3D Printer Training (SLA Printer; Form2)

This training covers only the SLA printers. For FDM printer training, please refer to the FDM (Makerbot/Dremel) training document.

SLA: Stereolithography (aka stereolithography apparatus, optical fabrication, photo-solidification, or resin printing) is a form of 3D printing technology used for creating models in a layer by layer fashion using photochemical processes by which light causes chemical monomers (in the form of liquid resin) to link together to form solid polymer shapes.

Our SLA Printers:

- 1x Formlabs Form2 SLS Resin Printer

Our printers use proprietary manufacturer software to set up prints.

Formlabs Form2: Formlabs Preform 

Most 3D printer setup software, commonly referred to as “slicing” software, are similar in use, though each will have its own set of criteria to accomplish a good print.

Getting Started:

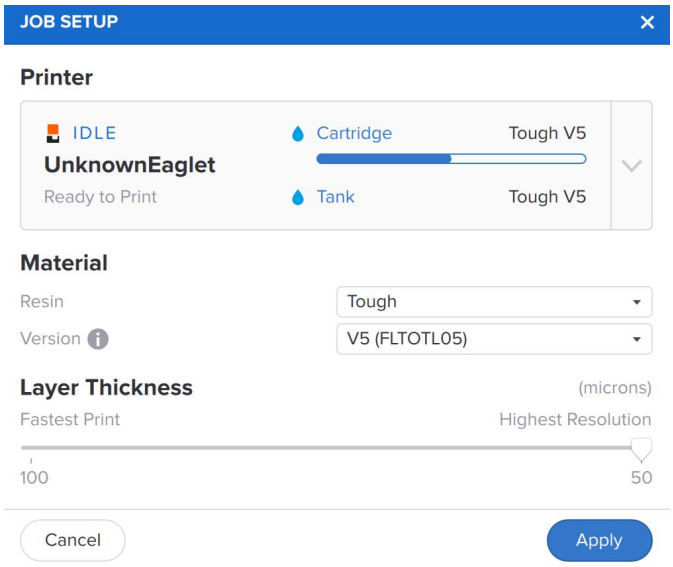
- Acquiring your 3D model:
  - Make a model using a 3D modeling software (e.g. Fusion 360)
  - Alternatively, download a file from an online resource (e.g. Thingiverse)
  - This can be in a variety of file types, but the most common and recommended are those with the .stl extension (stereolithography). Most 3D modeling software can output to this file type.

- Picking the right printer:
  - The Form2 is much more expensive and difficult to use than most of our other 3D printers. While it can accomplish much finer detail than our FDM printers, it is reserved for prints that cannot be accomplished in another way. Primarily this means very small parts or models that have extremely fine detail. This printer should never be used for simple geometric shapes or models that can readily be printed/milled on another machine.

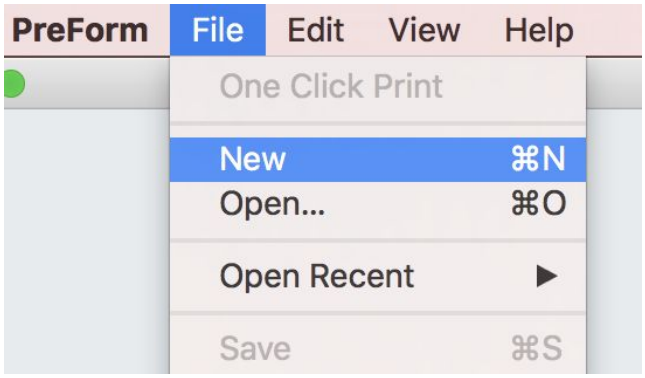
- Choosing the right resin:
    - **ALL resins must be matched to a tank that already contains the same resin as the installed cartridge.** If you would like to use a resin other than the resin currently loaded, please notify a staff member. Do not change the resin cartridge or tank yourself.
    - Black/Grey/White/Clear resins are the easiest to use and will result in the finest detail with the least chance of structural failure. These are the resins to use for 90% of prints done on this machine.
    - Durable resin is ideal for any piece that requires a small amount of flexibility, or will be experiencing stress of any kind. This is generally reserved for prototype parts or small components.
    - Tough resin is very finicky but parts built using it can endure larger impacts than other resins without breaking. This is reserved for special use and must be cleared by a staff member before printing.
    - Flex resin allows for prints that are highly flexible. The overall material quality is similar to a large pencil eraser, bendable but not stretchy. This material can be very delicate because of its flexibility, but detail on larger flat surfaces will still be excellent. This is reserved for special use and must be cleared by a staff member before printing.
    - Formlabs offers many additional resins, such as dental or ceramic, but we do not stock these. You are welcome to purchase your own resins to use in the printer, but keep in mind that you will also have to purchase a reservoir tank that will match your resin.

Printing with Form2:

- Open Preform, select printer and resin type if not already correct

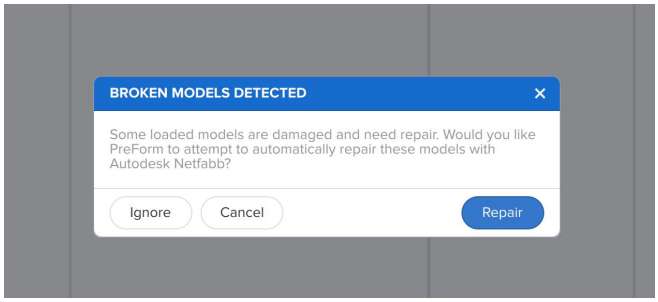


- If there is already a model in the workspace, select: File → New



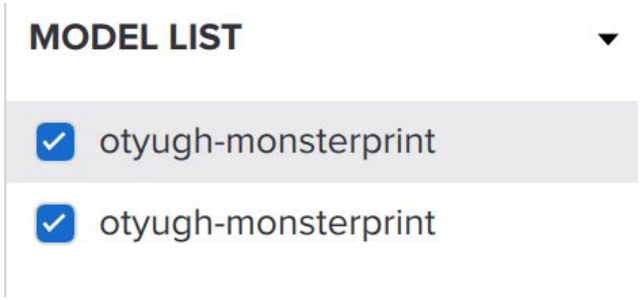
- File → Open, then select your file *OR* Drag file directly into workspace

- Many files may appear to be “broken”, simply use the Repair function



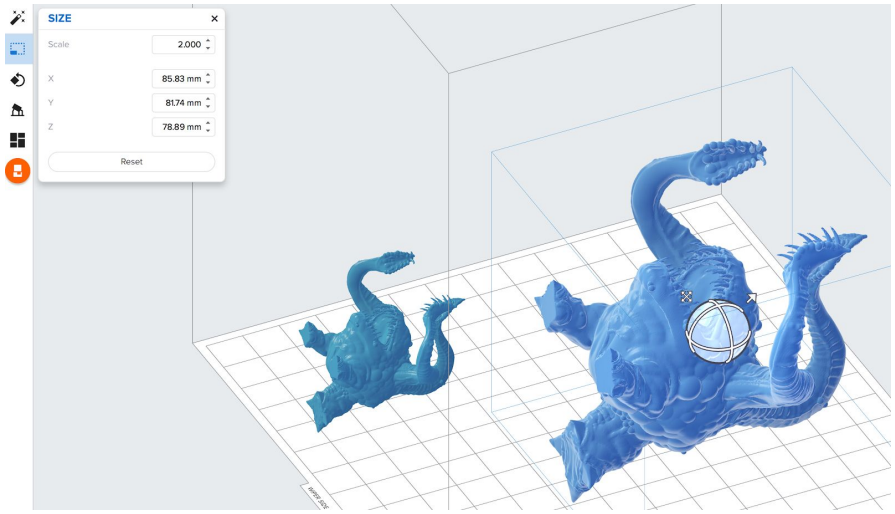
- While in the workspace, you may interact in the following ways:
    - Hold Right Click + Drag will rotate the entire workspace
    - Hold Left Click + Drag will allow you to box select multiple models
    - Hold Middle Click + Drag will pan the workspace

- Select your model(s), multiple models can be selected via box selection or Shift + Left Click. You can also select models on the bottom right hand side under Model List. Selected models will be highlighted.

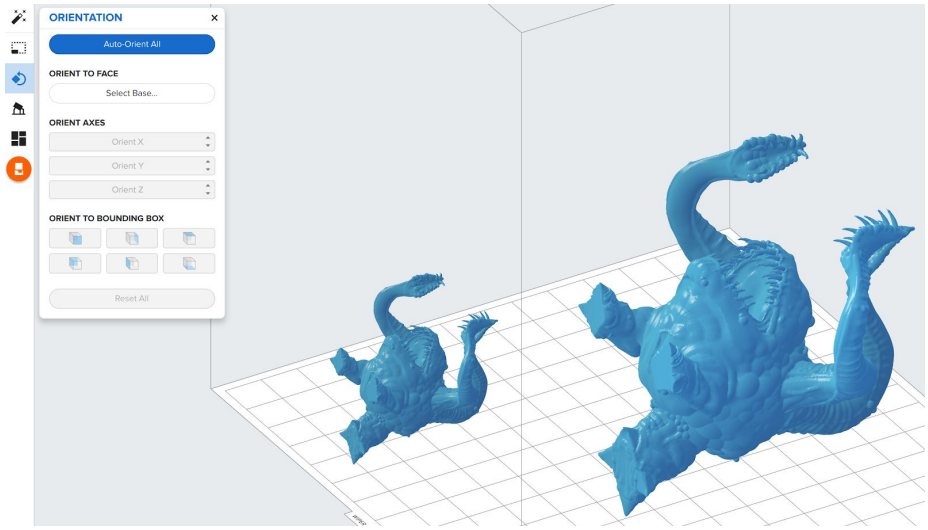


- Do not use the Magic Wand tool on the top left of the workspace. It will automatically arrange, orient, and generate supports, and generally does not do a very good job.

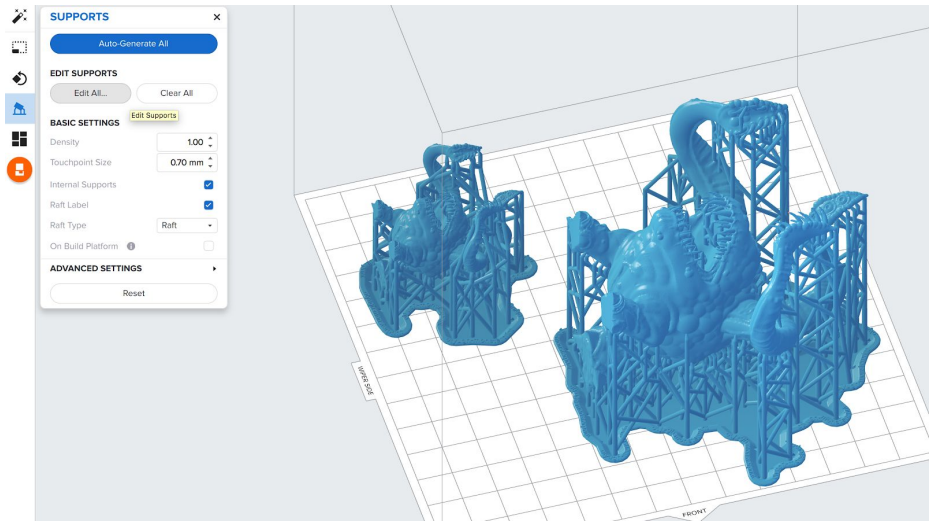
- Scale your model using the Size menu. You can scale by editing either the scale factor field, or by adjusting X/Y/Z (the remaining dimensions will be scaled proportionally)



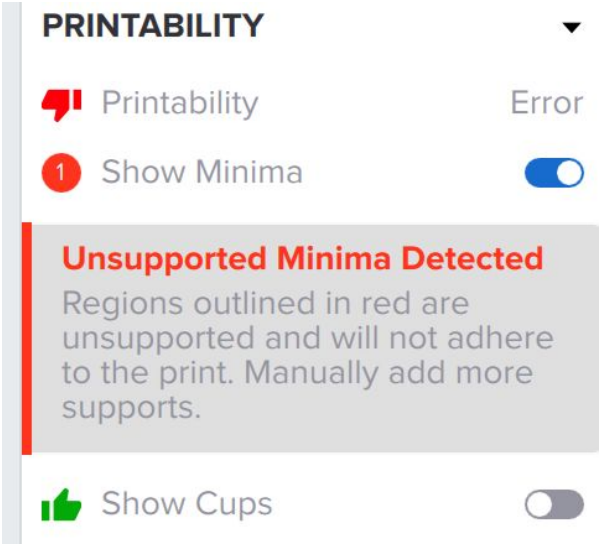
- Orient your model using the Orientation menu. You can drag the three rotation bars around the center of your model, or use any of the options in the menu to align your model as desired. In general, leave the most detailed surface facing upwards, as this will prevent the generation of supports on those features and will make cleaning much easier. Do not use the Auto-Orient All button, as it will generally do the exact opposite.



- Generate supports for your model using the supports menu. Click Auto-Generate All, and then check the Printability assessment on the right hand side once the supports are done calculating.

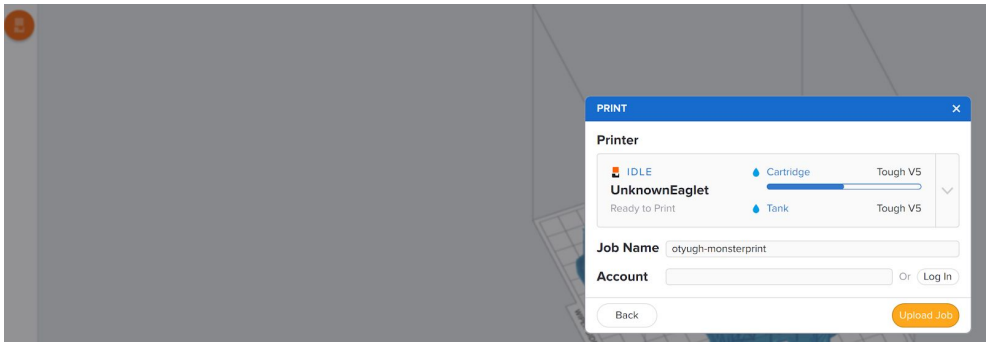


- If your Printability says “Error”, turn on Show Minima and Show Cups, and inspect your print for red areas.



- In this case, the only issue is a very small detail that will not affect any other parts of the model, so we’ll ignore it and move on to the next step.

- Click the Print button on the left, verify your printer and material are selected correctly, and select Upload Job when ready.



- Follow the prompts on the printer to finish your print!