

Guide for 3D Model Repair

3D models must be valid if your intention is to use them for physical output such as 3D printing. Before submitting your models to the AOC's technician-run printers, you should always do your cost estimates and validation checks in the appropriate software - see our 3D Printing Guide for a walk-through of this process. When validating, you may see messages like the ones pictures to the right. In these cases, follow this guide for tips on simple model repair done through **free, Mac and Windows based** software.

Error: STL has missing facets or reversed normals.

Catalyst Software for the AOC's Dimension printer

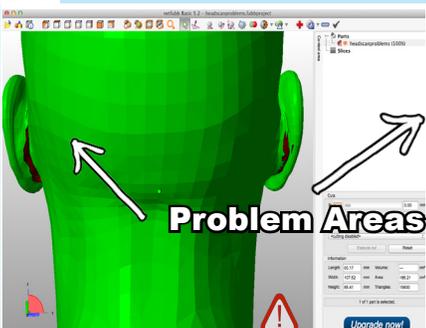


Objet Studio Software for the AOC's Objet printer

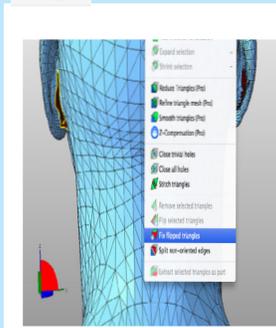
General Information:

- A model has to be watertight and closed in order to print. If you have holes or missing geometry, the software will alert you that you have **“missing facets”** or **“unclosed contours”**.
- **“Reversed normals”** are also common. This means all or parts of your model are flipped inside out.
- Always make sure your model meets the **size and thickness** requirements for the printer. The printer software will not always alert you if a part is too thin. Check the AOC website or the printer's documentation for these specifications.

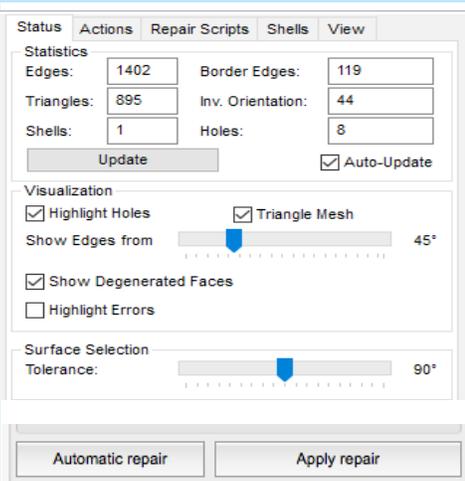
Netfabb Standard:



Right click anywhere in the Netfabb program and choose **“Add Part”**. Once your model loads in, you will see orange areas and red areas. Red areas are signaling that those parts of the model are invalid.

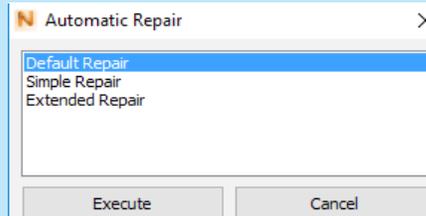


Click the red cross, on the right side of the tool-bar, to enter repair mode and begin fixing the model's errors. The model will turn blue and holes will be outlined in yellow.



In repair mode, you will see a repair panel in the lower right. Under **“Status”** you can view statistics about edges and holes.

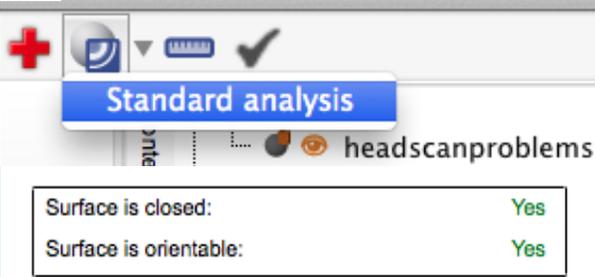
At the bottom of this panel, you will see an **“Automatic repair”** button. Pressing this will bring up Simple or Default repair.



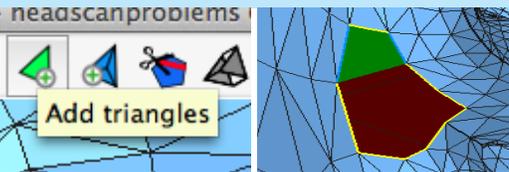
These repair functions both work globally to prepare the model for physical output.

“Simple repair” focuses only on filling holes and fixing flipped triangles (areas of geometry with inverted normals). **“Default repair”** is more in-depth and also removes doubled triangles and tiny flecks of geometry. **“Extended repair”** looks at even more factors and can take a longer time to run. Execute one of the functions, hit **“Apply repair”** and delete the old part. This will close repair mode and your model will turn orange again. Check to make sure all red areas have turned orange.

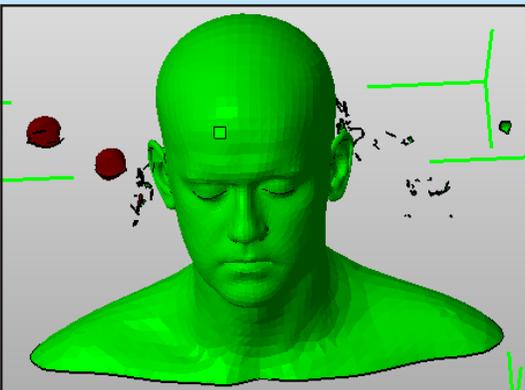
Netfabb Standard continued:



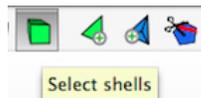
Run an analysis on your part. This will bring up a panel in the lower right corner that will give you information about your model and will let you know if the surface is closed. It will also tell you the **size and volume**, which can be useful for 3D printing cost estimates. Remember, you can also go to “**Part**” → “**Convert Units**” to scale your object to millimeters or inches. To free scale your part, go to “**Part**” → “**Scale**” and type in your scaling factor or target size with “**Fix scaling ratio**” on.



If you encounter a stubborn hole that won't auto-fill, you can use the “**Add triangles**” tool to draw polygons from edge to edge to bridge that hole. Find this tool in the top menu row while in repair mode.



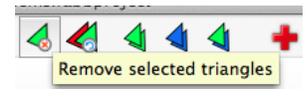
Sometimes parts will not validate because there are unwanted floating bits of geometry, referred to often as “outliers”, included in the file. Follow these steps while in repair mode to quickly rid the file of this geometry:



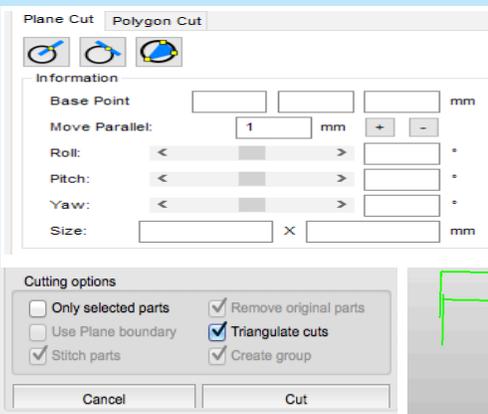
1. Hit the button above and select the object you want to keep.



2. Click the button above to select the inverse.

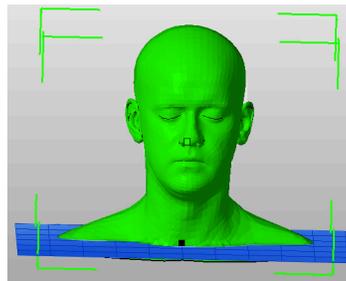


3. Click the button above to delete the floating geometry.

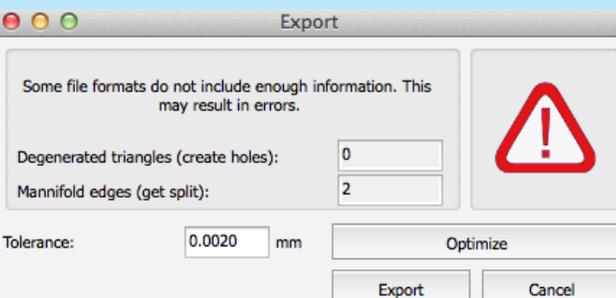


Another useful feature is **planar cuts**. This feature allows you to trim off an uneven bottom in a 3D scan or to simply cut your large model into small parts that can be 3D printed and adhered together later.

In normal view, you will see the “**Cuts**” panel on the right side of the software. Click in the box next to one of the three planes to activate a preview. You can then use the sliders to position your plane before hitting “**Execute Cut**”.



On the next screen, you will see a blue plane positioned through your model. If you are happy with the placement, press “**Cut**”. Your model is now two separate parts under the parts list in the upper right corner. Feel free to right-click the parts you don't want in this list and click “**Remove**”.



Once your file is valid and you are done editing, right-click the part in the parts list and choose “**Export part**” → and then choose your file format. STL files are used for 3D printing at the AOC. OBJs or VRMLs are for full-color printing.

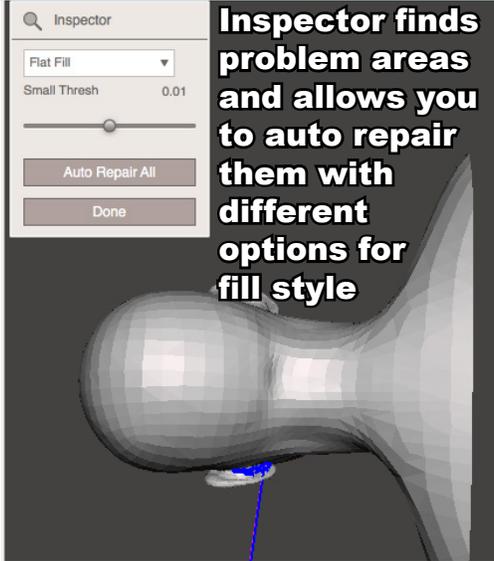
If you get a message alerting you to a problem with your file as seen to the left, hit “**Optimize**” and that should remove manifold edges. The file can now be exported with the “**Export**” button.



Meshmixer has a lot of the same capabilities as Netfabb such as hole filling, auto repair, and analysis tools. In most cases all of the basic hole filling and normals-inversions can be done in Netfabb, but Meshmixer does have many unique features that are outlined below.



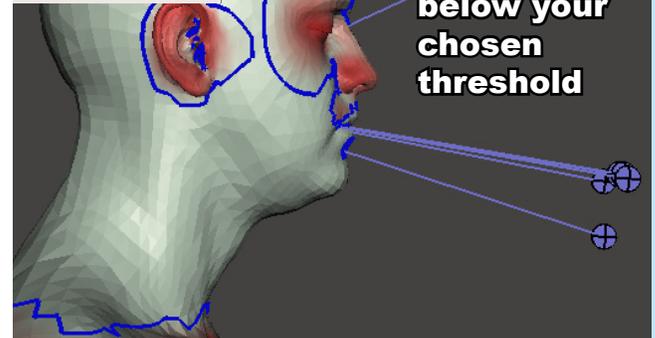
Import your model. Jump first into the “Analysis” panel by clicking on the icon shown above. A panel will spawn in the upper left corner displaying many analytic functions including “Inspector”, “Thickness”, and “Measure”.



Inspector finds problem areas and allows you to auto repair them with different options for fill style



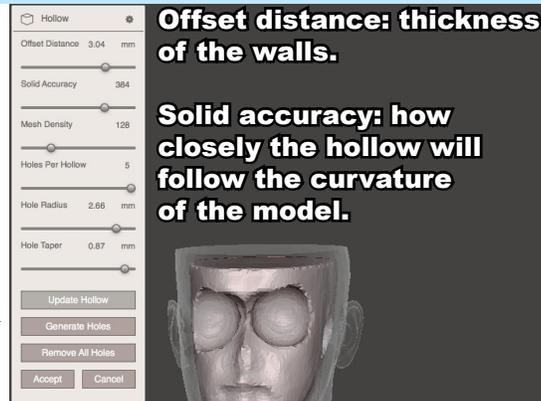
Thickness allows you to see which areas fall below your chosen threshold



Helpful for 3D printer constraints

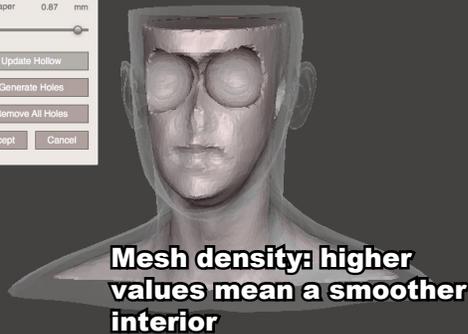


The “Edit” panel has a lot of functions to physically change your model. Shown to the right is the “Hollow” function which will save you lots of money in 3D printing and is essential for larger Form 1+ prints.



Offset distance: thickness of the walls.

Solid accuracy: how closely the hollow will follow the curvature of the model.



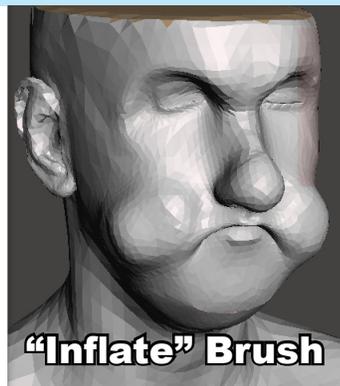
Mesh density: higher values mean a smoother interior



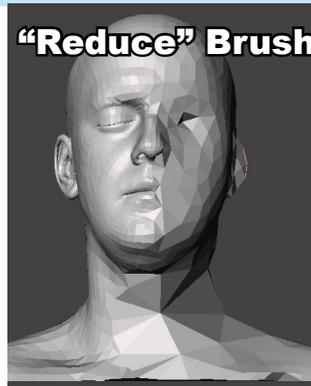
Drainage holes are needed for hollow Form 1+ models. Choose your number of holes per hollow, and then press “Generate holes”. Place the red dots by dragging them and press “Accept” to hollow.



The “Sculpt” panel allows you to smooth, decimate, add on to, or shrink parts of your model with an editable brush.



“Inflate” Brush



“Reduce” Brush

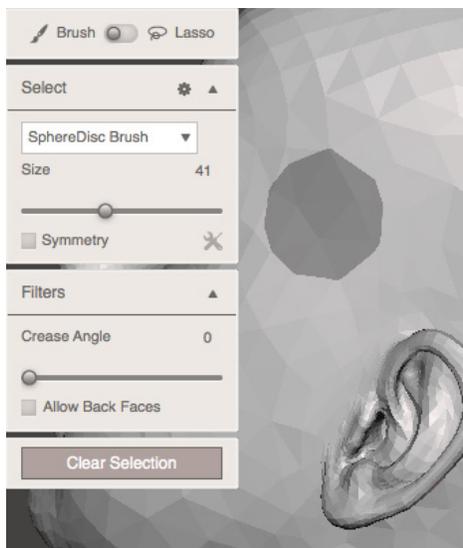
If this sculpting based modeling appeals to you, check out the free Mac-friendly software called **Sculptris** found at Pixologic.com which allows you to start with a 3D ball of clay and use advanced brushes to create a soft-edged model. For hard-edged, extrusion-based modeling from scratch, check out **123Ddesign**, another free Autodesk program.

Meshmixer continued:



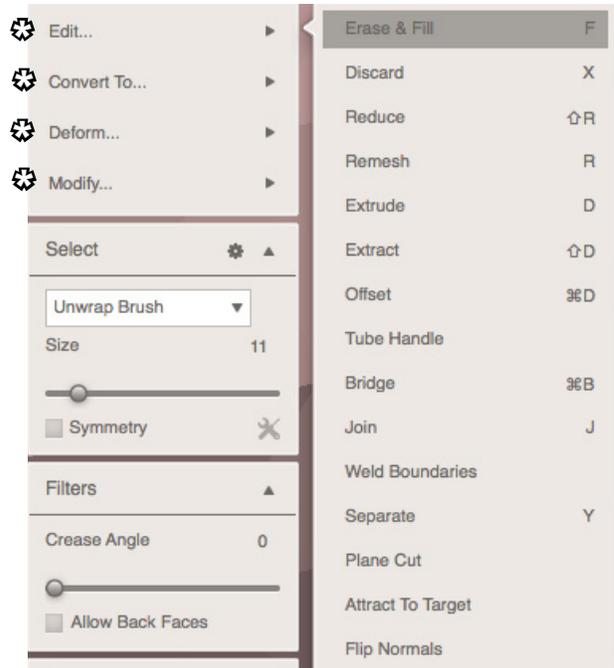
Within the “Select” panel, you can select portions of the model with a brush and then modify that section in a bunch of different ways.

For repairs, see the “Edit...” sub-menu for filling holes, reducing resolution, flipping normals and joining areas of the model.



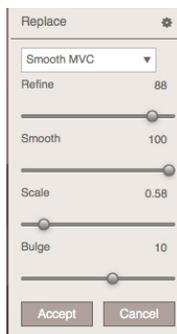
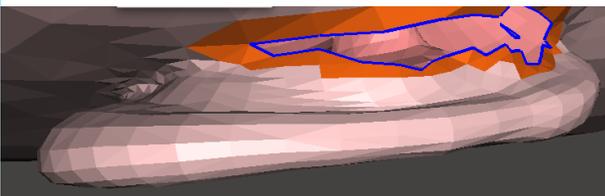
Select the area with the brush and the sub-menus will show up

Sub-menus:



Sometimes, the “Auto repair all” function in the “Analysis” panel can miss certain holes due to an irregularity in the shape or path of the hole. If you encounter a stubborn hole like this, you can fill it in the “Select” panel.

Double-click the edge of the hole to select the entire hole. Next, choose “Edit...” and “Erase & Fill”.



Choose a filling method from the drop-down box. Choose your settings with the sliders and hit accept.

If your model is a textured OBJ file, the software will also try to copy the surrounding color info.



When you are finished repairing and editing your model, you must export it as an STL for 3D printing at the AOC. Go up to “File” and then “Export”.

It is worth mentioning that there are plenty of websites that offer hundreds of free STL models for download. This could be a good place to start if you want to try 3D printing but you have no modeling or 3D scanning knowledge. A lot of the models on these sites are prepared for 3D printing and will require little to no repair.

Sites with free downloadable models:

- **Thingiverse.com** - most comprehensive and user-friendly
- **Sketchfab.com** - need to make a free account; lots of textured models
- **YouMagine** - also has a great blog and tutorials
- **Google Warehouse** – Sketchup needed to export to OBJ or STL



Download This Thing!