RHINO

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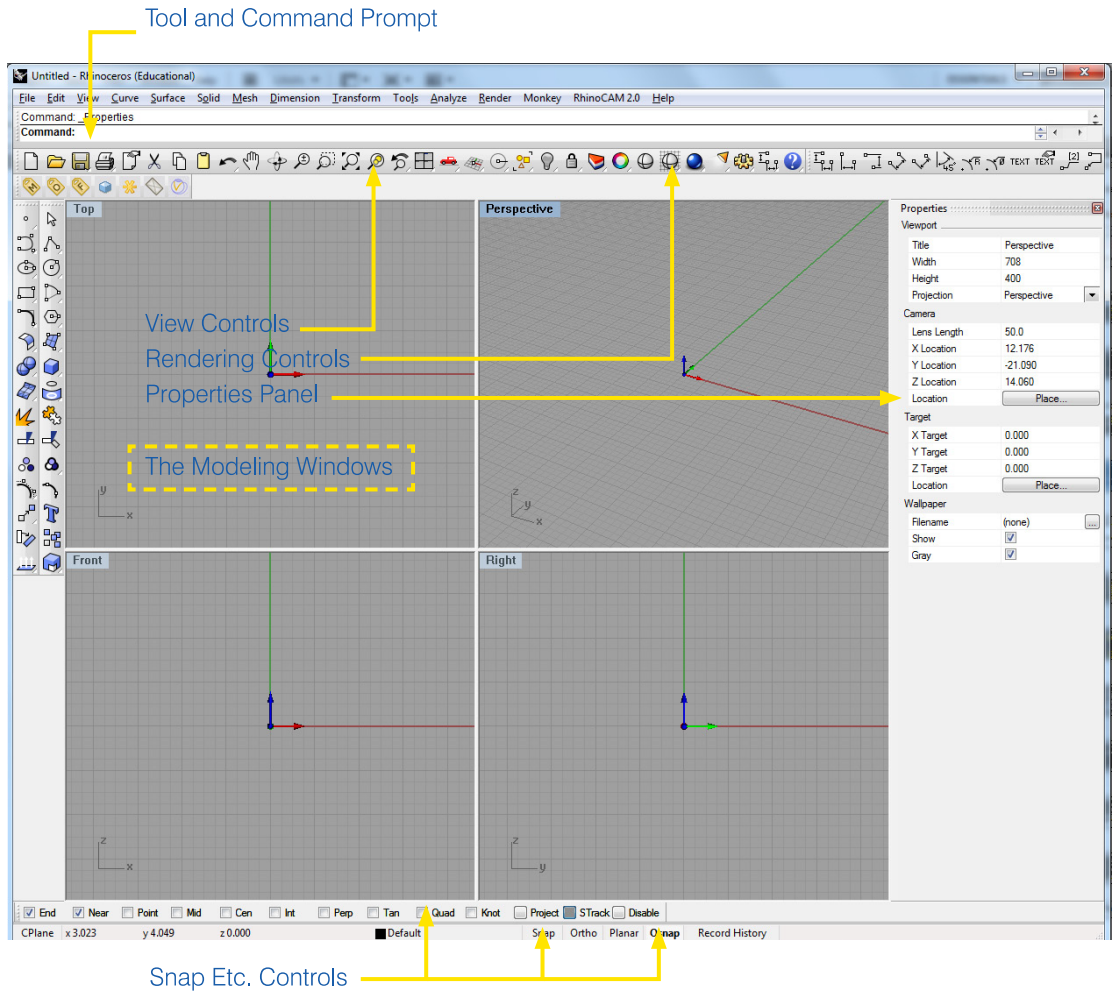last edited February 9, 2012 12:01 AM

Interface:

Icons, Menus, Pallettes, View Manipulation, Command Help

Rhino is designed as a Nurbs Modeler. Nurbs stands 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 malleable for the designer. 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. 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.
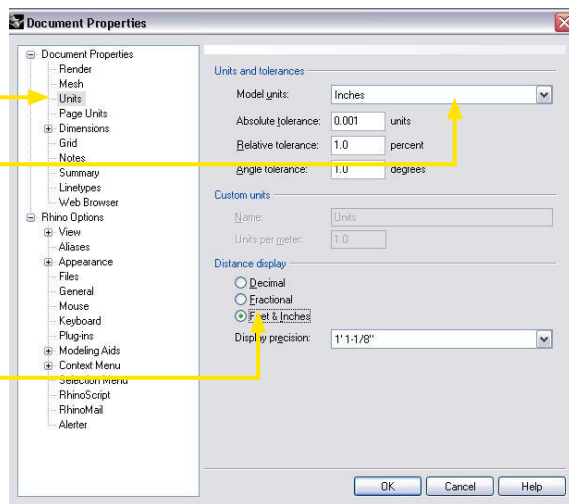
Tool and Command Prompt

View Controls

Rendering Controls

Properties Panel

The Modeling Windows

Snap Etc. Controls

Setting up grids and units for accurate modeling:

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

Units Tab

Units Setting

Display Setting
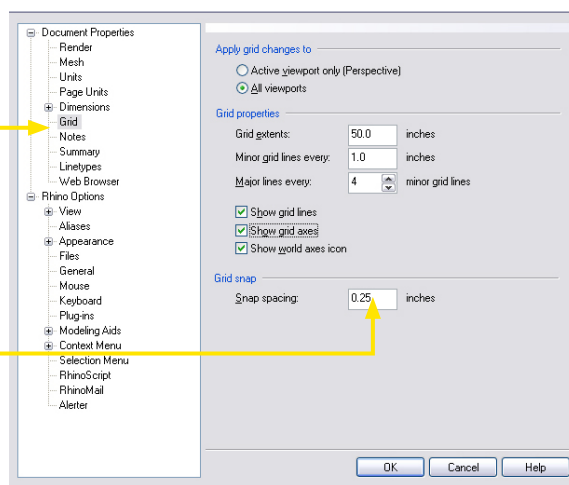
Document Properties

- Document Properties
  - Render
  - Mesh
  - Units
  - Page Units
  - Dimensions
  - Grid
  - Notes
  - Summary
  - Linetypes
  - Web Browser
- Rhino Options
  - View
  - Aliases
  - Appearance
  - Files
  - General
  - Mouse
  - Keyboard
  - Plug-ins
  - Modeling Aids
  - Context Menu
  - Selection Menu
  - RhinoScript
  - RhinoMail
  - Alerter

Units and tolerances

Model units:	Inches
Absolute tolerance:	0.001 units
Relative tolerance:	1.0 percent
Angle tolerance:	1.0 degrees

Custom units

Name:	Units
Units per meter:	1.0

Distance display

- ○ Decimal
- ○ Fractional
- ⦿ Feet & Inches
- Display precision: 1' 1-1/8"

OK   Cancel   Help

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

Grid Tab

Snap Spacing

- Document Properties
  - Render
  - Mesh
  - Units
  - Page Units
  - Dimensions
  - Grid
  - Notes
  - Summary
  - Linetypes
  - Web Browser
- Rhino Options
  - View
  - Aliases
  - Appearance
  - Files
  - General
  - Mouse
  - Keyboard
  - Plug-ins
  - Modeling Aids
  - Context Menu
  - Selection Menu
  - RhinoScript
  - RhinoMail
  - Alerter

Apply grid changes to

- ○ Active viewport only (Perspective)
- ⦿ All viewports

Grid properties

Grid extents:	50.0	inches
Minor grid lines every:	1.0	inches
Major lines every:	4	minor grid lines

- ☑ Show grid lines
- ☑ Show grid axes
- ☑ Show world axes icon

Grid snap

Snap spacing:	0.25	inches

OK   Cancel   Help

Snap, Ortho, Planar, OS-nap, Record History, and S-Track:

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

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

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

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

- Disable: Disables the O-Snap functions.

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

- Osnap: (Object Snap) activates the snap menu. It will snap lines to geometry as you draw.

- Record History: Allows you to link lines to surfaces. If you change the line, it will update the surface (Kind of Works)

- 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 found in the Menus palette at the top or the screen. In the Menus palettes there are 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.
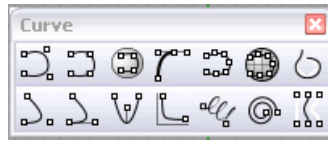
- Surface Tools: This palette is used to generate surfaces from Points and Curves. Lofts, Surface from Curve Network, Sweep Along one and two Rails, Revolve, Patch, and Drape Functions.

- 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 include Difference, Union, and Intersection, as well as Fillet, Cap Holes, and Extract Surface.
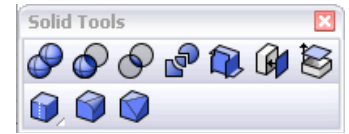
- Curve From Object Palette: This is used to generate curves from objects that then can be used to generate more surfaces. You can also generate a series of contours from an object, section curves, etc.
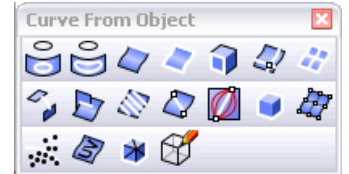
- Mesh Tool Palette. This is used to extract mesh surfaces and objects from nurbs surfaces. This is useful for exporting into other programs. You can also create direct mesh objects without creating a nurbs object first.
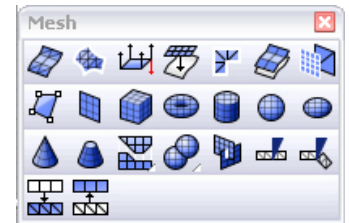
Rhino, as mentioned above, is primarily a Nurbs Modeler. Nurbs stands for Non-Uniform Rational B-Splines. This basically means that smooth surfaces can be created through mathematical formulas.

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.
In Rhino, points can be connected up to form multi-segmented lines called curves. The shape of the curve depends on the mathematical 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.

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.

Solids

In Rhino, surfaces can be joined to-gether to form solids. Surfaces that have been joined to each other are called polysurfaces. There are many methods for 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 "Rect-angle" and then "ExtrudeCrv." If the capped option is active, the extru-sion 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.
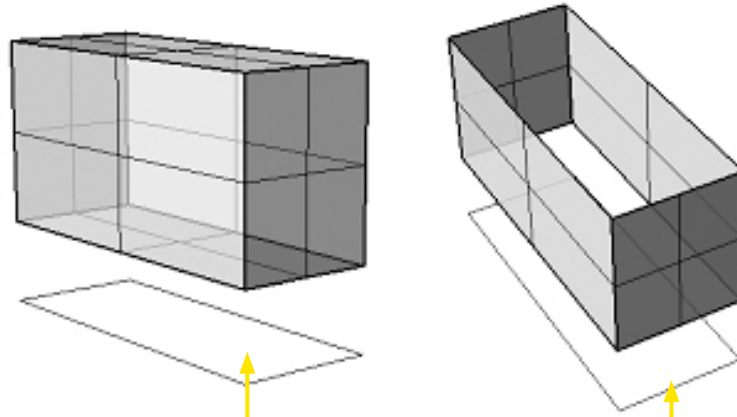
"Slab" command:

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.
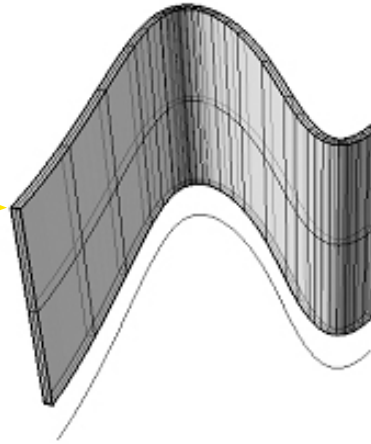
"Join" "Group" and "Explode" commands:

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 are related organizationally not geometrically: 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.
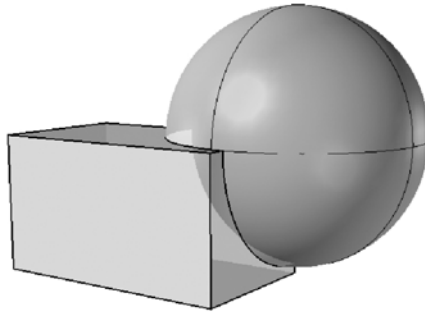
"BooleanUnion,"
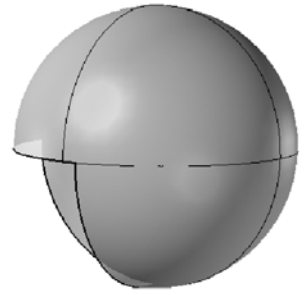
"BooleanDifference 1"
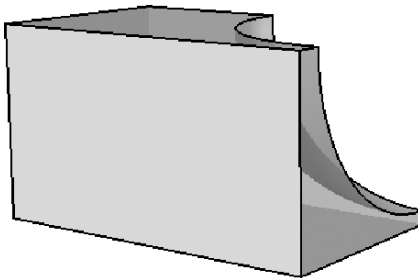
"BooleanDifference 2"
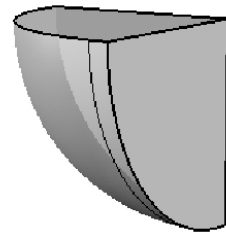
"BooleanIntersection"

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:

Orients an object to another by moving, copying, rotating and scaling 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.
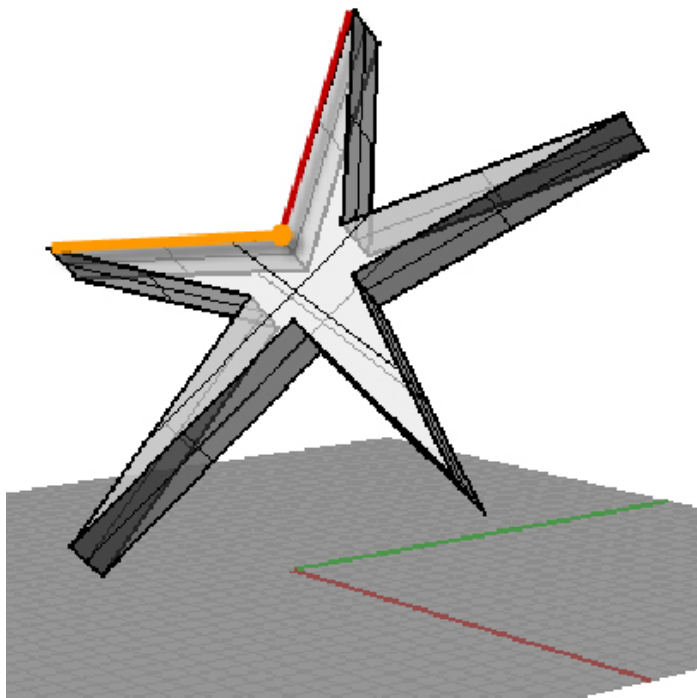
Surfaces

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:
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
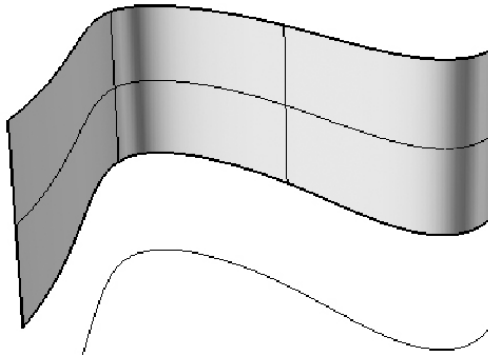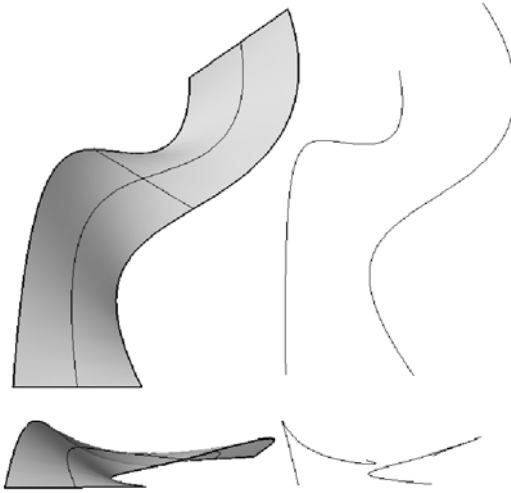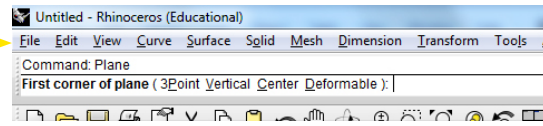
Sweep1:

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 Command Help for an exhaustive description of the options.
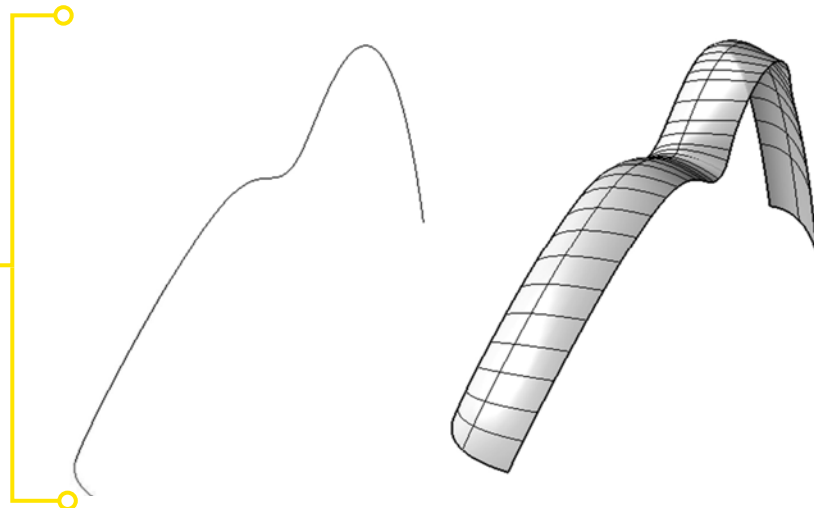
Sweep2:

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.
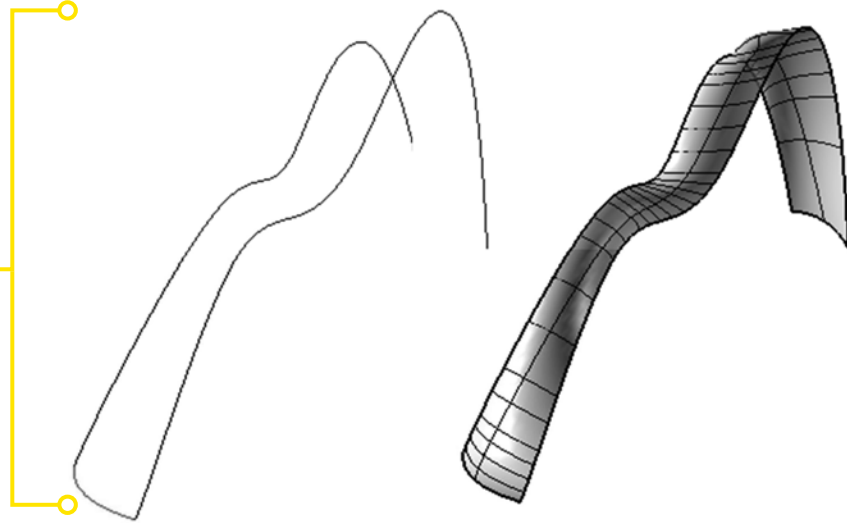
Revolve:

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. Then click Revolve. 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.
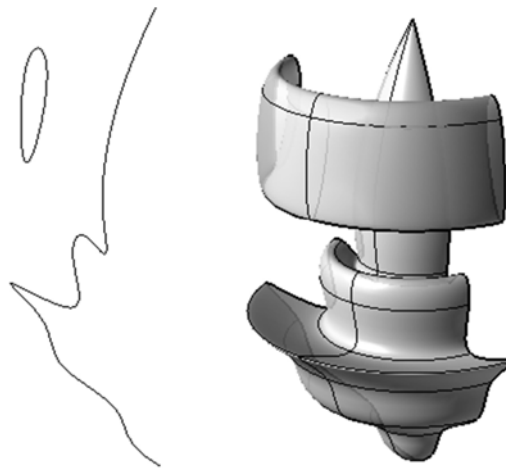
BlendSrf

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 Command-Help. Invoke the BlendSrf 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 affects 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 blended surface.
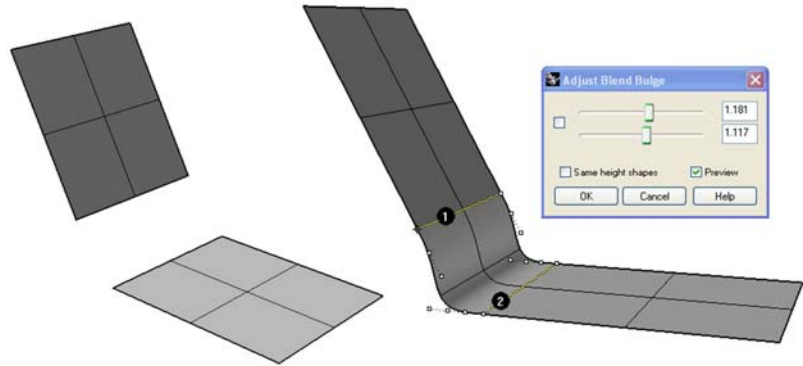
ChamferSrf:
Chamfer creates a beveled edge on two adjacent surfaces. Invoke the ChamferSrf command. Rhino will then prompt you for the two surfaces to be chamfered. Choose the two surfaces and hit enter. After hitting enter, Rhino then prompts you for the distances to chamfer. You can cut more from one surface than the other if you make these distances unequal.
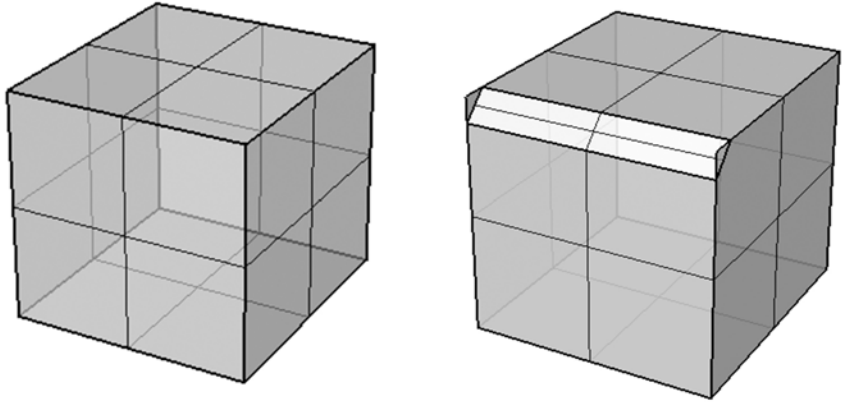
FilletSrf:
Fillet is much like chamfer in that it acts on two adjacent surfaces. Instead of producing a bevel, or cut; however, fillet produces a radius joint. Invoke the FilletSrf command. Rhino will prompt you for two surfaces.
Choose the two surfaces and hit enter. Rhino will then prompt you for a radius to cut. Choose your radius and hit enter.
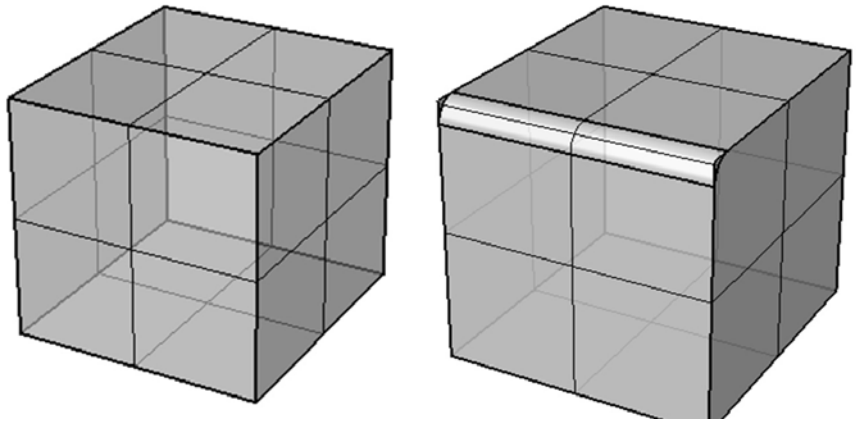
Contour
Contour creates a series of cut planes through an object and makes curves at those locations. It is possible to choose the distance between contours and their direction. Choose an object for contour. Invoke the command. Choose the base point for the contour plane (it does not matter where this is). Choose a second point. The second point determines the direction that the contours will be created, 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.
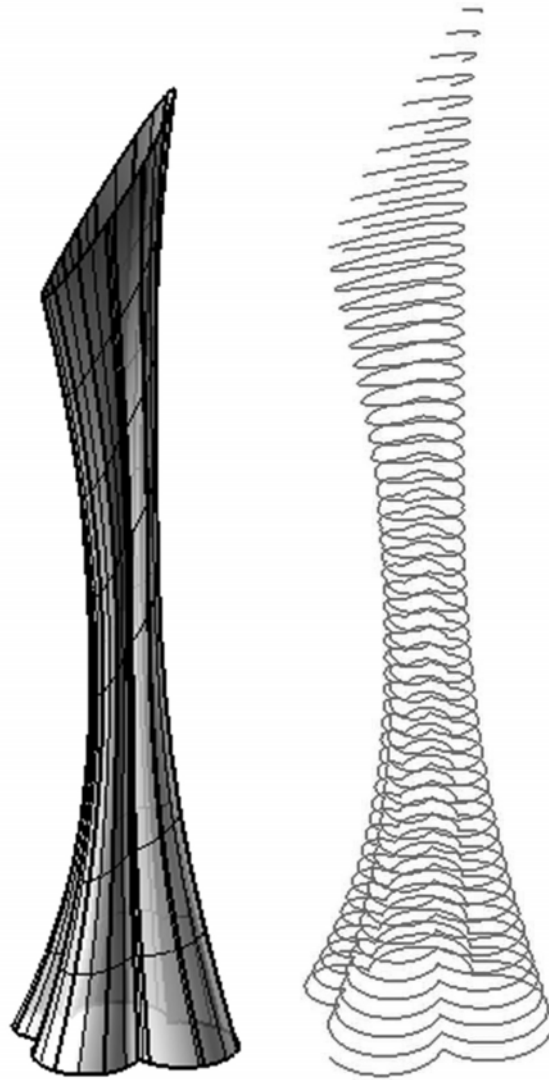
Explode:

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:

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), mirror, array, 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

To create an object in Rhino, click on one of the Solids from the solids tool pallet:
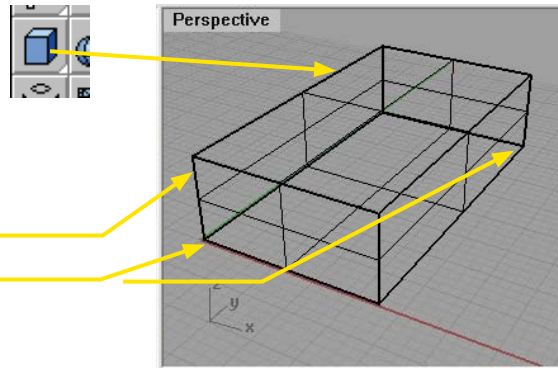
This box is defined by two points and a height.

- 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.
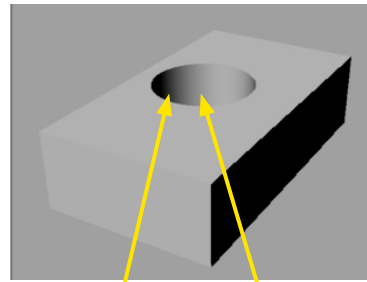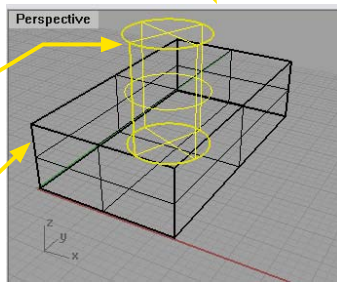
Creating complex objects in Rhino is easy to do using the boolean operations. Simply build the object you want to cut and place it in the correct location relative to the object you want to keep.

- Object to cut (Shown high lighted)
- Object to keep
- Cut object

Perspective

Perspective

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

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 surfaces..

- Control points on, and some points selected:

- Selected points moved and sphere is deformed
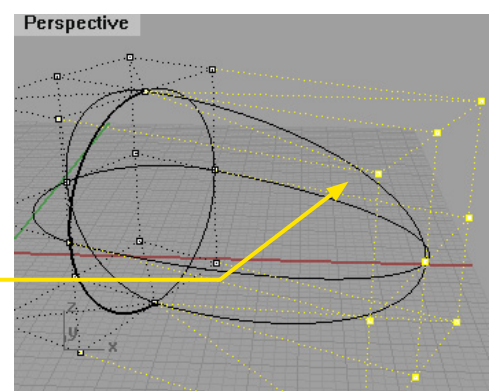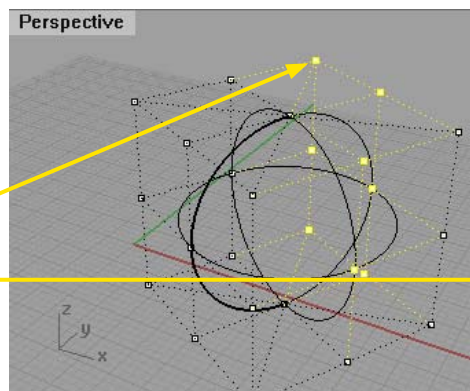
- Rendered Sphere.

This process can be applied to almost any Nurbs surface.
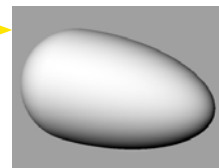
Advanced modeling features

Some of the features below are very specific to Rhino. Others will appear as variations of tools in other modeling programs.

Lofting a Surface

The term lofting comes 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.
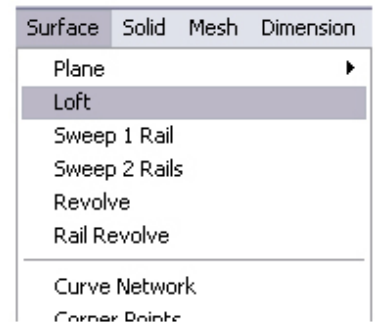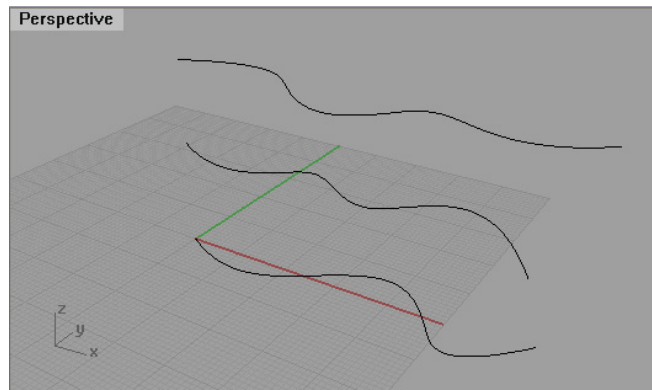
Select Surface>Loft. At the command prompt, click on the splines in the correct order. Click Enter or Right Click when done. The following window will appear: (Note, click to one side of the lines to set the direction of the loft) Now in plan or front view select a series of points. Then click and drag the points stretching out the sphere.
• The 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 that 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.
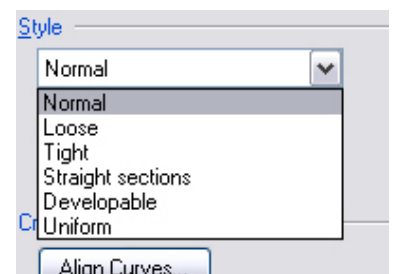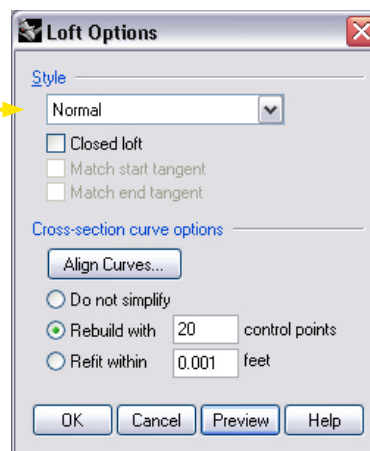
- Selecting this will create a closed lofted surface. Check and click preview to see.

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

- Rebuild With 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.
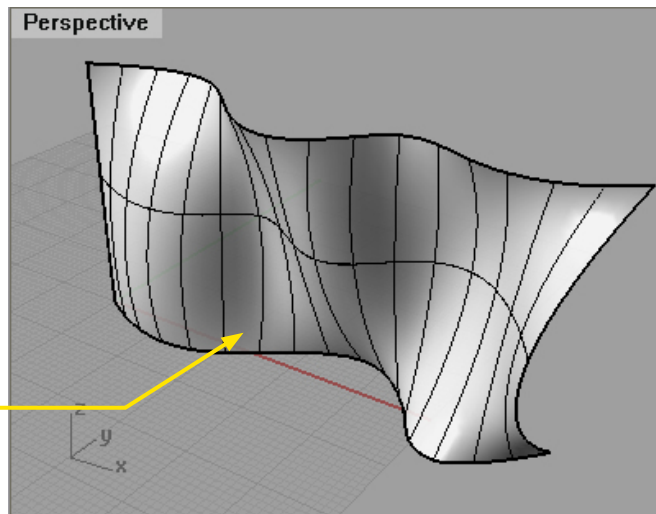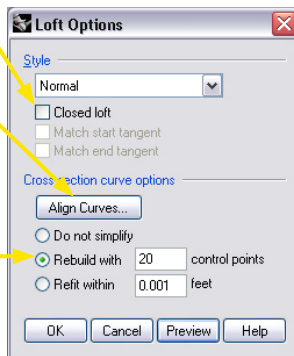
- The Lofted Surface

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 curve 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)

Select a surface or curve, then click on Edit>Rebuild
- Change the number of points in both the U and V Directions
- Keep the Curve Degree to 3 or lower typically
- Click Preview to preview the new Curve or Surface. Click OK to rebuild.
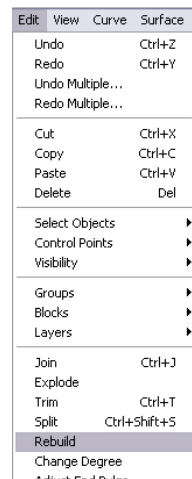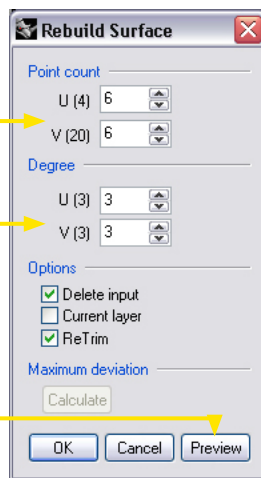
- Rebuilt Curve with only 6 control points

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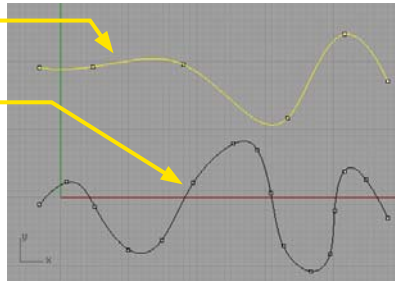
- Original Surface with 20 Control Points

- Same Surface "Rebuilt" with only 6 control points
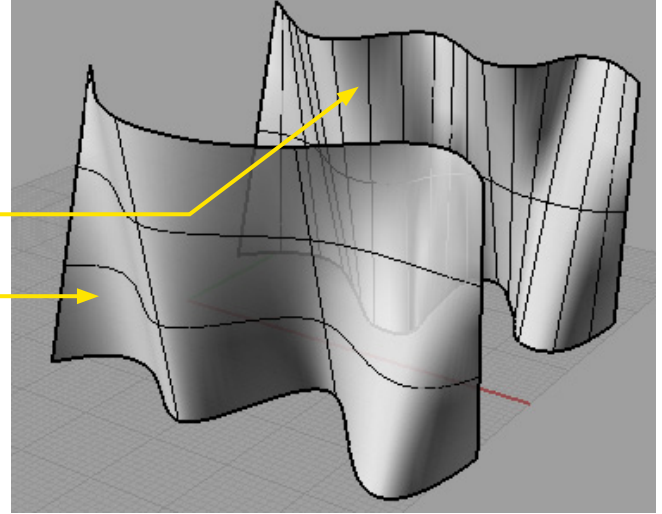
Trim and Split:

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 are 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.
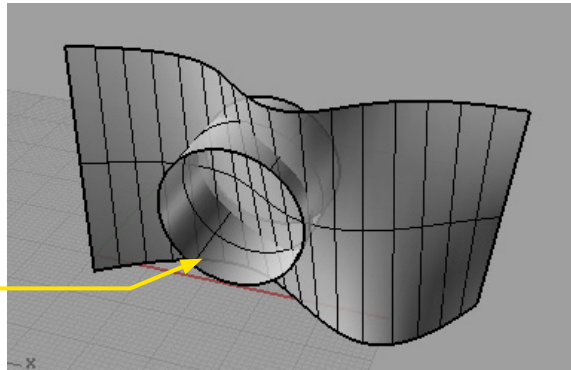
- Click on the Split Tool.

At the prompt, click on the lofted Surface--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 but there are two objects remaining.)

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 El-evation, otherwise the lofted shape would be cut through the perspec-tive 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.
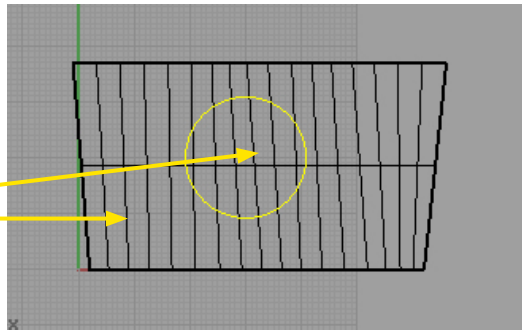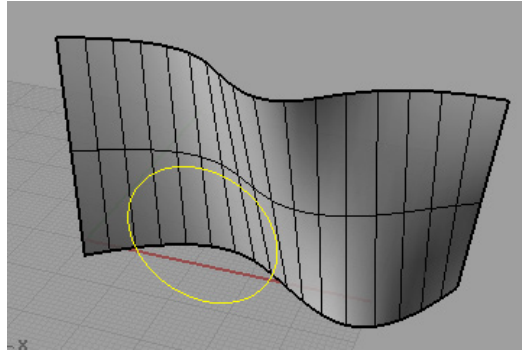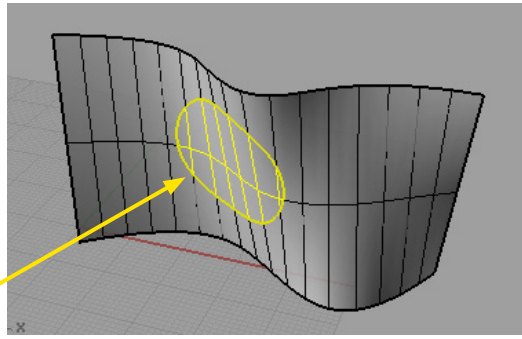
- Trimmed by clicking inside the circle
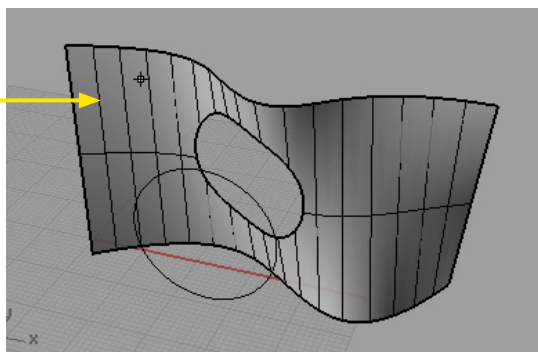
- Trimmed by clicking outside the circle
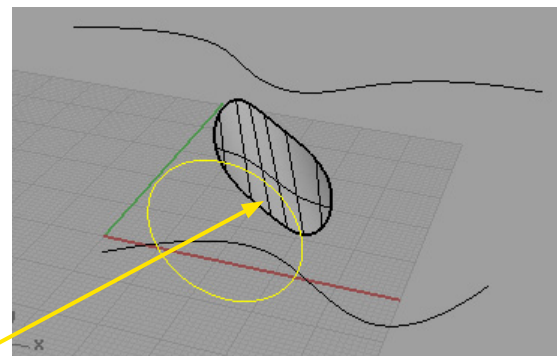
Patching a Surface:

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.

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.

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

Extend Surface

Patch Surface Options

General
Sample point spacing: 1
Surface U spans: 20
Surface V spans: 20
Stiffness: 1
☑ Adjust tangency
☑ Automatic trim

Starting surface
Select Starting Surface...
Starting surface pull: 1
☐ Preserve edges
☑ Delete input

OK   Cancel   Preview

Perspective

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
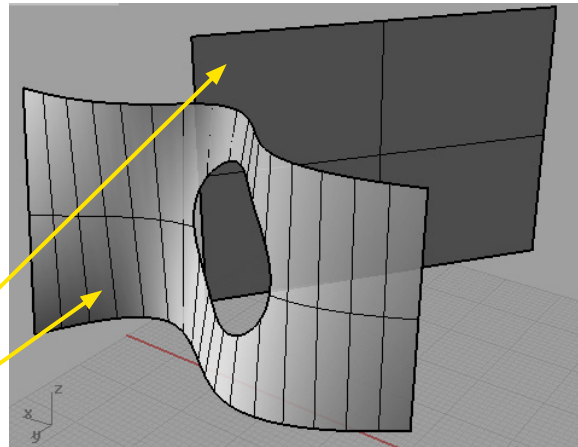
Step One:

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.
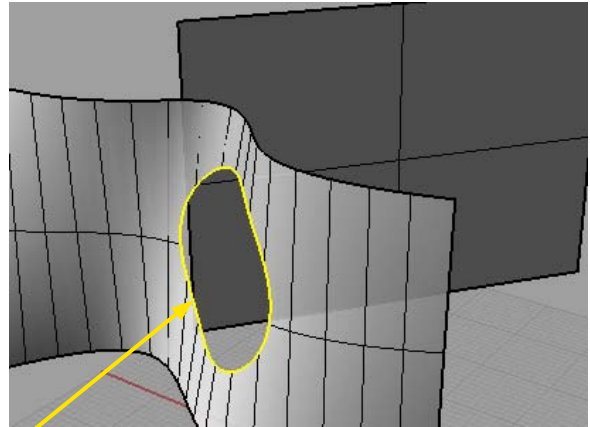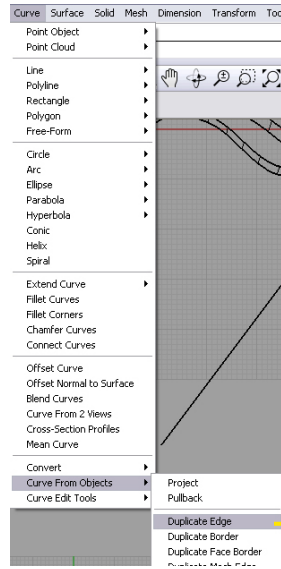
Step Two:
Project the duplicated edge curve onto the surface.

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.

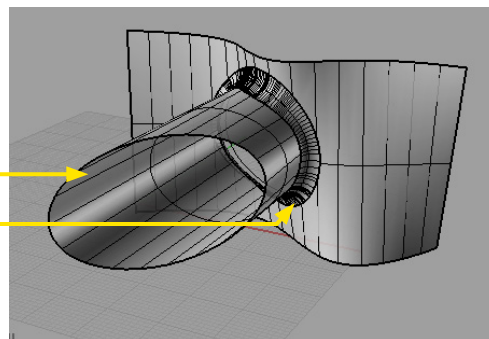- Click the Surface to project onto in an orthogonal view
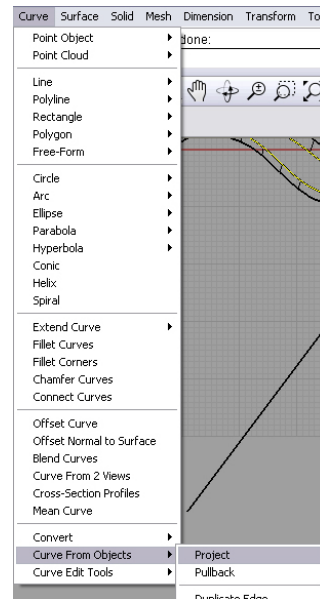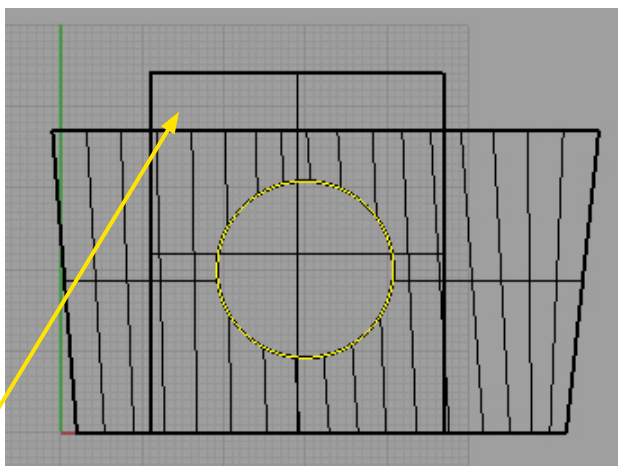
- The curve projected onto the surface.

Delete the surface and you are left with the projected curve.
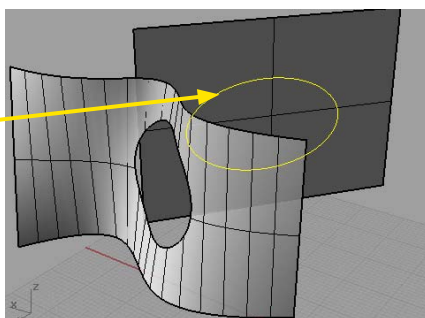
- Lofted Surface

- 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.)
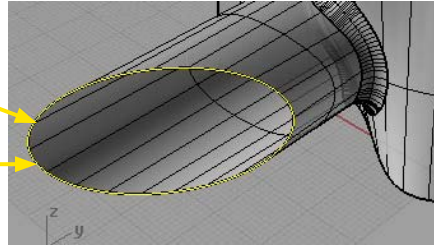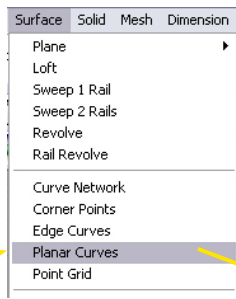

- Capped Surface

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.

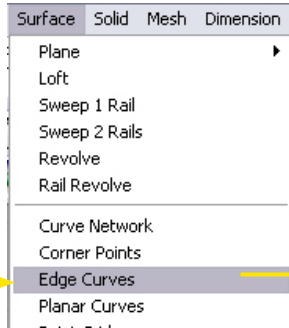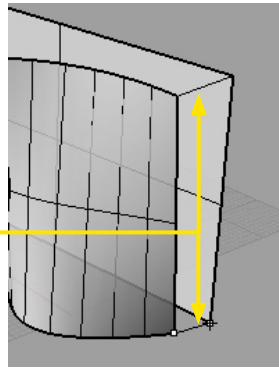A common use for the Surface From Edge Curves is to box out two surfaces.

- Use this tool to create surfaces along the two edges of the existing surfaces. (Note: You could loft it but it is hard to do it on all four sides.
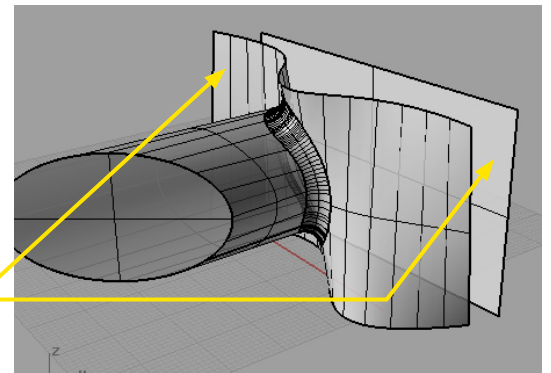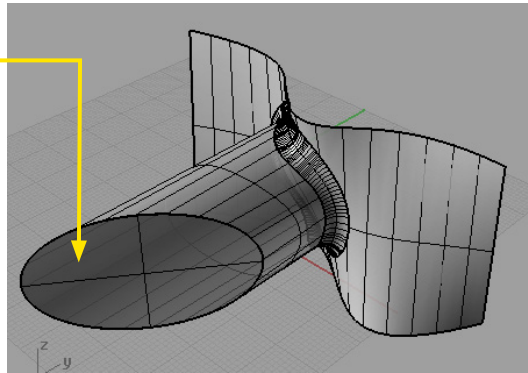
Step One: Snap Curves between the two Surfaces.

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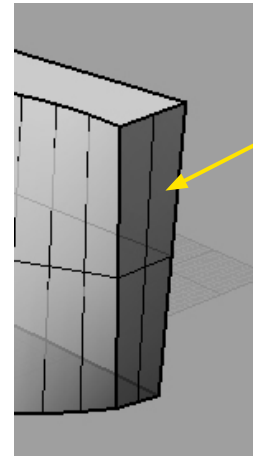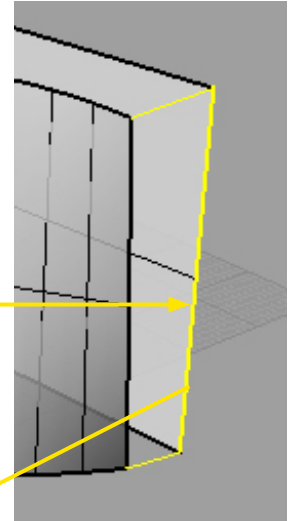
- Two Curves Drawn between the two surfaces.

Surface | Solid | Mesh | Dimension
- Plane ▶
- Loft
- Sweep 1 Rail
- Sweep 2 Rails
- Revolve
- Rail Revolve
- Curve Network
- Corner Points
- Edge Curves
- Planar Curves

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

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

The surface is created.

- Model with all four surfaces created
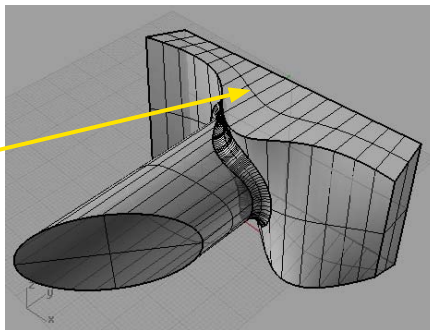
Caging allows you to transform a complex object using a small number of edit points.

Caging is a two step process, first you define the shape (Cage) that will be used to box around the object and transform it.

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
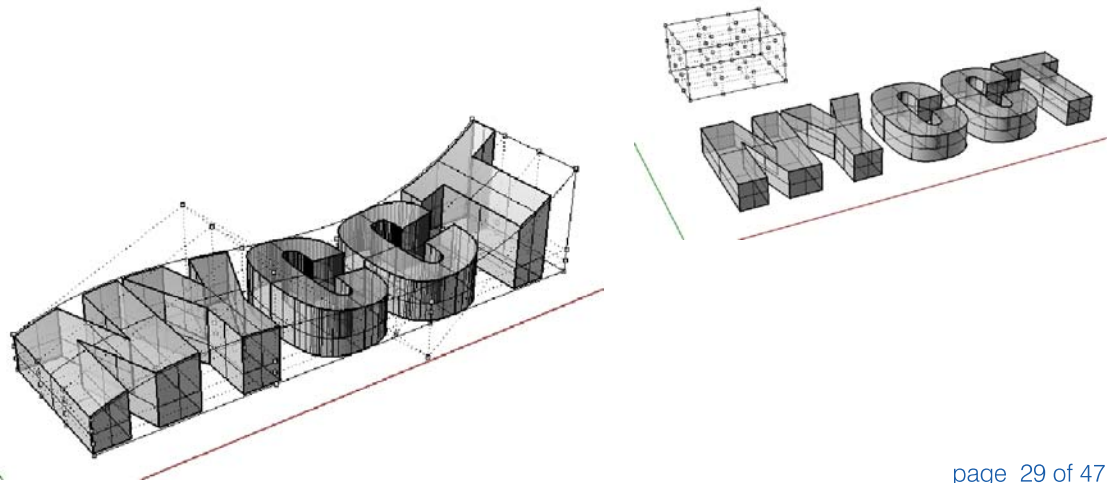
• Transformed Object
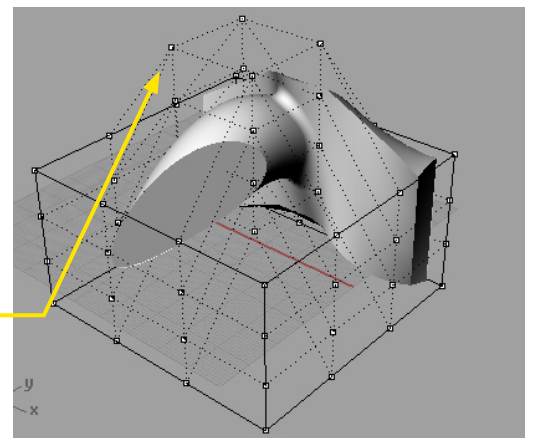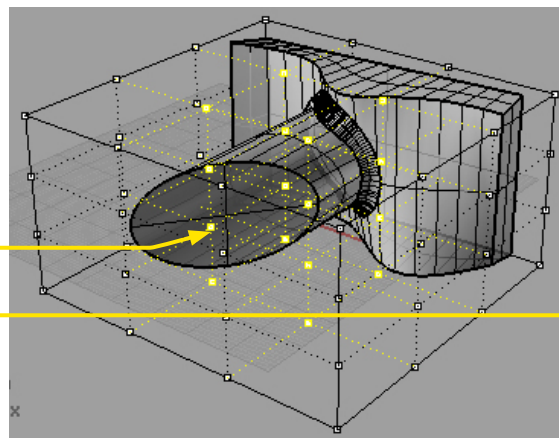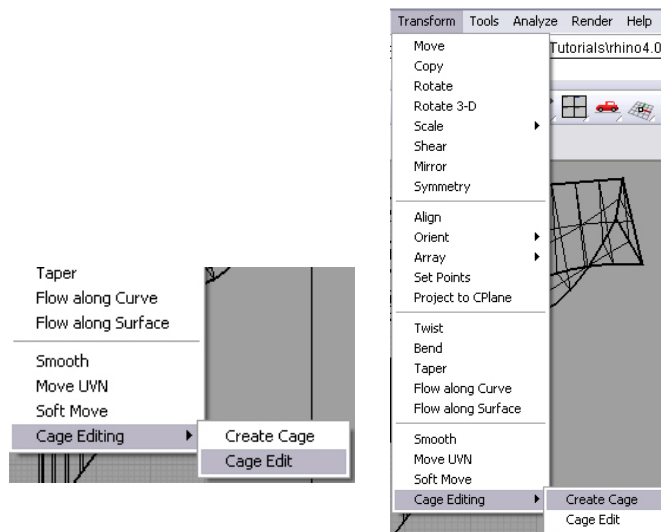
We will explore the options involved in a CageEdit in this section. While we will cover these commands in a detailed way, there is additional explanation and video help available in the CommandHelp in Rhino. Start with a text object: it has a fairly complex geometry, and since the shape of letters is familiar, it will be easy to distinguish the changes taking place. TextObject will allow us to make 3D text. Now invoke the CageEdit command.
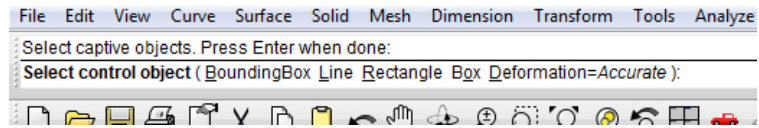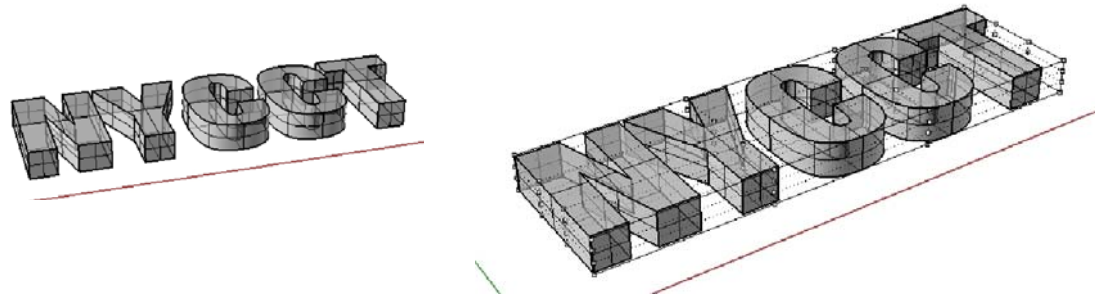
The following options will appear

The control object is the object that we will edit in order to change the target objects. Let's choose Bound-ingBox for the time being. Bound-ingBox automatically generates a box that surrounds the objects selected. Rhino will then prompt for a coordinate system. World is the default, but you can also choose C-plane, if you have defined a custom C-plane in your viewport. We will be working with the world coordinates for these examples. Rhino now prompts you for the number of cage points in the x, y, and z points. These directions correspond to the world or c-plane axis depending on your previous choice. Choose the default of 4 points and hit enter. A cage with corresponding points will appear in Rhino. Pulling these points will change the shape of all of the letters within the cage.

File   Edit   View   Curve   Surface   Solid   Mesh   Dimension   Transform   Tools   Analyze

Select captive objects. Press Enter when done:
Select control object (Boundingbox Line Rectangle Box Deformation=*Accurate*):

Now try it with a different control object. Use the Box option to create a box that is located away from the text. For all of the remaining options use the same settings so it is possible to see what just this single change can do. When prompted, choose Global as the Region to Edit. When we deform the control object, the changes to the target object are very drastic. As a rule, the further away from the control object you are, the stronger the changes.
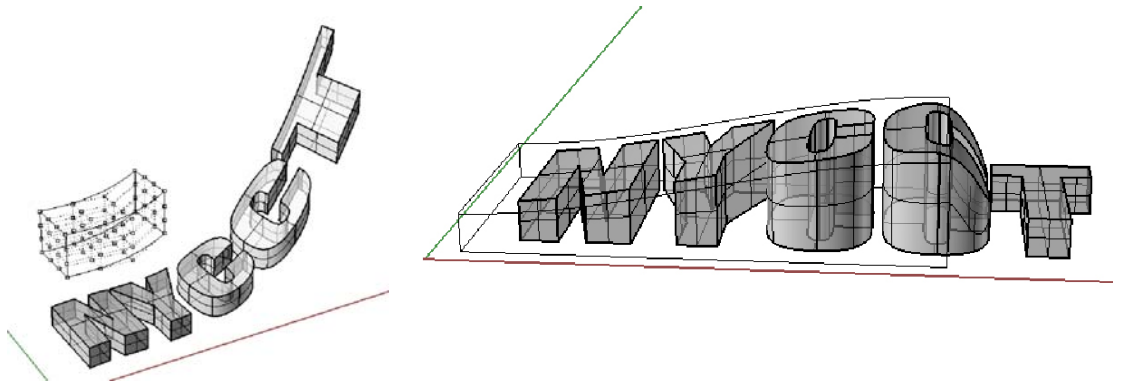
Now create another CageEdit using Box as the control geometry. Place a portion of the object within the box and a portion of the text outside of it. Again, choose the same options as before, but this time, choose Local as the Region to Edit. You will be prompted for a falloff distance. The larger this number, the more of the geometry outside of the box will be affected. The upper image was created with a smaller falloff than the image below.
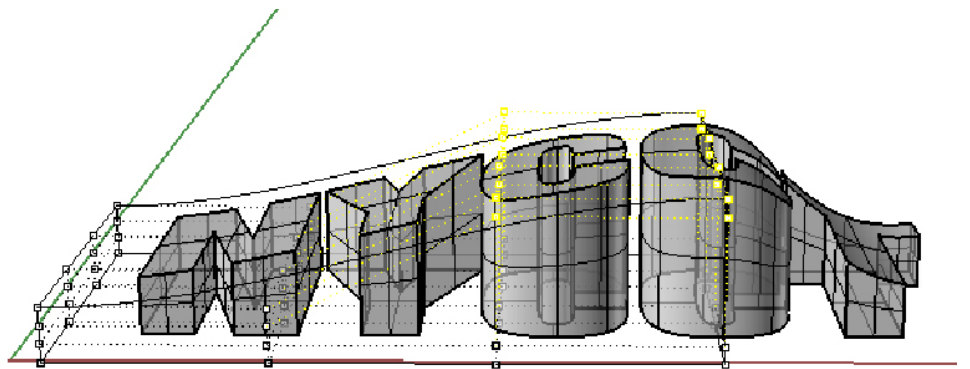
In addition to controlling one geometry with another, we can directly edit the points that make up a surface or a curve, as we saw last week. Move, scale, bend, rotate, and all many other tools also work on control points. To turn on the control points for a surface, hit F-10. To turn them off hit control F-11.
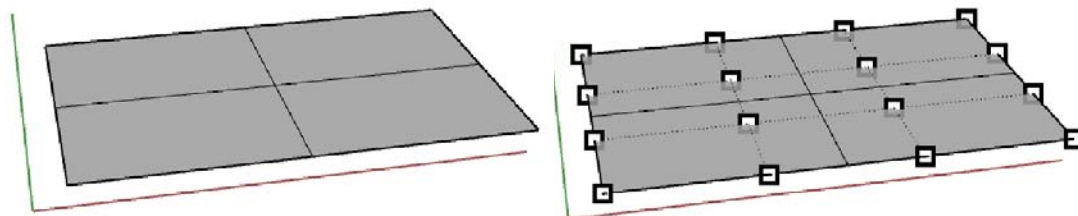
Selu/Selv
You can select an entire row of control points by selecting a single point, and then running the SelU or SelV command.

SetPt
You can align a series of points by running the SetPt command. Choose the points you would like to align and then enter an orthogonal viewport. Invoke the SetPt command. A dialog will appear. Choose the axis to align on, hit ok, and then choose a single location for the group of aligned points in the viewport.
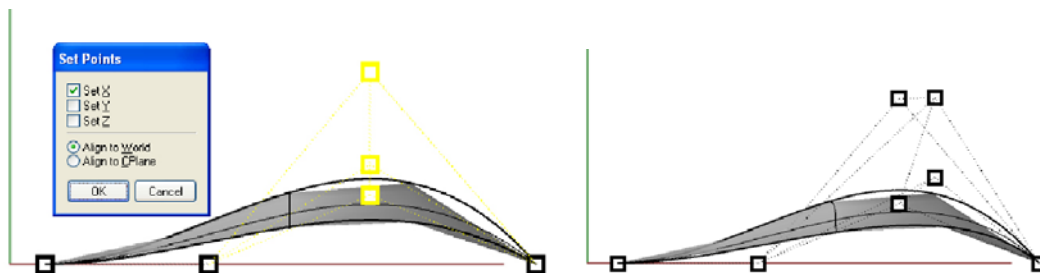
Smooth:

It is also possible to smooth a surface. Choose the control points to smooth, and then invoke the Smooth command. A dialog box appears asking which direction to smooth. You can choose any combination of x, y, and z. The smooth factor controls how much the surface control points are smoothed.

SoftMove

Weight – control point weight dialog

Edit>Control Points>Insert control point u or v toggle direction, changes shape
InsertKnot inserts control points and maintains shape of surface
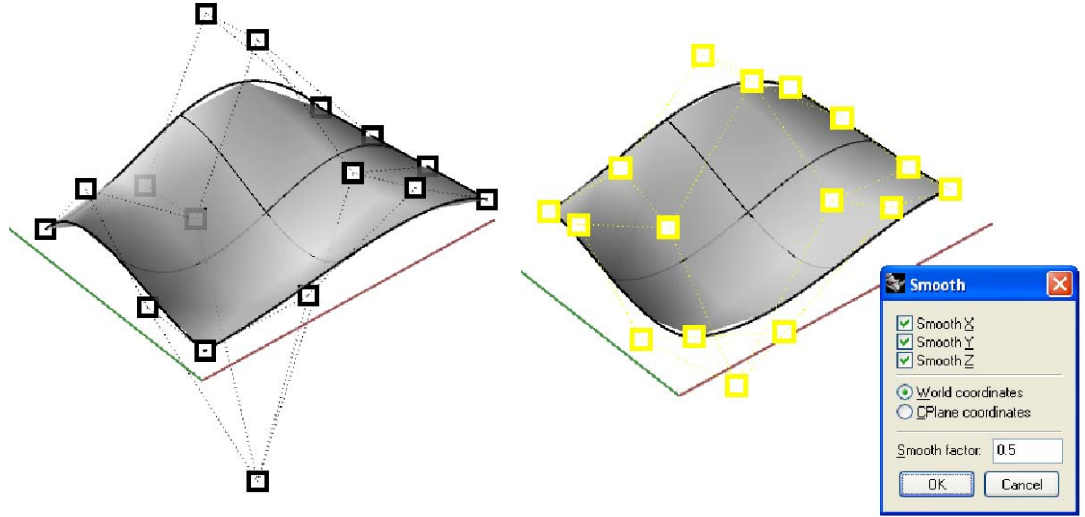
Blend Surface Feature:

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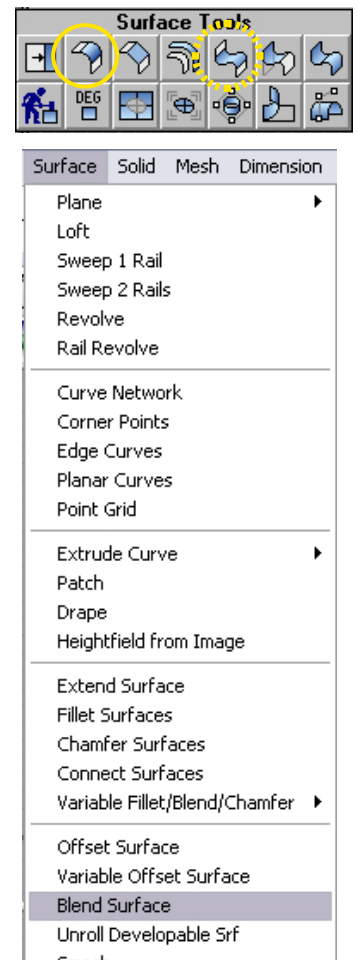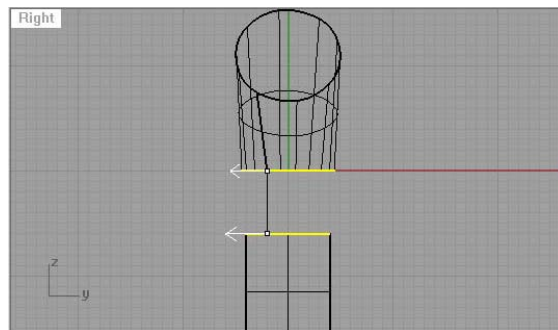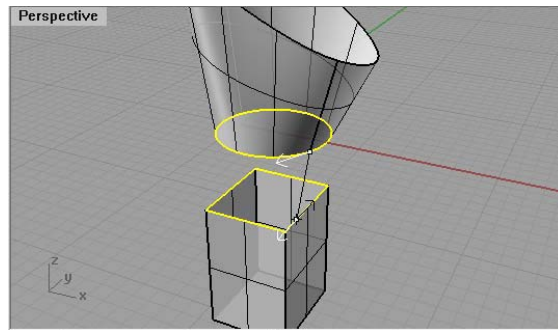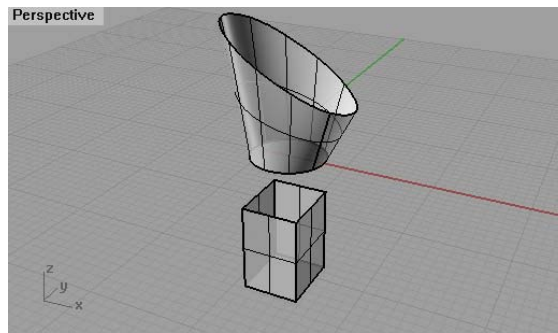
The command prompt will now say "Select Second Set of Edges". Select the second set of edges, and hit return or right click.

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.
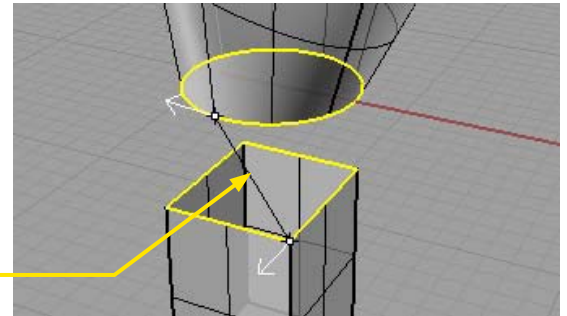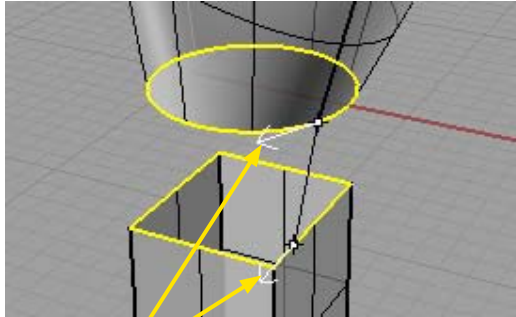
- Two Arrows

- 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.
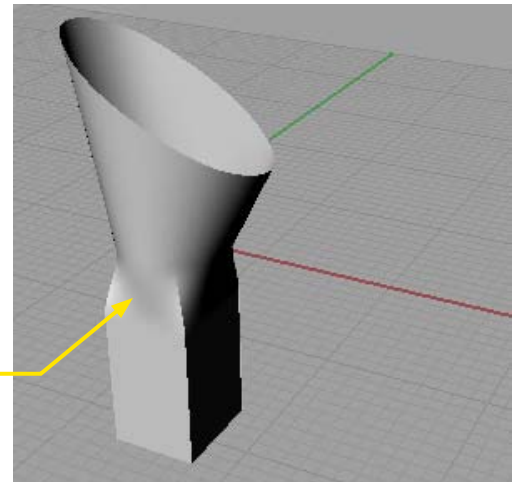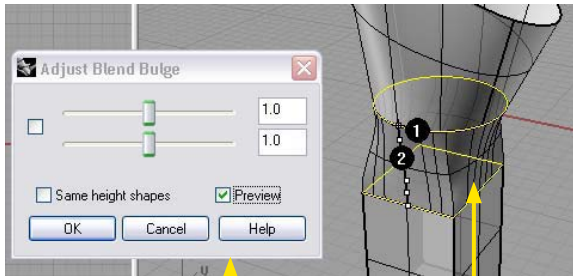
- A preview button will appear where you can adjust the settings.

- Generated Surface

Adjust Blend Bulge

1.0

1.0

Same height shapes   Preview

OK   Cancel   Help

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.
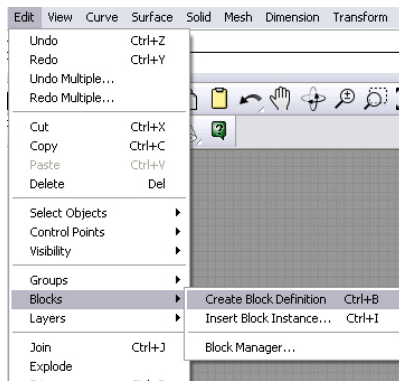
• Complex Form that will be turned into a block.
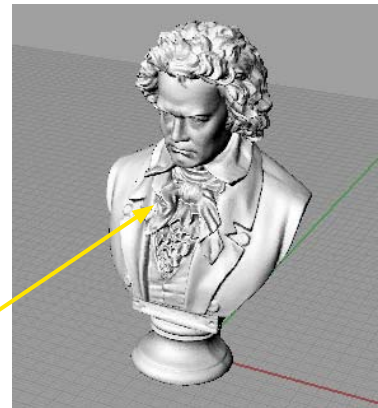
Select Edit>Blocks>Create Block Definition.

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.
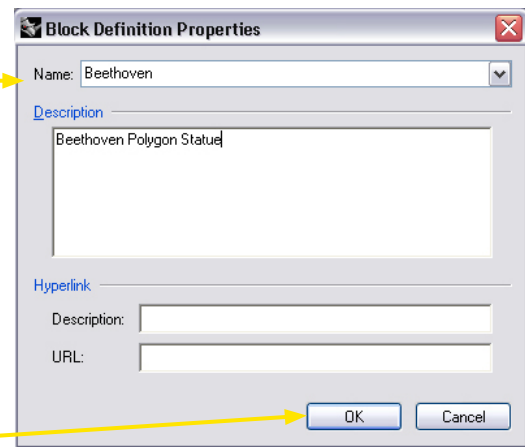
Inserting a Block Definition:

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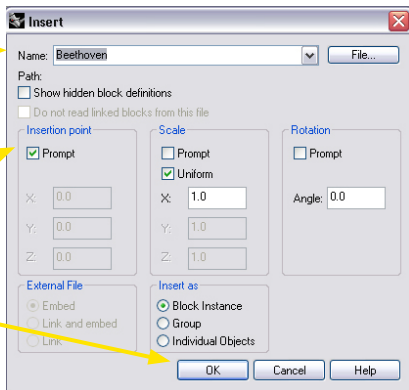
- The block will be bound to the mouse and when clicked, will be inserted
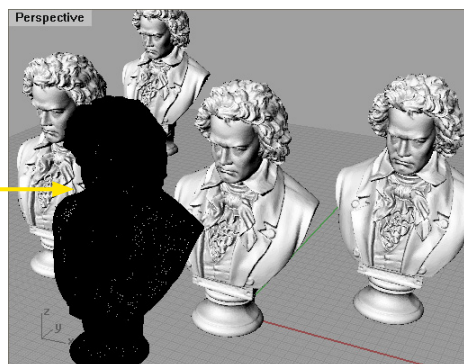
Linking External Models as Blocks:

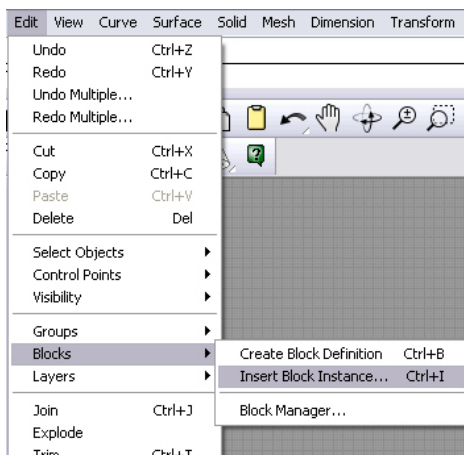Another powerful way to use blocks is to link a series of external models into a "Master" Rhino Model. This allows updates in different models to be automatically updated in the master model.
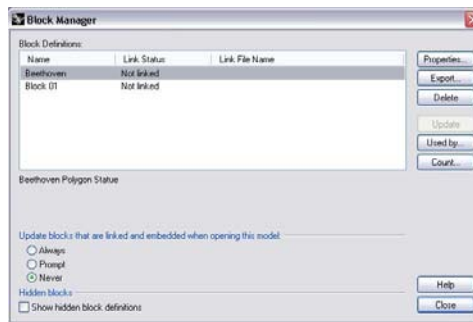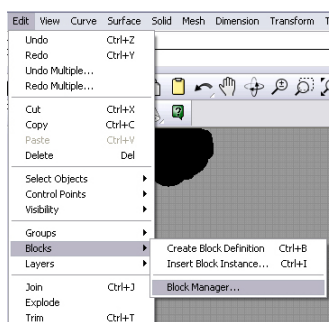
- Step One: Create a Block from an external file. Select Edit>Blocks>Insert Block Instance
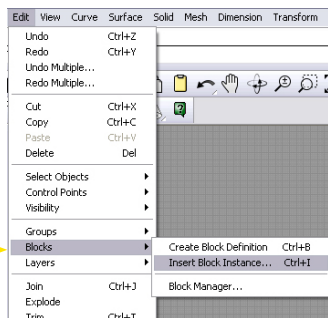
- Select File

- Select a Rhino Model to insert
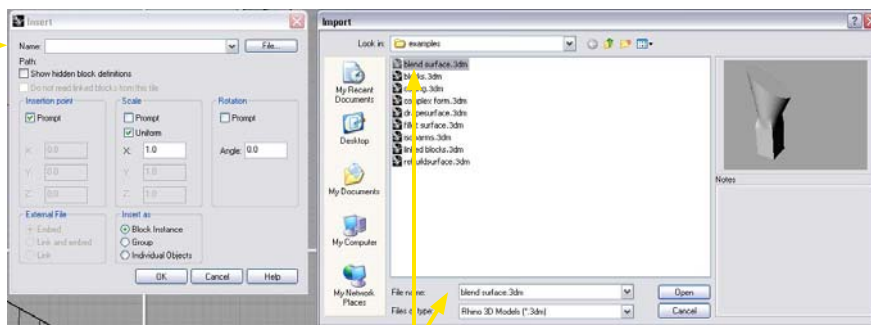
- Linked File Name and Pathway is displayed

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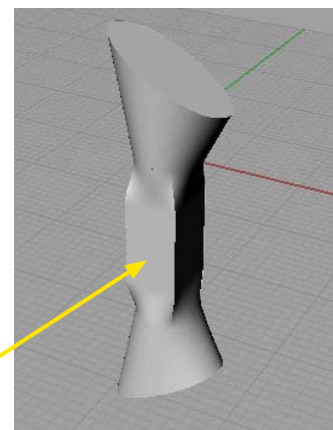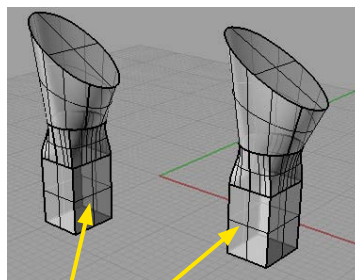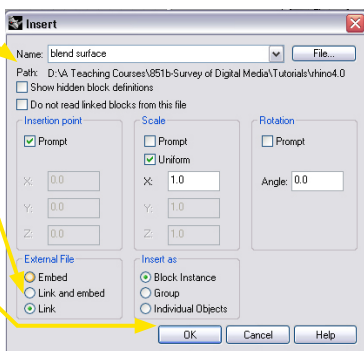
If changes are made in the original model the changes will appear in the model with the block definitions.

- Change Made to Original Model File
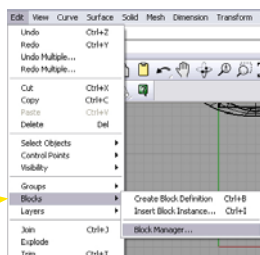
- To update the Link Model, click on Edit>Blocks>Block Manager

- Select the Block to Update, and click on Update.
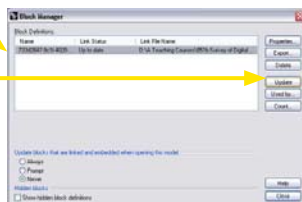
- Blocks are Updated
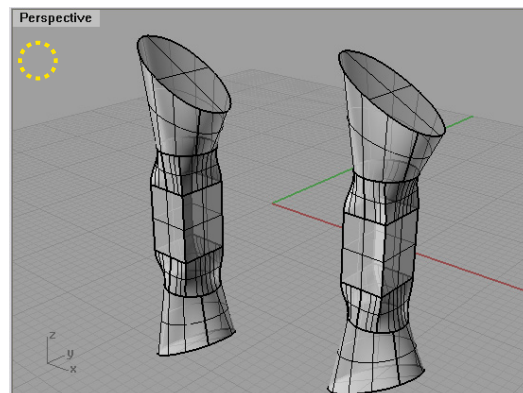
Make A 2D Drawing:

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 drawing, click on Dimensions>Make 2-D Drawing.
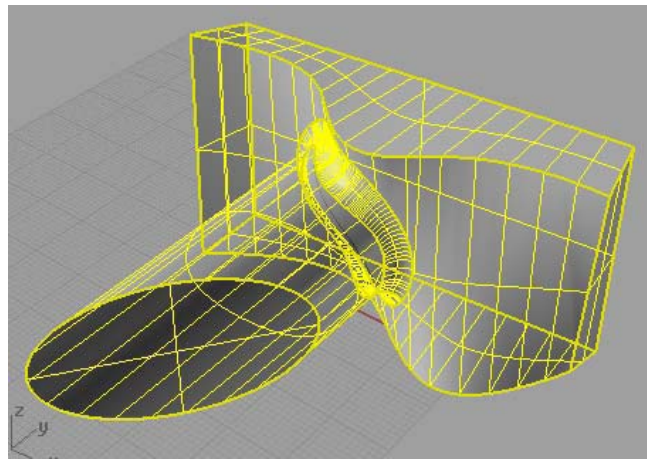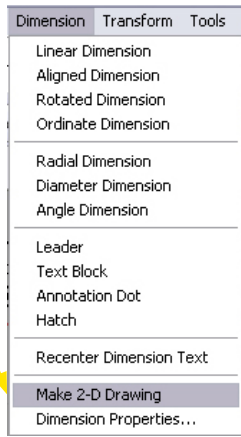
At the prompt, select objects to draw, and select the objects.

The following Window will appear:

- Choose 4 View USA

- Click OK

- Four Views, Top, Front, Left, and Perspective are created.
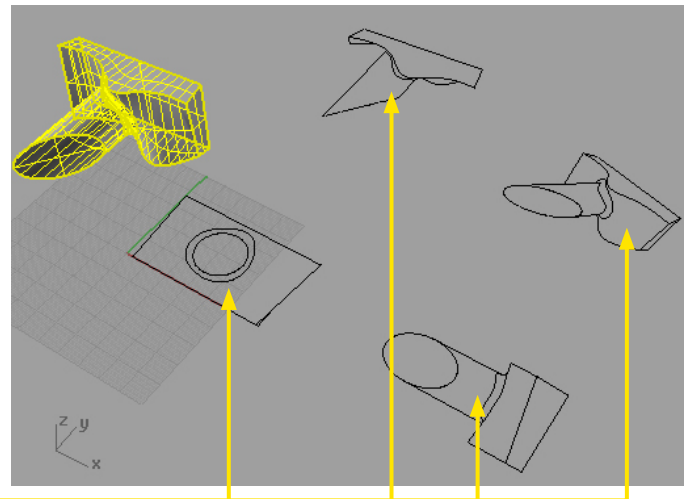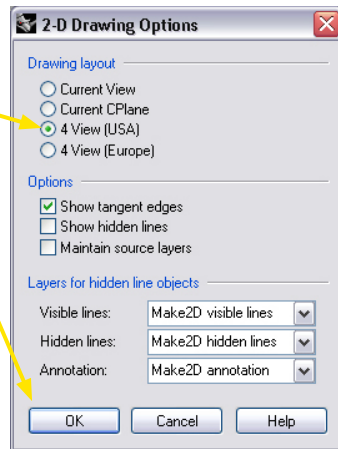
These views are located on the ground plane. If you select a view and export it from the top view it will be oriented in the correct direction.

Basic Rendering

Key – The key light is the main light and should be approximately twice as bright as your fill light for regular scenes, but can be up to 8 times as bright for dramatic shadows.
Fill – The fill light helps to bring up the level of the shadows.
Back – The back light should be bright enough to create a light halo around the back and side of your rendered object.

Basic Lights:

Directional – Light that is similar to the sun (it fills the render space with a uniformly intense amount of light) and creates a directional shadow

Spot – Light that is similar to a stage spot-light. This has a limited range of effect, so objects outside of it will not be lit.

Point – Like a candle flame or light bulb. Light radiates from a point. Limited range.

Rectangular – Light radiates from a plane.

Linear – Light radiates from a line.

Set the Renderer to Rhino renderer by going to : Render> Current Renderer> Rhino Render

In this dialogue, you can change the resolution of your image, the background color, set a background image, and change other options.
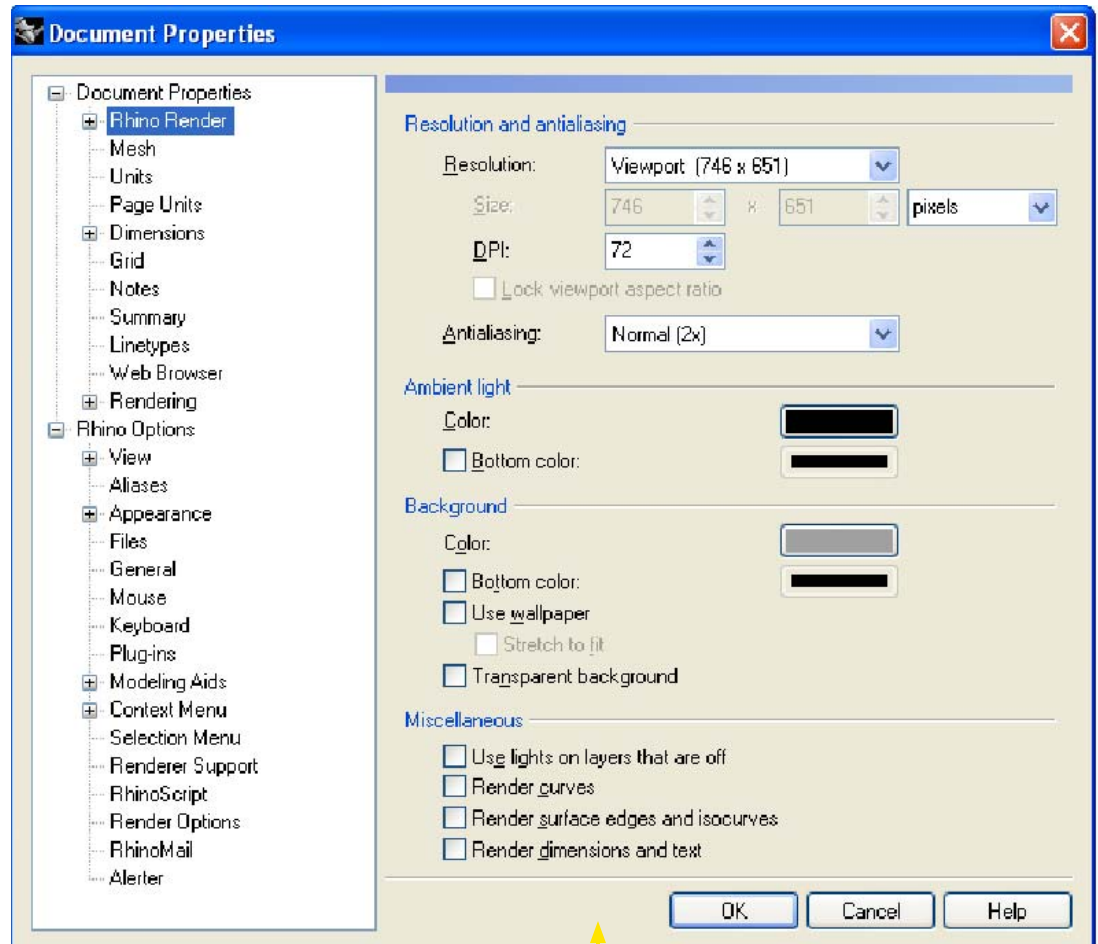
Constructing Page Layouts for
Printing, Background colors, and
Orienting objects

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.

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 choos-
ing the "Select View" icon, change to a
different view.

4. To lock at 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.
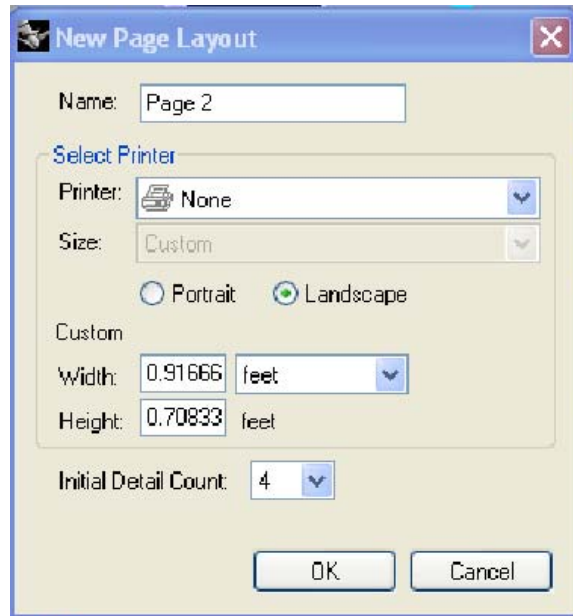
Background color
Background colors for the screen are
found in the Options > Appearances >

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
It is recommended that you choose AutoCAD 2004 as your filetype when you SaveAs. The 2007 and 2010 filetypes WILL typically 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 Auto-
CAD drawing file in the type drop-
down.
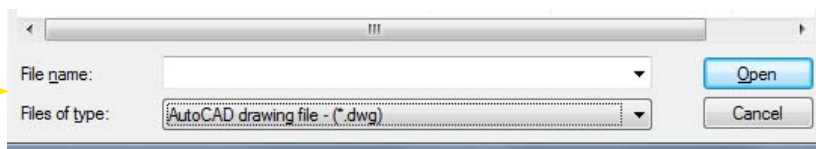
Your geometry should appear in
your Rhino model. Before doing
anything else, group the drawing.
This will allow you to select it as an
object, rather than line by line.
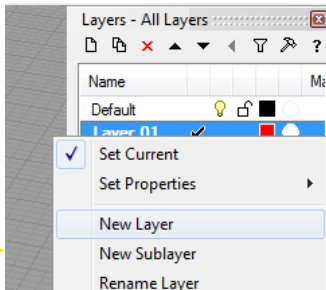
Create a new layer (right click on
the layer palette> New Layer):

and place your plan on it (select the
plan, right click the layer you made
and choose Change Object Layer):
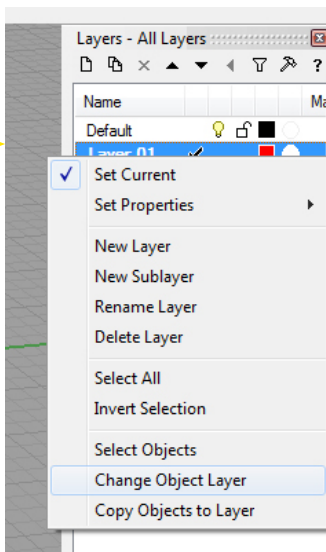
Verify your units and dimensions
match in AutoCAD and Rhino by us-
ing the distance command in both
AutoCAD and Rhino on the same
geometry.

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

For some of you who are coming from AutoCAD 3D or SketchUp, the workflow in Rhino may be a bit foreign. This worksheet will describe two possible methods for drawing in 3D space.

This is the default setup for viewports, which will be familiar to many of you. Each view is linked to a camera, and can be changed by right clicking on any of the viewport names.

It is best to understand how to draw in 3D fairly early, usually right after exploring the interface. In this viewport, I've drawn a line, which appears to be starting on the grid, and extending upward in the Z direction.

This is the reality. In general, when you draw in a viewport. Unless you specifically tell Rhino to draw in the 3rd dimension or OSnap to another object, you will be drawing on the 2D construction plane (the plane of the gray grid—called the CPlane from here on) shown in the view.
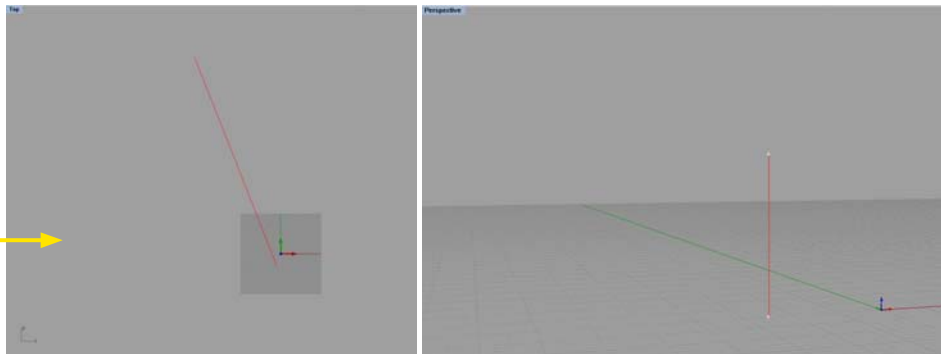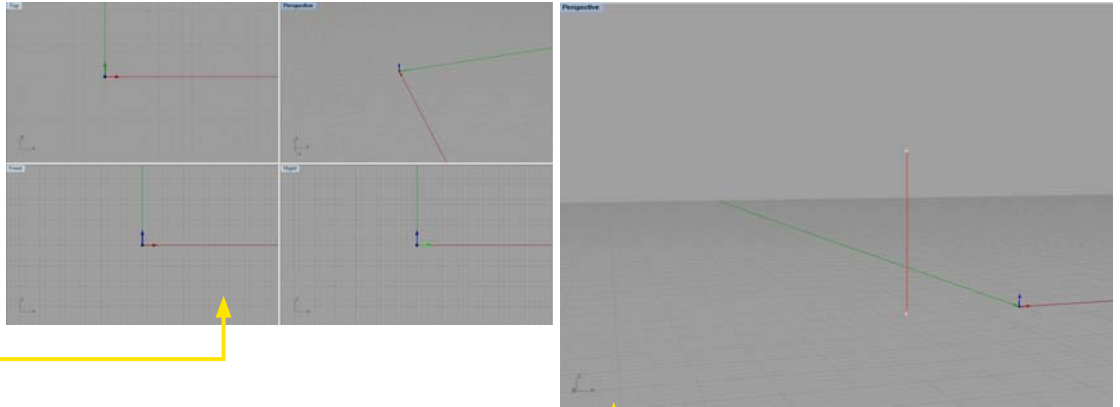
The first way to draw vertically (or in the 3rd dimension not represented by the CPlane, as this is possible in any viewport) is to use the vertical command line option when invoking a command. Many very basic commands have this option.

The second is the coordinate system. When drawing a curve, for example, you are able to input coordinates for points using the command line in the x,y,z format. This is going to use the coordinate system defined by the CPlane of the viewport you are working in. If you modify the CPlane, make note of this!
The third is to draw objects to use as a template flat on the CPlane and to simply rotate them in 3D the appropriate amount. This can take some time (you are still only drawing in 2D and then often need to model further after placing your templates) but can be a valid method for working with certain object profiles.
The fourth involves modifying the CPlane and can be difficult for some students to master. (This is generally an intermediate skill: introduced this after one has grasped the idea of how the viewports are working, and feels comfortable creating 3D geometry). You can create a mobile construction plane tied to a piece of geometry that you can then use to edit your project:

Right clicking on a viewport brings up the following context menu where you can change viewport properties and edit which camera is being used. It also provides us with the option of creating a mobile CPlane.

If we have the following geometry, and we want to create a mobile CPlane in the Front view, we invoke the command and see the following prompts:

Command: Line
Start of line (Normal Angled Vertical FourPoint Bisector Perpendicular Tangent Extension BothSides):

Command: _MPlane
Select object

Command: _MPlane
Choose mobile plane option (Attach Detach Modify Refresh Show):
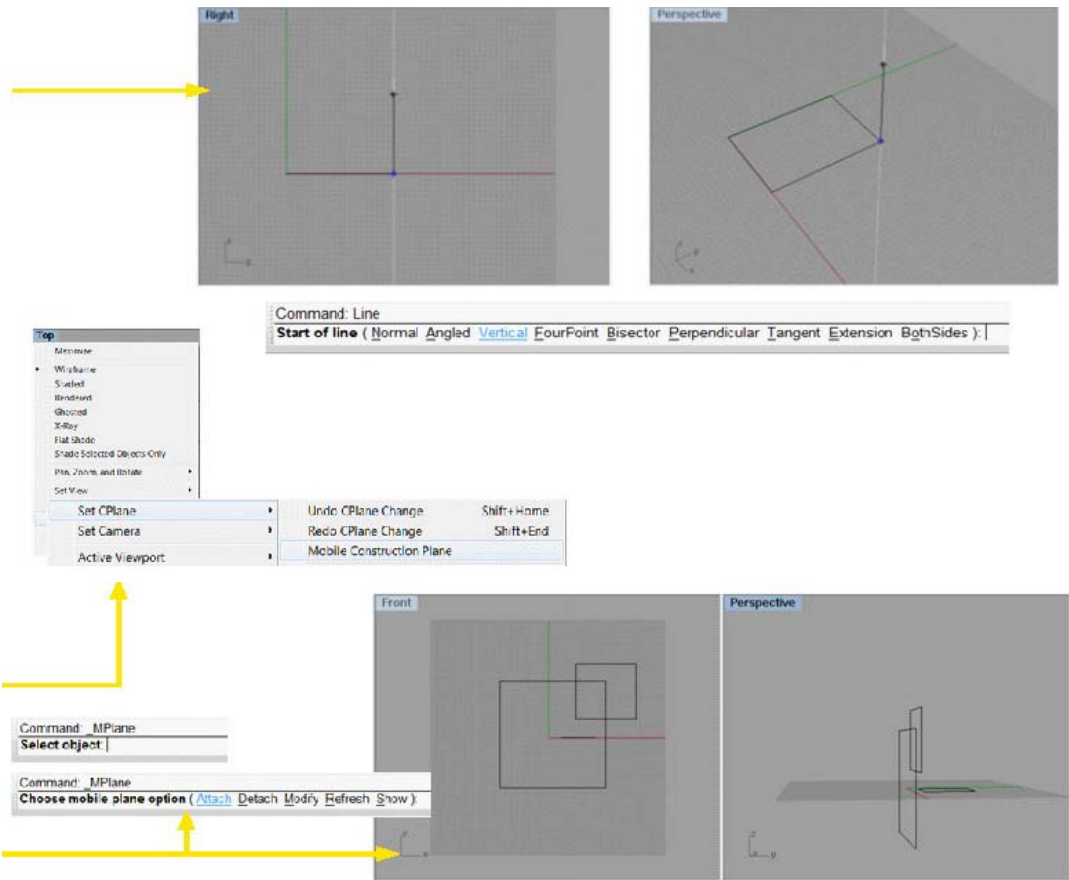
We then choose the geometry to represent our mobile CPlane. I've chosen the small rectangle.
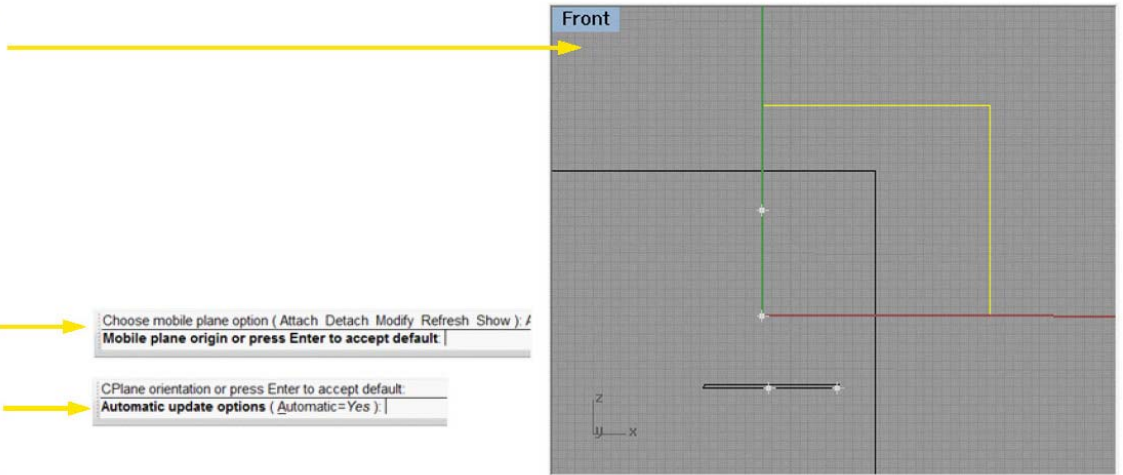
We will then be prompted to choose a coordinate system starting with the origin, and proceeding to the axes.

Choose mobile plane option (Attach Detach Modify Refresh Show): /
Mobile plane origin or press Enter to accept default: |

We will then be prompted if we want the CPlane to automatically update when we move the geometry.

CPlane orientation or press Enter to accept default:
Automatic update options (Automatic=Yes): |
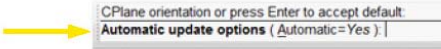
Saying yes will cause the CPlane to move when the geometry moves, as can be seen here.
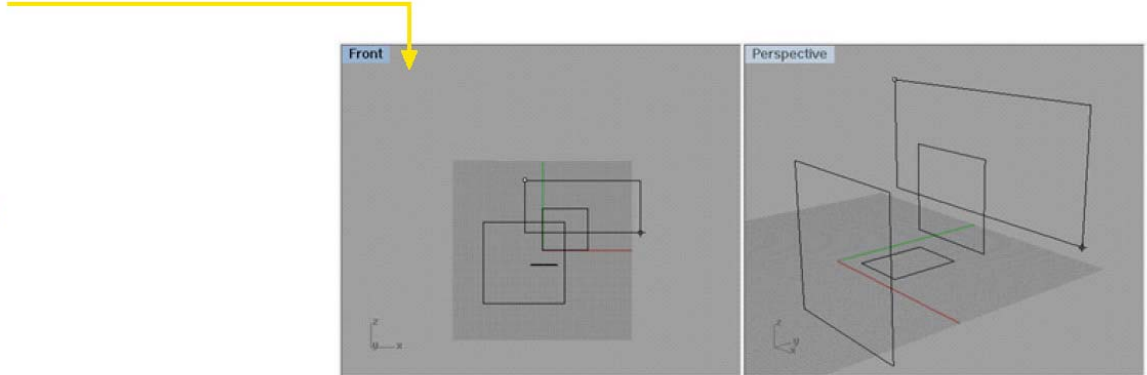
Remember to think about breaking down modeling problems as a set of procedures. There are many ways to create a piece of geometry and it is best to approach any object that needs to be created as a problem that needs to be solved. Often one will see the end goal and not be able to get beyond it's complexity. Even the most complex objects can be broken down into a series of simpler operations.

Tips for Users of Rhinoceros for Creating Models for 3D Printing

Rhino can be a valuable tool for both the creation of 3D mesh and as a tool for conversion of parametric based geometry to polygon geometry. Rhino also imports a large number of file formats, which can be converted to the .stl file format.

Exporting to .stl

Prior to exporting to .stl format follow these diagnostic tool to check the quality of the solid.
.stl Mesh Export Diagnostics
For some rapid prototyping machines, .stl files must contain completely closed (watertight) polygon mesh objects. You might want to do this to make sure the meshes really do fit together before exporting them for use in an expensive .stl job. Use Join, then Weld (angle=180), UnifyMeshNormals to turn a group of meshes into a single watertight mesh object. Then use SelNakedMeshEdgePt to find the open (naked) edges.

Watertightness

• Select the mesh objects.
• From the Edit menu, click Join.
• Conceptually this gets all the triangles into one bag, but does not glue the edges together. (The situation is similar to having a bunch of surfaces that all fit together but have not been joined into a Solid.)
• Select the new mesh object.
• From the Tools menu, click Polygon Mesh, then click Weld.
• At the Angle tolerance prompt type

180. An angle tolerance of 180 tells the Weld command to glue adjacent triangle points together no matter what.
• From the Tools menu, click Polygon Mesh, then click Unify Normals. This changes all the triangles so they are oriented the same way, that is, if two triangles share an edge, then they have the same idea of up.
• To see if the result has any holes or gaps, type SelNakedMeshEdgePt. If a mesh point is highlighted, then it is part of a "naked" triangle edge.

Note

When exporting NURBS objects to .stl, Rhino converts the NURBS objects to polygon mesh objects. If Rhino cannot create a closed .stl file, the Export Incomplete Solids dialog box asks if you want to export anyway.
The tolerances you set in the Document Properties dialog box, Units page do not affect the .stl export tolerance. Instead, to control .stl accuracy, use the mesh tolerance settings from the Detailed Controls section of the mesh dialog that appears during .stl export. Set the Max distance edge to srf value to the .stl tolerance you desire.

Weld Polygon Meshes

• Select the group of polygon meshes objects.
• At the Angle tolerance prompt, enter a maximum angle between mesh polygon normals where welding of points should occur. If two naked mesh edge points of a selected mesh are coincident and their neighboring faces are within the angle tolerance of each other, they are replaced by a single mesh

point. If the two points were from different "sub-meshes" of a joined mesh, the meshes become one mesh that cannot be exploded.
• To perform a similar operation with control points, see the SetPt command. Welding polygon meshes affect rendering, texture mapping on mesh objects, and file export for stereolithography.

Unify Normals

UnifyMeshNormals is primarily for making sure the order of the vertices in all polygons in welded polygon meshes are the same.
• Select a single unjoined mesh. The normals of all the faces of the mesh now point to one side of the mesh. The UnifyMeshNormals command changes the direction of the surface normals of a mesh object so they all face the same direction. This command is useful for tidying up your mesh objects for export into 3D Studio.
To test if the mesh needs to be unified:
• File menu: Properties
• On the Render page, clear the Render Backfaces checkbox.
• Render your mesh. If some of the faces that should render disappear, their normals are flipped. Use UnifyMeshNormals to make them all match.

Note

If UnifyMeshNormals does not seem to affect your mesh, explode it. Once the normals are correct, re-join the meshes.
A polygon can have two types of normals: vertex normals and a face normal. All polygons have a face normal direction, but many polygon meshes do

not have vertex normals. For example, 3DFace object, all mesh primitives and all polygon meshes imported in formats other than 3DM and 3DS do not have vertex normals at all.
In general, the order of the vertices in a polygon determines the face normal direction. The order of the vertices should be either clockwise or counterclockwise. The normal direction is determined from the vertices using the right hand rule.

Select Naked Edge Points

The selected naked edge points command shows where polygon mesh vertices are not completely surrounded by faces. Joined meshes, such as are made by Mesh Box, have naked mesh edge points where the sub-meshes are joined.
To eliminate naked mesh edge points, move them to make them match in pairs, then weld them to make a single mesh that cannot be exploded. Edges that formerly rendered sharp will shade rounded, unless you use FlatShade.
• .stl Tools toolbar: Show and Select Naked Mesh Edge Points
Command line: SelNakedMeshEdgePt
• Select polygon meshes for naked edge point search prompt, select polygon meshes and press Enter. Points at edges of polygon meshes are selected.