# Model Preparation in Rhino

These instructions assume a basic working knowledge of modeling in Rhino. If you are unsure about your modeling skill, we recommend that you work with a faculty member who can guide you through the modeling process.

Units

We recommend modeling in either inches or millimeters, especially if accuracy of size is important to your part. The units you use during modeling must correspond to the actual units used during output. To set the units, click on 'File -> Properties…' then click on 'Units.' Set the 'Model Units:' to either inches or millimeters.
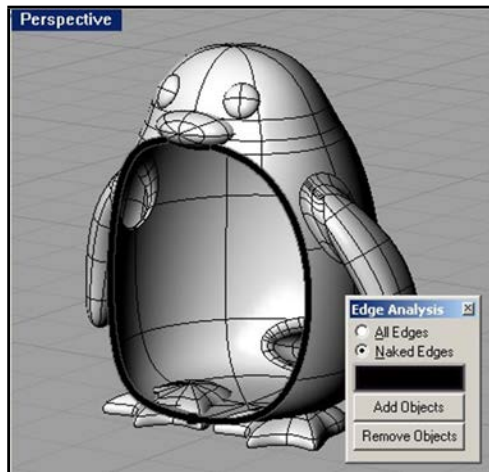
Checking for Naked Edges

In order to output a correct .STL file for prototyping, NURBS objects in Rhino must be closed solids/polysurfaces (a.k.a 'watertight' objects). This means your object can have no naked or unjoined edges. Examples of 'watertight' objects:
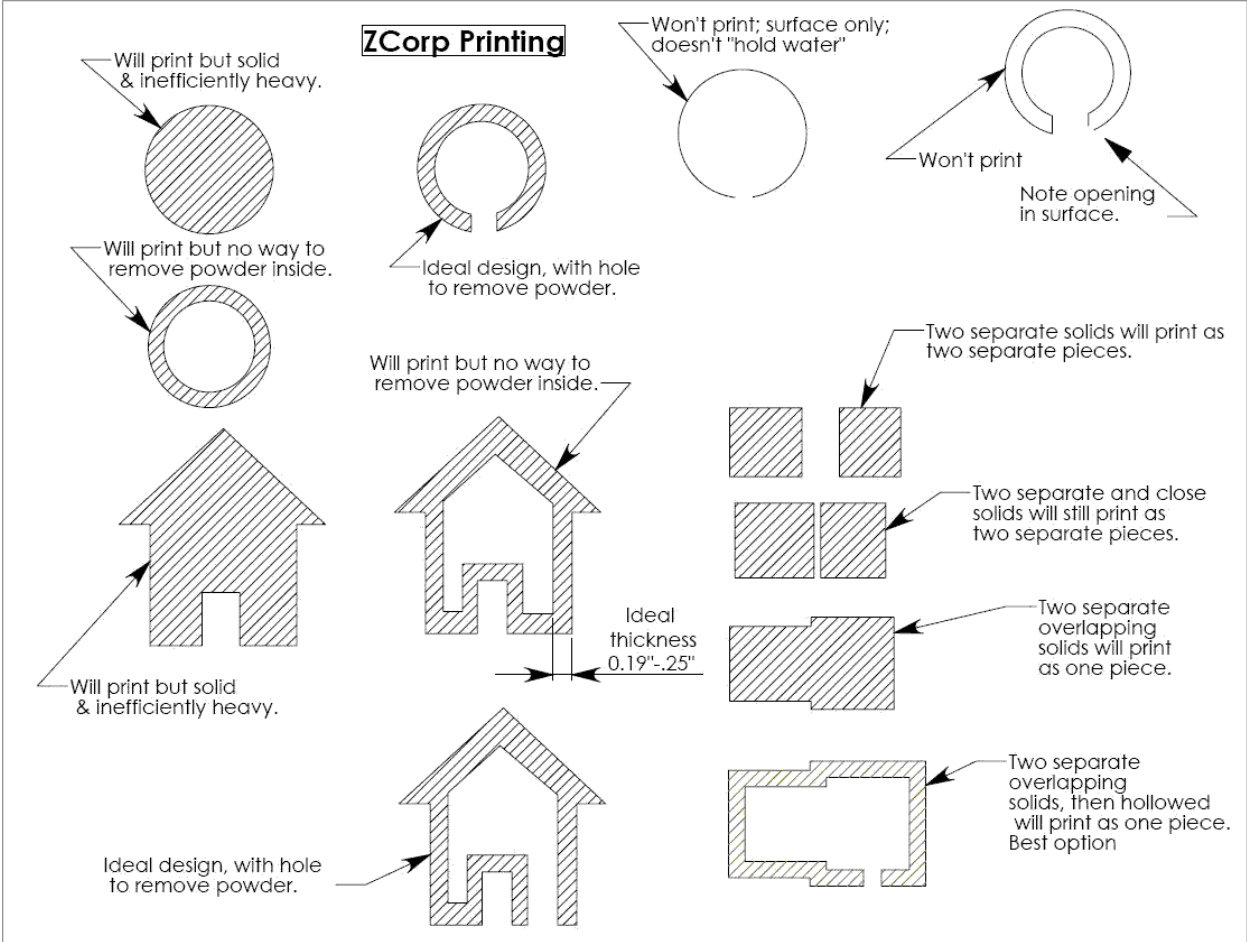
Here is one way to check for naked/unjoined edges during the modeling process:

• Type the command ShowEdges into the command line, then hit Enter.

• You will be prompted to select surfaces, polysurfaces or meshes for edge display. Select the object(s) you want to check, and then hit Enter.

• This will bring up the Edge Analysis dialog. Select the radio button next to Naked Edges. All naked edges will be displayed highlighted in the color shown.

Edge Analysis

Here is another way to check for naked/unjoined edges during the modeling process (Fig. 2):

• Type the command What into the command line, then hit Enter.

• You will be prompted to Select objects. Select the object(s) you want to check, and then hit Enter.

• This will bring up the What dialog. Look under the heading 'Geometry'; the object is described as open or closed (closed & valid = good; open &/or invalid = bad).
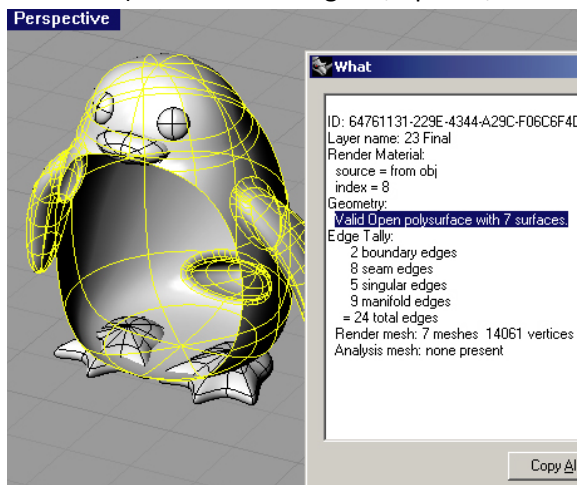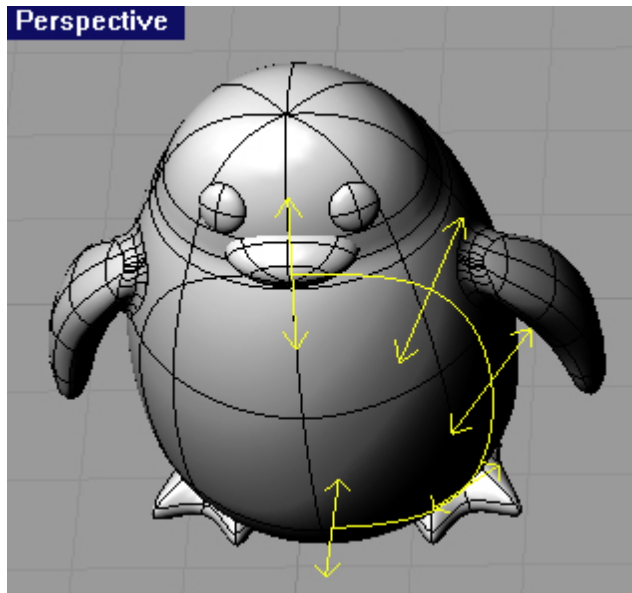

What Command

Repairing Naked Edges

To join two naked edges once they have been located:

• Type the command JoinEdge into the command line, then hit Enter.

• You will be prompted to Select two unjoined edges. Select the edges you want to join, then hit Enter.
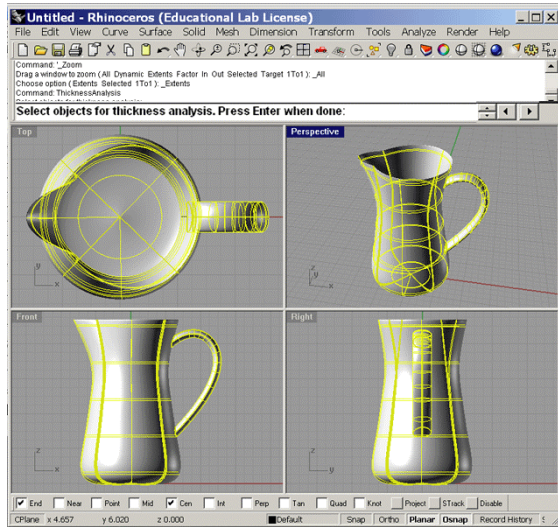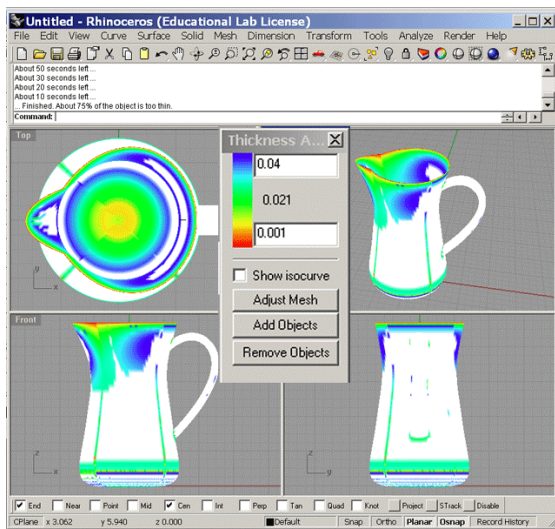

Thickness Analysis

Solid models can still fail to print correctly if they have parts that are too thin (less than 0.1"/2.5 mm). Rhino's built-in thickness analysis tool can give you a visual representation of parts that may be too thin.
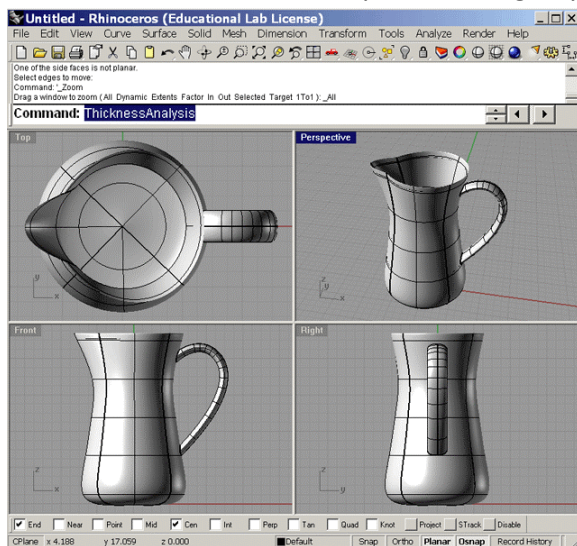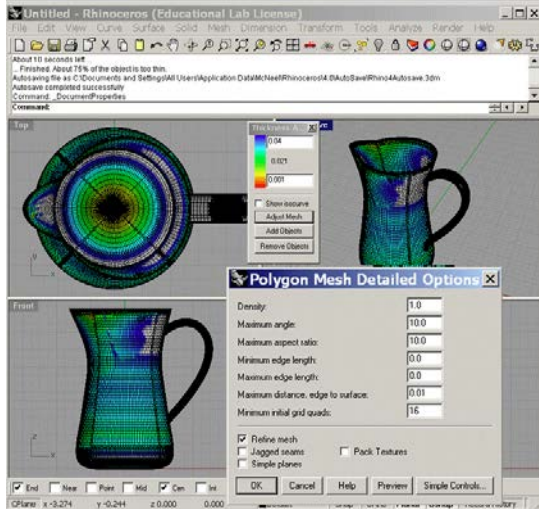

1) Enter ThicknessAnalysis in the command line.

2) Select the object you want to analyze.

3) Upper range: 0.1" lower range: 0.001"   Any part highlighted with color is too thin to print. Thickness analysis can take several minutes.

 4) It is also recommended to use the AdjustMesh option to duplicate the mesh settings you plan to use on export.

.STL Export

Once your model has been made into a valid closed surface with no thin parts, you can export it to the .STL file format.

• Select the object you want to export. Models that will end up being physically separate pieces must be exported as individual STL files.

• Click on 'File -> Export Selected...'

• Name your file and set the 'Save as Type:' option to 'Stereolithography [*.STL],' then click Save.

• In the 'Polygon Mesh Detailed Options' dialog, enter the settings for polygon conversion.

• You can preview the mesh settings, or export directly to a .STL in binary format.
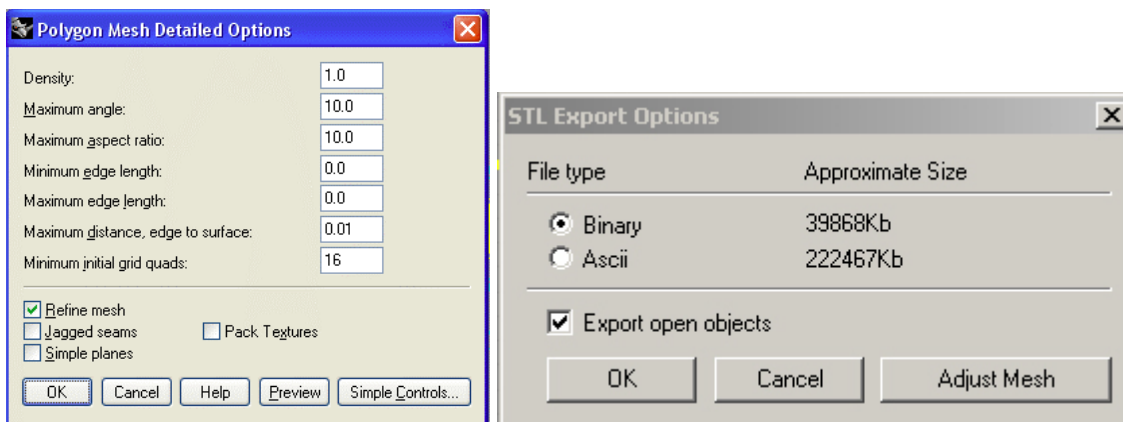
NOTE: These settings are recommendations only, and may not be ideal for all parts. These settings may produce extremely large .STL files. Files over 50MB will not be accepted.

Checking Meshes

Sometimes errors in mesh export occur, even if the original model is a valid, closed polysurface. Bad meshes can lead to output problems including missing parts and the inability of the printer to process the file. Here's how to check for common problems in your exported mesh:

• Start a new file in Rhino. Make sure to use the same units as your original file.

• Click on 'File -> Import…' and select the .STL you just saved out. The mesh will import.

• Highlight the mesh by clicking on it, then type 'CheckMesh' into the command line. A message box will appear listing any problems with the mesh.

• Open the Rhino help file, and search for 'CheckMesh.' Refer to the help file page for specific commands to correct any problems in your mesh.

• Meshes are not always able to be fixed using CheckMesh. Sometimes it is best to go back to the original NURBS model to fix meshing problems.

• Once your mesh is fixed, save as a .STL.

NOTE: CheckMesh may report a good mesh with minor problems listed under **Important things…**. Some of these minor problems, such as reversed normals, can prevent the 3D printer from properly processing a file.

For more information on Rhino mesh settings please refer to these web pages:

http://wiki.mcneel.com/rhino/meshfaq

http://wiki.mcneel.com/rhino/meshsettings