3D printing on SLA machines

Introduction

by

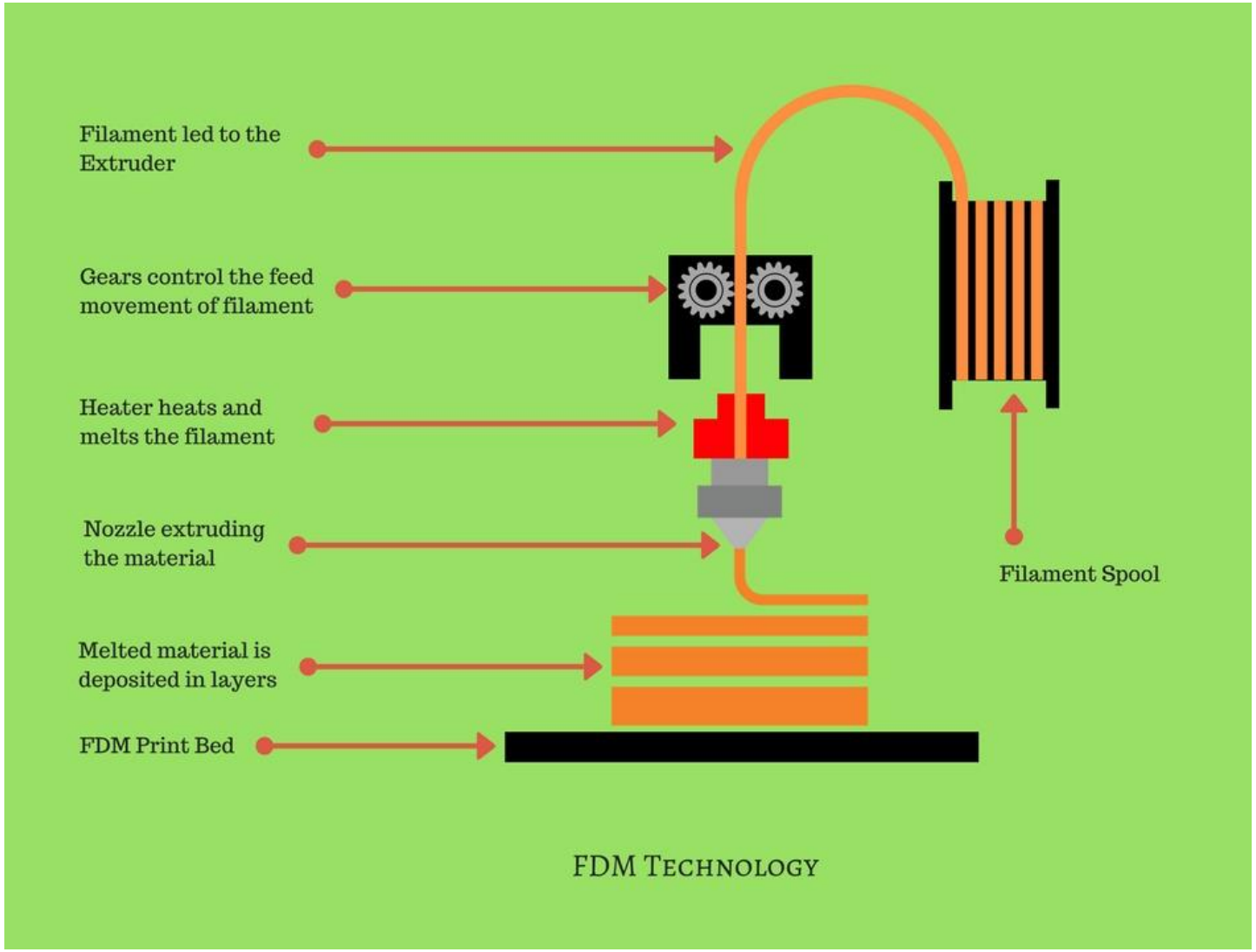
Niloufar Emami

Additive manufacturing: 3D printing

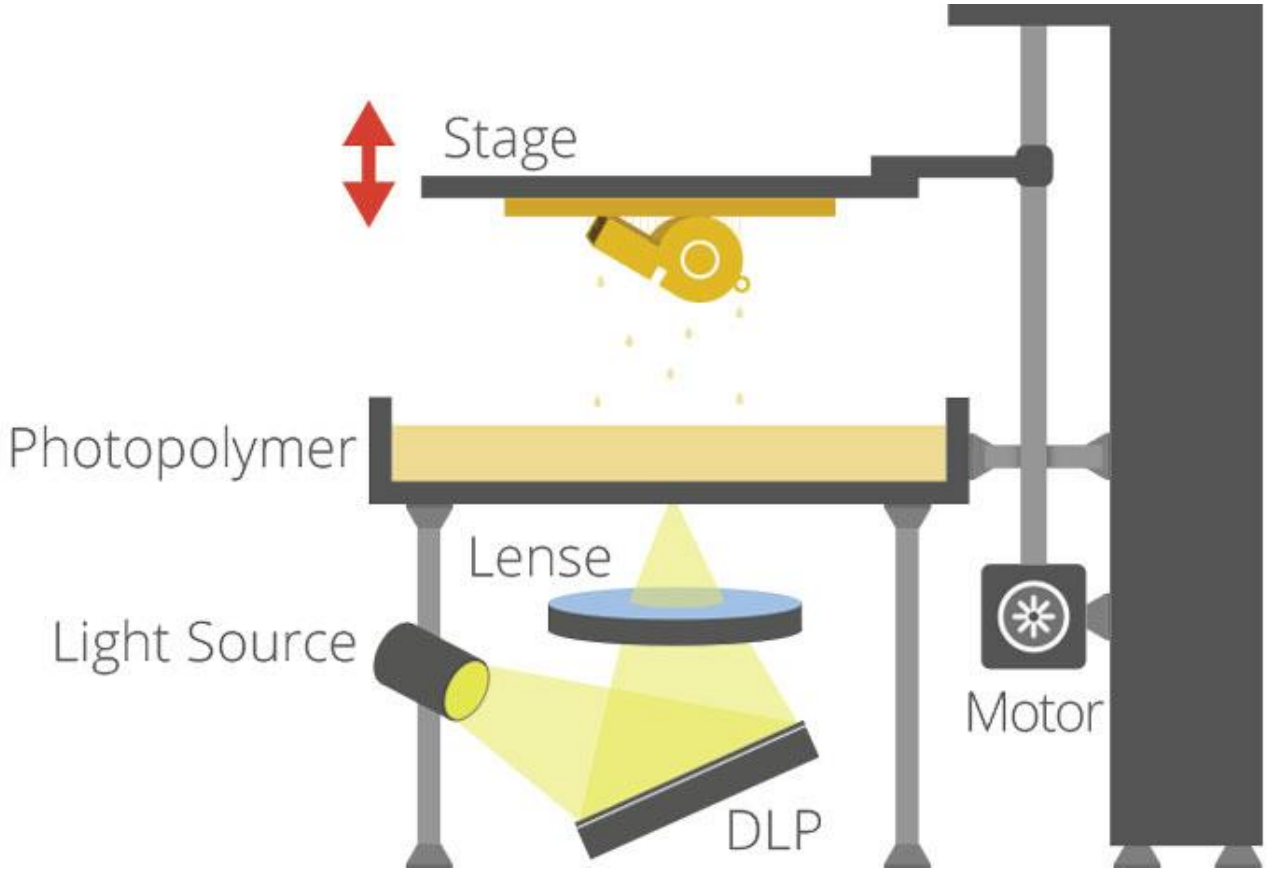
- An object is created by adding material layer by layer in 3D printing
- The benefits of 3D printing is that you can prototype complex geometries without using a mold or other techniques
- There are an array of materials that can be 3D printed

Fused Filament Fabrication 3D printing

- FFF (Fused Filament Fabrication) or FDM (Fused Deposition Modeling). There is no difference between the two, it's only two different trademarks.
- The filament is melted and deposited onto a bed.



Stereolithography (SLA) 3D printing



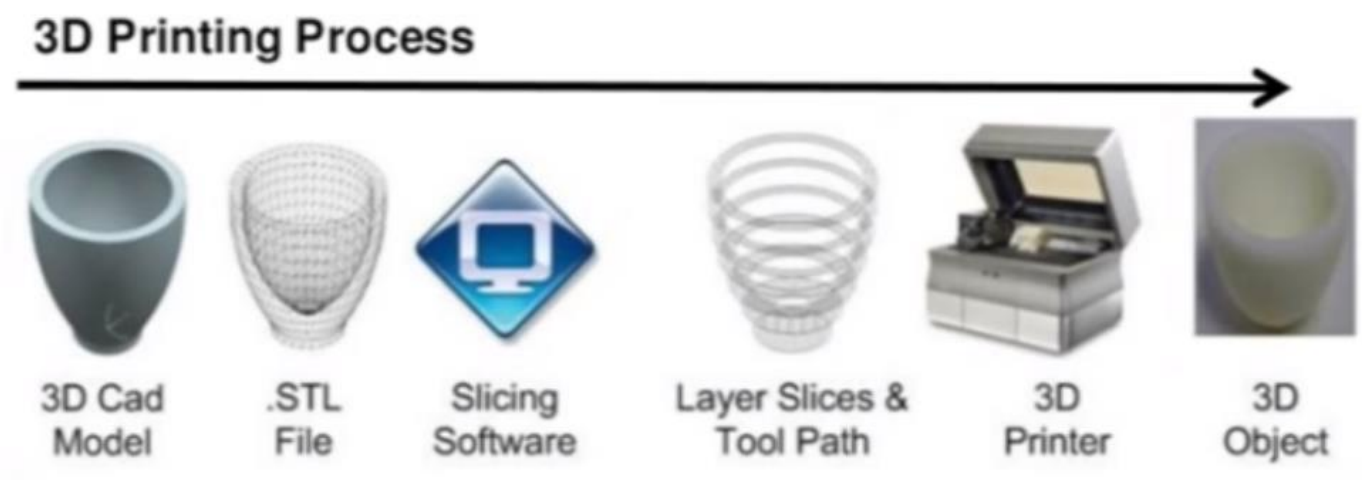
In SLA printing, an ultraviolet (UV) laser is focused onto a vat of photopolymer resin. The resin is sensitive to UV, and is solidified when the UV hits it. The CAD model controls where the UV is being projected, therefore different shapes can be 3Dprinted.

3D printing procedure

IBM Market Development & Insights

IBM

3D Printing begins with a CAD 3D Design Model, which is then used to send specific instructions to the printer about the amount, location and type of material to use.



Using a 3D digital CAD model as the "blueprint," successive layers of material are precisely deposited or fused by a computer controlled "print head" into the desired 3 dimensional shape. No machining of the part is required, thus no raw material is wasted.

Step 00: create your model in a CAD software

- You can use any software (Rhino, 3dsMax, etc) to create your object

Or

- You can go to websites that have free objects and download them

https://www.thingiverse.com/

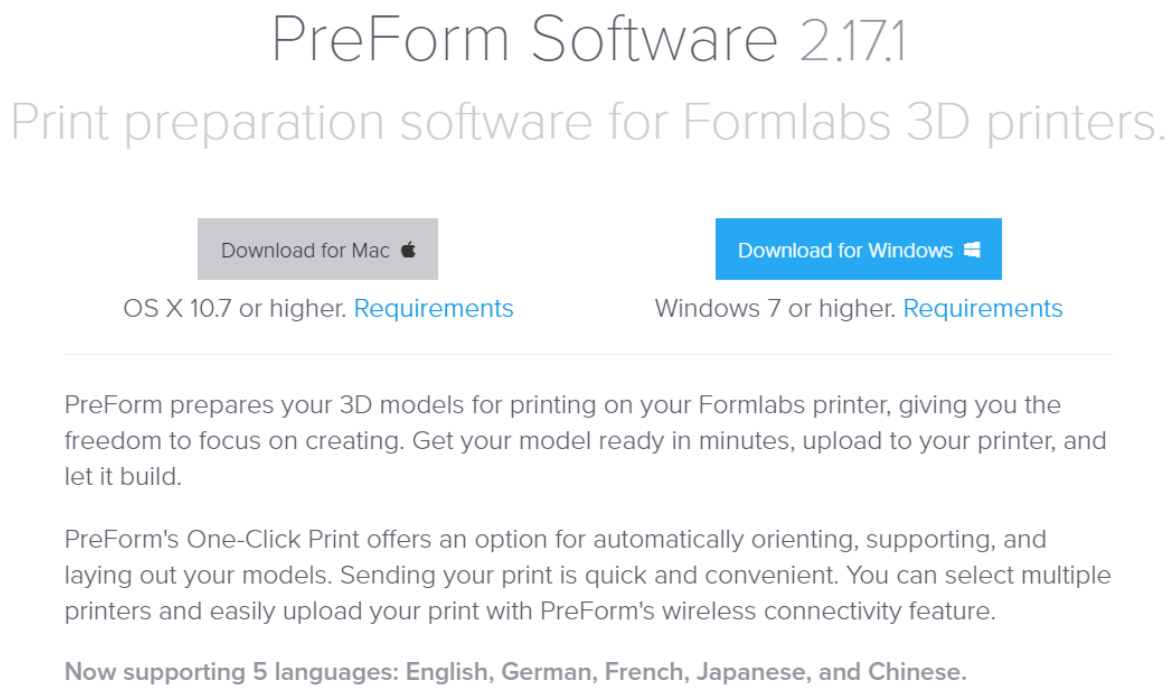
https://www.myminifactory.com/

Step 01: export your CAD model as .stl

- Once you have your model, you can export it from Rhino as an “.stl” file

Step 02: open the file in a slicing software

- As said before, 3D objects are created through layering. Printer adds one layer of the object at a time until the full object is formed.
- We will be using Formlab 2 which is a SLA 3Dprinter, and the slicing software that we use is “PreForm.”
- https://formlabs.com/tools/preform/



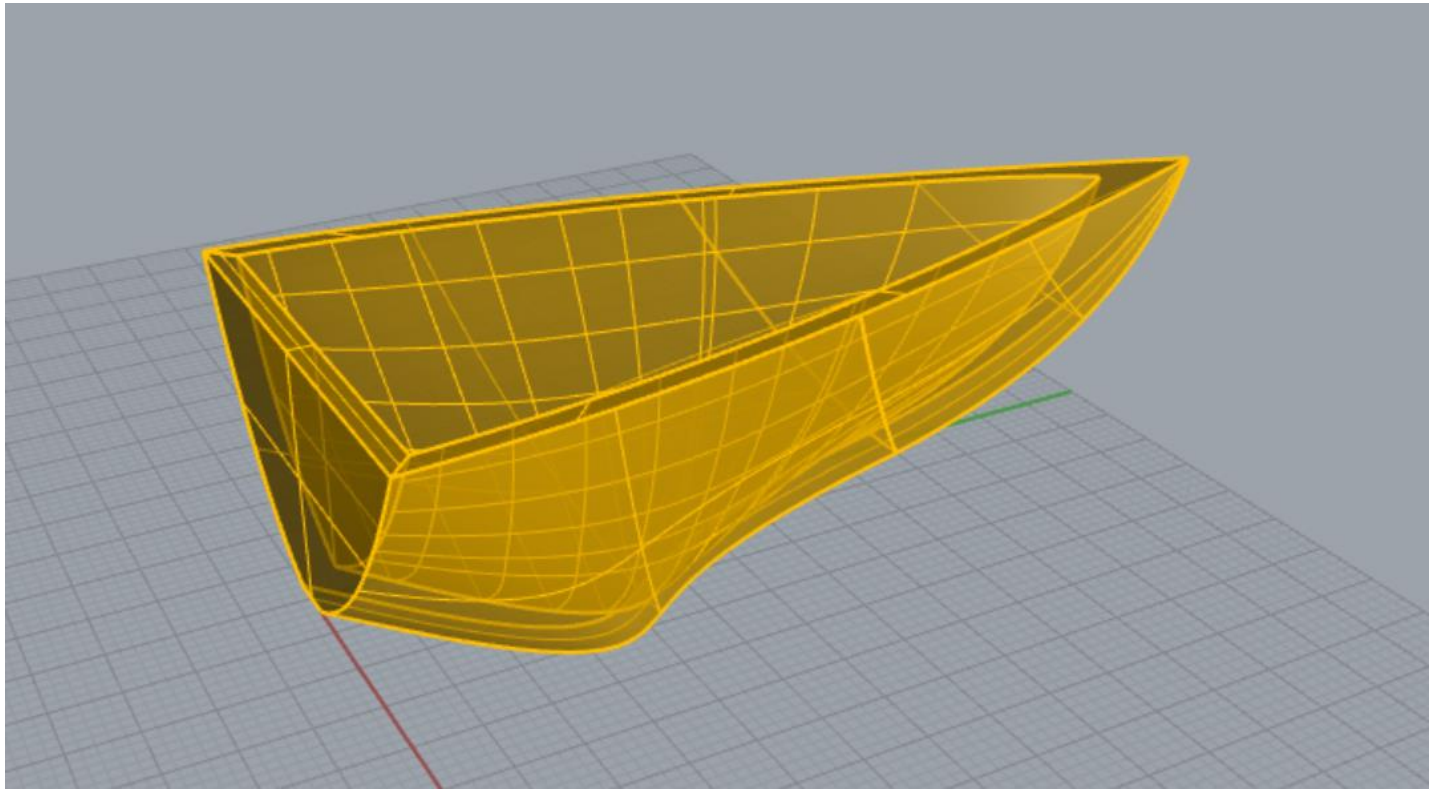
Step03: Send the G-code created through the slicing software to the 3D printer

- Hit print
- It shows you an approximation of the time needed for printing
- After it is completed, the object will be removed from the bed and cleaned in the alcoholic bath. Afterwards, it will be cured.
- Voila! Your object is ready

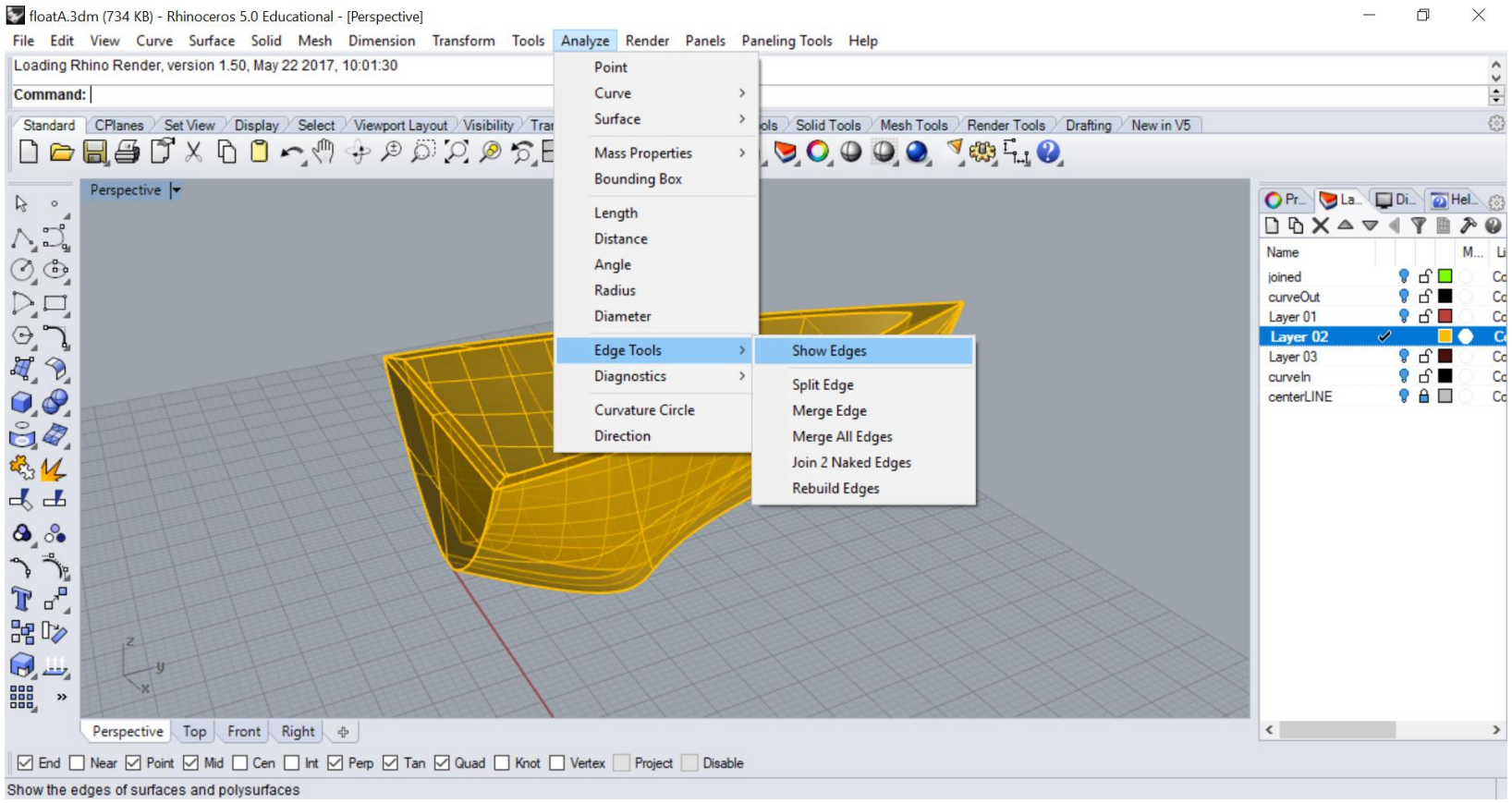
Tutorials on an example file

Check files in Rhino

After you have modeled a form in Rhino, join the surfaces.

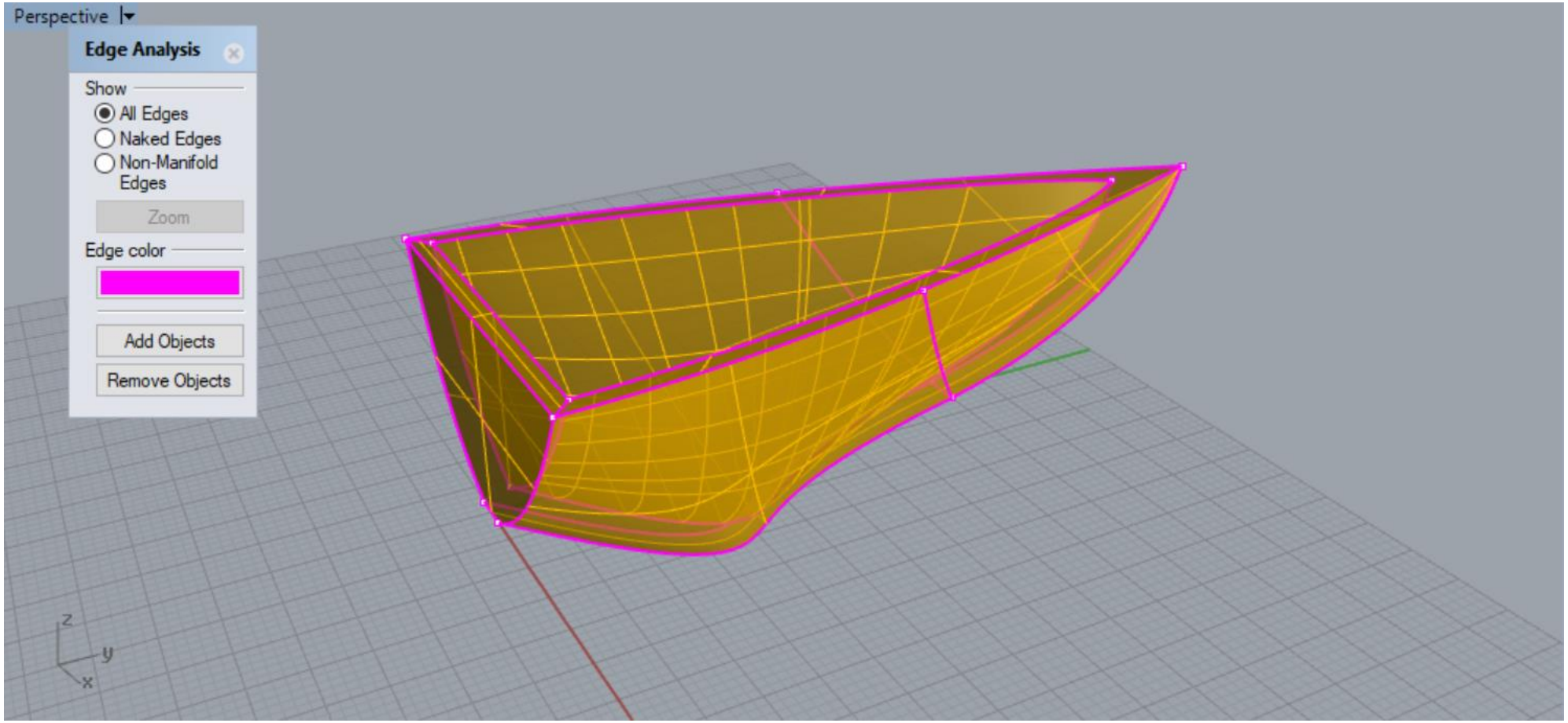


Go to Analyze, Edge Tools, show edges

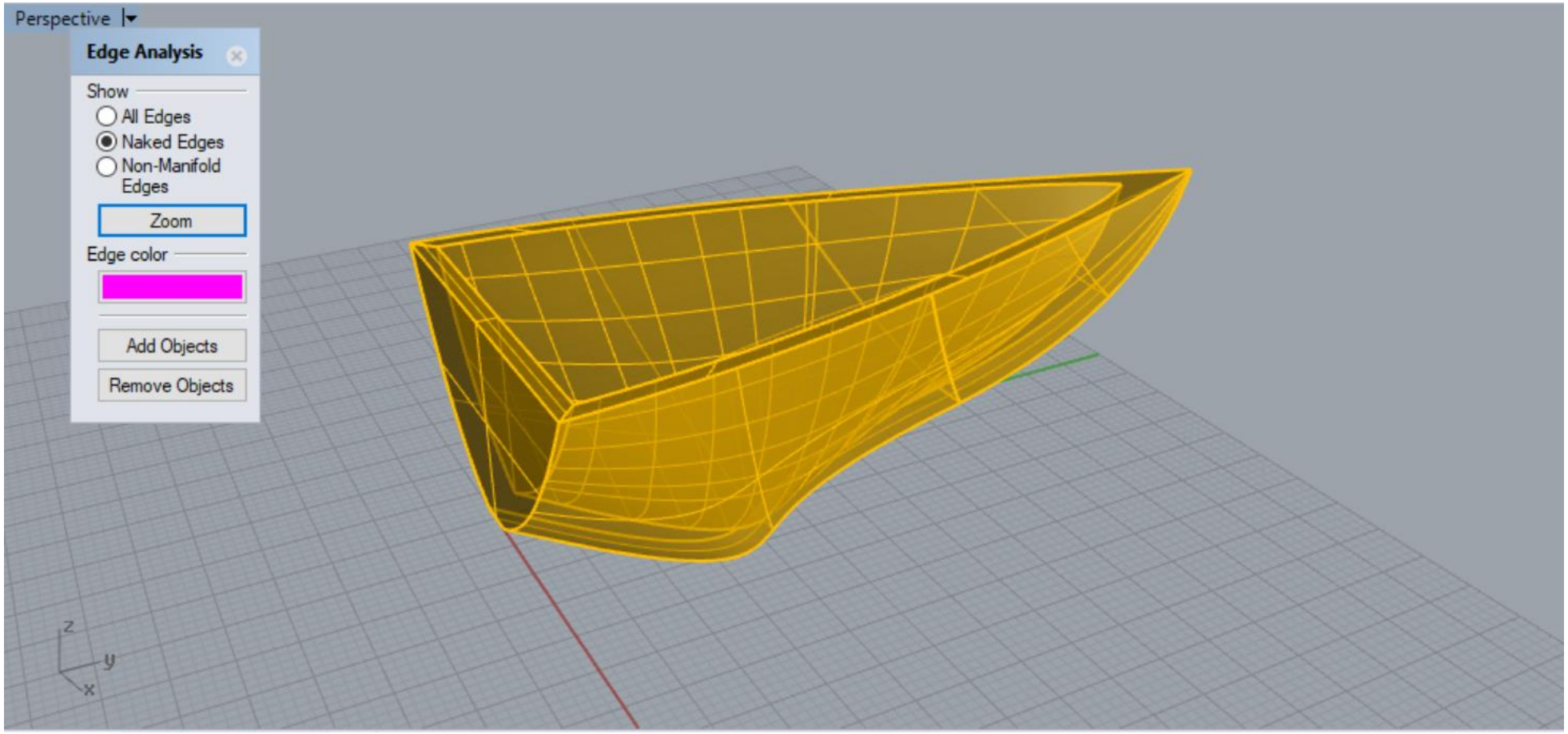


You want to make sure it is water tight (all edges are closed)

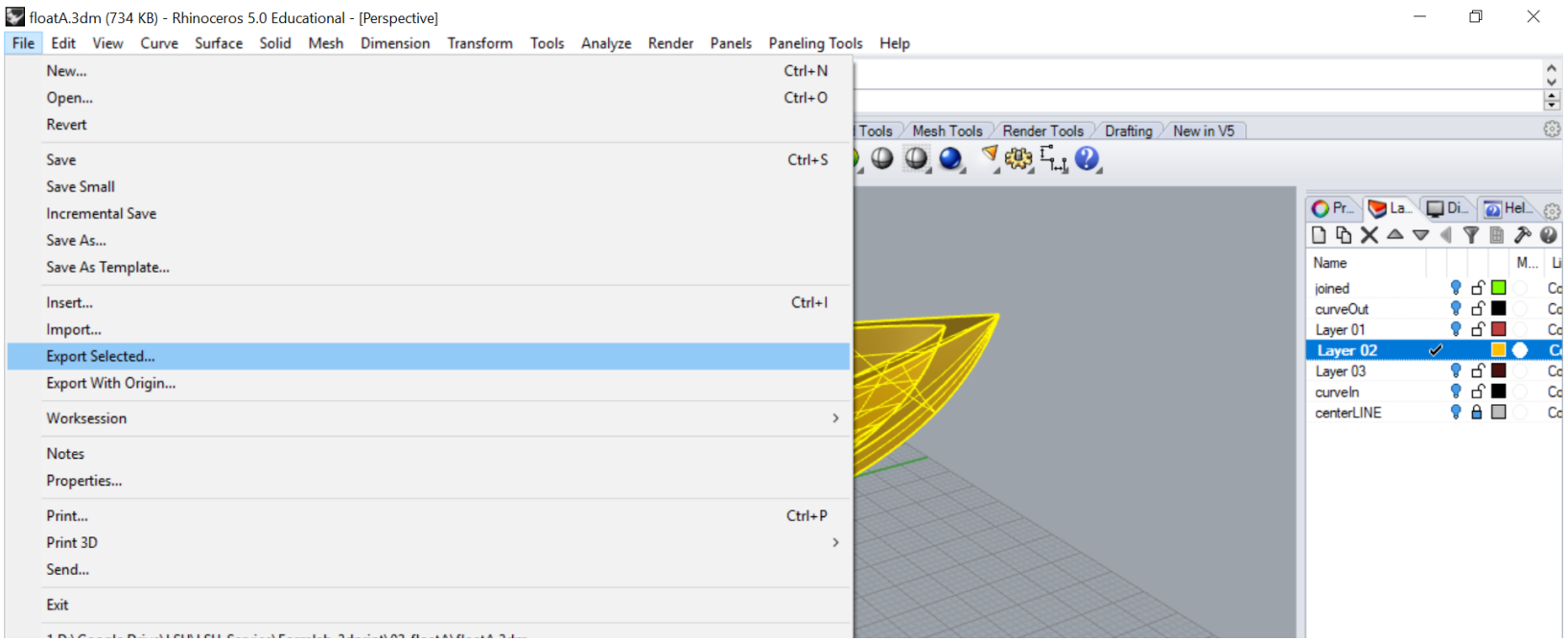
All edges: if you select this, it will display all the edges



Naked edges: you should not see any “naked edges”. If seen, try to fix it through joining surfaces, blending surfaces, or other techniques



Once the form is checked to be watertight, select the job and export it as .stl



Other modeling considerations

https://formlabs.com/media/upload/formlabs-design-guide.pdf

MINIMUM SUPPORTED WALL THICKNESS

Recommended: 0.4 mm

A supported wall is one that is connected to other walls on two or more sides. A supported wall smaller than 0.4 mm may warp during the peel process.

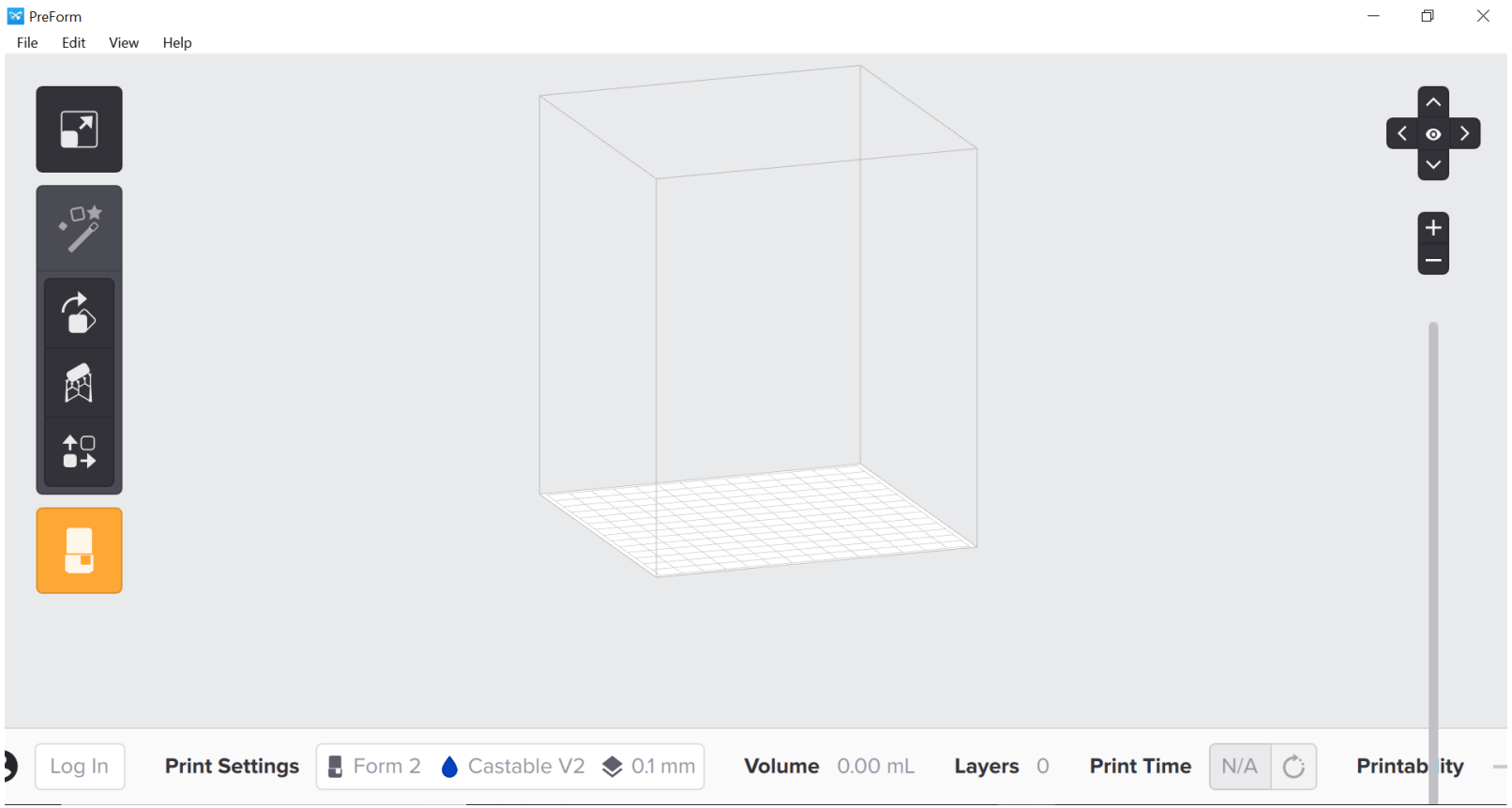
Note: Washing Thin Walls

Care should be taken when washing thin walls, as they may absorb IPA and swell during the cleaning process, leading to deformation of the part. Minimizing time immersed in IPA will limit this effect.

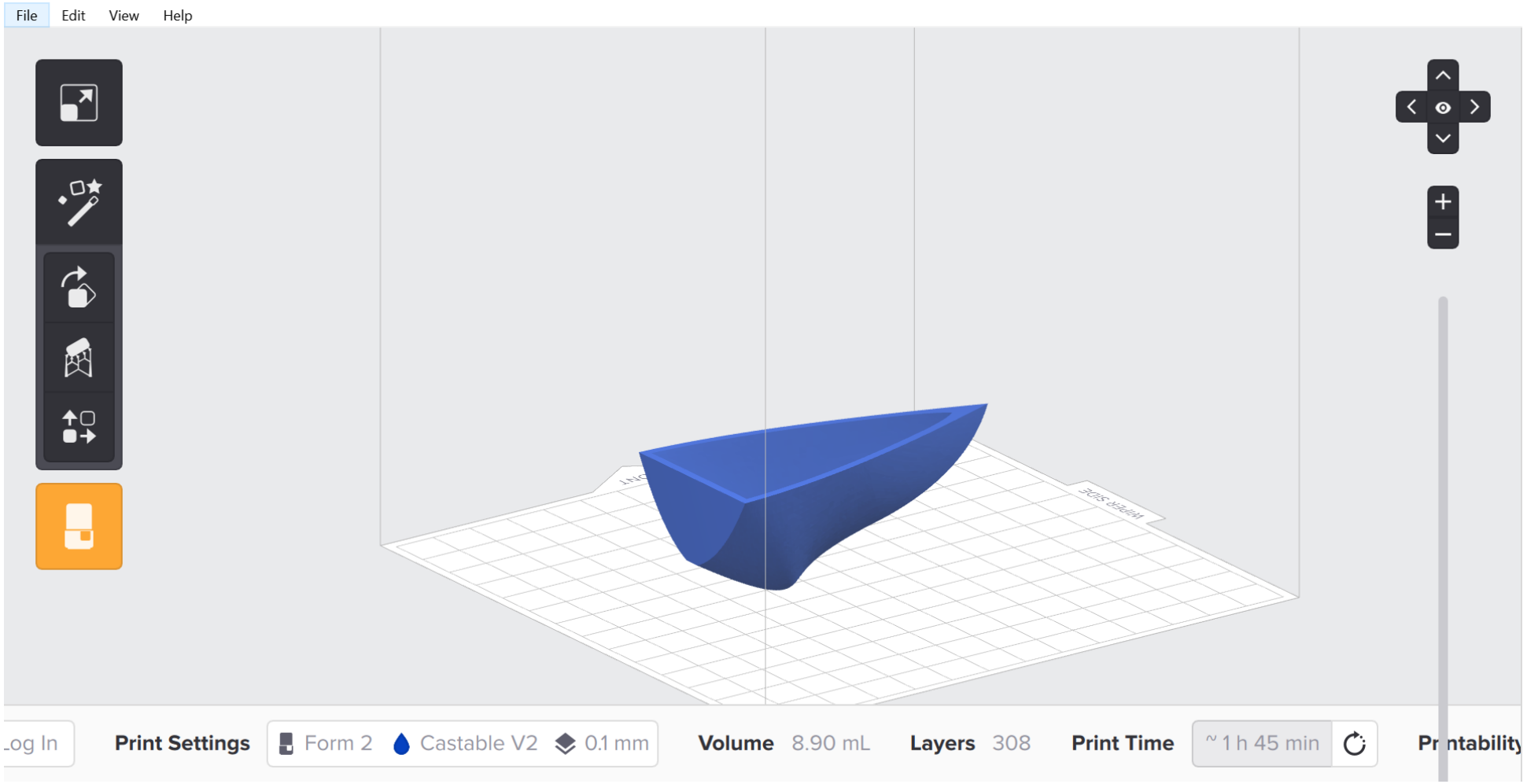
0.4 mm = 0.015 inches

Now open it in Preform (this will be done in the FabFac, but you can try installing the software on your machines. In this case, skip the printer setup as your laptop is not connected to a 3Dprinter)

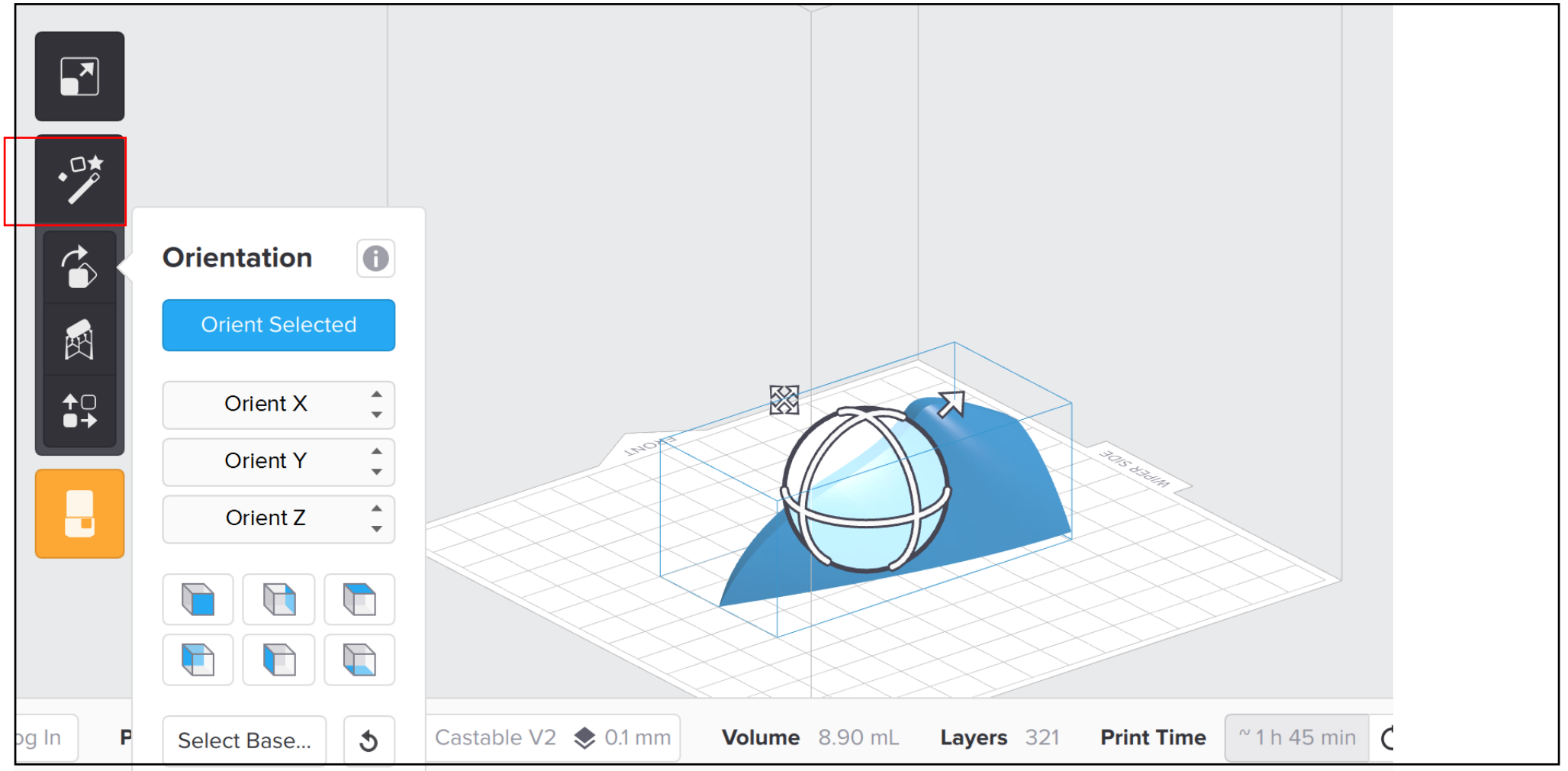
Bed size: 14.5” x 14.5”



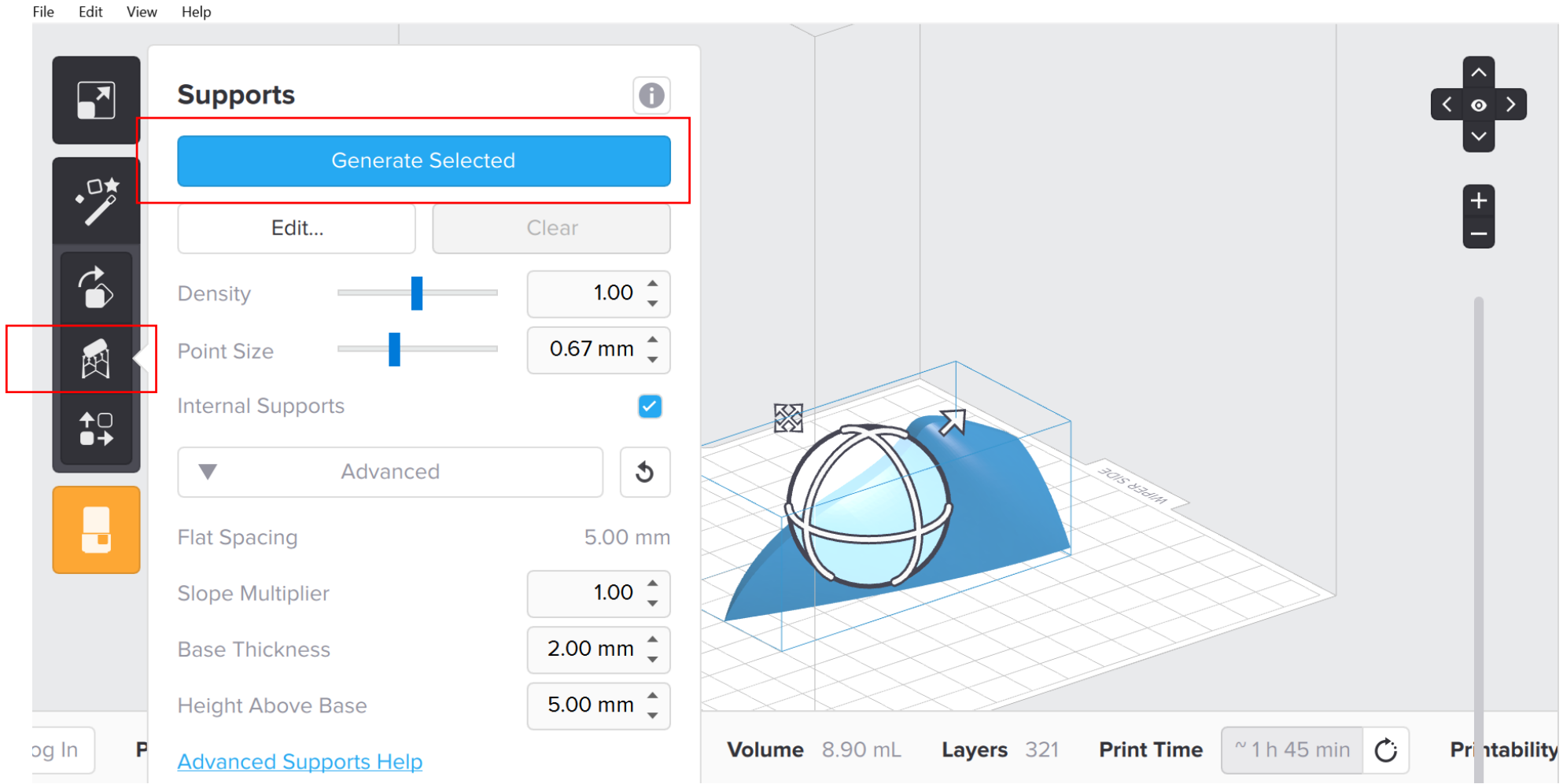
Imported object to Preform



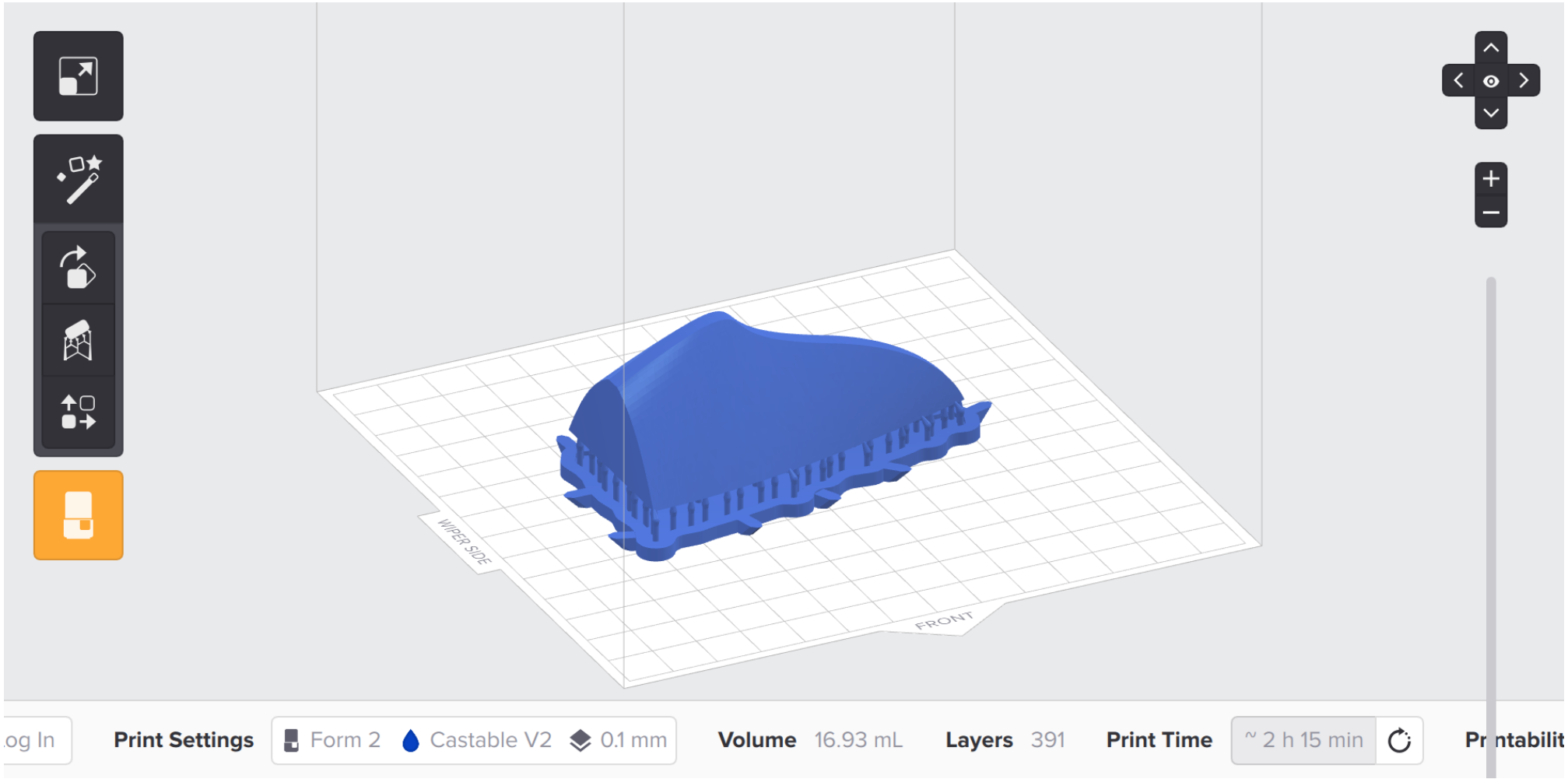
Rotate the object to determine which part will be attached to supports



Supports- set them it
Then hit “generate supports” to have them



Generated supports



Hit “start a print”
note that you can see the print time approximation at the bottom