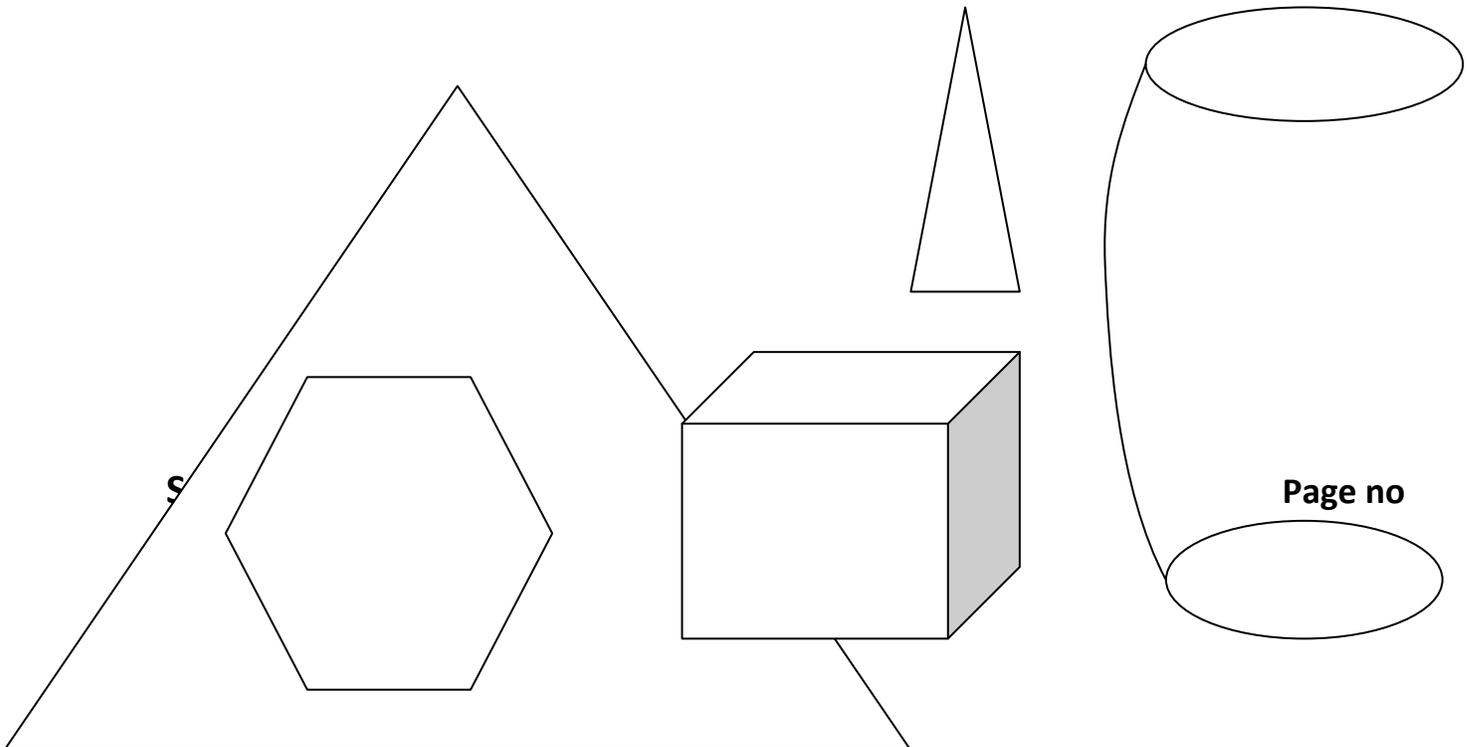


COMPUTER AIDED DESIGN

CURRICULOM RHINO

BASED 3D DESIGN



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INTRODUCTION

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.

Course Objectives

- 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 Smart Track™ 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

1

Why Rhinoceros is useful to Designing?

When we make any product, first of all we need to design. If a product is made perfectly, what types of things shows that it is perfect, to prove it, it is necessary that its angle, dimension, and measurement should be perfect. A Design expresses its full Vertical and Horizontal sides as its length and width and its angle also. A big products design can be expressed on the paper and it would come in existence easily.

For what purpose to learn Rhinoceros?



- To learn the basic knowledge of designing.
- How to design a product.
- To understand of various products produced by different manufactured.
- To draw the design according to the buyers demand.
- To check the basic requirement to draw design.
- To draw the design according to the given size.
- To learn how to apply the different types of finishes.
- To learn knowledge of units used in design.

2

WORKING WITH 3D MODELS

With 3d modeling, you can design using solid, surface, and mesh models. Create new 3D solids and surfaces, or sweep, combine, and modify existing objects. There are some solid based commands through which we can directly make solid object; there are some solid based command.

- Create a Solid Box
- Create a Solid Cone
- Create a Solid Cylinder
- Create a Solid Pyramid
- Create a Solid Torus
- Create a Solid Wedge

There are some special commands that make 3d from 2d and can change the shape of object, now you can learn these commands.

Object types

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.

Polysurface

Rhino also has objects which consist of two or more surfaces joined together.

Solid

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

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.

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.

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.

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.

Menus

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

Toolbars

Rhino toolbars contain buttons that provide shortcuts to commands.

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.

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.

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.

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.

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.

Creating and Editing Geometry

To draw line segments:

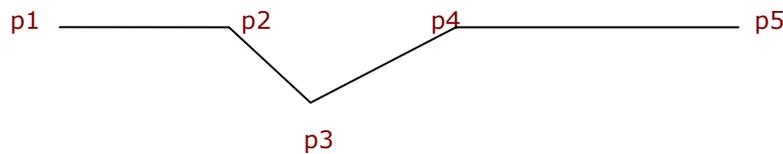
- 1** From the **Curve** menu click **Line**, and then click **Line Segments** to begin the **Lines** command.
- 2** Pick a point in a viewport.
- 3** Pick another point in a viewport. Click p1 and drag the mouse click next point p2.

A line segment appears between the two points.

p1 ————— p2

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. Click first point p1 , next click p2 then p3 and continuo p4,p5 .
- This makes an open polyline. A polyline is made of line segments that are joined together. It is one object.



To draw a single line segment:

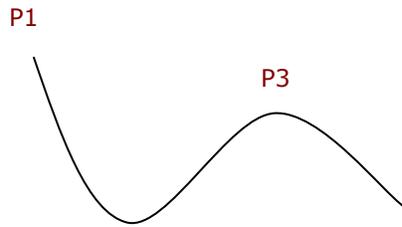
- 1 From the **Curve** menu click **Line**, and then click **Single Line** to begin the **Line** command.
 - 2 Pick a **Start** point.
 - 3 Pick an **End** point.
- The command ends after one segment is drawn



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



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.

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.

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.

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**.

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.

To make a layer current:

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

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.

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.

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.

Deformable Draws an approximation of a circle with a defined number of control points..

Circle Around Curve Draw a circle perpendicular to a curve at the chosen point. Vertical Draw a circle perpendicular to the construction plane.

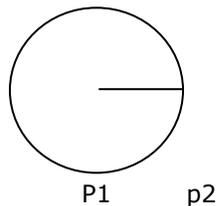
To draw a center, radius circle:

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 draw a diameter circle:

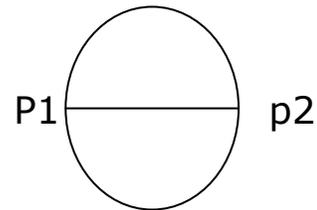
1 Turn On the **Point** osnap.

2 From the **Curve** menu click **Circle**, and then click **2 Points**.

3 For the **Start of diameter** pick snap p2

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.



To draw a tangent, tangent, radius circle:

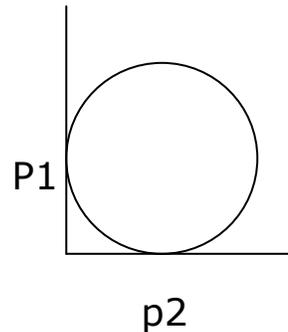
1 From the **Curve** menu click **Circle**, and then click **Tangent, Tangent, Radius**.

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 .



To draw a rectangle from diagonal corners:

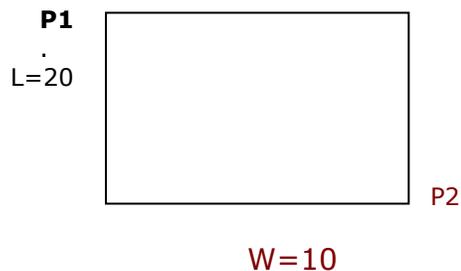
1 From the **Curve** menu click **Rectangle**, and then click **Corner to corner**

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**

L



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.

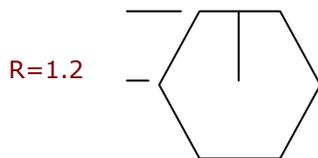
Radius=3

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 Make a six sides polygon.



To make the rectangle solid:

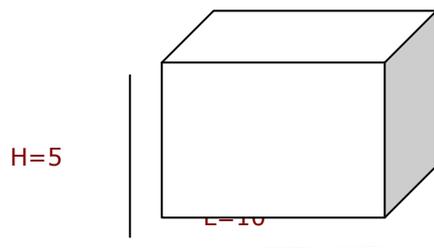
1 Select the rectangle.

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 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

Edit and Modify the Shape

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

.

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.

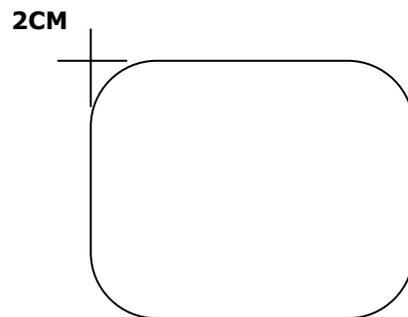
4 Select an outer vertical line

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.



T

Chamfer

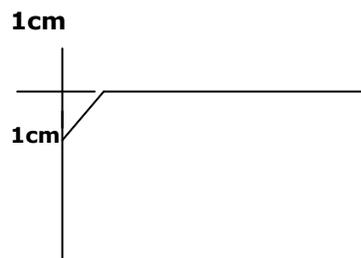
Chamfer connects two curves by extending or trimming them to intersect or to join with a beveled line. Chamfer works on convergent or intersecting curves.

□ Open the model Chamfer.3dm.

To chamfer lines:

1 From the Curve menu click **Chamfer Curves** .

2 For the **Select first curve to Chamfer**, type **1,1** and press **Enter**, to set the distances.



Move

Use Move to move objects without changing orientation or size.

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

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.

To copy using object snaps for placement:

- Select the small circle at the lower left of the object.
- From the **Transform** menu click **Copy**.
- For the **Point to copy from**, snap to the center of the small circle.
- For the **Point to copy to**, snap to the center of the arc at the upper left of the object.
- Pick to place the object, and press **Enter**.

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.

Group

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

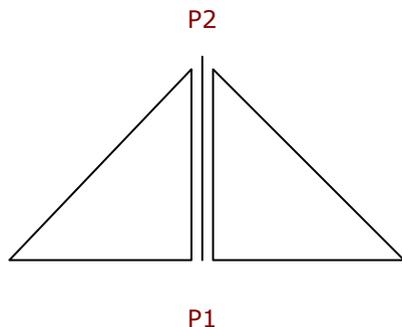
- 1 Select the two circles that you placed.
- 2 From the **Edit** menu click **Groups**, and then click **Group**.

Mirror

Mirror creates a copy of the objects flipped over a specified axis **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. Click first axis p1 then click second axis p2 on the construction plane.



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.

Join

- 1** Select the two polylines.
- 2** From the **Edit** menu click **Join**.
- 3** Click first line then click second line.



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.

–Scaling objects

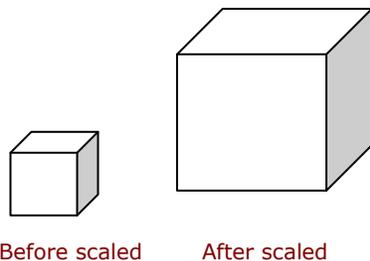
- 1** From the **Edit** menu click **Select Objects**, and then click **Curves**.
- 2** From the **Transform** menu click **Scale**, and then click **Scale 2-D** .
ommands.

3 For the **Origin point**, type **0** and press **Enter**.

4 For the **Scale factor**, type **.75** and press **Enter**.

5 Select small object and drag the mouse.

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



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.

—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**.



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**.

2 Maximize the Top viewport.

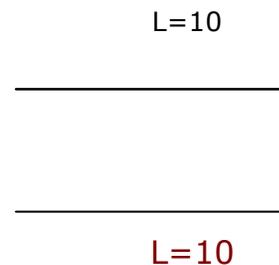
L10CM

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 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.

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

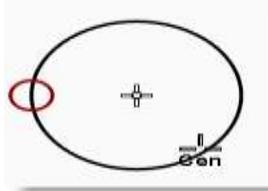
Exercise 49—Polar Array

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 .



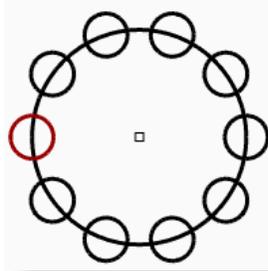
then click **Polar**.

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



3D COMMANDS AND THEIR USES

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.

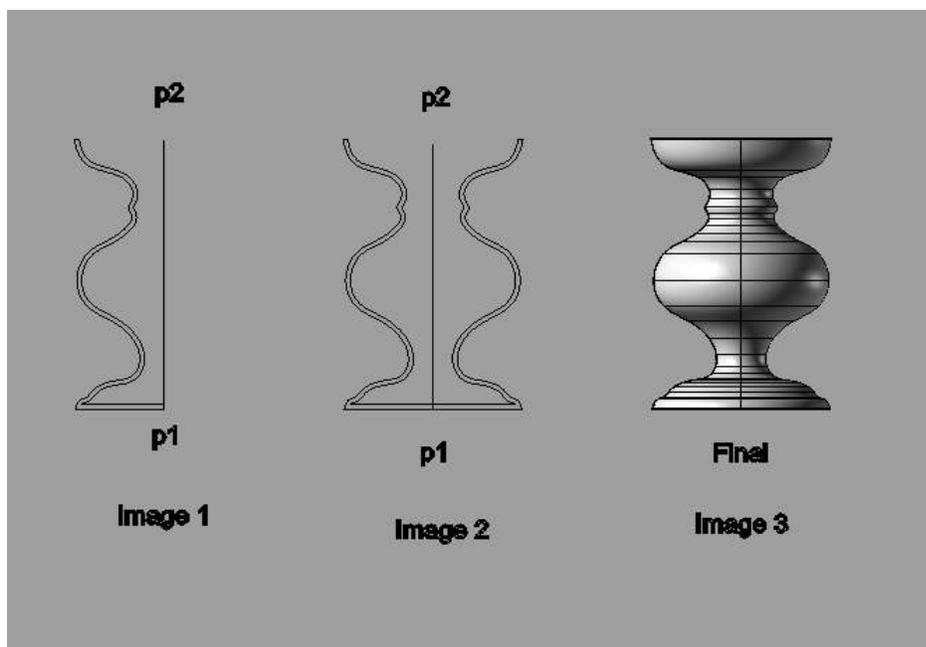
BooleanDifference = Boolean operation in Rhino to subtract surfaces and solids from one another.

BooleanIntersection = Boolean operation in Rhino to create the intersection of two surfaces or solids.

HOW TO MAKE 3D MODELS FROM A 2D ?

There are some useful command through which you can draw 3d design from a 2d . We learn every step of using 3d commands.

REVOLVE COMMAND = Here are an image of Flower vase , now you learn how to make it.



1 – Start a 2d drawing of flower vase as (image 1) .

2 – On the Surface menu, click the Revolve option.

3 – Select the (image 1) then click p1 and p2 , it will look as (image 2) press , Enter.

4 – Now you can see final (image 3).

RAIL REVOLVE COMMAND = Here are some images of star , now you can how to make it .

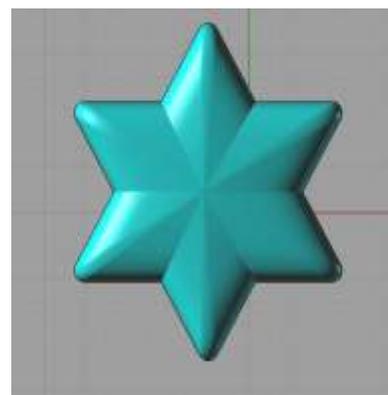


Image 1

Image 2

Image 3

1 – Start a 2d drawing of STAR as (Image 1)

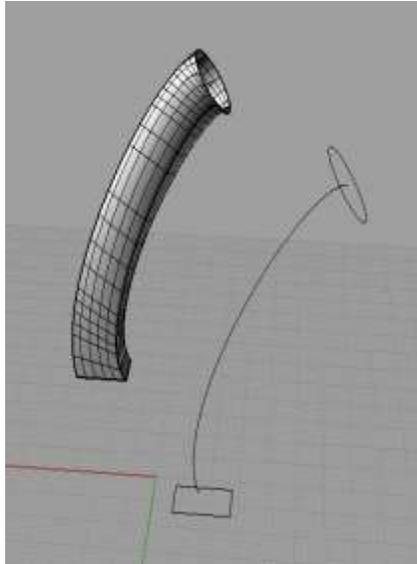
2 – Draw a curve on the star as showed (image 2) the Red line and Yellow line is a extra curve to applied the command .

3 – On the Surface menu, click Rail Revolve option.

4 – Select red line on the star . Select star then click first point an end point of the yellow line.

Now you can see the final image of STAR as (Image 3).

SWEEP 1 RAIL COMMAND = Here is an image of Pipe that has 2 different shape , now you will learn how to make it.



- 1 – Start a 2d drawing of Pipe as showed image above .
- 2 – On the Surface menu , click the Sweep 1 RAIL.
- 3 – Select the curve, then select both the curve press Enter .

SWEEP 2 RAIL COMMAND = To make a Bottle , we use sweep 2 rail command .

- 1 – On the Surface menu , click sweep 2 rail curve .

2 – Select both green curve and select both red curve , press Enter.(image 1)

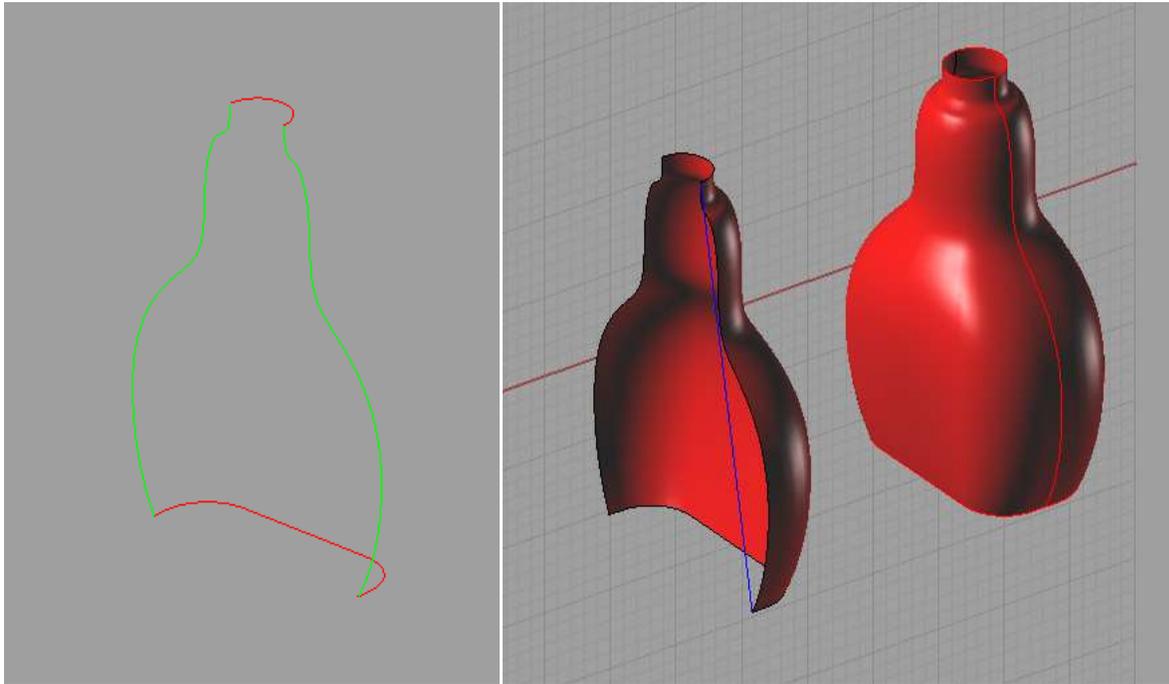


Image 1

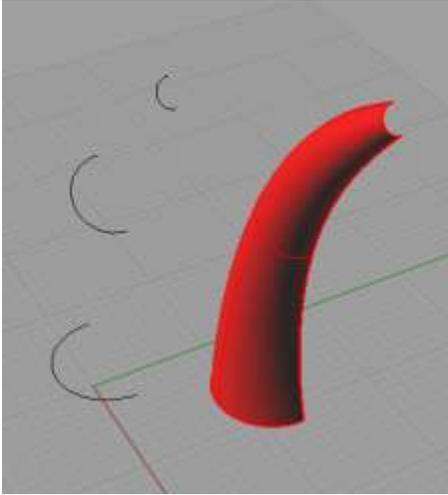
Image 2

3 – Now the half image is ready of Bottle . Make mirror of the half object , it would be a Bottle after mirror .

LOFT COMMAND = Loft command is used to make 3d from many different types of shape . You can see an image given bellow .

1 – On the Surface menu , click the LOFT option.

2 – Draw a 2d to make Loft object as shown the image.



3 – Select 2d , press Enter .

You **Annotating your model**

Rhino can also produce 2D drawings of your models.

Rhino has these annotation objects:

- Dimensions
- Annotation Text
- Leaders
- Dots
- Hatching

Dimensions

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.

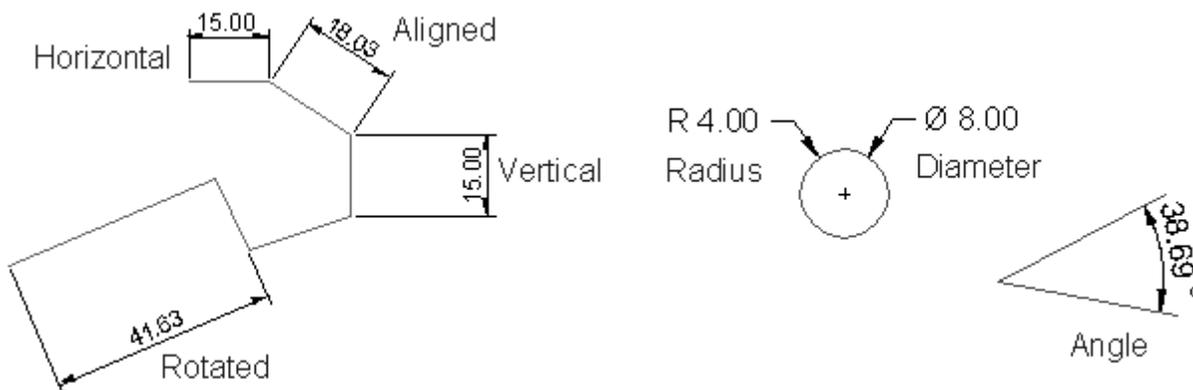
The dimension text can show decimal, fractional, or feet and inches numbers. A new model will open with Default as the dimension style.

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 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.

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.

Practice rendering a model

1 Open the model **Render**.

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.

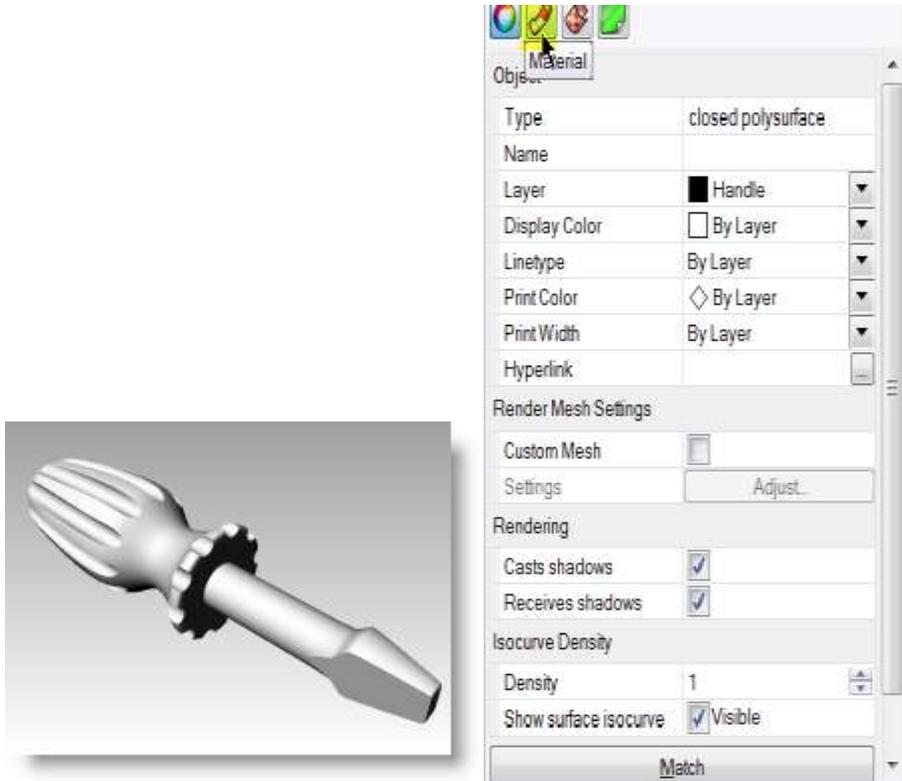
To assign a material to the handle by object:

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**.

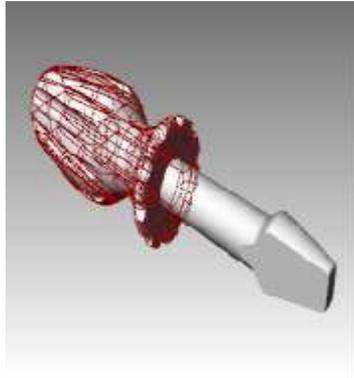
4 From the **Material** page, in the name field type **Red_glossy**.

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



Printing and Layouts

Printing

The Rhino Print command will let you print one viewport at a time.

Printing the model

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

2 Make the Top viewport active and from the **File** menu click **Print**.

3 In the **Print Setup Destination** dialog, select a printer, **Letter Size** paper, **Landscape** orientation, **Raster Output**, and **Display**

4 In the **View and Output Scale** area, select **Scale** of **2:1**, **On Paper 1.0 Millimeter**, and **In Model 0.5 Millimeter**.

Rhino will print .5 mm of the model in every 1 mm on the printed sheet.

