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 colleague, or work through some of the excellent tutorials that are installed with Rhino. Remember modeling is a skill; it can take months and even years of practice to become proficient.

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.
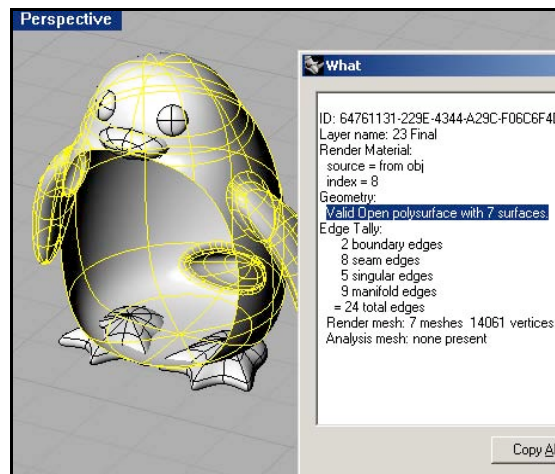
Checking for Naked Edges

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.

This is one way to check for naked/unjoined edges during the modeling process (Fig. 1):

- 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, 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.
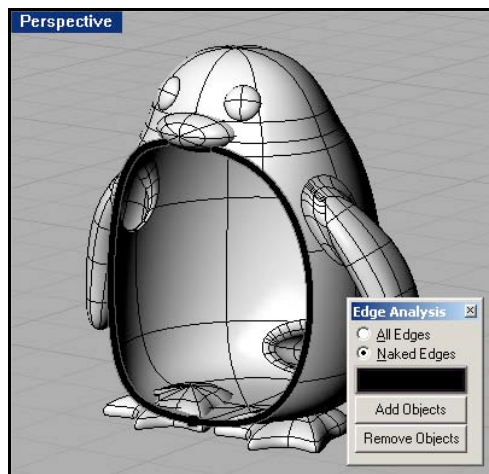
(Fig. 1) Edge Analysis   (Fig. 2) What Command

This 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, 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).

NOTE: We strongly recommend a periodic check of your model for naked edges as you work. Don't wait until you think your model is finished; check often and fix naked edges as you find them.

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.
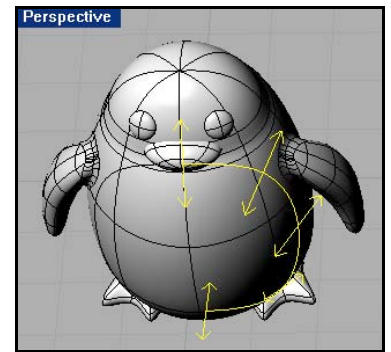
Thickness Analysis

Solid models can still fail to print correctly if they have parts that are too thin (less than 0.04"/1mm). Rhino's built-in thickness analysis tool can give you a visual representation of parts that may be too thin.
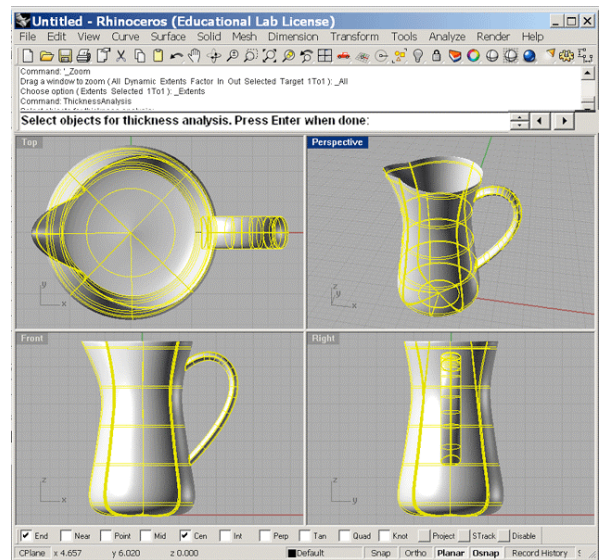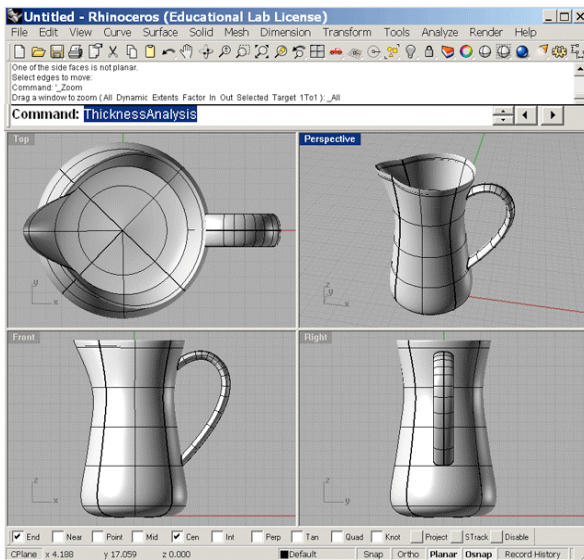
1) Enter **ThicknessAnalysis** in the command line.

2) Select the object you want to analyze.
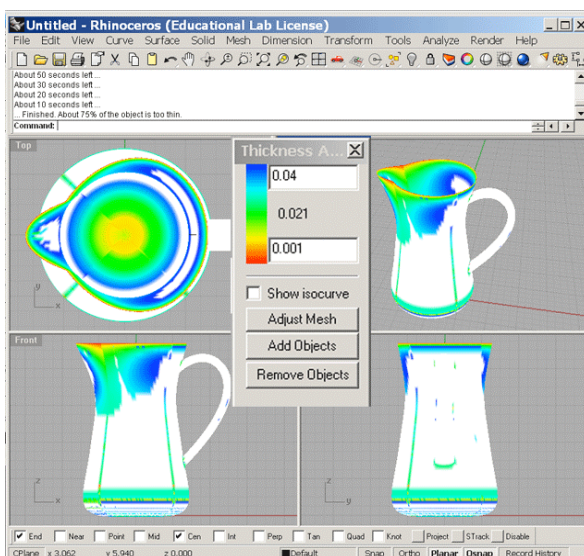
3) **upper range:** 0.04" **lower range**: 0.001"
Any part highlighted with color is too thin to print. Thickness analysis can take several minutes.

4) It is also recommended to use the **Adjust Mesh** 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.
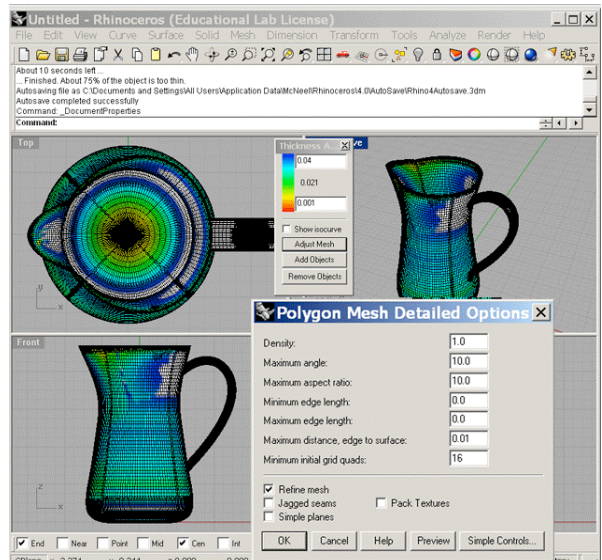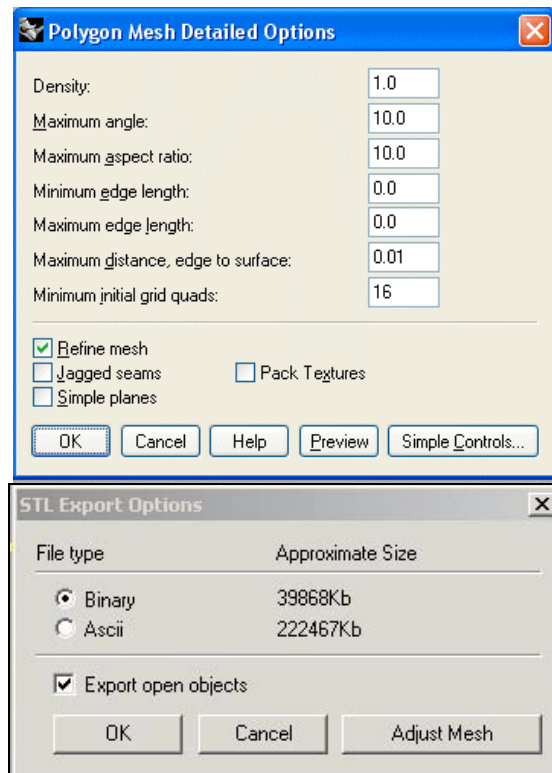
- Select the object you want to export. Models that will end up being physically separate pieces must be exported as individual STL files.

- 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 an .STL in **binary format**.

Polygon Mesh Detailed Options

Density:	1.0
Maximum angle:	10.0
Maximum aspect ratio:	10.0
Minimum edge length:	0.0
Maximum edge length:	0.0
Maximum distance, edge to surface:	0.01
Minimum initial grid quads:	16

☑ Refine mesh
☐ Jagged seams   ☐ Pack Textures
☐ Simple planes

OK   Cancel   Help   Preview   Simple Controls...

STL Export Options

File type   Approximate Size
◉ Binary   39868Kb
○ Ascii   222467Kb

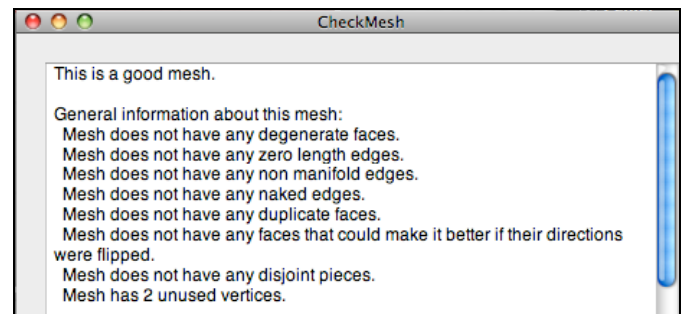☑ Export open objects

OK   Cancel   Adjust Mesh

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.
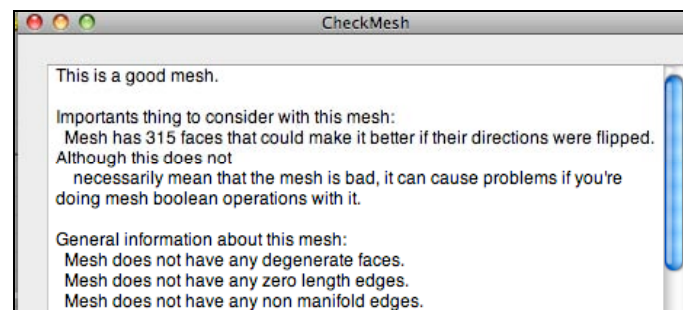
Checking Meshes

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 an .STL and proceed to the Pre-Processing instructions.

CheckMesh

This is a good mesh.

General information about this mesh:
  Mesh does not have any degenerate faces.
  Mesh does not have any zero length edges.
  Mesh does not have any non manifold edges.
  Mesh does not have any naked edges.
  Mesh does not have any duplicate faces.
  Mesh does not have any faces that could make it better if their directions were flipped.
  Mesh does not have any disjoint pieces.
  Mesh has 2 unused vertices.

A perfect mesh.

CheckMesh

This is a good mesh.

Importants thing to consider with this mesh:
  Mesh has 315 faces that could make it better if their directions were flipped. Although this does not
    necessarily mean that the mesh is bad, it can cause problems if you're doing mesh boolean operations with it.

General information about this mesh:
  Mesh does not have any degenerate faces.
  Mesh does not have any zero length edges.
  Mesh does not have any non manifold edges.

A good mesh, with problems.

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.