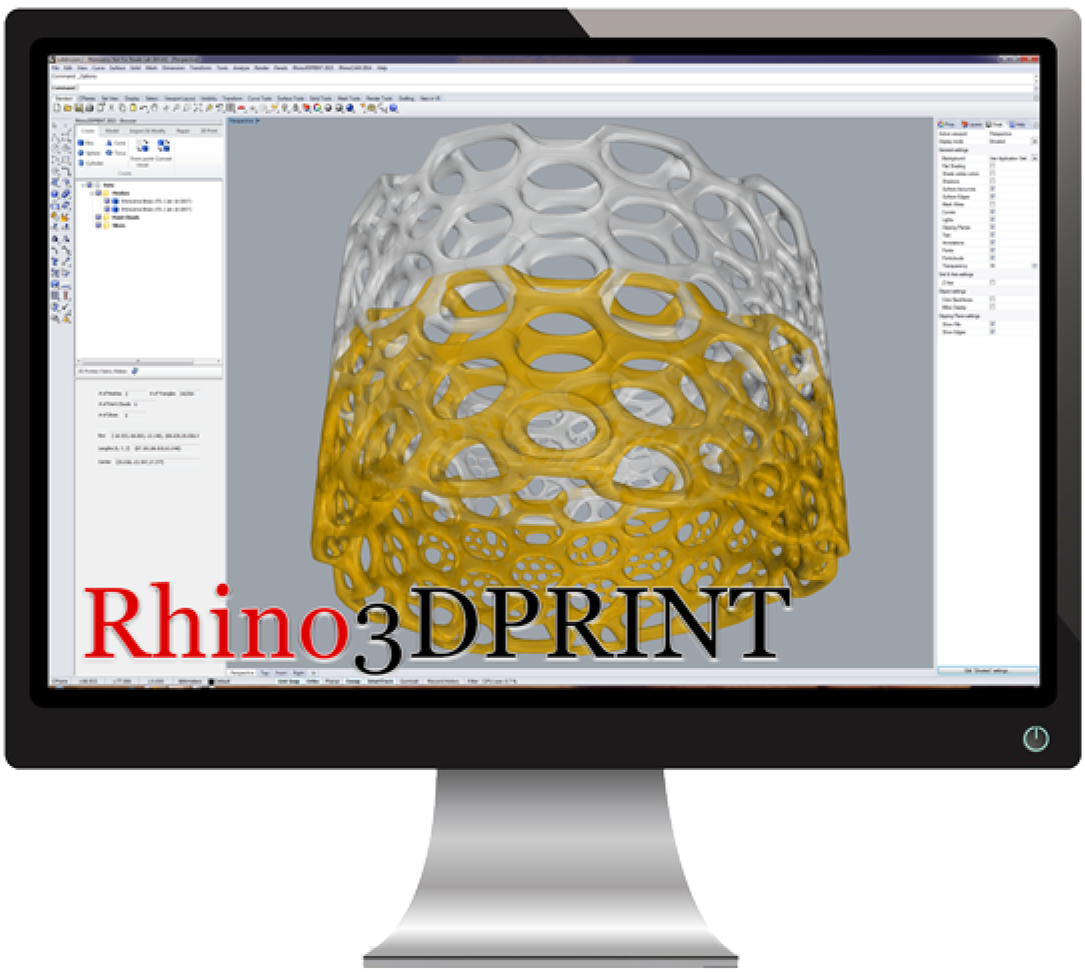
Rhino3DPRINT® 2017 Quick Start Guide



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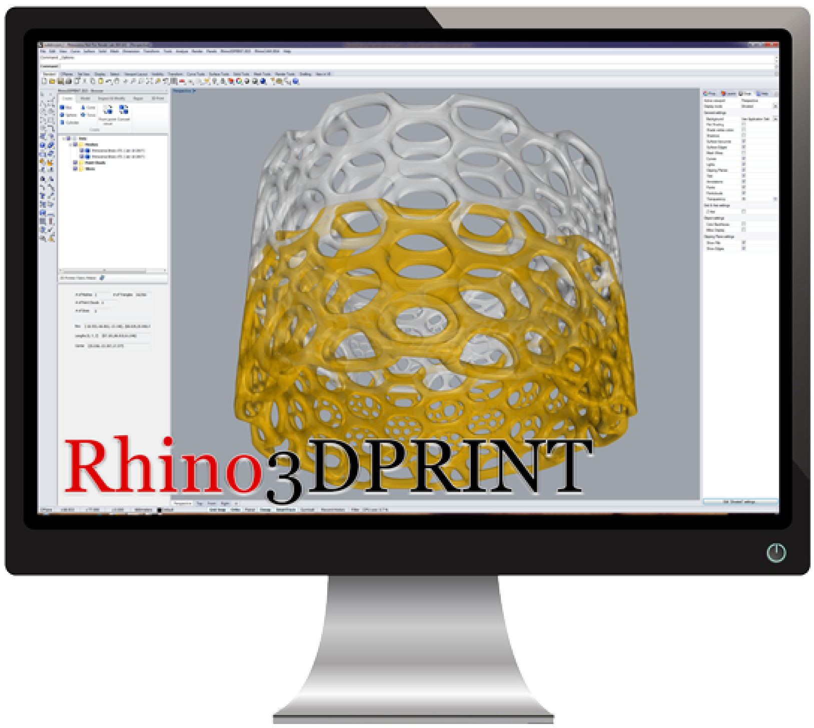
Chapter 1: About

1.1 About Rhino3DPRINT 2017

Rhino3DPRINT 2017 is a product from MecSoft Corporation that runs as a plug-in inside of the Rhino modeling system. This product offers efficient, easy and automatic tools for the preparation of 3D data, such as point clouds and meshes, that otherwise cannot be directly 3D printed. The combination of Rhino's modeling tools, extensive file import functionality and Rhino3DPRINT 2017 data preparation tools enables you to 3D print models from almost any input source and any major 3D format.



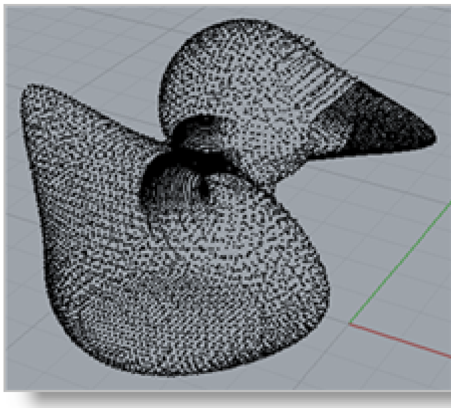
1.2 Why Rhino3DPRINT 2017

Rhino3DPRINT 2017 exists as a bridge between users' 3D data and the 3D printer. 3D data comes in many formats, such as CAD data, scanned data etc., that cannot simply be sent to the 3D printer. This 3D data needs to be converted to a format that the 3D printer not only understands but also can handle. Rhino3DPRINT 2017 serves to convert such 3D data and make it amenable for 3D printing.

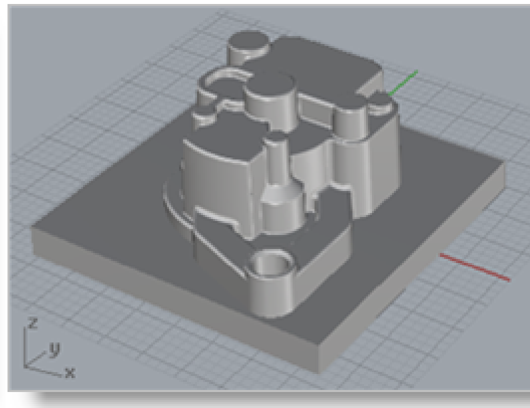
1.3 Using this Guide

This guide will show you how to use Rhino3DPRINT 2017 to prepare various forms of 3D data and geometry for 3D printing. You will learn how to analyze, repair, modify and setup 3D mesh models for a variety 3D printers. You will learn how to use point cloud data to create and refine a mesh model. You will learn how to offset, split and cap a mesh to create a uniform wall thickness as well as Boolean mesh operations to create a cavity block.

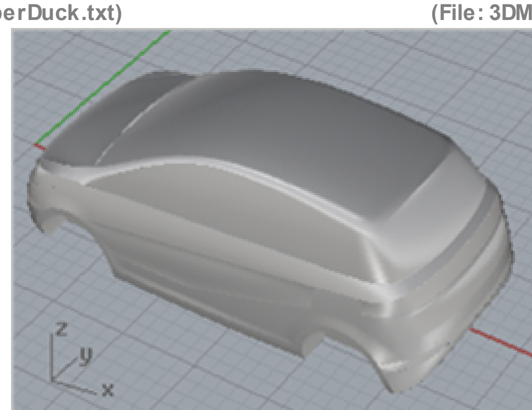
This guide has three associated Rhino files that you can find located in the QuickStart folder under the installation folder of RhinoCAM.



Chapter 1: Point Clouds
(File: RubberDuck.txt)



Chapter 2: Offset and Split
(File: 3DMold.3dm)



Chapter 3: Cavity Block
(File: CarModel.stl)

1.4 Useful Tips

Here are some useful tips that will help you use this guide effectively.

1. Copy the tutorial part files in a location other than the installation folder to make sure you have read/write privileges to the files.
2. Once you start working with the tutorial file, save your work periodically!
3. Don't stress out too much if you are having trouble with the tutorial. Call us or send us an email and we can help you out.
4. Most of all have fun!

Chapter 2: Getting Ready

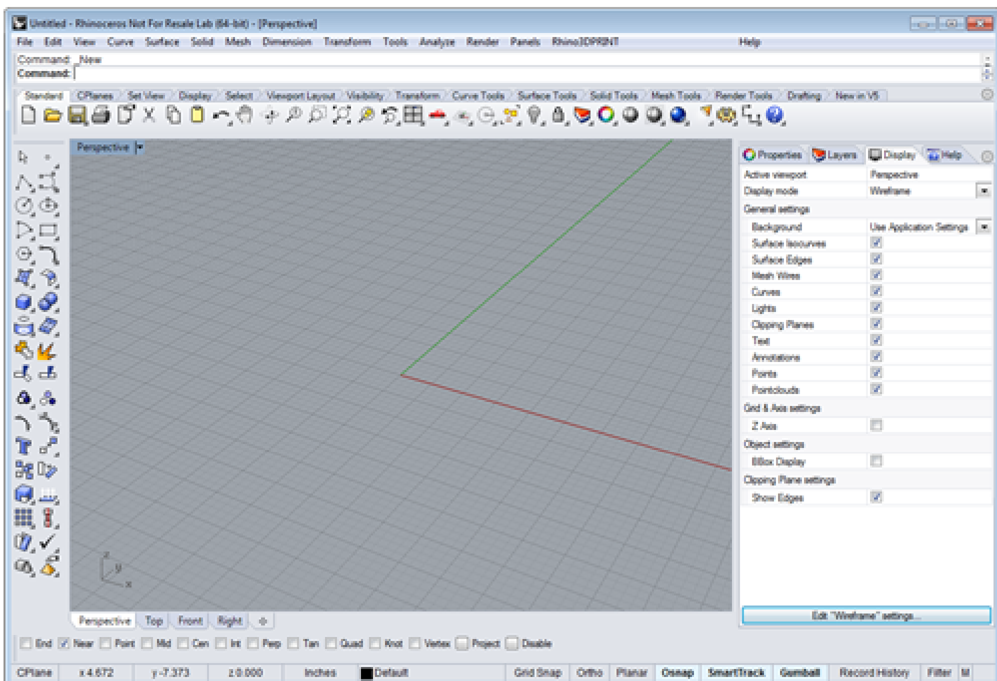
2.1 Running Rhino3DPRINT 2017

Locate the Rhino 5 shortcut on your desktop and double click to launch the application. Alternatively you can also click on the Windows Start button and select All Programs. Go to the program group containing Rhino 5. (The name of this program group will usually be called Rhinoceros 5, unless you specified otherwise during setup.) Once you locate the program group, select it and then select Rhinoceros 5 to launch the application.

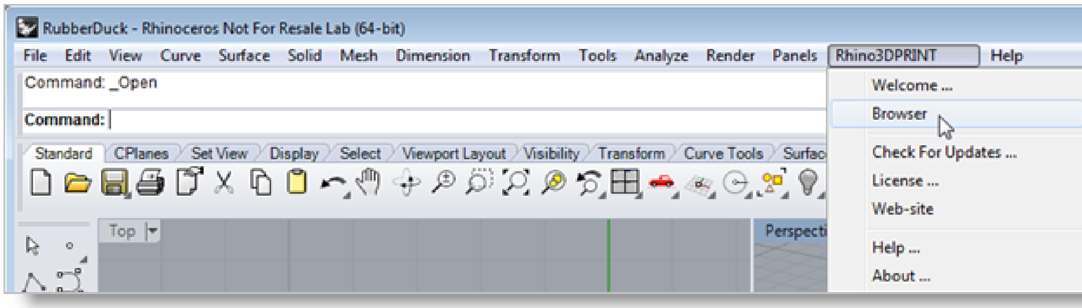
2.2 About the Display

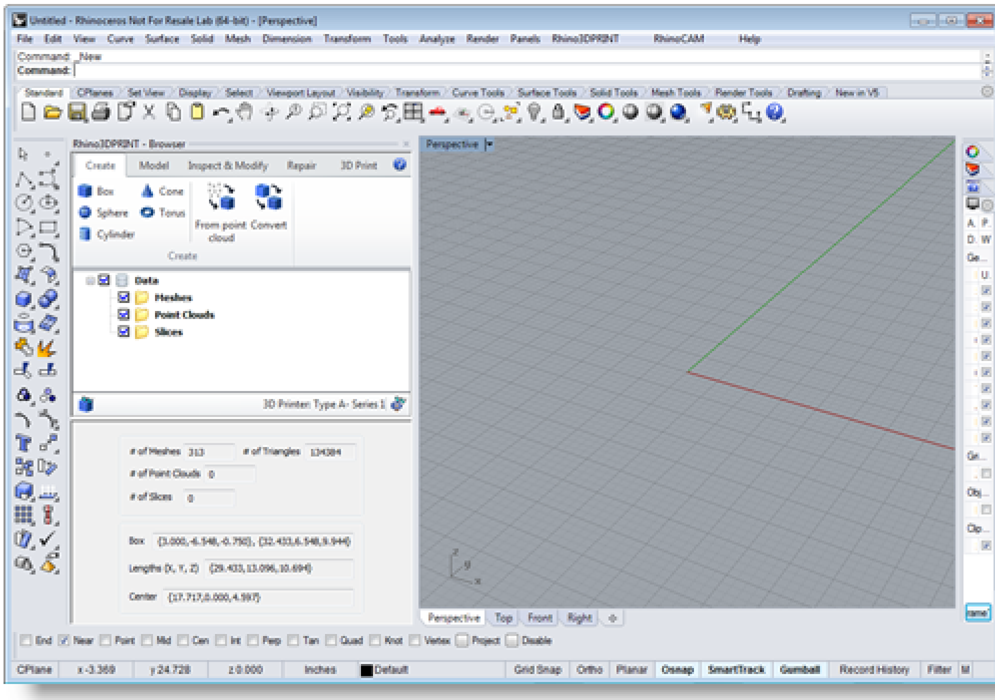
Before we begin, let's talk a bit about the Rhino display.

1. When you run Rhino, your screen may look this.

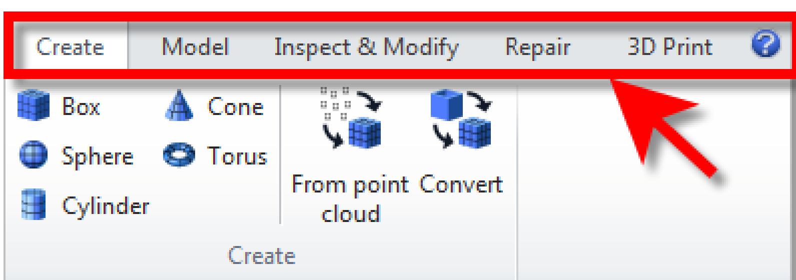


2. Now go to the Main Menu in Rhino and find the Rhino3DPRINT 2017 menu. Dropdown the menu and then select Browser. This will display the Rhino3DPRINT 2017 Browser.





3. There is a ribbon bar at the top of the Rhino3DPRINT 2017 Browser with 5 tabs, Create, Model, Inspect & Modify, Repair and 3D Print.



The Create pane contains mesh creation tools.

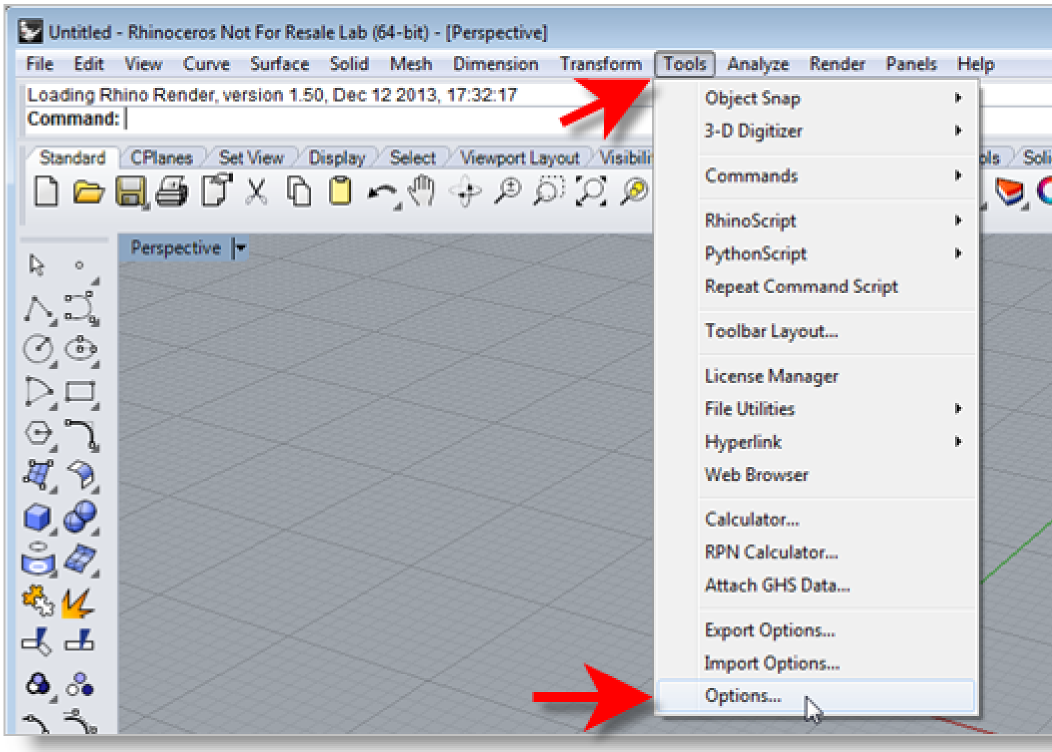
The Model pane contains mesh editing tools.

The Inspect & Modify pane contains analysis and additional editing tools.

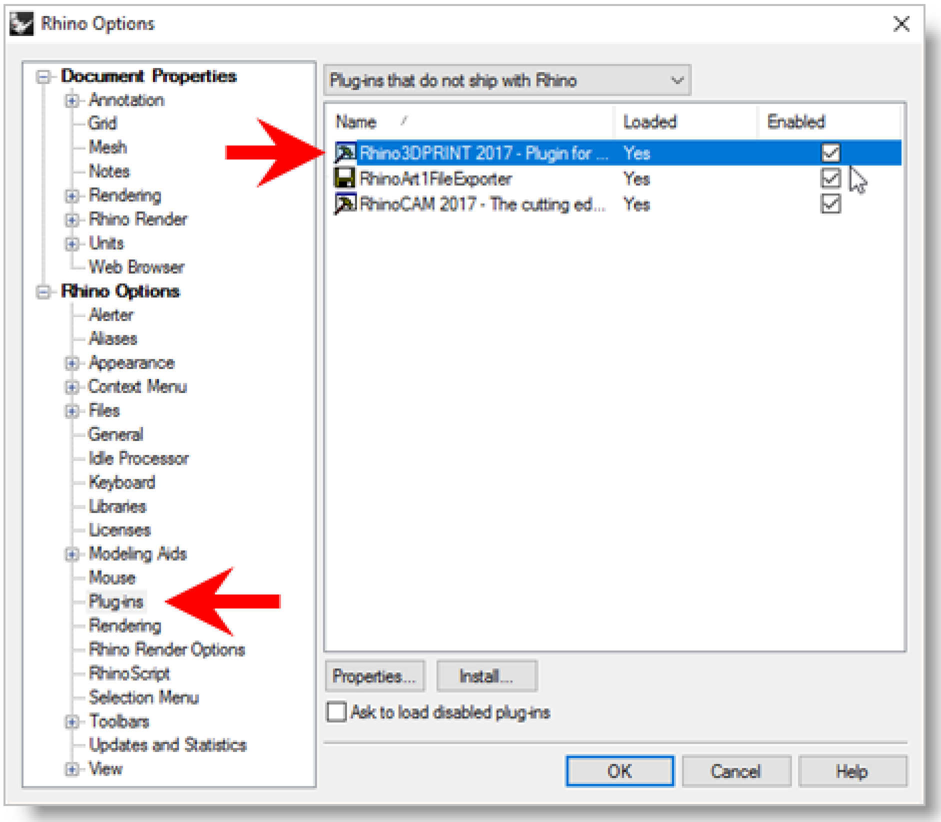
The Repair pane contains various tools for fixing your mesh.

The 3D Print pane contains tools for Preparing, Checking and Printing your mesh to a 3D printer.

4. If the Rhino3DPRINT 2017 menu does not appear on the Rhino Main Menu, select Tools and then Options.



5. Select Plug-ins from the left pane and then locate and check the box next to Rhino3DPRINT 2017 and then pick OK.



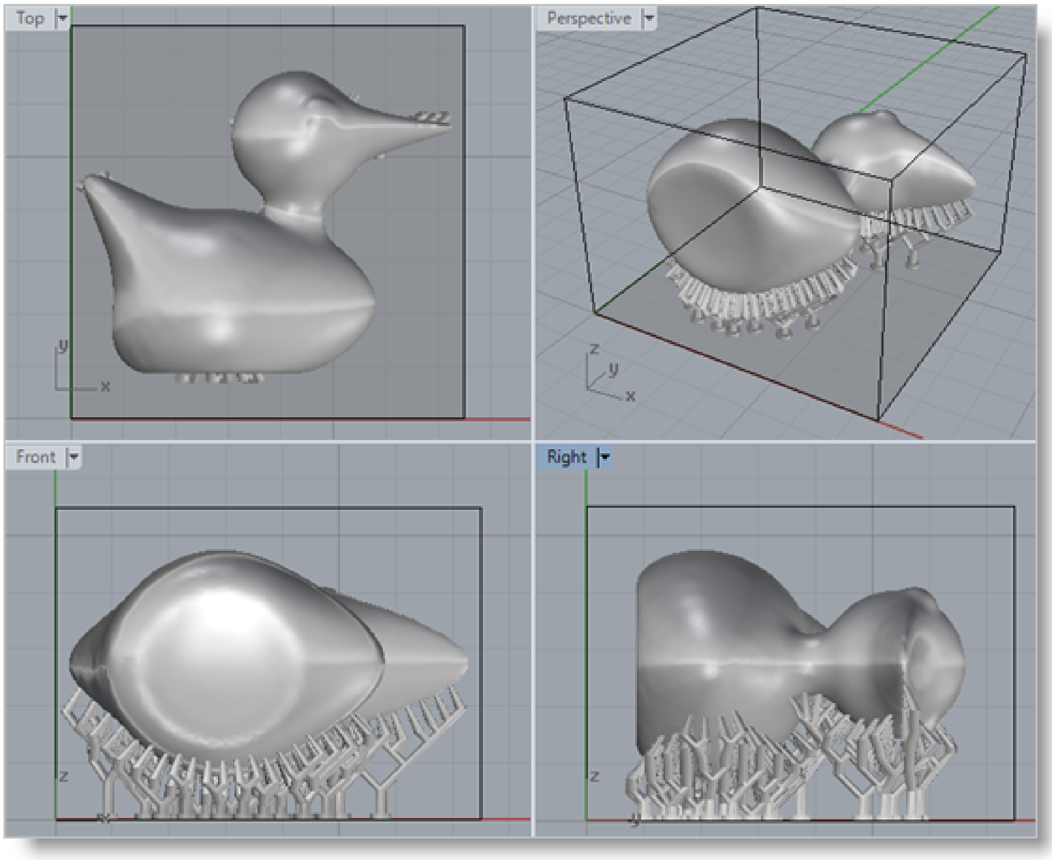
6. You will then need to close and re-open Rhino and Rhino3DPRINT 2017 will then load automatically.

Chapter 3: Point Clouds

In this chapter we will work with Point Cloud data to create and prepare a Mesh model for 3D Printing. We will perform the following basic steps to complete this chapter:

1. Load the Point Cloud Data file.
2. Create a Mesh model from the Point Cloud.
3. Auto Fix the mesh.
4. Re-Mesh the part.
5. Smooth the mesh.
6. Analyze the mesh.
7. Slice the mesh.
8. Select a 3D Printer.
9. Rotate and Fit the mesh to the printer.
10. Check for Printing
11. Generate a Support Structure
12. Export to an STL file.
13. Generate a G-Code file for 3D Printing

    Our completed part will look like this:

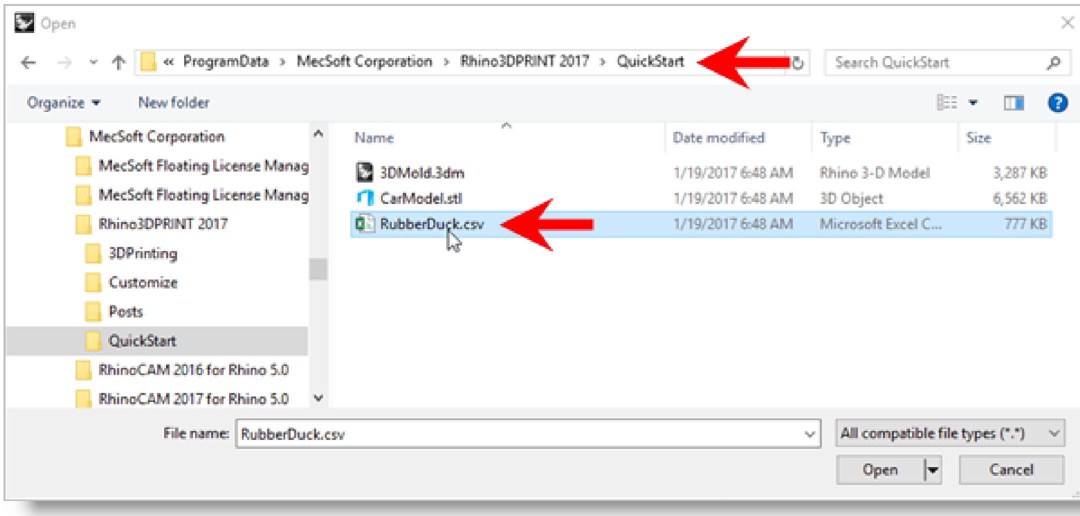


3.1 Load the Part File

Now, let's load the point cloud data for this tutorial.

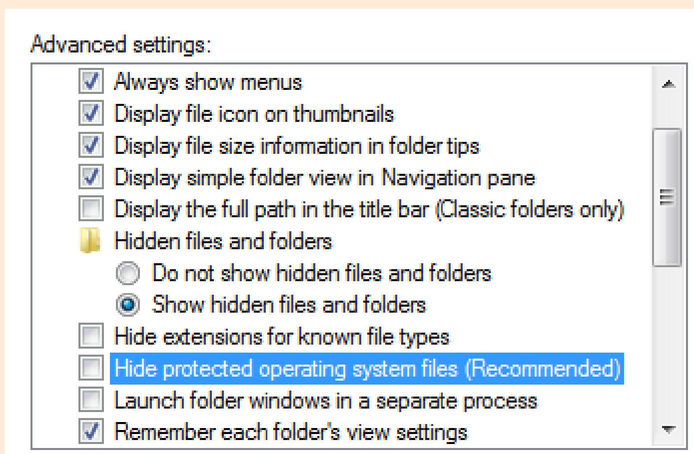
1. From the Rhino Standard toolbar, select the Folder icon.

2. Now, let's load the point cloud data for this tutorial.
3. Find the part file named *RubberDuck.csv* located in the QuickStart folder and then pick Open.



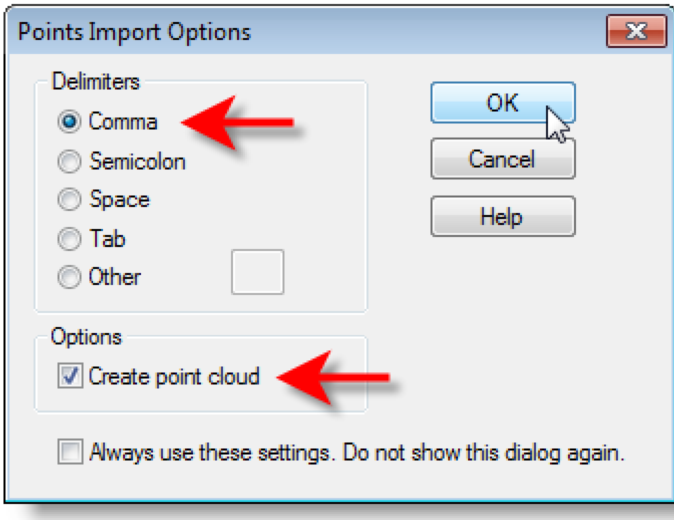
By default, the ProgramData folder is "hidden" from view. Here are the steps to Show hidden files and folders:

1. For Windows7/8/10 users: Go to Control Panel > Appearance and Personalization > Folder Options (Windows XP users can locate folder options under Control Panel).
2. Select View tab and under advanced settings select Show Hidden files and folders, clear the check boxes for:
   - Hide extensions for known file types
   - Hide protected operating system files (Recommended)



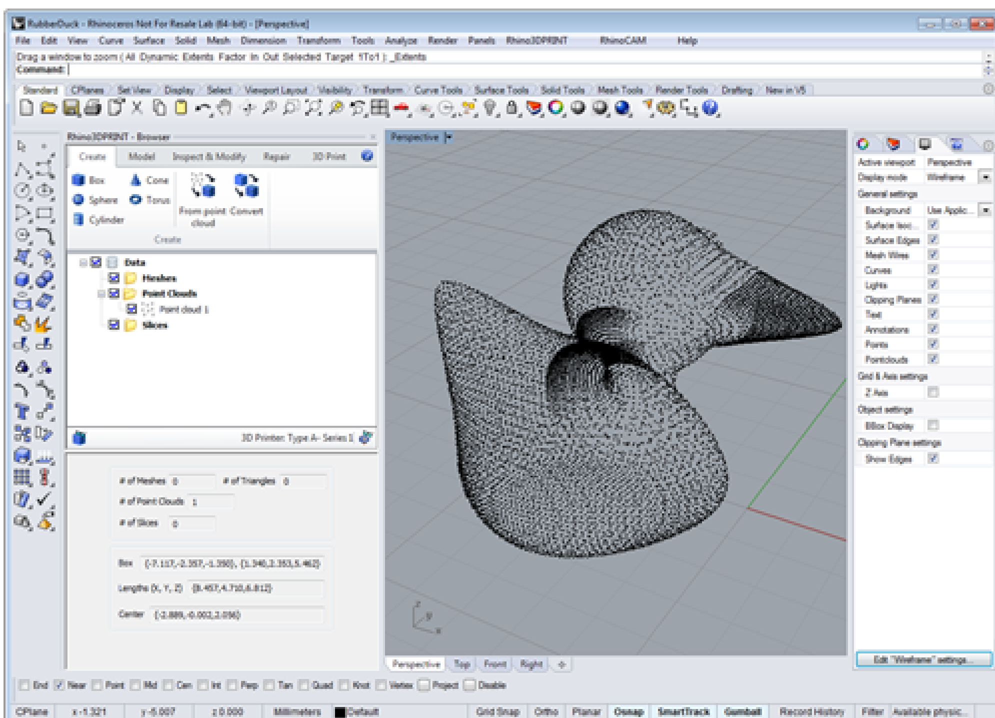
3. Click Apply and OK.

4. From the Points Import Options dialog:



Select Comma under Delimiters
Check Create point cloud under options
Pick OK to import the point file

5. Once imported, select the Perspective view to work in.

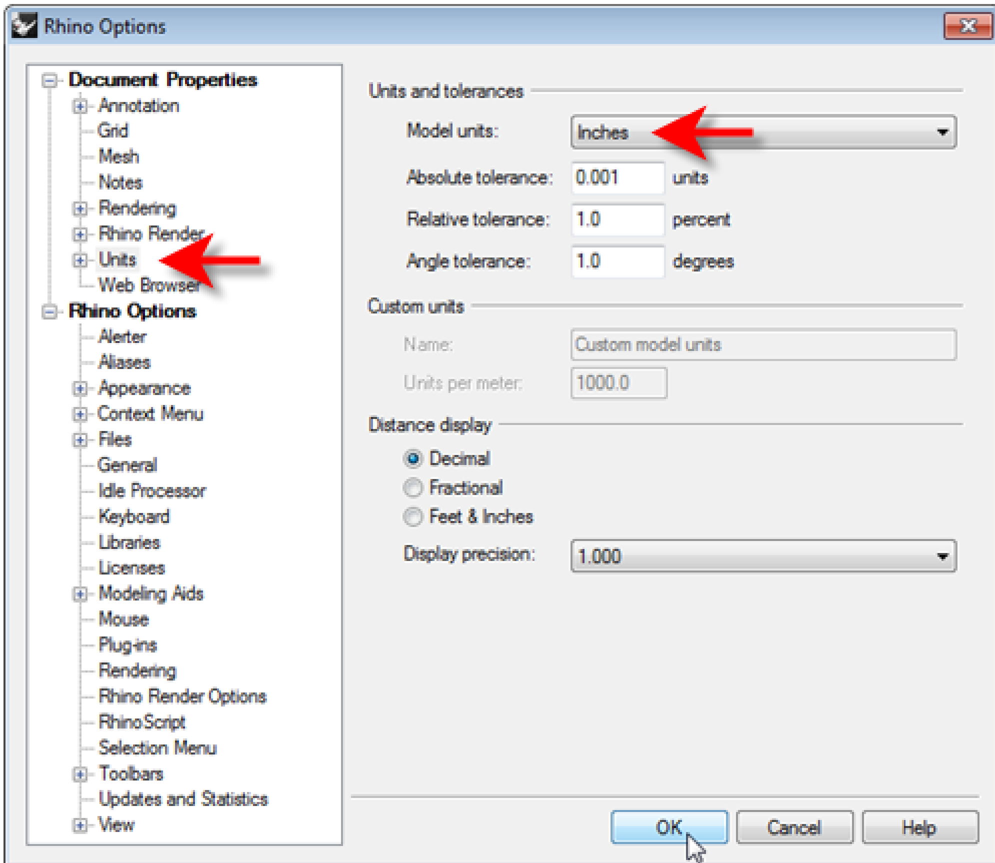


3.2 Mesh from Point Cloud

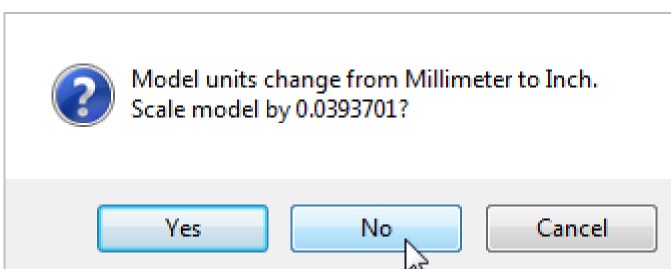
First, let's set the correct Part Units in Rhino.

1. From the Rhino Main Menu, select Tools and then Options.

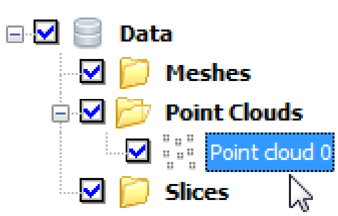
2. Select Units section from the tree and then set the Model Units to Inches and pick OK.



3. Pick No when it asks you to scale the model.

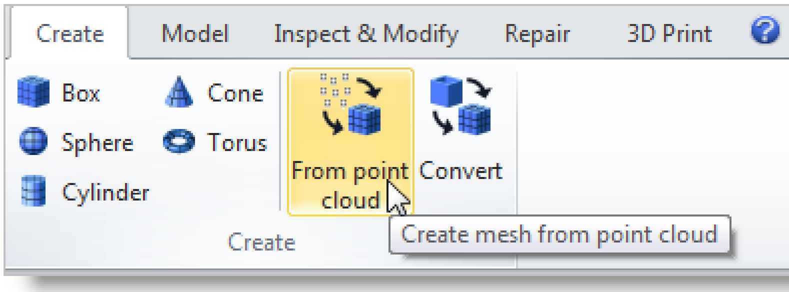


In the Browser we see that we have one point cloud.

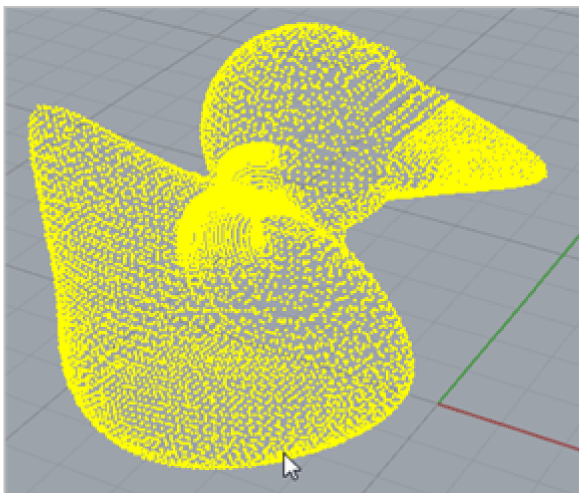


4. From the Create tab of the Rhino3DPRINT 2017 Browser, select From Point Cloud to

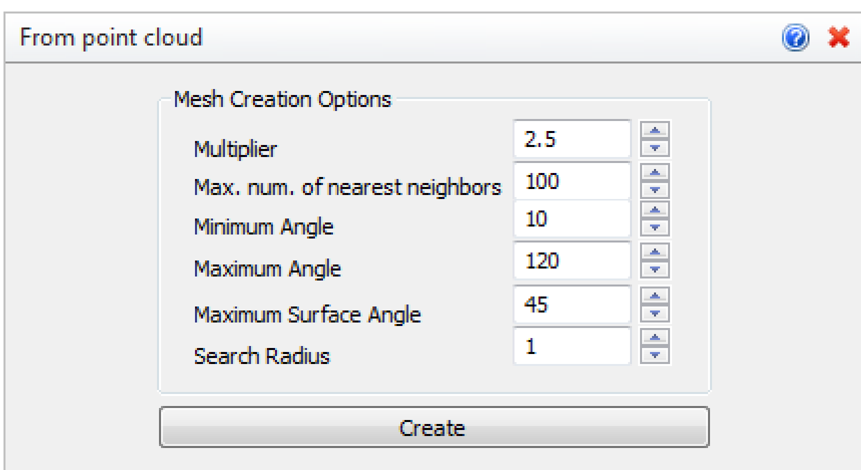
create a Mesh model from the point cloud.



5. Select the point cloud and press Enter.
Note: If the Point Cloud is already selected from the previous step, just press Enter.

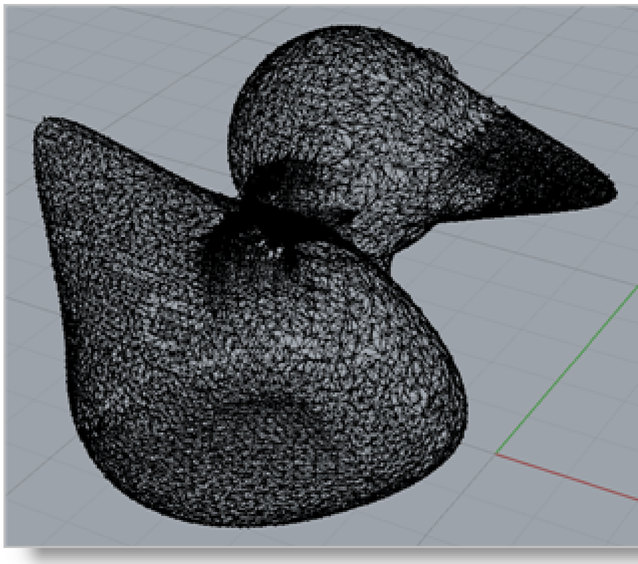


Notice that an options dialog appears below the Browser.

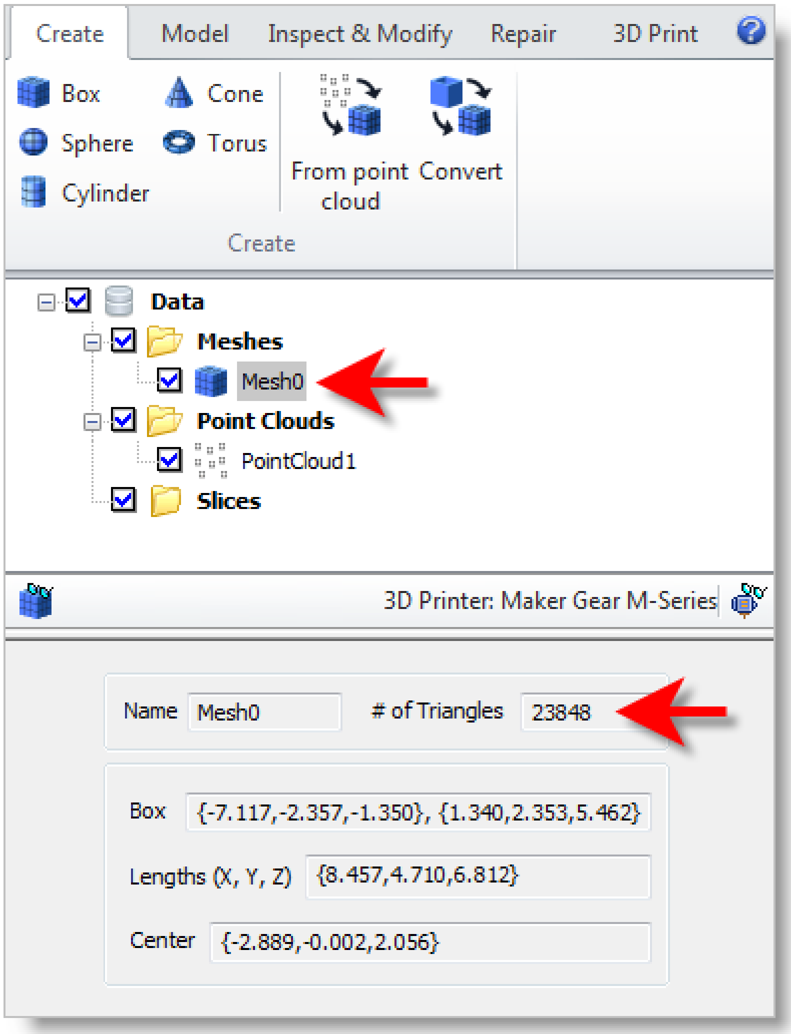


6. Accept the default options and then pick Create.

From the Browser we see that we now have 1 Point Cloud and 1 Mesh with over 23,000 Triangles. It is also displayed on the screen.

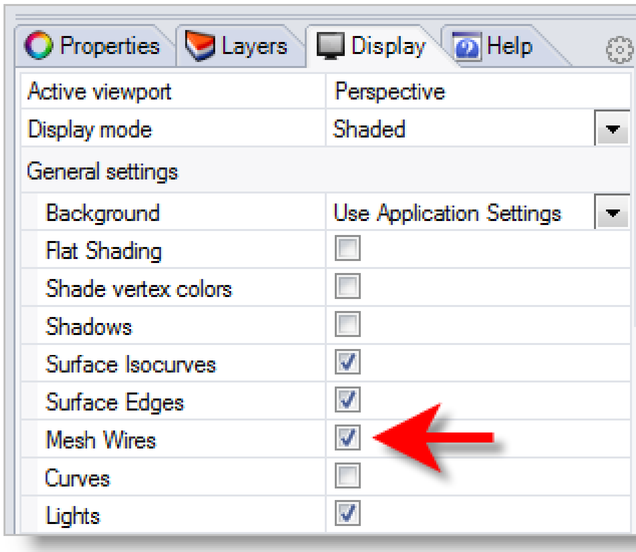


Mesh from Point Cloud



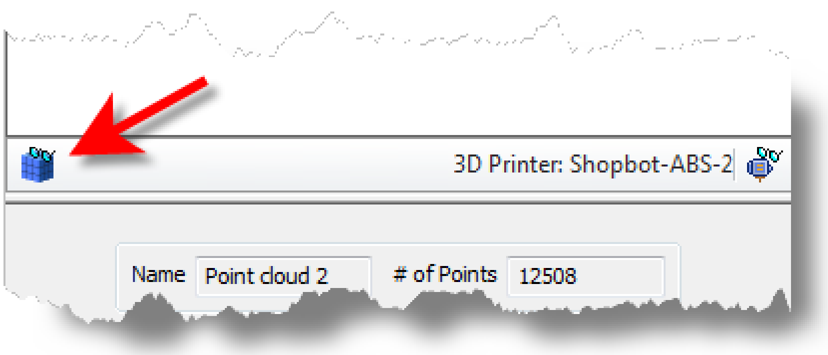
7. From the Rhino Standard toolbar, select Shaded Viewport.

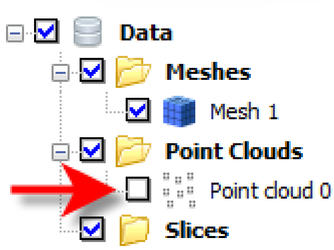
8. From the Rhino Display tab check the box next to Mesh Wires. This will allow for better viewing of the mesh.



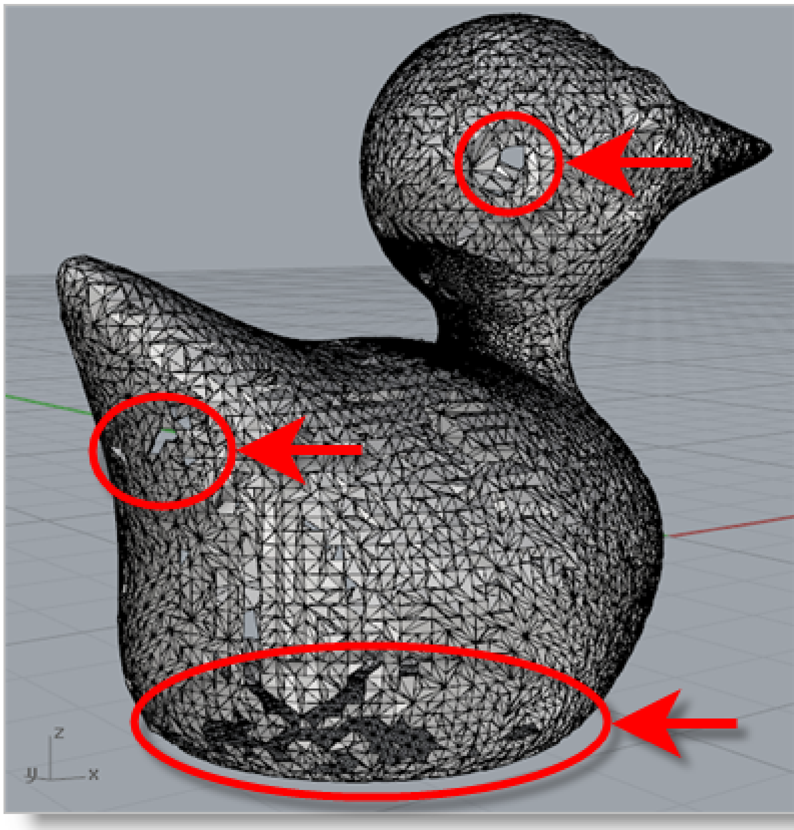
Display Mesh Wires in Rhino

9. Then from the Rhino3DPRINT 2017 Browser we'll first make sure that the icon for Toggle Hidden Objects in Tree is turned Off, then we will hide the point cloud (by unchecking it) and take a look at the mesh that was created.





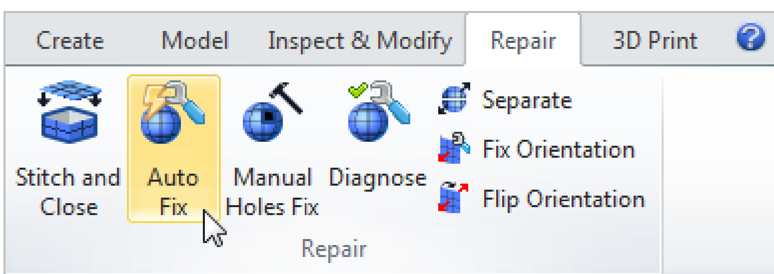
We notice that the point cloud data is incomplete, leaving the model incompatible for 3D printing.



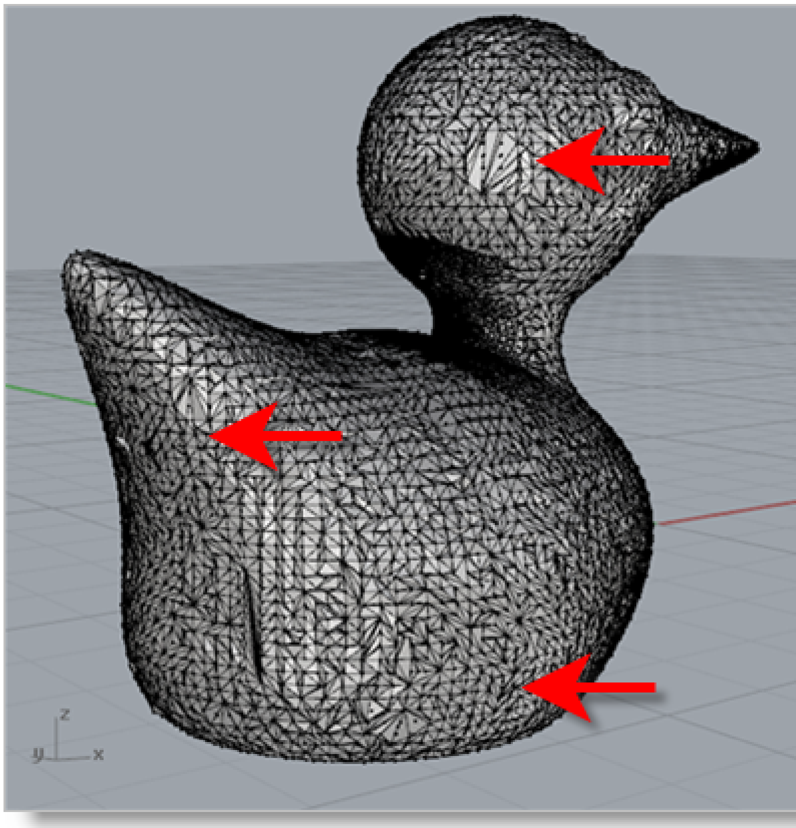
3.3 Auto Fix

We can repair the mesh automatically.

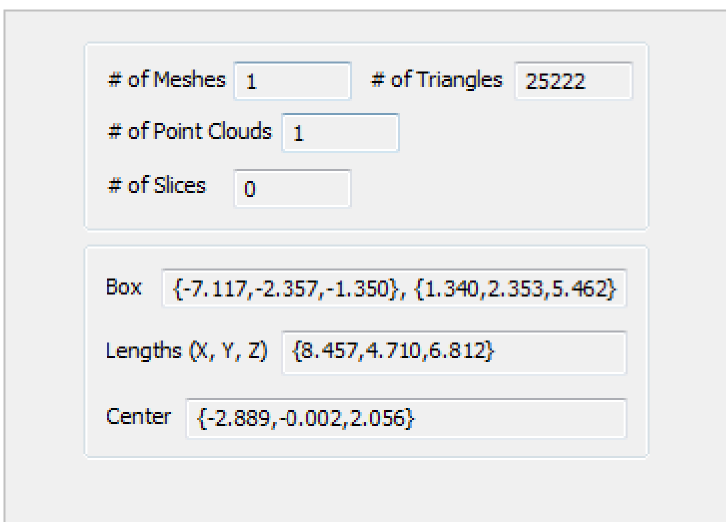
1. From the Rhino3DPRINT 2017 Browser select the Repair tab and then select Auto Fix. This will automatically fix holes, self-intersecting facets and more.



2. Select the mesh model and then press Enter. We see now that all of the gaps are closed.



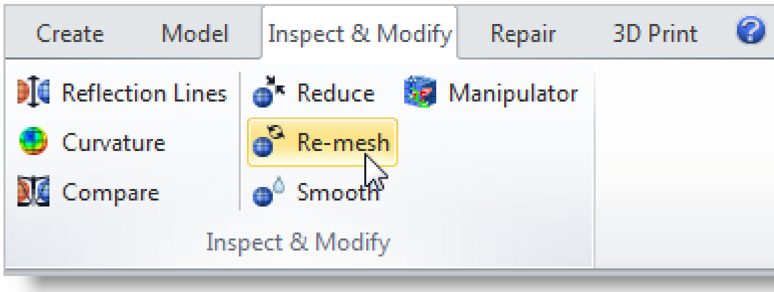
The browser now reports that the mesh has over 25,000 triangles. The exact # of Triangles reported may differ slightly depending on the exact build version of Rhino3DPRINT 2017 you are running.



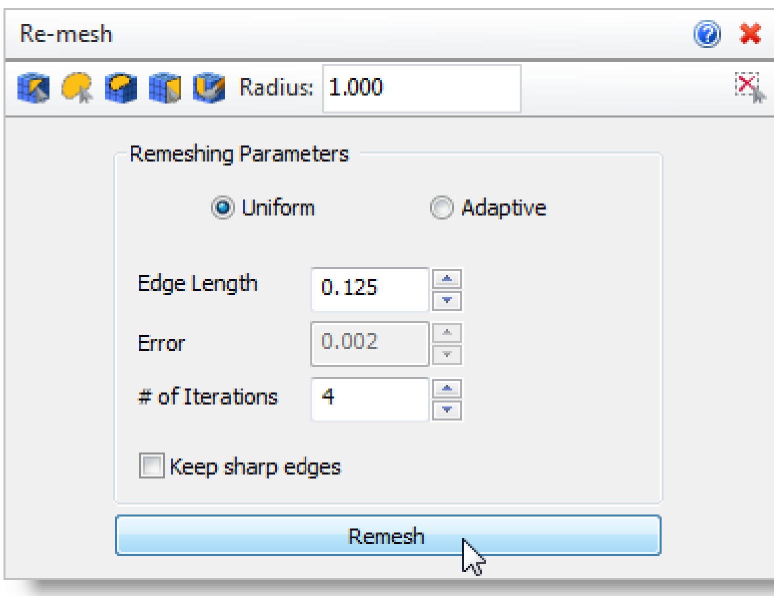
3.4 Re-Mesh

Next we'll re-mesh the model creating a more uniform mesh.

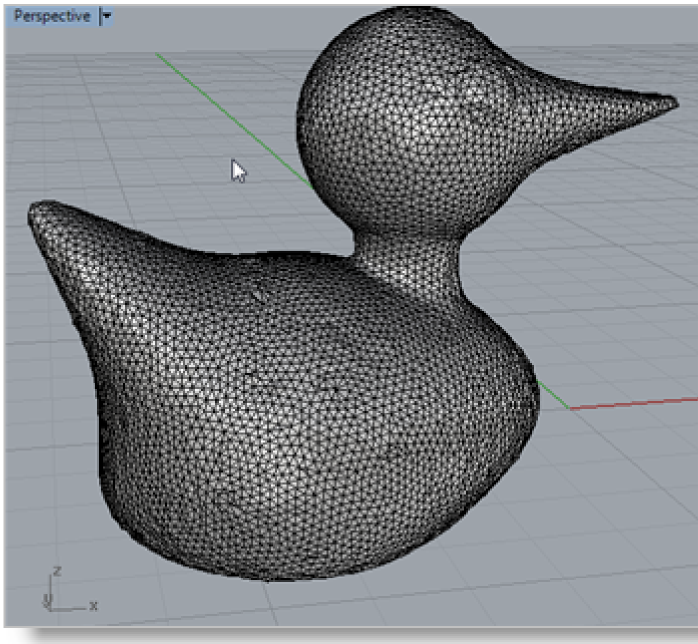
1. From the Inspect & Modify tab, select Re-mesh.



2. Now we select the mesh model and press Enter to display the Re-mesh dialog.
3. Selecting the Uniform method will re-mesh the model based on a fixed Edge Length.
4. Then set the Edge Length to 0.125 and the # of Iterations to 4.
5. Then uncheck Keep sharp edges and then pick Remesh.

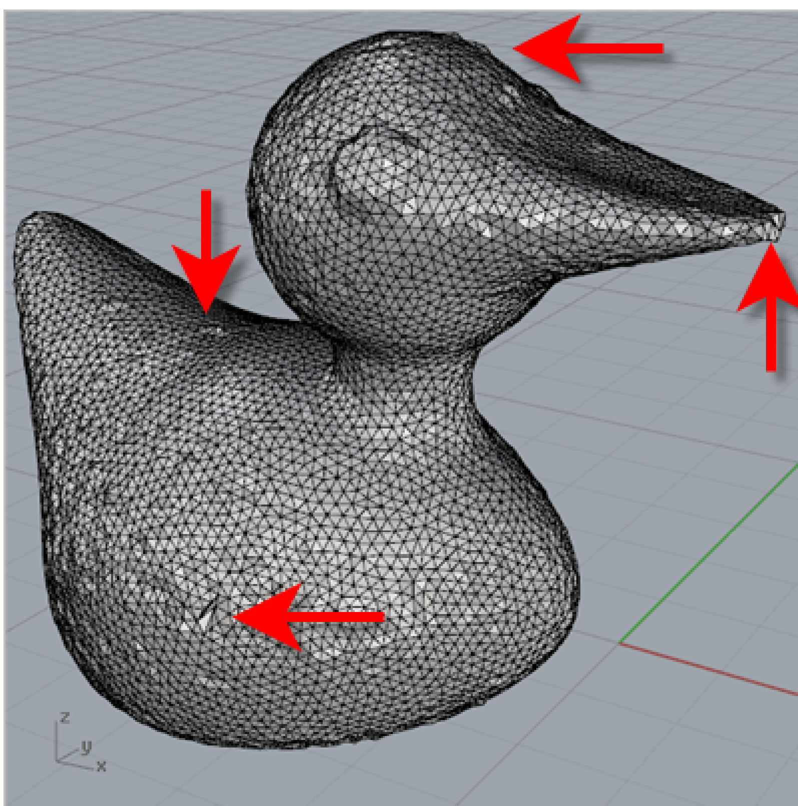


With Mesh Wires checked from the Rhino Display tab, we now see that each facet in the mesh model is uniform.

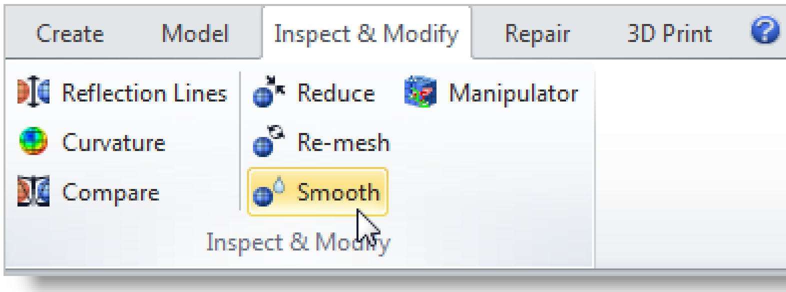


3.5 Smooth

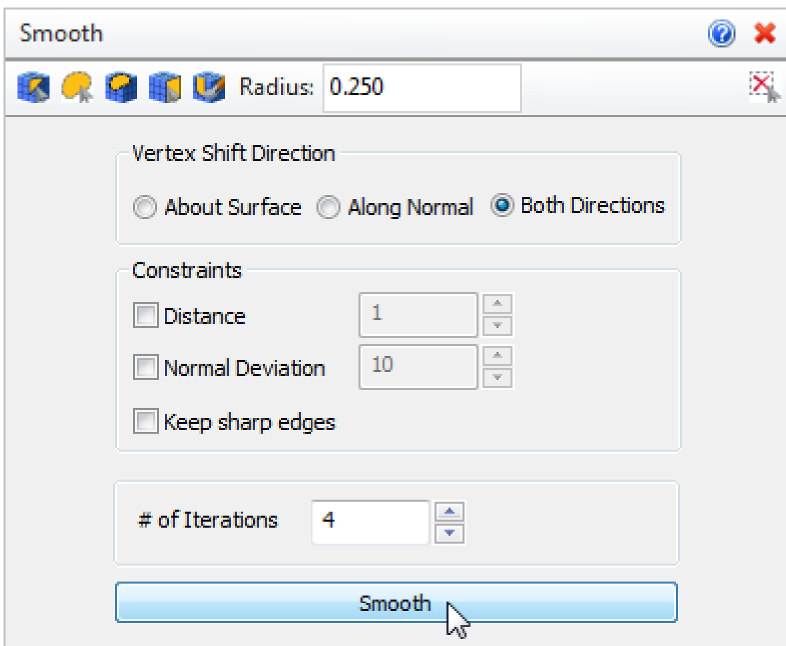
Although uniform, we still see areas in the mesh that are rough. Let's smooth these out a bit.



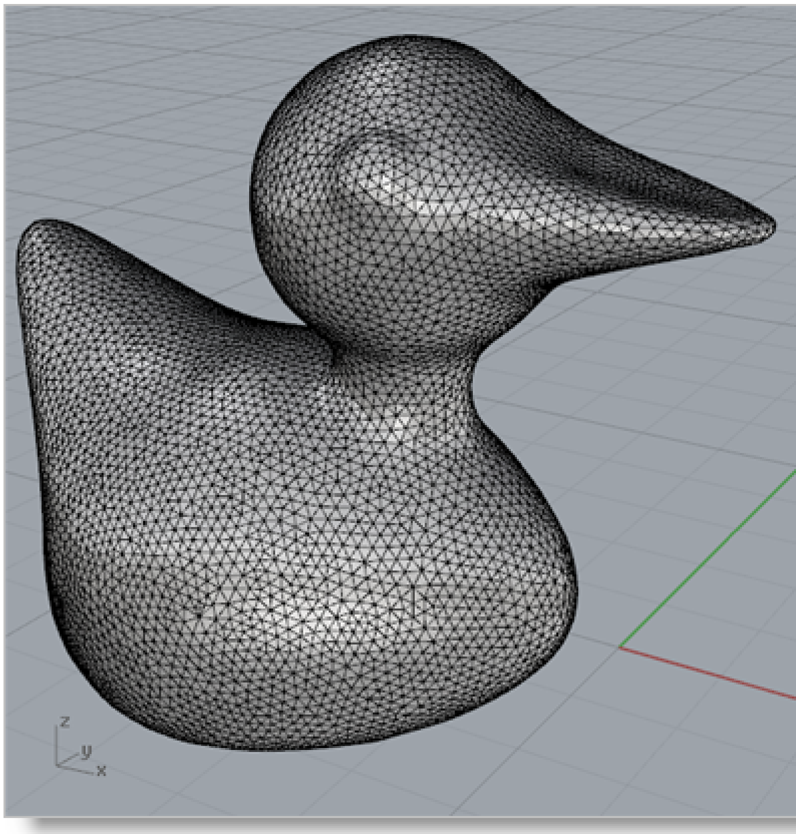
1. From the Inspect & Modify tab, select Smooth.



2. Now we select the mesh model and press Enter to display the Smooth dialog.
3. Next, we'll allow the Vertex of each facet to shift in Both Directions (inward and outward). This means that any smoothing will both add and remove facets as needed.
4. We'll set the # of Iterations to 4. This means that the second iteration will Smooth the results of the first and so on.



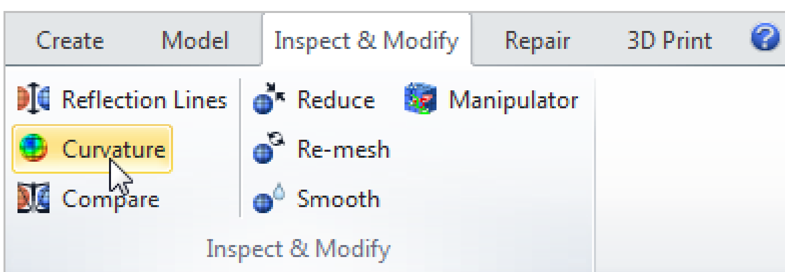
5. Now we'll pick the Smooth button and review the results. We see that the mesh looks much smoother now.



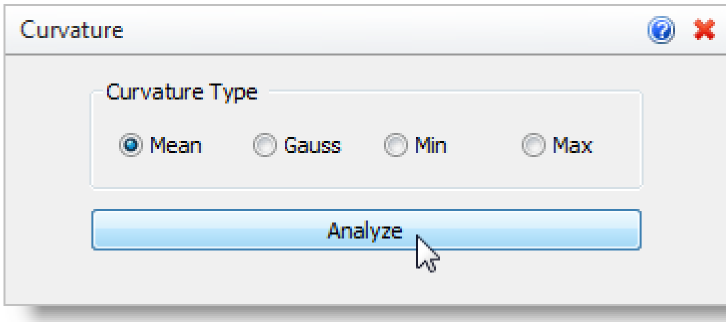
3.6 Analyze

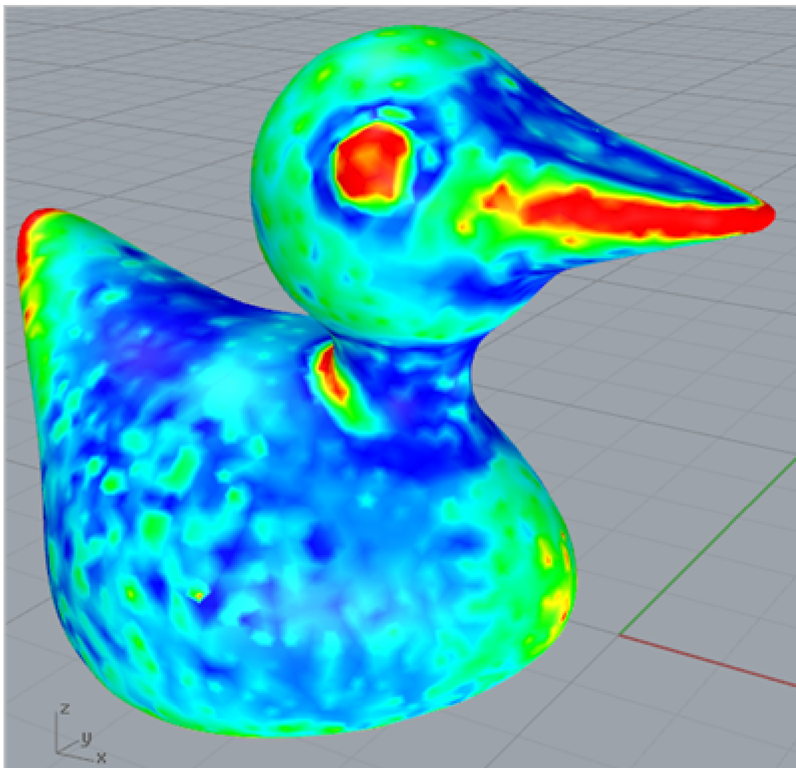
Next we'll analyze the mesh model using a couple of different methods.

1. From the Inspect & Modify tab, select Curvature. This will highlight areas of curvature across the model.

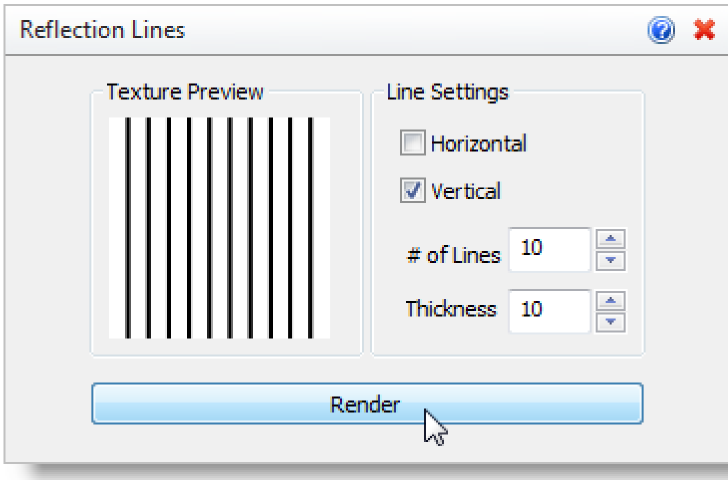


2. Now we select the mesh model and the Curvature dialog is displayed automatically.
3. We'll leave the Curvature Type set to Mean and then pick Analyze to see the results.





4. You can pick the Cancel icon ✖ to close the command dialog leaving the mesh selected.
5. Now, from the Inspect & Modify tab, select Reflection Lines. This method will indicate any irregularities or tangent discontinuities.
6. Just press Enter since we left the mesh selected from the previous command.

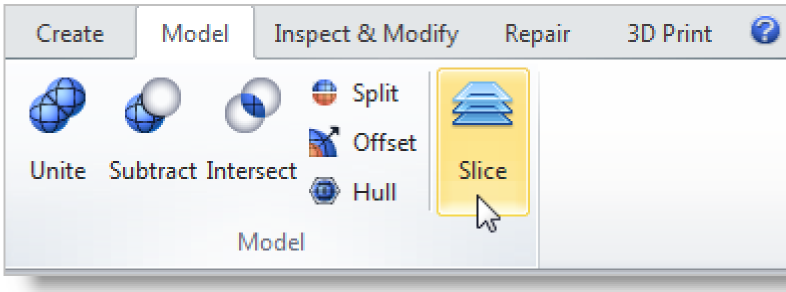


7. Again, you can pick the Cancel icon ✖ to close the command dialog leaving the mesh selected.

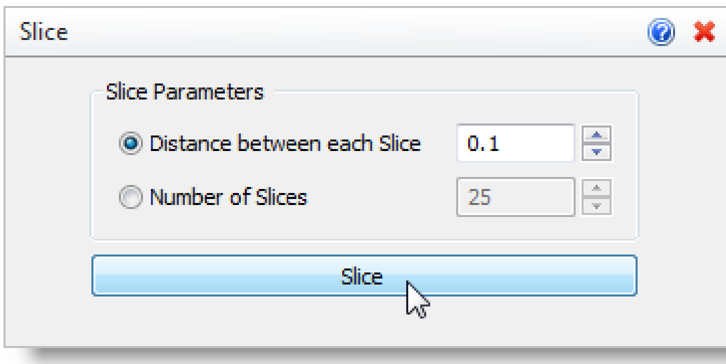
3.7 Slice

Next, we'll visually Slice the mesh model to simulate levels of 3D printing that will occur.

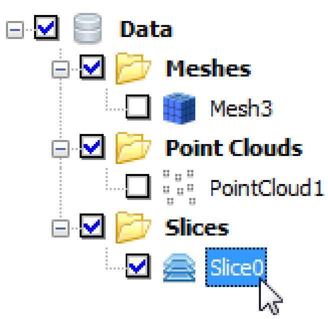
1. With the mesh still selected from the previous step, from the Model tab, select Slice.

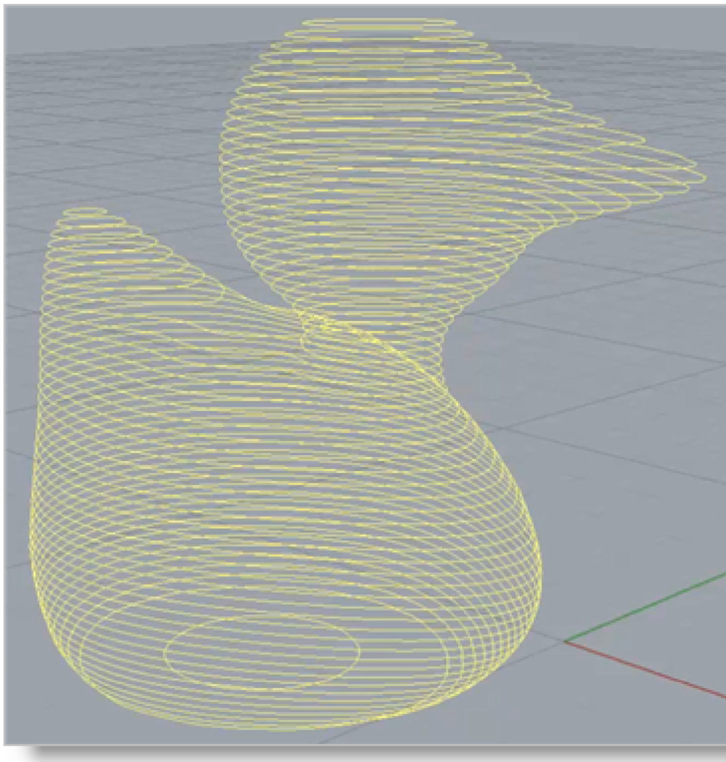


2. From the Slice dialog, we'll set the Distance between each slice to 0.1 and then pick Slice.

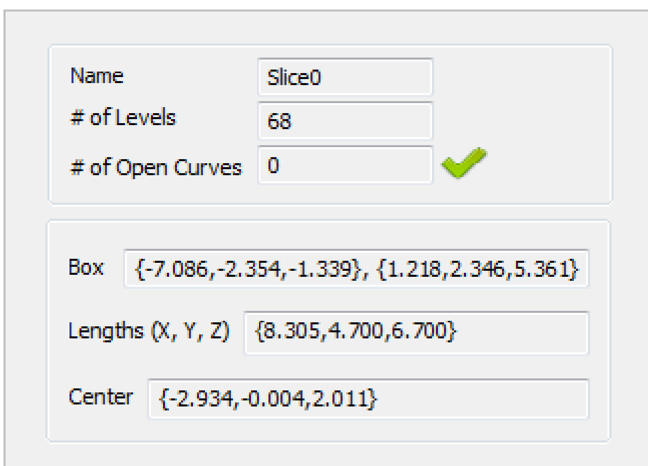


3. From the Browser, first check the Slice object and then uncheck the Mesh object. The Slice will be isolated and clearly highlighted on the screen.



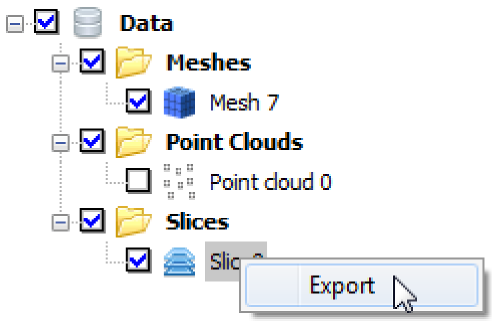


The Browser displays information about the Slice. Notice that the  icon is displayed indicating that NO open curves are present.

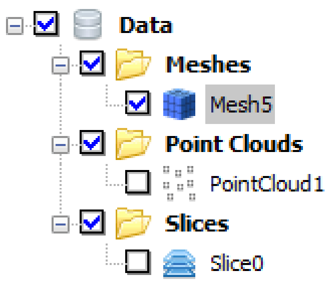


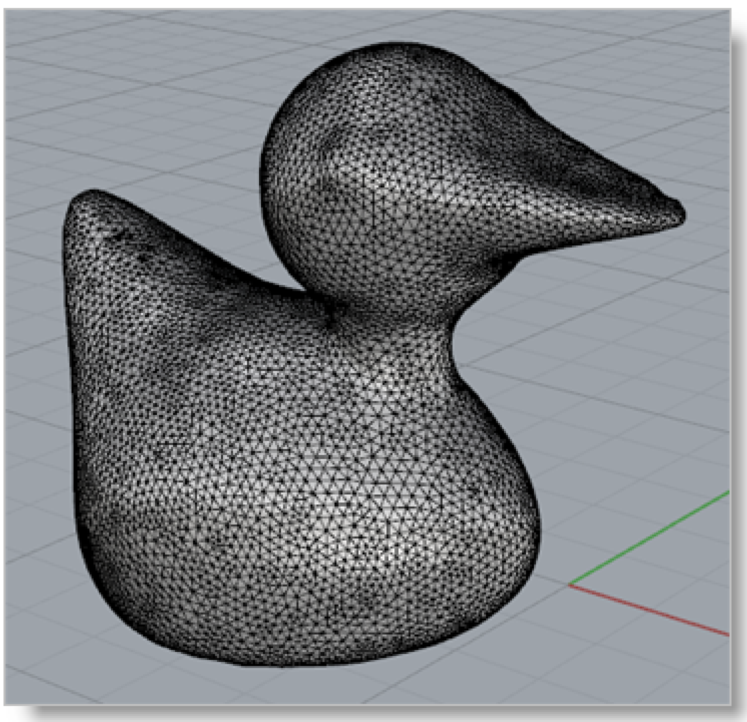
Name	Slice0
# of Levels	68
# of Open Curves	0
Box	{-7.086,-2.354,-1.339}, {1.218,2.346,5.361}
Lengths (X, Y, Z)	{8.305,4.700,6.700}
Center	{-2.934,-0.004,2.011}

4. If you right-click on the Slice object from the Browser you can Export it to actual curve geometry later if needed.



5. For now, we'll hide the Slice from view by unchecking it and display the mesh again by checking it.

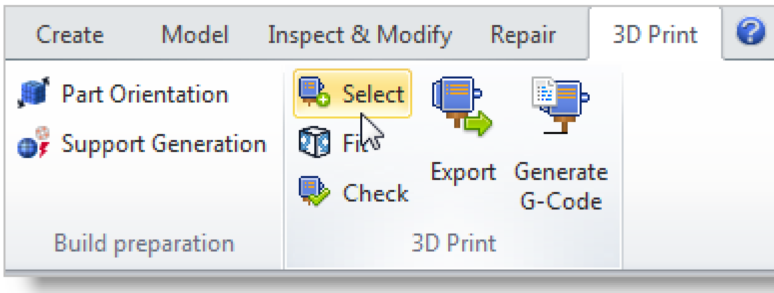




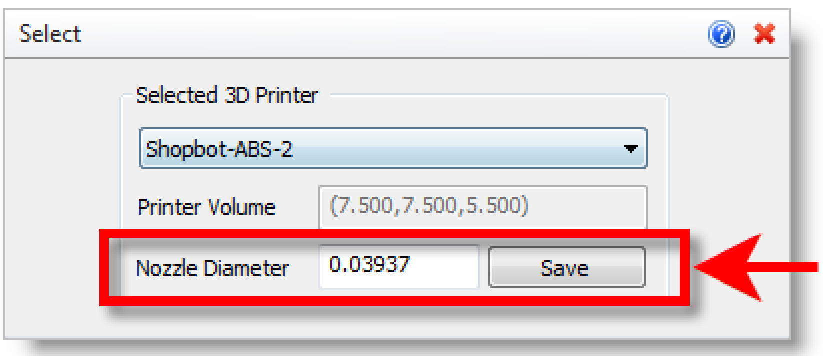
3.8 Select Printer

Now, we'll setup the model for 3D printing.

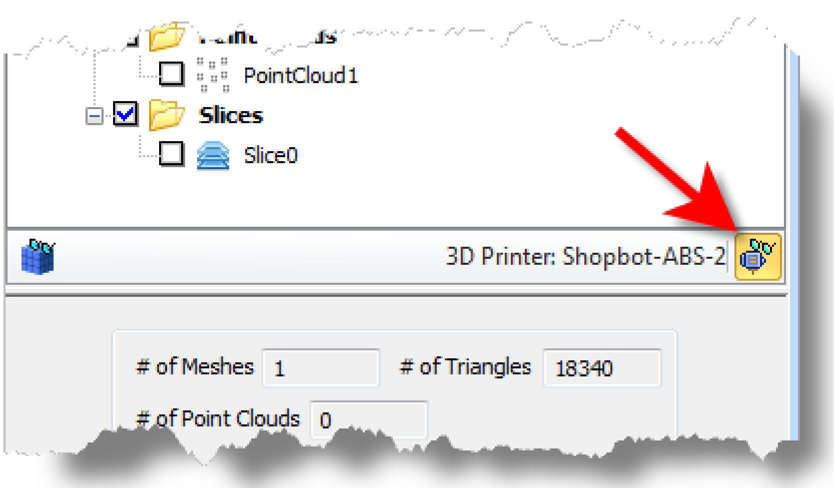
1. From the 3D Print tab, pick Select.



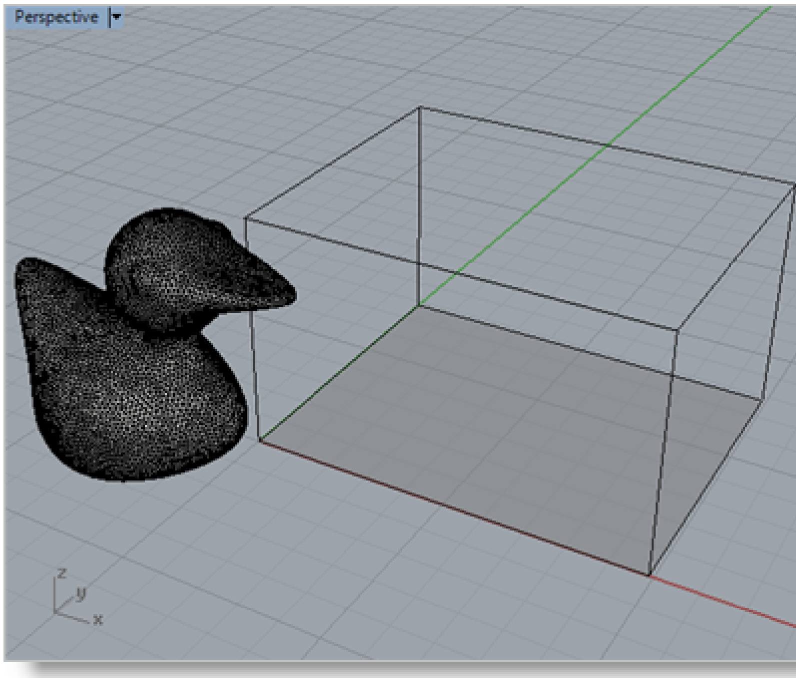
2. From the list of 3D printers we select our target printer to be ShopBot-ABS-2. It has a printer volume of 7.5" x 7.5" x 5.5" inches, just perfect for our rubber duck.
3. Make sure to set the Nozzle Diameter value to match the actual nozzle currently installed on your 3D printer.



4. From the Browser toolbar, select the icon to Toggle the visibility of the 3D printer's volume ON.



We now see it on the screen but notice that the mesh is positioned outside of the 3D printer's volume!

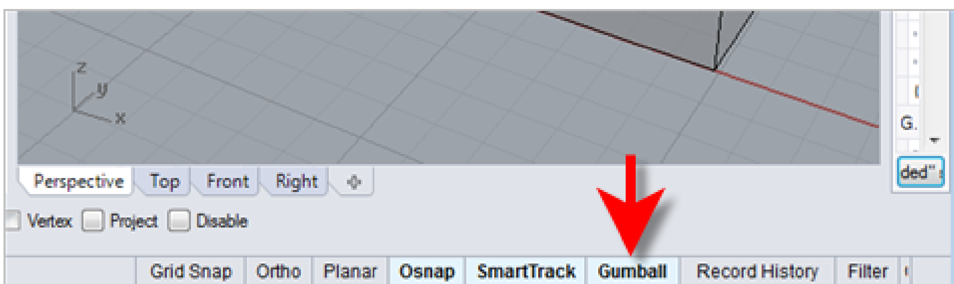


5. Pick the Cancel icon ✖ to close the Select command dialog.

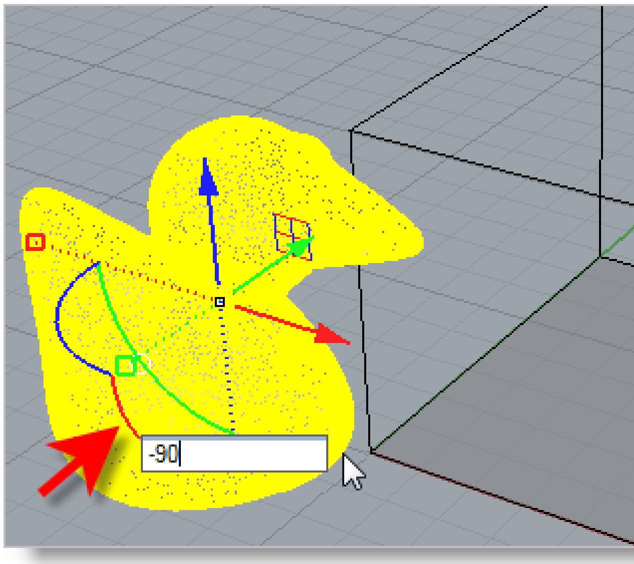
3.9 Rotate and Fit

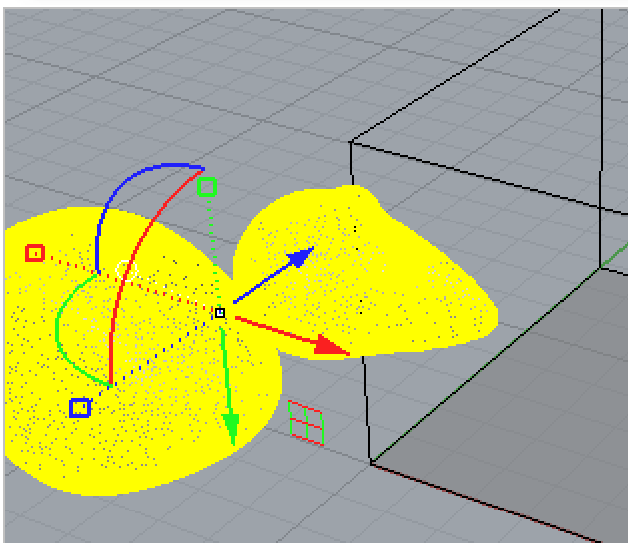
For this print we want to Rotate the rubber duck so that it lies on its side. Then we will Fit it to the 3D Printer's volume automatically.

1. From the Rhino Status Bar, toggle ON the Gumball.

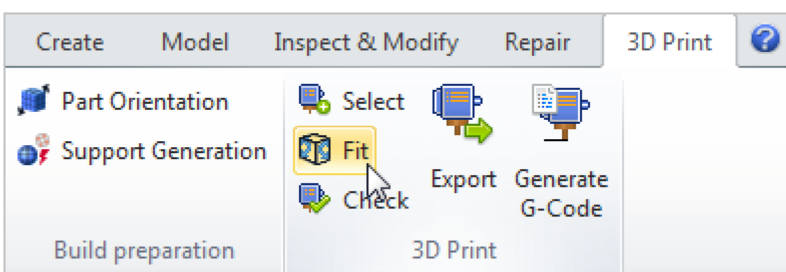


2. Now select the mesh and the Manipulator will display. Select the X Rotation arc and enter -90 and press Enter. The mesh will rotated about the X axis by -90 degrees.

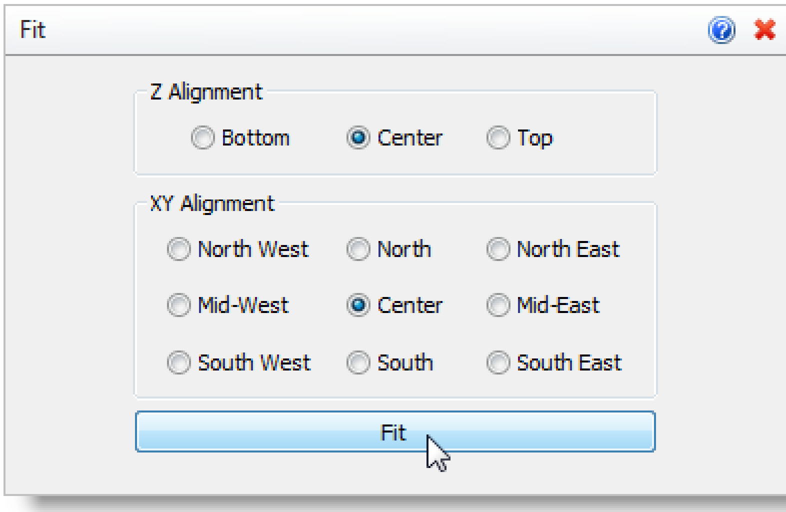




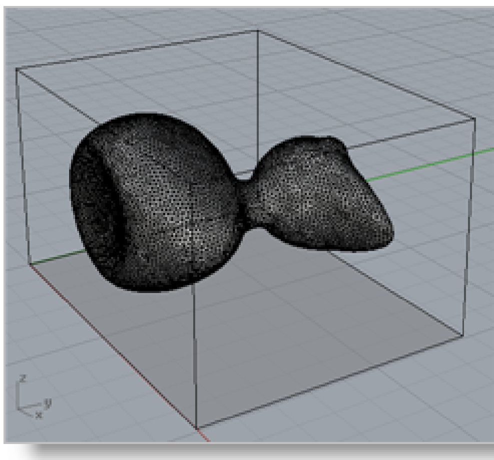
3. Now we will Fit the mesh to the printer. Select the 3D Print tab and then pick Fit.



4. We will be generating a support structure for the rubber duck so we don't want it to lay at the bottom of the printer. So for the Z Alignment we select Center and for XY Alignment we'll select Center and then pick Fit.



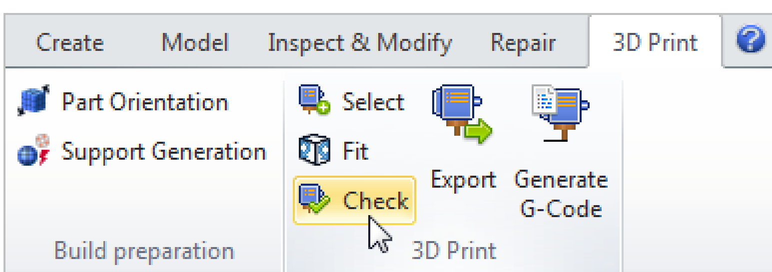
The rubber duck is automatically placed at the center of the printer's volume.



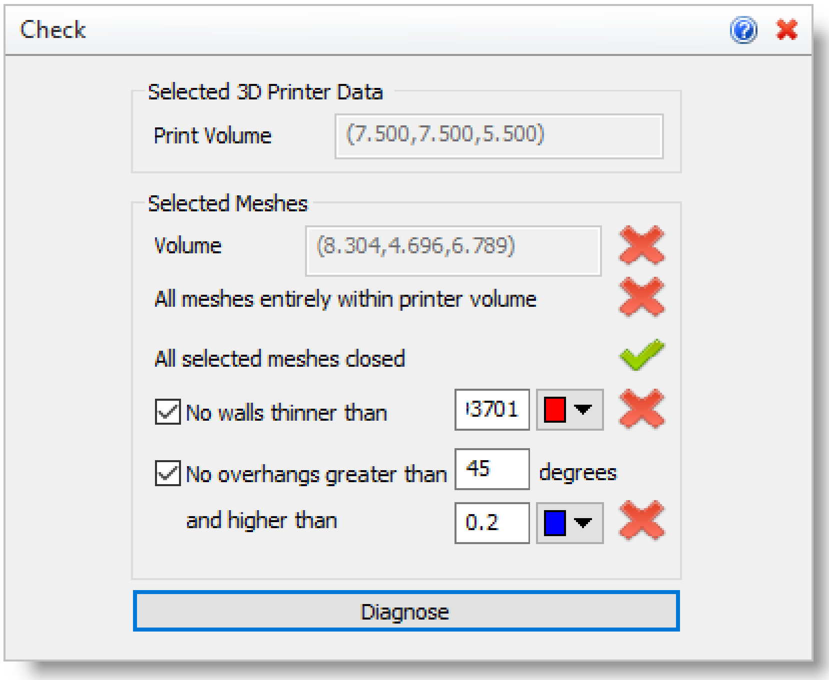
3.10 Check

Now let's check the mesh model to see if it's truly ready for printing.

1. From the 3D Print tab, pick Check. Select the mesh and then press Enter to display the dialog.



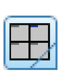
2. Notice that the icons tell us the following:

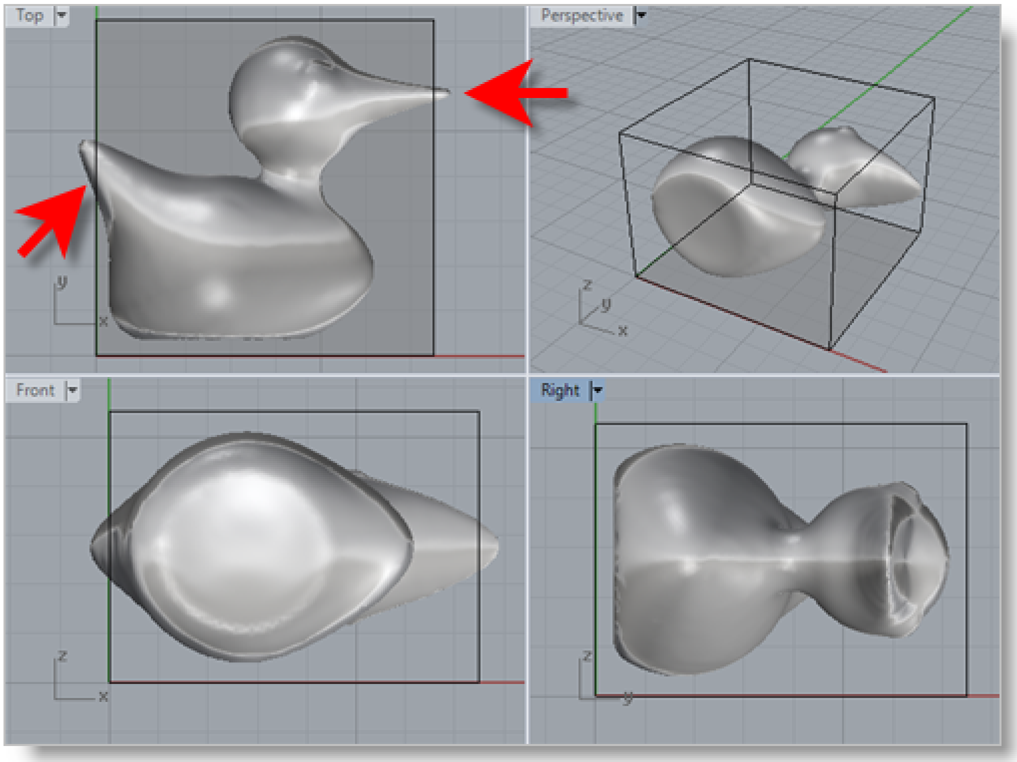


Check		Comment
	The mesh volume exceeds the printer's volume.	We need to fix this
	Mesh is outside the printer's volume.	We need to fix this
	There are walls thinner than 0.1.	This is acceptable
	There are overhangs greater than 45 degrees and higher than 0.2.	We will add supports

3. Pick the Cancel icon to close the dialog.

4. Now let's have a look to see what the problem is. From the Rhino Standard toolbar, select 4 Viewports. We see that the mesh extends past the printer's volume on two sides.





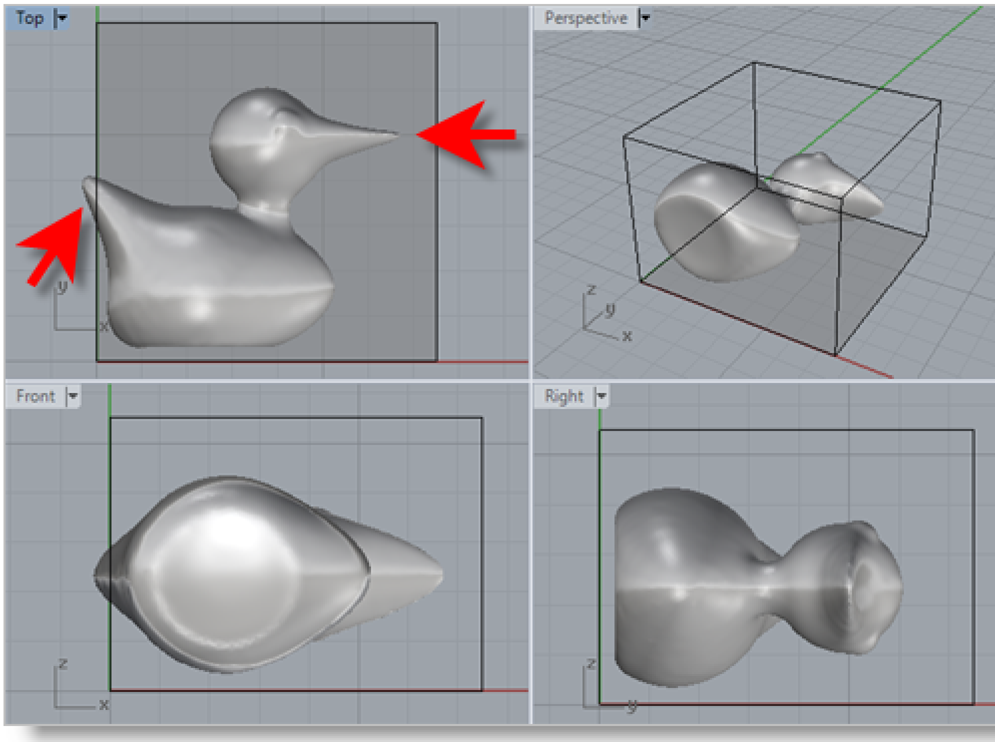
3.11 Scale

We need to scale the part to fit into our 3D Printer.

1. First select the mesh, then from the Rhino Main Menu select Transform > Scale > Scale 3-D.

   For the Origin enter 0,0,0 and press Enter
   For the Scale Factor enter 0.85 and press Enter

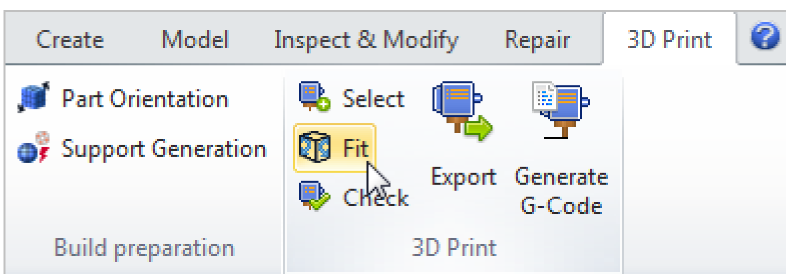
2. Your part should now look like this:



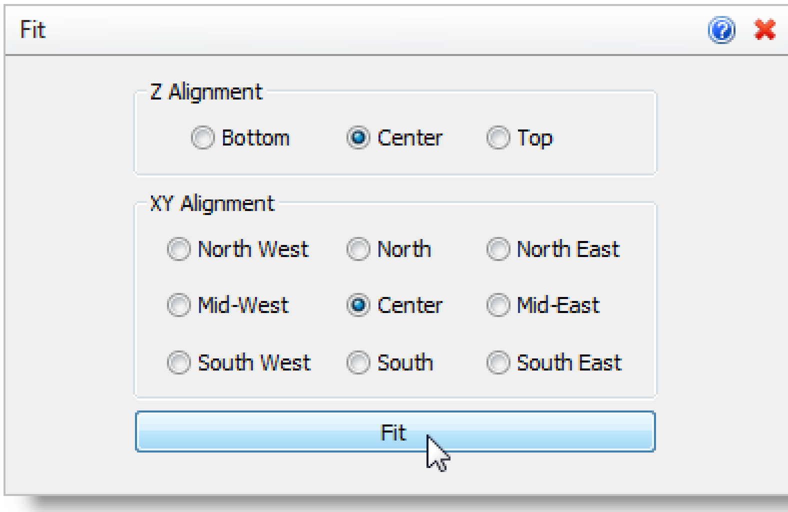
3.12 Re-Check

Now let's check the mesh model again to see if it passes the 3D Printer Check. First we need to again Fit the mesh to the 3D Printer's volume.

1. Select the 3D Print tab and then pick Fit.

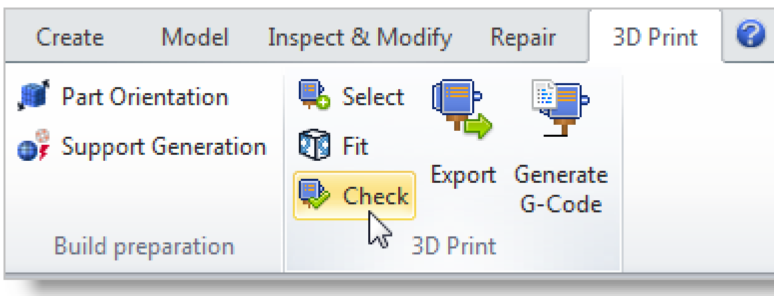


2. Both the Z Alignment and the XY Alignment should default to Center. Make sure they are set and then pick Fit.

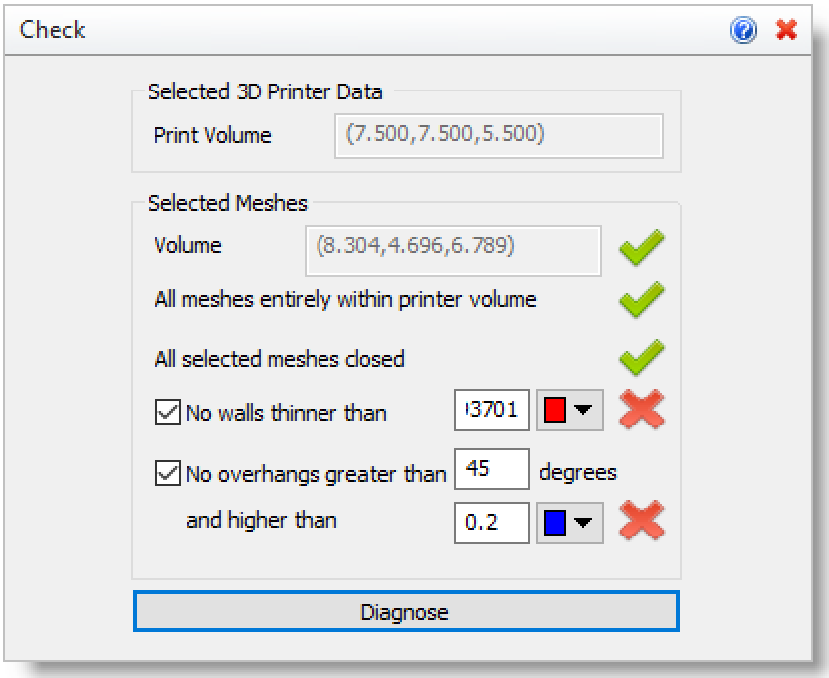


The rubber duck is again automatically placed at the center of the printer's volume.

3. Now let's Check again. From the 3D Print tab, pick Check. Select the mesh and then press Enter to display the dialog.



4. Notice that the  icons tell us that the mesh is now ready to generate a support structure.



5. Pick the Cancel icon to close the dialog.

3.13 Support Structure

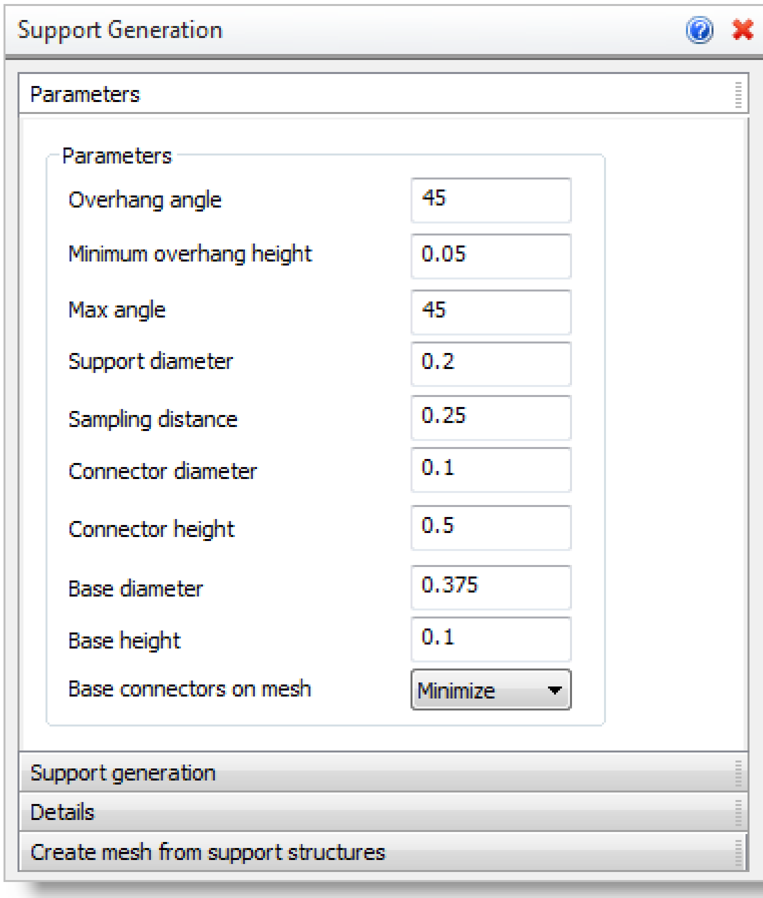
A cool new feature in Rhino3DPRINT 2017 is the ability to organically *grow* a support structure under the part to support it during 3D printing. Using a support structure may eliminate the need to split your part into separate pieces for printing. The command has many options but we will only discuss the basics here. To learn more, select the help icon at the top of the command dialog.

> **Always use Fit & Check before adding Supports**: Using this command "grows" a support structure from the base of your printer volume up to the part. This is why it is important to make sure your part is properly positioned (using the Fit command) and also passes the Check command.

1. Since the mesh has been Fit and Checked in a previous step, we can toggle the printer volume visibility Off to work on the support structure.

   Select from the Rhino3DPRINT 2017 Browser toolbar to hide the printer volume.

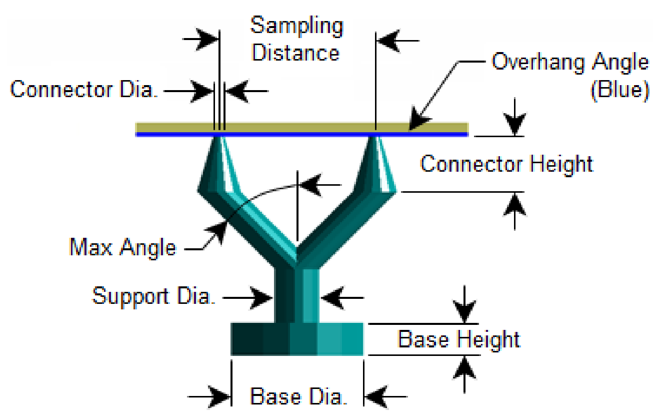
2. From the 3D Print tab, select the Support Generation icon.
3. Now we select the mesh model and the Support Generation dialog will display.



4. This dialog has 4 tabs called Parameters, Support generation, Details and Create mesh from support structures. We will work from top to bottom beginning with the Parameters tab.

For Parameters, enter the following values.

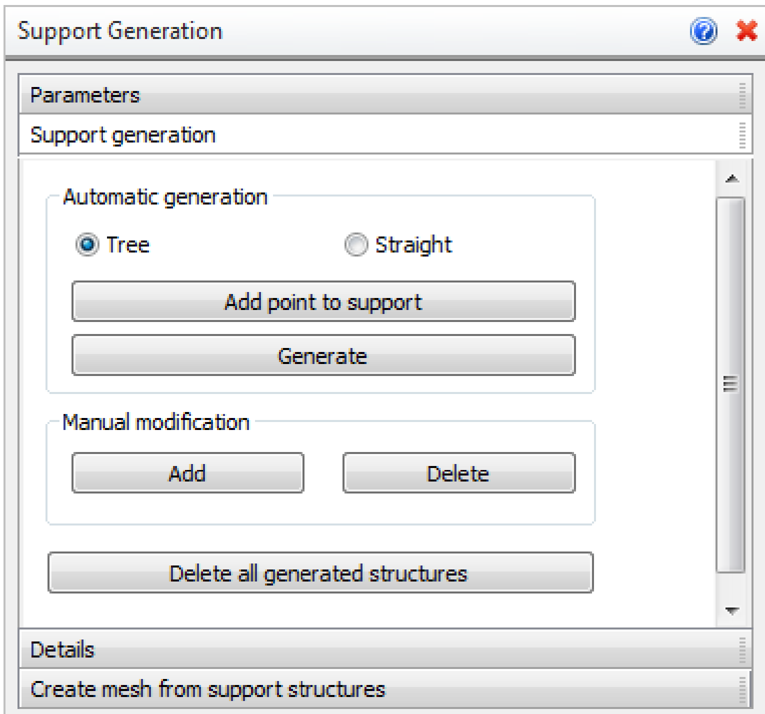
Refer to the image to the right:

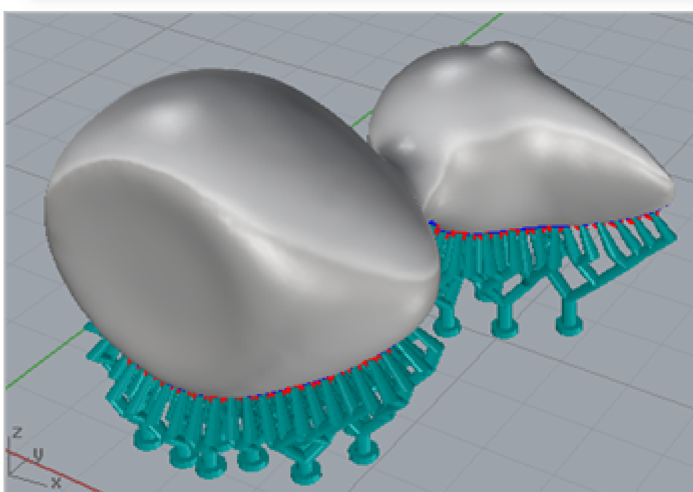


Overhang angle: 45
Minimum Overhang Height: 0.05
Max angle: 45
Min Overhang Height: 0.2
Support diameter: 0.2
Sampling distance: 0.25
Connector diameter: 0.1
Connector height: 0.5
Base diameter: 0.375
Base height: 0.1
Base connectors on mesh: Minimize

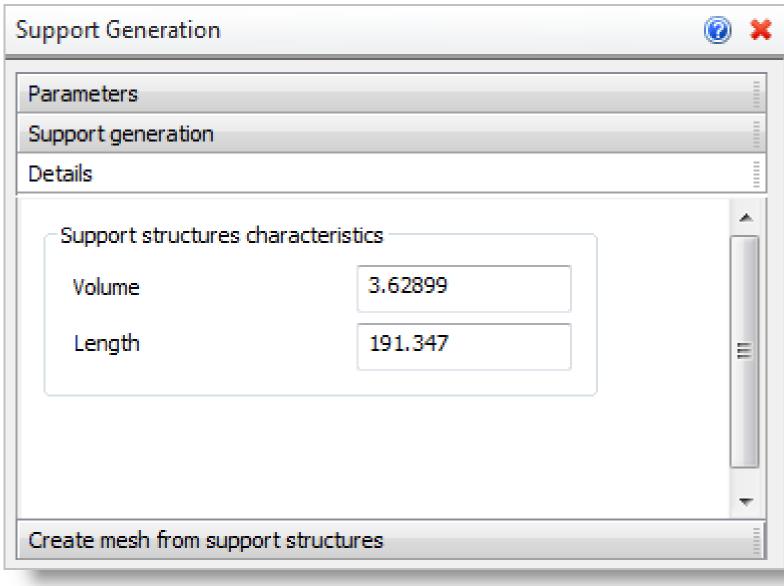
5. Now select the Support generation tab. Under the Automatic generation section, select Tree and then pick Generate.

   The support structure is calculated and displayed on the part. As you see in the image below, the structure is *grown* from the base of the 3D printer volume up to the part. The Overhand angle and Sampling distance will determine how many Connectors are generated.

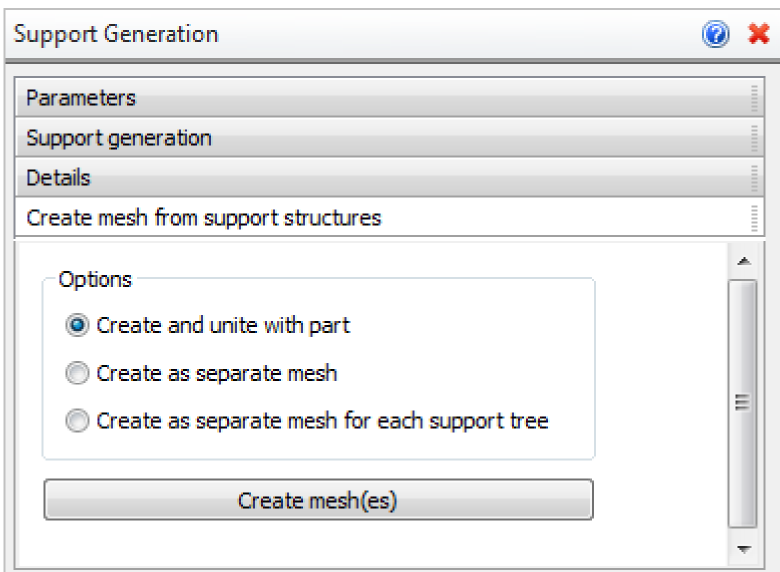




6. Now select the Details tab to calculate and review characteristics about the support structure.



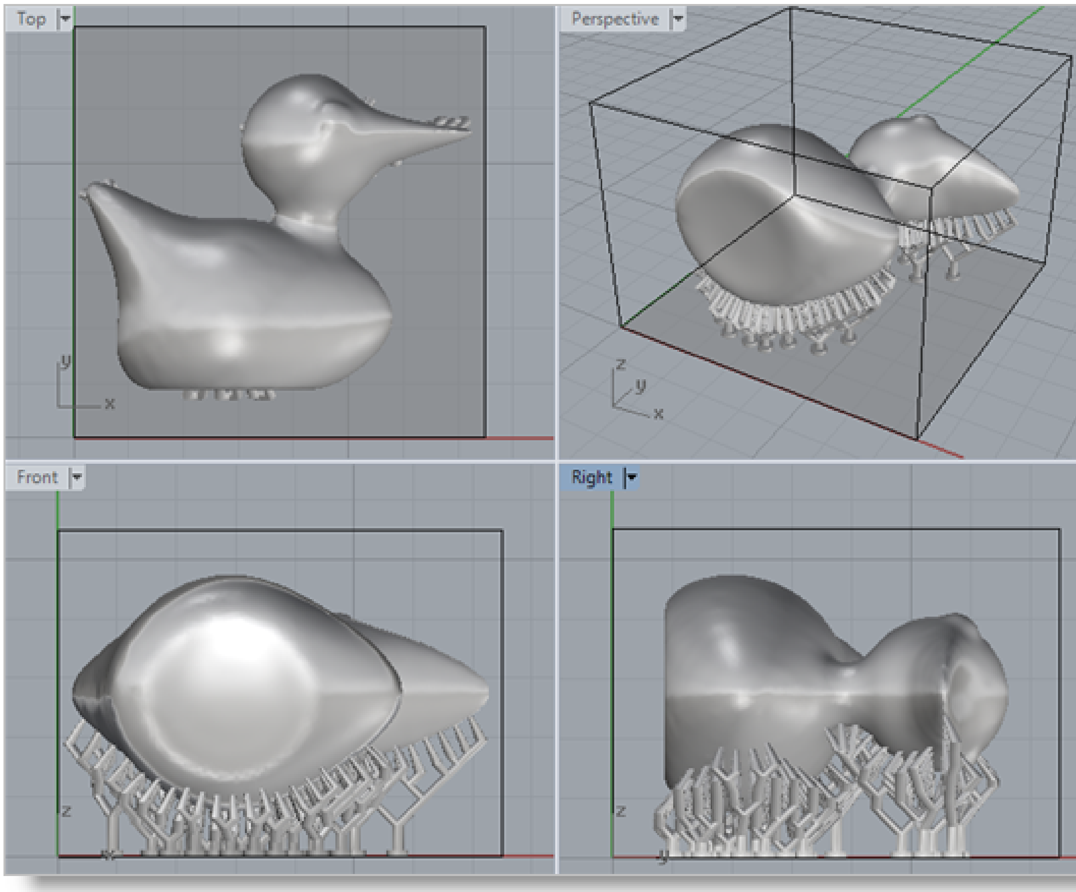
7. Now select the Create mesh from support structures tab to see additional options before creating the support structure mesh. Here we will select Create and unite with part and then pick the Create mesh(es) button.



Now pick the Cancel icon  to close the Support generation dialog

8. Now let's view the view the support structure.

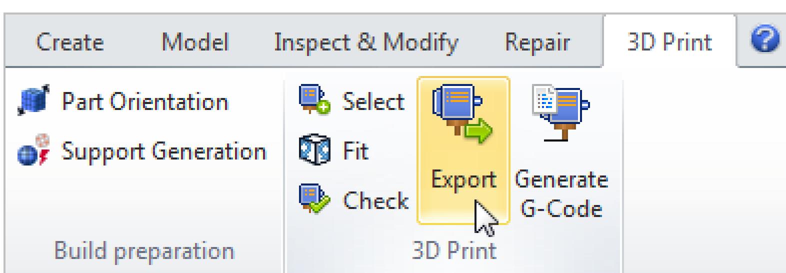
First, pick  from the Rhino3DPRINT 2017 Browser toolbar again to show the 3D printer volume. Then changed to the Quad Viewport display. Your part should look like this. Notice the organic support structure created between your part and the 3D Printer's bed:



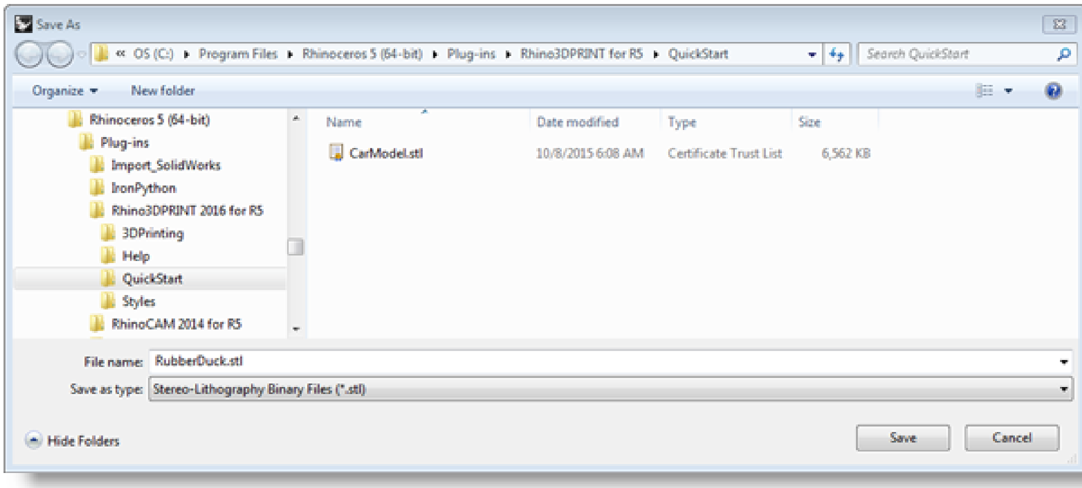
3.14 Export

Now, we're ready to create an STL file that you can send your 3D Printer or service.

1. Select the mesh model if its not already selected.
2. From the 3D Print tab, pick Export.



3. From the Save As dialog, select a folder to save the file to, set the Save as Type to: *Stereo-Lithography Binary (*.stl) files* and then pick Save.

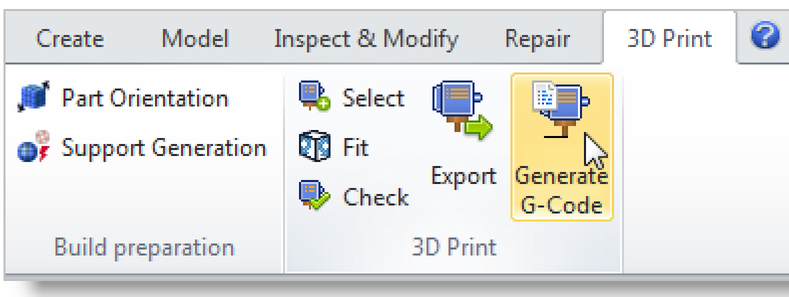


This file can now be sent to your 3D printer or service center for printing!

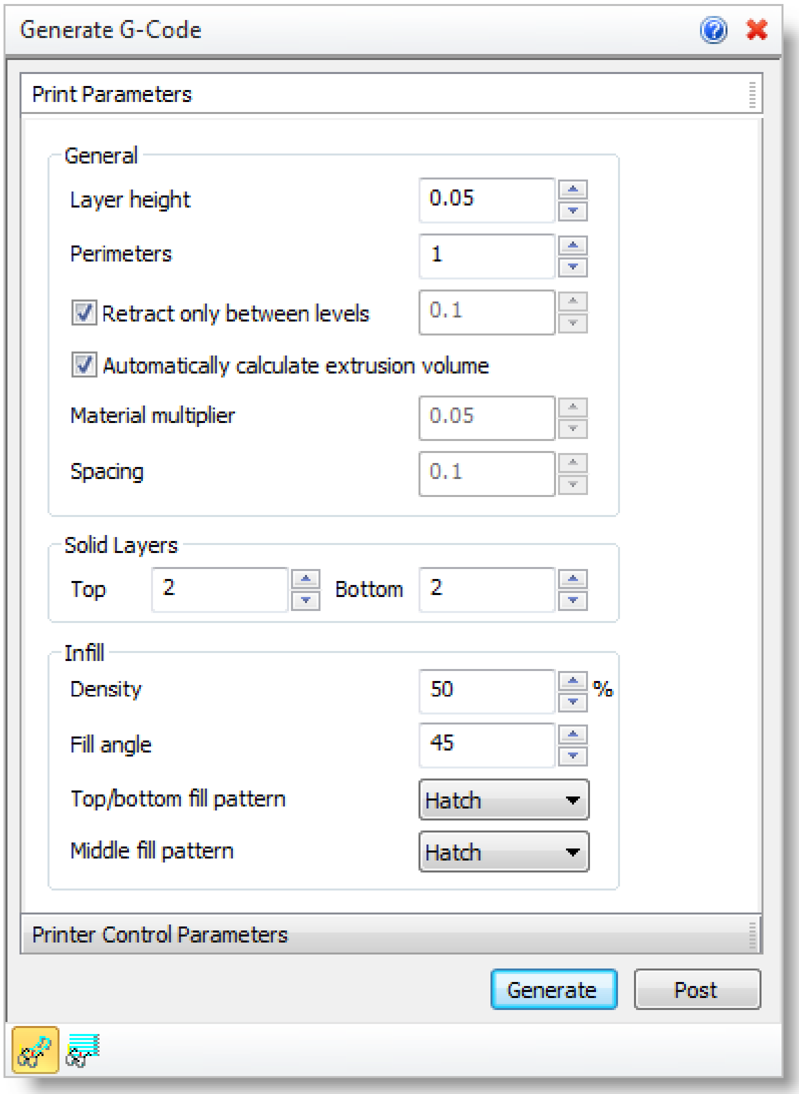
3.15 G-Code

Optionally, you can Generate G-Code that is compatible with some printers.

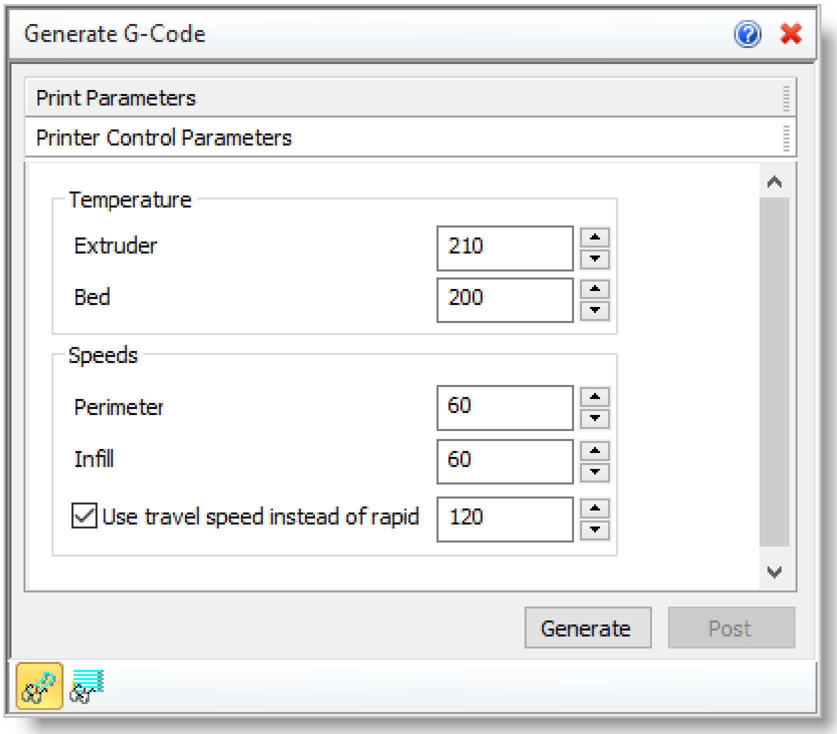
1. From the 3D Print tab, pick Generate G-Code.



2. Select both meshes (the part mesh and the support structure mesh) and then press Enter. The Generate G-Code dialog will display. Set your parameters to those shown in the dialog below.



3. Now select the Printer Control Parameters tab. Set your parameters to those shown in the dialog below.



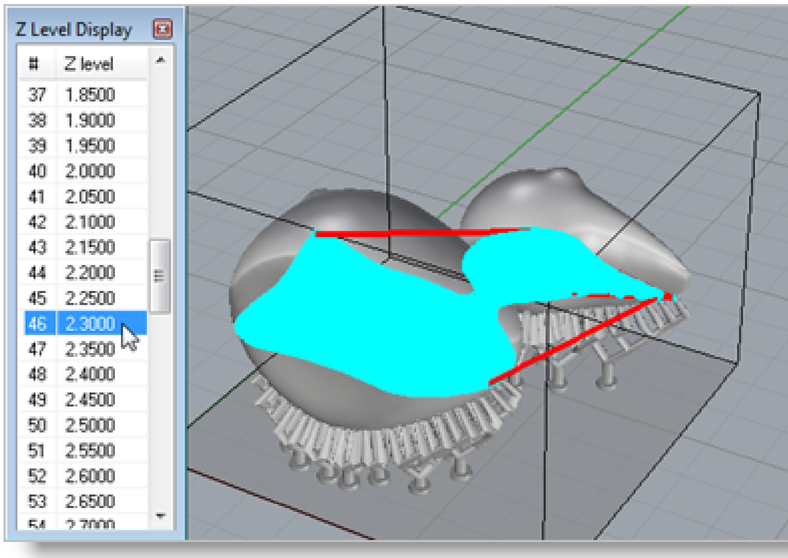
Printer Control Parameters tab

Check your 3D Printed before Generating G-Code: Before generating G-Code it is always a good practice to check to make sure you have the correct 3D Printer selected and that the Nozzle Diameter is set correctly to match the nozzle currently installed on your printer. See Select Printer for more information.

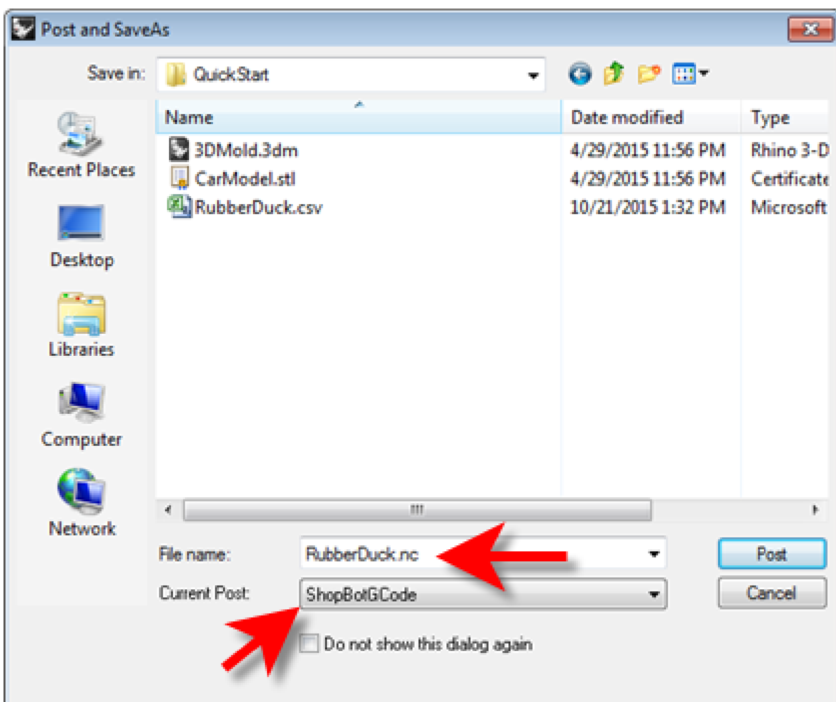
4. Now pick the Generate button. The G-Code is calculated and displayed on the part. Once completed you can use the following icons located at the bottom of the dialog to view your 3D Print paths and levels.

 Toggle Path Visibility

 Display Path Levels



5. When you are satisfied with your 3D Printer paths, select the Post button to create your G-Code file.
6. From the Post and Save As dialog select the following:

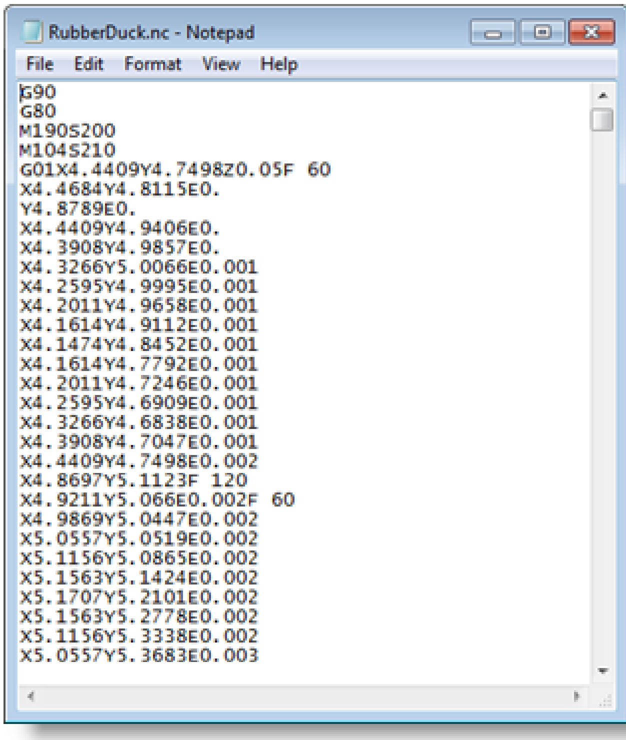


Save in: The program defaults to the same folder where your part is located.

File name: The program defaults to the name of the current file. Then enter *.nc* for the file extension for the G-Code file (ex: *RubberDuck.nc*). Note that in the future you can change the file extension to match what your printer supports (i.e., **.txt*, **.nc*, etc.).

Current Post: Select ShopBotGCode for the post processor.

7. Now select Post to generate and save the G-Code file. It will be displayed in Notepad.




```
G90
G80
M190S200
M104S210
G01X4.4409Y4.7498Z0.05F 60
X4.4684Y4.8115E0.
Y4.8789E0.
X4.4409Y4.9406E0.
X4.3908Y4.9857E0.
X4.3266Y5.0066E0.001
X4.2595Y4.9995E0.001
X4.2011Y4.9658E0.001
X4.1614Y4.9112E0.001
X4.1474Y4.8452E0.001
X4.1614Y4.7792E0.001
X4.2011Y4.7246E0.001
X4.2595Y4.6909E0.001
X4.3266Y4.6838E0.001
X4.3908Y4.7047E0.001
X4.4409Y4.7498E0.002
X4.8697Y5.1123F 120
X4.9211Y5.066E0.002F 60
X4.9869Y5.0447E0.002
X5.0557Y5.0519E0.002
X5.1156Y5.0865E0.002
X5.1563Y5.1424E0.002
X5.1707Y5.2101E0.002
X5.1563Y5.2778E0.002
X5.1156Y5.3338E0.002
X5.0557Y5.3683E0.003
```

8. If the G-Code text file does not display automatically then your Notepad is not recognizing the *.nc text file extension. Select Post again from the Generate G-Code dialog to display the Post and Save As dialog again. Right-click on your G-Code file and select Open. Windows will ask you what program to use for opening *.nc files.

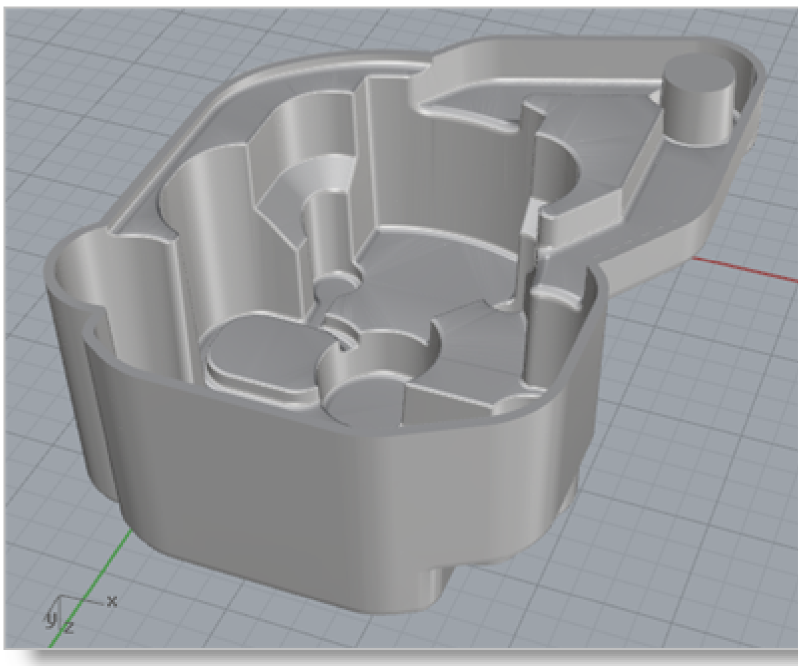
This file can then be sent you your 3D Printer for printing.
This completes Chapter 3 of the Quick Start Guide for Rhino3DPRINT 2017.
For further assistance you can visit the Online Help supplied with the program or visit www.mecsoft.com for additional tutorials.

Chapter 4: Offset & Split

In this chapter we will use the Offset and Split tools to create a part with a uniform wall thickness.

We will perform the following basic steps to complete this chapter:

1. Load the part file.
2. Diagnose the mesh for possible problems.
3. Use Stitch & Close to combine and fix the mesh.
4. Use Offset to create a uniform wall thickness.
5. Use Split and review the final mesh.
6. Our completed part will look like this:

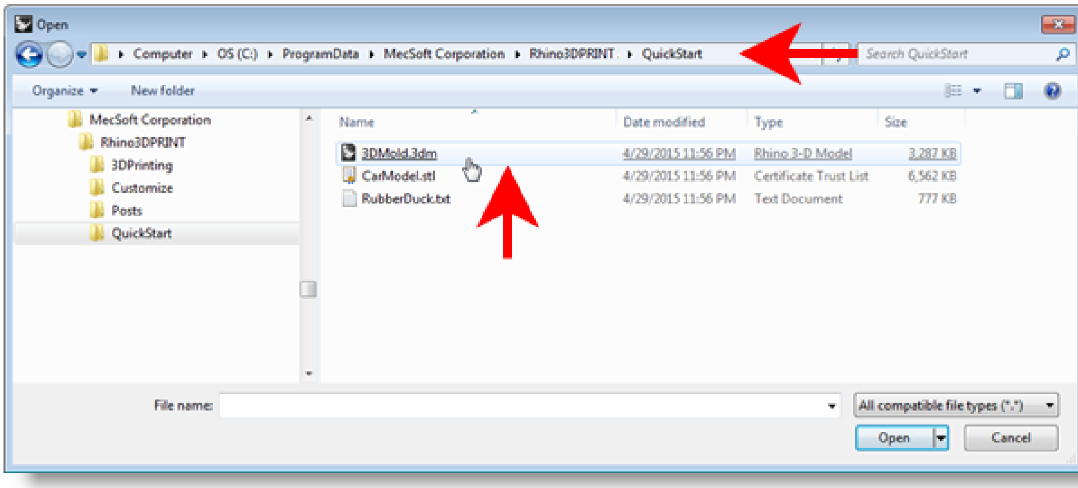


4.1 Load the Part File

Now, let's load the part file for this tutorial.

1. From the Rhino Standard toolbar, select the Folder icon.

2. Find the part file named *3DMold.3dm* located in the QuickStart folder and then pick Open.

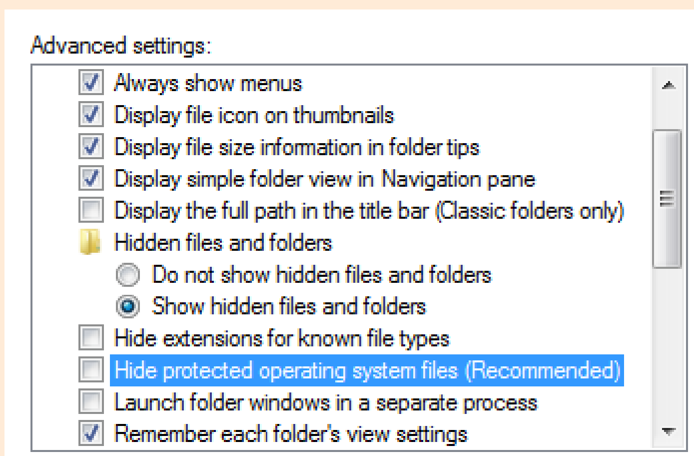


By default, the ProgramData folder is "hidden" from view. Here are the steps to Show hidden files and folders:

1. For Windows7/8/10 users: Go to Control Panel > Appearance and Personalization > Folder Options (Windows XP users can locate folder options under Control Panel).

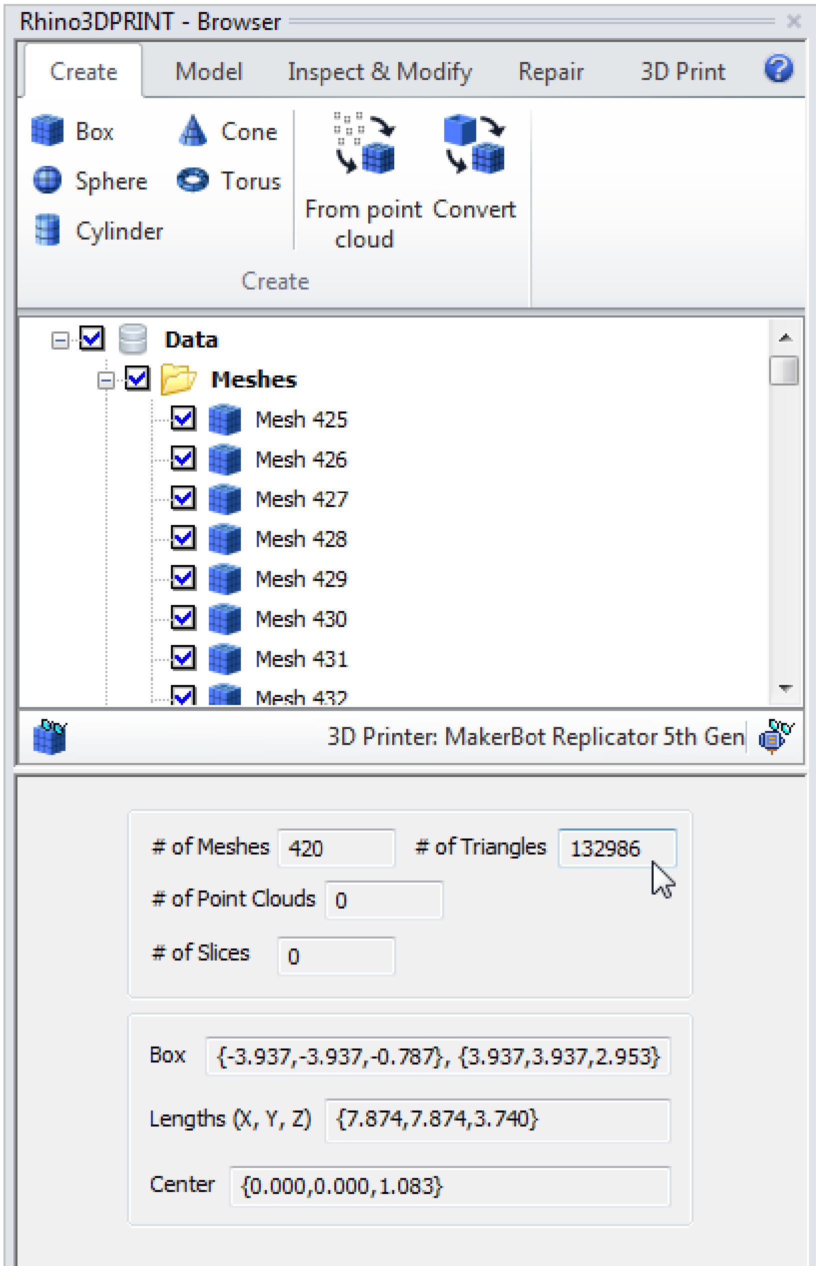
2. Select View tab and under advanced settings select Show Hidden files and folders, clear the check boxes for:

   - Hide extensions for known file types
   - Hide protected operating system files (Recommended)

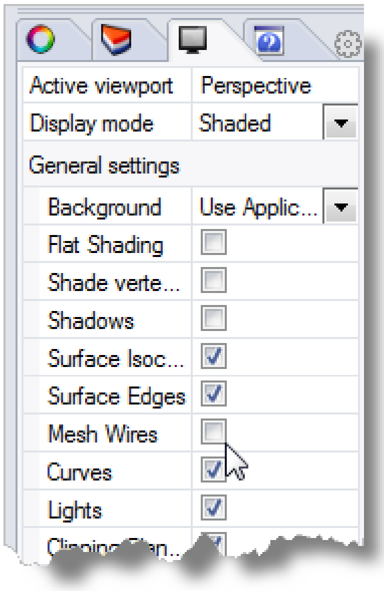


3. Click Apply and OK.

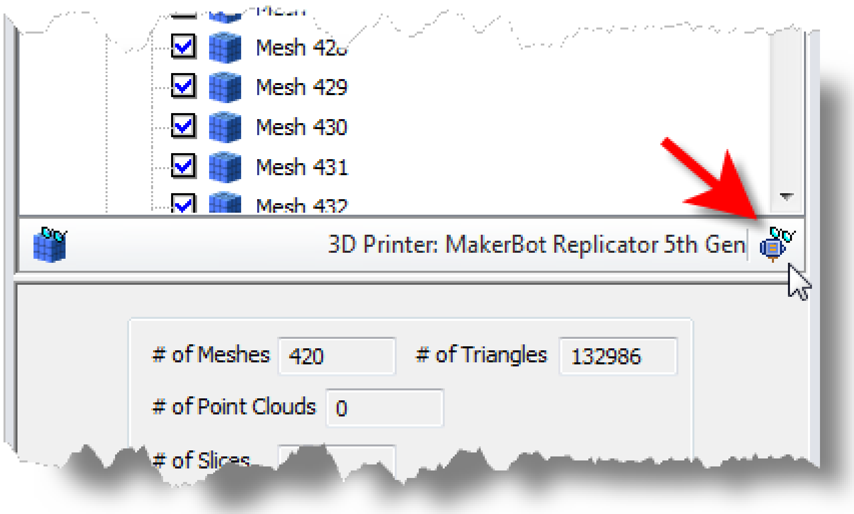
3. From the Rhino3DPRINT 2017 Browser we see that this model contains over 400 individual mesh objects and over 130,000 facet triangles.



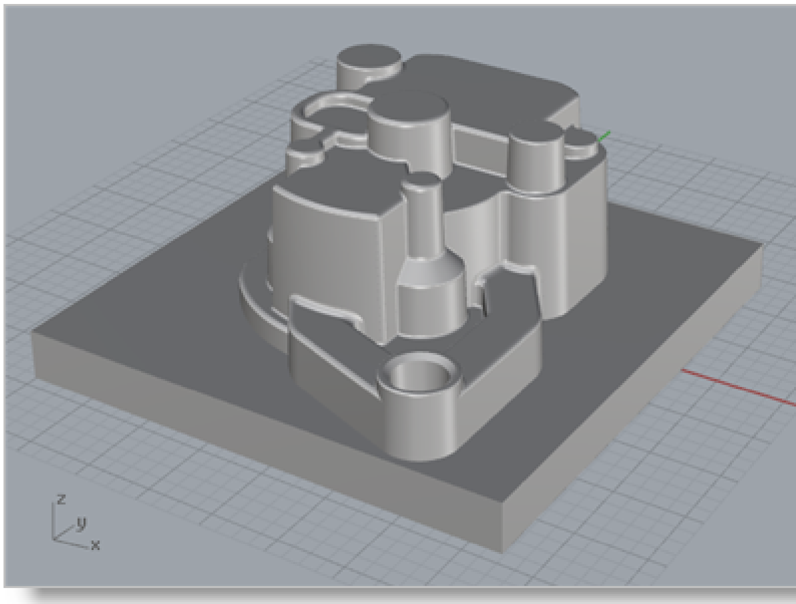
4. From the Rhino Display Properties tab, uncheck Mesh Wires so we can see the model more clearly.



5. From the Rhino3DPRINT 2017 Browser toolbar, select the icon for Toggle Printer Volume Visibility to make sure the display of the printer volume is turned off.



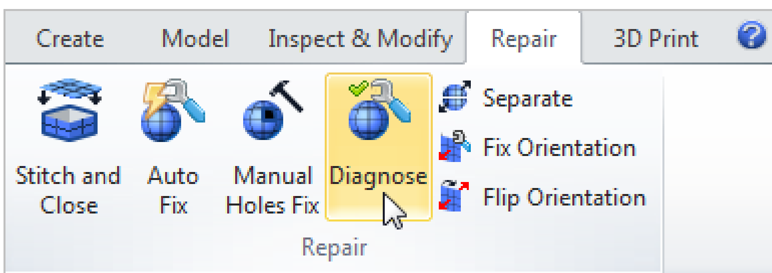
Your display should now look like this:



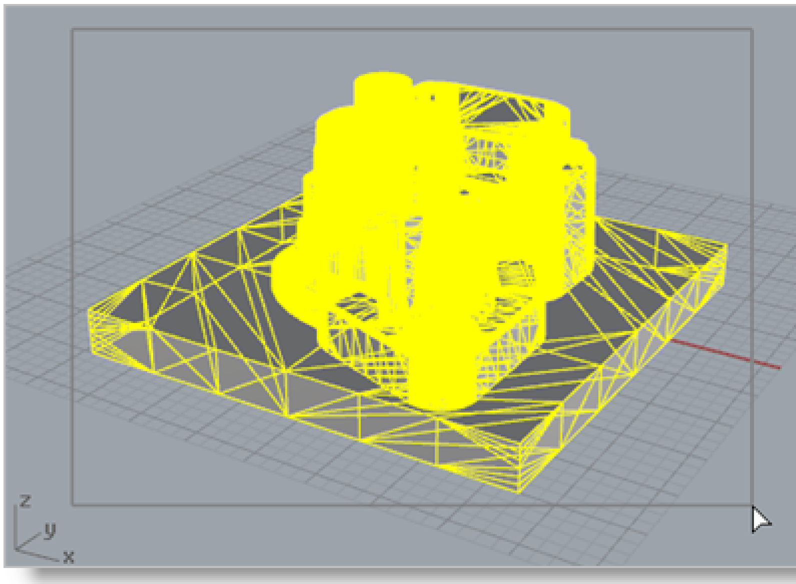
4.2 Diagnose

The first thing we want to do is perform some diagnostics.

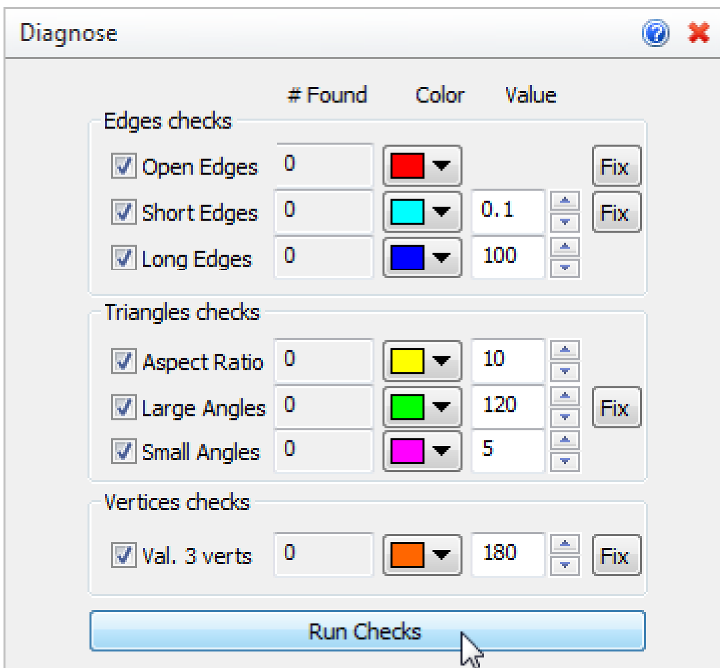
1. From Rhino3DPRINT 2017 Browser select the Repair tab and then select Diagnose.



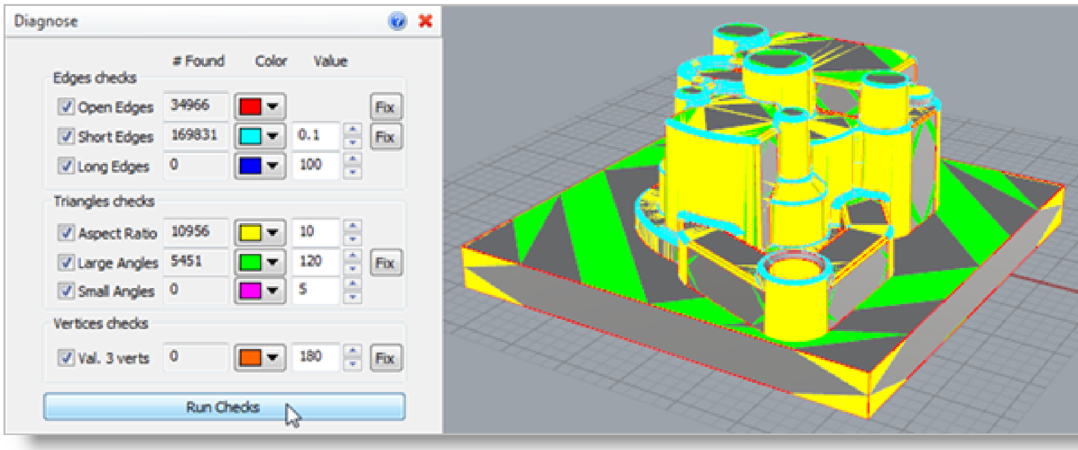
2. Now window select the entire mesh model and then press Enter.



3. The Diagnose dialog will appear below the Browser.



4. Using the default selections, pick the Run Checks button.
5. We see that three sets of checks are performed. These include Edges, Triangles and Vertices.
6. The results of each check are color coded for easy recognition on the mesh model.

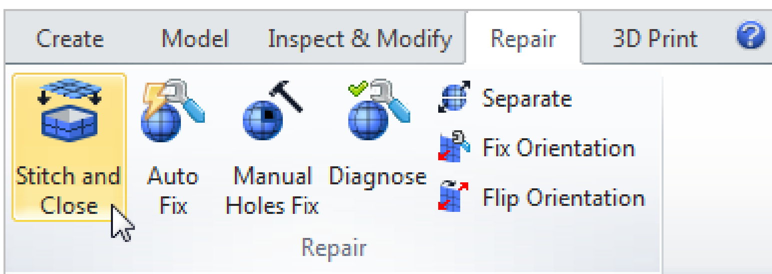


7. Open Edges are the most important check as these will cause the 3D print to fail. We see that Diagnose found over 30,000 open edges which are assigned the color **red** by default.
8. For now, we'll pick the Cancel icon ✖ to close the Diagnose dialog.

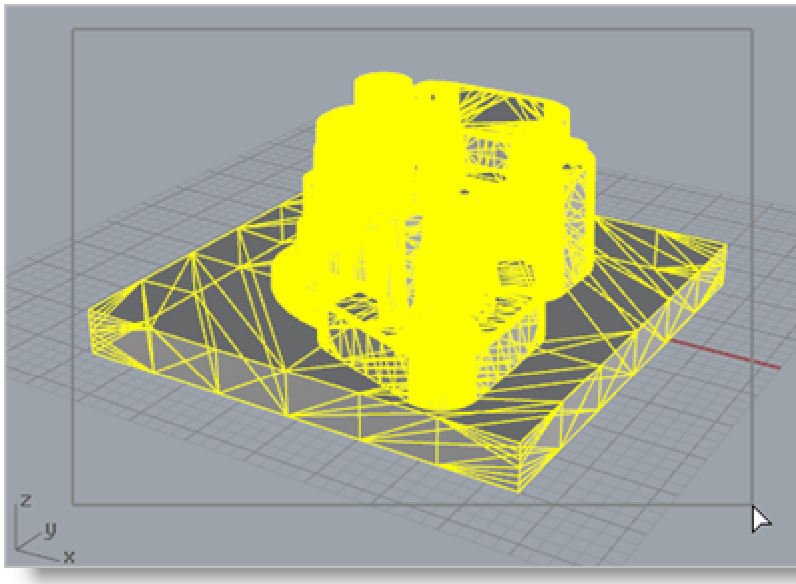
4.3 Stitch & Close

Rhino3DPRINT 2017 allows you to fix the open edges automatically.

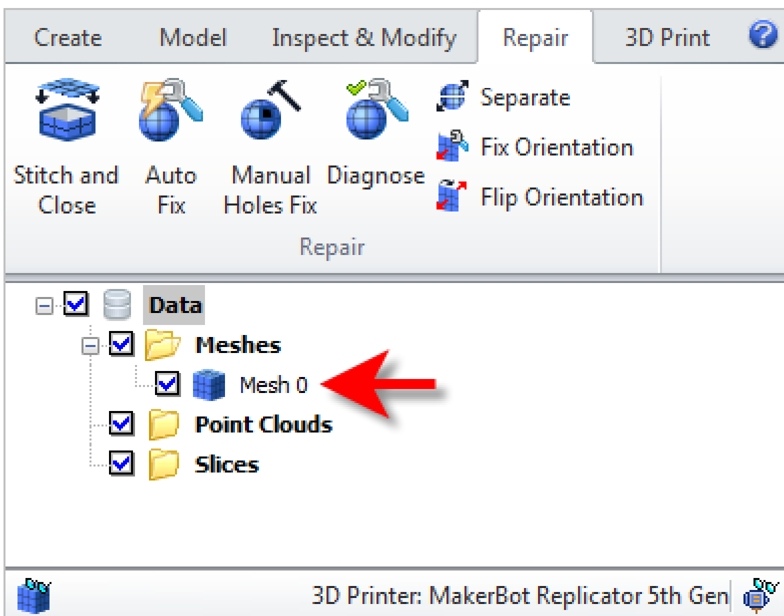
1. From Rhino3DPRINT 2017 Browser select the Repair tab and then select Stitch and Close.



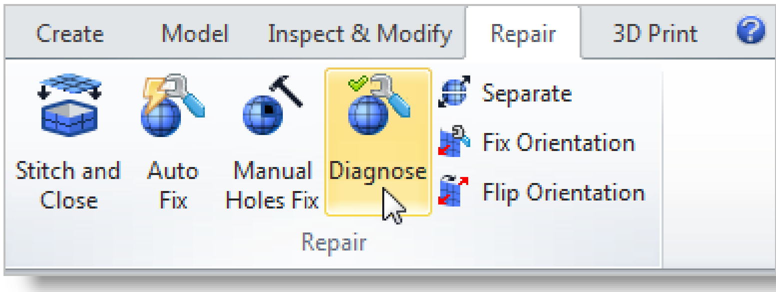
2. Now window select the entire mesh model again and, press Enter. The mesh is repaired automatically.



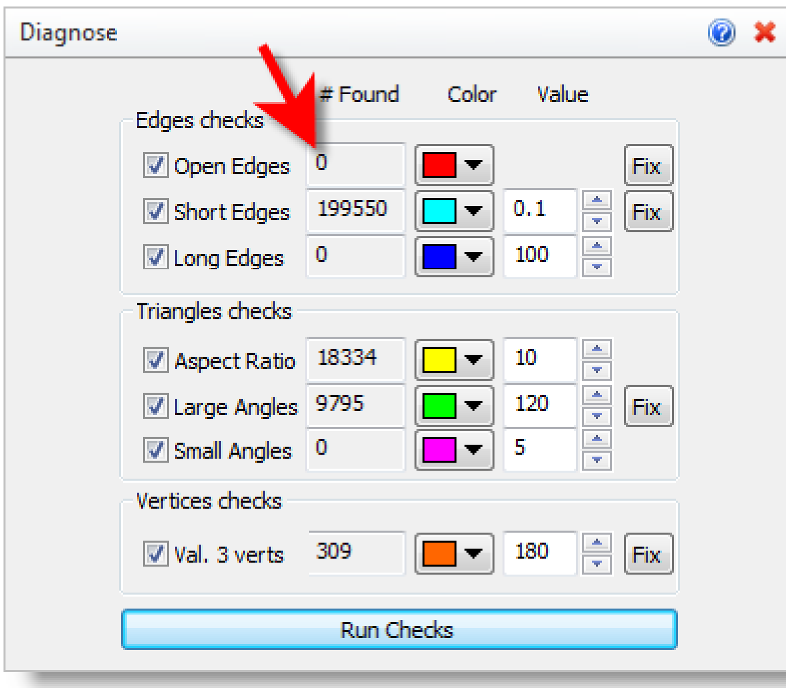
3. Notice that in the Browser there is now only one mesh in the Data Tree.



4. From the Repair tab, we'll select Diagnose again. Window select the entire mesh model again, press Enter and then pick Run Checks to see the new results.



5. We see that we now have zero open edges!!

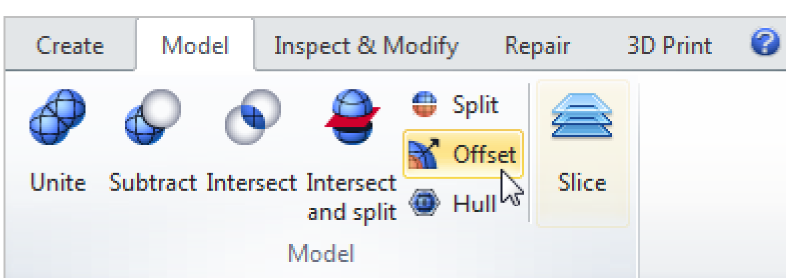


6. Pick the Cancel icon ✖ to close the Diagnose dialog.

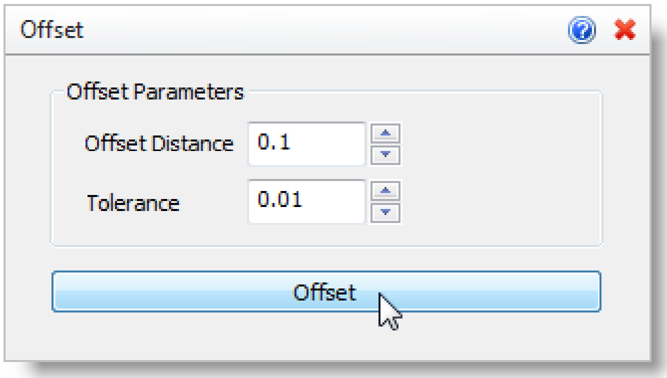
4.4 Offset

Now that we have only one mesh, let's Offset to create a wall thickness.

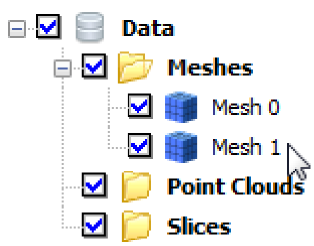
1. From the Rhino3DPRINT 2017 Browser select the Model tab and then select Offset.



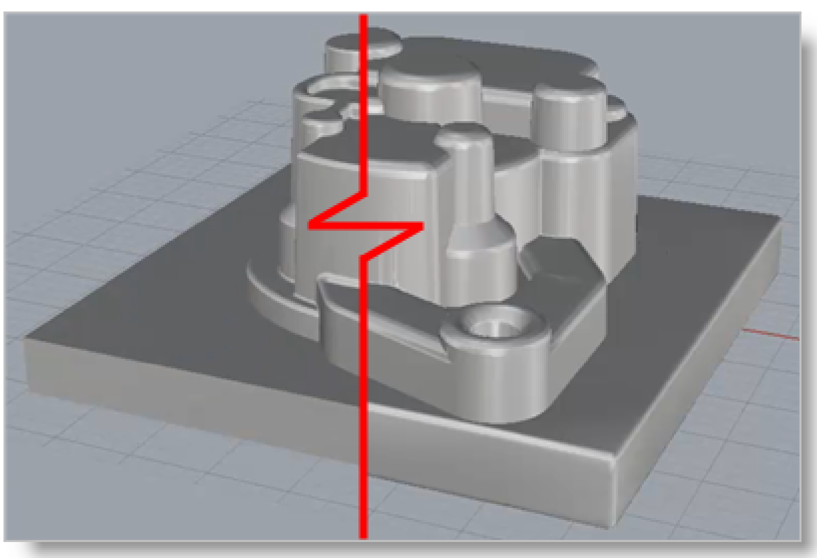
2. Select the mesh and the Offset dialog will display.



3. We'll set the Offset Distance to 0.1
4. Then we'll set the Tolerance to 0.01 and then pick Offset.
5. Now we see that a second mesh was added to the Data Tree that is a positive offset of the first.



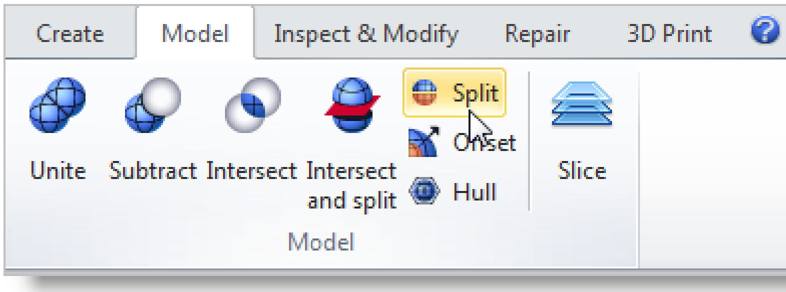
6. We can see the difference clearly on the screen by unchecking and checking each mesh individually.



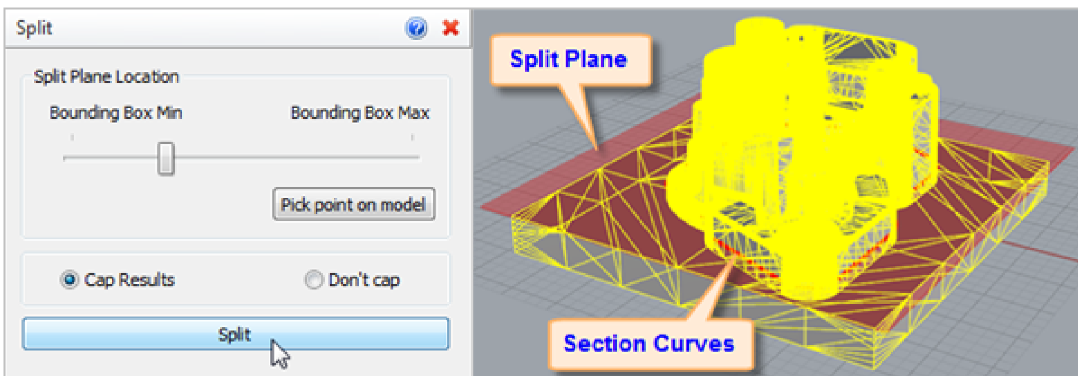
4.5 Split & Cap

Now that we have a uniform wall thickness, let's Split the mesh.

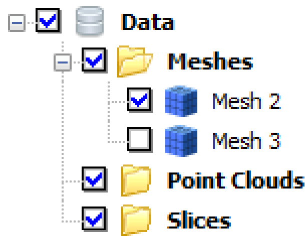
1. We'll go back to the Model tab and select Split.



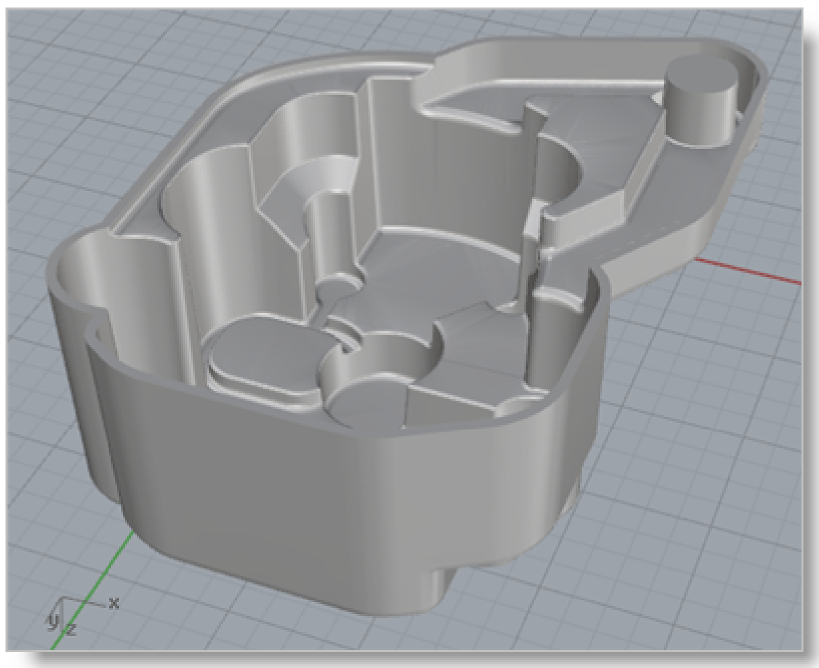
2. Window select **both meshes** or select them both from the Rhino3DPRINT 2017 Browser Data Tree and then press Enter to display the Split dialog.
3. The upper portion of the dialog contains a slider that allows you to control the Split Plane Location.
4. Move the Split Plane, shown graphically on the screen, by adjusting the slider between the Min and Max of the mesh's Bounding Box.



5. Section curves shown in **Red**, and also displayed dynamically, update as you adjust the slider. Move the slider until the section curves are located approximately as shown in the example above.
6. We are provided with a Cap Results option. Selecting this will ensure that once split, each section of the mesh will be a capped closed volume.
7. Now we pick Split.
8. Once completed, we'll uncheck all but the upper mesh from the Data Tree.



9. Now in Rhino, right-click in the display area, hold and drag to rotate the upper mesh so that the cavity and offset wall thickness can be clearly seen. Mesh offsets can be used in a variety of applications.



This completes Chapter 4 of the Quick Start Guide for Rhino3DPRINT 2017.

For further assistance you can visit the Online Help supplied with the program or visit www.mecsoft.com for additional tutorials.

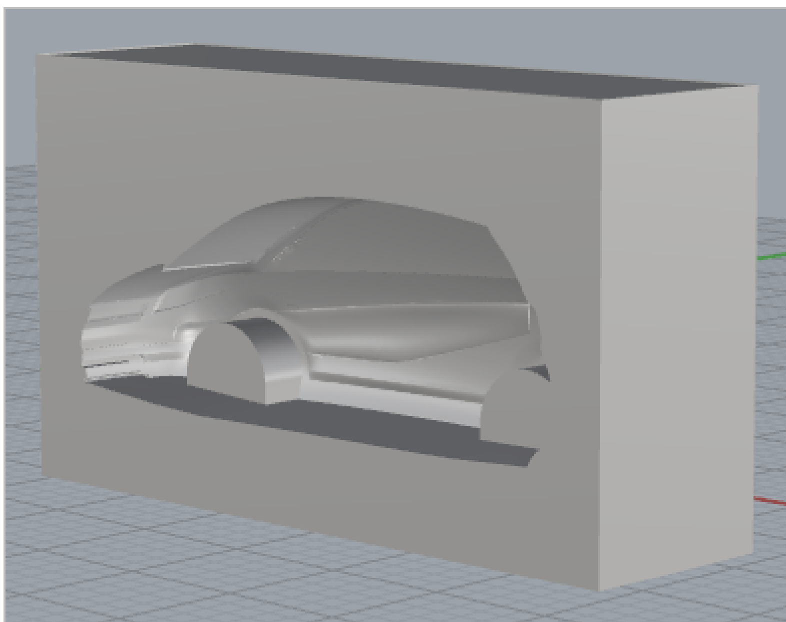
Chapter 5: Cavity Block

In this chapter we'll be working with an STL file to create a core and cavity block.

We will perform the following basic steps to complete this chapter:

1. Open the STL part file.
2. Diagnose the mesh for possible problems.
3. Use Stitch & Close to combine and fix the mesh.
4. Use Reduce to simplify the mesh.
5. Create a Box for the care and cavity.
6. Subtract the original mesh from the cavity block.

   Our completed part will look like this:



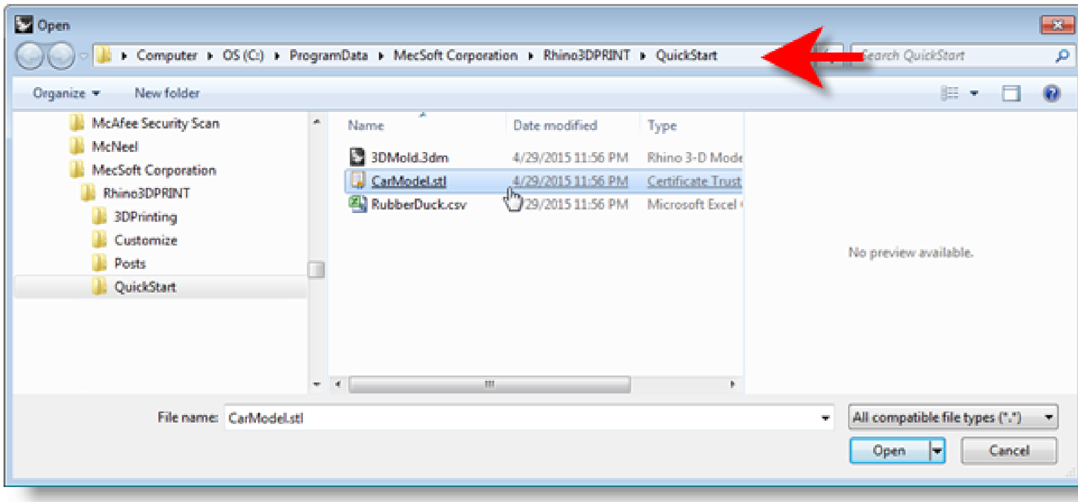
5.1 Load the Part File

Now, let's load the part file for this tutorial.

1. From the Rhino Standard toolbar, select the Folder icon.

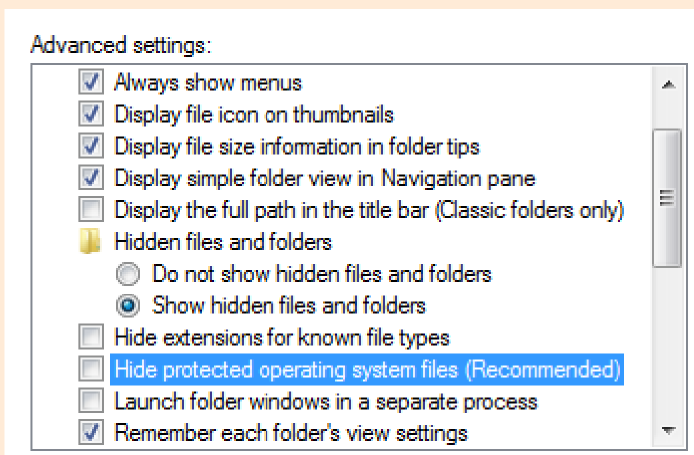
   

2. Find the part file named *CarModel.stl* located in the QuickStart folder and then pick Open.



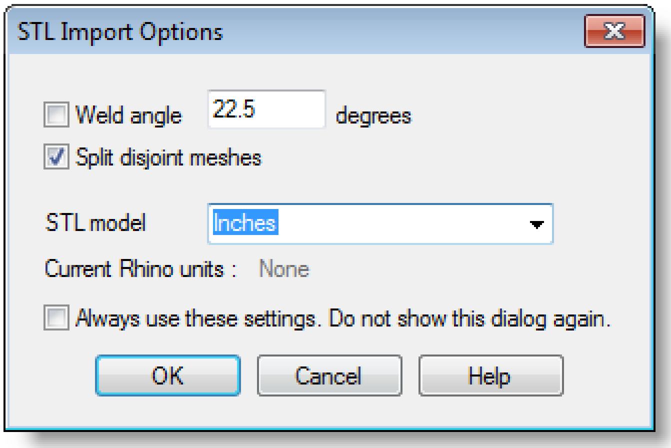
By default, the ProgramData folder is "hidden" from view. Here are the steps to Show hidden files and folders:

1. For Windows7/8/10 users: Go to Control Panel > Appearance and Personalization > Folder Options (Windows XP users can locate folder options under Control Panel).
2. Select View tab and under advanced settings select Show Hidden files and folders, clear the check boxes for:
   - Hide extensions for known file types
   - Hide protected operating system files (Recommended)

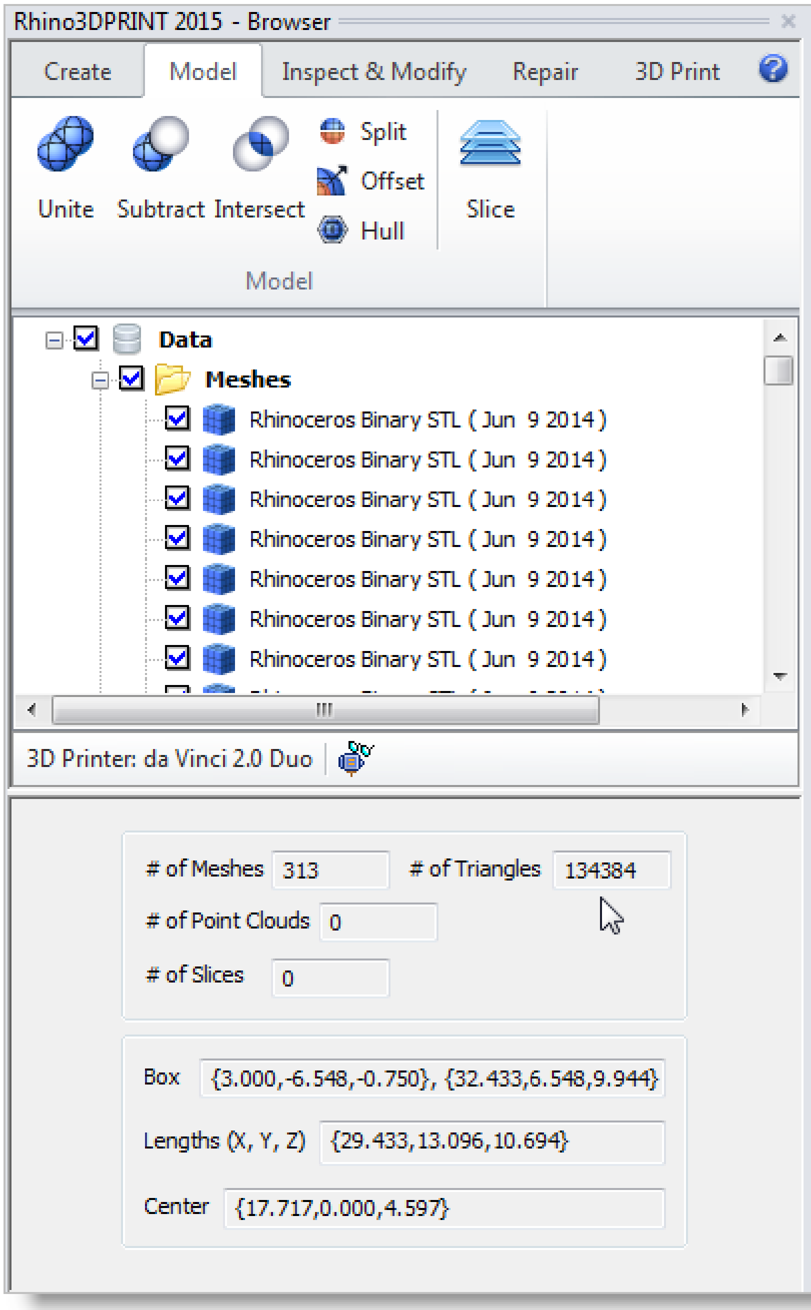


3. Click Apply and OK.

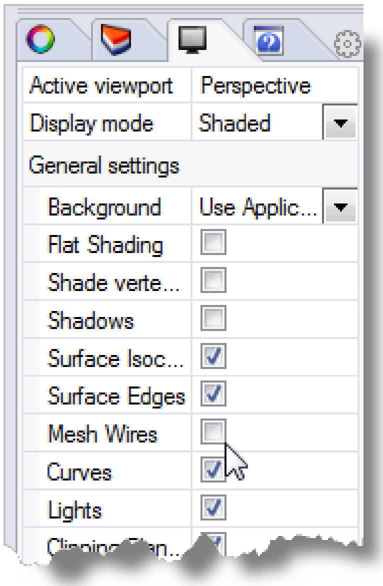
3. From the STL Import Options dialog we check the box for Split disjoint meshes, set the units to Inches and then pick OK.



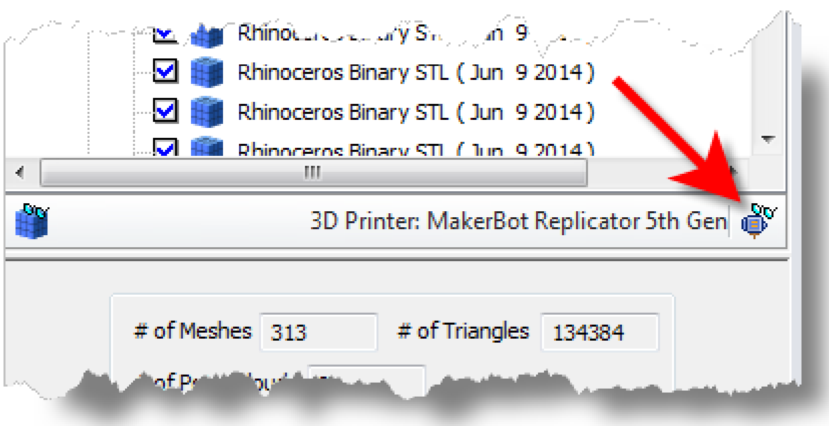
5. From the Rhino3DPRINT 2017 Browser we see that this model contains over 400 individual mesh objects and over 130,000 facet triangles.



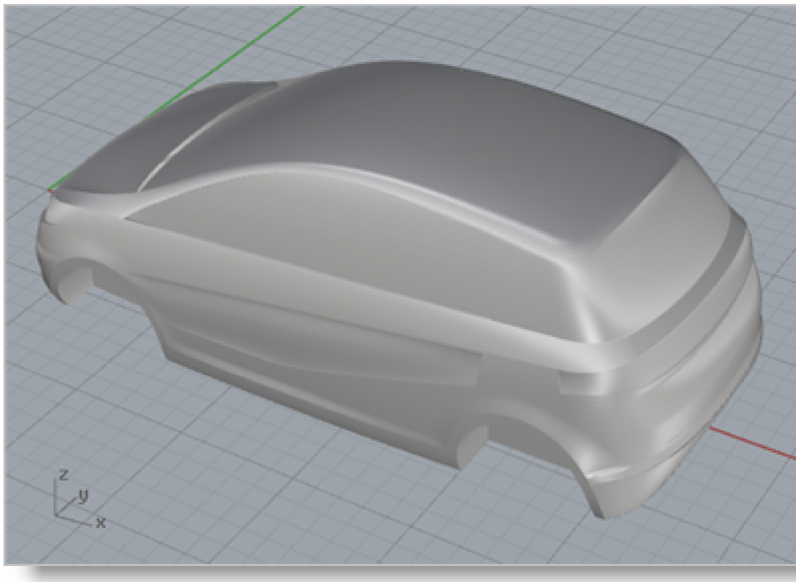
4. From the Rhino Standard toolbar, select Shaded Viewport  to change to a shaded display. Then from the Rhino Display Properties tab, uncheck Mesh Wires so we can see the model more clearly.



5. From the Browser toolbar, select the icon for Toggle Printer Volume Visibility to make sure the display of the printer volume is turned off.



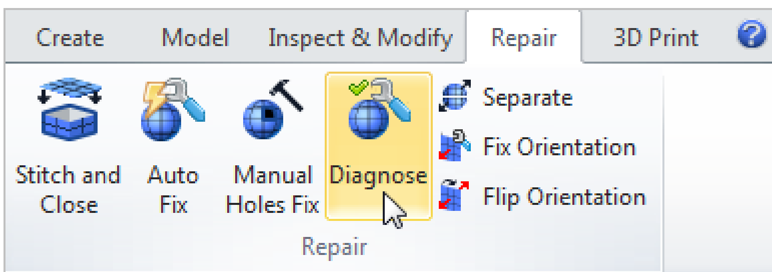
With Toggle Printer Volume Visibility turned off, the display looks like this:



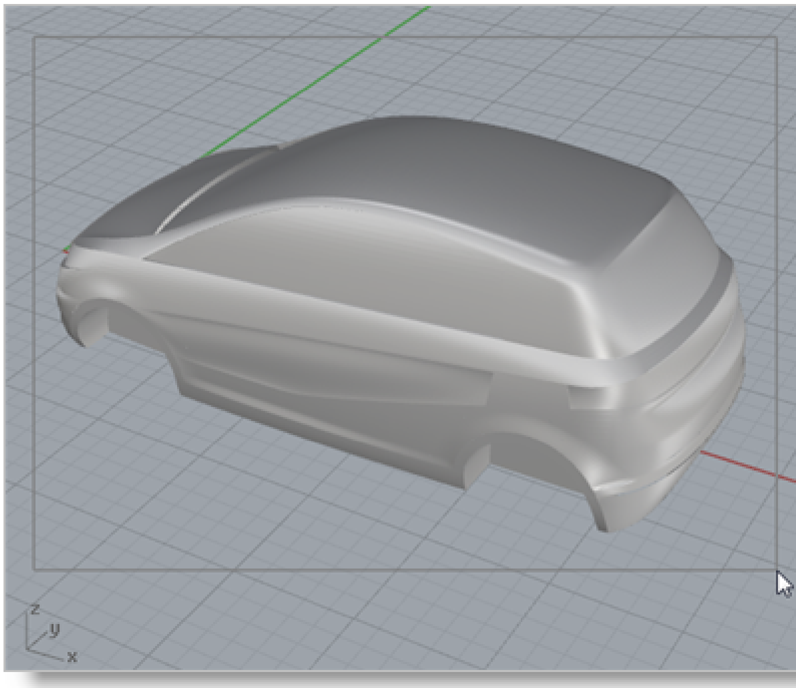
5.2 Diagnose

The first thing we want to do is perform some diagnostics.

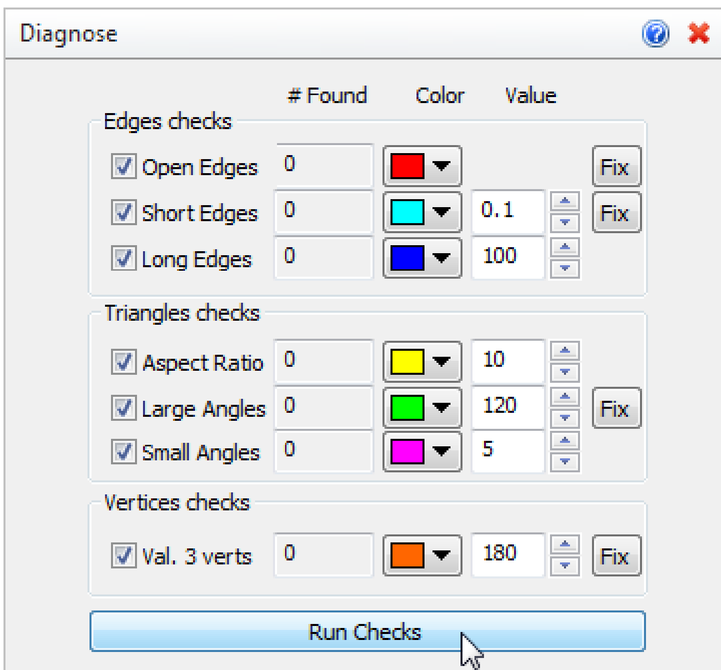
1. From Rhino3DPRINT 2017 Browser select the Repair tab and then select Diagnose.



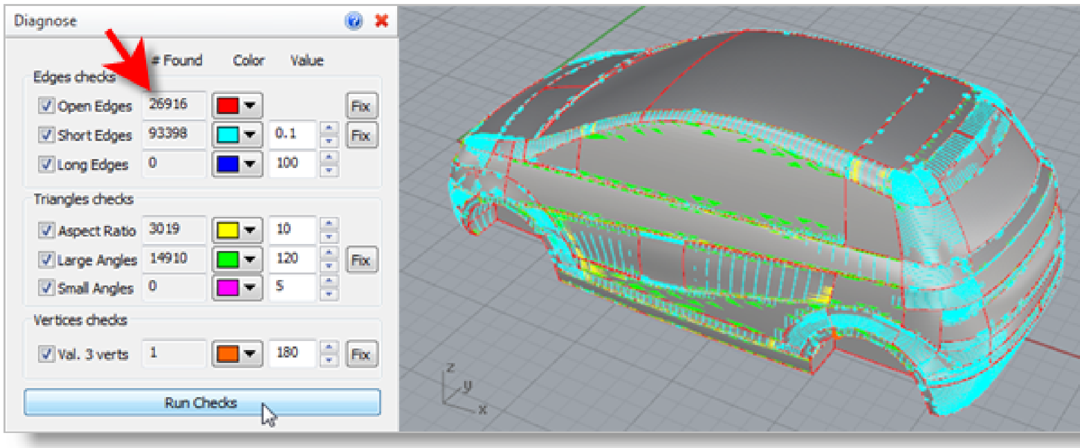
2. Now window select the entire mesh model and press Enter.



3. The Diagnose dialog will appear below the Browser.



4. Using the default selections, pick the Run Checks button.
5. We see that three sets of checks are performed. These include Edges, Triangles and Vertices.
6. The results of each check are color coded for easy recognition on the mesh model.

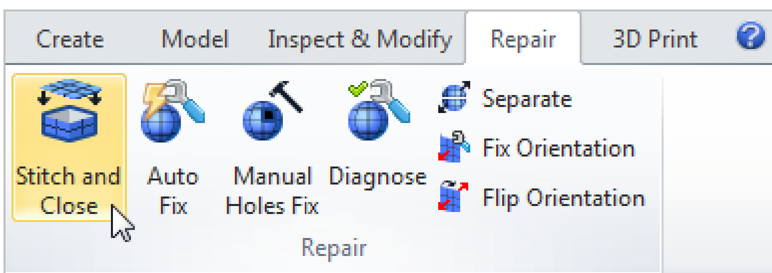


7. For example, Diagnose found over 25,000 open edges which are assigned the color red by default.
8. For now, we'll pick the Cancel icon ✖ to close the Diagnose dialog.

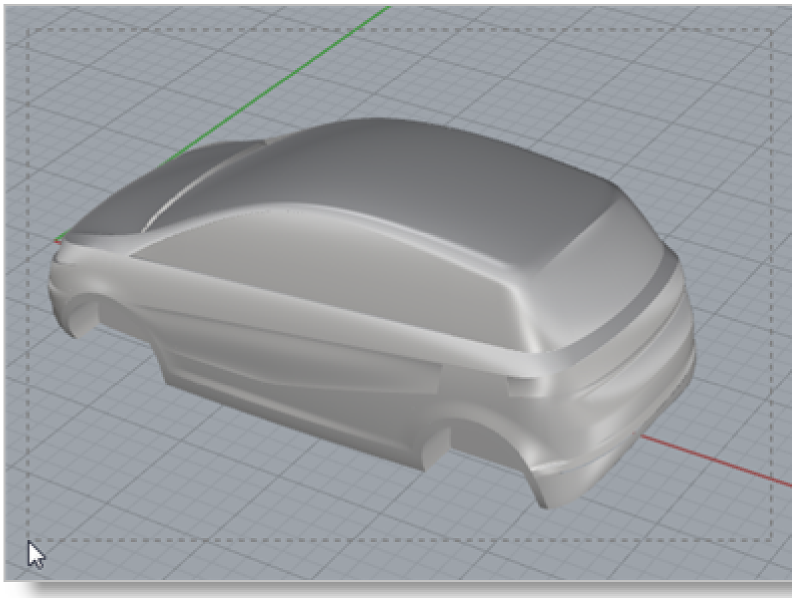
5.3 Stitch & Close

Rhino3DPRINT 2017 allows you to fix the open edges automatically.

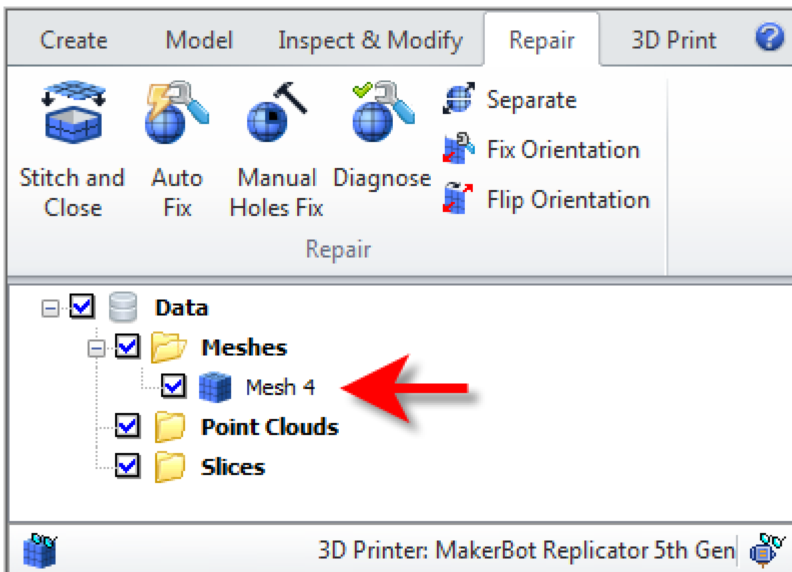
1. From Rhino3DPRINT 2017 Browser select the Repair tab and then select Stitch and Close.



2. Now window select the entire mesh model again and press Enter. The mesh is repaired automatically.

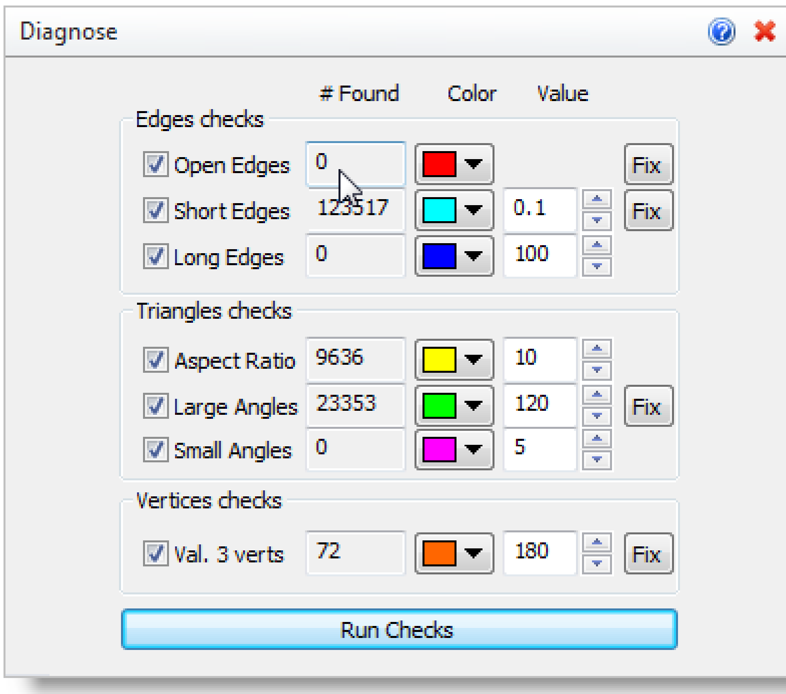


Notice in the Browser that there is now only one mesh in the Data Tree.



3. Now from Rhino3DPRINT 2017 Browser select the Repair tab and then select Diagnose again.

4. Window select the entire mesh model and then press Enter.
5. We see that we now have zero open edges!!

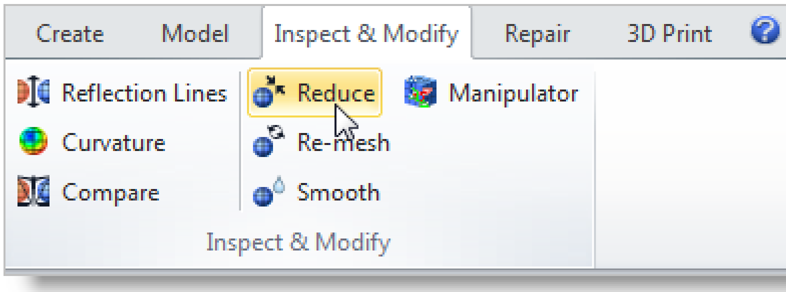


6. Pick the Cancel icon ✖ to close the Diagnose dialog.

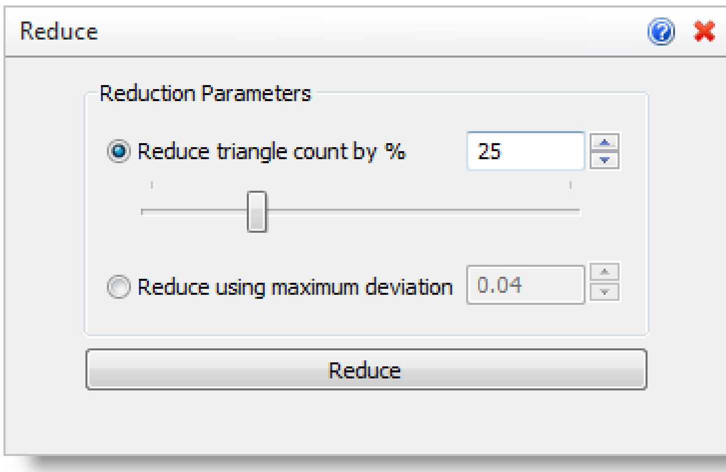
5.4 Reduce

Now that the mesh is fixed, let's simplify it some before moving on.

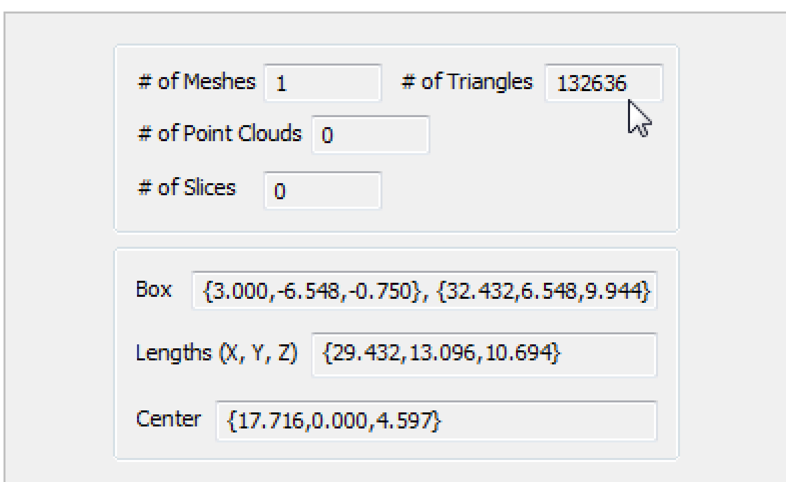
1. From Rhino3DPRINT 2017 Browser select the Inspect & Modify tab and then select Reduce.



2. Select the mesh and then press Enter to display the Reduce dialog.



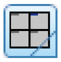
3. We'll adjust the slider to reduce the mesh by a percentage of the total facet triangle count. In this case we'll set it to 25% and then pick Reduce.
4. With the mesh selected in the Rhino3DPRINT 2017 Browser we now see that the total number of facet triangles is reduced.



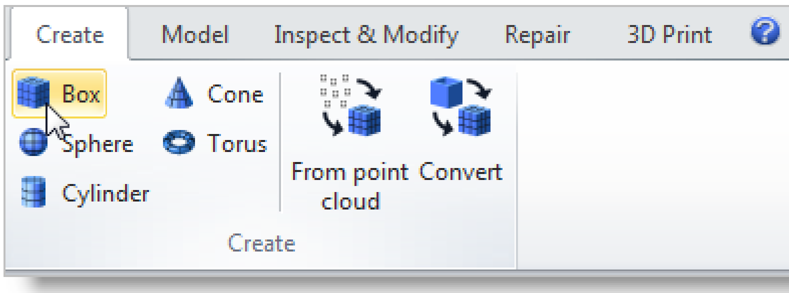
5.5 Create a Box

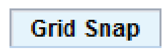
Next we'll create our cavity block by.

1. First we'll go to the Rhino Standard toolbar, select 4 Viewports.

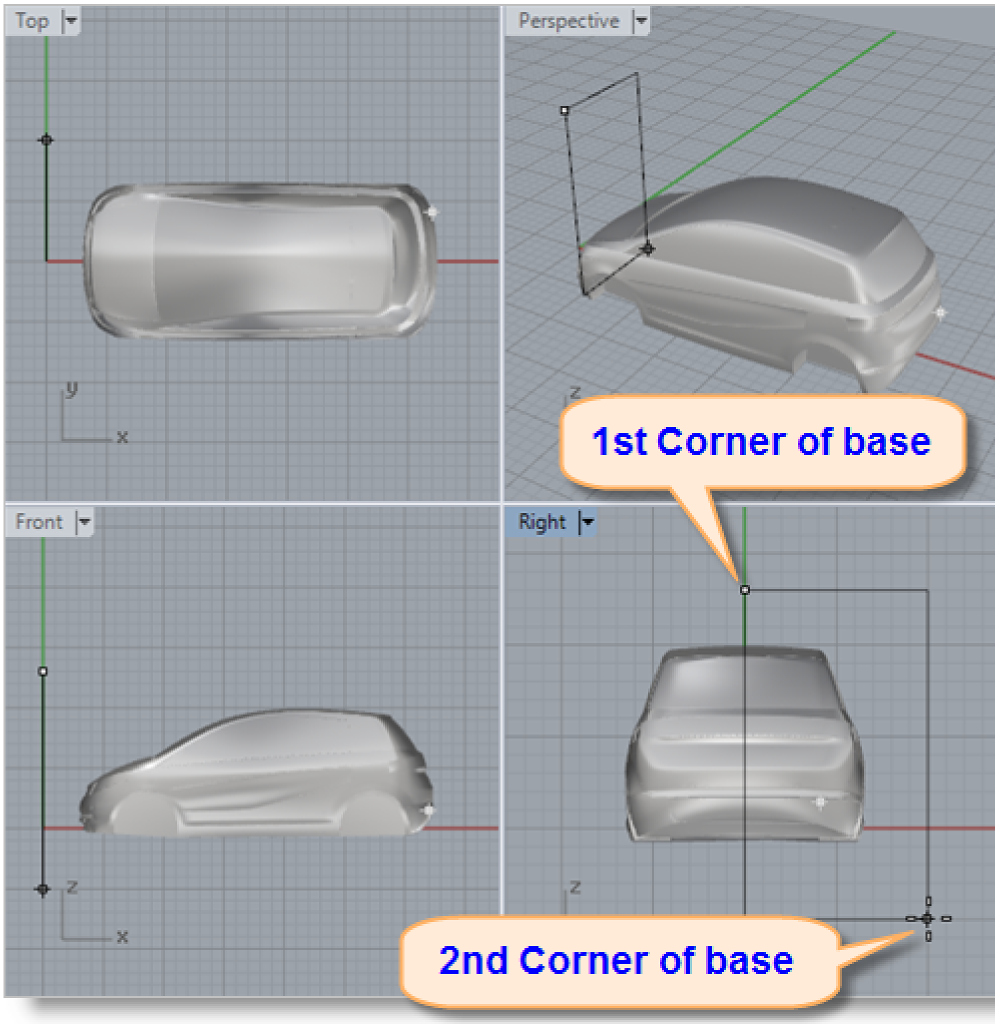


2. Then from the Create tab of the Rhino3DPRINT 2017 Browser, we select Box.



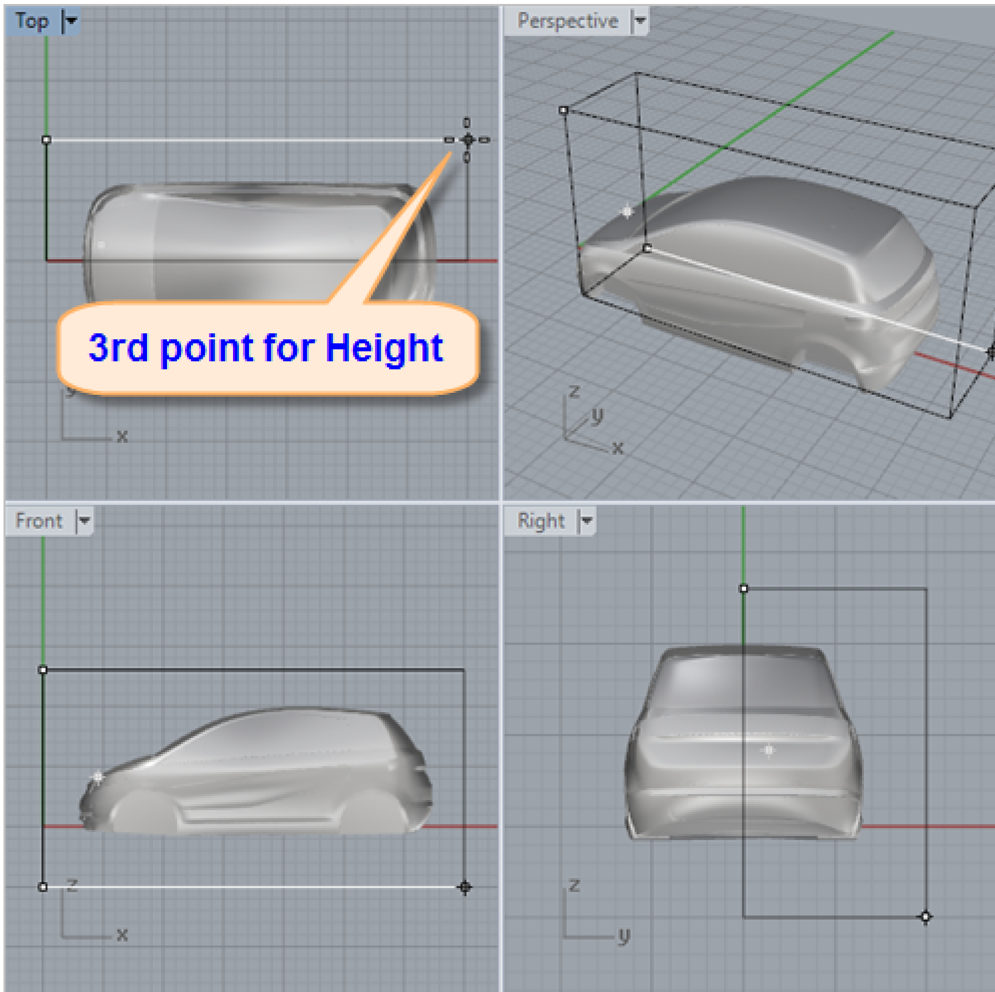
3. We want to pick two diagonal points to define the base of the box.
4. First, from the Rhino Status Bar, select Grid Snap 

and then activate the Right view to work in.
5. For the first corner of the base, we select a grid point that is above and on the centerline of the mesh.

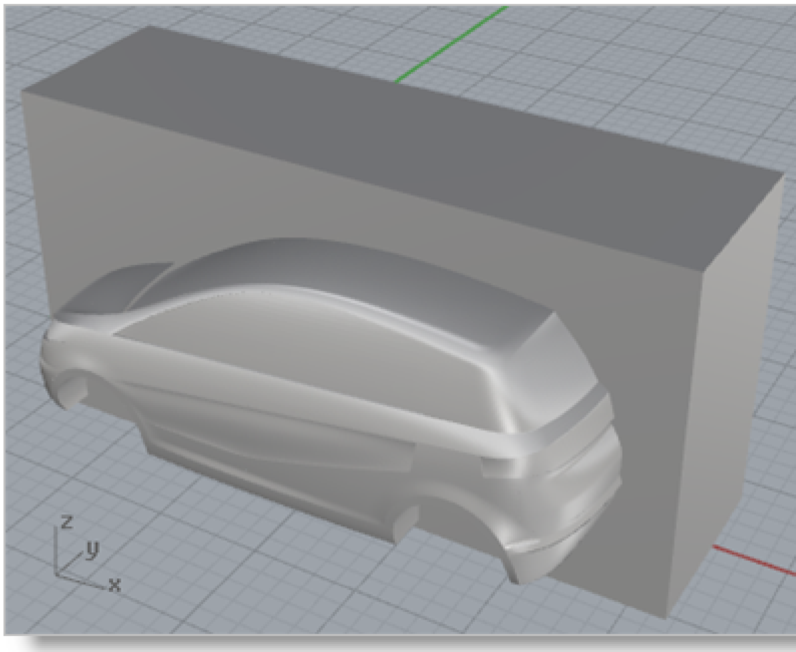


6. For the second corner of the base, we select a grid point that is below and to the right of the mesh, making sure that half of the mesh is completely enclosed by the base of the box.
7. Now a third point is needed to define the height of the box.

   For this we switch to the Top view and select a grid point that is behind and to the right of the mesh, again making sure that half of the mesh is completely enclosed by the height of the box.



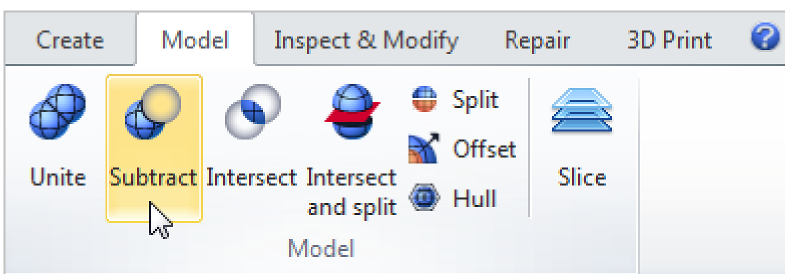
8. Now we activate the Perspective view to see the box clearly.



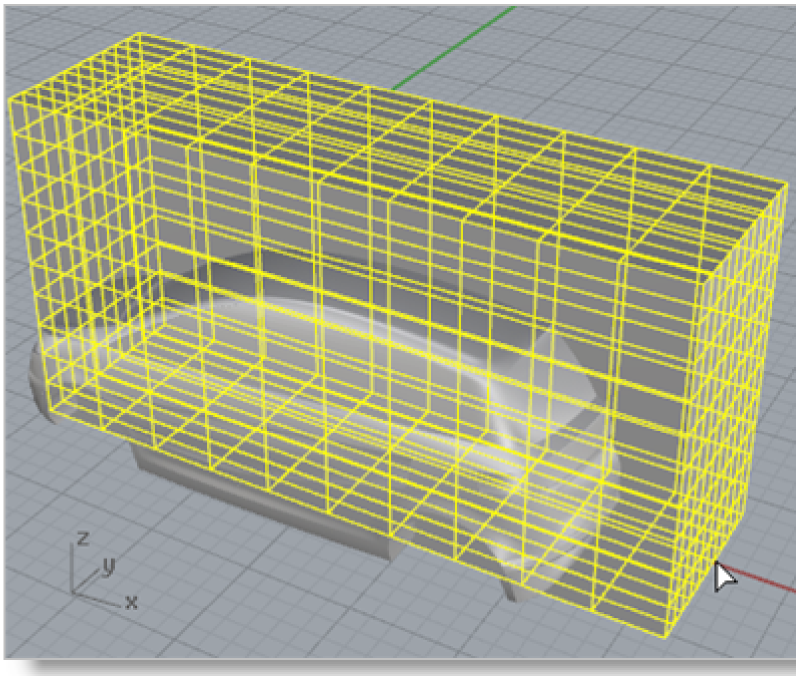
5.6 Subtract

To create our cavity block, we will subtract the mesh from the box we just created.

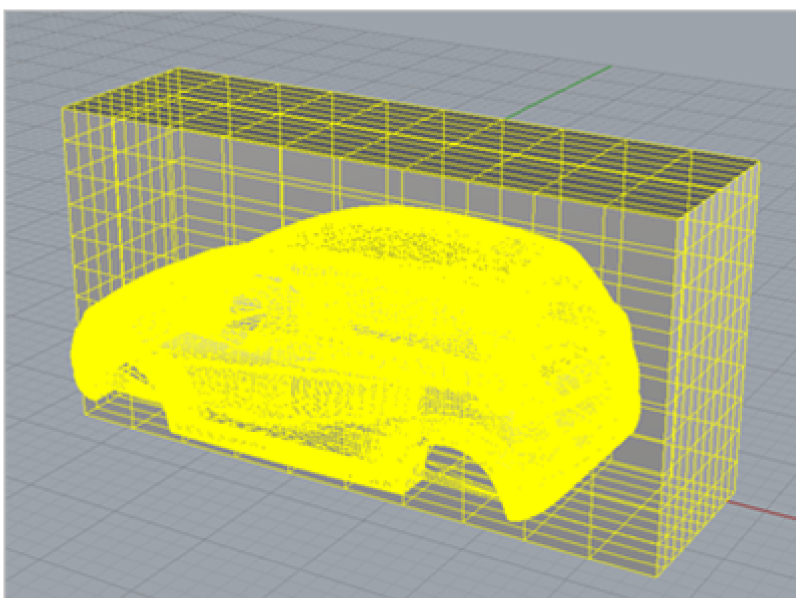
1. In the Rhino3DPRINT 2017 Browser, we'll switch to the Model tab and then pick Subtract.

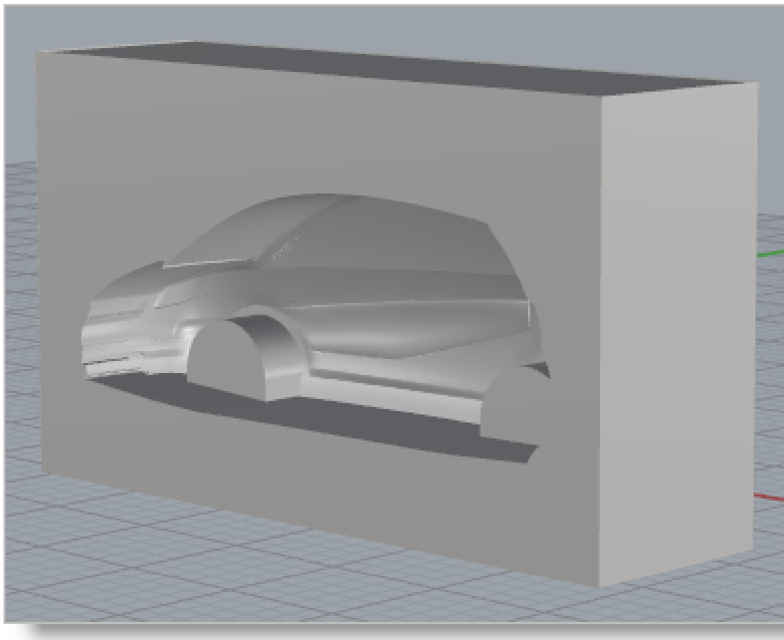


2. The command prompt says "*Select mesh set to subtract from:*" For this we select the box and then press Enter.



3. Now the prompt says "*Select mesh set to subtract:*" For this we select the mesh and then press Enter to see the results.





This completes Chapter 5 of the Quick Start Guide for Rhino3DPRINT 2017.

For further assistance you can visit the Online Help supplied with the program or visit www.mecsoft.com for additional tutorials.

Where to go for more help

We have come to the end of the Quick Start Guide of the Rhino3DPRINT 2017 module. This tutorial is just the tip of the proverbial iceberg of the various functions and controls available in the Rhino3DPRINT 2017 module. Please explore the product in more depth to get a feel for how these functions and controls operate.

If you need additional help please use the following resources:

- The on-line help distributed with the product is a great resource to find reference information on the various functions available.
- Apart from the on-line help system you can download other tutorials and projects from MecSoft Corporation's web site at www.mecsoft.com.
- If you need additional help, or if you have any questions regarding Rhino3DPRINT 2017, you may contact us via e-mail at support@mecsoft.com
- MecSoft offers Online training as well as personalized full day training sessions. Please look up our website or email us at sales@mecsoft.com for further details
- Please do continue to visit our home page to learn about the latest updates to Rhino3DPRINT 2017 and any other help material.

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