MESH
Quick Start Guide

RhinoCAM 2022

Sunday, February 13, 2022

MecSoft Corpotation

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RhinoCAM-MESH 2022 Quick Start Guide

by MecSoft Corporation

User Notes:

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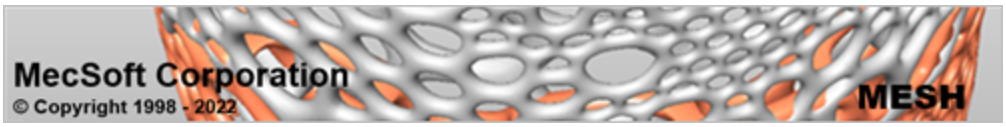
Quick Start

RhinoCAM 2022



Prefer Printed Documentation? Check Here!

Quick Start Guides for each RhinoCAM module are available in both PDF and Video format. Refer to the following information to access these resources:

What's New!

What's New in RhinoCAM 2022

Watch the What's New in 2022 Webinar!

The Complete Quick Start Video Play List

Here is a link to the complete 2022 Video Play List

How to Access the Quick Start Guide Documents

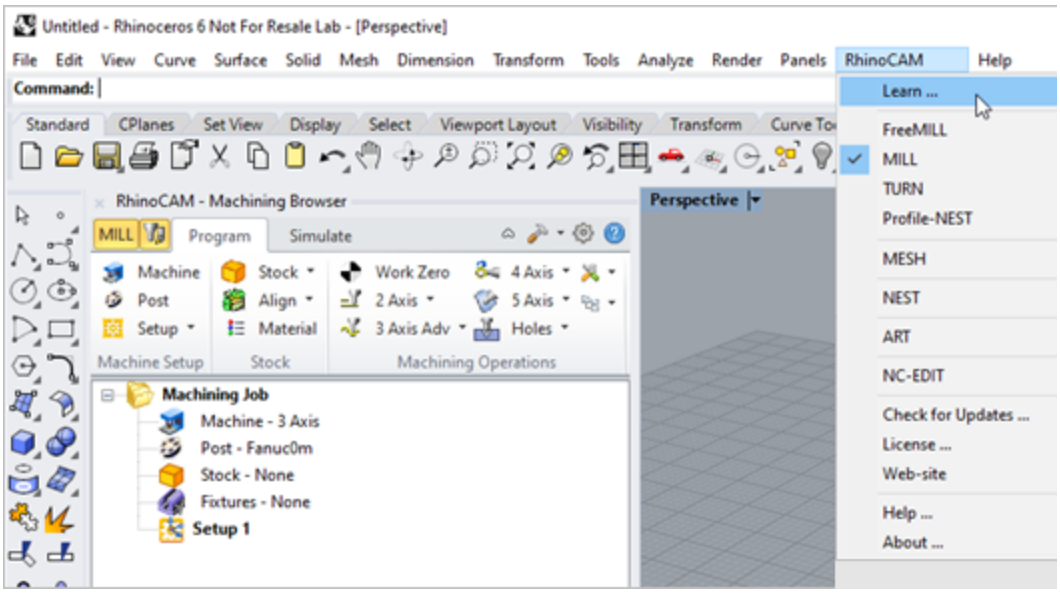
To help you quickly get started in working with each module, select one of the Help buttons located on the RhinoCAM Learning Resources dialog.

You will find:

- Quick Start Guides
- What's New documents
- Online Help links

The Quick Start Guides will help you step through an example tutorial which will illustrate how to use the module. To access the Learning Resources dialog:

1. From the Rhino Main Menu, drop down the Main menu and select Learn ...



To access the Learning Resources dilog in RhinoCAM

2. Select a document from the Learning Resources dialog to get started using the module of your choice.

You can also select the Open Quick Start Files Folder button located at the bottom of the dialog to open the Quick Start folder where the source files (start and completed versions) are located.

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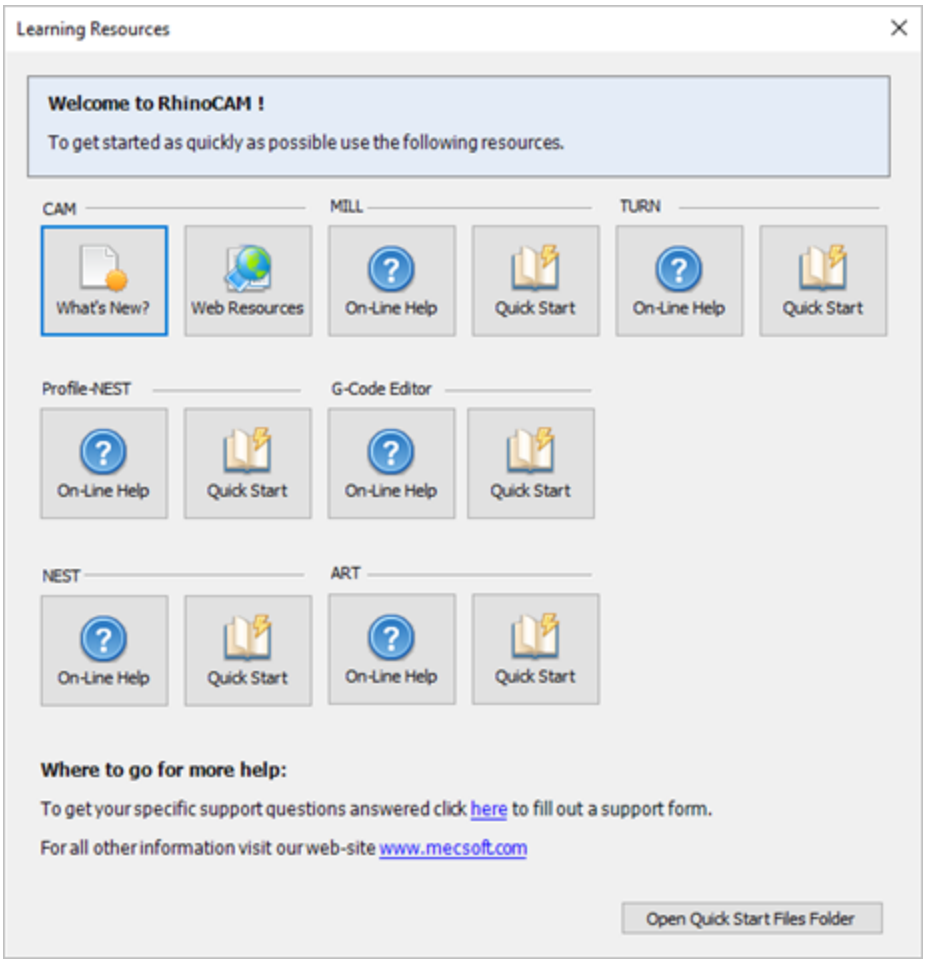
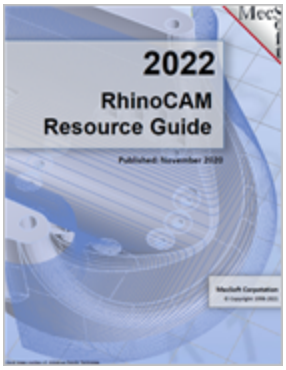


Learning Resources Dialog

Resource Guide

Download this PDF Guide for a list of the available RhinoCAM Resources.

2022 RhinoCAM Resource Guide



The 2022 RhinoCAM Resource Guide!

18 Pages

Lists PDF downloads and Online resources including Quick Start Guides, Reference Guides, Exercise Guides, Tutorials and More.

Prefer Printed Documentation? Check Here!

About this Guide

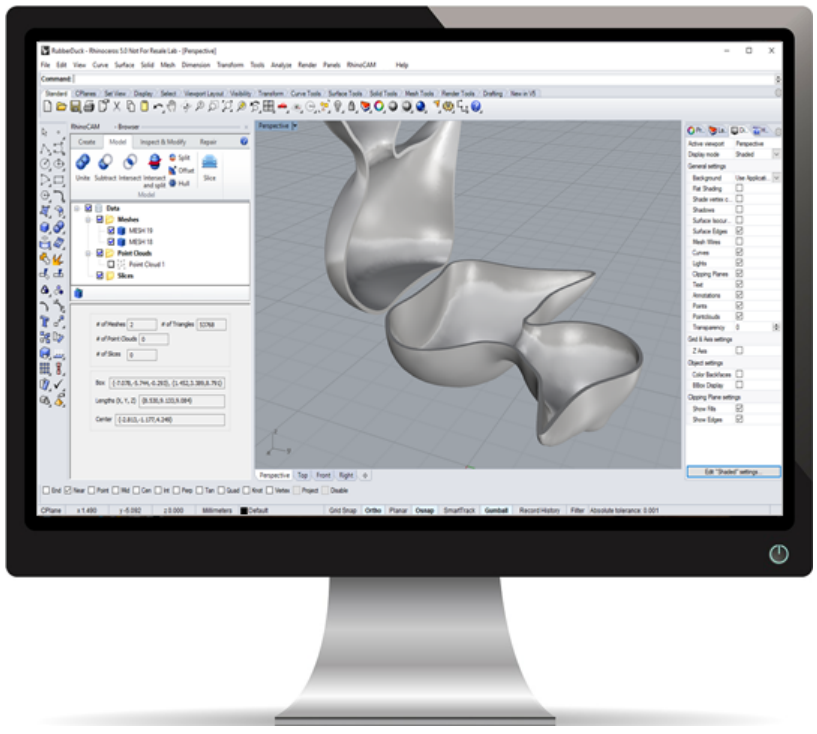
3.1 Useful Tips

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

1. For purposes of brevity, Rhino refers to both Rhinoceros 6 or Rhino 7.
2. Copy the tutorial part files in a location other than the installation folder to make sure you have read/write privileges to the files.
3. Once you start working with the tutorial file, save your work periodically!
4. Don’t stress out too much if you are having trouble with the tutorial. Call us or send us email and we can help you out.
5. Most of all have fun!

3.2 About the MESH Module

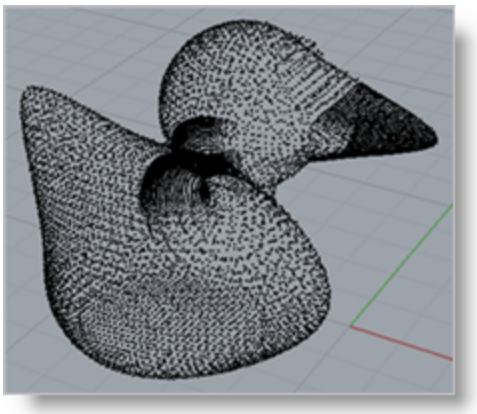
RhinoCAM is a product from MecSoft Corporation that runs as a plug-in inside of the Rhino modeling system. This product offers efficient, easy and automatic tools for creating, analyzing and repairing large 3D mesh data sets such as point clouds and other faceted mesh files.



3.3 Using this Guide

This guide will show you how to use RhinoCAM to create and modify mesh geometry. You will learn how to analyze, repair and modify 3D mesh models. You will learn how to use point cloud data to create and refine a mesh model. You will also learn how to offset, split and cap a mesh to create a uniform wall thickness.

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



Point Clouds (File: RubberDuck.csv)

3.4 Watch the Video!

Want to see a video demonstration of this quick start guide? Just click on the play list below and watch the MESH Quick Start Guide video.

Here is a link to the complete 2022 Video Play List

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Getting Ready

4.1 Running RhinoCAM

Locate the Rhinoceros 6 or Rhinoceros 7 shortcut on your desktop and double click to launch the application.

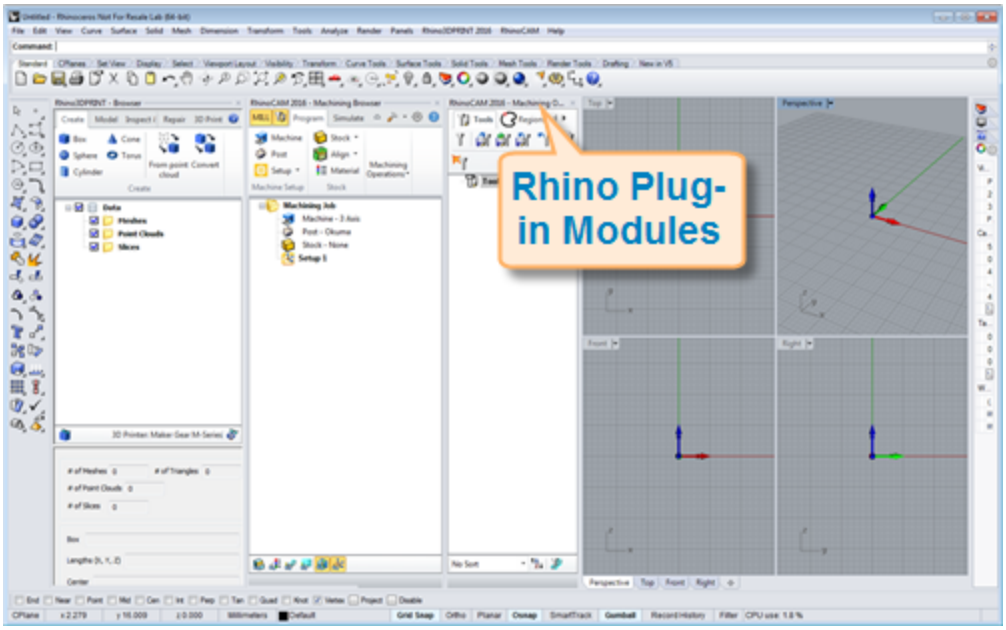
Alternatively you can also click on the Windows Start button and select All Programs. Go to the program group containing Rhinoceros. (The name of this program group will usually be called Rhinoceros, unless you specified otherwise during setup.)

Once you locate the program group, select it and then select Rhinoceros to launch the application.

If the installation was successful, upon launching of Rhinoceros you should observe a menu entry called RhinoCAM in the main menu bar of Rhino.

4.2 About the RhinoCAM Display

Before we begin, let's talk a bit about the RhinoCAM display. When you run RhinoCAM for the very first time, your screen may look this.



These windows on the left belong to plug-in modules that are currently loaded. For now, let's close all of them.

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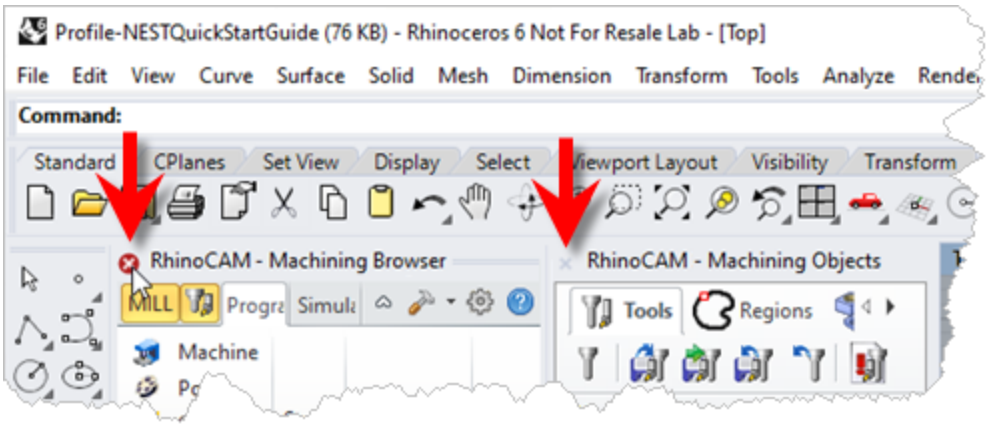
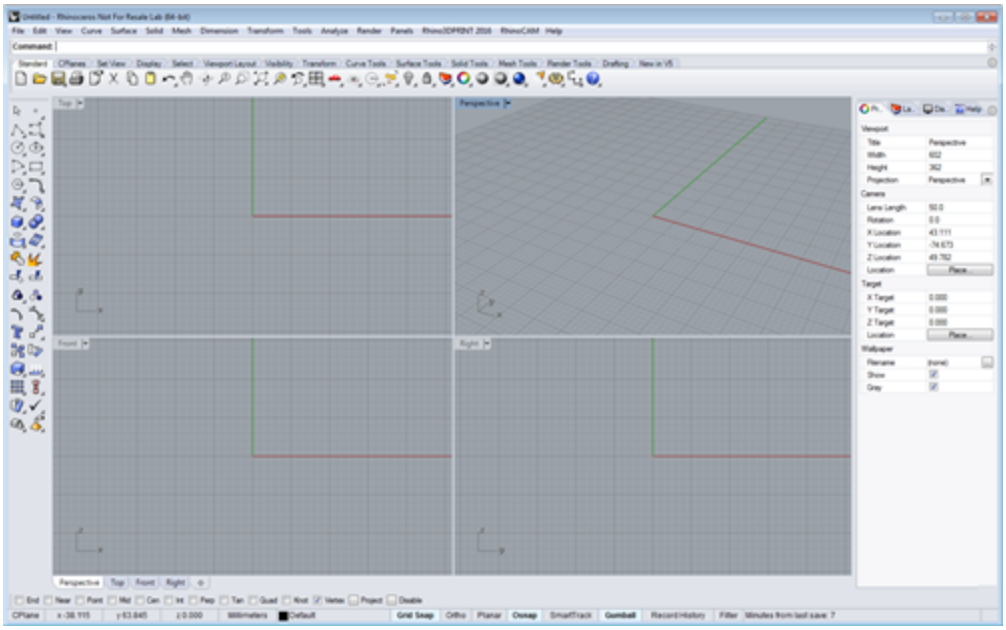


With all plug-in modules closed your screen will look like this:



4.3 Launch the MESH Module

Now, let's begin by launching RhinoCAM.

1. From the Rhino main menu bar, you will see the RhinoCAM 2022 menu item.
2. To load the module, drop-down the menu and pick MESH.

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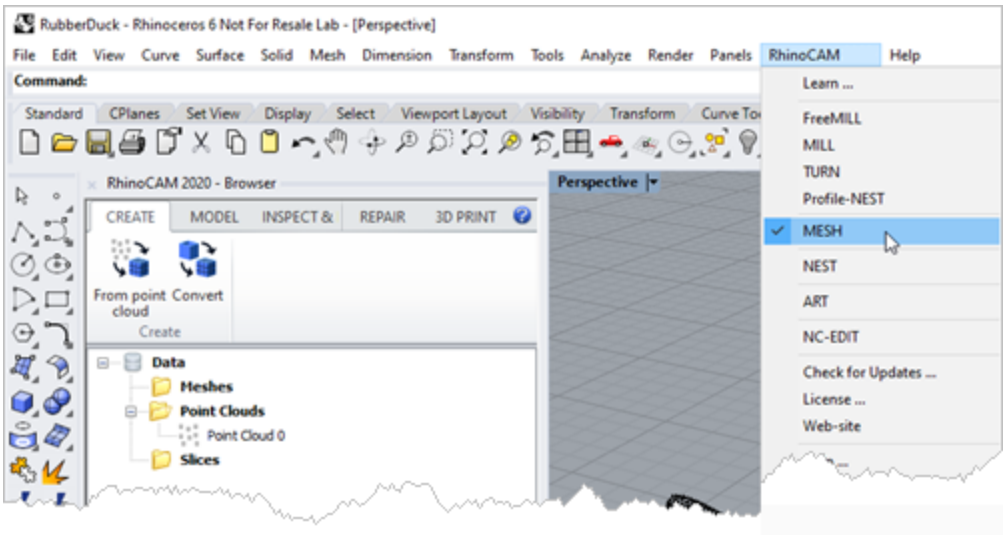
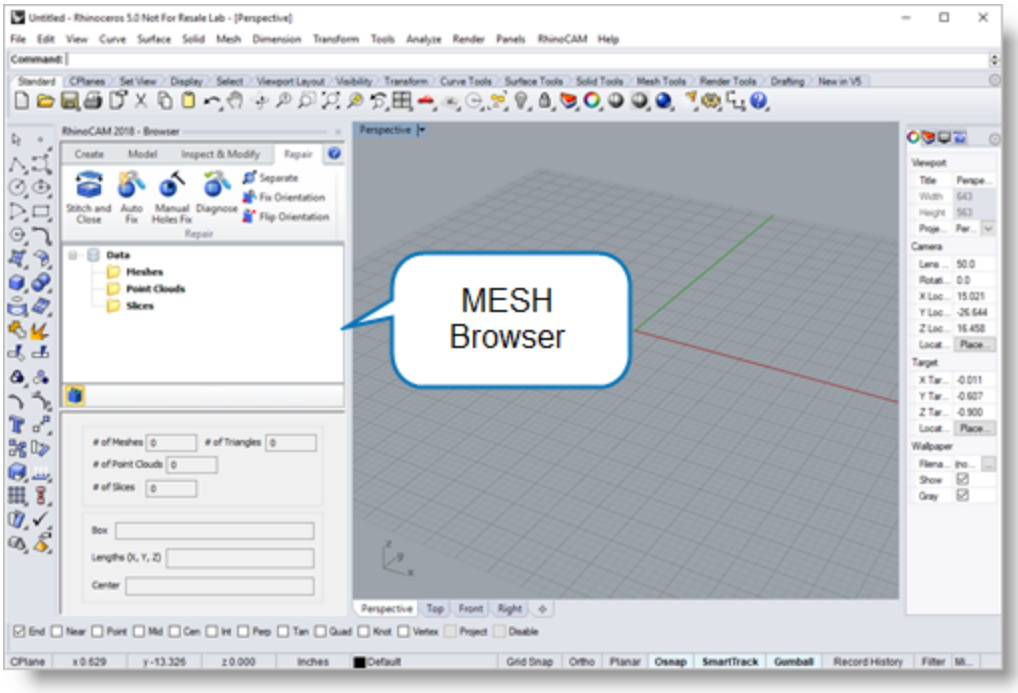
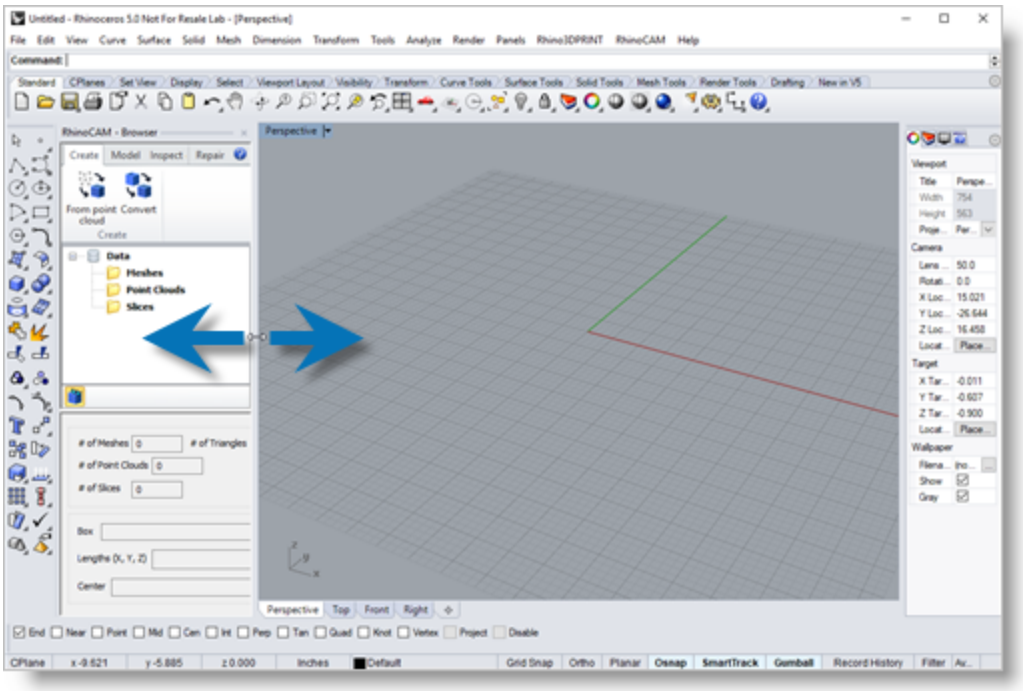


3. Docked on the left you will see the Mesh Browser.



4. You can re-size the width of the browser making sure that all of the command icons and menus are easily accessible.

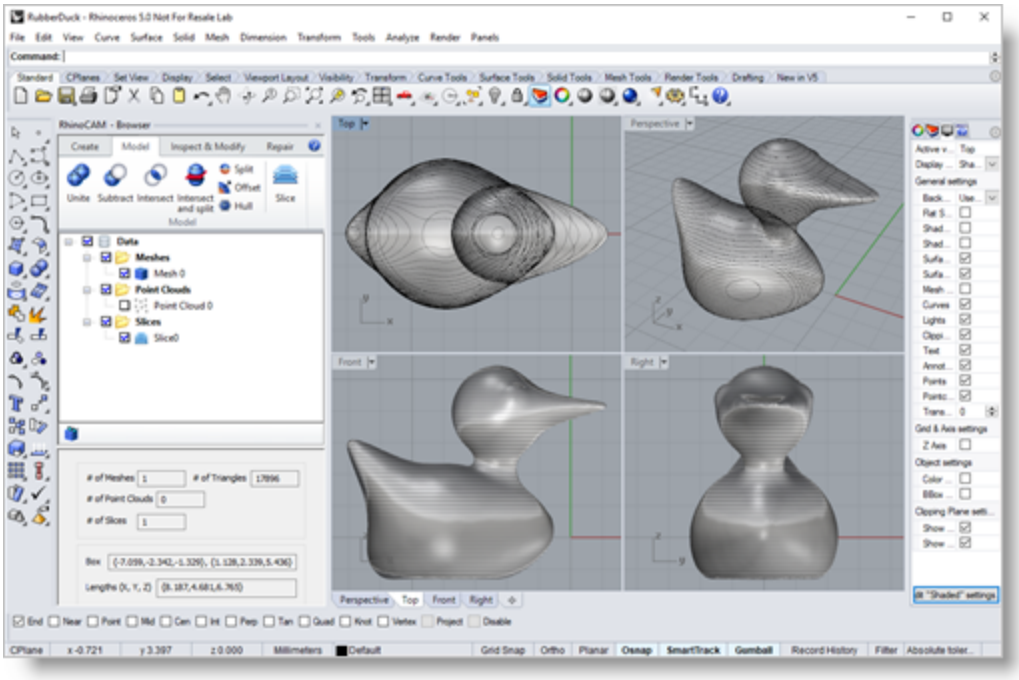

Untitled - Rhinoceros 5.0 Not For Resale Lab - [Perspective]
File Edit View Curve Surface Solid Mesh Dimension Transform Tools Analyze Render Panels Rhino3DPRINT RhinoCAM Help
Command:
RhinoCAM - Browser
Create Model Inspect Repair
From point cloud
Convert
Create
Data
Meshes
Point Clouds
Slices
# of Meshes
# of Triangles
# of Point Clouds
# of Slices
Box
Lengths (X, Y, Z)
Center
Perspective
Top
Front
Right
End Near Point Mid Cen Int Perp Tan Quad Knot Vertex Project Disable
CPlane
Default
Grid Snap
Ortho
Planar
Osnap
SmartTrack
Gumball
Record History
Filter

Inspect & Modify a Mesh

In this section we will work with Point Cloud data to create, analyze and repair a Mesh model.

We will perform the following steps:

1. Open a Point Cloud data file.
2. Create a Mesh model from the Point Cloud.
3. Auto Fix the mesh.
4. Perform a Re-Mesh.
5. Smooth & Analyze the mesh.
6. Slice the mesh.
7. Our completed part will look like this:



5.1 Load a Point File

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

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

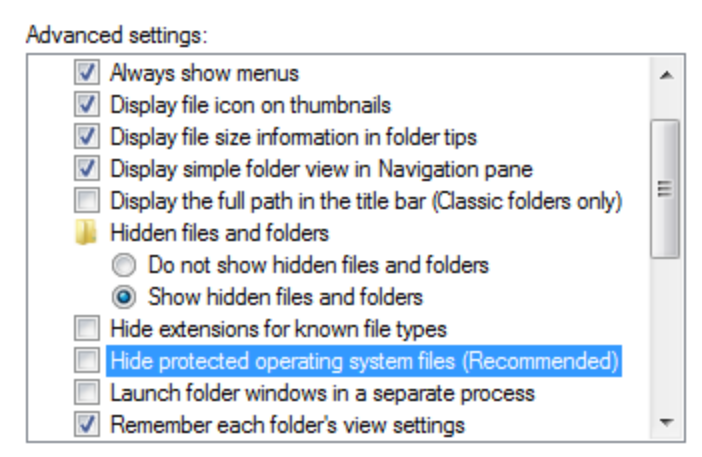
2. Find the part file named *RubberDuck.csv* located in the QuickStart folder and then pick Open. From the Open dialog box, select the file from the C:\ProgramData\MecSoft

Corporation\RhinoCAM 2022 for Rhino x.x\QuickStart\ folder. It is advisable to make a copy of this part at a suitable alternative folder so that you have write privileges to modify the part.

Note: This is a *.csv file so be sure to set the File Type to *.* (all files) from the File Open dialog.

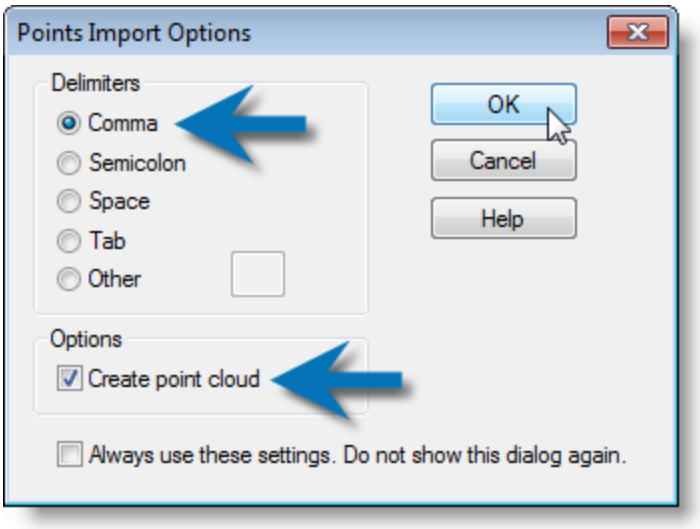
By default, the ProgramData folder is "hidden" from view. Here are the steps to Show hidden files and folders:

1. Go to Control Panel > Appearance and Personalization > Folder Options.
2. Select View tab and under advanced settings:
   - Select Show Hidden files and folders
   - Uncheck Hide extensions for known file types
   - Uncheck Hide protected operating system files (Recommended)



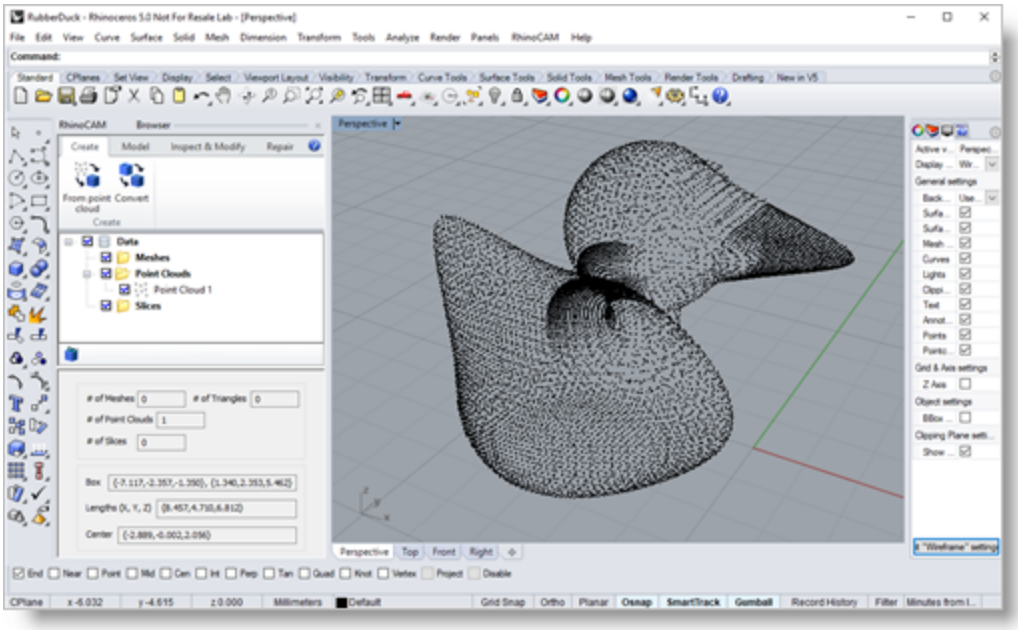
3. Click Apply and OK.

3. From the Points Import Options dialog:



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

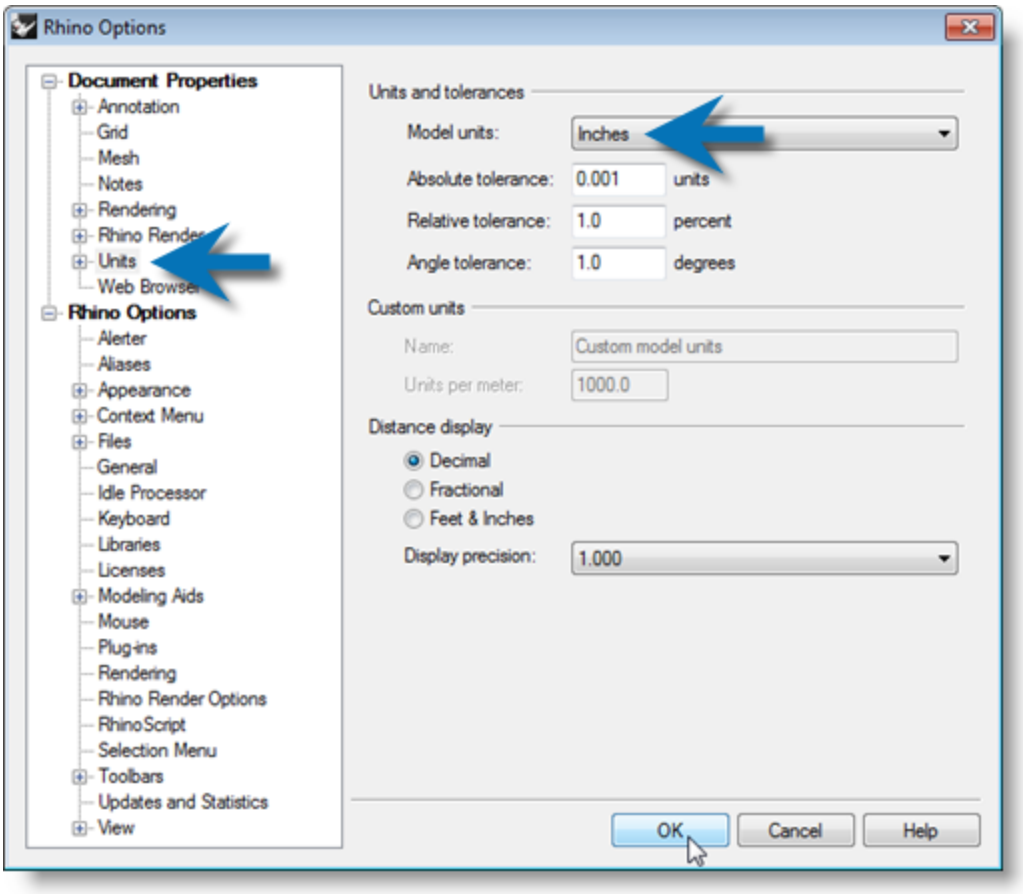
4. Once imported, select the Perspective view to work in.



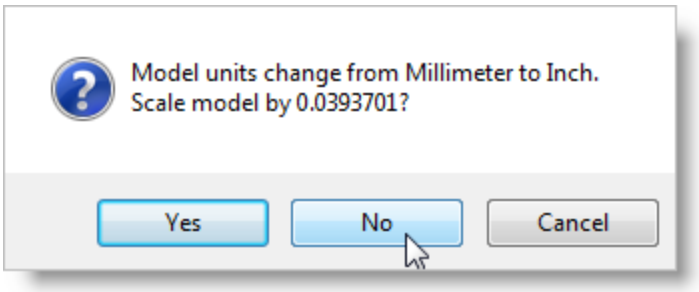
5.2 Mesh from Point Cloud

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

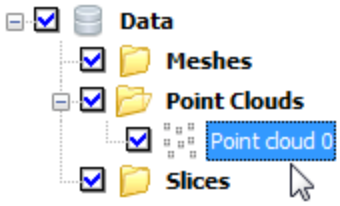
1. From the Rhino Main Menu, select Tools and then Options.
2. Select Units from the tree o the left and then set the Model Units to Inches and pick OK.



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



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



4. From the Create tab of the MESH Browser, select From Point Cloud to create a Mesh model from the point cloud.

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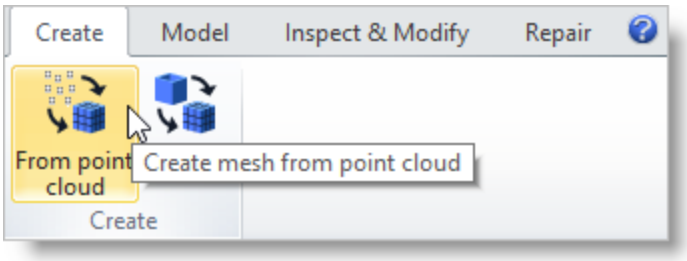
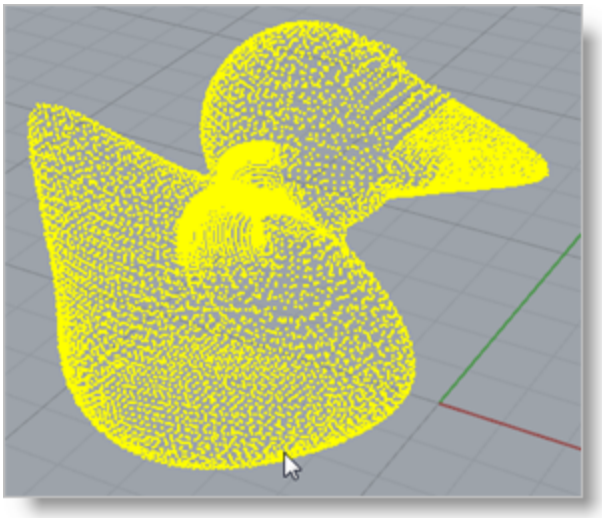
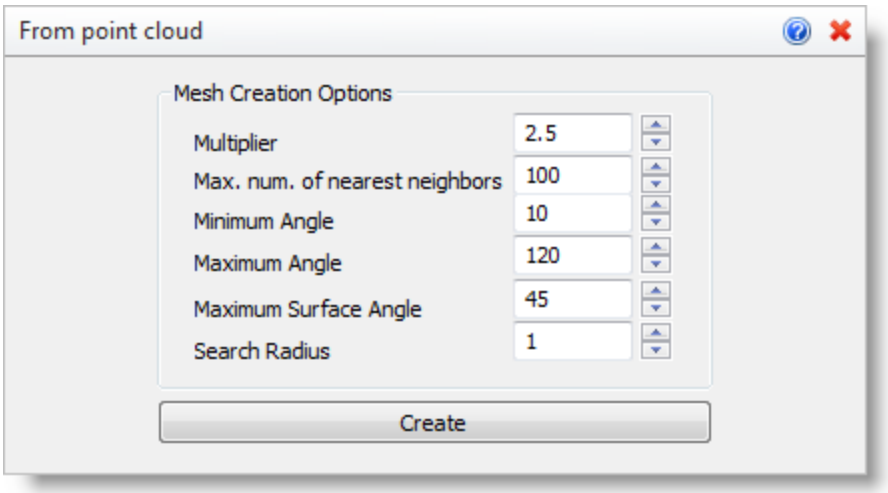


5. Select the point cloud and press Enter.
   Note: If the Point Cloud is already selected from the previous step, just press Enter.



Notice that an options dialog appears at the bottom of the MESH 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.

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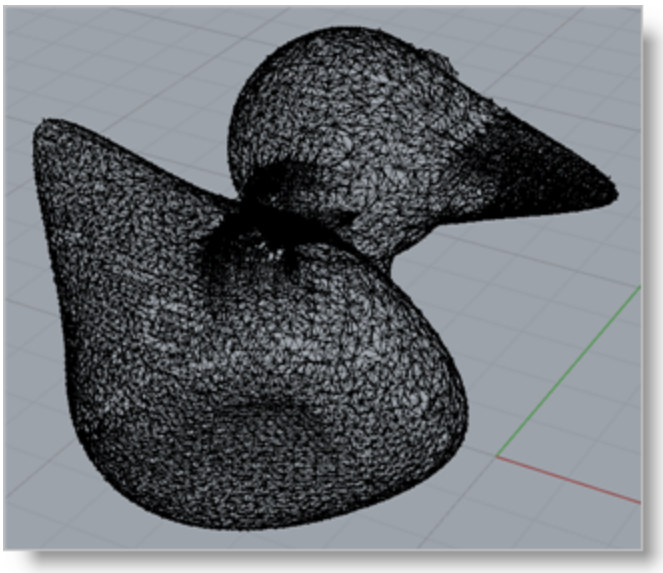

Mesh from Point Cloud

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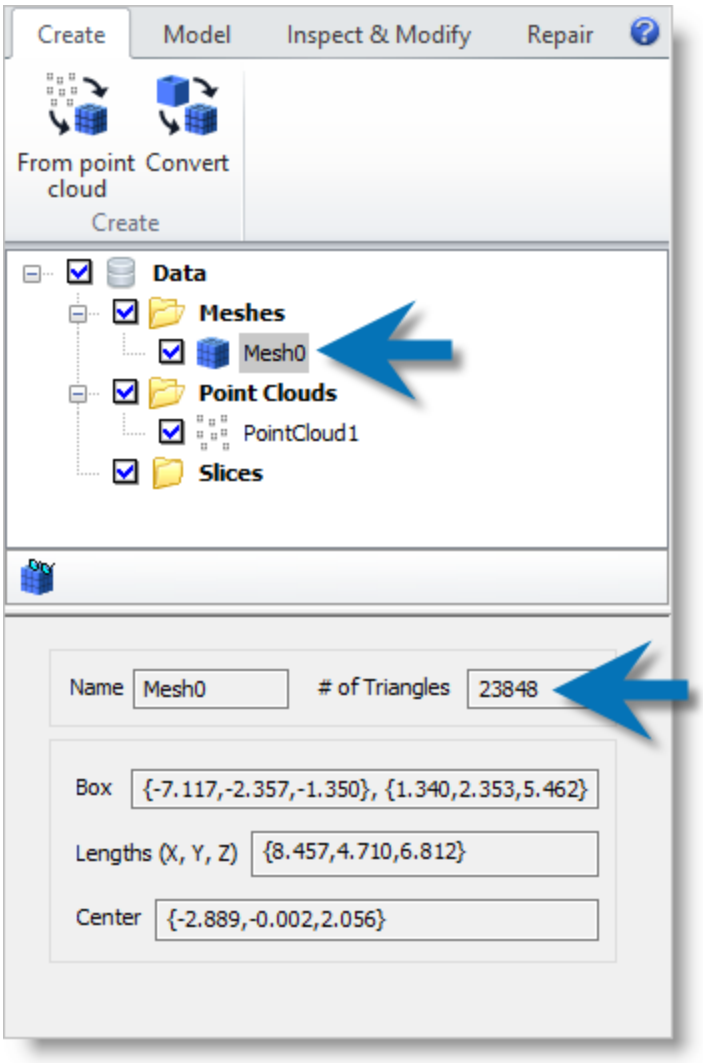

7. From the Rhino Standard toolbar, select Shaded Viewport.

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

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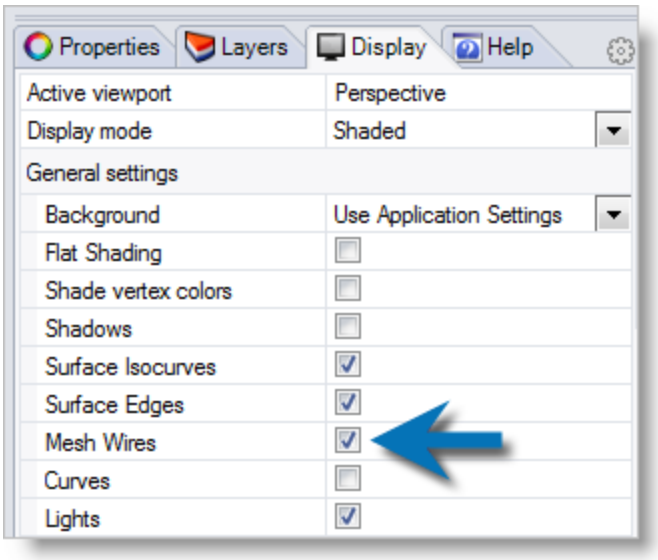
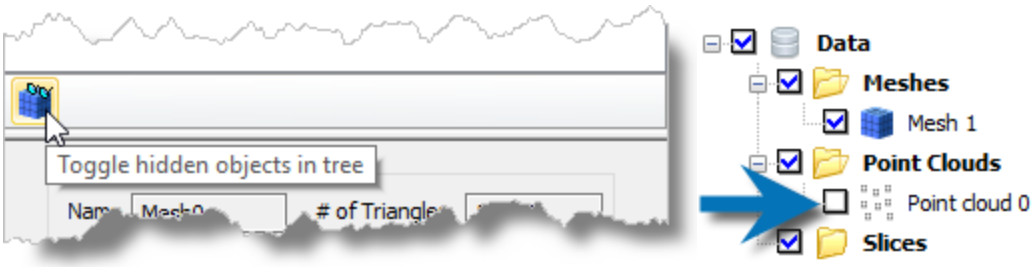


Display Mesh Wires in Rhino

9. Then from the MESH 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 un checking 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 many applications, such as 3D printing.

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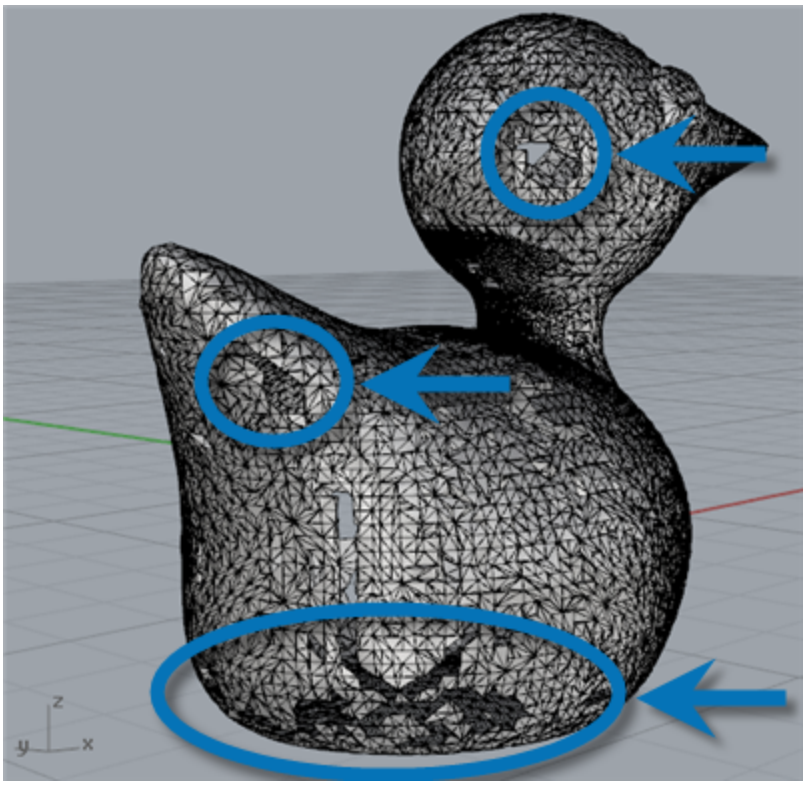
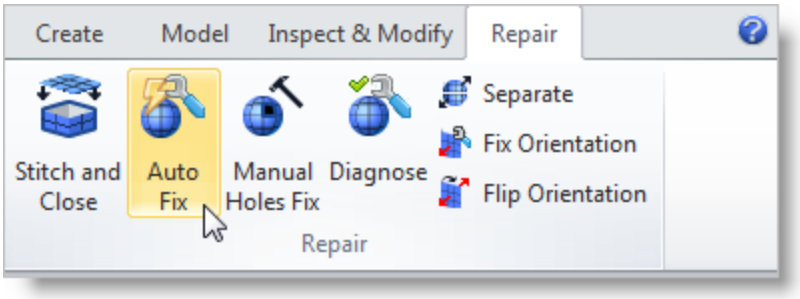


5.3 Auto Fix

We can repair the mesh automatically.

1. From the MESH 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.

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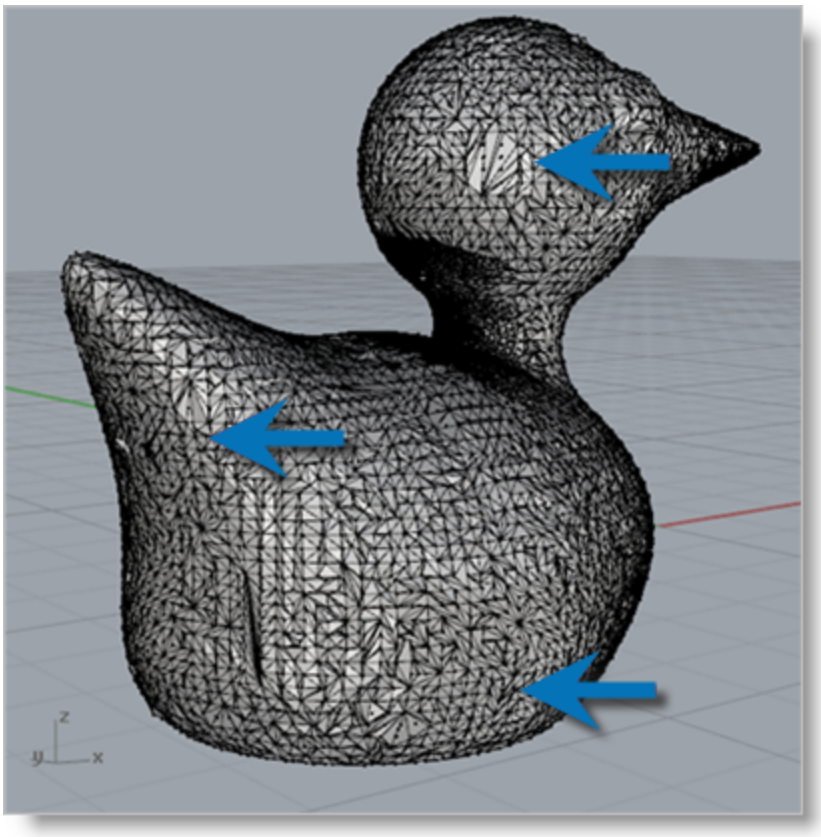
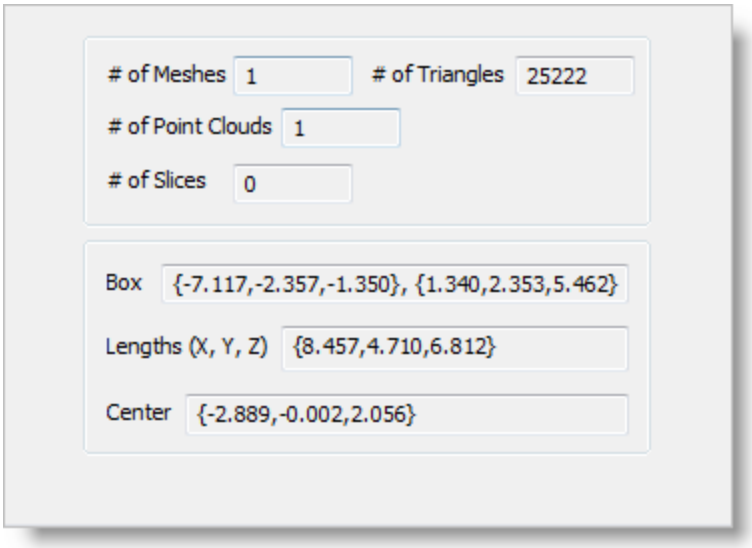


The browser now reports that the mesh has about 25,000 triangles. The exact # of Triangles reported may differ slightly depending on the exact build version of RhinoCAM you are running.



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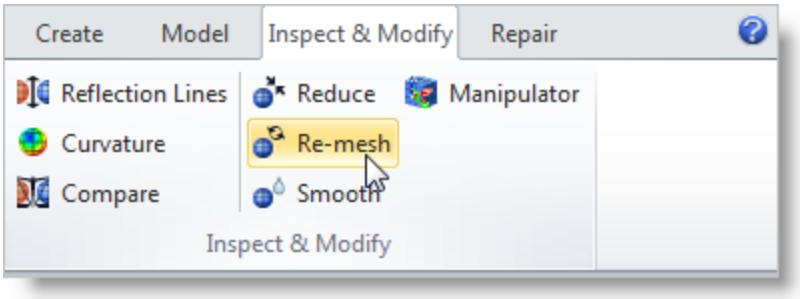
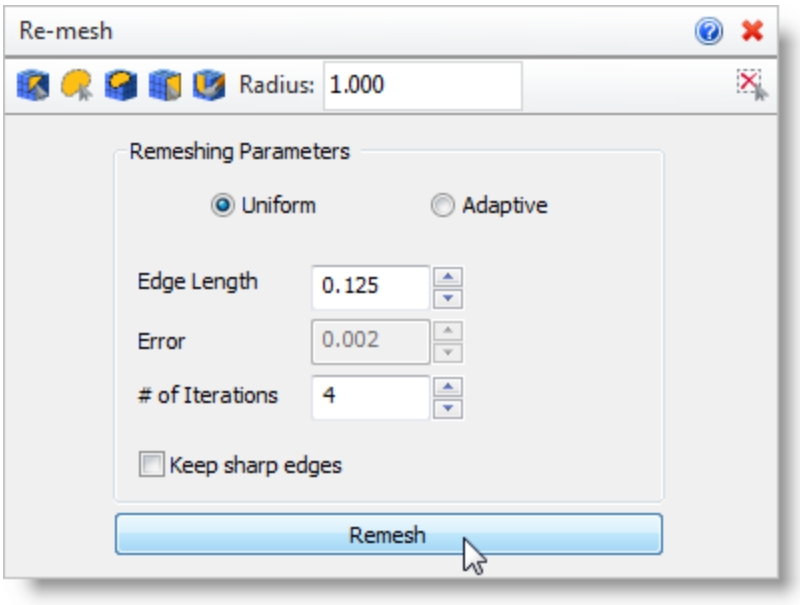
5.4 Re-Mesh

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.

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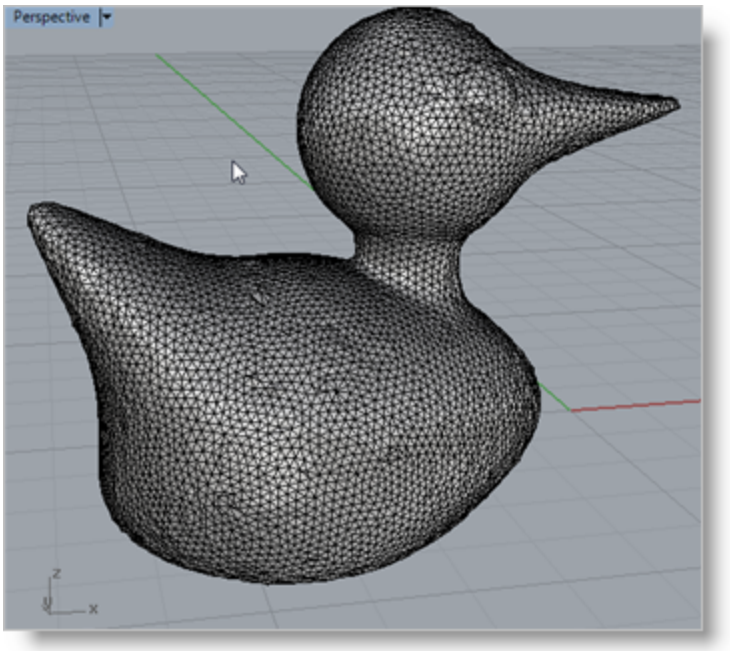

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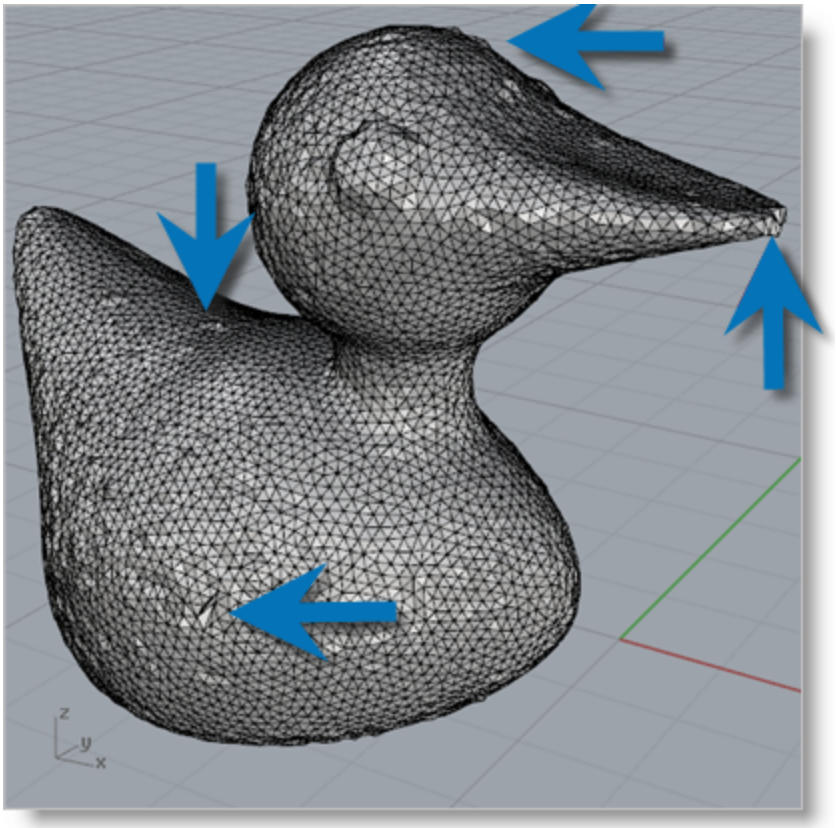
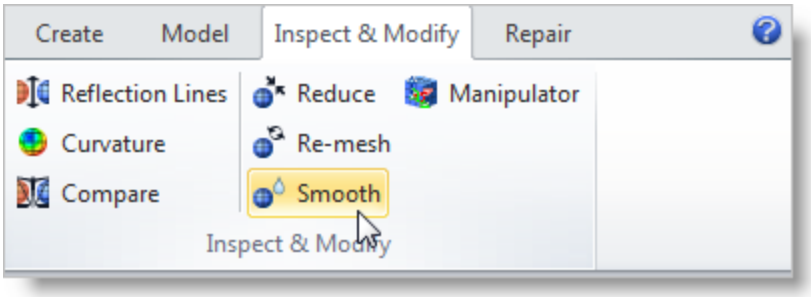
5.5 Smooth

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.

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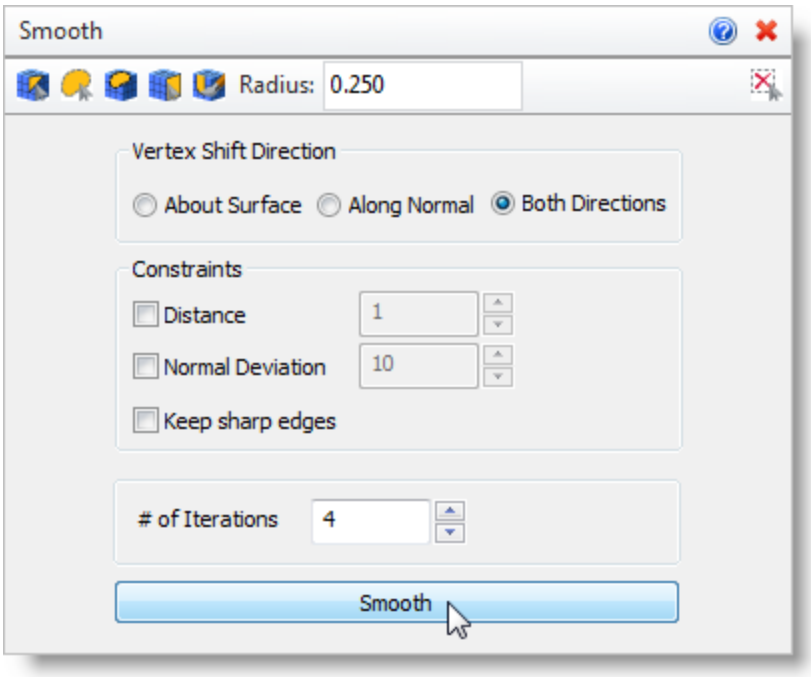
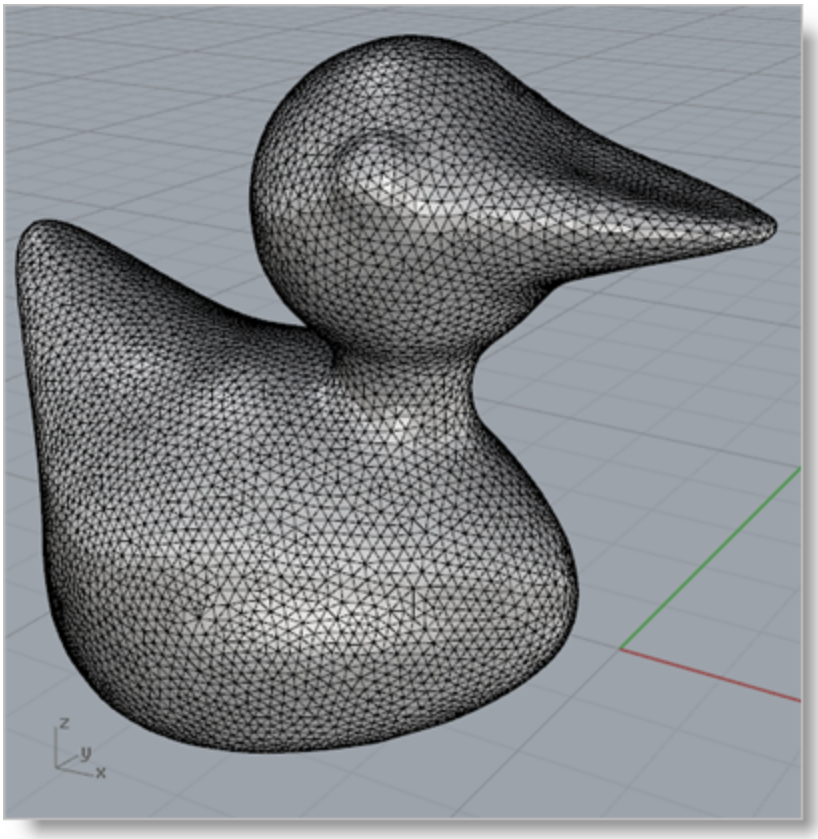


5. Now we'll pick the Smooth button and review the results. We see that the mesh looks much smoother now.



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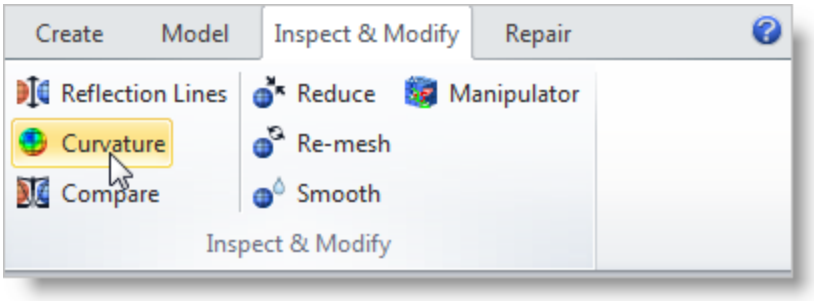
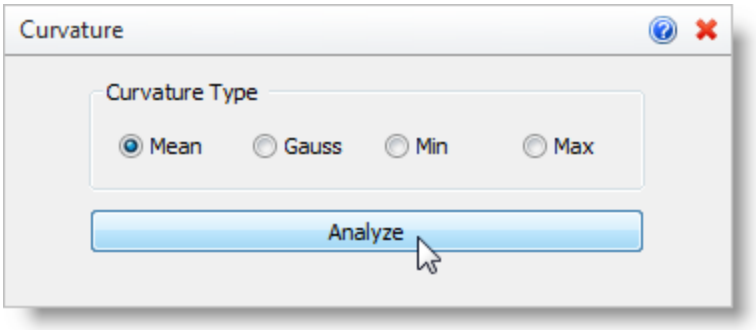
5.6 Analyze

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.



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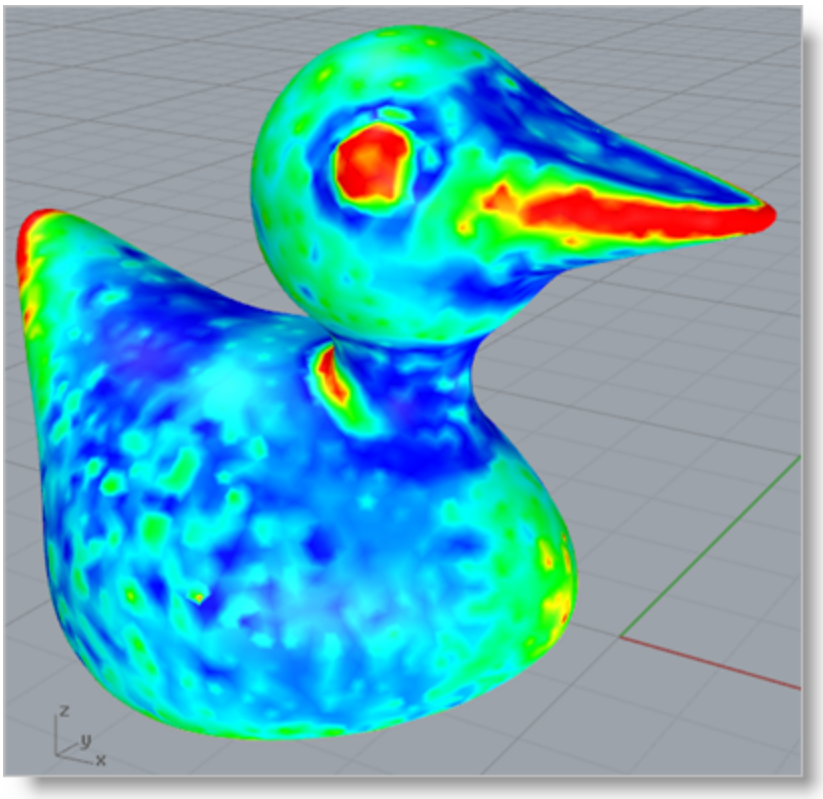
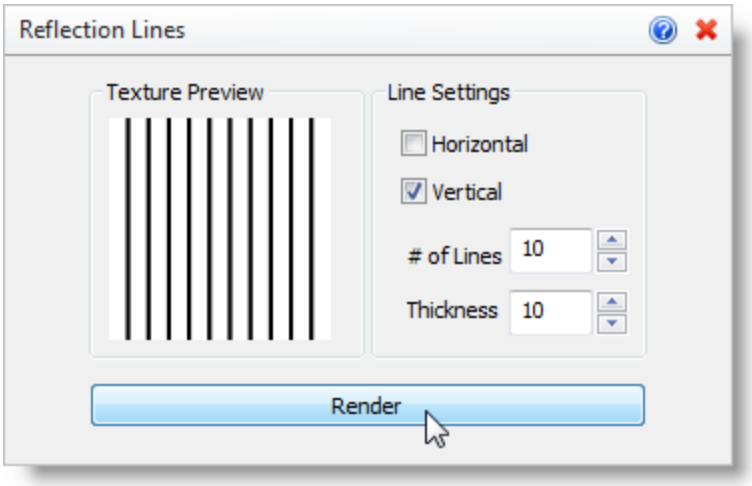


4. You can pick the Cancel icon ✖ to close the command dialog leaving the mesh selected.
5. Now, from the Inspect & Modify tab, select Reflection Lines. This method will indicate any irregularities or tangent discontinuities.
6. Just press Enter since we left the mesh selected from the previous command.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



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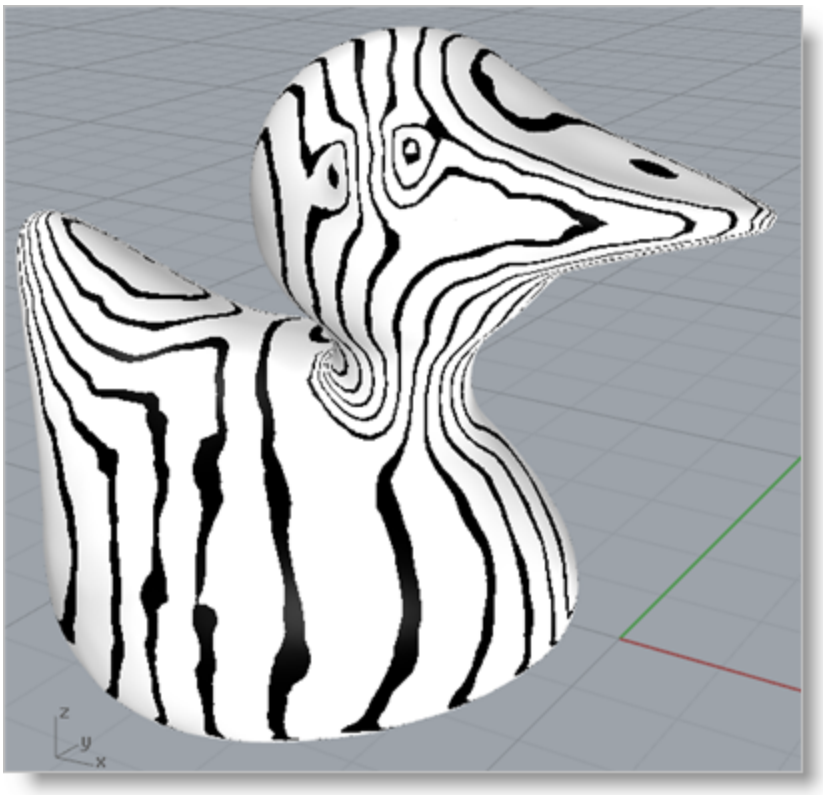
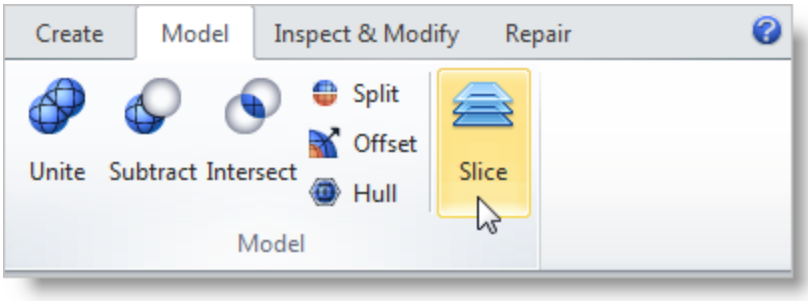


7. Again, you can pick the Cancel icon ✖ to close the command dialog leaving the mesh selected.

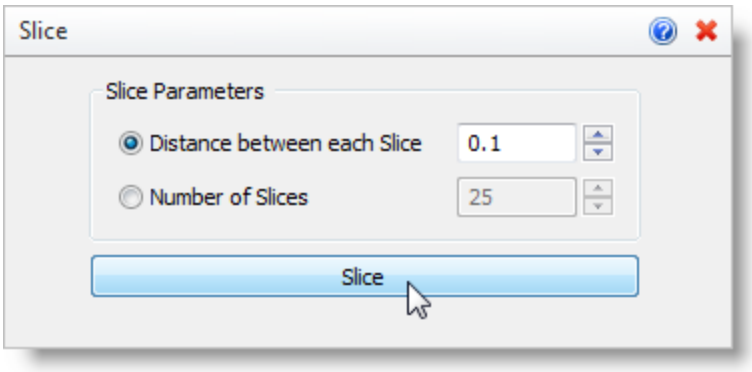
5.7 Slice

Next, we'll visually Slice the mesh model.

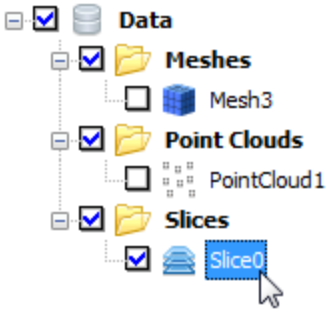
1. With the mesh still selected from the previous step, from the Model tab, select Slice.

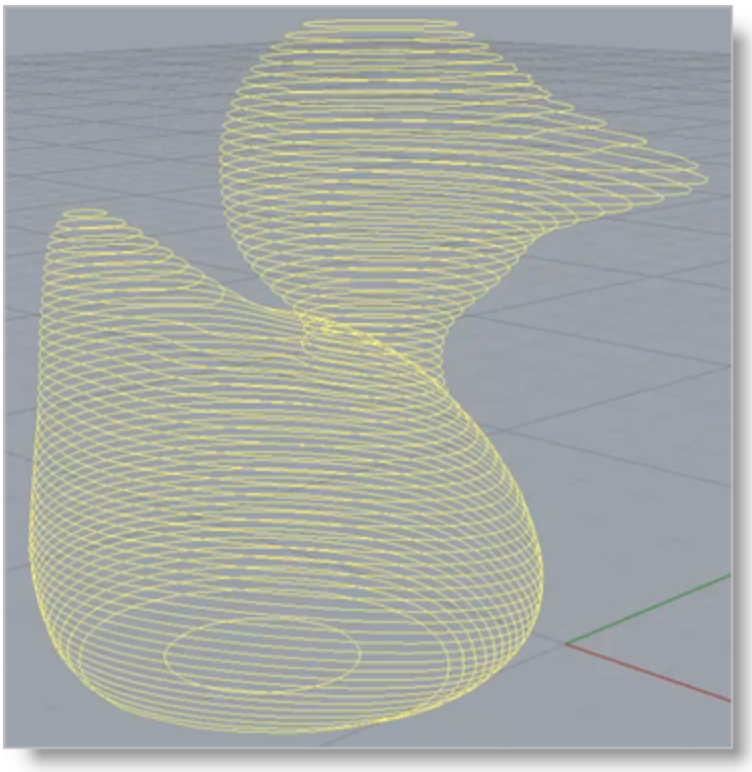


2. Select the mesh (if it is not already selected) and press <Enter>. From the Slice dialog, we'll set the Distance between each slice to 0.1 and then pick Slice.

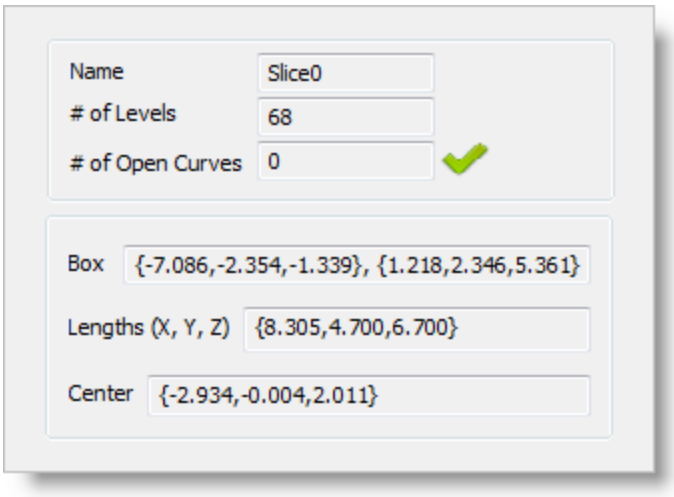


3. From the MESH Browser, first check the Slice object and then uncheck the Mesh object. The Slice will be isolated and clearly highlighted on the screen.

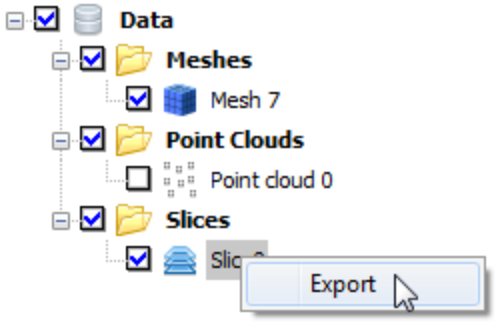




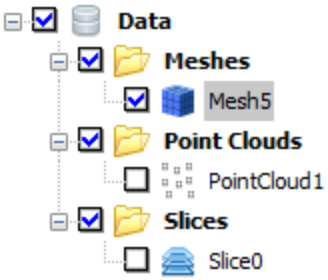
The MESH Browser displays information about the Slice. Notice that the ✔ icon is displayed indicating that NO open curves are present.



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



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



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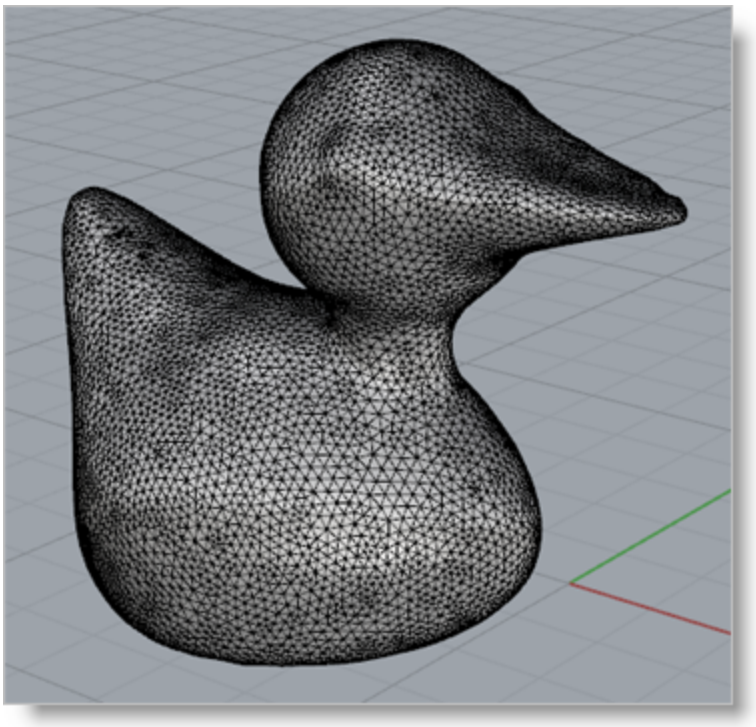

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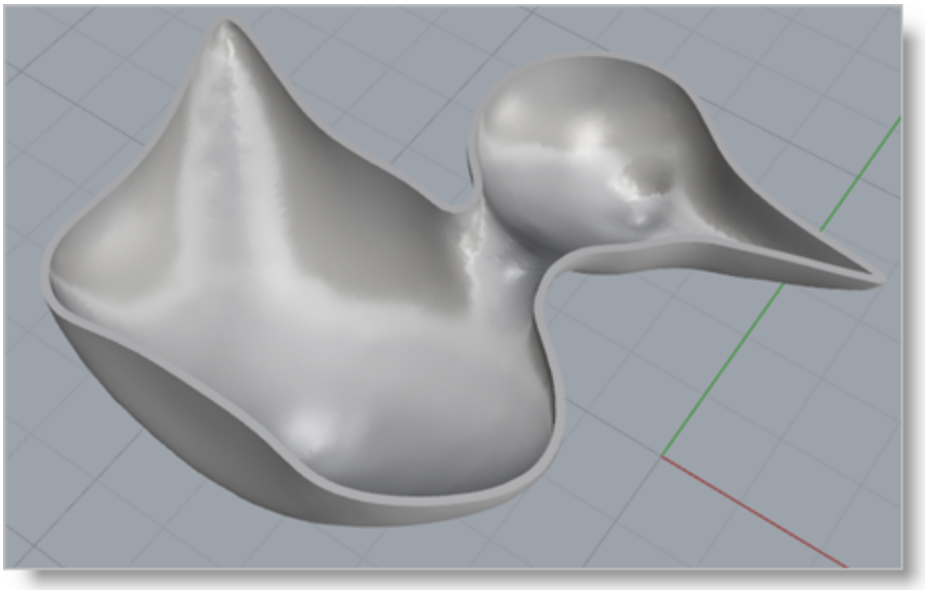
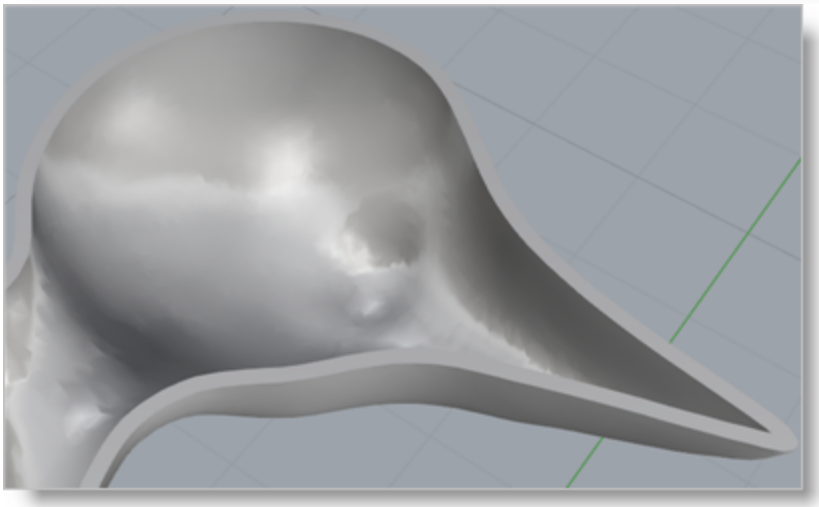
Offset & Split a Mesh

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

We will perform the following basic steps:

1. Use Offset to create a second a uniform distances away from the first mesh.
2. Use Split and CAP the two resulting mesh halves.
3. Our completed part will look like this:





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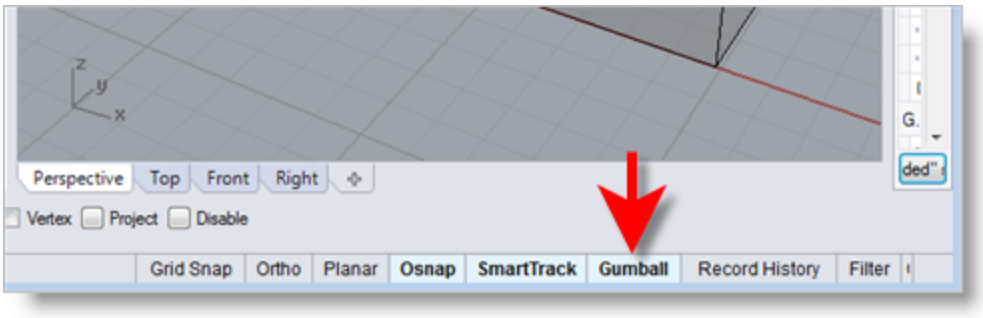
6.1 Offset

Now that we have a nice mesh from our point file, let's Offset it to create a wall thickness.

1. First we will rotate our current mesh onto its side using the Gumball in Rhino.
2. From the Rhino Status Bar, toggle ON the Gumball.



3. 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.

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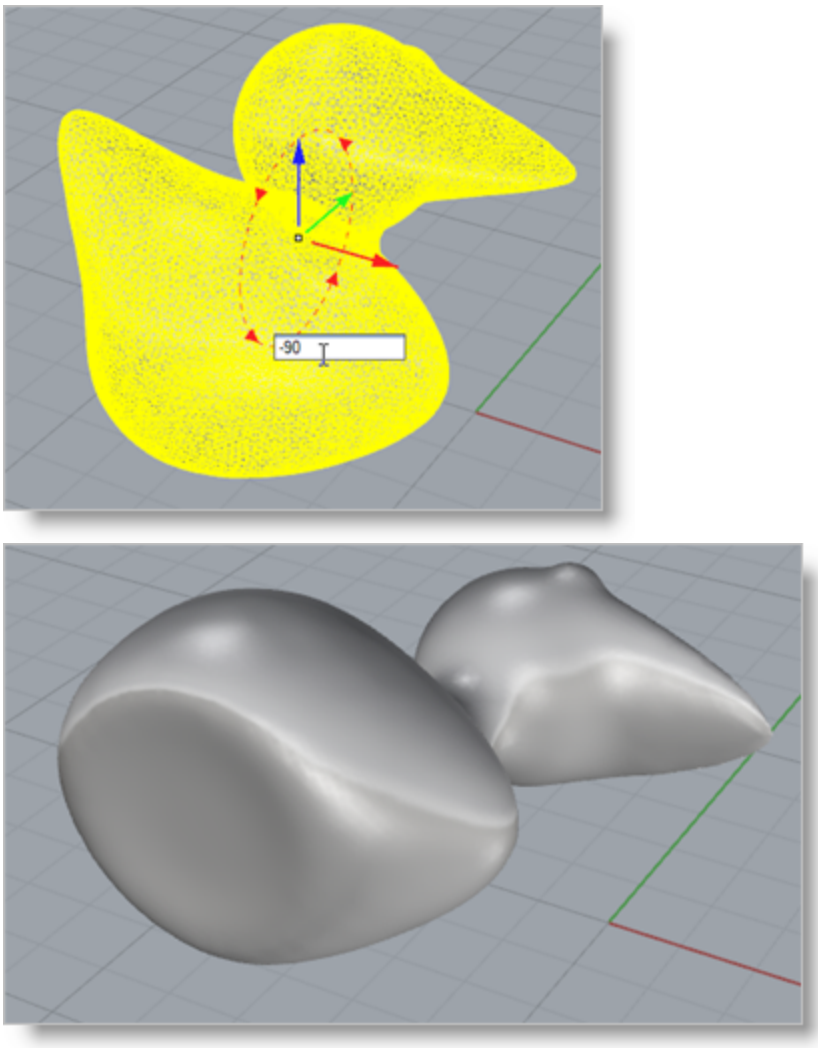
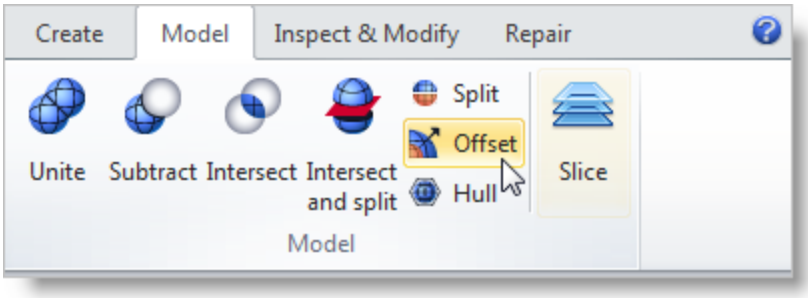


4. From the MESH Browser select the Model tab and then select Offset.



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

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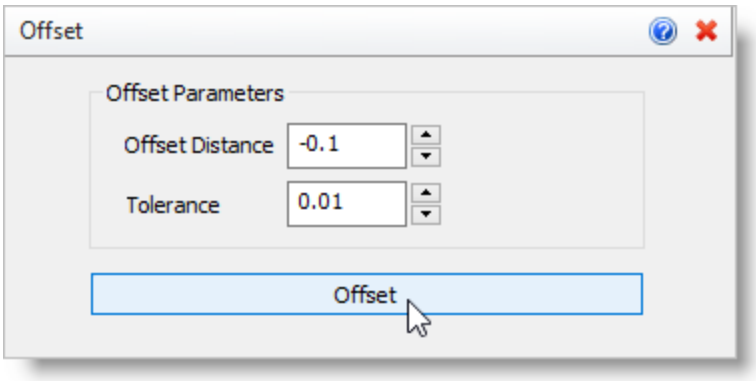
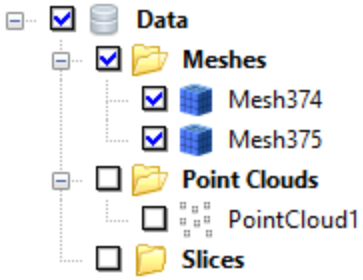


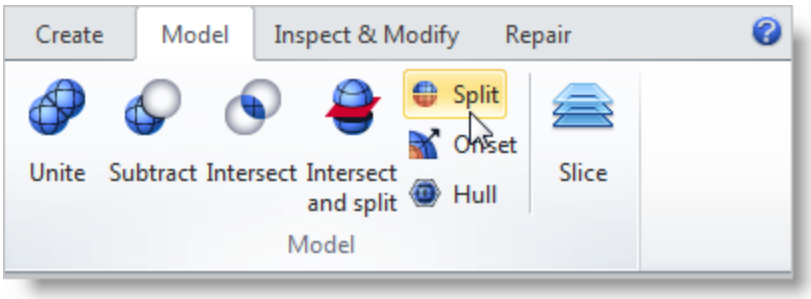
6. We'll set the Offset Distance to negative 0.1 (-0.1)
7. Then we'll set the Tolerance to 0.01 and then pick Offset.
8. Now we see that a second mesh was added to the Data Tree that is a negative offset of the first.



6.2 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 MESH Browser 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.

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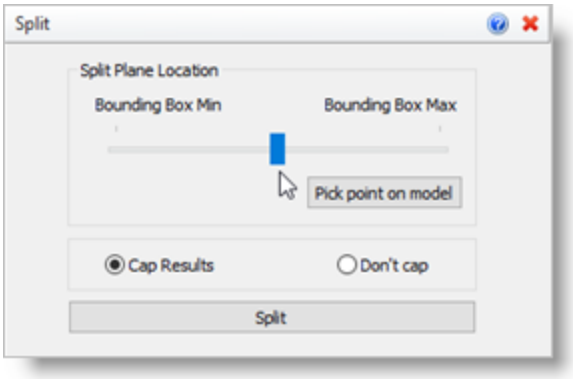
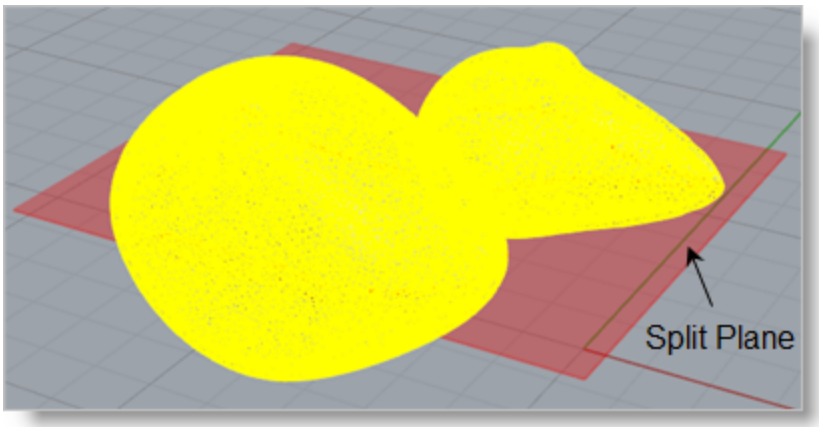
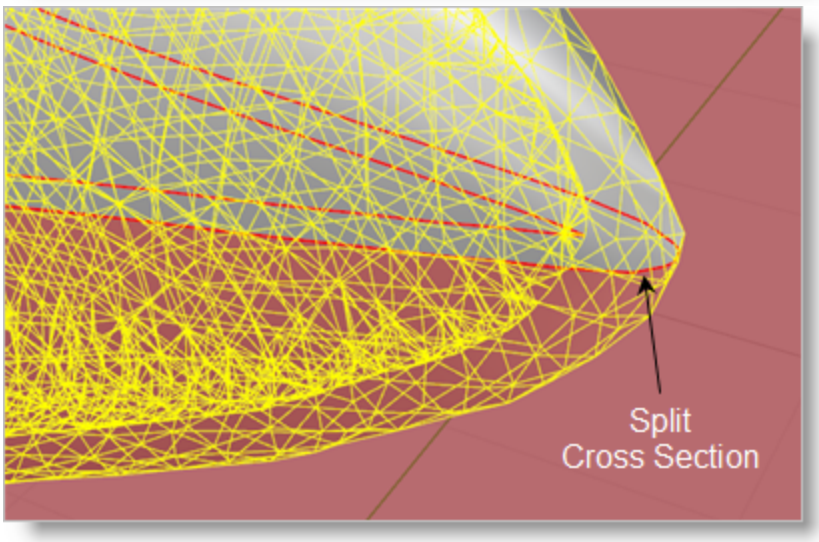
4. Move the Split Plane, shown graphically on the screen, by adjusting the slider between the Min and Max of the mesh's Bounding Box.







5. The Split Preview (section curves) are shown in **Red**, and also dynamically update as you adjust the slider. Move the slider until the section curves are located approximately as shown in the example above.

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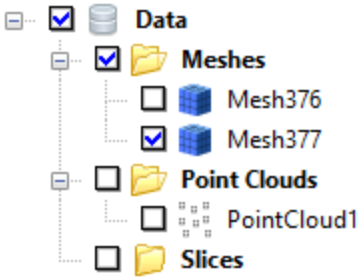
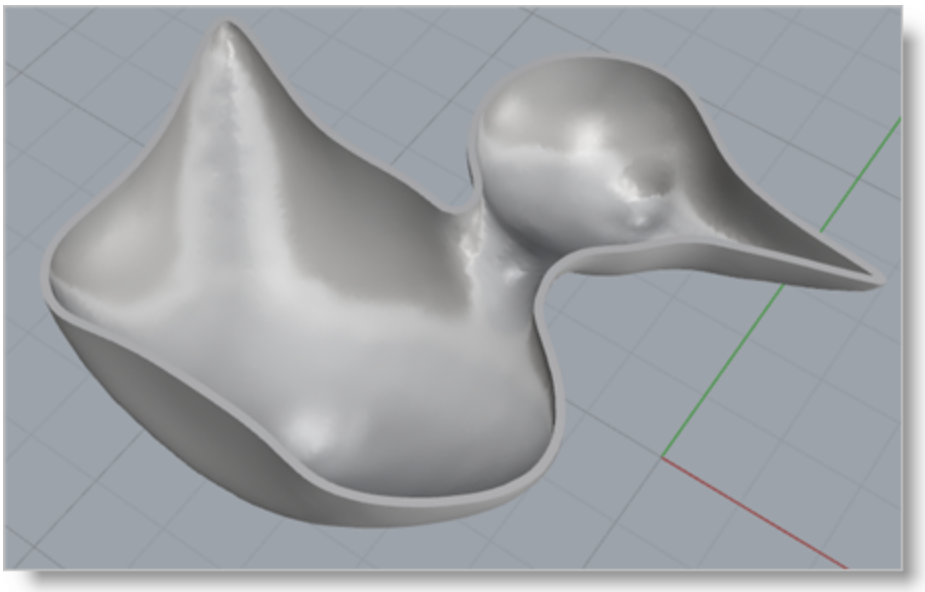
6. Now select the Cap Results option. Selecting this will ensure that once split, each section of the mesh will be a capped closed volume.
7. Now we pick Split.
8. Once completed, we'll uncheck the upper mesh to toggle its display off.



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.



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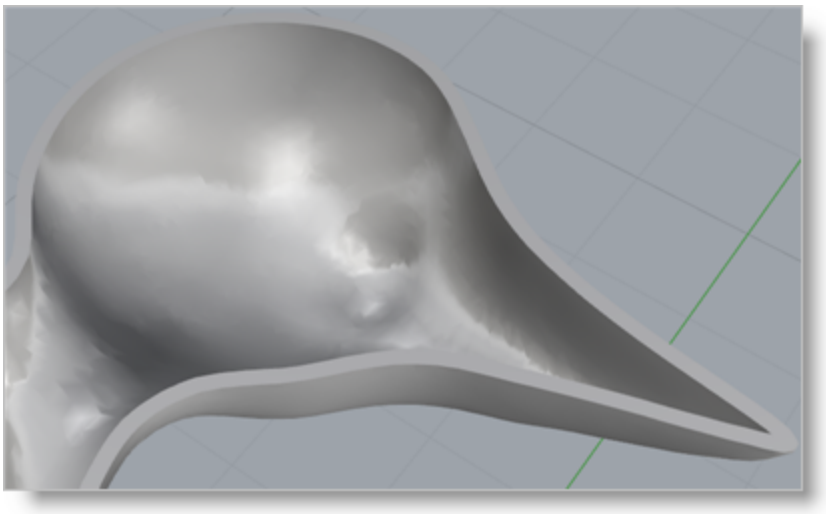

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Where to go for more help

Download this PDF Guide for a list of the available RhinoCAM Resources.

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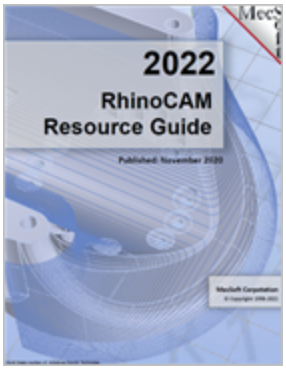

The 2022 RhinoCAM Resource Guide!

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Lists PDF downloads and Online resources including Quick Start Guides, Reference Guides, Exercise Guides, Tutorials and More.

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