

### Simple Healing with MeshLab


Before you generate your G-Code in Repetrel, you begin with a STL file. Some STL Files that you will work with may have some imperfections that require repair prior to converting to a printable G-Code.

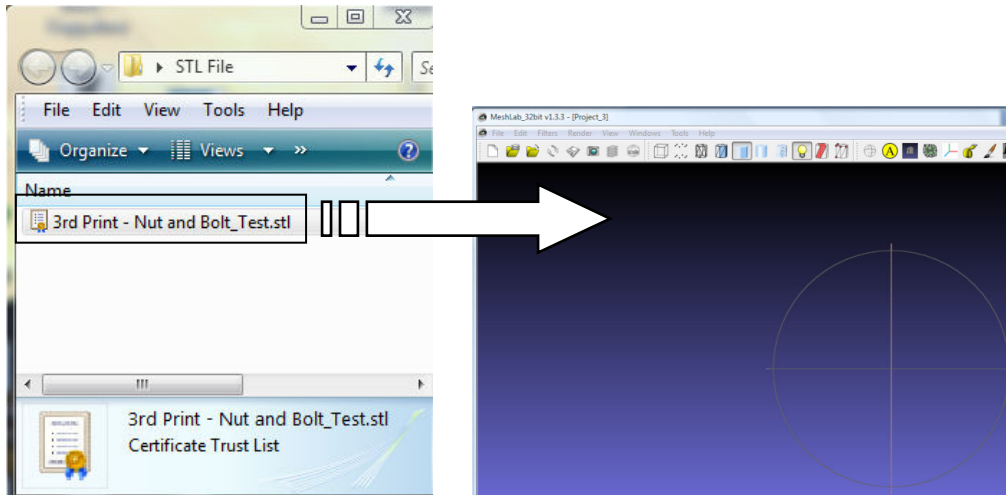
MeshLab is a good tool to use to correct your STL 3D Object file before you convert it to G-Code. We call the process of repairing a STL file in MeshLab as “Healing” your STL file. The basic process of healing STL files in MeshLab is pretty simple.

The following details how to use basic MeshLab tools to heal your STL file.

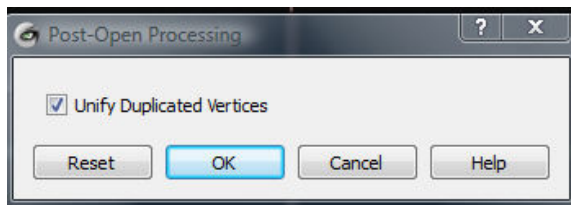
See Hyrel’s YouTube Video. <https://www.youtube.com/watch?v=eLGIRAgLz4w>

### Open and view your 3D Object

1. **Open the MeshLab application** by Double Clicking on the MeshLab Icon .
2. **Import your STL File into MeshLab** by either method below.
  - a. Use traditional method from the menu path: **File => Import STL** or
  - b. **Drag and Drop your STL** from Windows Explorer into an active session of MeshLab.

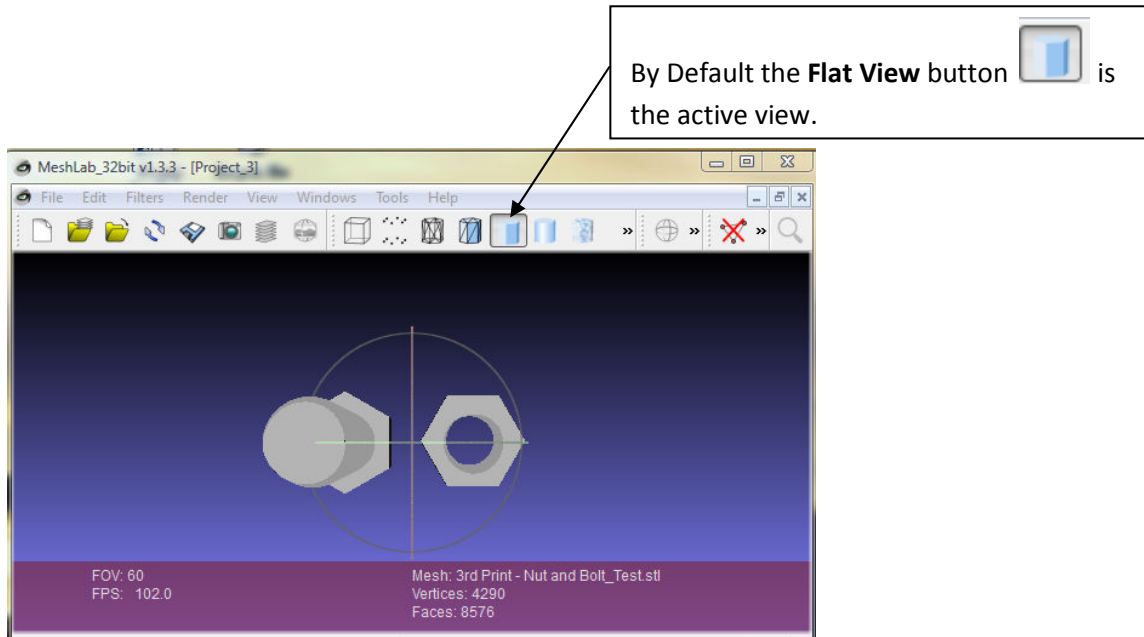


3. The Post Open Processing Prompt appears.

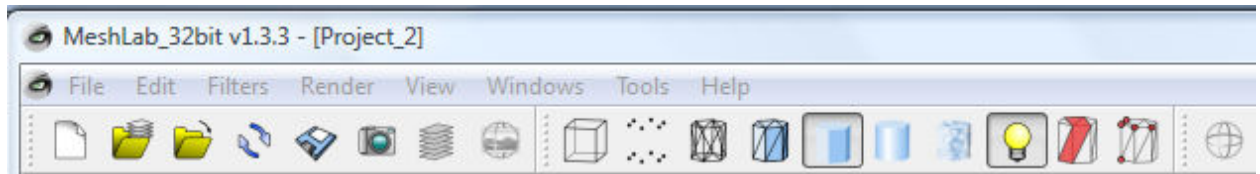


- a. Leave the **Unify Duplicate Vertices** checked.
- b. Click **OK**.

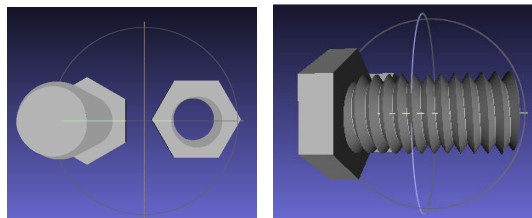
Your Object Model appears in MeshLab.




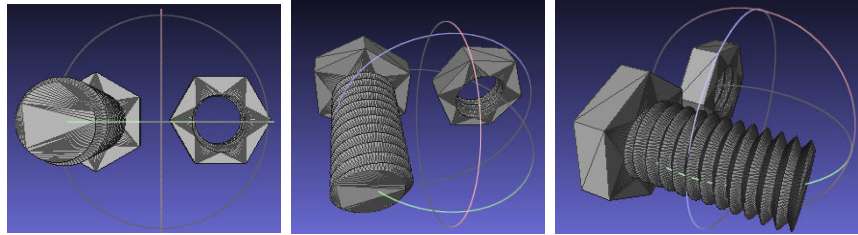
### MeshLab Tool Bar and Menu



- 4. **To Navigate** different views of your STL Model:
  - a. **Left Click, Hold, and Drag** the model around the screen.



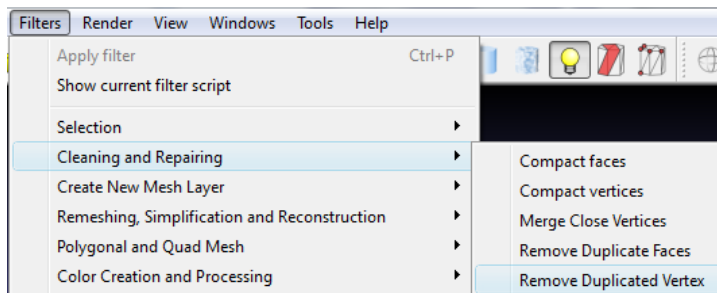
5. You can see every triangle that defines your mesh by clicking the **Flat Lines** button .



### Healing the STL File

To begin the Clean up and Repair Process take the following steps.

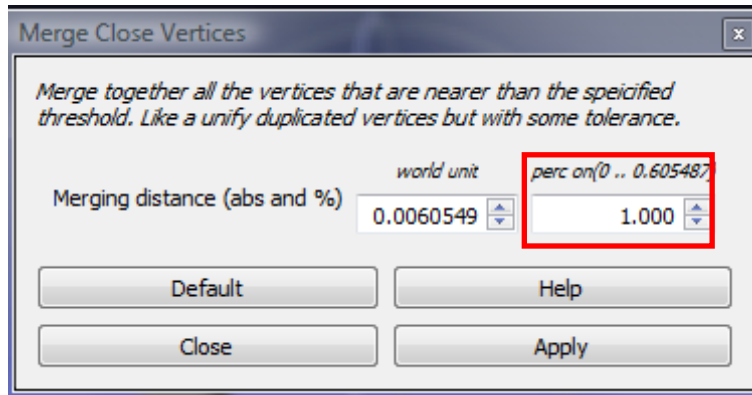
1. Use Menu Path: **Filters => Cleaning and Repairing**
  - a. Select Option (5) **Remove Duplicated Vertex**.



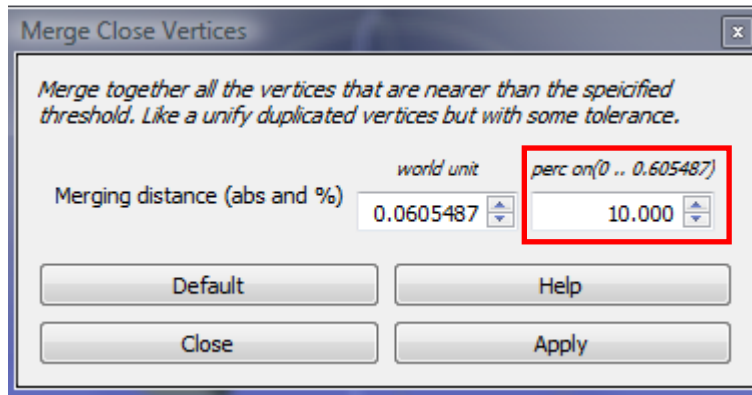
Note: After each repair, MeshLab displays a short confirmation at the bottom left of the screen: **“Filter has successfully completed”**.

2. Use Menu Path: **Filters => Cleaning and Repairing**
  - a. Select Option (4) **Remove Duplicated Faces**.
3. Use Menu Path: **Filters => Cleaning and Repairing**
  - a. Select Option (3) **Merge Close Vertices**.

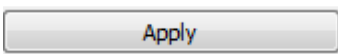
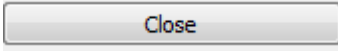
The following window appears.



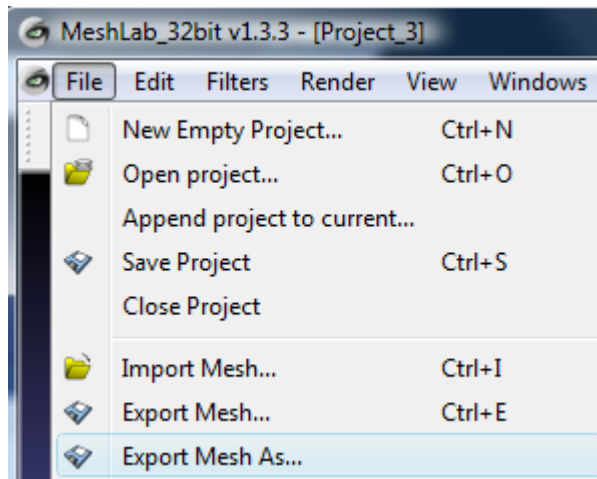
4. Enter 10 – in the Percentage Box to the right side.



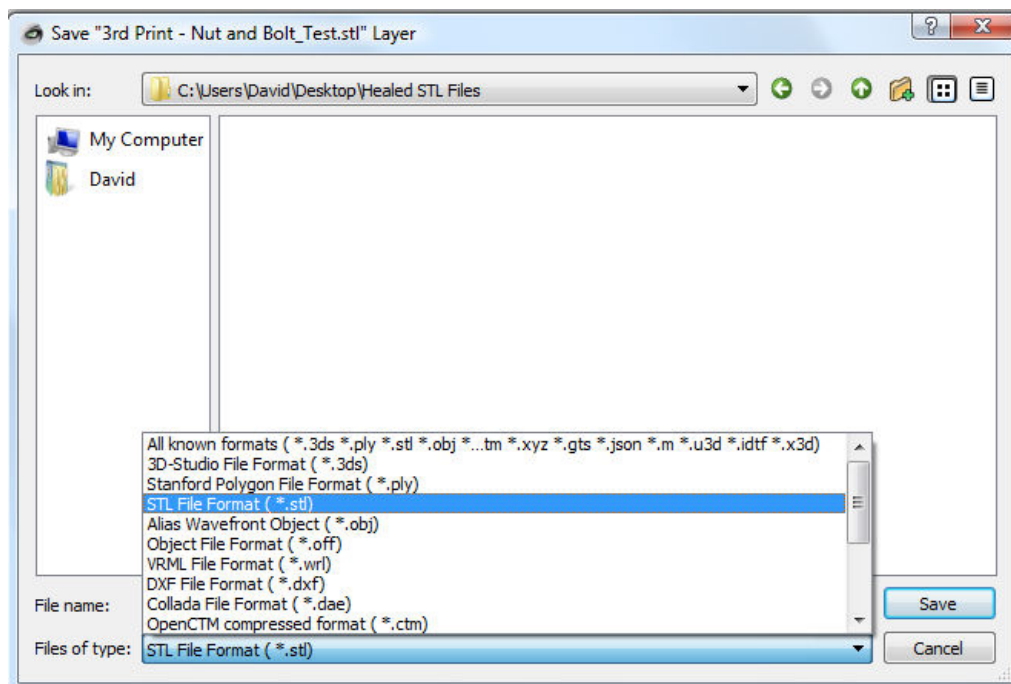
- Typically 10% is a good number.
  - This setting is mainly helpful for a very detailed 3D Model Build.

- a. Click  .
- b. Click  .

5. **Export the new Mesh File as a STL File**, but rename the STL file to indicate you healed it.
  - a. Use menu path: **File => Export Mesh As**

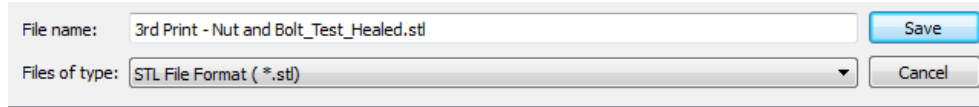


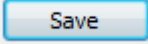
The Save File Window appears.



- b. **Choose a folder** to save the new STL File.
- c. **Click the Files of Type Pull Down window and Select the STL Format.**
  - Helpful when selecting an STL file name that already exists in a folder.
- d. **Rename the STL File** to indicate that you have **"Healed" the file.**
  - To help identify healed STL files, **add "Healed"** at the end of the file name.

- Example: **3<sup>rd</sup> Print – Nut and Bolt\_Test\_Healed.STL**



6. Click  .
7. **Exit MeshLab**, using menu path: **File => Exit**

---

For more information on MeshLab, visit their Web Site. <http://meshlab.sourceforge.net/>

Explore the various YouTube Video Tutorials on specific features of MeshLab.

YouTube has basic introductions to Meshlab as well as more advanced MeshLab Repairs.

---

### **Summary of remaining steps to Print in Repetrel**

(See **REPETREL Printing Steps** document.)

1. **Open the Healed STL** in Repetrel.
  - a. **Remove your old STL File** from the Object Tree.
  - b. **Right Click** on the **old STL** file, then **Select Delete**.
2. **Select or Verify your Slicer Recipes** (for ABS Printing Material - example)
  - **Click the Slicer Tab** to see Recipe selection.
3. **Slice the Healed STL file** to generate the printable **GCode**.
  - **Click the Process pull down button**.
  - **Click Slice with Slic3r**
4. **Click the Print button** at the top main menu to print the GCode.
  - (Assumes a **Calibrated Z Height** - distance between print head nozzle and print bed.
  - **Option:** While printing your first layer, you can **adjust your Z Height** using:
    - **Z Height Fine Tune Adjust buttons (+ or -)**.
      - To further dial in your extruded filament bead.