Move Model to Origin:

1>   Select Move tool.

2>   When prompted select your model and hit return or RMB.

3>   When prompted select any point on your model and hit return or RMB.

4>   When prompted type 0 in the command line and hit return or RMB.

5>   Your model will now be on the origin.

          RMB = Right Mouse Button
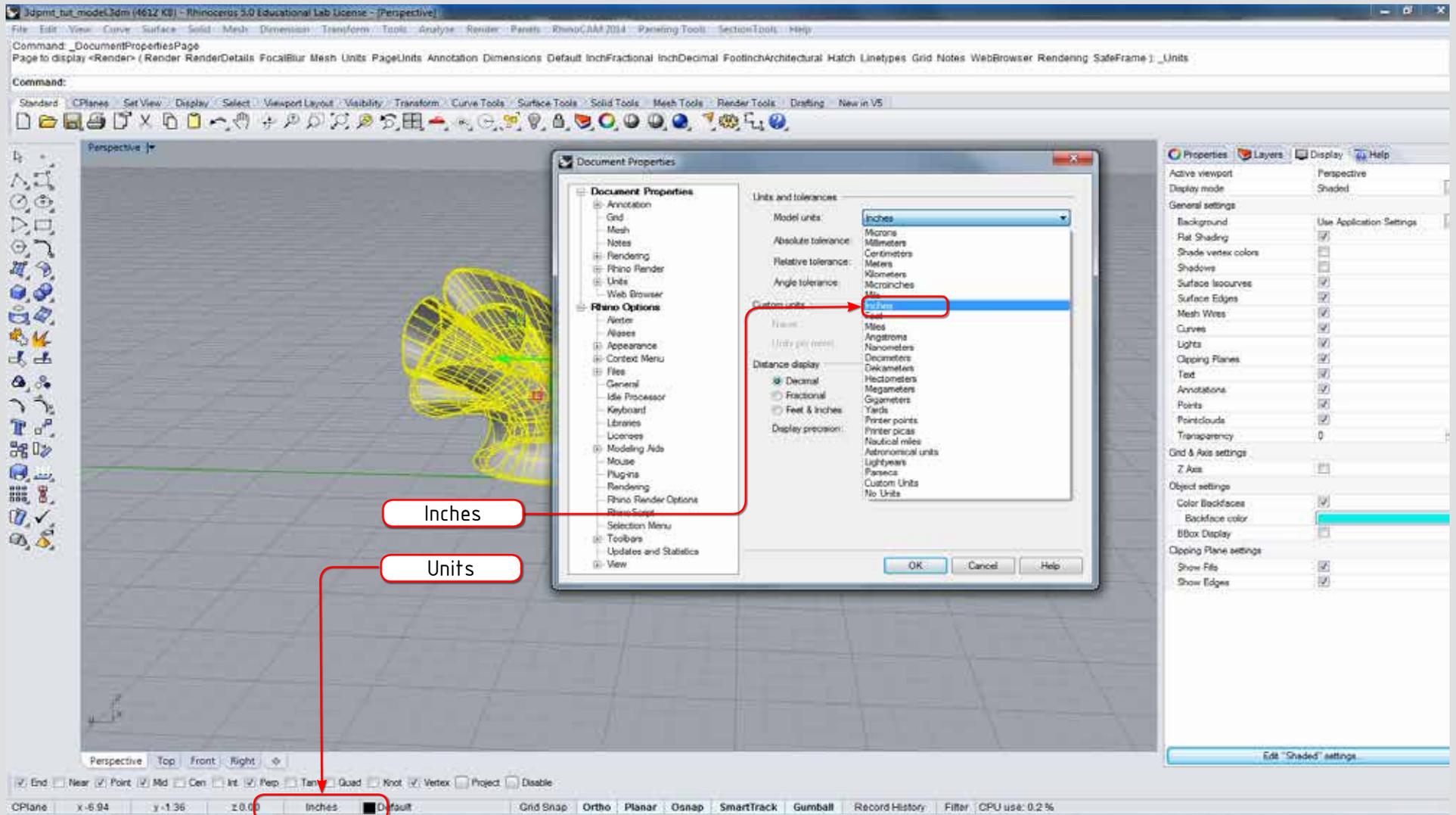
Check Units:

1>   Make sure Units Display in bottom info bar reads inches.

2>   If model is not in inches, type Units in command line.

3>   From drop-down in units window, select inches.

4>   In pop up window click Yes to scale model, unless you thought you were modeling in inches all along only to find out you were not. In this case chose No and your model will remain the same size; just the units will change.
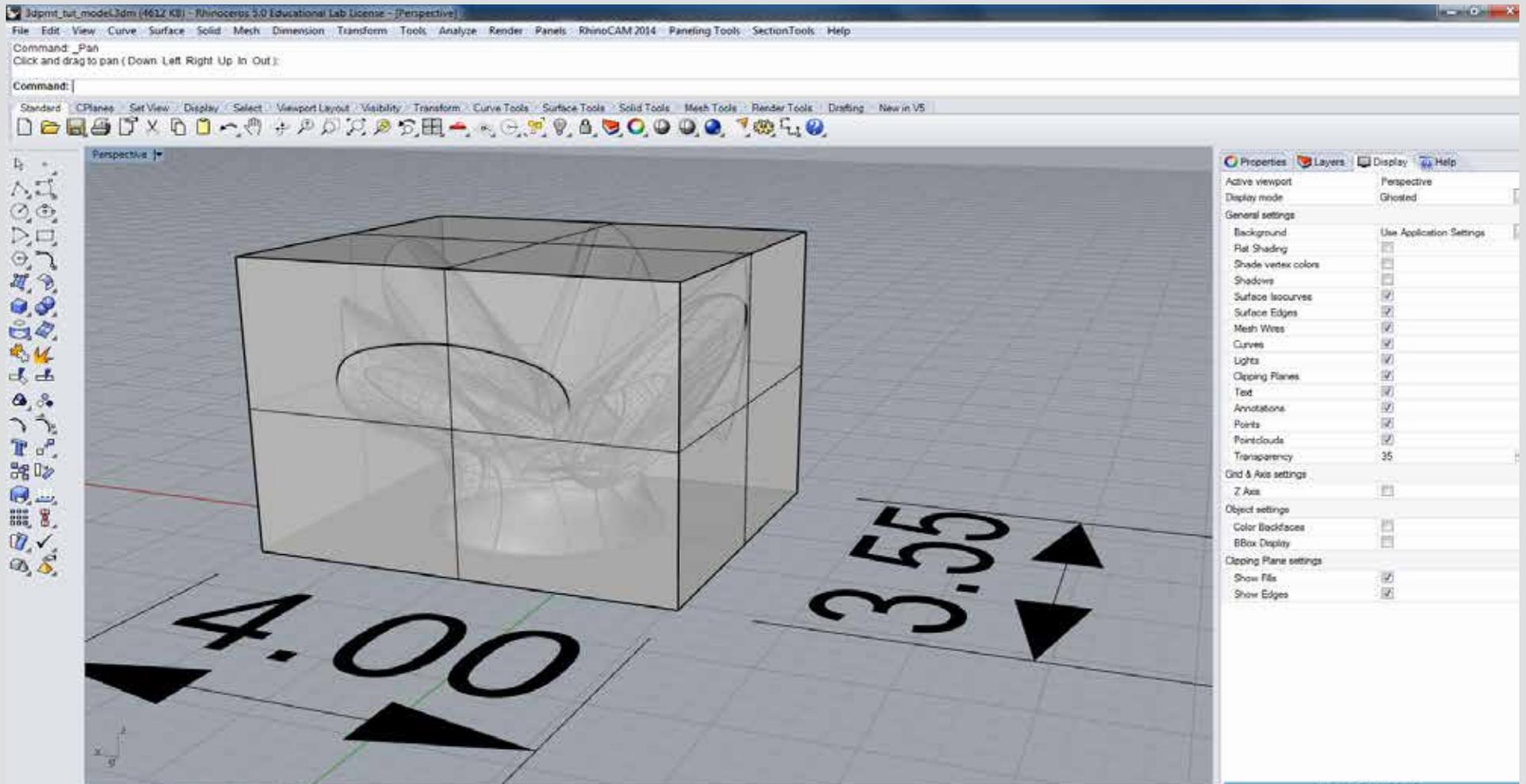
Check size of model:

1> Type BoundingBox into command line.

2> At prompt select entire model and hit return or RMB and Rhino will draw a box that encloses your model.

3> Using Distance command or Dimension Lines measure length (X), width (Y) and height (Z).

4> Make sure all dimensions of model fit within the build envelope of the printer.

    a.   MakerBot (PLA plastic) 10.5"x5.5"x6"

    b.   Type A (PLA plastic) 11"x11"x11"

    c.   Zprinter (powder) 9.75"x6.5"x7.5"

    d.   FormLabs (resin) 5.7"x5.7"x6.9"

5> If model is too big, scale to fit.
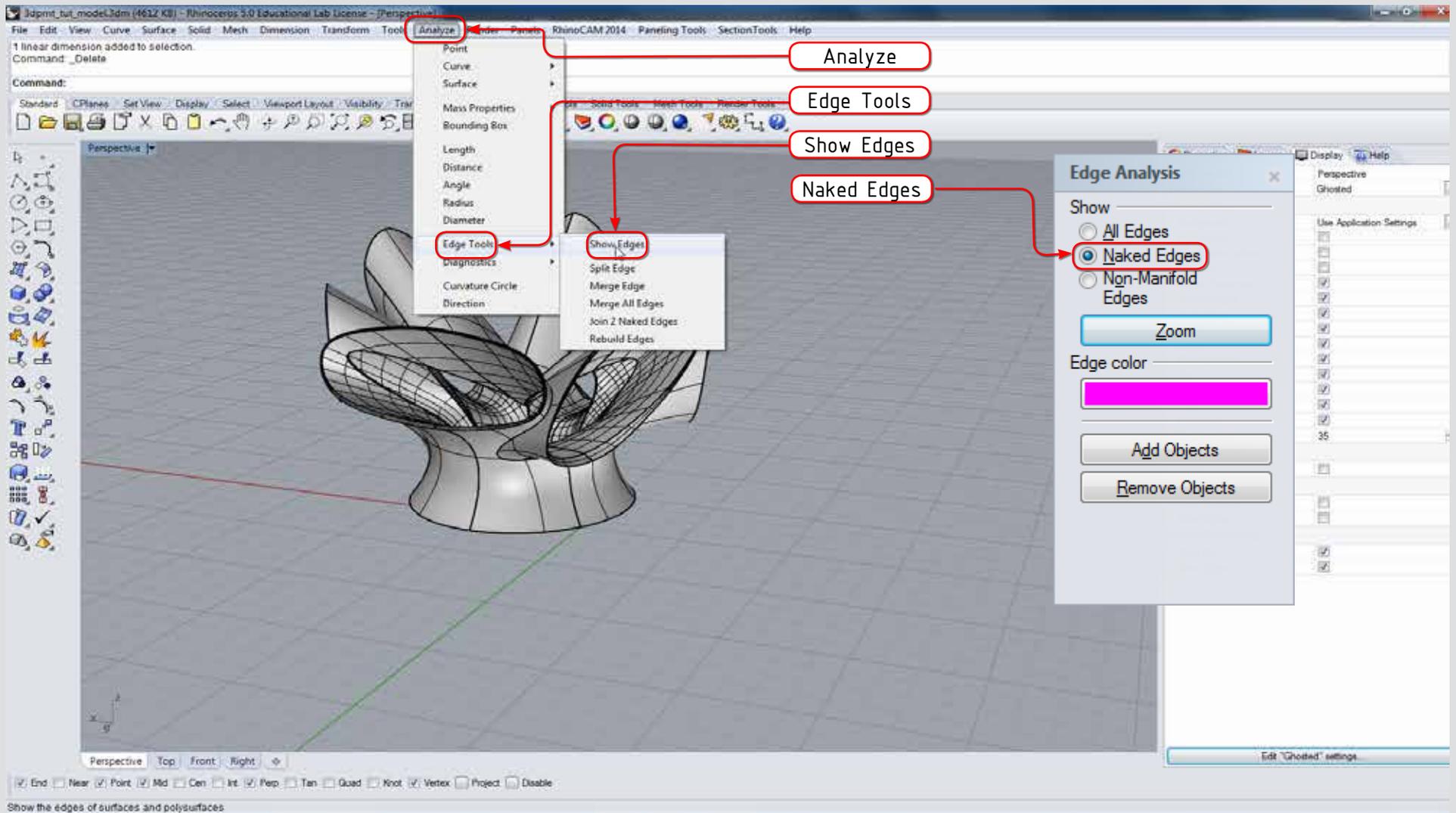
Check for naked edges:

1>   Select Show Edges tool or type ShowEdges into command line.

2>   When prompted select your model and hit enter or RMB.

3>   In pop-up window select Naked Edges; any surface edge that is not joined to another will be marked. Naked Edges means your model is not a solid closed polysurface and will not print. All models to be 3D printed must be solid to print successfully.
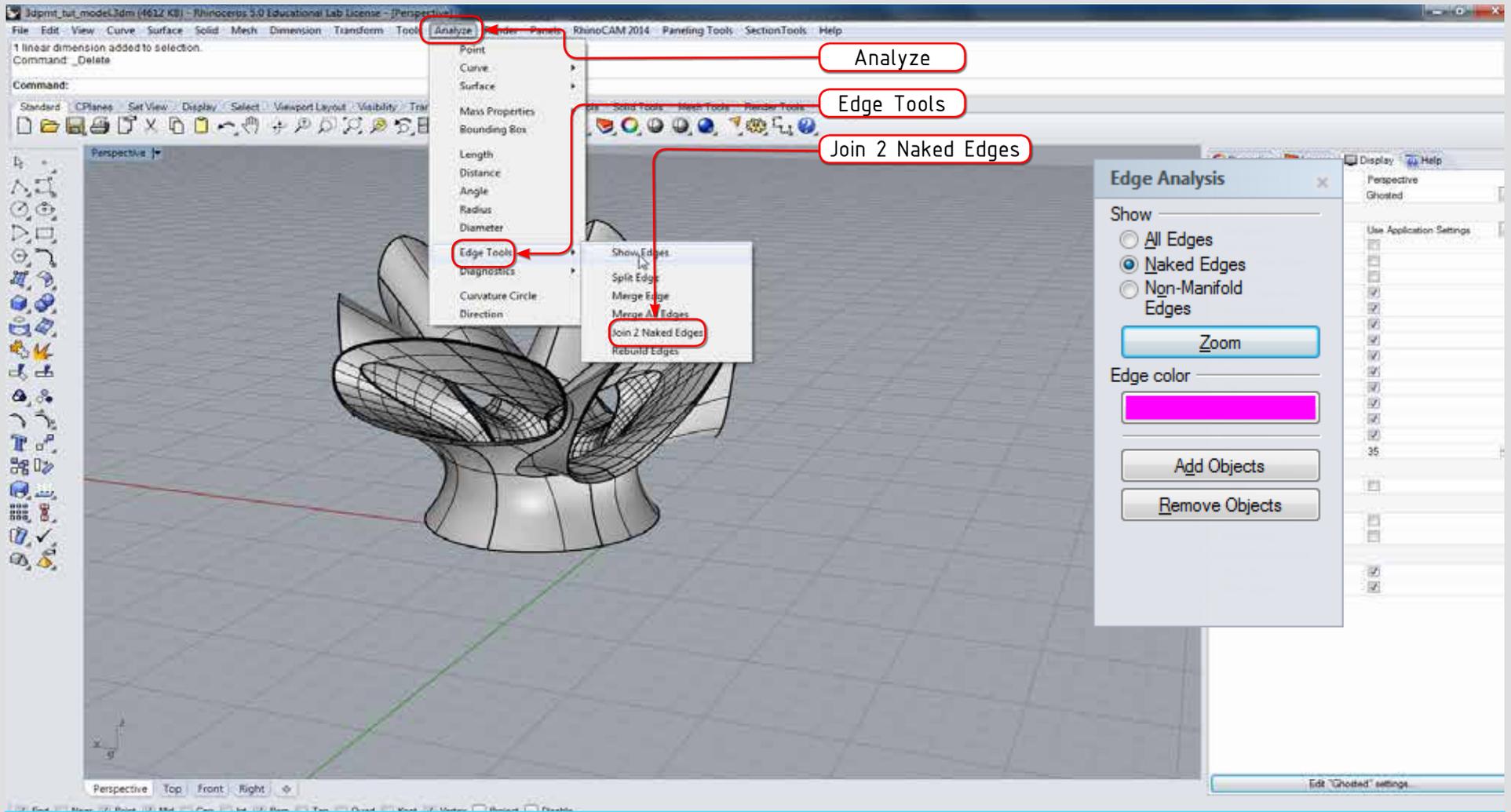
Fix naked edges:

1>   Select your model and hit the Join button or type Join in command line. Any edges that are still naked need to be joined using Join 2 Naked Edges.

2>   Select Join 2 Naked Edges.

3>   When prompted select one edge in a pair of naked edges.

4>   When prompted select the other edge in a pair of naked edges.

5>   Repeat until all naked edges are fixed. If you have edges that will not join, you can try to fix them after converting to STL in a later step. Otherwise your best option is to go back into the model and look at how surfaces that will not join are built and try rebuilding them.
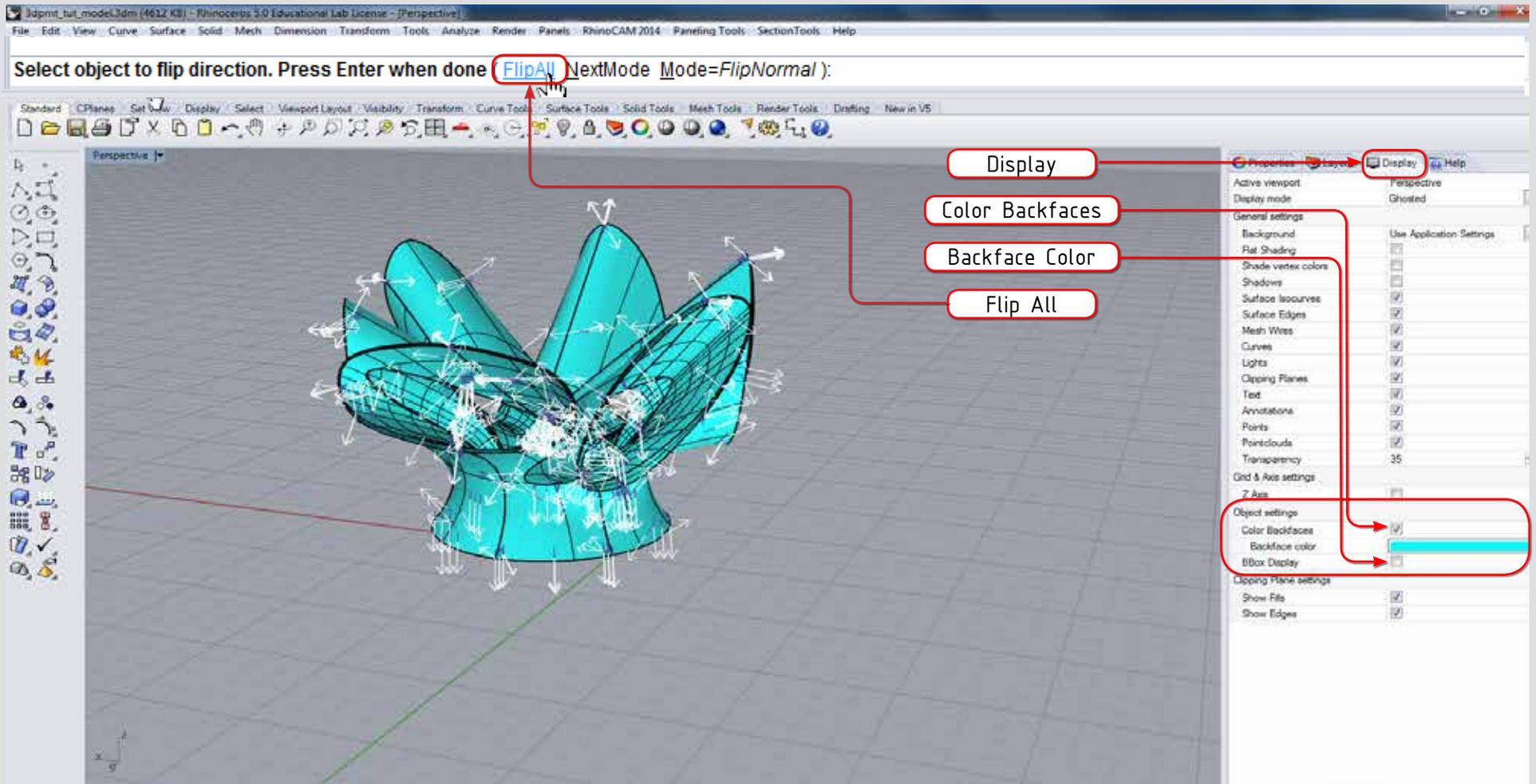
Check Normals:

1>   Click on the Display tab

2>   Under the Object settings click the Color Backfaces check box

3>   Under the Backfaces color choose the most obnoxious color you can stand so it will be easy to see. Any part of your model that has changed color is inside out [the Normals of surface point to the inside of the model]. To print successfully your model must be all right side out [Normals point to outside of model].

4>   To change direction of any surface that has normals pointing to the inside of the model [inside out] select them.

5>   Type Dir into the command line then return or RMB. All selected surfaces will now have white arrows showing the direction the normal is pointing.

6>   In the command line click Flip or FlipAll depending on what is showing and then enter or RMB. Your model should no longer be the color you chose for the Backface color.
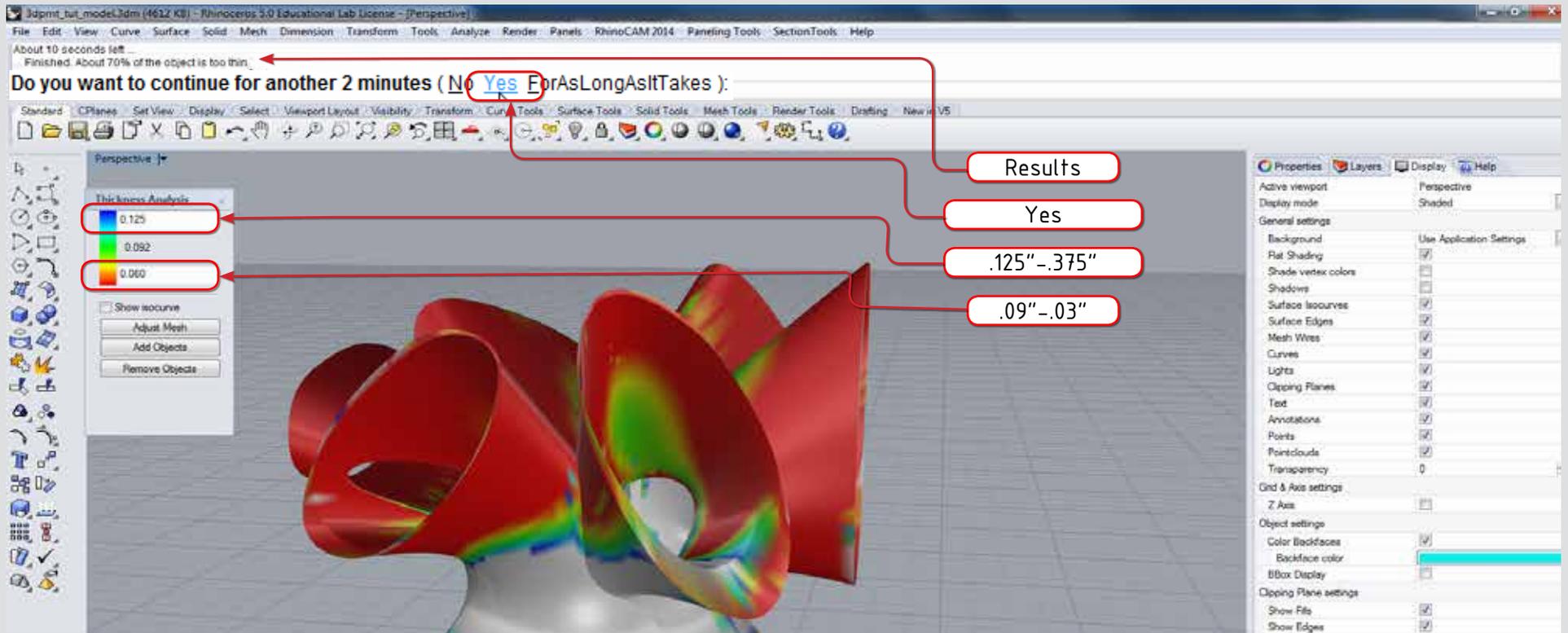
Check Thickness:

1>   Type ThicknessAnalysis into command line.

2>   When prompted select model and hit enter or RMB.

3>   If prompted "do you want to continue for another 2 minutes" click yes.

4>   When pop-up window opens enter numbers as shown in table below.

Printer	Max Thickness [BLUE]		Min Thickness [RED]	
MakerBot /Type A [PLA]	.125 in	3 mm	.06 in	1.5 mm
Zprinter [Powder]	.1875 in	5 mm	.09 in	2 mm
FormLabs [Resin]	.09 in	2 mm	.03 in	.75 mm

5>   If prompted "do you want to continue for another 2 minutes" click yes.

6>   Your model will now be colored in a gradient from Blue to Red.

7>   Any part of your model that is in the Blue to Green range or the original model color is OK and ready to print [to print successfully your whole model should be in this range].

8>   Any part of your model that is in the Yellow is borderline too thin to print.

9>   Any part of your model that is Red is too thin to print.

10>   Unfortunately the only way to fix this is to either scale up the model or go back and re-model the thin parts thicker.
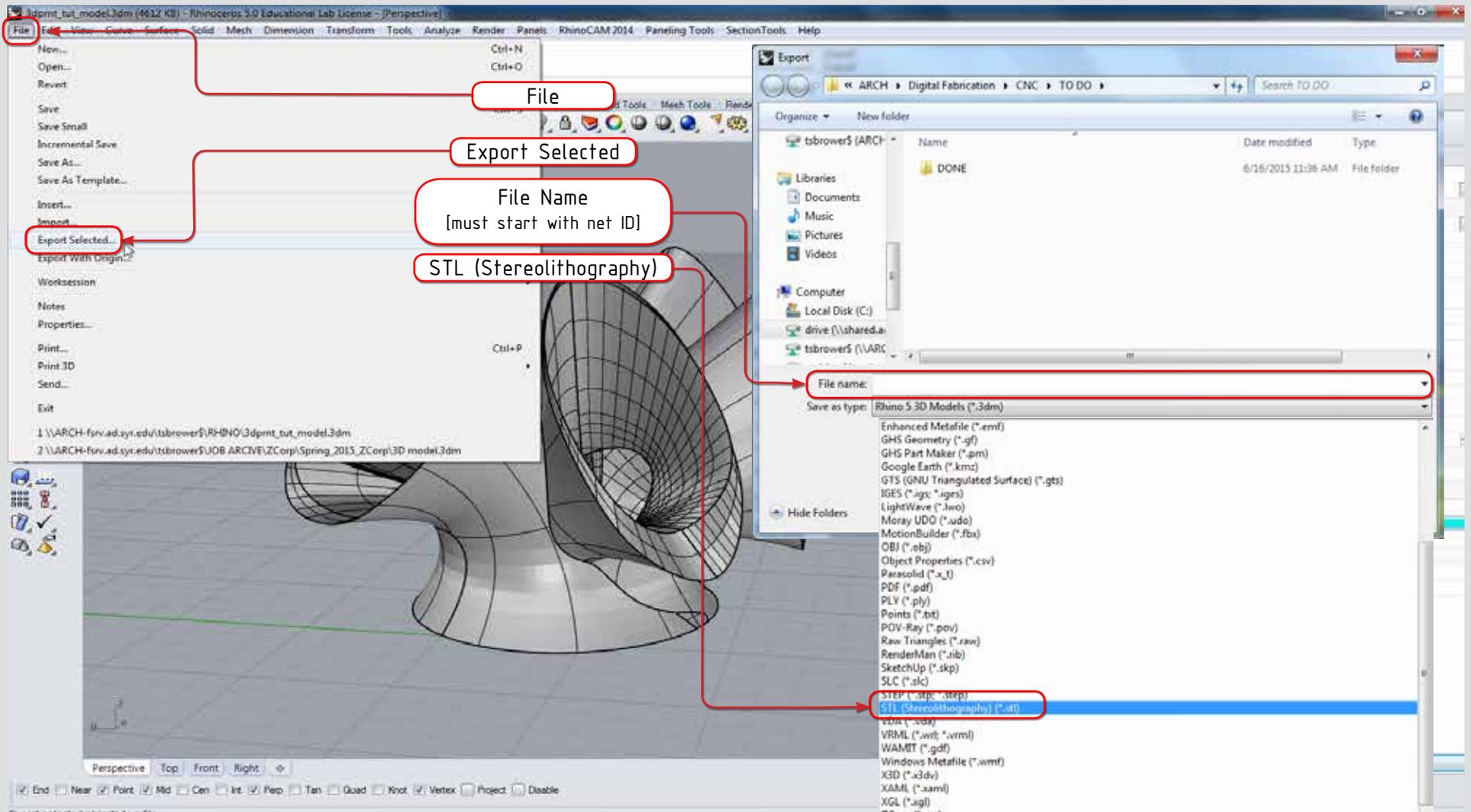
Export STL:

1>   Under file menu select Export Selected.

2>   When prompted select your model and hit return or RMB.

3>   Choose a name for your file. Remember your file name must start with your Net ID [everything before the @ in your SU email address]. It is recommended to end your file name with numbers [001] that you can increment with different versions of the file. This way if you have to submit multiple versions of the file it is easy to tell them apart.

4>   Select STL from the Save As Type drop-down of available file types.

5>   Hit Save button.

Detailed Control

STL Mesh Export Options

Tolerance
The maximum distance between the original surface or solid and the polygon mesh created for the STL file.

0.004 inches

OK | Cancel | Preview | Help | Detailed Controls...

Maximum angle:
[START AT 15 CAN BE 7-35]

Minimum edge length:
[START AT .0001" or .0025mm]

Maximum distance, edge to surface:
[.004" or .1mm]

Minimum initial grid quads:
[.16 OR 0]

Polygon Mesh Detailed Options

Density: 0.0
Maximum angle: 15.0
Maximum aspect ratio: 0.0
Minimum edge length: 0.0001
Maximum edge length: 0.0
Maximum distance, edge to surface: 0.004
Minimum initial grid quads: 16

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

OK | Cancel | Help | Preview | Simple Controls...

Export continued:

6>   In STL Mesh Export Options pop-up window, hit Detailed Controls button.

7>   In the Polygon Mesh Detailed Options pop-up window is where you control the quality of your export mesh. There are four parameters that you will set here. Any parameter that has a value of 0 will be ignored by Rhino while meshing.

    a.   The most important parameter is Maximum distance, edge to surface [which is the desired maximum distance between the midpoint of any edge of a polygon and the true surface]. This should always be set to about the resolution of the printer being using. This should be set at .004" or .1mm.

    b.   Maximum angle: The smaller the number the more dense the mesh.   A good place to start is 15. If you are having problems with large file sizes try something like 35 or turn off the parameter all together by entering a 0. Not enough detail try something like 7 but watch out for large file sizes.

    c.   Minimum edge length: This parameter helps to keep file size down by stopping subdivision when edge length reaches input threshold. A good place to start is .0001" or .0025mm. The larger the number the more coarse the mesh.

    d.   Minimum initial grid quads: This parameter sets the starting point for subdivision. 16 is good starting point but also can be turned off with 0 without much effect.

8>   Make sure Refine mesh is checked. [Note: all these settings should be remembered by Rhino so you should only have to do this once. It is however worth checking to make sure the first few times you export a file.]

9>   Hit OK button.
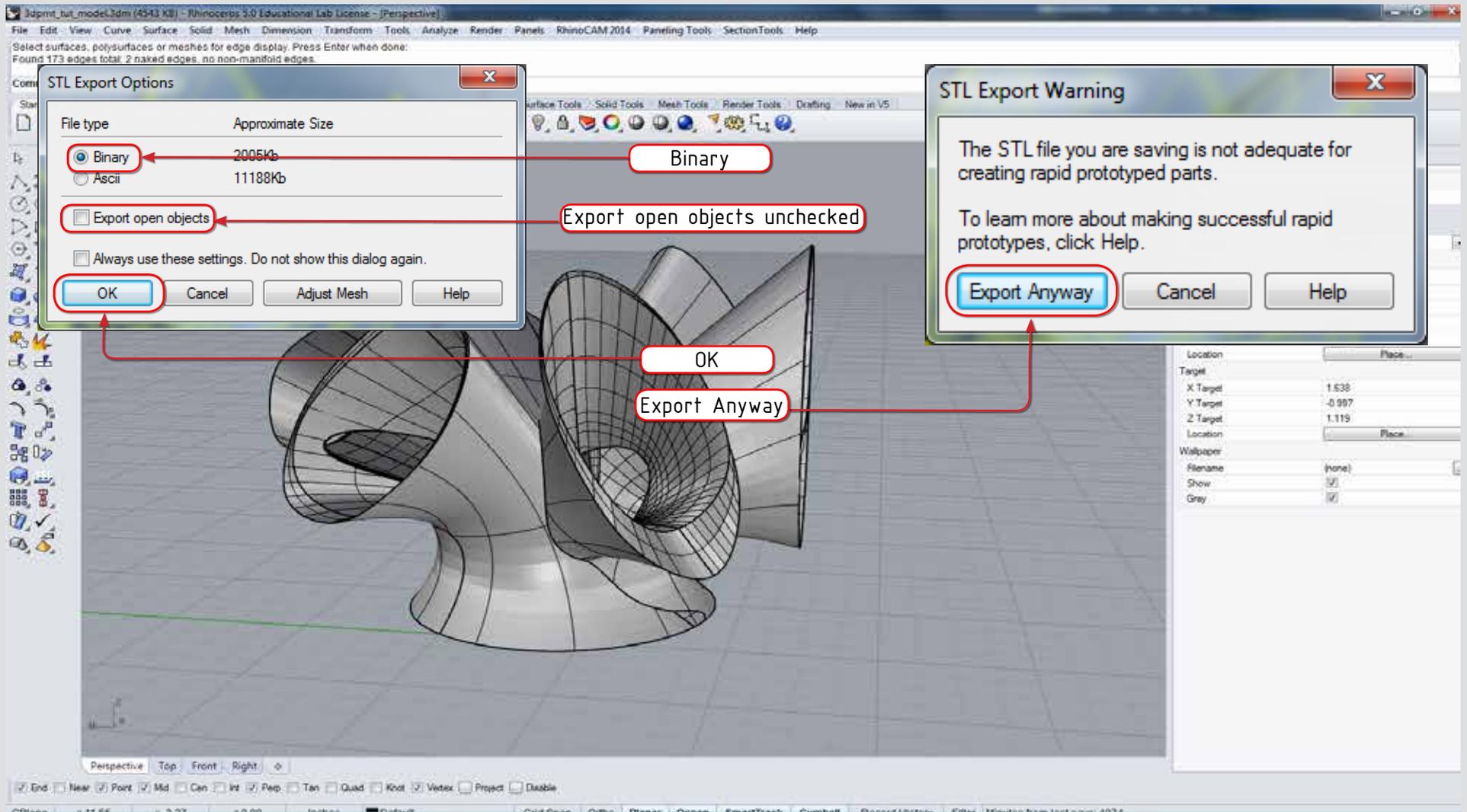
Export continued:

10>   In pop-up window select Binary.

11>   Uncheck Export open objects if checked.

12>   Hit OK button, Return or, RMB.

13>   If STL Export Warning pop-up window opens it means your mesh is not closed [You have naked or non-manifold edges]. You have two choices at this point: either abort the export and try to fix model in Rhino, or continue with export and try to fix the model in mesh repair software [Magics, MeshMixer, etc…].