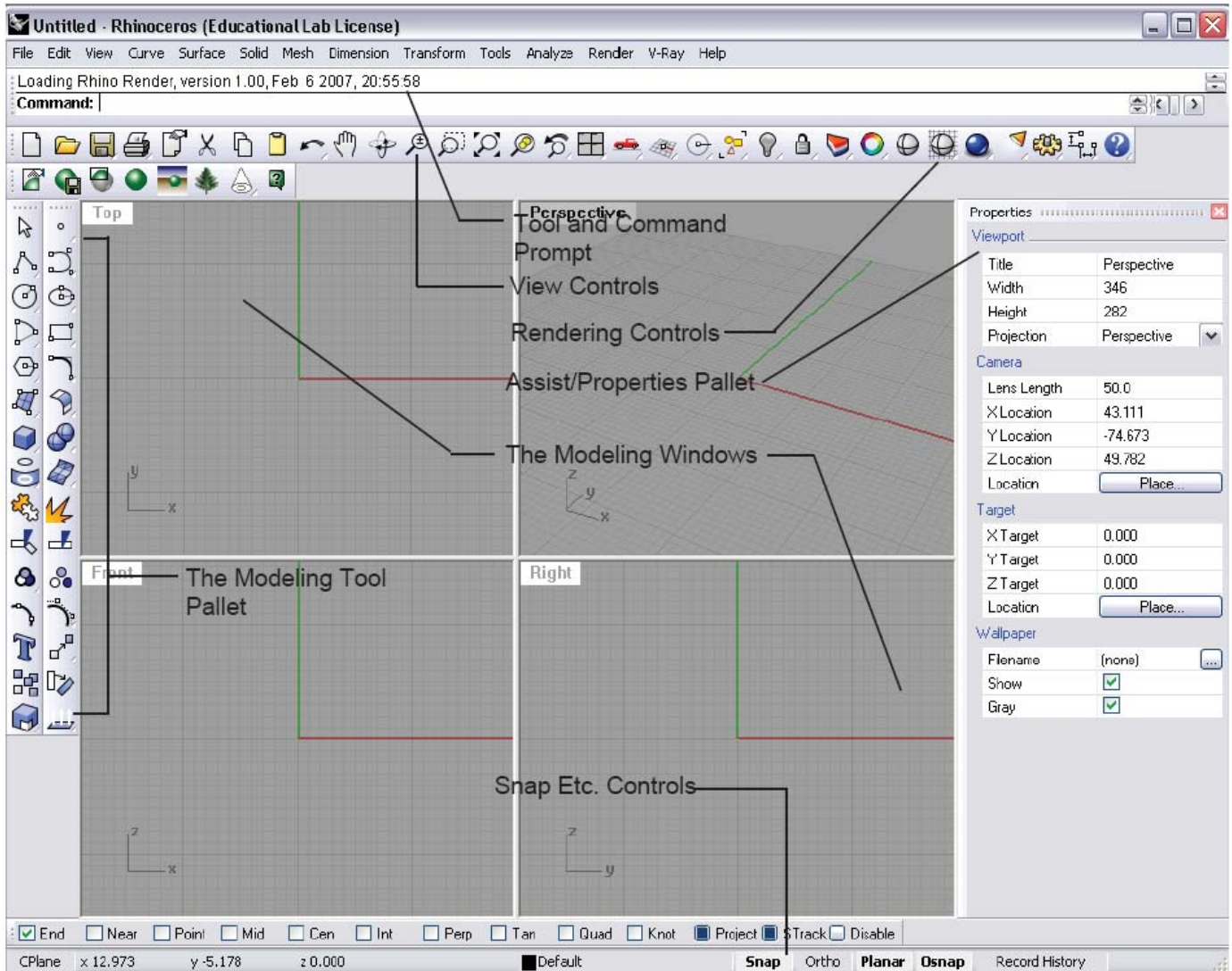ARCH 2491
Computer Generated Architectural Presentation Drawings

CUNY - New York City College of Technology
Architectural Technology Department
Instructor: Anne Leonhardt

CLASS 1

Rhino Introduction – Points, Curves, Surfaces, Solids, Booleans

Interface will not be covered explicitly in this handout.
Remember that a brief introduction to the **interface** of Rhino is available here:

Rhino Interface:
Rhino Interface/Tutorial 1
http://www.paramod.net/post043/rhino001.html

Rhino is primarily a Nurbs Modeler. Nurbs stand for Non-Uniform Rational B-Splines. This basically means that smooth surfaces can be created through mathematical formulas.
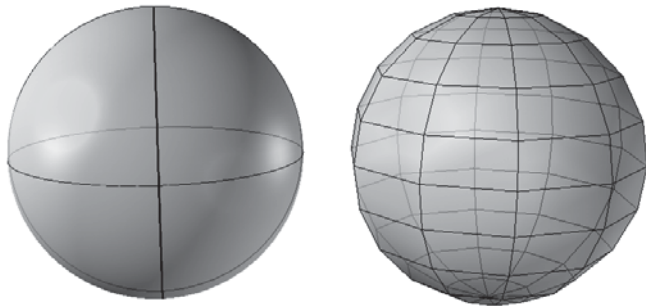
In traditional Modelers, surfaces are defined by a series of flat planes joined together to create a faceted surface.

Nurbs modelers allow you to create smooth complex surfaces by defining a couple of points that can then be manipulated.

Points

[A point shows an exact position or location on a plane surface. It is important to understand that a point is not a *thing*, but a *place*. We draw a point by placing a dot with a pencil. This dot may have a diameter of 0.2mm, but a point has no size. No matter how far you zoomed in, it would still have no width.]

Points are not very exciting in Rhino, but they are important to keep in mind as we explore the basic tools they support. Points can be placed, but do not appear when rendering.

Curves

NURBS curve knot vectors as a result of varying degree

A D^1 nurbs curve behaves the same as a polyline. It follows from the knotcount formula that a D^1 curve has a knot for every control point. Thus, there is a one-to-one relationship.

A D^2 nurbs curve is in fact a rare sighting. It always looks like it is over-stressed, but the knots are at least in straightforward locations. The spline intersects with the control polygon halfway each segment. D^2 nurbs curves are typically only used to approximate arcs and circles.

D^3 is the most common type of nurbs curve and -indeed- the default in Rhino. You are probably very familiar with the visual progression of the spline, eventhough the knots appear to be in odd locations.

D^4 is technically possible in Rhino, but the math for nurbs curves doesn't work as well with even degrees. Odd numbers are usually preferred.

D^5 is also quite a common degree. Like the D^3 curves it has a natural, but smoother appearance. Because of the higher degree, control points have a larger range of influence.

D^7 and D^9 are pretty much hypothetical degrees. Rhino goes all the way up to D^{11}, but these high-degree-splines bear so little resemblance to the shape of the control polygon that they are unlikely to be of use in typical modeling applications.

In Rhino, points can be connected up to form multi-segmented lines called curves.

The shape of the curve depends on the mathe-matical formula used to generate it. To the left you can see how the same control points generate different geometries based on the degree of the curve. The larger the degree of the curve, the greater the range of influence of the control points.

Curve Primitives – "Circle", "Arc", "Ellipse", "Parabola", "Hyperbola", "Conic", "Helix", "Spiral"

All of these curve primitives will produce their associated curves in your Rhino model and have specific options related to their function that appear in the command prompt.

Surfaces

Just like points can be connected to form lines and curves, curves can be connected to form surfaces. Surfaces in rhino can be created in multiple ways. We will go through a few now. You can enter any of the following commands without the "" quotes in order to initiate the command.

"Plane" command:

Command: Plane
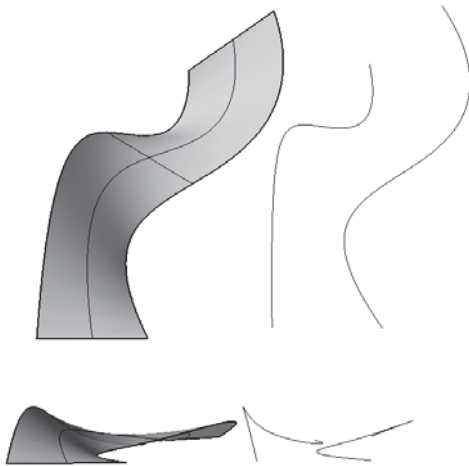**First corner of plane** (3Point Vertical Center Deformable):

Creates a planar surface element based on the coordinates entered.

"PlanarSrf" command:
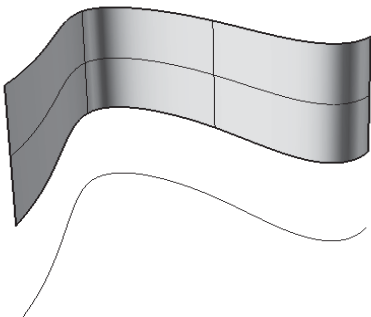Creates a planar surface element based on a collection of planar curves.

"Loft" command:
Loft creates a surface between two or more curves that act as control geometry. Lofts can be planar (flat) or have 3D contours.

"ExtrudeCrv" command:
Allows a surface to be extruded up from a profile curve

Solids

In Rhino, surfaces can be joined together to form solids. Surfaces that have been joined to each other are called polysurfaces. There are many methods to creating solids, and here we will cover a few of them.

Solid Primitives – Box, Sphere, Cylinder, Cone, Truncated Cone, Pyramid
All of these commands will produce their related solid primitive. Check the command prompt for options related to each specific command.

"ExtrudeCrv" command:
Solids can also be created from a closed curve. Create a "Rectangle" and then "ExtrudeCrv." If the capped option is active, the extrusion will have end caps as shown on the left. If the capped option is set to "no" then the extrusion will be open, as shown on the right.
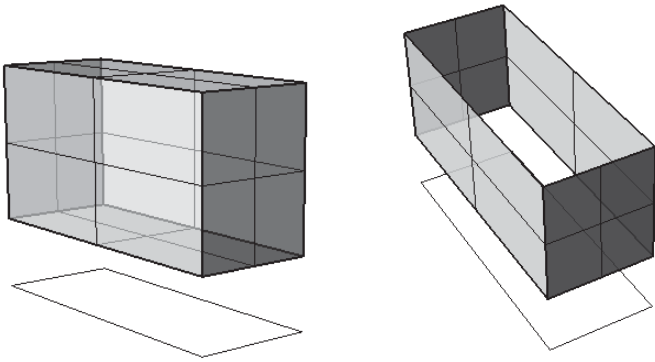
"Slab" command:
Slab creates a "thickness" and "height" from a curve. You will be prompted to enter the thickness, the side where the offset should go, and the height all at the command prompt.
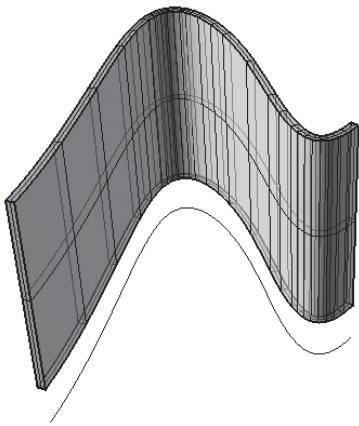
"Join" "Group" and "Explode" commands:
Make a box and "Explode" it. Notice that it is all separate planes. Select all of the planes and "Join" them. Now they are together again. Note that "Join" is about geometry grouping. If you "Group" the objects, they will all select together, and *look* exactly like a solid but they are NOT a solid, and you can NOT perform solid editing commands on a group.

"ExtractSrf" command:
Allows you to remove a surface from a joined object without exploding it. Enter the command, pick the surface you want to extract, and hit enter. The surface will un-join the original solid.

"Cap" command:
Takes an open surface with planar holes and caps them creating a solid object. Extrude a rectangle with the caps option set to "no". Select the resulting surface and enter the "Cap" command. The open rectangular polysurface should close into a solid.

"BooleanUnion," "BooleanDifference" "BooleanIntersection"
Create a sphere and a box and intersect them. Now let's Boolean these objects using each of the three commands.

|			
BooleanUnion	BooleanDifference	BooleanDifference2	BooleanIntersection
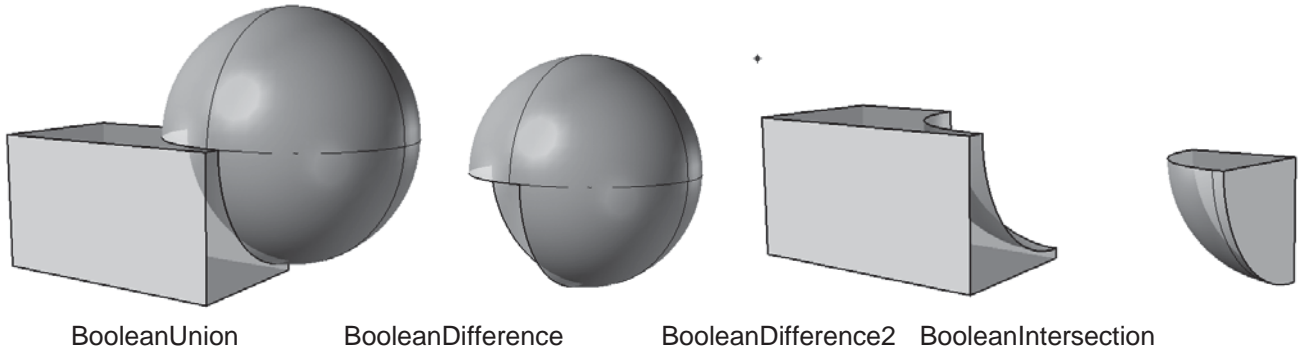
Remember that with BooleanDifference, the object you choose FIRST is the object that is kept, and the object you choose SECOND is the object that is carved out.

Object Manipulation

"Rotate 3D" command:

In order to rotate objects in 3D space you must use the rotate3D command. Execute the command then choose your objects to rotate. Choose the axis of rotation by specifying the first and second points to rotate around. This line WILL NOT MOVE. The object will rotate AROUND it. Imagine the earth. It spins in space, but at the absolute north pole, and the absolute south pole, the coordinate do not change, only the orientation. The first line you are drawing is this axis of spin. You will then choose a point which will determine the reference angle for rotation. The last point you pick will determine the angle of rotation.
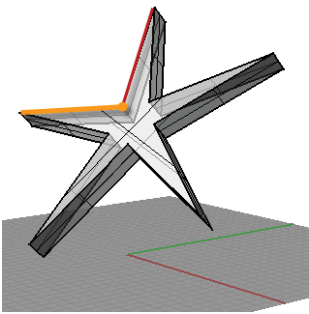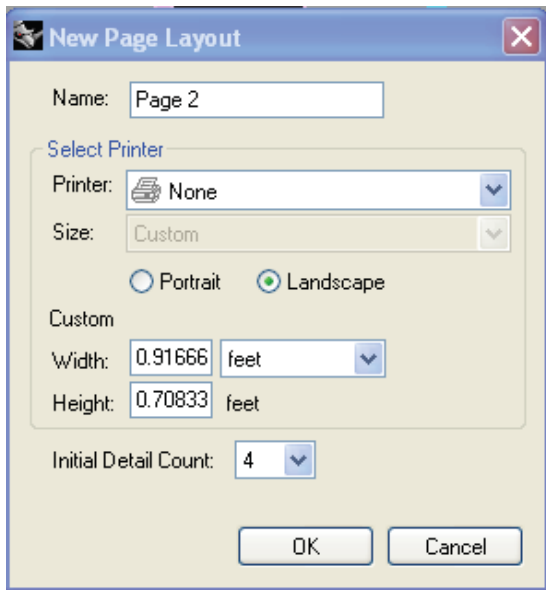

Orient Command

To **orient** an object to another: moves, copies, rotates and scales the object using two reference and two target points.

1. Select one object.
2. Pick two reference points on the object.
3. Pick two target points on the second object: the first and second reference points will match these locations.

Constructing Page Layouts for Printing, Background colors, and Orienting objects

1. Select View > Page Layout > New Page Layout
2. In the New Page Layout dialog, choose printer (or None), set the paper size to 11 x 17" landscape, set the Initial Detail Count to 4, and click on the OK button. The four viewports open at this time in your drawing will appear in the layout viewports.

New Page Layout

Name: Page 2

Select Printer
Printer: None
Size: Custom
○ Portrait   ● Landscape

Custom
Width: 0.91666 feet
Height: 0.70833 feet

Initial Detail Count: 4

OK   Cancel

The Initial Detail Count refers to the number of detail viewports that will be constructed automatically by the system. If you choose 0, you will have a blank page layout.

3. To manipulate the viewports, activate it by double clicking inside it: you can rotate, pan, zoom, and orbit, or by choosing the "Select View" icon, change to a different view.

4. To lock a detail, so the selected view is not altered, select the edge of its box (it will turn yellow), go to the Properties list, select detail, and in the dialogue box and check "Lock".

5. To add shading to a viewport, double click on the viewport to enable it. Then left click on the shaded viewport command icon.
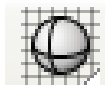
Background color

Background colors for the screen are found in the Options > Appearances > Background color

Rhino 3D an Overview

New York City College of Technology

Rhino is designed as a Nurbs Modeler. Nurbs stand for Non-Uniform Rational B-Splines. This basically means that smooth surfaces can be created through mathematical formulas. What this means to us is that complex smooth forms can be created that are highly adjustable and mailable for the designer. In traditional Modelers, surfaces are defined by a series of flat planes joined together to create a facetted surface. Nurbs modelers allow you to create smooth complex surfaces by defining a couple of points that can then be manipulated. However, when it comes to simple modeling, Rhino is a very easy to use and a highly intuitive modeler.

Rhino 3D Basic Modeling Tools:
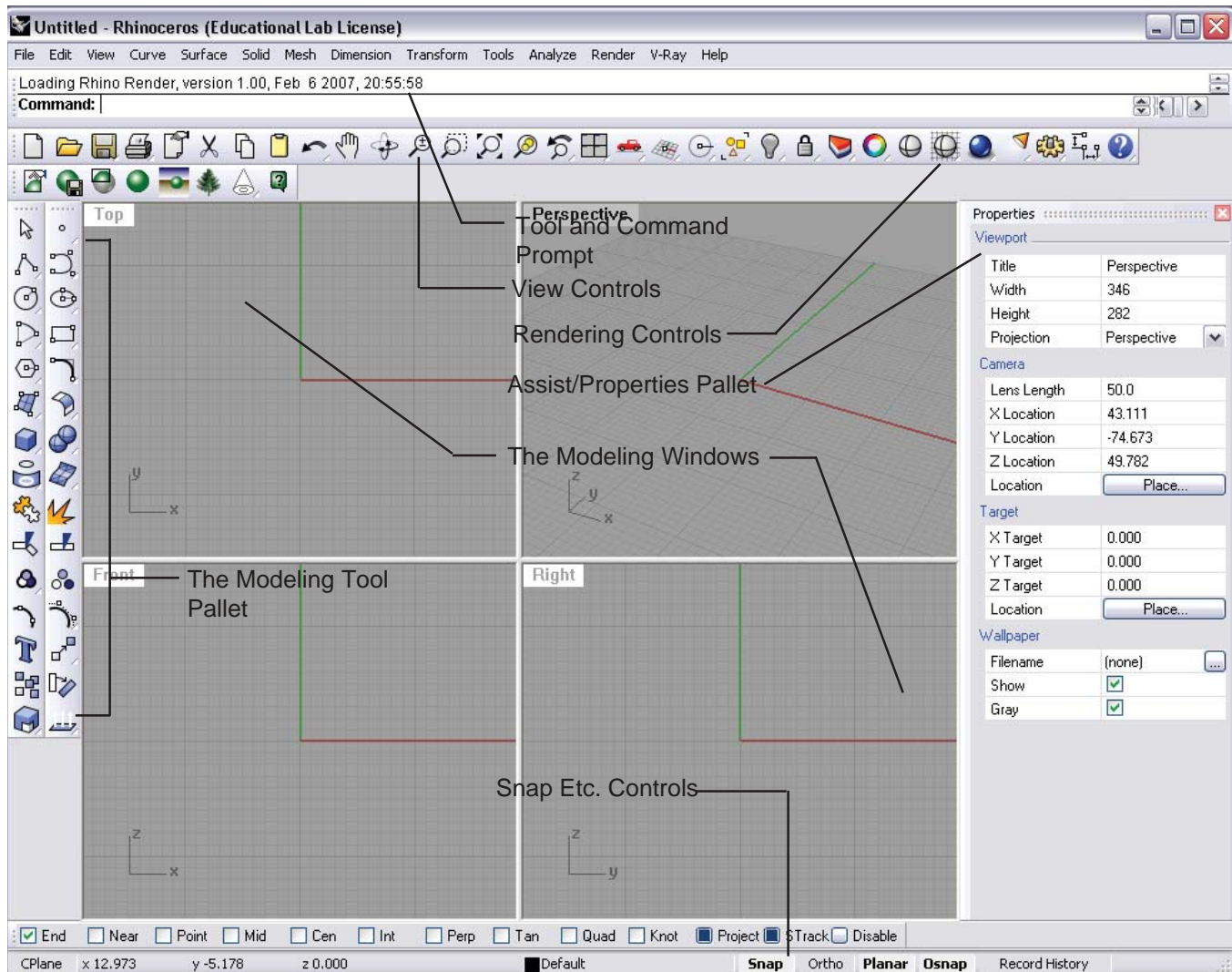
Rhino has a typical Modeling interface. By default it is composed of four view points and a tool palette on the left. At the bottom are the ortho and object snap controls. At the top are the Pan and Zoom Controls, as well as the tool command prompt.

Setting up grids and units for accurate modeling:

Click on **File>Properties** to open the settings window.

Click on the units Tab, and select either Millimeters or Inches. Match your Unit setting with the Distance Display

Click on the Grid Tab, and set a grid snap and grid spacing. Note: if you are using Feet and inches a grid unit of 8 is best. If you are using metric, then a grid unit of 10 is best.
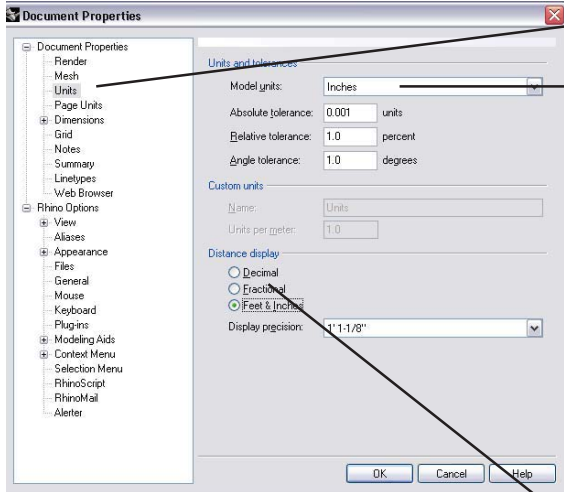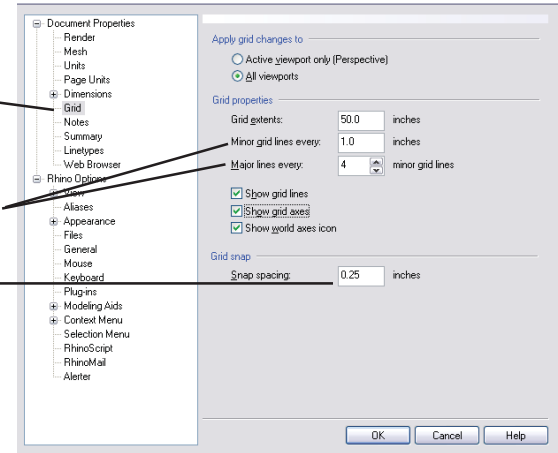

Units Tab

Units Setting

Grid Tab

Major Grid Lines
Match your units

Snap Spacing

Display Setting

Snap, Ortho, Planar, OSnap, Record History, and S-Track:

The bottom of the Rhino interface is a number of drawing aids. These allow you to lock lines and Snap as needed.

Project: Will project line snapped to geometry onto the ground plane, useful for creating plan outlines.

S-Track: Allows you to snap projection line from multiple types of geometry.

Disable: Disables the O-Snap functions.

Record History: Allows you to link lines to surfaces. If you change the line, it will update the surface (Kind of Works)
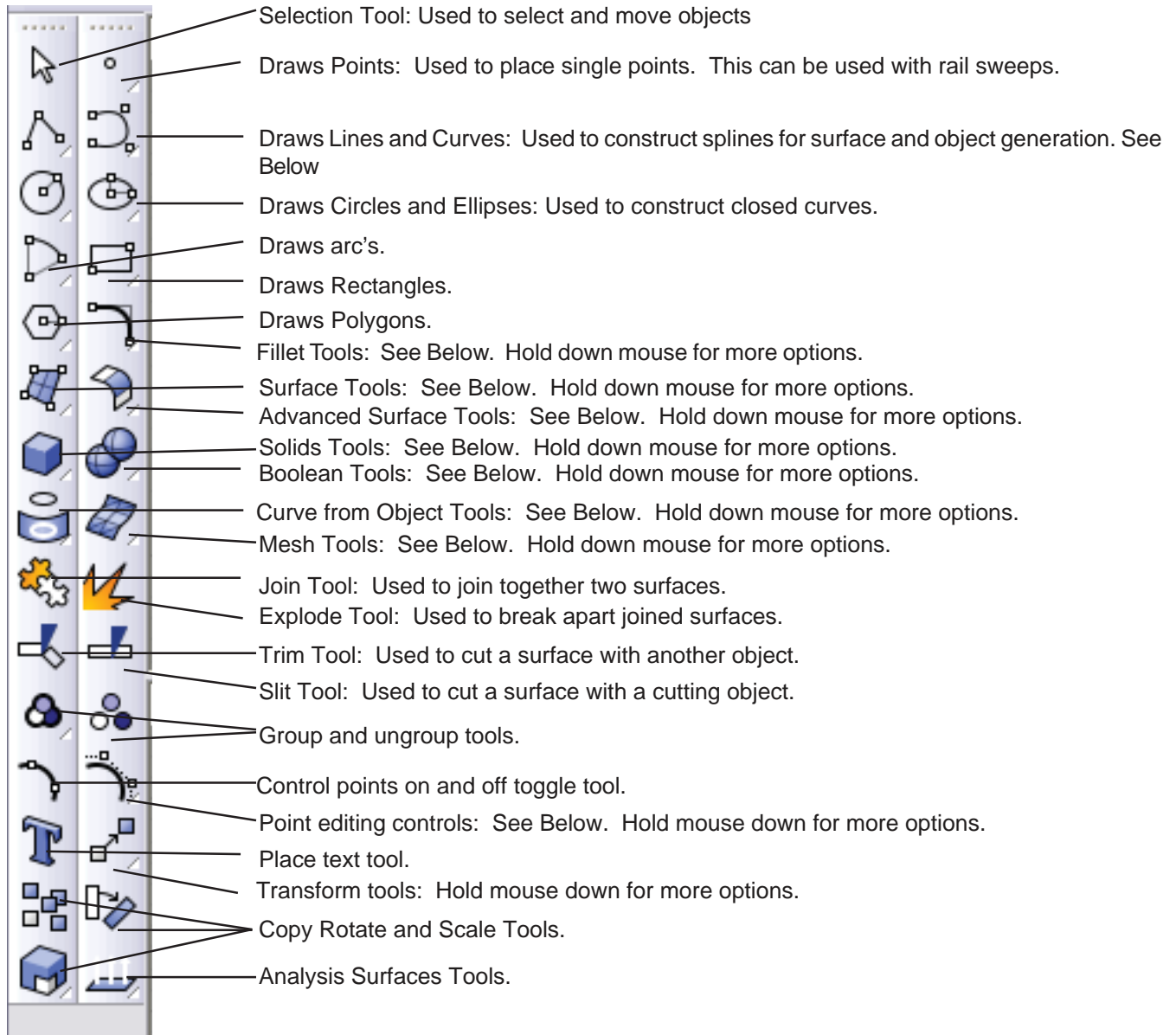
Osnap: (Object Snap)Activates the snap menu. Will snap lines to geometry as you draw.

Planar: Keeps curves in a set plane as you draw from one view to another.

Ortho: Locks lines to 90 Deg. Hit F8 to toggle or hold down shift key while you draw.

Snap: Toggles the grid snap on or off that we just set.

The Tool Palette:

Selection Tool: Used to select and move objects

Draws Points: Used to place single points. This can be used with rail sweeps.

Draws Lines and Curves: Used to construct splines for surface and object generation. See Below

Draws Circles and Ellipses: Used to construct closed curves.

Draws arc's.

Draws Rectangles.

Draws Polygons.

Fillet Tools: See Below. Hold down mouse for more options.

Surface Tools: See Below. Hold down mouse for more options.

Advanced Surface Tools: See Below. Hold down mouse for more options.

Solids Tools: See Below. Hold down mouse for more options.

Boolean Tools: See Below. Hold down mouse for more options.

Curve from Object Tools: See Below. Hold down mouse for more options.

Mesh Tools: See Below. Hold down mouse for more options.

Join Tool: Used to join together two surfaces.

Explode Tool: Used to break apart joined surfaces.

Trim Tool: Used to cut a surface with another object.

Slit Tool: Used to cut a surface with a cutting object.

Group and ungroup tools.

Control points on and off toggle tool.

Point editing controls: See Below. Hold mouse down for more options.

Place text tool.

Transform tools: Hold mouse down for more options.

Copy Rotate and Scale Tools.

Analysis Surfaces Tools.

Most of the items in the tools palette can also be accessed in the menus palette. In the Menus palettes there are also more specific tools associated with its menu item not listed in the tools palette,

Curve Tools are used to adjust and fix existing curves. These include Fillets, Chamfers, Off Sets, Fix Points, Simplify and Rebuilding Curves.

Surface Tools: This palette is used to generate surfaces from Points and Curves. Lofts, Surface from Curve Network, Sweep Along one and two rails, Revolve, Patch, and Drape Functions.
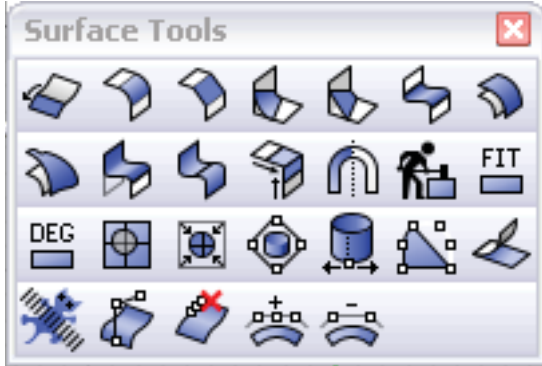
Surface editing tools: These are used to edit, adjust, fix and rebuild surfaces.


Solid Tools Pallet. This is a typical pallet of object primitives that can be created.

Solid Editing Tools: These includes difference, union, intersection. As well as fillet, cap holes, and extract surface.

Curve From Object Palette: This is used to generate curves from objects that then can be used to generate more surfaces. You can also generate a series of contours from an object, section curves, etc.
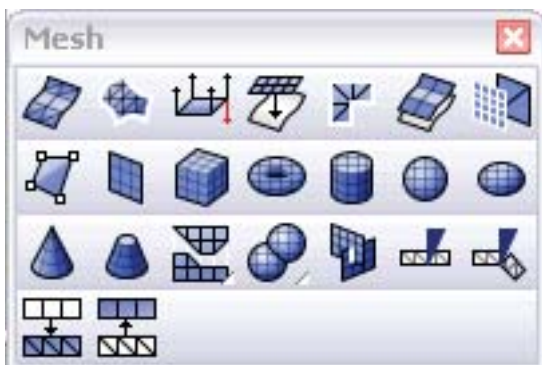
Mesh Tool Palette. This is used to extract mesh surfaces and objects from nurbs surfaces. This is useful for exporting into other programs. You can also create direct mesh objects without creating a nurbs object first.

Creating Objects Using Boolean Commands:
To create an object in Rhino, click on one of the Solids from the solids tool pallet:
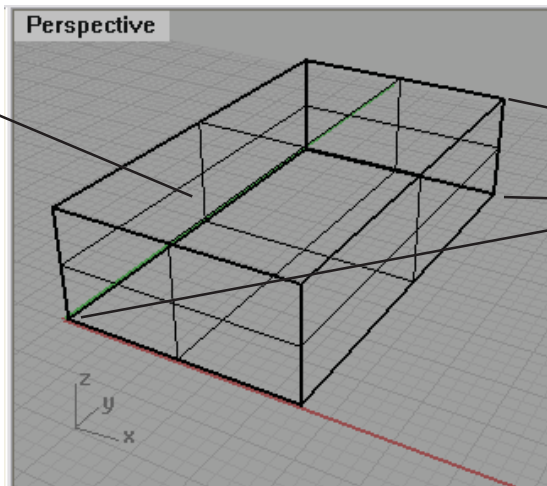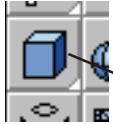
Perspective

This box is defined by two points and a height.

Height

Two base points

Note, use multiple windows to create your box. Define the base points in either the top view or the perspective view, then activate ortho at the bottom of the modeling window, and then extrude the box in either front or side view.

Creating complex objects in Rhino is easy to do using the boolean operations. Simply build the object you want to cut and place it in the correct location relative to the object you want to keep.
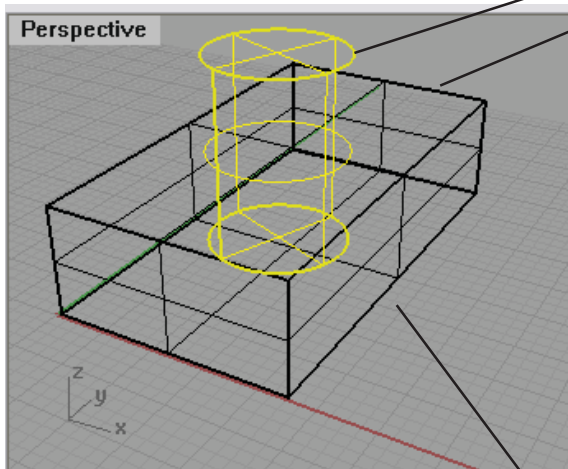
Object to cut (Shown High lighted)

Object to Keep

Perspective

Deselect all objects, and click on Solid, then Difference. Note in this same menu you have Union and Intersection as well.

Click on the object you want to keep, first. Hit return, or right click. Then click on the object or objects to want to cut from with. Hit return or right click again.

Solid	Mesh	Dimension	Transform
Box	▶		
Sphere	▶		
Cylinder			
Cone			
Truncated Cone			
Pyramid			
Ellipsoid	▶		
Paraboloid	▶		
Tube			
Pipe			
Slab			
Torus			
Text...			
Extrude Planar Curve	▶		
Extrude Surface	▶		
Extrude Surface to Boundary	▶		
Boss			
Rib			
Fillet Edge	▶		
Cap Planar Holes			
Extract Surface			
Union			
Difference			
Intersection			
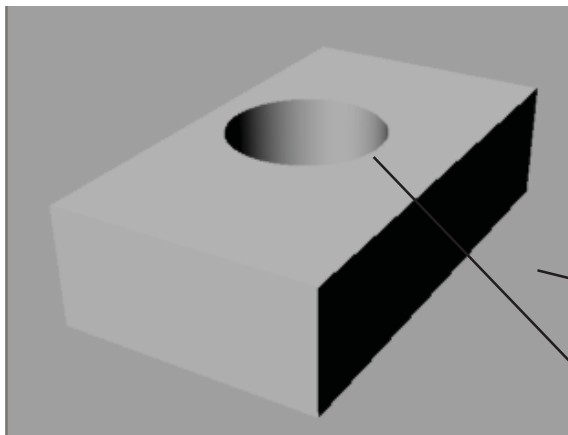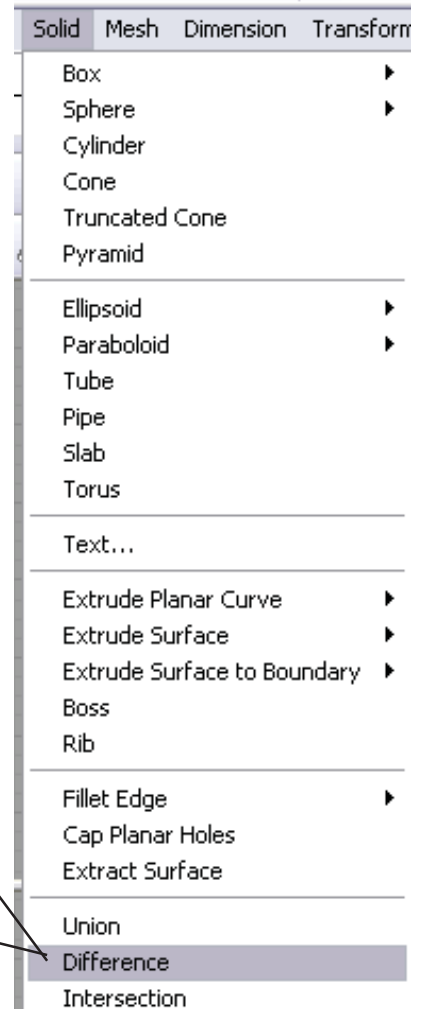
Cut Object

Adjusting Surfaces by Editing its Control Points:

Along with normal objects, you can adjust "blobs" Start with a
sphere. Click on the solids menu>Sphere>Center, Radius.
Click in the screen and draw a sphere.

You can now move the control points that define the sphere
by clicking on the point editing tool. With the sphere
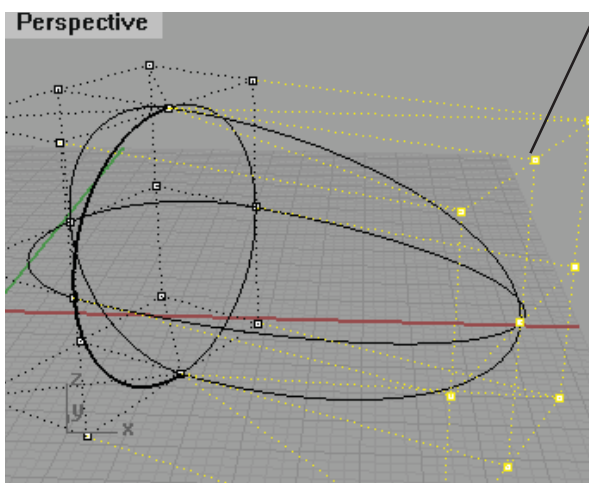selected, click on this tool to turn on point editing.

Now in plan or front view select a series of points. Then
click and drag the points stretching out the sphere.

Note: This will not work with rectangular solids, but will work
with spheres and surface.
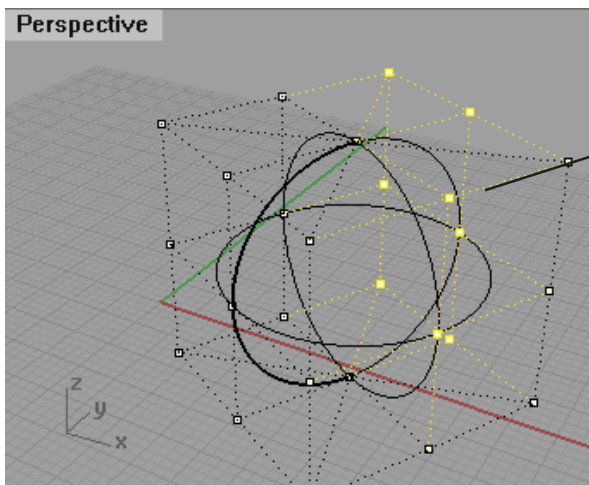
Control points on, and some points
selected:

Selected points moved and sphere is
deformed

Rendered Sphere.
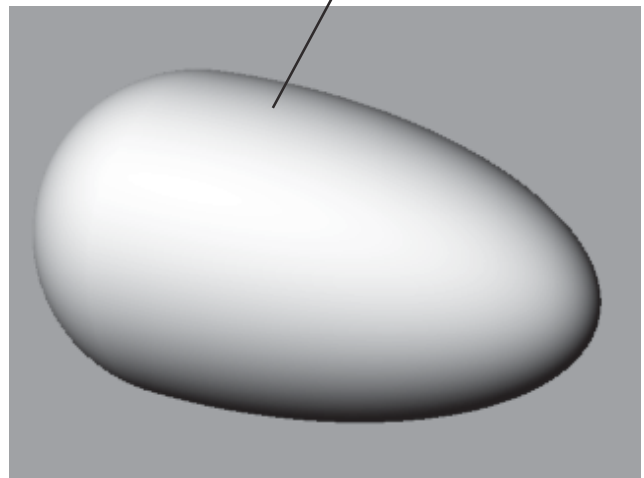
This process can be applied to almost any Nurbs surface.

Visual Studies I: Arch 1191

Assignment Number: 9

Computer Program(s): McNeel Rhinoceros Cont'd

Student Learning Objectives:

Upon successful completion of this assignment, the student will:

1. Understand how to model in 3D a simple geometric object in McNeel Rhinoceros (Rhino).
2. Know how create a Rhino model file.
3. Know how to toggle between the four standard view points.
4. Know how to set up units.
5. Know how to set up a grid.
6. Know how to use the solid tools palette.
7. Know how to use Boolean operations (addition, subtraction) in order to create more complex forms.
8. Know how to use more complex modeling tools: surfaces, meshes, etc.

Assessment:
To evaluate the student's achievement of the learning objectives, the professor will do the following:

1. Evaluate the student's understanding McNeel Rhinoceros by examining both the digital file and the hard copy output.

Project Description:
In this assignment, you will model in three dimensions digitally your assignment 9 from Arch 1110, Extruded Geometries.

Process:

1. Create a new file in Rhino.
2. Choose units, inches and match units with Distance Display.
3. Choose grid tab, and set grid snap and grid spacing: a grid of 8 is best when working in inches.
4. Choose one view port to begin your drawing.
5. Draw the 48' square that is the limits of your drawing.
6. Reconstruct your chosen square design in Rhino, Autocad, or import a raster of your design and trace over it.
7. Create the solids as defined by the rules of shape generation:
   a. At point/intersections, create a column, either a 2 foot diameter cylinder or a 2 foot square prism centered on the point/intersection.
   b. At single lines, walls are extruded and must be 2 feet wide and centered over the defining line. The length should be one foot more at each end of the line.
   c. Volumetric elements are defined by some combination of lines, both straight and curved, that enclose an area.
8. Use Boolean Operations to create volumetric elements:
   a. Volumetric elements are defined by some combination of lines, both straight and curved, that enclose an area.
   b. Positive volumes are defined by extending one foot past the defining lines.
   c. Negative volumes are defined by extending one foot inside the defining lines.
   d. Maximum height of any positive element is 24 feet above grade.

e. Negative volumes may not extend more than 4 feet below grade.
f. All vertical dimensions must be in one foot increments.
g. The tops and bottoms of all extruded elements must be flat horizontal surfaces.
h. Create a minimum of two cases of intersecting elements of different heights.
i. See additional rules in Arch 1110's Assignment 9.

9. Complete three dimensional model of Extruded Geometries design.
10. Create page layout for printing:
    a. Select view>Page Layout>New Page Layout.
    b. Choose printer.
    c. Choose page size: 11x17.
    d. Choose orientation.
    e. Set initial Detail Count to 4, hit OK. Detail count is equal to the number of viewports in Autocad. Note Autocad will automatically create a viewport for each tab.
    f. Manipulate viewports:
        i. Double click inside the viewport.
        ii. Rotate, pan, zoom, and orbit or use the select view icon.
        iii. Lock chosen view.
        iv. Shade chosen view.
        v. Choose background color.
11. Render your composition:
    a. Set the renderer to Rhino renderer.
    b. Choose resolution
    c. Background color
12. Export images: screen capture or export file type.
13. Import images into Adobe Illustrator.
14. Format images with title block.
15. Print hard copy. What are the differences between working in three dimensions by hand and working digitally? Pros and cons of each?

Rhino 3D - Advanced Features

New York City College of Technology

This tutorial covers a series of advanced modeling features. Some of the features are very specific to Rhino. Others will appear as variations of tools in other modeling programs.

Lofting a Surface: The term lofting come from the ship building industry. This is the method of placing a surface over a ship skeleton. In Rhino you generate a lofted surface over a series of curves spaced out. To loft a surface, create a series of splines, and arrange them to define the soon to be formed surface.
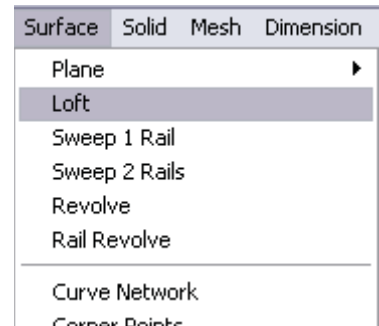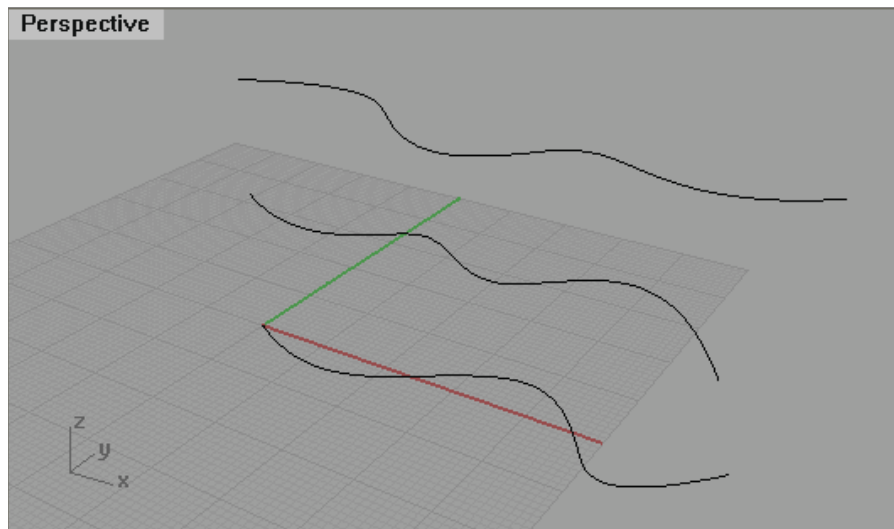
Select Surface>Loft. At the command prompt, click on the splines in the correct order. Click Enter or Right Click when done. The following window will appear: (Note, click to one side of the lines to set the direction of the loft)
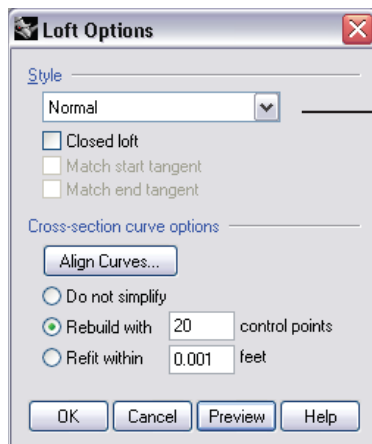
The following Loft Options box will appear.

Under the Style drop down you will have a number of loft options.
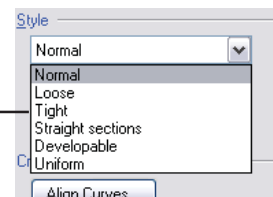
Normal: The most mathematically efficient loft
Loose: Will create a smoother loft that is not based exactly off of the loft curves, but averaged.
Tight: Constructs a loft the is exactly on the various loft curves
Straight Sections: Will create a series of ruled surfaces between loft lines
Developable: Will create lofts that can be flattened out on a single sheet.
Uniform: Guarantees that every control point affects the surface exactly the same way.

Loft Options

Style
Normal

☐ Closed loft
☐ Match start tangent
☐ Match end tangent

Cross-section curve options
[Align Curves...]

○ Do not simplify
◉ Rebuild with 20 control points
○ Refit within 0.001 feet

[OK] [Cancel] [Preview] [Help]
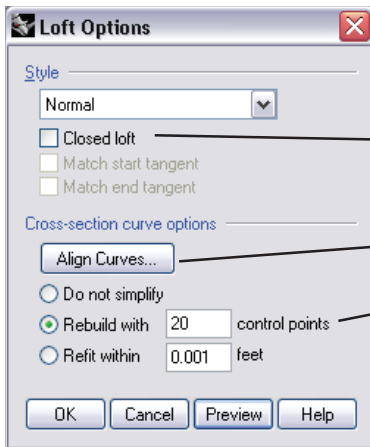
Selecting this will create a closed lofted surface, check and click preview to see.

Align Curves: This is used to flip the direction of a curve.

Rebuild With XX Control Points. This allows you to set a number of curve divisions refining the smoothness of the surface.

Set the loft to rebuild with 20 points and click Preview to see the surface. If you like it, click OK.

Perspective
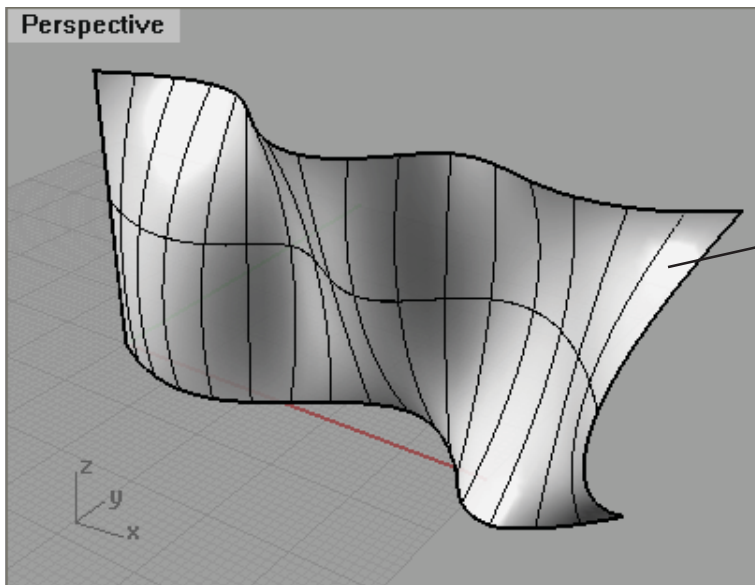
The Lofted Surface

Rebuilding Surfaces: Often after you create a curve or surface, you will need to rebuild it. Rebuilding a curve or surface changes the number of points in a curve or the number of points in a profile curves used in creating a surface. Each point then has a line (Iso-curve) running through it. Rebuilding a curve or surface changes the number of points in each line. This will in effect make the curve or surface smoother (reducing points) or more flexible (increasing curves)

Edit	View	Curve	Surface	S
Undo			Ctrl+Z	
Redo			Ctrl+Y	
Undo Multiple...				
Redo Multiple...				
Cut			Ctrl+X	
Copy			Ctrl+C	
Paste			Ctrl+V	
Delete			Del	
Select Objects			▶	
Control Points			▶	
Visibility			▶	
Groups			▶	
Blocks			▶	
Layers			▶	
Join			Ctrl+J	
Explode				
Trim			Ctrl+T	
Split			Ctrl+Shift+S	
Rebuild				
Change Degree				
Adjust End Bulge				
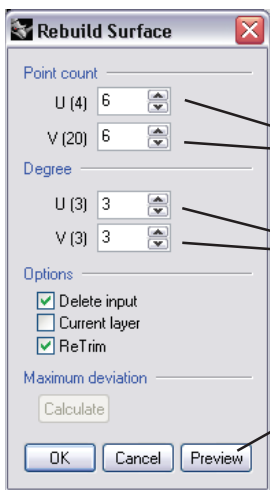
Select a surface or curve, then click on Edit>Rebuild

Rebuild Surface

Point count
U (4) 6
V (20) 6

Degree
U (3) 3
V (3) 3

Options
☑ Delete input
☐ Current layer
☑ ReTrim

Maximum deviation
[Calculate]

[OK] [Cancel] [Preview]

Change the number of points in both the U and V Directions

Keep the Curve Degree to lower than 3

Click Preview to preview the new Curve or Surface. Click OK to rebuild.
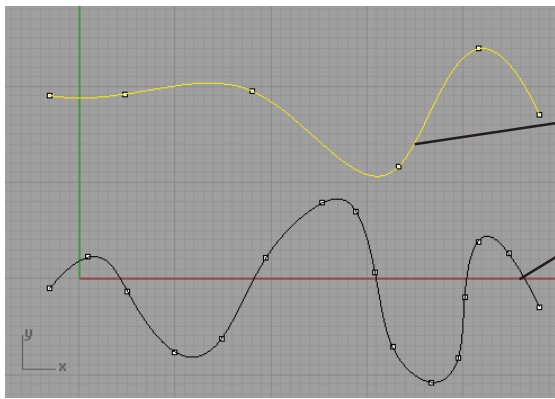
Rebuilt Curve with only 6 control points

Original Curve with 16 Control Points
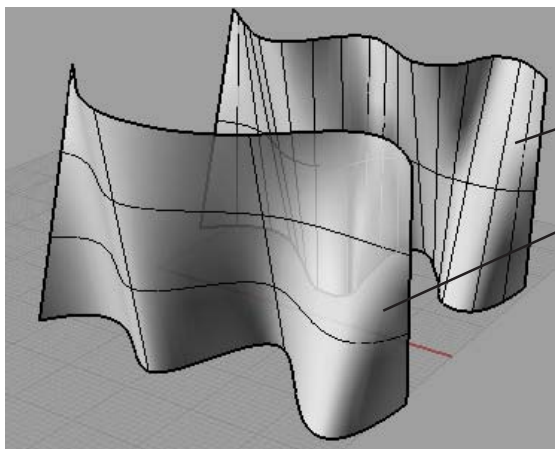
Original Surface with 20 Control Points

Same Surface "Rebuilt" with only 6 control points

Trim and Split:

When dealing with complex surfaces, you will most likely need to cut holes and trim surfaces. This can be done with the boolean tools, but there are times when they do not provide the level of detail that the trim and split tool offer.

Trim and Split is basically the same tool, the only difference is that the trim tool will delete the piece being cut. Split will just cut the objects.

Trim
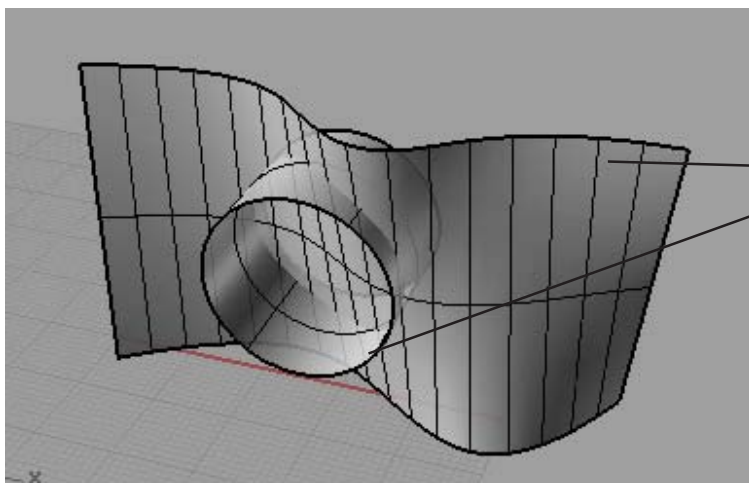
Split

In this example, I have placed a cylinder through a lofted surface.

Click on the Split Tool.

At the prompt, Click on the lofted Surface, this is the object to be split.

Click Enter.

Then Click on the Cylinder. This is the object to cut.

Click Enter to split the object

The Object Split (Note: I deleted the cylinder)
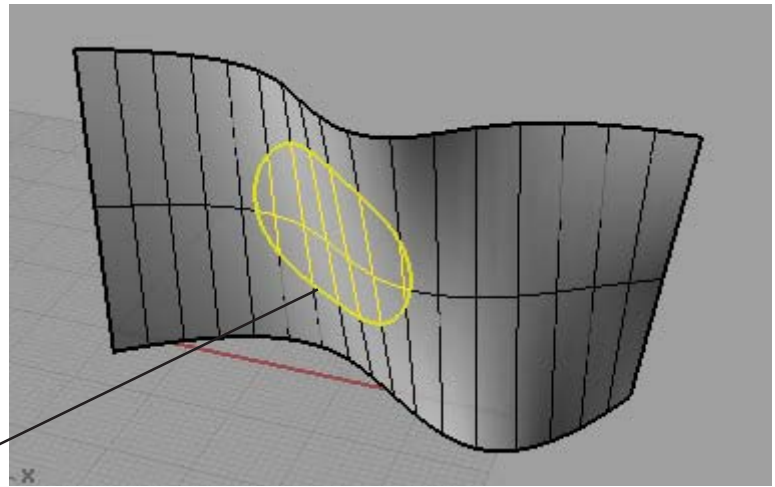
The Trim Tool will automatically delete the cutting object. Also in this example, I just have the circle shape. I do not need to extrude a cylinder through the lofted shape.

Click on the Trim Tool

At the Prompt: Select Cutting Objects, choose the Circle and hit Enter.

At he Prompt: Select object to Trim, select the lofted surface

NOTE: Do this through the Front Elevation, otherwise the lofted shape would be cut through the perspective view.
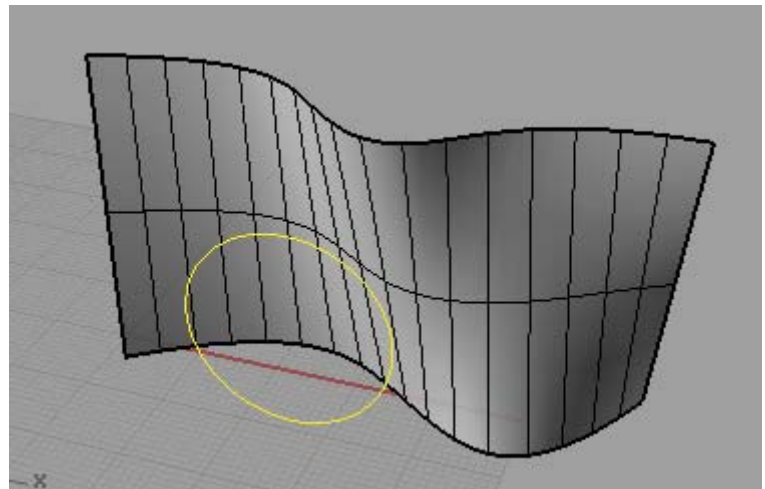
Note: If you click on the lofted shape inside the circle, the inside circle will be deleted. If you click outside the circle, the outside will be deleted.
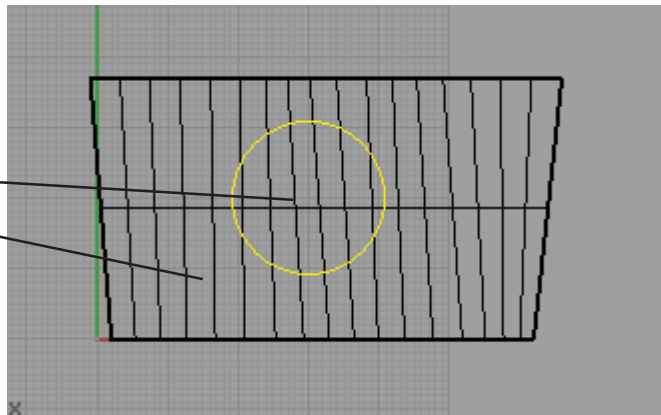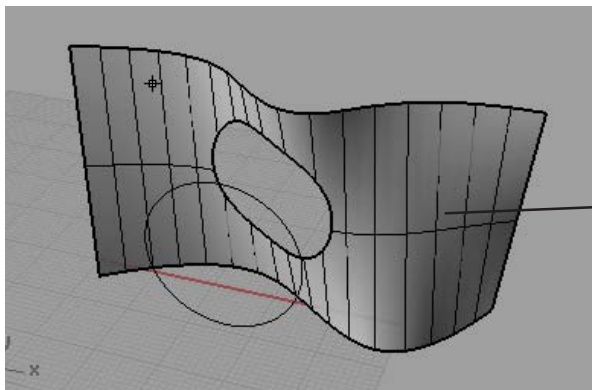
Trimmed by clicking inside the circle
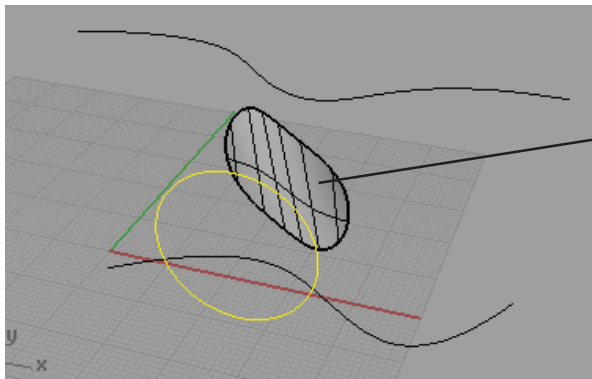
Trimmed by clicking outside the circle

Patching a Surface: Patching a surface is used to create a uniform surface from a series of closed curves. The most common use for this tool is generating terrain models.
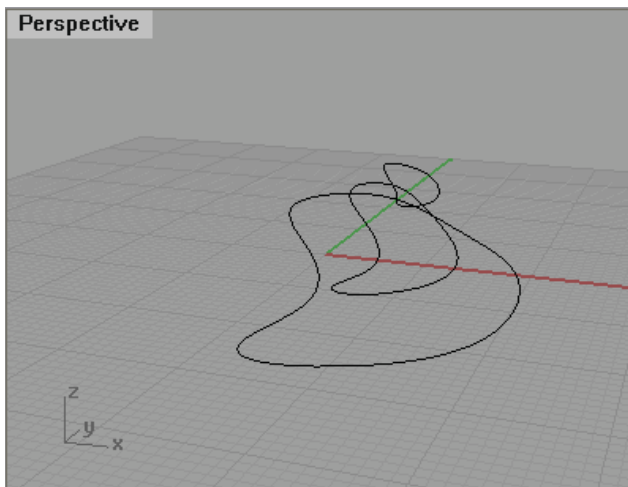
Select Surface>Patch

At the Patch prompt, click in order the closed curves to be patched.

Surface	Solid	Mesh	Dimension
Plane			▶
Loft			
Sweep 1 Rail			
Sweep 2 Rails			
Revolve			
Rail Revolve			
Curve Network			
Corner Points			
Edge Curves			
Planar Curves			
Point Grid			
Extrude Curve			▶
Patch			
Drape			
Heightfield from Image			
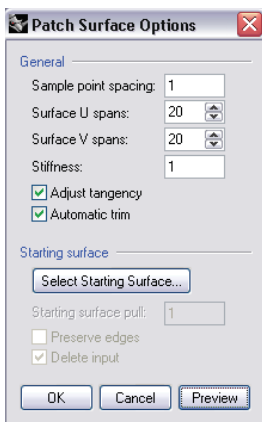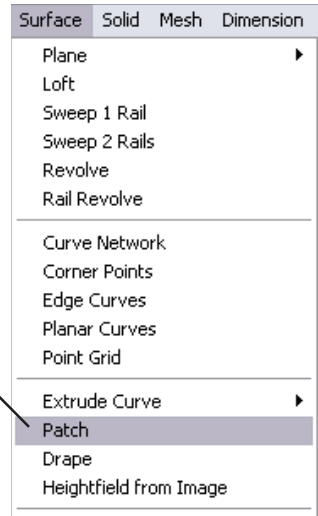
Adjust the setting to get the desired result. Click on the preview button to see a preview of the patched surface.

Click OK to generate the patched surface.
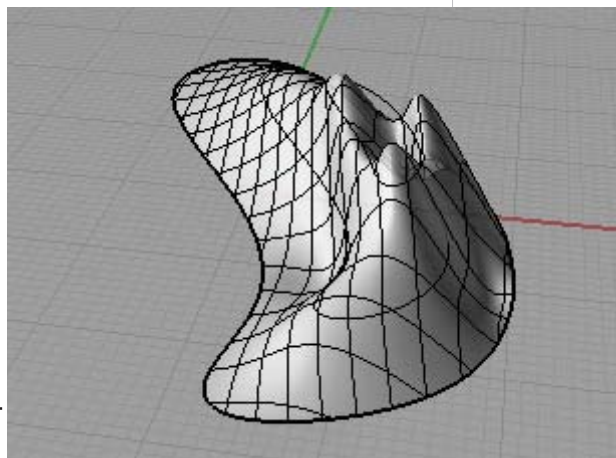
Duplicating and Projecting Curves onto Surfaces:

You can generate curves from existing geometry and then project that onto other geometries.

In this example, I have added a plane to the model with the trimmed lofted surface. I am going to project the holes onto the surface.

Surface to project the curve onto

Lofted Surface with the hole
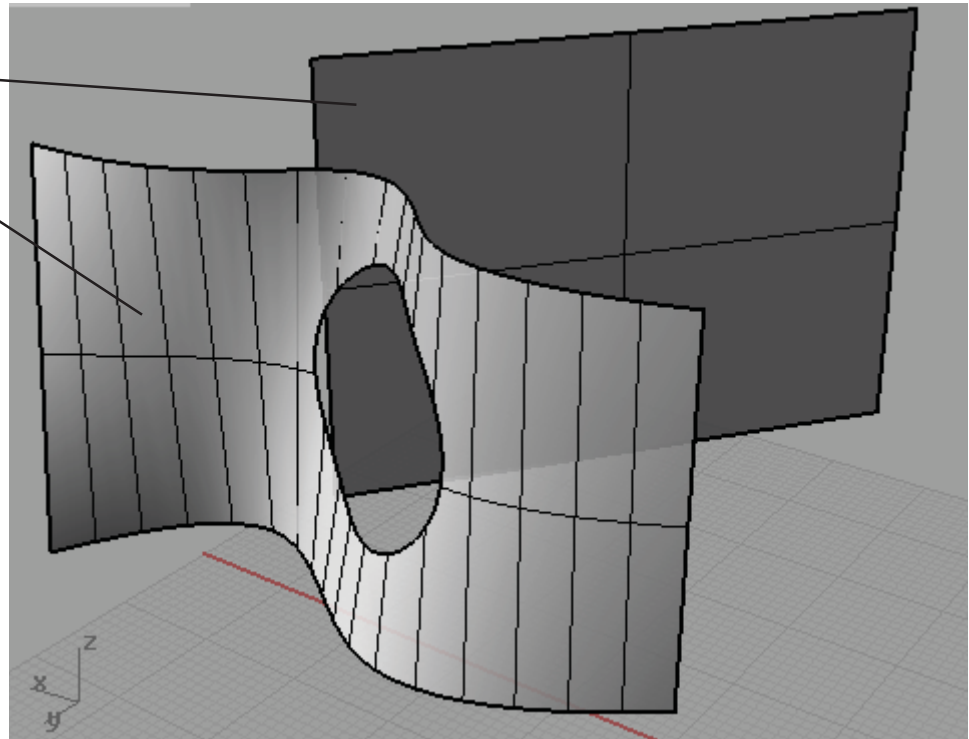
Step One: Duplicate the Edge of the hole. Click on Curve>Curve From Objects>Duplicate Edge
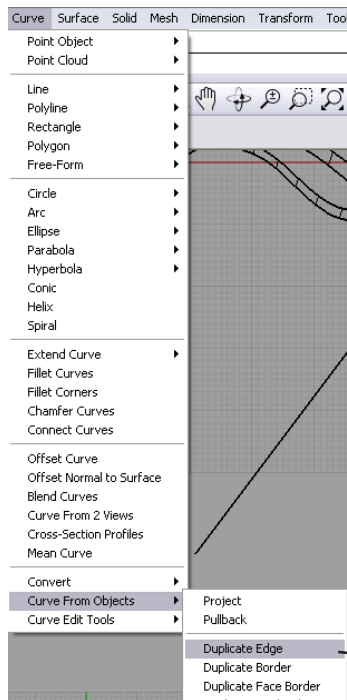
At the Command Prompt, select the edge to duplicate. Note: Sometimes you will have to click on many segments of an edge to duplicate. If this is true, you will also need to join the segments after they are duplicated into one singe curve.
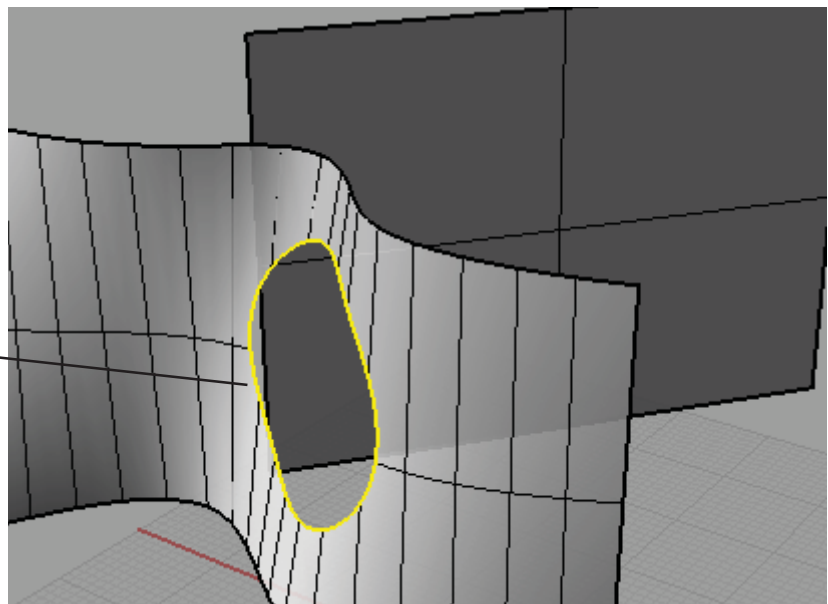
Step Two: Project the duplicated edge curve onto the surface.

Select Curve>Curve From Objects>Project.

At the Command Prompt, Select Surfaces...to project onto, select the surface.

NOTE: You have to do this through an orthogonal view. If you do it in perspective view, it will perspectively project the curve.

Click the Surface to project onto in an orthogonal view
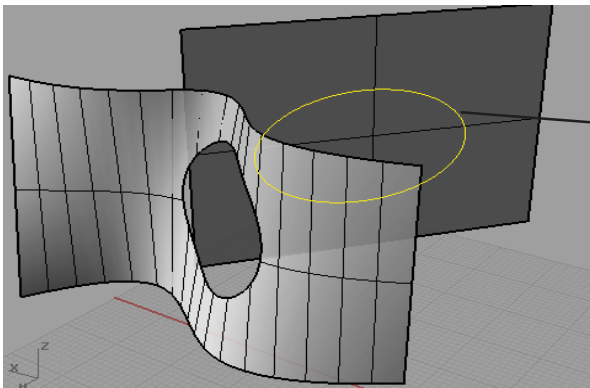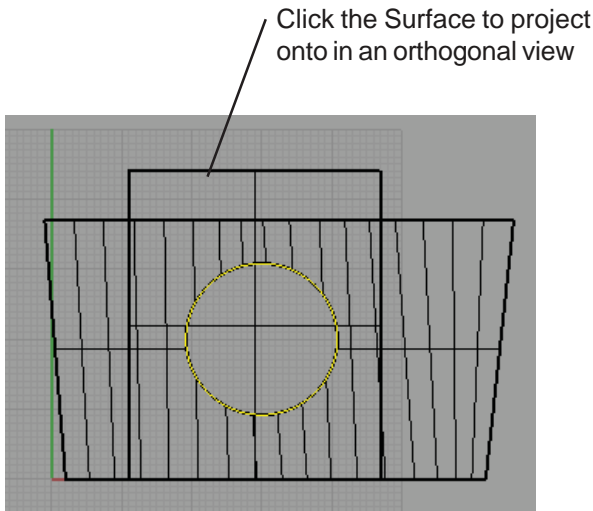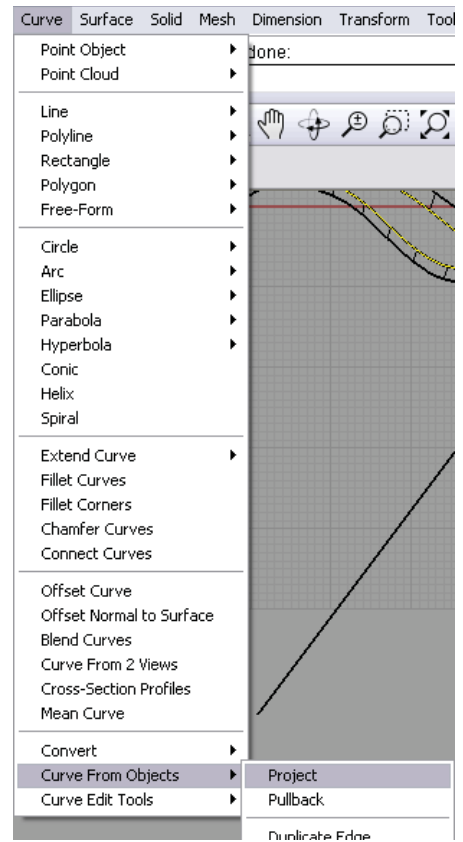
The curve projected onto the surface.

Delete the surface and you are left with the projected curve.

Lofted Surface

Filleted Surfaces

I lofted the two Edge curves and filleted the intersection of the two lofted surfaces.
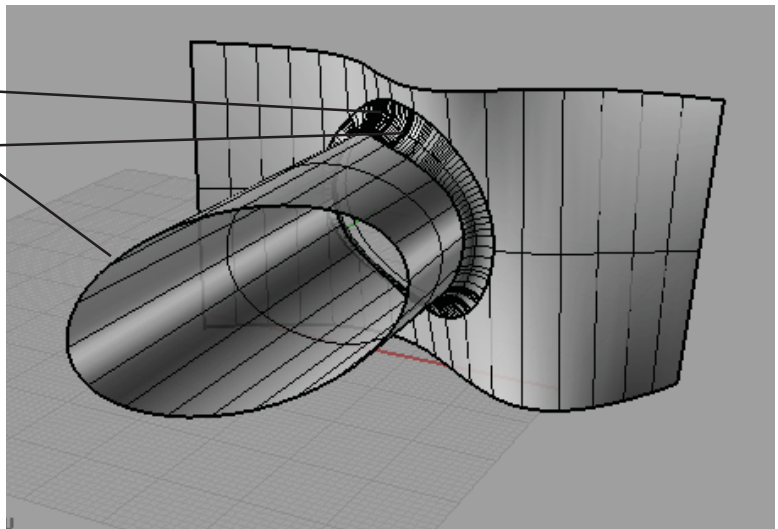
Capping Holes and Generating Surfaces from Edge Curves:

Often you will need to cap holes. If the curves that make up the edge of the hole are planar (They all fall inside a single plane, you can use the Planar Holes tool.

Select Surface>Planar Curves.

Select the curves to cap and Hit Enter.
(Note: You can select more than one curve.)

Surface	Solid	Mesh	Dimension
Plane			▶
Loft			
Sweep 1 Rail			
Sweep 2 Rails			
Revolve			
Rail Revolve			
Curve Network			
Corner Points			
Edge Curves			
Planar Curves			
Point Grid			
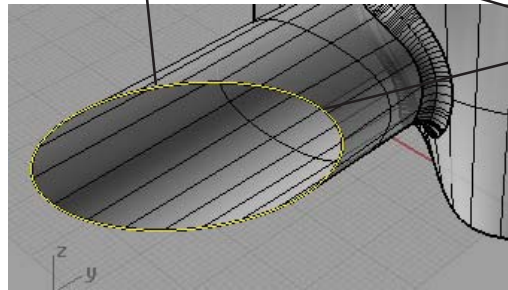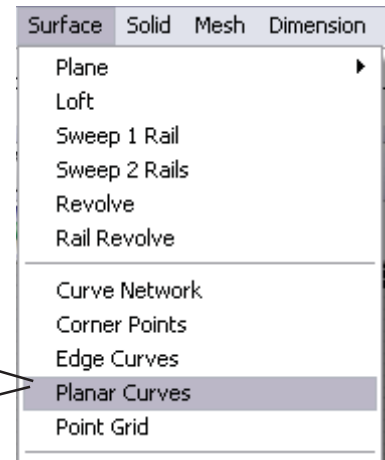
Capped Surface
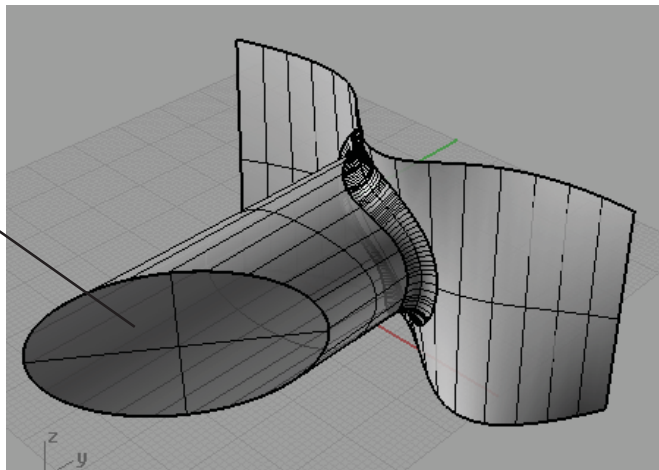
To cap a non-planar surface, you will need to use the Edge Curves tool. The problem with the edge curve tool is there is a limit to the number of curves you can select (only 2-4 curves) To get around this you can join additional curves. However, the result can be less then satisfactory.
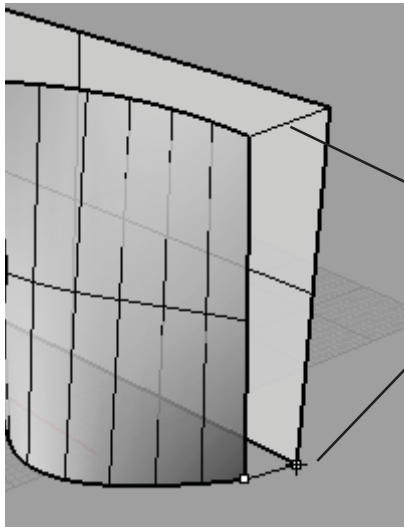
A common use for the Surface from edge curves is to box out two surfaces.
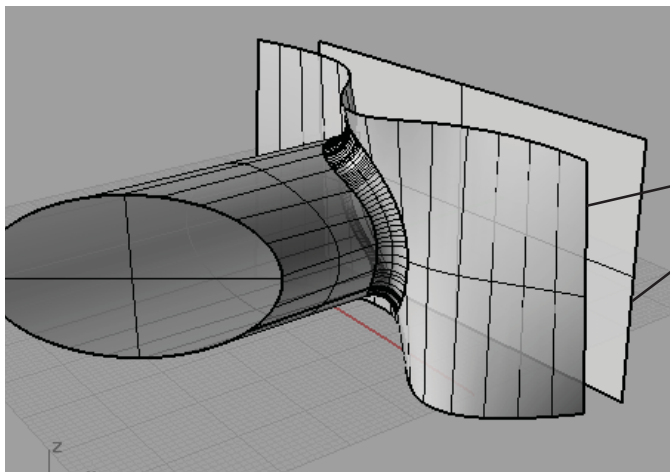
We will use this tool to create surface along the two edges of this surface.
(Note: We could loft it but it is hard to do it on all four sides.
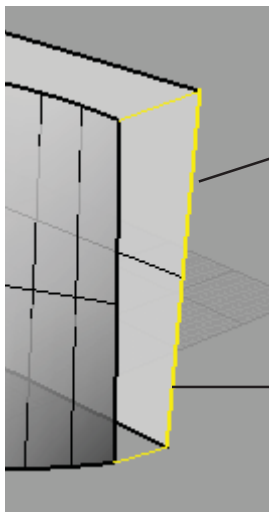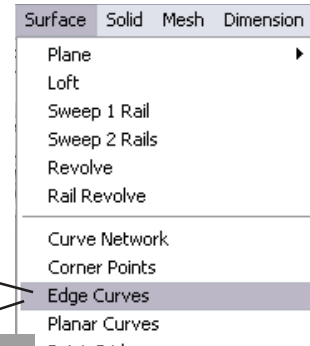
Step One: Snap Curves between the two Surfaces.

Using the curve tool, and with the end point O-Snap active and Project off, snap two lines between the two surfaces.
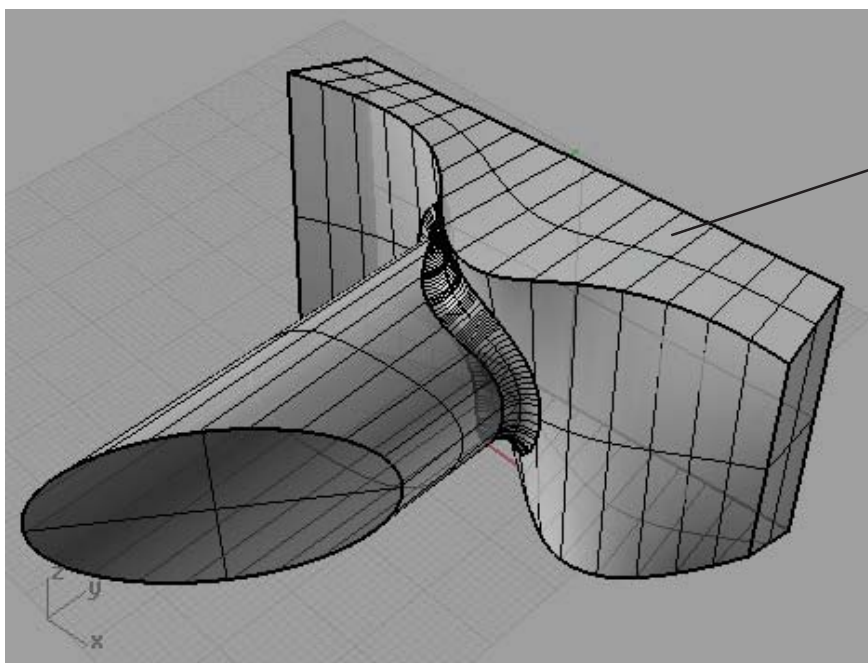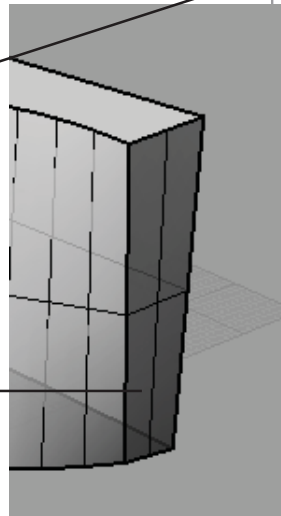

Two Curves Drawn between the two surfaces.

Step Two: Activate the Edge Curves tool by Clicking on Surface>Edge Curves

Surface   Solid   Mesh   Dimension

Plane   ▶
Loft
Sweep 1 Rail
Sweep 2 Rails
Revolve
Rail Revolve

Curve Network
Corner Points
Edge Curves
Planar Curves

Select the 2 Curves and 2 Edges that will make up the surface.

The surface is created.

Model with all four surfaces created

Object Manipulation using Caging:

Caging allows you to transform a complex object using a small number of edit points.
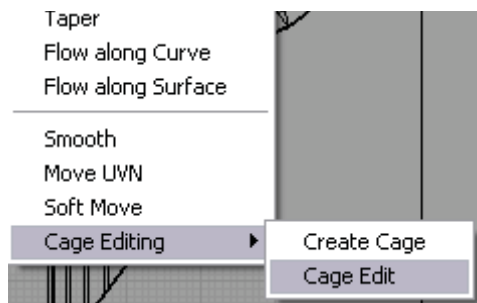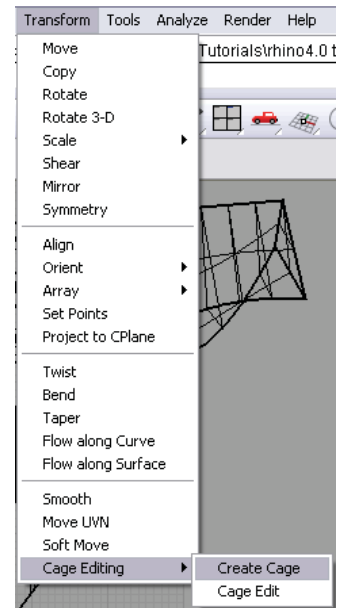
Caging is a two step process, first you define the shape (Cage) that will be used to box around the object and transform.

To do this select Transform>Cage Editing>Create Cage

At the prompt, draw a 3-point box around the object and hit Enter. A box is formed around the object.

The next step is to Edit the Cage which will transform the object.
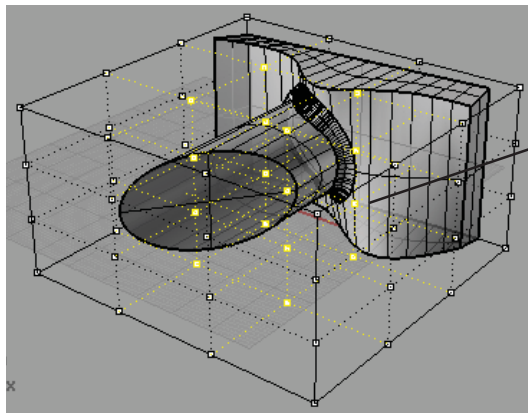
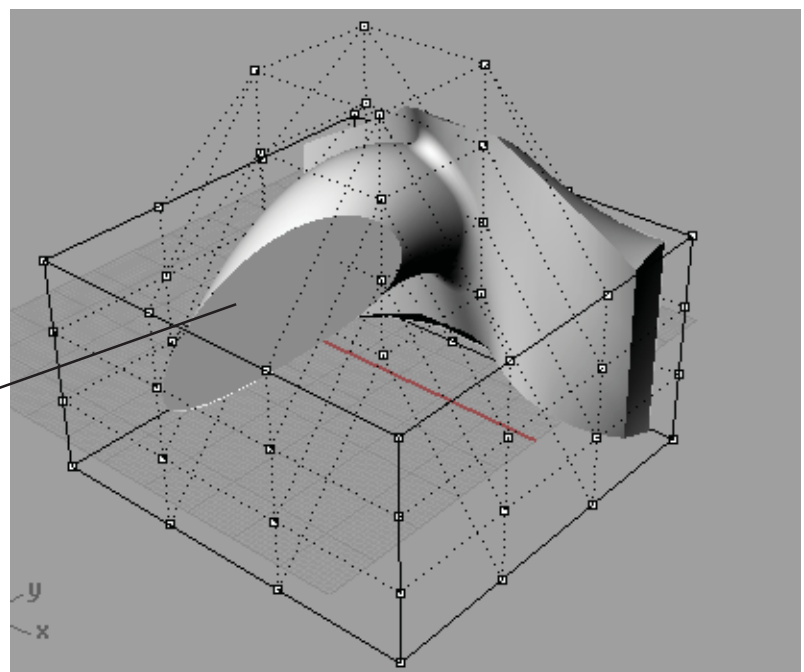Click Cage Editing>Cage Edit.

At the prompt, Select Captive Objects, select the objects to transform.

At the next Prompt, Select Control Object, select the caged box.

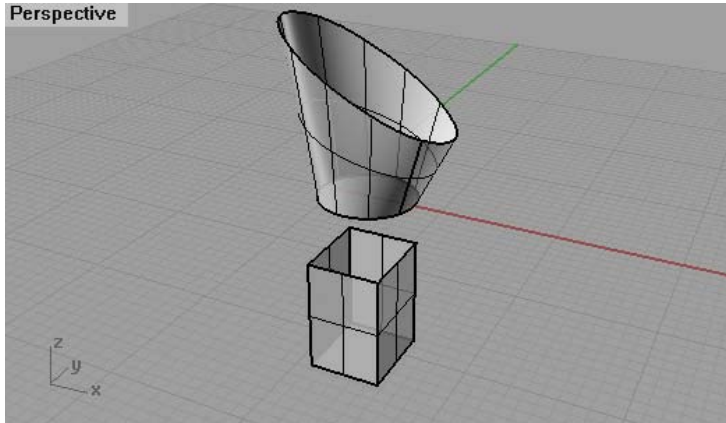At the next Prompt, select Global, and points are created throughout the cage.

Select Points in the cage and move the points
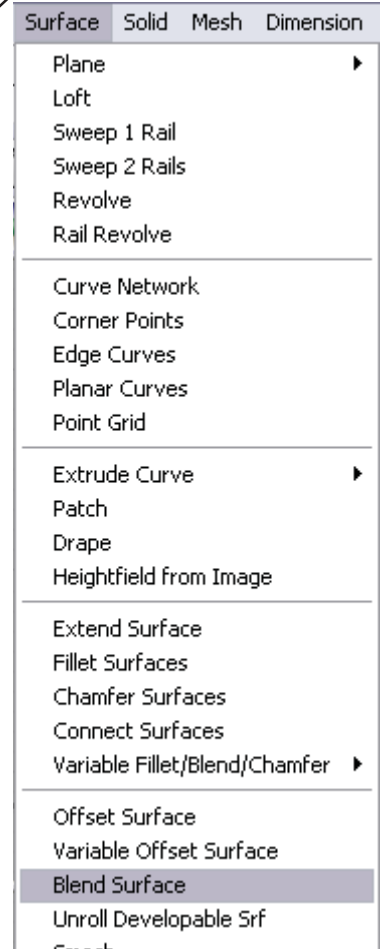
Transformed Object

Blend Surface Feature:

Rhino has a number of tools to match up two separate surfaces. These are located in the surface tools pallet, or in the surface pull down menu.
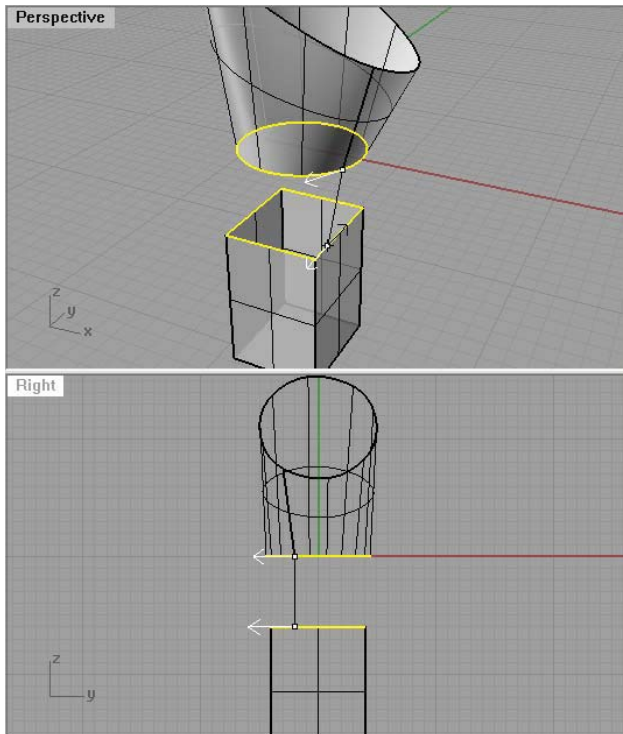

Fillet Surface

Blend Surface

These are two surfaces that I want to blend together to create a single surface that will transition the two surfaces.

Step One: Select "Surface" from the menu bar, and then "Blend".


Step Two: In the command prompt at the top of the window, under "Select First Set of Edges", click on the edges of the first surface to blend. After you have selected all of the edges, hit return or right click.

The command prompt will now say "Select Second Set of Edges". Select the second set of edges, and hit return or right click.

Step Three: At the command prompt "Select Seam Point to Adjust. Press Enter When Done" you will get some additional options. You will also notice that you will get two arrows. These arrows allow you to adjust the location of the seam to the blended surface and adjust the direction of the surface.
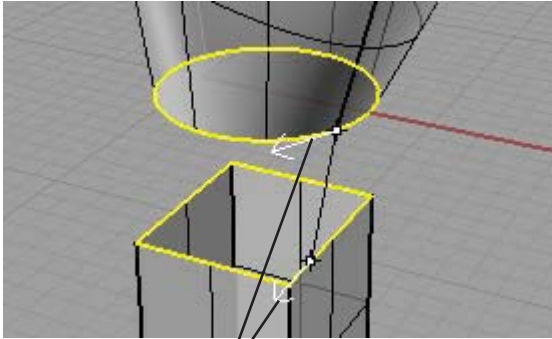

Two Arrows
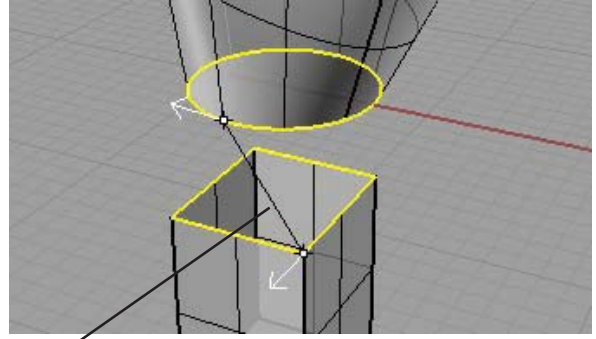
Seam of Blended Surface Adjusted.

Also in this command prompt you will notice three options "Flip Direction, Automatic, and Natural" The flip direction option allows you to orient the arrows in the same direction to avoid a twisted surface. Automatic will generate the surface automatically. Natural will give a more relaxed look to the surface. Click Enter to Continue.
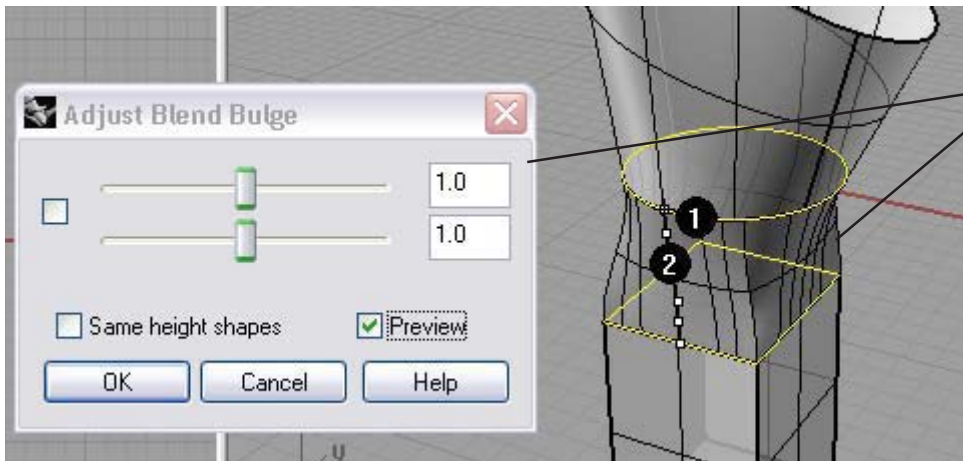
A preview button will appear where you can adjust the settings.
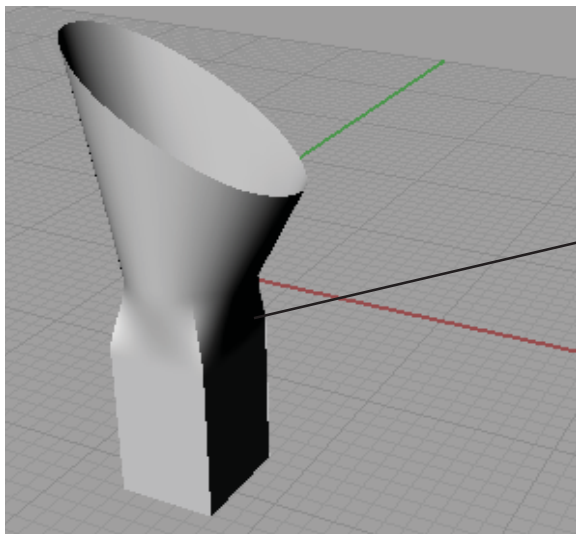

Generated Surface

Using Blocks in Rhino:

Often you will use the same part created in Rhino many times. If this is the case it is best to use blocks. Blocks have two major advantages:

1. It keeps the files sizes small and thus you can create complex models without having large files sizes.

2. If you need to update the part, all the instances of that part will be updated.
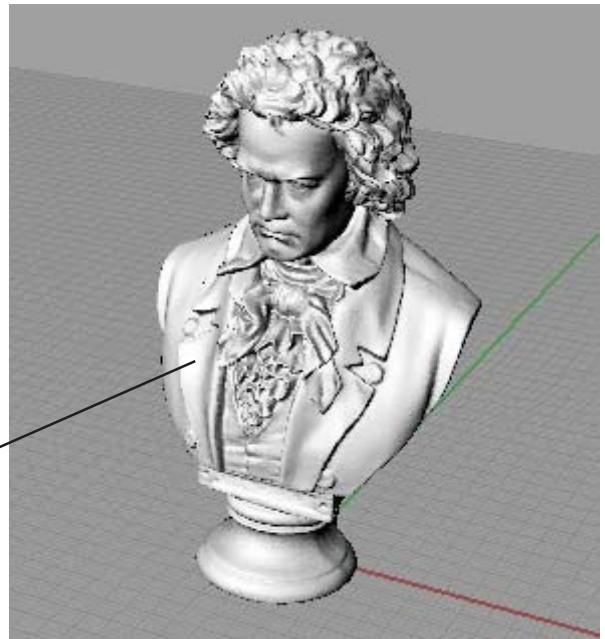
Complex Form that will be turned into a block.
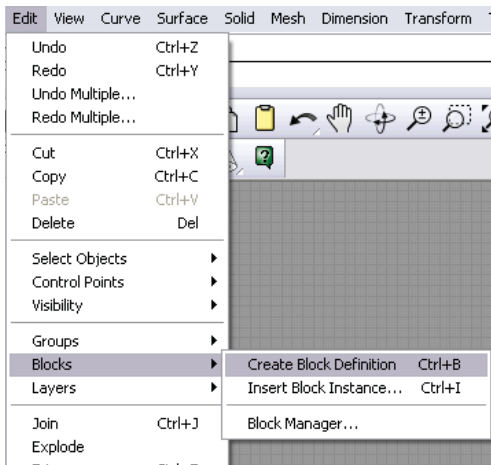
Select Edit>Blocks>Create Block Definition.

At he prompt, select objects to define block, click on objects and hit Enter.

At the Prompt, define base point, click to set an insertion point for the block.

The following window will appear:
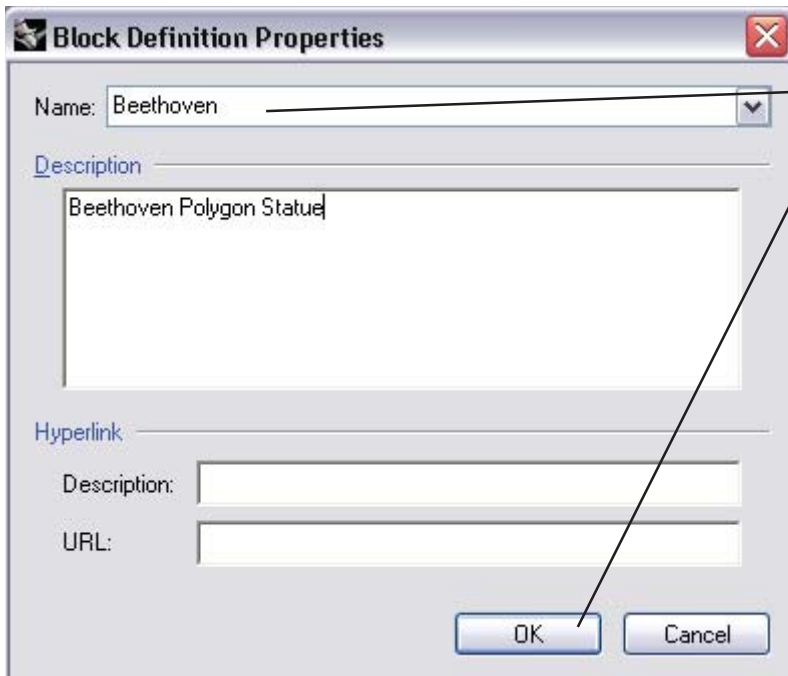
The block Definition properties will appear.

Give the block a Name and hit OK.

Inserting a Block Definition:

After you create a block, you can then insert instances of the block into the Rhino model. An instance is a reference of an original. This is how you can keep the file sizes down.
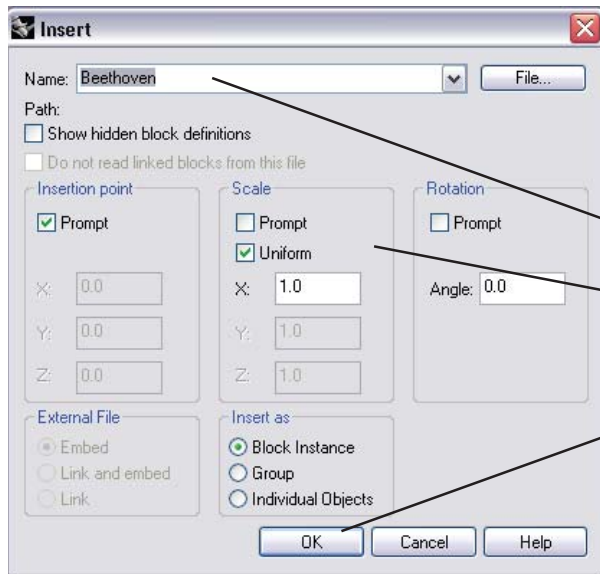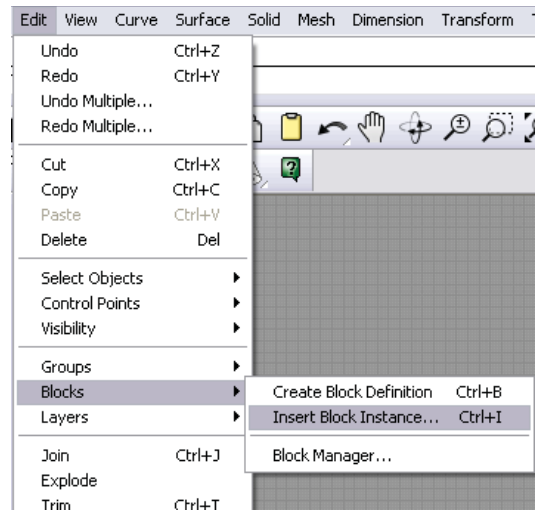
Choose the Block to insert

Select Various insertion settings

Click OK to insert

The block will be bound to the mouse and when clicked, will be inserted
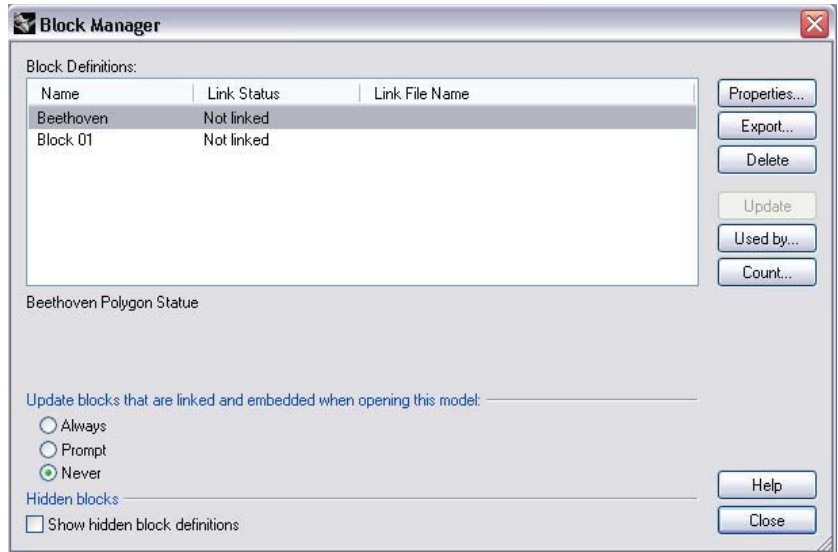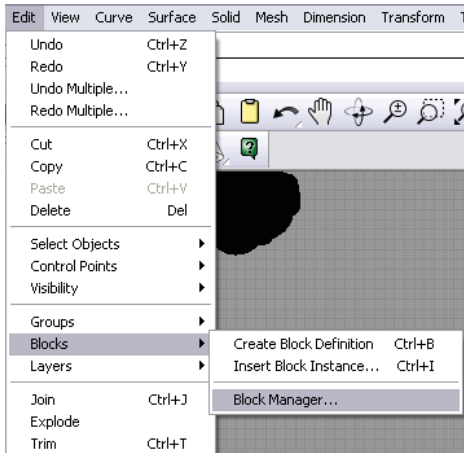
Linking External Models as Blocks:

Another powerful way to use blocks is to link a series of external models into a "Master" Rhino Model. This allows updates in different models to be automatically updated in the master model.

Step One: Create a Block from an external file. Select Edit>Blocks>Insert Block Instance
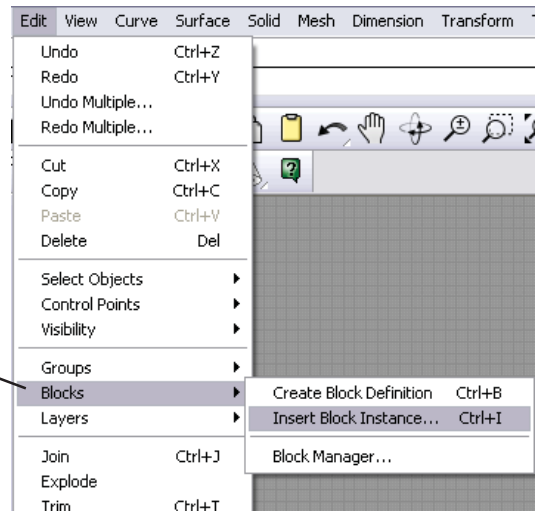
Select File
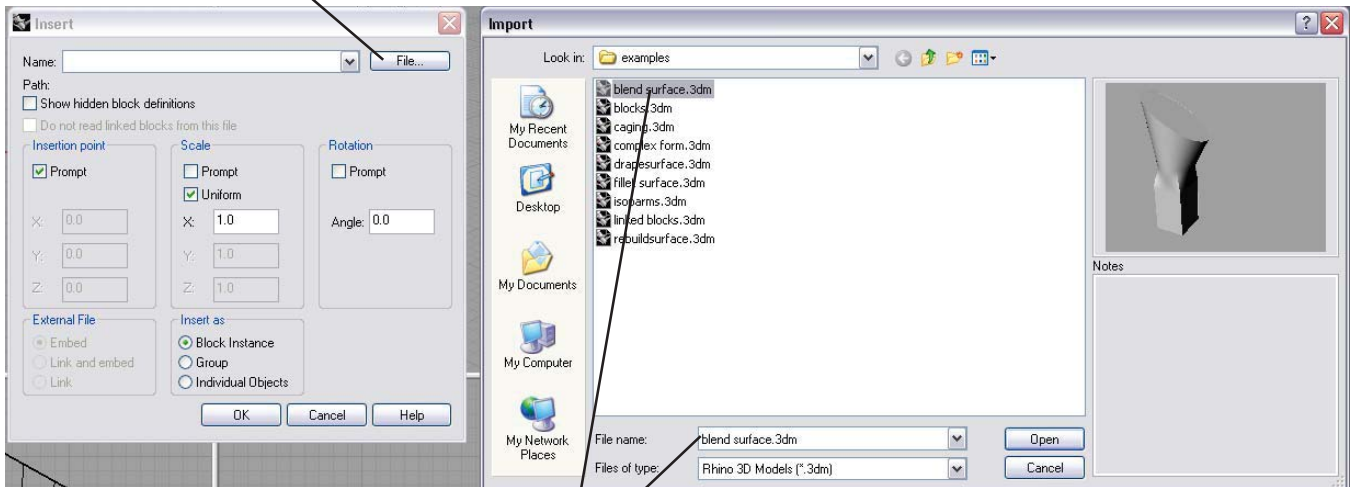
Select a Rhino Model to insert
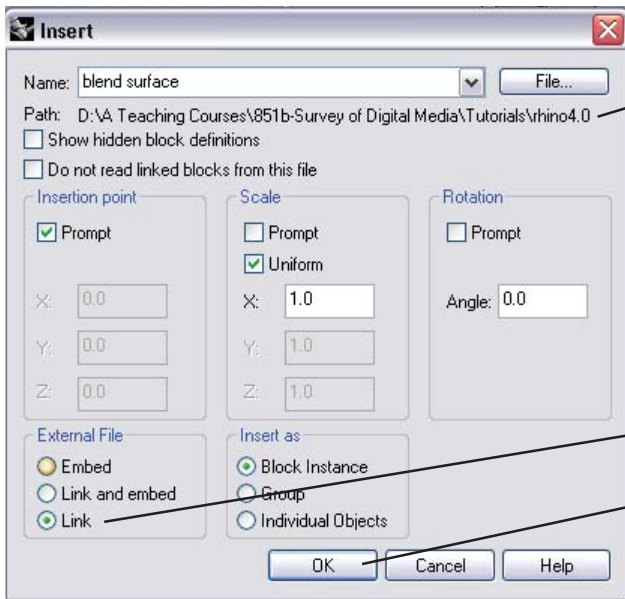
Linked File Name and Pathway is displayed

(NOTE: It is a good idea to keep all files in the same folder to avoid broken links)

Under External File, Choose Link

Click OK to place in model
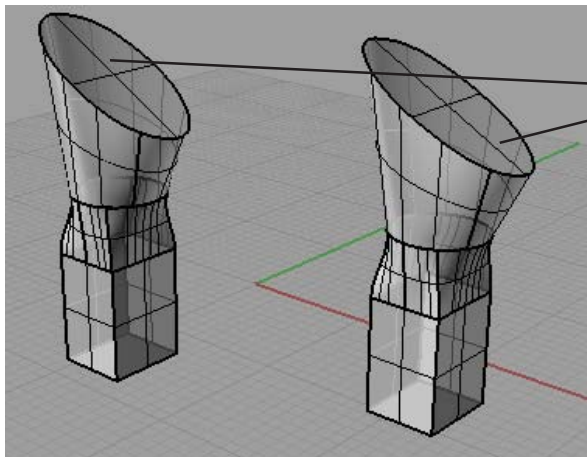
Linked Models

If changes are made in the original model the changes will appear in the model with the block definitions.

Change Made to Original Model File
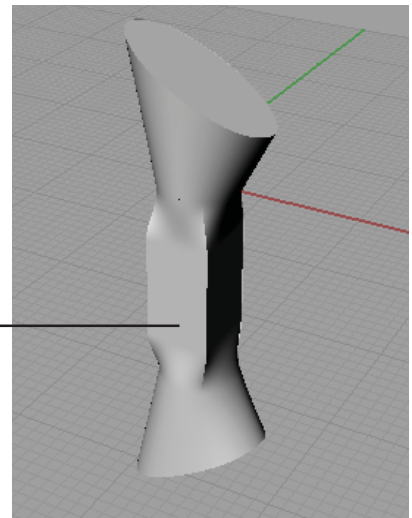
To update the Link Model, click on Edit>Blocks>Block Manager
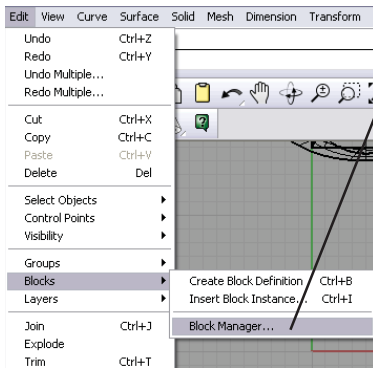
Select the Block to Update, and click on Update.
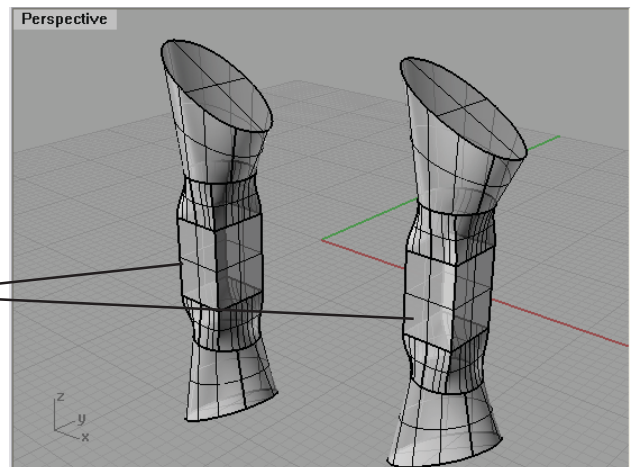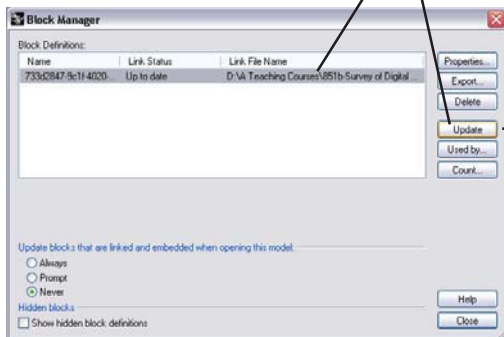
Blocks are Updated

Make A 2D Drawing:

In Rhino you can generate a series of 2D Drawings from 3D forms these drawings are a series of 2D curves that can be imported into illustrator or flash.

To make a 2D drawings, click on Dimensions>Make 2-D Drawing.

At the prompt, Select objects to draw, select the objects.

Dimension	Transform	Tools
Linear Dimension		
Aligned Dimension		
Rotated Dimension		
Ordinate Dimension		
Radial Dimension		
Diameter Dimension		
Angle Dimension		
Leader		
Text Block		
Annotation Dot		
Hatch		
Recenter Dimension Text		
Make 2-D Drawing		
Dimension Properties...		

2-D Drawing Options

Drawing layout
- ○ Current View
- ○ Current CPlane
- ⦿ 4 View (USA)
- ○ 4 View (Europe)

Options
- ☑ Show tangent edges
- ☐ Show hidden lines
- ☐ Maintain source layers

Layers for hidden line objects

Visible lines:	Make2D visible lines
Hidden lines:	Make2D hidden lines
Annotation:	Make2D annotation

OK   Cancel   Help

The following Window will appear:

Choose 4 View USA

Click OK
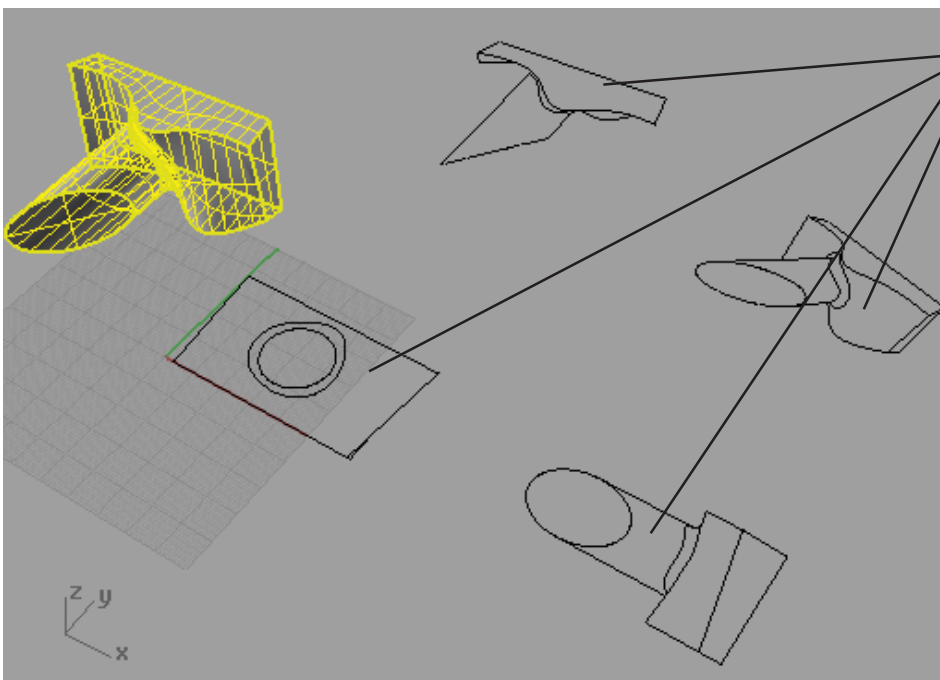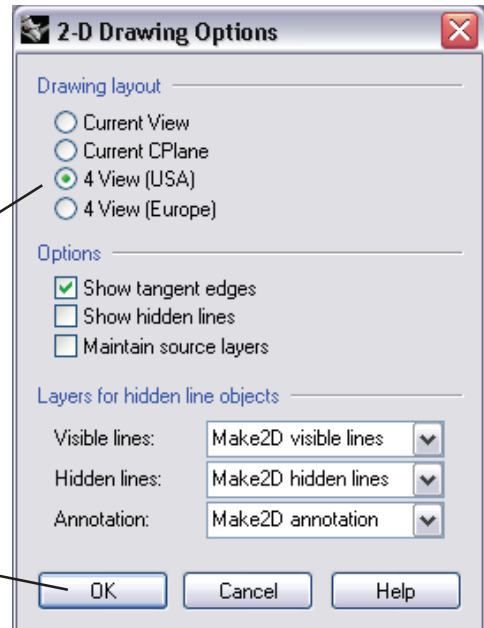
Four Views, Top, Front, Left, and Perspective are created.

These views are located on the ground plane. If you select a view and export it from the top view it will be oriented in the correct direction.

Computer Generated Architectural Presentation Drawings

CUNY - New York City College of Technology
Architectural Technology Department
Instructor: Joseph Lim

AutoCAD to Rhino

Cleaning your AutoCAD drawing is one of the most important parts of the process. A clean .dwg will allow you to easily produce a 3D model, while a messy .dwg can cause many headaches in the long run.

Make a Copy of your Files
Before you make any changes to your file, it is recommended that you use **File>SaveAs** to create a second set of files that will become your underlay for modeling by appending something to the end of the filename. This is important as you do not want to modify your working drawings accidentally.

NOTE: Make sure that you choose AutoCAD 2004 as your filetype when you **SaveAs**. The 2007 and 2010 filetypes WILL NOT import into Rhino.

Delete Unneeded Information
It is important that we remove any extraneous information from the drawing:

Text and Dimensions
Delete all text and dimensions. You will not need them in Rhino and they will only serve to cause you headaches if you leave them in your drawing.

Layers
It is time to leave your layers behind. Turn on all of your layers (assuming that you need all of them in your model) and change all of your layers to layer 0. You'll likely find that you have some objects that still appear to be on a layer. These are probably blocks that have been improperly made. We will deal with blocks next.

Blocks
You may find that the drawings you are working with use blocks for washrooms or bathrooms, kitchens, furniture, scalies (people) and other non-architectural elements. It is best to remove all of these blocks from the drawing by deleting them. Blocks are also often used to simplify the process of drawing complex geometry. These blocks should be simplified (by editing the block directly or using **bedit**) and then exploded using **explode**.

You may need to **Edit>Select All** and change the layer of your objects to 0 one more time, as blocks can have objects set to different layers nested inside them.

NOTE: It is better to delete your unneeded blocks before exploding them as you will often have a hard time cleaning up your drawing should you explode first.

Hatches
Next, we will want to make sure that we clean up any hatches from the drawing. You should just delete these objects unless they are highly specific (a brick pattern that has been PROPERLY applied at the right scale.) IF and ONLY IF you absolutely need a hatch (think long and hard about this), you can explode that hatch and it will import into Rhino.

Helpful Commands:
Purge, Qselect, Overkill

After deleting the above objects (block instances, hatches, and objects on unneeded layers) you will want to **purge** the file. This helps ensure that your Rhino file will not have hundreds or possibly even thousands of unused block definitions, layers, hatch definitions, etc. taking up space and slowing down your model.

Open Rhino and Choose **File>Import:**
Make sure that you choose AutoCAD drawing file in the type dropdown.

| File name: | | ▼ | Open |
| Files of type: | AutoCAD drawing file - (*.dwg) | ▼ | Cancel |
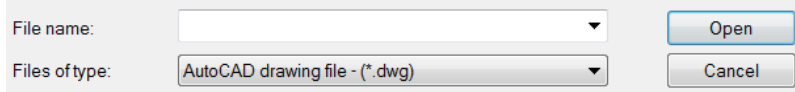
Your geometry should appear in your Rhino model. Before doing anything else, **group** the drawing. This will allow you to select it as an object, rather than line by line.

Create a new layer (right click on the layer palette> New Layer):

Layers - All Layers

| Name | Ma... | Linetype |
| Default ✔ | ■ ○ | Continuo... |

		us
✓	Set Current	us
	Set Properties ▶	us
	New Layer	us
	New Sublayer	us
	Rename Layer	
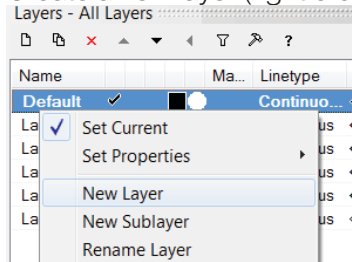
and place your plan on it (select the plan, right click the layer you made and choose Change Object Layer):

| Default ✔ | ■ ○ | Continuo.. |
| Plan 1 💡🔒■ ○ | Continuous |

	Set Current	us
	Set Properties ▶	us
		us
	New Layer	us
	New Sublayer	us
	Rename Layer	
	Delete Layer	
	Select All	
	Invert Selection	
	Select Objects	
	Change Object Layer	
	Copy Objects to Layer	
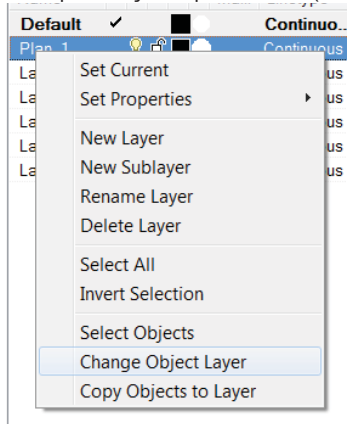
Verify your units and dimensions match in AutoCAD and Rhino by using the distance command in both AutoCAD and Rhino on the same geometry.

Now we want to simply repeat the process with each of the underlay drawings you will be using.

ARCH 2491
Computer Generated Architectural Presentation Drawings

CUNY - New York City College of Technology
Architectural Technology Department
Instructor: Joseph Lim

CLASS 2   02.08.2011
Rhino 2 – Surfaces continued, Advanced tools – Sweep, Revolve, BlendSrf, ChamferSrf, Fillet, Contour

Interface will not be covered explicitly in this handout.
Remember that a brief introduction to the **interface** of Rhino is available here:

Rhino Interface:
Rhino Interface/Tutorial 1
http://www.paramod.net/post043/rhino001.html

Surfaces
Sweep1
Create a rail curve and a profile. Think of the rail curve as the path that will be traveled, and the profile curve as the curve that will be travelling on it (like a rollercoaster). The red path represents the profile curve in the example below. The options box controls may require some tweaking to get the desired effect. Use CommandHelp for an exhaustive description of the options.
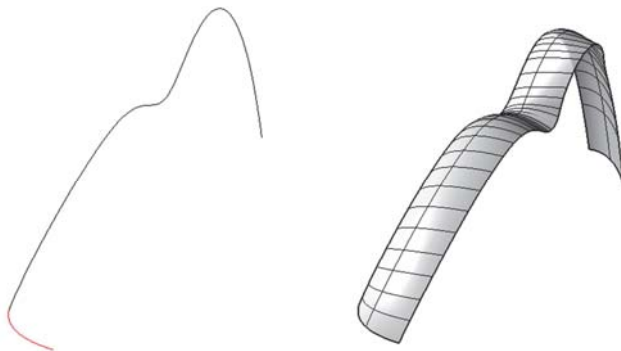
Sweep2 is very similar to sweep1, however you can select 2 rail curves to control your profile curves. Remember, the order is important. Select the 2 rail curves FIRST, then select the profile curve.
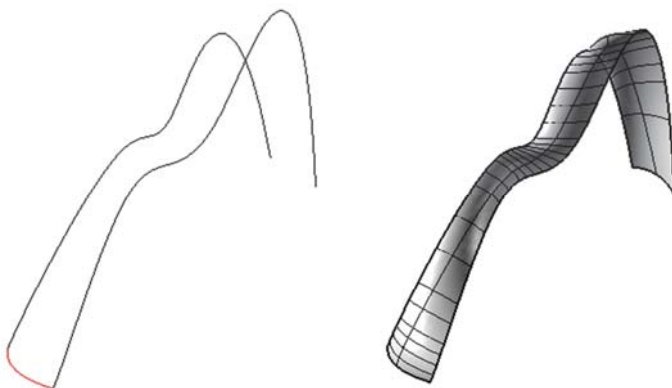
Revolve
Revolve is a command that creates a surface from profile curves, a revolution axis, and angle. First draw a profile curve that you want to revolve. Invoke the command. Rhino then asks you for the curves to revolve. You can select more than one. Normally these curves are 2-dimensional, or flat, but they can also be 3D. Then choose a revolution axis. This is the axis around which our objects will rotate (think of a CD. A line drawn straight up and down through the center of the hole would be the axis of rotation). The length of this line is not important, only the direction. Then choose the angle through which you want to rotate the curves. 360 is a full circle, 180 half a circle, and so on.
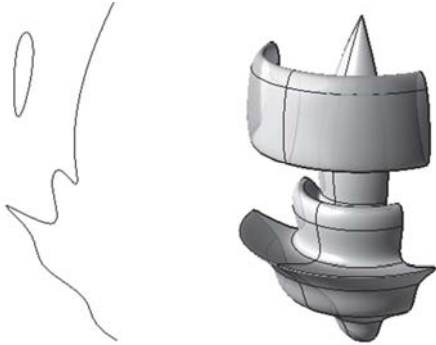

BlendSrf
Blend is a command that takes 2 different surfaces and creates a third surface that literally blends between the first two. There are different options available for the blend, which we will cover here and that are also covered in the CommandHelp. Invoke the command. Rhino will ask you to choose the edges to blend together. Choose the two edges facing one another. Hit enter. Rhino will then present you with an options box, and several options from the command line. Notice the Continuity options are set to G2. The number describes how "strong" the influence is from that edge's surface. Change the Continuity _1 from G0, one by one, to G4 to see how this effects the shape of the resulting surface. Move the sliders to change the strength of influence for each edge. You can also edit the handles directly to change the shape of the blend surface.
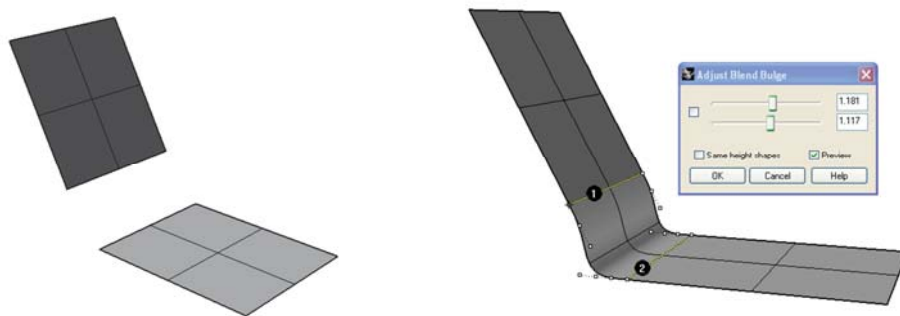

ChamferSrf
Chamfer creates a beveled edge on two adjacent surfaces. Invoke the ChamferSrf command. Rhino will then prompt you for the two surfaces to be chamfered. Choose the two surfaces and hit enter. After hitting enter, Rhino then prompts you for the distances to chamfer. You can cut more from one surface than the other if you make these distances unequal.
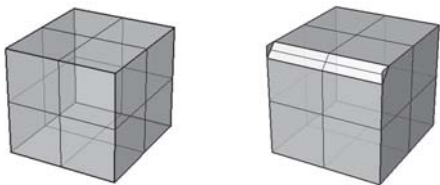
FilletSrf

Fillet is much like chamfer in that it acts on two adjacent surfaces. Instead of producing a bevel, or cut, however, fillet produces a radius join. Invoke the FilletSrf command. Rhino will prompt you for two surfaces. Choose the two surfaces and hit enter. Rhino will then prompt you for a radius to cut. Choose your radius and hit enter.
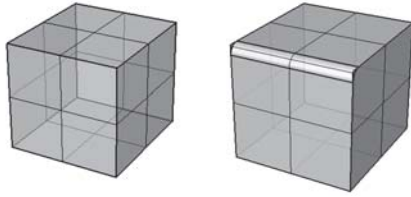
Contour

Contour creates a series of cut planes through an object and makes curves at those locations. It is possible to choose the distance between contours and their direction. Choose an object for contour. Invoke the command. Choose the base point for the contour plane (it does not matter where this is). Choose a second point. The second point determines the direction that the contours will be made, so if you choose a point directly above the first point on the z axis, your contours will all be made in the x-y direction. If you set the JoinCurves option to ByContourPlane, each contour curve will be a single object. When you hit enter, the command will generate contour curves.
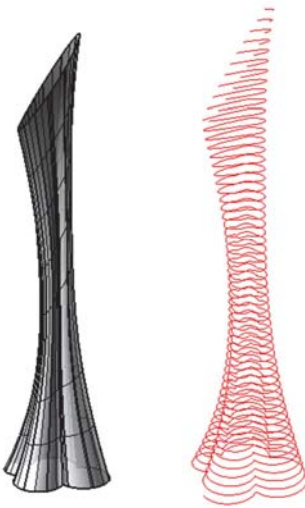
Explode

Explode will take a solid object and separate it into its constituent parts. For instance a cube would become 6 separate surfaces when exploded. Sometimes this is useful because we want access to each side individually. More often than not, however, it is easier to work with one side at a time. We can extract sides individually by using:

Extractsrf

Extractsrf will pull a surface from a solid for editing. The remainder of the solid will stay together in a polysurface (poly meaning many).

Join

Will take a series of surfaces and join them into one polysurface.

Group

In Rhino, objects can be organized into groups using the "group" command. Grouping objects together creates an organizational relationship. This relationship does NOT create solids or new geometry. Surfaces grouped together using the group command are not polysurfaces and have only an organizational relationship. These surfaces are not related geometrically.

Commands that work similarly in AutoCAD and Rhino for you to try:
Trim, extend (works with surfaces as well), stretch, scale, rotate (in addition to rotate3d), chamfer and fillet (works with curves as well),

Commands that are new but work like commands we have already covered to try:
Curveboolean – Booleans with curves
Moveedge – Move the edge of a surface
Maelstrom – Rotates part of a surface, solid, or curve leaving the rest stationary
Arrayhole – Creates an array of holes in a solid or surface
Deletehole – Removes a hole in a solid or surface
Rotateedge – Rotates an edge of a surface
Rotateface – Rotates a face in a solid
Rotatehole – Rotates a hole in a surface or solid

Visual Studies I: Arch 1191

Assignment Number: 10

Computer Program(s): McNeel Rhinoceros Cont'd and V-Ray

Student Learning Objectives:

Upon successful completion of this assignment, the student will:

1. Understand the differences between creating a perspective by hand using the office method/plan projection method and setting up cameras and rendering digitally in McNeel Rhinoceros and V-Ray.
2. Understand how to apply materials in V-Ray.
3. Understand how to create lighting in V-Ray.

Assessment:
To evaluate the student's achievement of the learning objectives, the professor will do the following:

1. Evaluate the student's understanding McNeel Rhinoceros and V-Ray by examining both the digital file and the hard copy output.

Project Description:
In this assignment, you will create and render perspectives of your digital version of Extruded Geometries, Assignment 9 and compare to your plan projection perspectives by hand. Remember that perspectives are the only drawings that will describe the experience of a space. It is important for this reason to make sure that the horizon line is at 5'6, the average eye level of a human being. While worm's eye views can be interesting, they do not describe how we experience most spaces, unless perhaps it is a yoga studio and we are often lying on the floor. Bird's eye views are great for larger planning exercises as found in urban planning.

Process:

1. Create cameras at different positions around your model.
    a. Be sure that your camera's are at a height of 5'-6" above floor level for your model.
    b. Be sure that the target for your camera is also at a height of 5'-"6".
2. Check perspectives by doing a simple render:
    a. Set the renderer to Rhino renderer.
    b. Choose resolution
    c. Background color
3. Use V-Ray for advanced rendering.
    a. Apply materials to model
        i. Apply default material.
        ii. Edit default material so that you have the following three versions of your model:
            1. One version should be a simple white model
            2. Another version should play with solid, translucency, and transparency.
            3. Another version should use textures.
    b. Apply lighting in V-Ray:
        i. GI or Global Illumination does not create shadows as there is not direction.
        ii. Add depth with Rectangular Light: Click on Spot light icon, select Rectangular light, create light in top view and place it in elevation at a height of five times the elevation of your model.