Step 1: Install FreeCAD

To start off, make sure you have FreeCAD downloaded and installed

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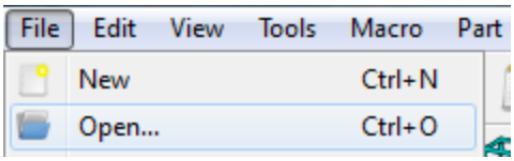
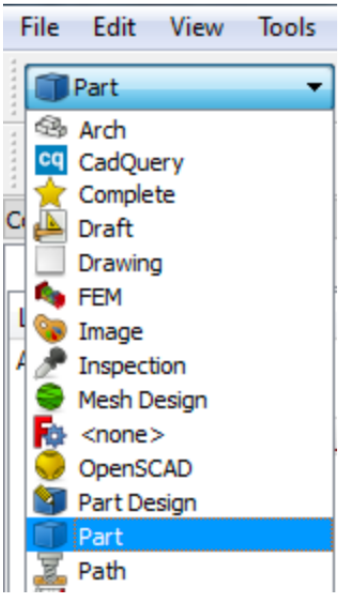
Step 2: Import STL

Open your STL part in FreeCAD

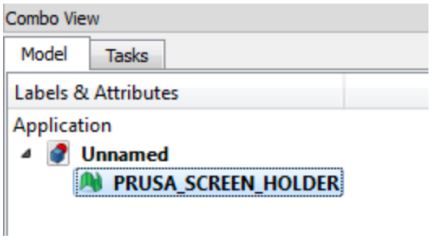


Go to "Part" workbench

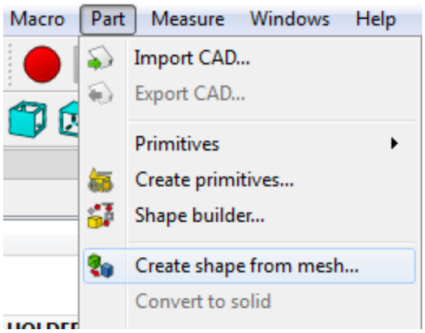


Step 3: Convert from Mesh to Shape

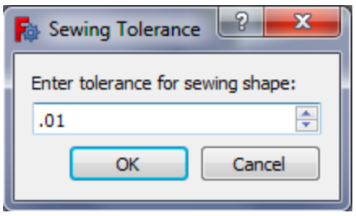
Now, select the imported STL part in main view or the model tree.



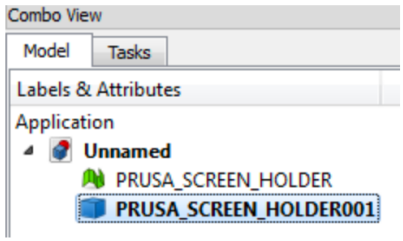
Go to Part / Create shape from mesh...



Set your mesh sewing tolerance in the pop up window



FreeCAD will now create a new shape in the model tree



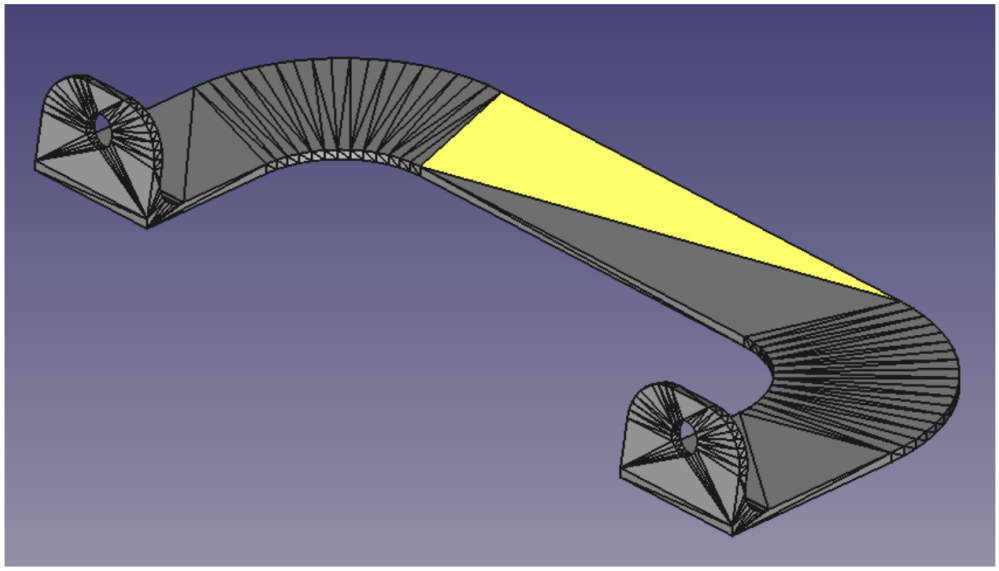
Limitations:

- There will be no analyzing or validating of the mesh object.
- Analyzing and repairing of the mesh (if needed) should be done manually before conversion.
- Appropriate tools are available in the Mesh Workbench.

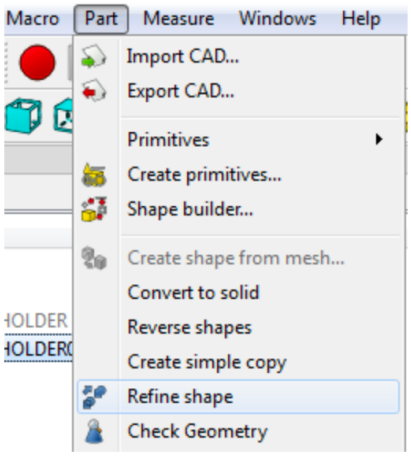
If you're looking for a fast solution to get a Step exported now would be the time to export it. However, if you want a less triangulated solid or Boolean operations performed, then follow these next steps...

Step 4: Simplify Shape

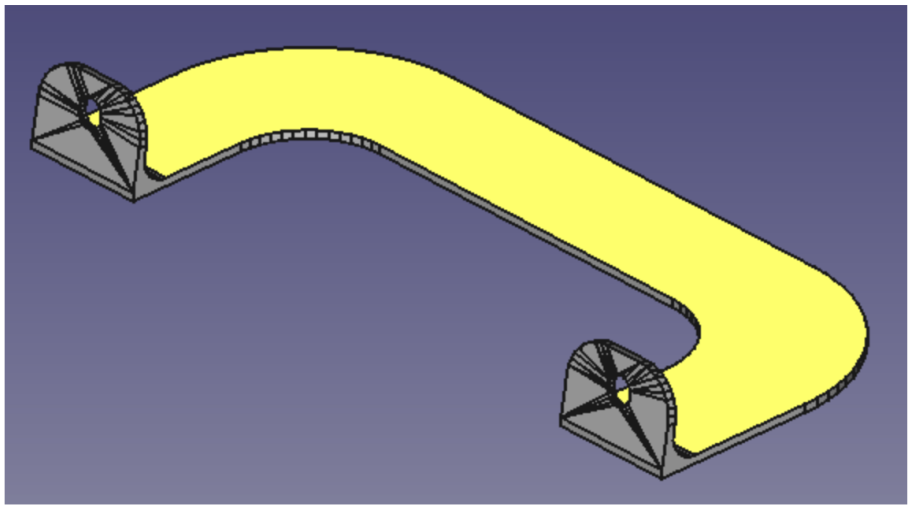
So you now have a shape to work with but as you can see there's still allot of triangulated faces



To simplify this part, now select the shape and go to Part / Refine shape



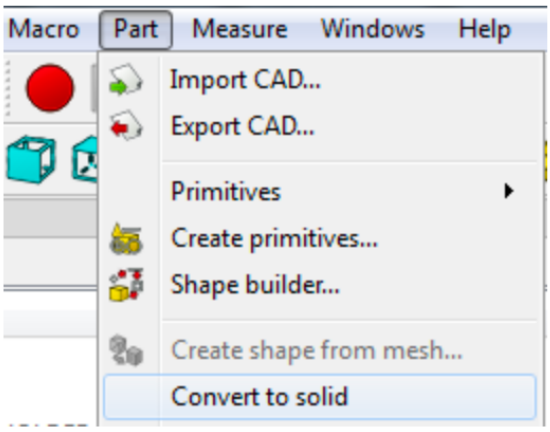
Yet another part will be added to your model tree, now simpler!



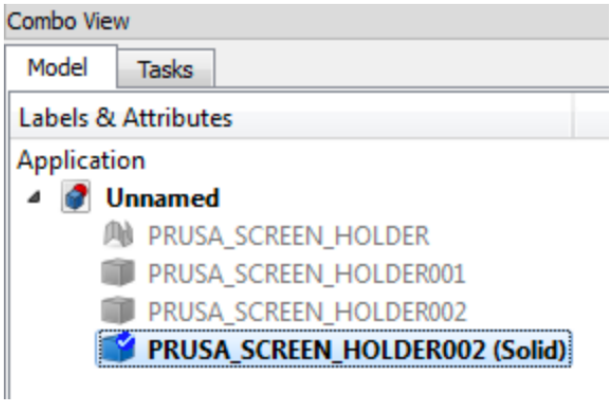
Step 5: Convert Shape to Solid

Now, to convert the shape to a solid that Boolean operations can be performed on...

Select your now simplified part and go again to the Part menu, this time selecting "Convert to solid"



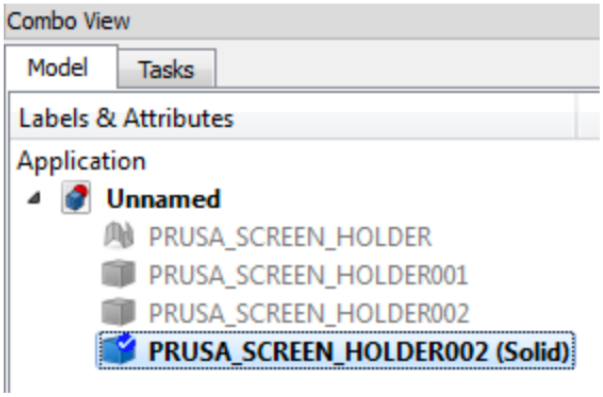
And one more time a new solid will show up in your model tree



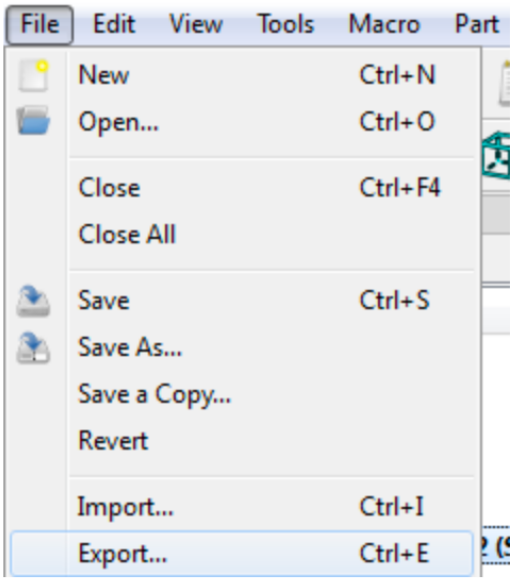
Quick Tip: Selecting a part in the model tree and pressing space will hide that selected part. This makes it easier to see the newly made shape. Hidden parts will now appear greyed out as seen in the above picture.

Step 6: Export

Select the model you want to export, in this case the fourth part in the model tree



File / Export



And you're done! You now have a Step file from a STL file!