selected point. It doesn't work on the first point that a command prompts you to pick.
Tan		Tangent To snaps to the point on a curve that makes a tangent to the last selected point. It doesn't work on the first point that a command prompts you to pick.
Quad		Quad snaps to the quadrant point. The quadrant point is the maximum or minimum direction on a curve in the x or y construction plane direction.
Knot		Knot snaps to knot points on curves or surface edges.
Project		Projects the snap point to the construction plane.
SmartTrack		SmartTrack is a system of temporary reference lines and points that is drawn in the Rhino viewport using implicit relationships among various 3-D points, other geometry in space, and the coordinate axes' directions.
Disable		Temporarily turns off persistent object snaps, retaining settings

Notes:

Exercise 21 – Project Constraint

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

To use the Project Constraint:

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

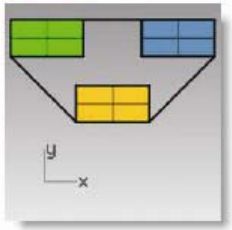
2 Make sure that the **Ortho** constraint is **On.**

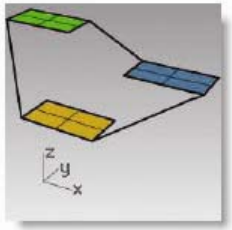
3 Turn **Layer 01** off and turn **Layer 02** on.

The surfaces on Layer 02 are located at different elevations.

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

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



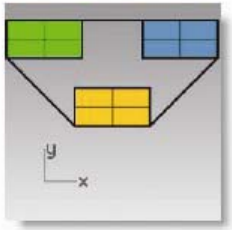


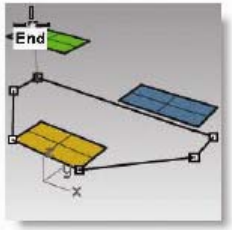
5 **Delete** the Polyline.

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

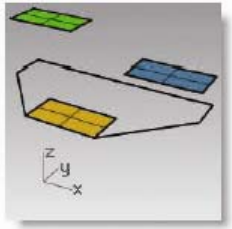
7 Again, working in **Top** viewport, draw a **Polyline** around the perimeter of the three rectangles.

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





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



8 **Save** the File

Notes:

Exercise 22 – Planar Constraint

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

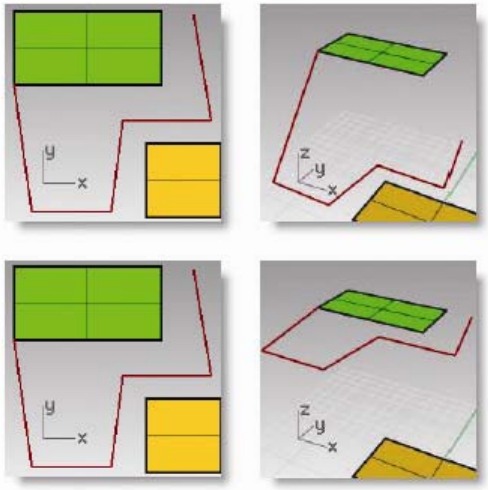
To use the Planar Constraint:

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

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

   Look at the Perspective viewport and notice how the polyline pulls back onto the construction plane after the initial point.
6. To override this behavior and create a planar curve **delete** the **Polyline** and turn on the **Planar** constraint.
7. Draw the **Polyline** again.

   Note how it now remains planar.
8. **Delete** the **Polyline**.



Notes:

4. Edit Lines Loft

5 Editing Objects

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

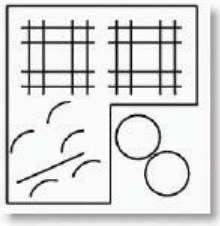
Fillet

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

Option	Description
Radius	Sets the fillet radius. A radius of 0 extends or trims the curves to a corner, but does not create a fillet.
Join	**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.**



Rules to follow when filleting curves:

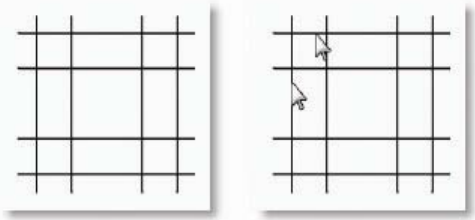
The curves must be coplanar.

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

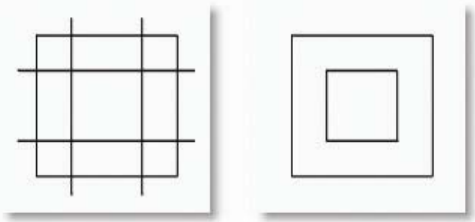
The radius cannot be so large that it runs past the end of the curve.

To fillet lines at a zero radius:

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



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



To join the filleted objects:

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

Notes:

Fillet Curves

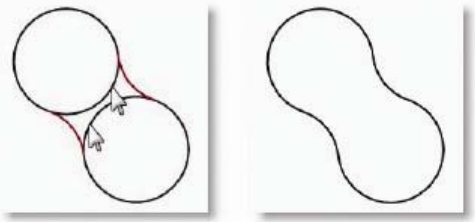
Remember to pick on the part of the line that you want to keep.

Join

To fillet circles:

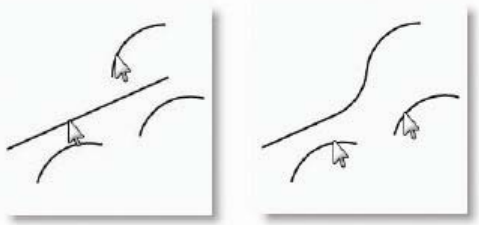
1. From the **Curve** menu, click **Fillet Curves.**
2. Type **3** and press **Enter.**
3. Select the right edge of a circle.
4. Select the right edge of the other circle.

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.

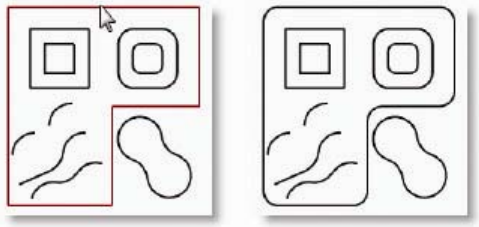


Notes:

To fillet the corners of a closed polyline:

1. Select the closed polyline.
2. From the **Curve** menu, click **Fillet Corners**.
3. For the **Fillet** radius, type **2** and press **Enter**.

   All corners are filleted at once.



Notes:

Fillet corners

Blend

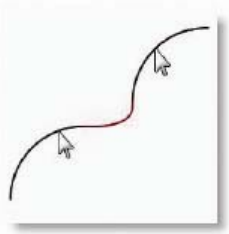
Blending is another method to connect lines, arcs, or curves. There are two blend commands that work on curves, **Blend** *(default blend)* and **BlendCrv** *(adjustable blend)*. Blend offers many additional options for how the curves connect.

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

To blend two curves:

1. From the **Curve** menu, click **Blend Curves**.
2. Select the **Curves to blend** near the ends that you want to connect.

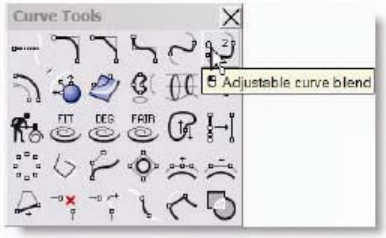
   The arcs are connected with a smooth curve. The default continuity is Curvature.
3. **Delete** the blend curve.



Blend Curves

To blend two curves with Adjustable curve blend:

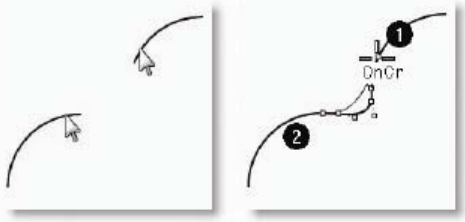
1 Since there is no menu pick for this command, type **BlendCrv** on the command line or select the **Adjustable curve blend** icon from the **Curve Tools** toolbar.



2 Select the **Curves to blend** near the ends that you want to connect.

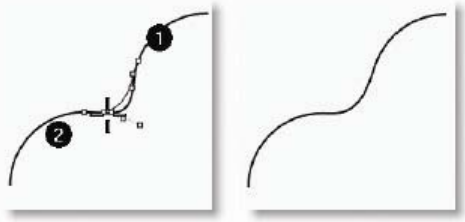
You will see a preview of the default blend with control points displayed.

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



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

5 After making your adjustments, press **Enter** to make the blend.



Notes:

Adjustable curve blend

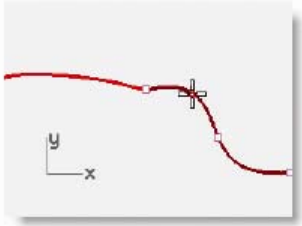
To add a knot:

Adding a knot or two to the curve will put more points near the end so that the third point can be nearer the end. Knots are added to curves and surfaces with the **InsertKnot** command.

1. Undo your previous adjustments.
2. Start the **InsertKnot** command *(Edit menu: Control Points > Insert Knot)*.
3. Select the magenta curve.
4. Pick a location on the curve to add a knot in between the first two points.

   In general a curve or surface will tend to behave better in point editing if new knots are placed midway between existing knots, thus maintaining a more uniform distribution.

   Adding knots also results in added control points.

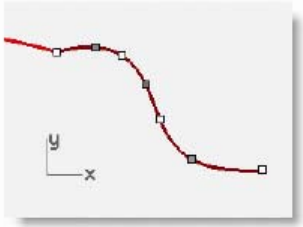


Knots and Contol Points are not the same thing and the new control points will not be added at exactly the new knot location.

The Automatic option automatically inserts a new knot exactly half way between each span between existing knots.

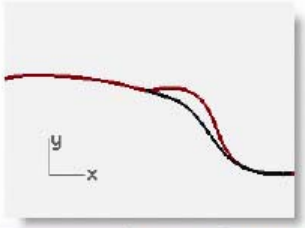
If you only want to place knots in some of the spans, you should place these individually by clicking on the desired locations along the curve.

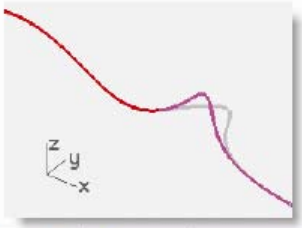
Existing knots are highlighted in white.



5. Match the curves after inserting a knot into the magenta curve.

   Inserting knots closer to the end of curves will change how much Match changes the curve.





Notes:

Insert Knot

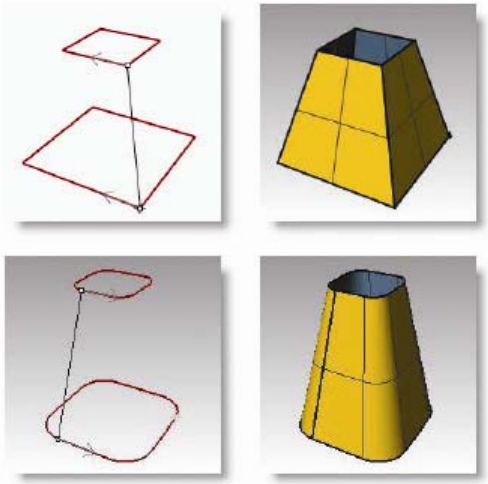
To make a lofted surface with closed curves:

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

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

   If the seams don't line up at corresponding points on the two curves, drag the seam point until it does.
5. Press **Enter**.
6. In the **Loft Options** dialog box, click **OK**.

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



Notes:

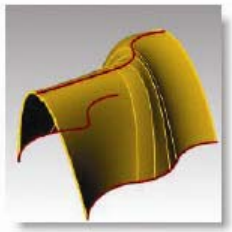
The curves in this model are at two different elevations. You are going to connect the curves at different elevations with a surface.

Loft

To make a lofted surface with open curves:

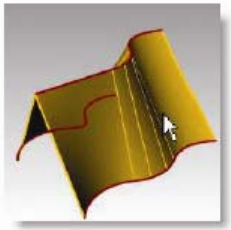
1 Repeat the **Loft** command for the three open curves.

2 In the **Loft Options** dialog box, change the **Style** to **Loose**, click **Preview**.



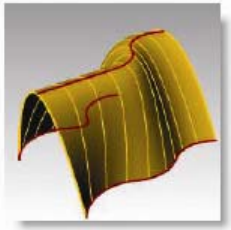
3 Change the **Style** to **Straight sections**, click **Preview**.

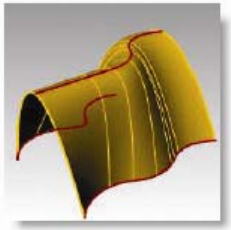
4 Change the **Style** to **Normal.**



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

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





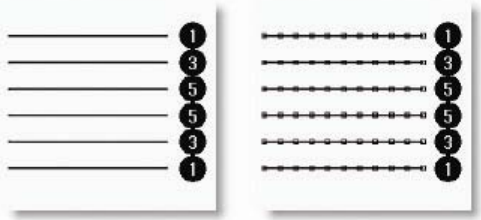
Notes:

Exercise 52—Control point editing

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

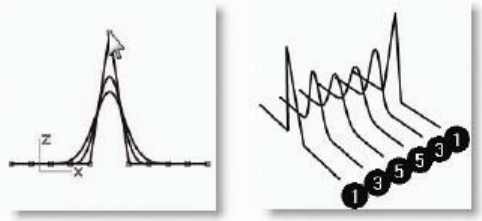
To edit control points:

1. **Open** the model **Control Point.3dm**.
   There are pairs of curves with different degrees in the model.
2. Turn **Ortho** and **Snap** on.
3. From the **Edit** menu, click **Select Objects**, and then click **Curves**.
4. From the **Edit** menu, click **Control Points**, and then click **Control Points On**. (Press **F10**.)



Control Points On

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 make a lofted surface:

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

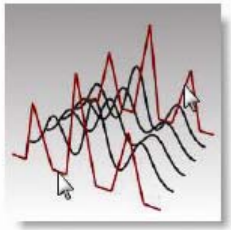
   Because the degree 1 curves were included in the loft, a polysurface is created with a seam at each kink.
4. Select the surface.
5. Turn on the control points.

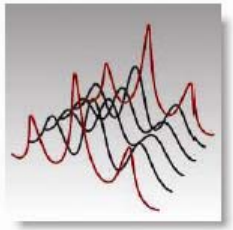
   The points do not turn on and the following message is displayed on the command line: **Cannot turn on points for polysurfaces.**
6. **Undo** the loft.

To change the polylines into curves without kinks:

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

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






Rebuild

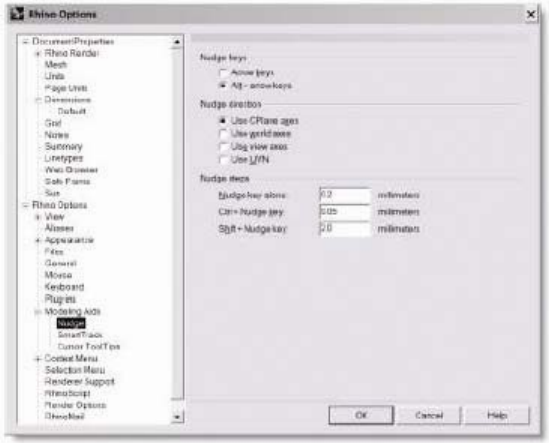
Nudge Controls

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 down the **Alt** key down and press an arrow key.

   Notice that it moves (nudges) the point a small amount.
3. Hold the **Alt** and the **Ctrl** key down and press another arrow key.

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

   The movement is magnified.
5. Hold the **Alt** and press the **PageUp** or **PageDown** key to nudge in the **Z** direction.

To use set points to adjust points:

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

4. In the Right viewport move the points and click.

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

Set XYZ Coordinates

5. Trim, Split, etc

Trim

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

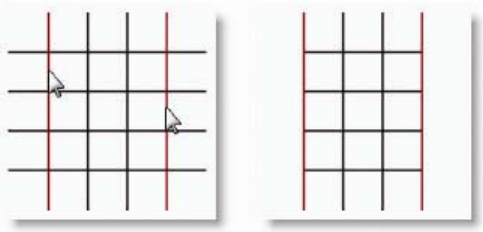
Exercise 44—Trim

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

To trim curves:

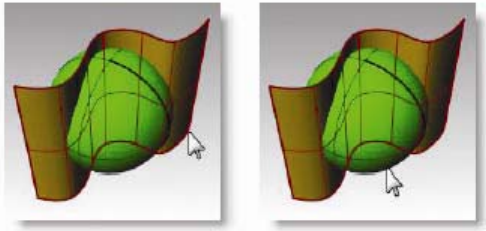
1. **Open** the model **Trim-Split.3dm**.
2. **Zoom Window** around the grid in the lower left corner of the **Top** viewport.
3. Pre-select the cutting objects by selecting the two outside vertical lines in the grid.
4. From the **Edit** menu, click **Trim**.
5. Select each of the horizontal lines at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

   The lines are trimmed to the cutting edges.
6. Press **Enter**.



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**.



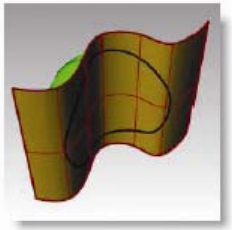
Notes:

Zoom Window

Trim

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

5 Press **Enter**.



Notes:

Split

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

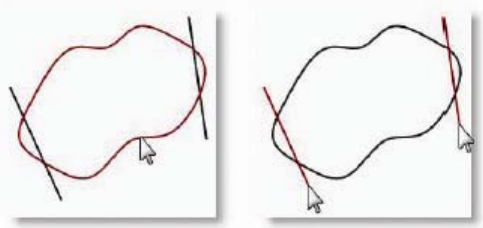
Exercise 45—Split

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

To split a curve:

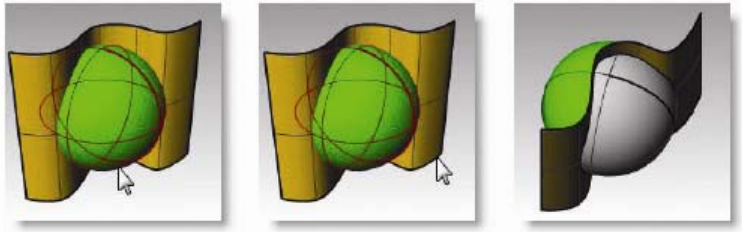
1. **Zoom Window** around the closed curve in the lower right corner of the **Top** viewport.
2. Select the closed curve.
3. From the **Edit** menu, click **Split**.
4. Select the lines and press **Enter**.

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



To split a surface:

5. From the **View** menu, click **Zoom**, and then click **Extents All**.
6. From the **Edit** menu, click **Split**.
7. Select the sphere, and press **Enter**.



8. Select the surface that intersects the sphere and press **Enter**.

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

Notes:


Split

Extend

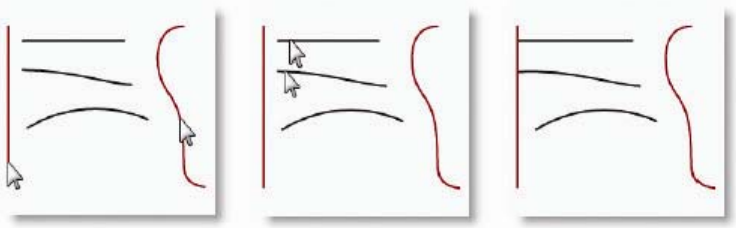
Extend lengthens an object to make it end precisely at its intersection with another object or you can lengthen an object when there is no intersection.

Exercise 46—Extend

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

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

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



4 Press **Enter**.

5 For the **curve to extend**, select the left end of the line and the middle curve.

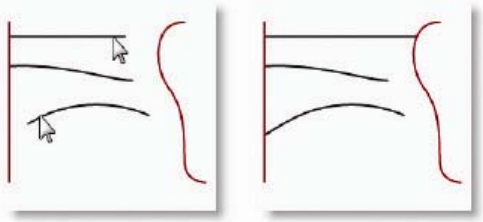
The line and curve extend to touch the boundary edge on the left.

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

7 Change to **Type=Line**.

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

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



Notes:

Extend Curve

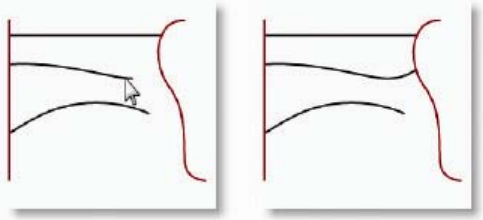
Notes:

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

10 Change to **Type=Arc**.

11 Select the right end of the middle curve.

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



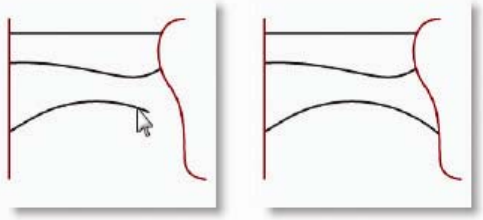
12 For the next **curve to extend**, click **Type=Arc** on the command line.

13 Change to **Type=Smooth**.

14 Select the right side of the bottom curve.

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

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



To extend with a set extension length:

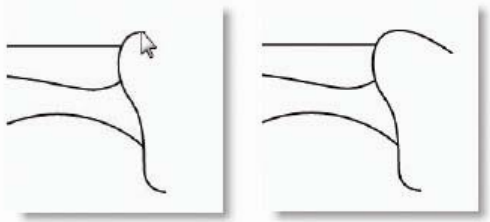
1 From the **Curve** menu, click **Extend Curve**, and then click **Extend Curve**.

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

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

The curve extends exactly 4 units.

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



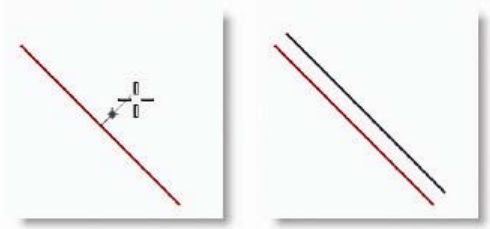
Offset

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

Exercise 47—Offset

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

   A parallel line is created.



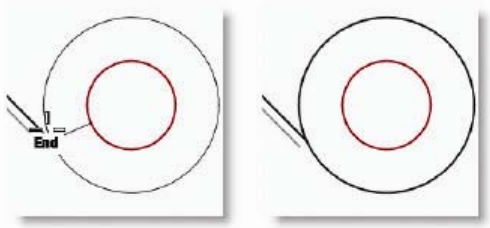
Notes:

Offset Curve

To offset with the through point option:

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

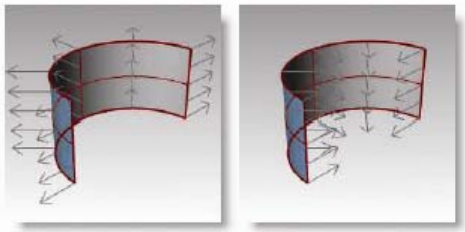
   A concentric circle is created that goes through the endpoint of the line.



To offset a surface:

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

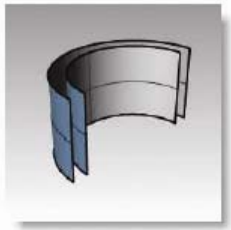
   Notice that the direction of the arrows changed. The normal direction of the surface was flipped by the mouse click. If you have a positive number for offset distance, the surface will be offset in the direction of the normal. If you have a negative number it will offset the opposite direction.



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

4. Press **Enter**.

   The surface is offset in the direction of the normals.

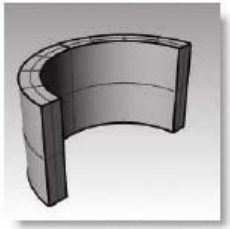


Notes:

Offset Surface

To offset a surface to a solid:

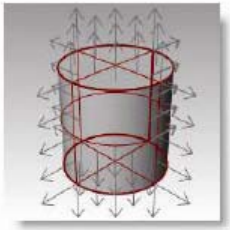
1. Select the other open surface.
2. From the **Surface** menu, click **Offset Surface**.
3. Click on the surface to change the normal direction if necessary.
4. Choose the **Solid** option.
5. Press **Enter** to create the offset surface and the surfaces needed to make the solid.

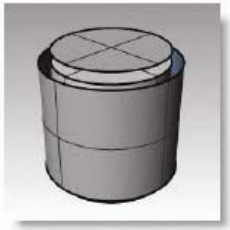


To offset a polysurface:

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

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

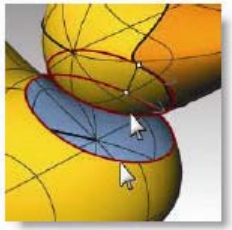




Notes:

To create the blend surface between the head and body:

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



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

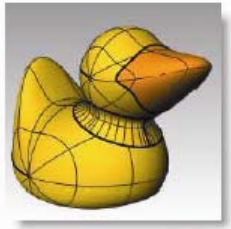
   The blend surface between the body and the head is displayed.
9. Make any adjustments you desire by moving the slider bars in the dialog box, click **OK** when finished.

   If you check the box at the left of the sliders the surface adjustments are symmetrical.
10. **Save** your model.

To join the parts:

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

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



Notes:

Blend Surface

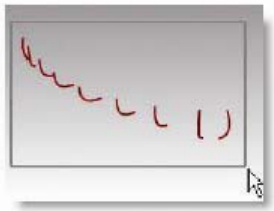
Join

Exercise 58—Lofted surfaces

Notes:

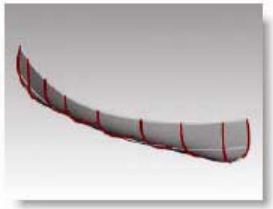
1 **Open** the model **Loft.3dm**.

2 Window select all of the curves.



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

A surface is fitted over the curves.



Loft

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.

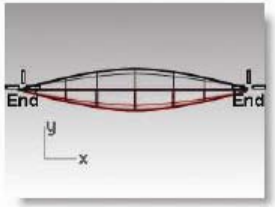
5 In the **Loft Options** dialog box, switch **Style** to **Loose**, and then click **Preview**.

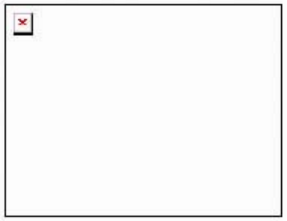
A surface is created that uses the same control points as the curves. The surface follows the curves more loosely.

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

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

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





Mirror

8 **Join** the two halves.

Join

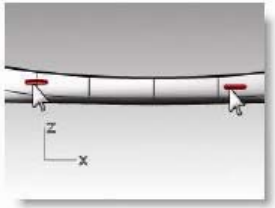
To make a seat:

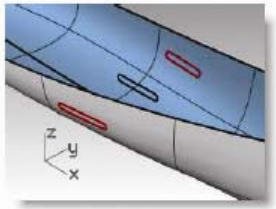
1 Turn the **Hull Curves** layer off and turn the **Seat Curves**, and **Seat** layers on.

2 Make the **Seat Curves** layer current.

3 In the **Front** view, select the rounded rectangles.

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





Project to Surface

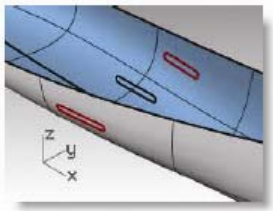
Project uses the current construction plane to determine the direction of projection. Make sure you select the curves and surface to project onto in the Front viewport.

Notes:

5 For the **Surface to project onto**, select the hull.

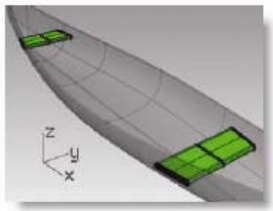
The curves will be projected to both sides of the hull surface.

6 Select the curves on the surface.



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

8 In the **Loft** dialog, click **OK**.



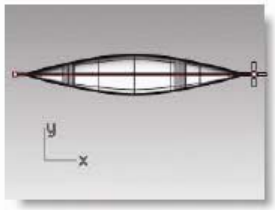
A surface is fitted over the curves that fits exactly with the shape of the hull.

To create section curves from the surfaces:

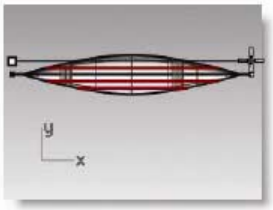
1 Select the hull.

2 Change to the **Sections** layer.

3 From the **Curve** menu, click **Curve From Objects**, and then click **Section.**

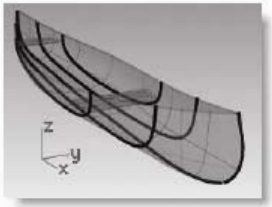


4 For the **Start of section**, in the **Top** viewport, pick a point to the left at the center of the hull.



5 For the **End of section**, with **Ortho** on, drag a line to the right and pick.

A curve is generated on the surface. Repeat this at various locations.

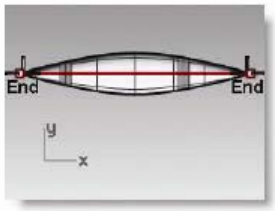


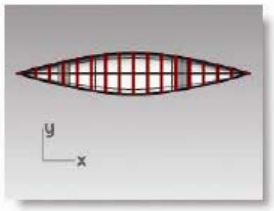
Notes:

Section

To create contour curves across the hull surfaces:

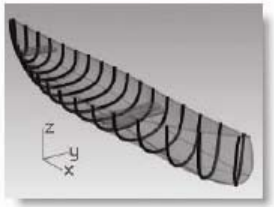
1. Select the hull.
2. Change to the **Contours** layer.
3. From the **Curve** menu, click **Curve From Objects**, and then click **Contour.**
4. For the **Contour base point**, snap to the left end of the canoe.





5. For the **Direction perpendicular to contour planes**, snap to the other end of the canoe.
6. For the **Distance between contours**, type **12** and press **Enter**.

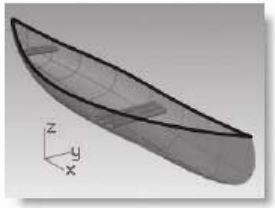
   A curve is generated every foot along the hull.

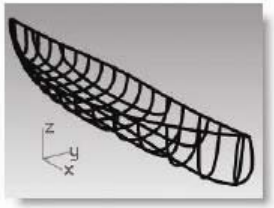


To create an edge curve from the surfaces:

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

   Two curves are generated at the edges of the hull





Notes:

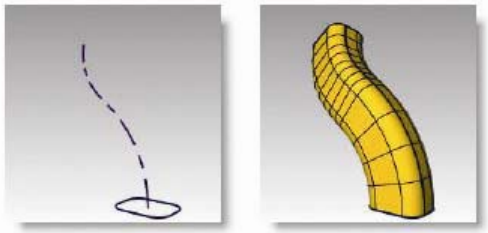
Contour

Duplicate Edge

Exercise 61—Using 1-rail sweeps to create surfaces

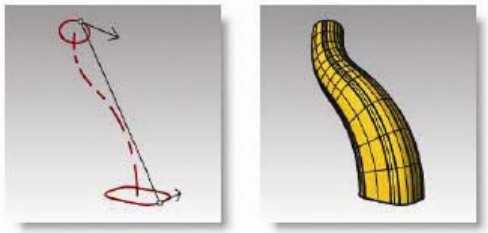
One cross-section:

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



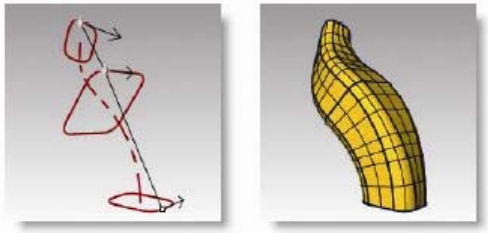
Two cross-sections:

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



Multiple cross-sections:

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

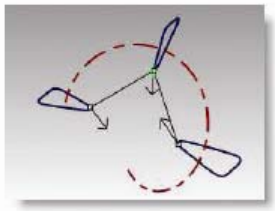


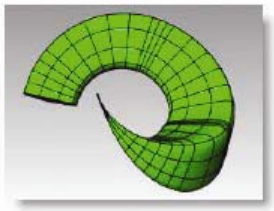
Notes:

Sweep 1 Rails

To create a 1-rail sweep to a point:

1. Make **Surface 02** layer group current and turn off **Surface 01** layer group.
2. From the **Surface** menu, click **Sweep 1 Rail**.
3. For the **Rail**, select the open free-form curve.
4. For the **Select cross section curves**, select the three closed curves, then click **Point** on the command line.

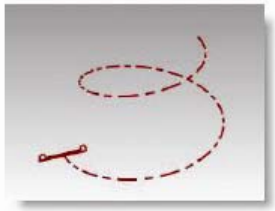




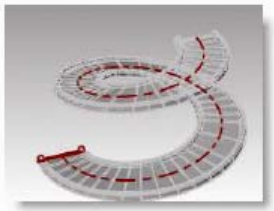
5. For the **Pick end point**, snap to the **End** of the free-form curve.
6. In the **Sweep 1 Rail Options** dialog, click **OK**.

To create a roadlike 1-rail sweep:

1. Make **Surface 03** layer group current and turn off **Surface 02** layer group.
2. Select the helix.
3. From the **Surface** menu, click **Sweep 1 Rail**.
4. For the **Select cross section curves**, select the closed curve, press **Enter**.



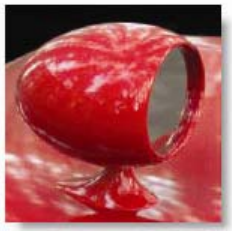
5. Change the style to **Roadlike Right**, click **Preview**.
6. Change the style to **Roadlike Front**, click **Preview**.
7. Change the style to **Roadlike Top**, click **Preview**, click **OK**.



Notes:

Exercise 62—Using 2-rail sweeps to create surfaces

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



To create the base —Part 1:

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

Notes:

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.

2
Sweep 2 Rails

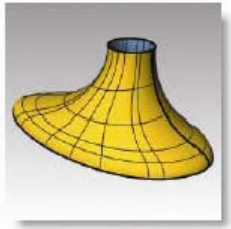
6. In the **Sweep 2 Rail Options** dialog, check **Maintain height**, click **Preview.**

   Notice that the cross-section maintains the same height thoughout 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**.

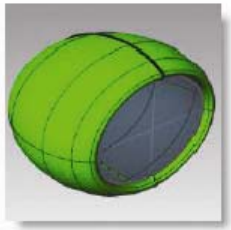
   A surface is created whose edges match the rail curves and the cross-section curves.



To create the housing:

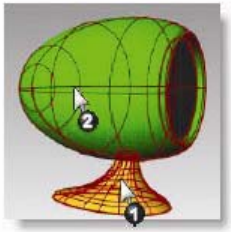
1. Turn the **Housing Surface**, **Housing Curves**, and **Mirror** layers on.
2. Make the **Housing Surface** layer current.
3. From the **Surface** menu, click **Sweep 2 Rails**.
4. Select the two **rail curves** (1).
5. For the **cross-section**, select the outer edge of the cylinder, press **Enter**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

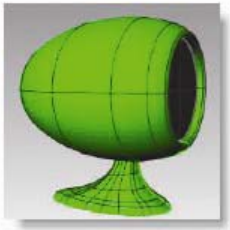
   A surface is created.



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.





Notes: