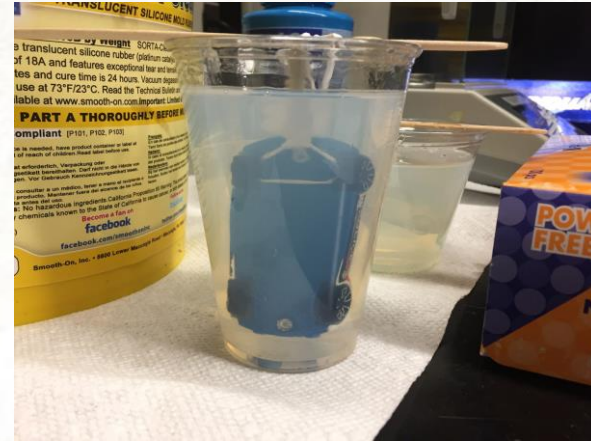
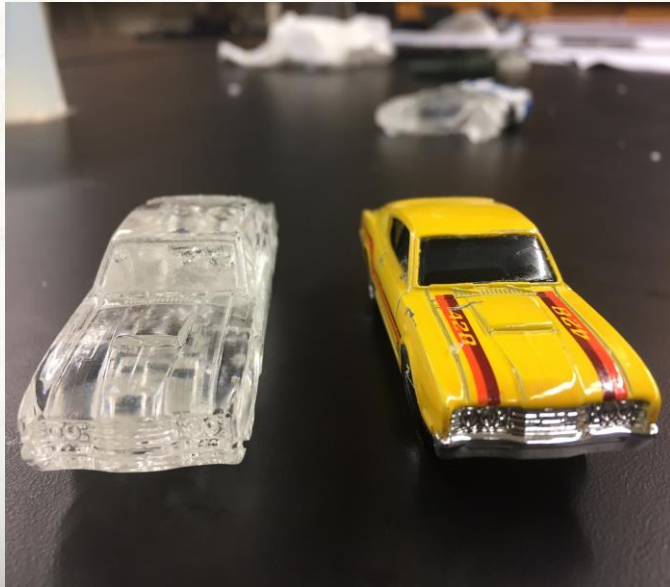


# How to make and fill silicone molds



# Part I



# Step 1- Materials to use

- Put on PPE, gloves and goggles if provided
- No shorts and no open toed shoes
- Regular and big plastic cups
- Make sure working area is organized and prepared



# Step 2 - Mixing for Molding

- Use big plastic cup for this process.
- Mix silicone (Part A) with reagent (Part B). Ratio 10:1 of part A and part B.
- Stir the solution slowly until it is well mixed.
  - Bubbles will form, it's okay.



# Important

- Do not pour more than 1/4th of the cup.
- You will need about 270 g of part A and 30 g of part B
- This must be split with two cups since the scale can not exceed 200 g

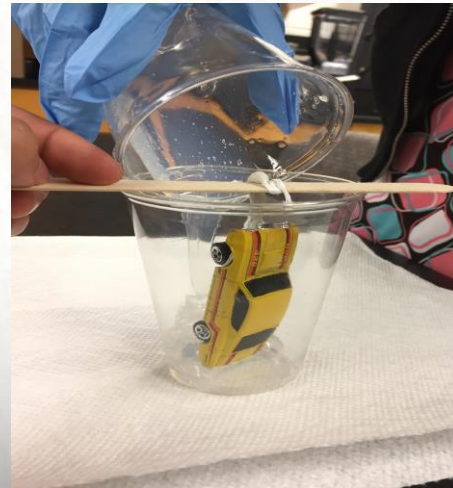


- Degas the mix of the molding to finish with a clear mold.
- Make sure to place a cap in case of an overflow.

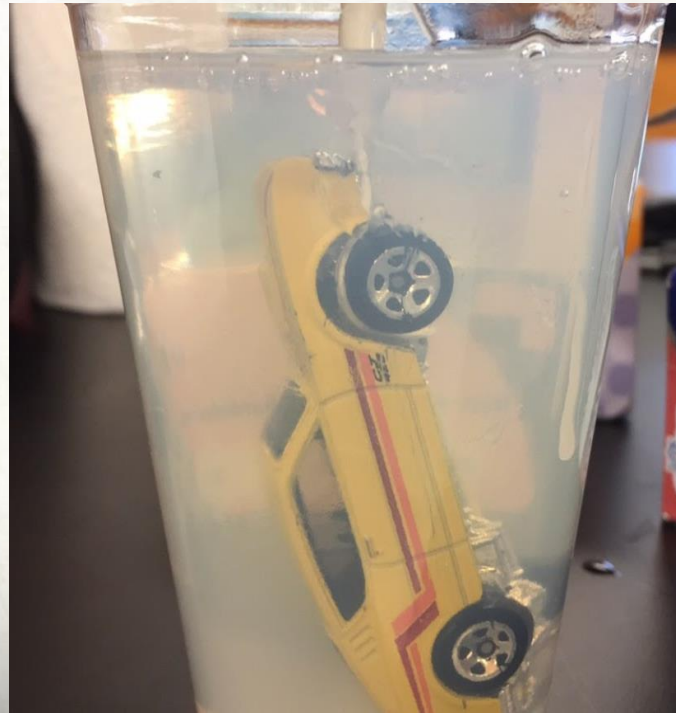


# Step 3- Placing toy car

- Create a cylinder clay structure as support for the car.
- Attach cylinder to bottom of car and stick to bottom of cup
- Attach clay to the wheels and around a popsicle stick



- Lastly pour the mix into the cup.
- Wait approx. 24hrs for the final result.





# Part II



# Step 1- Materials to use

- Put on PPE, gloves and goggles if provided
- No shorts and no open toed shoes
- Exacto knife
- Regular and/or big plastic cups
- Make sure working area is organized and prepared



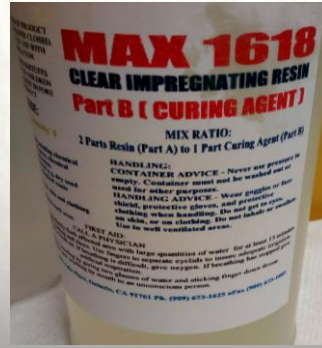
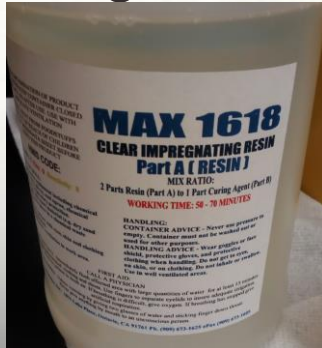
# Step 2 - Take out toy car

- Cut plastic cup in mold
- Make a cut line through the silicone mold to take out toy car
  - Make a solid cut line near the tires. Since tires are the most difficult to get out of mold, try not to damage the mold.

# Step 3 - Mixing

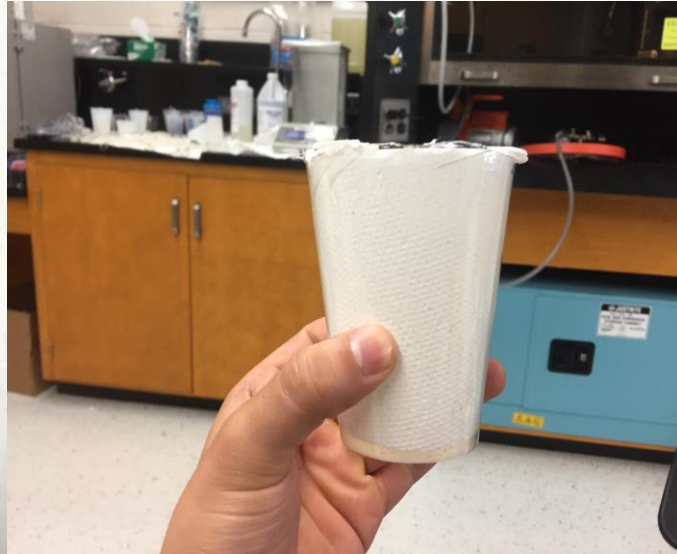
- Mix epoxy resin (Part A) with (Part B). Ratio of 2:1 of part A and part B.
- Stir the solution slowly until it is well mixed.
- Time to degas epoxy mix.

(The degas time tends to be longer than the silicone molds)



# Important

Wrapping the pre-made mold with towel allows for cleaner result.



# Step 4 - Filling epoxy

- Fill the epoxy resin mix into silicone molds
- When epoxy is almost at the top, put the key chain holder
- Pour a little more epoxy
- Wait 24 hrs for complete dryness.



Thank you  
for your attention!



## STEREOLITHOGRAPHY (SLA)

**Printer:** FormLabs Form 2

**Cost:** \$3000 (approx.)



**Material:** Clear Resin (One of the many acrylate resins available that can be used in the SLA printer)





# Software and Setup (Steps):



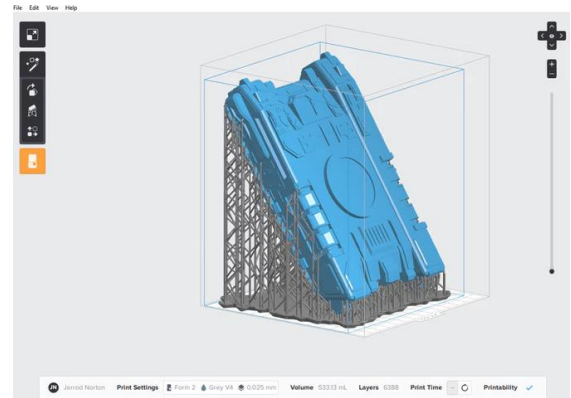
**Design:** Start with the object of your choice by either making a CAD model or 3D scanning and saving it as an STL file or OBJ file



*Export CAD Model*



**Process Setup:** FormLabs has a free downloadable program called **PreForm** that sends your STL file directly to the printer. This software will be used to setup the printing process after file import is successful.



*Transfer File to Printer*

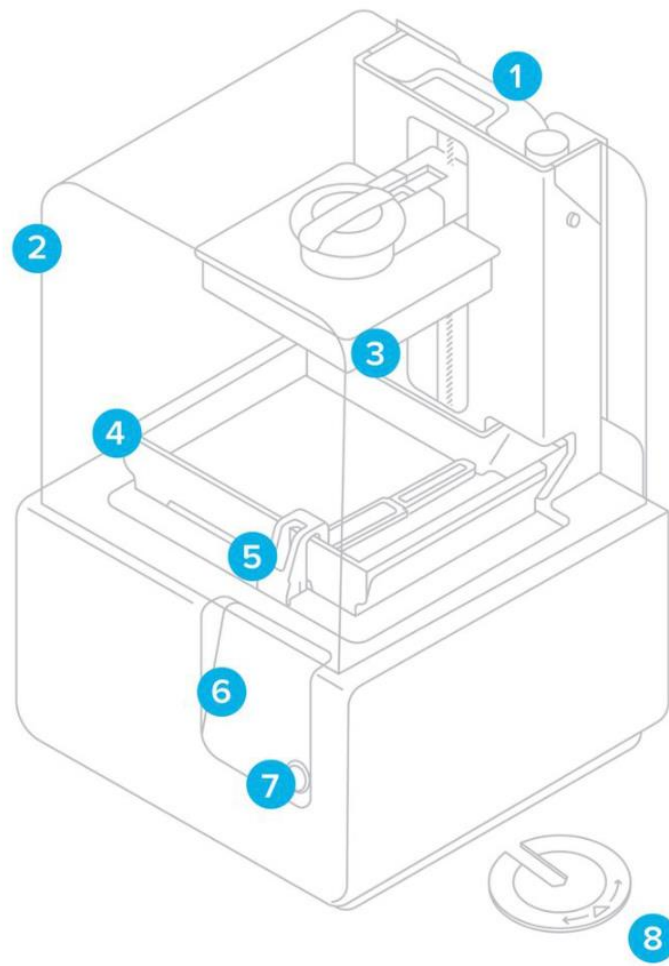


The file can be easily transferred to the printer using a USB cable. The rest of the process can be done by following the steps on the printer display



## The Printer:

The following are the 8 main parts of the FormLabs Form 2

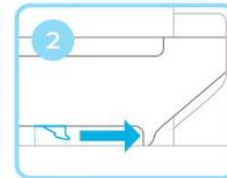
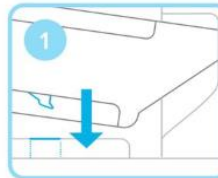
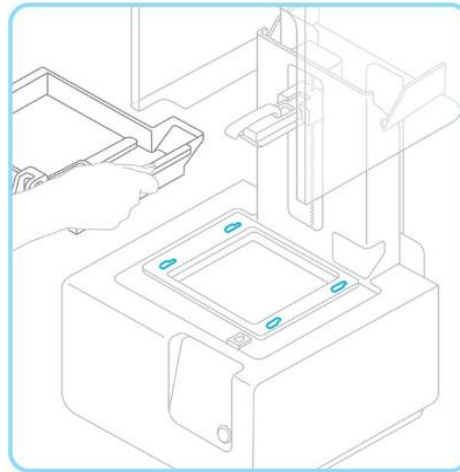


1. **Cartridge** – Carries the Resin
2. **Cover** – Enclosure for the printer during the printing process
3. **Build Platform** – Resin is cured on this surface as it moves up or down
4. **Resin Tank** – Material flows from the cartridge into the plate/tank and material is cured onto the build platform from here
5. **Wiper** – Sweeps the material present in the resin tank over the surface of the build platform for curing
6. **Touchscreen** – Display to aid process setup
7. **Button** – Used to navigate the UI
8. **Leveling Tool** – To make sure the Resin in the tank is flat and there are no inconsistencies

## Preparation:

### 1) Insert Resin Tank with Wiper

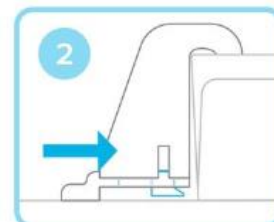
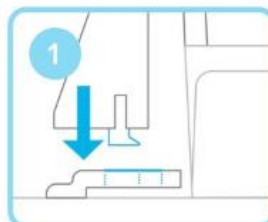
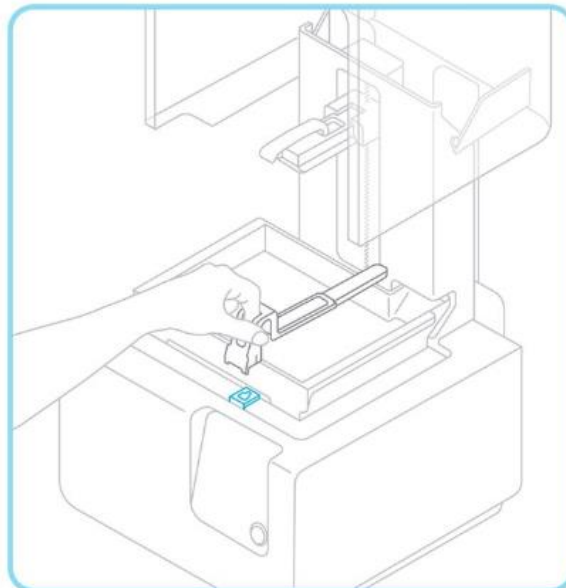
Remove the tank lid and align the four small feet of the resin tank with the corresponding holes in the tank carrier; push forward into place.



Remove the tank lid and align the four small feet of the resin tank with the corresponding holes in the tank carrier. Push forward into place with force.

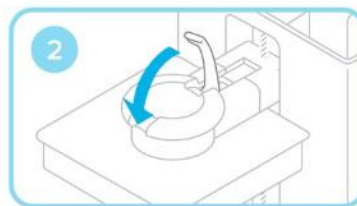
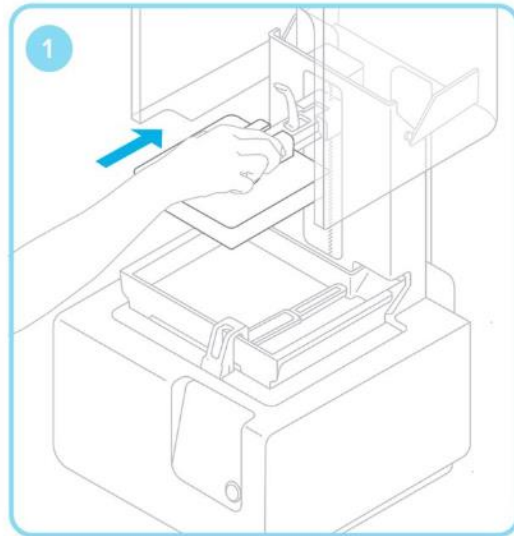
### 2) Lock Wiper

Ensuring the wiper is straight, align the foot of the wiper with the corresponding hole and push forward, snapping it into place.



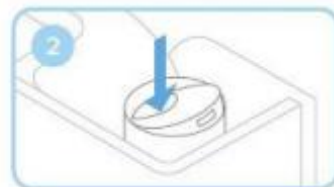
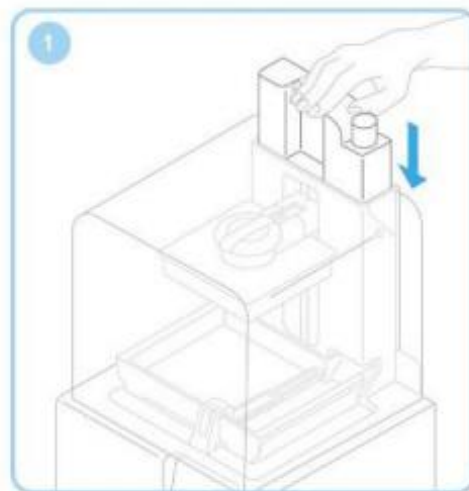
### 3) Insert Build Platform

Align the build platform with the platform carrier and push it into place, locking the handle down to secure.



### 4) Insert Resin Cartridge

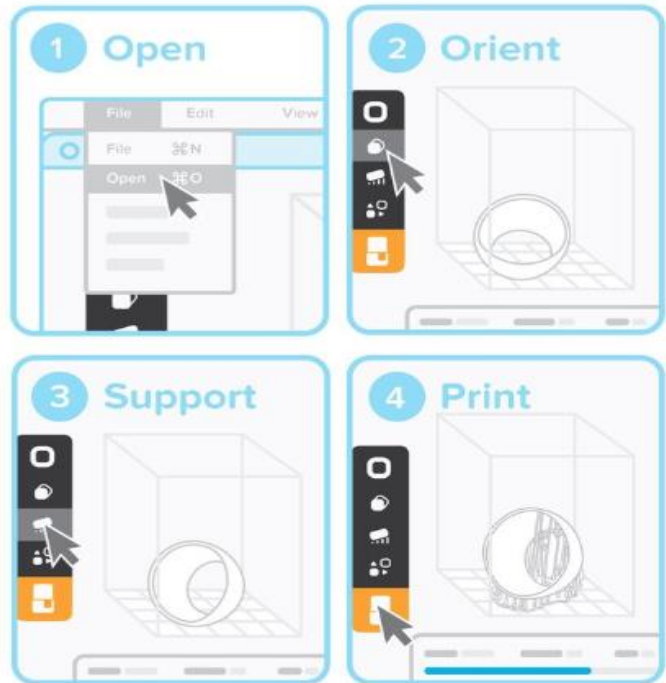
Align the cartridge with the opening in the back of the printer. Push down until it sits securely. Be sure to remove the orange valve cover from the underside and press open the cap before starting a print, so that your resin tank fills correctly.



## Print:

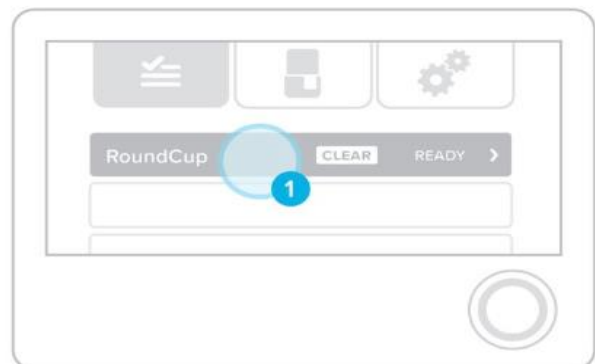
### 1) Prepare File for Print

Open your .STL or .OBJ files. Most parts print better when supported at an angle. For automatic set-up, use PreForm tools to “orient all,” “generate all,” and “layout all.” PreForm will do these steps in sequence if you select “One Click Print.” Confirm your print’s material and layer height, then click the orange print icon. Select which printer to upload the file to from the printer dialogue.



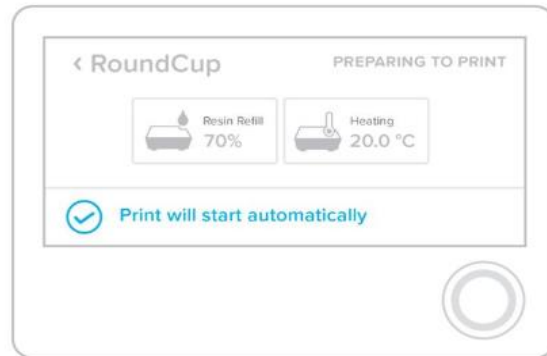
### 2) Confirm Print on Printer

You will see your file uploading to the print queue on your printer’s touchscreen. Select the file name and confirm the print by pushing the button on your printer



### 3) Pre-Print Checks

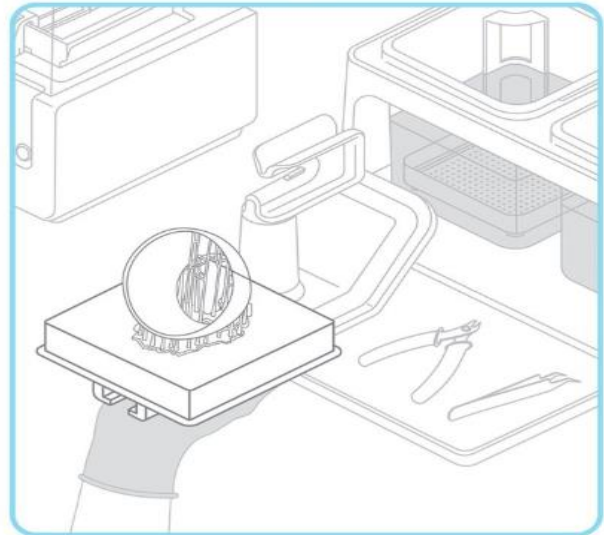
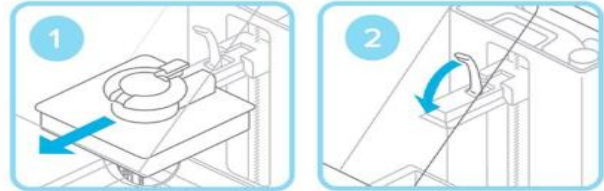
Before printing, the resin tank will fill and heat up to around 35° C. Once the tank is full, your print will automatically start.



## Finishing Up:

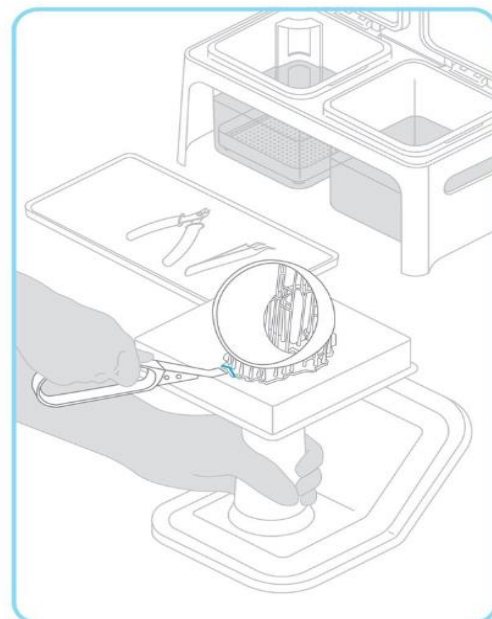
### 1) Remove the Build Platform

When your print is finished, wear gloves to remove the build platform. Turn the platform over to prevent dripping while transferring to the finish kit. Make sure to close the printer cover.



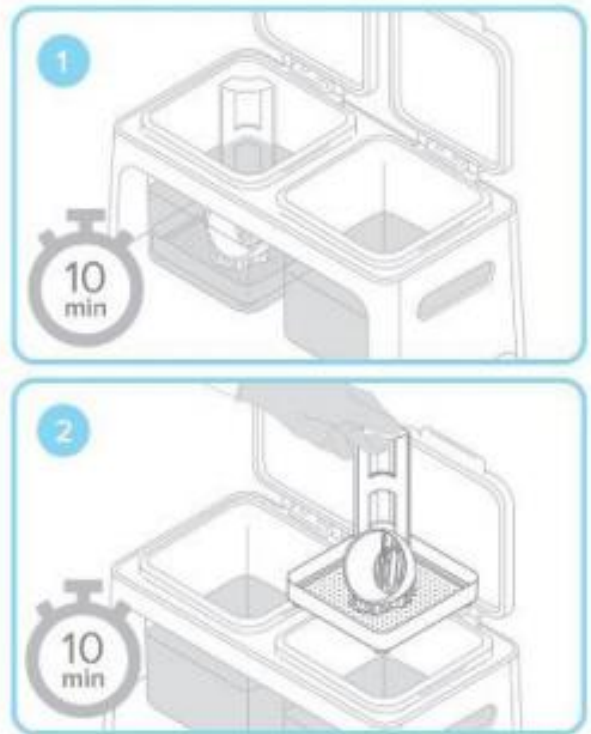
### 2) Remove Your Print

Attach the build platform to the jig and slide the removal tool under the base of your prints to release the part. Use the removal tabs for easy entry points.



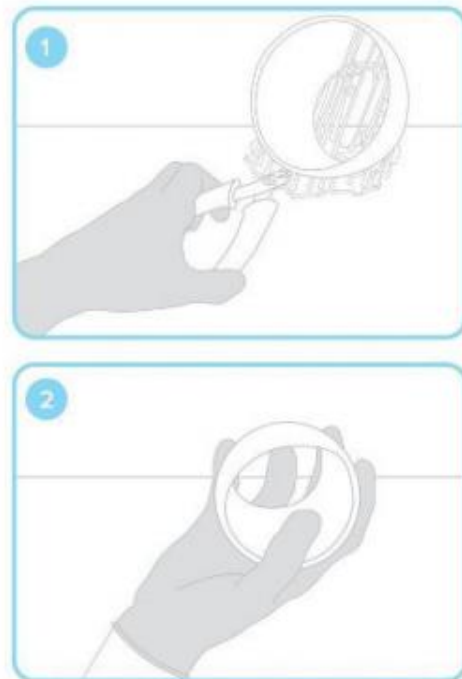
### 3) Wash Your Print

Fill two rinse buckets with isopropyl alcohol (IPA). Keep the lids closed and keep out of the reach of children; IPA is flammable. Drop your print in the rinse bucket and leave for 10 minutes. Agitate your part and repeat with the second rinse bucket. This two-step process will help your IPA last longer.



### 4) Finish Your Print

Once dry, use the flush cutters to remove supports from your print. You can sand support marks and polish your parts for a clean finish. You may need to UV post-cure your part if using functional resins.





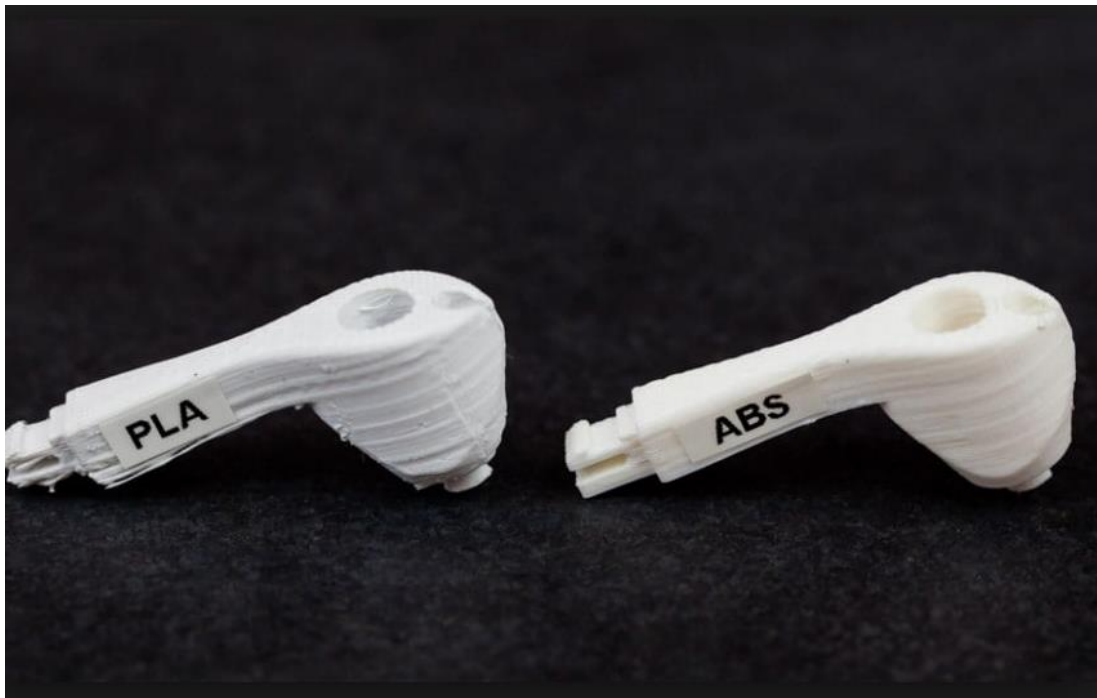
## FUSED DEPOSITION MODELING (FDM)

**Printer:** Dreamer FlashForge

**Cost:** \$1200 (approx.)



**Material:** ABS and PLA



## Software and Setup (Steps):



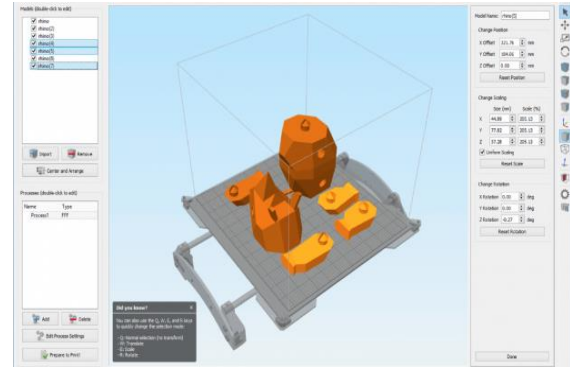
**Design:** Start with the object of your choice by either making a CAD model or 3D scanning and saving it as an STL file or OBJ file



*Export CAD Model*



**Process Setup:** Simplify 3D is a licensed software that can be used for a plethora of FDM printers. This software is very user friendly and will be used to setup the printing process after file import is successful.



*Transfer File to Printer*

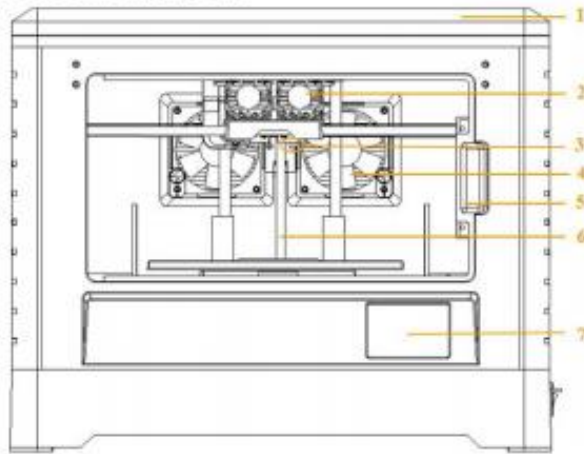


The file can be easily transferred to the printer using an SD card. The rest of the process can be done by following the steps on the printer display

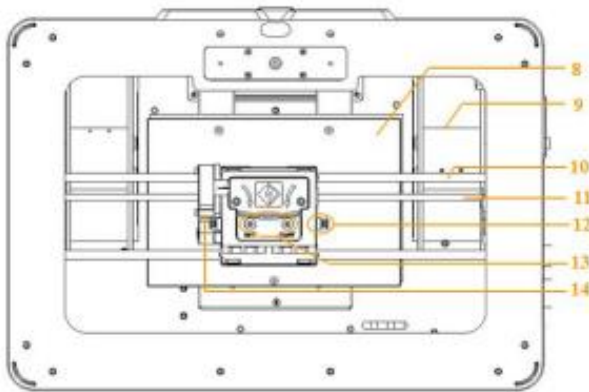


## The Printer:

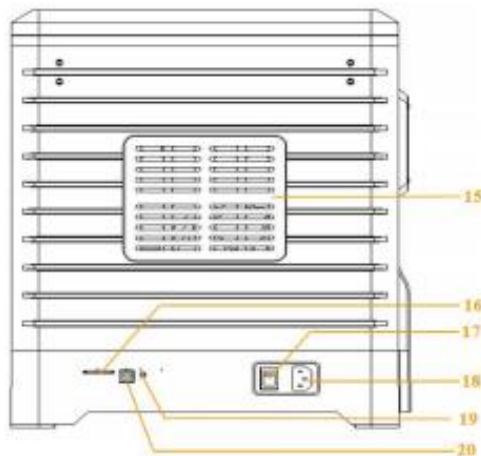
### 2.1.1 Main view:



### Top view:



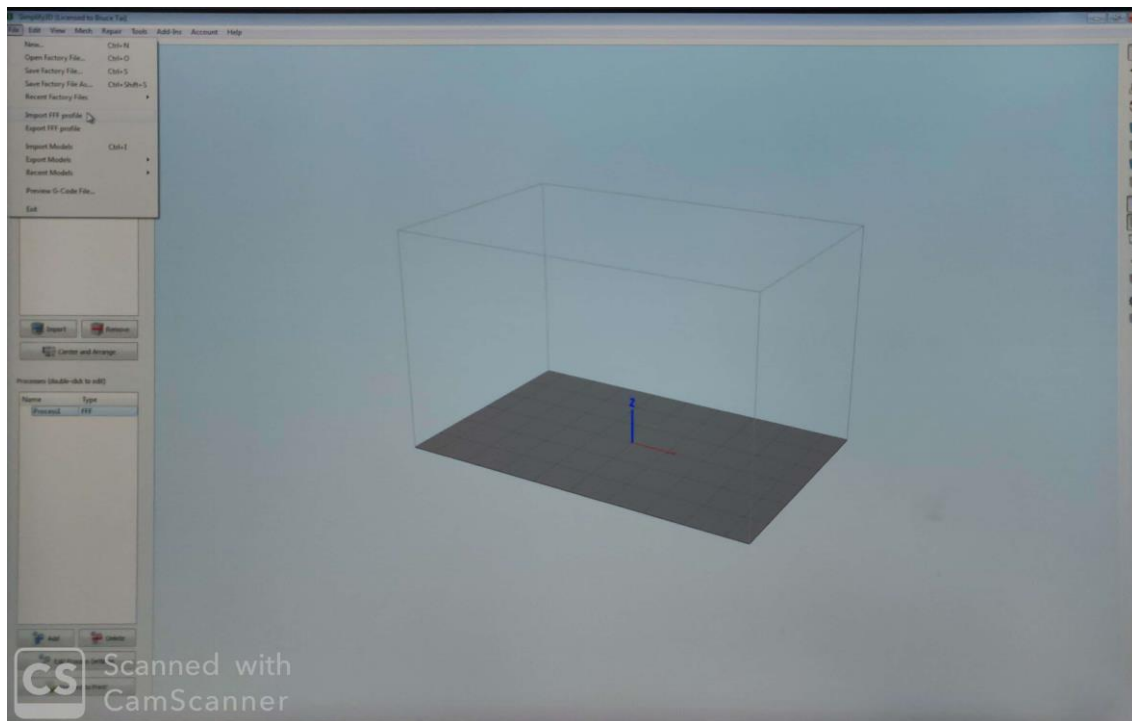
### Left view:



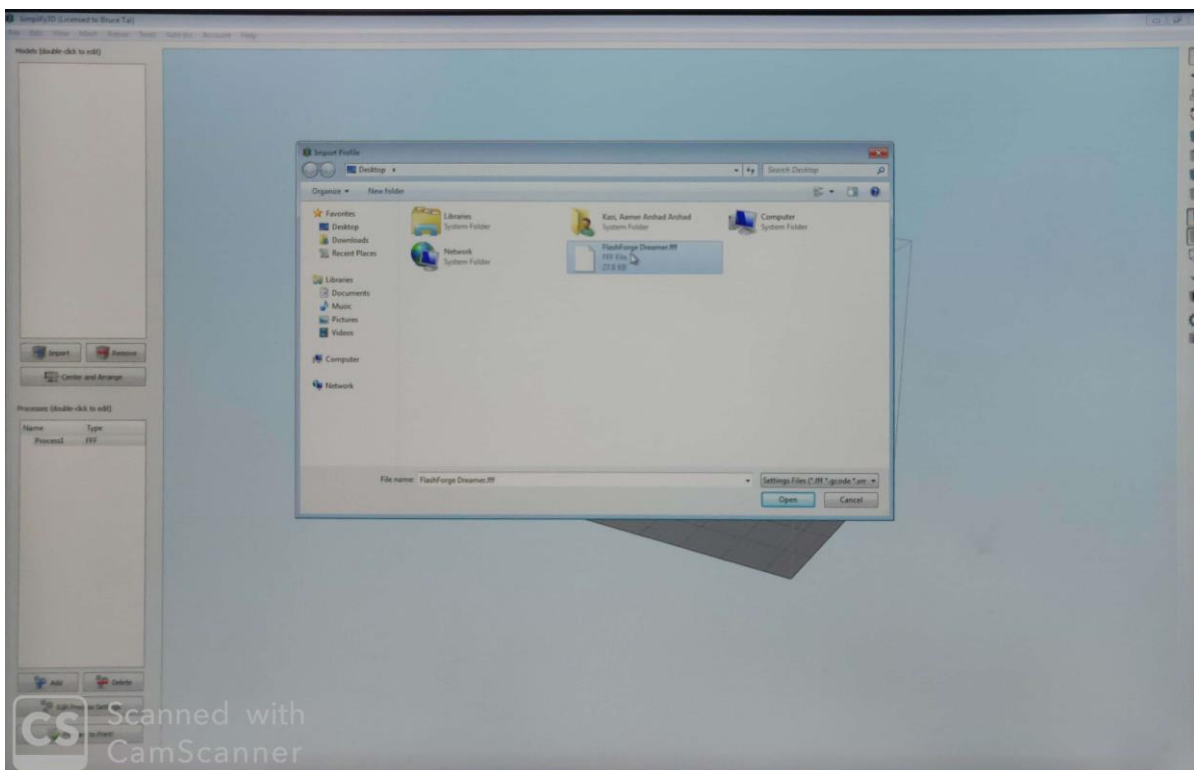
1. Lid
2. Cooling fan
3. Nozzle
4. Ventilating fan
5. Front door
6. Z axis guide rod
7. Touch screen
8. Build plate
9. Filament slot
10. X axis guide rod
11. Timing belt
12. Spring presser
13. Filament intake
14. Turbofan
15. Side panel
16. SD card slot
17. Power switch
18. Power input
19. Reset button
20. USB cable input

## Printing:

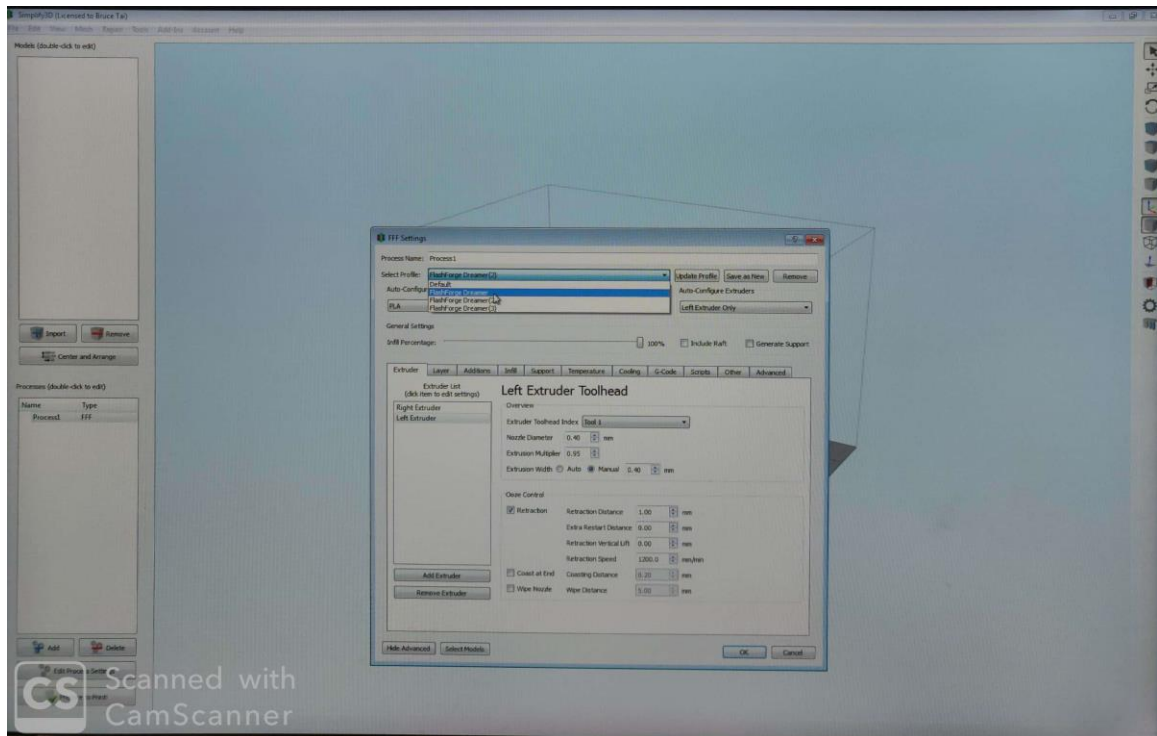
- 1) The first step is to import a printer profile for your printing process. These profiles are specific to each printer type and are used as a link between the model on Simplify 3D and the printer. Click on **File > Import FFF profile**



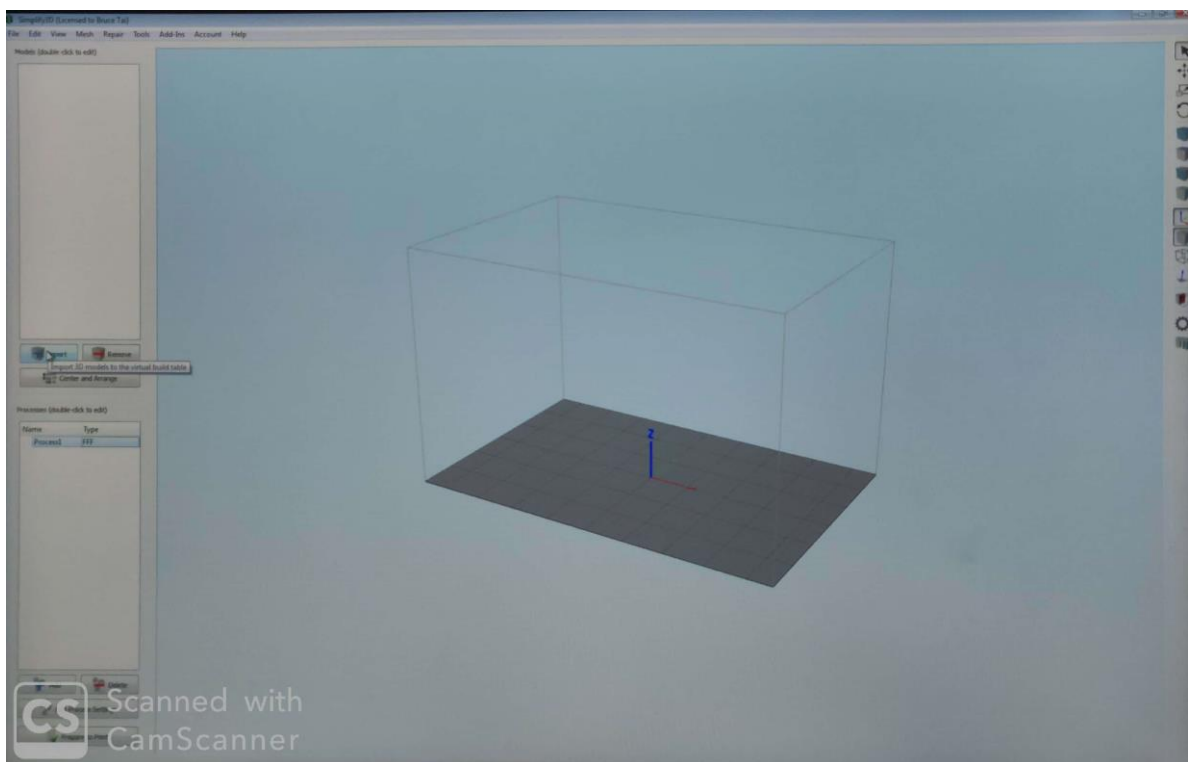
- 2) Find the printer profile on the computer and select it



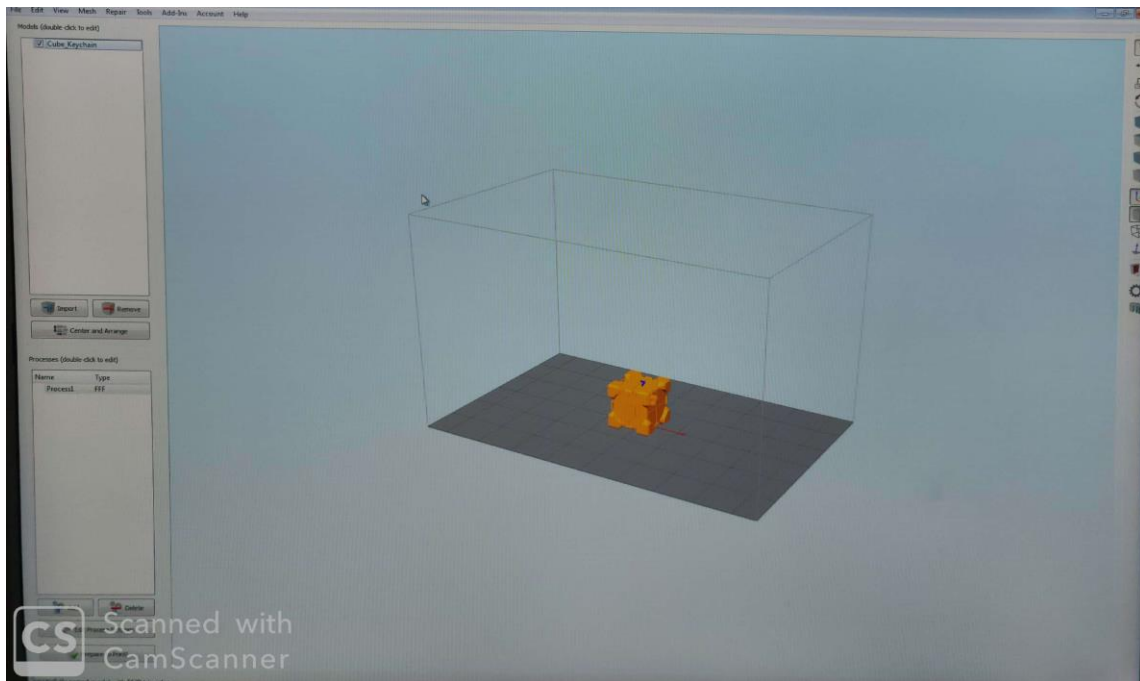
- 3) Click on **“Edit process settings”** in the bottom left corner and a dialog box will appear. Select the printer profile in the drop down menu as shown below and click OK. There are many different settings you can change within Simplify3D. Generally for simple parts, you will need to make very few or almost no changes (It does depend on the printer material and part geometry though.)



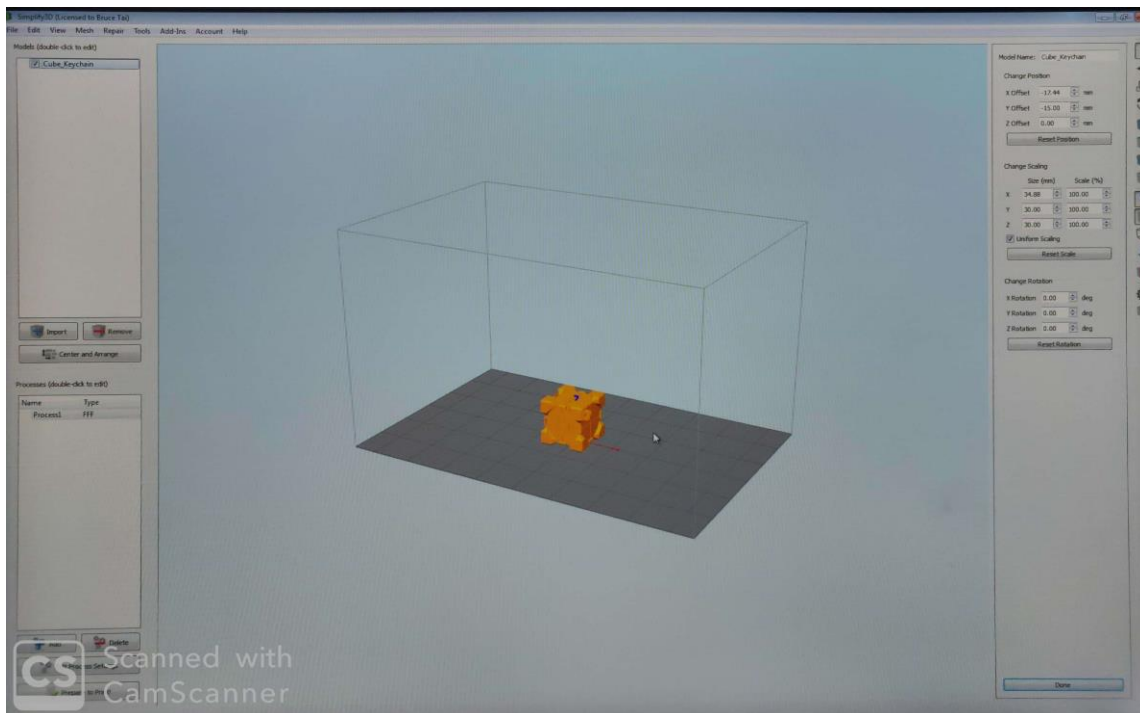
- 4) After the printer profile has been successfully imported and recognized, the next step is to import the CAD model STL file. Click on **“Import”** in the left column as shown in the figure below.



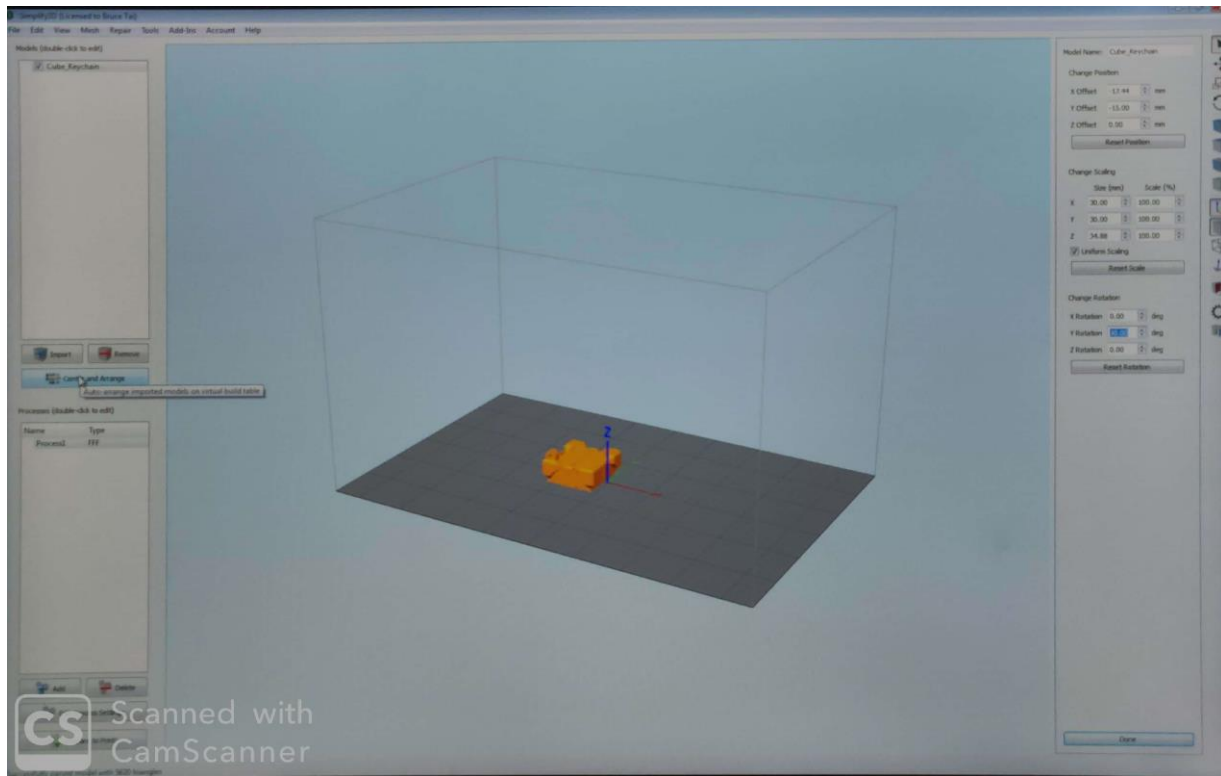
- 5) Browse your system and select the STL file that needs to be printer. It should appear on the screen immediately after



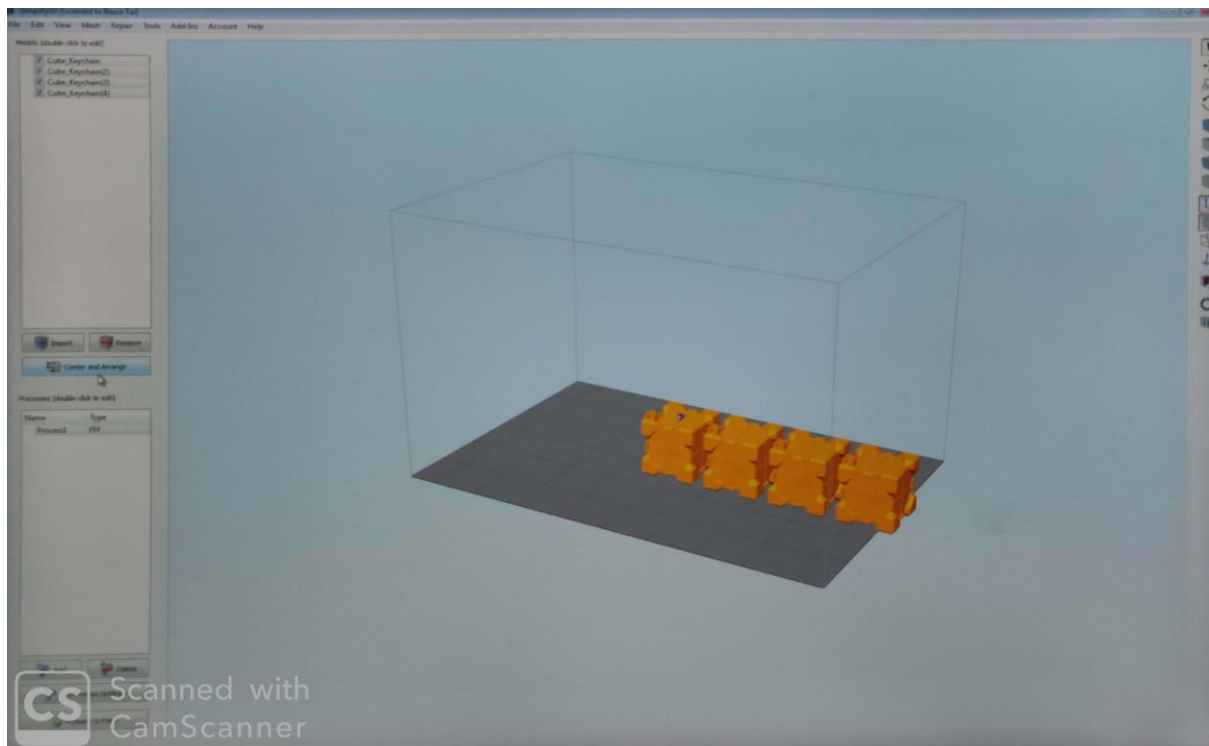
- 6) There are options to change the position, size and orientation of the model on the print bed. To do so, double click on the model and a bar should appear on the right of your screen. Adjust the parameters according to your printing needs.

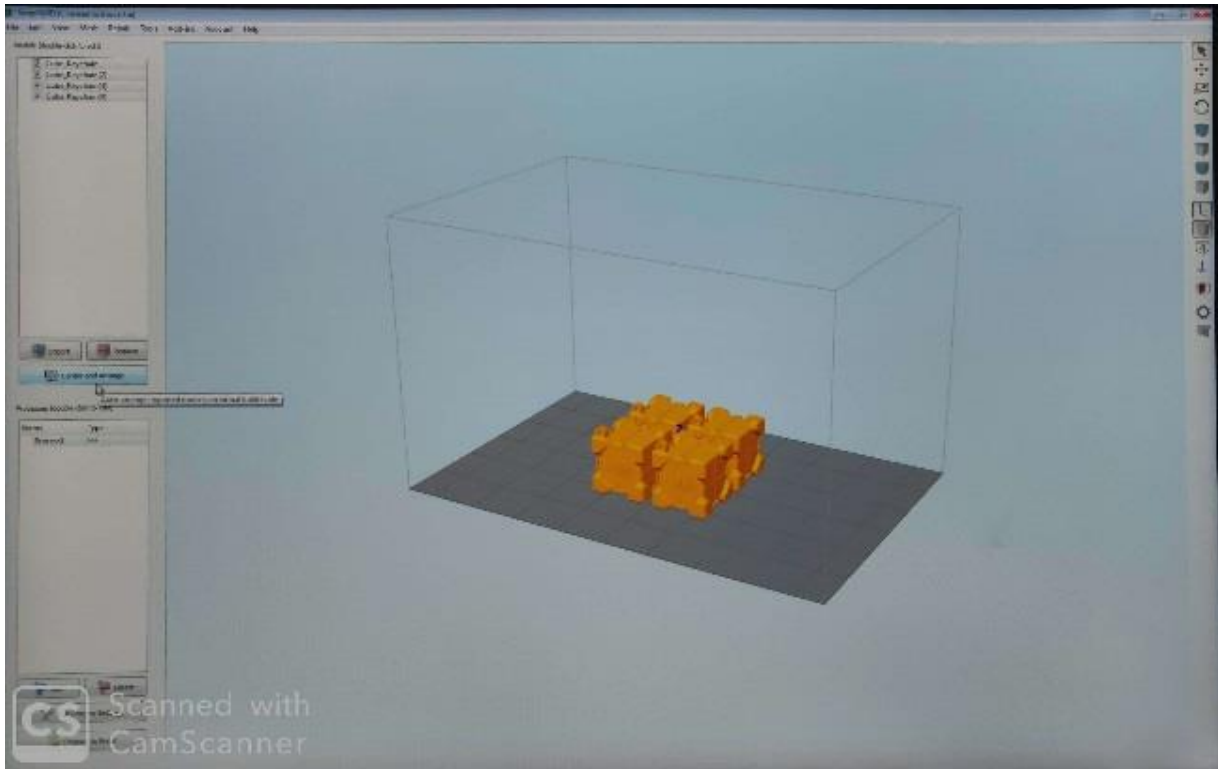


- 7) After adjusting the part parameters, it is necessary to bring the part back into place on the build platform. The **“Center and Arrange”** option will automatically bring the part into position

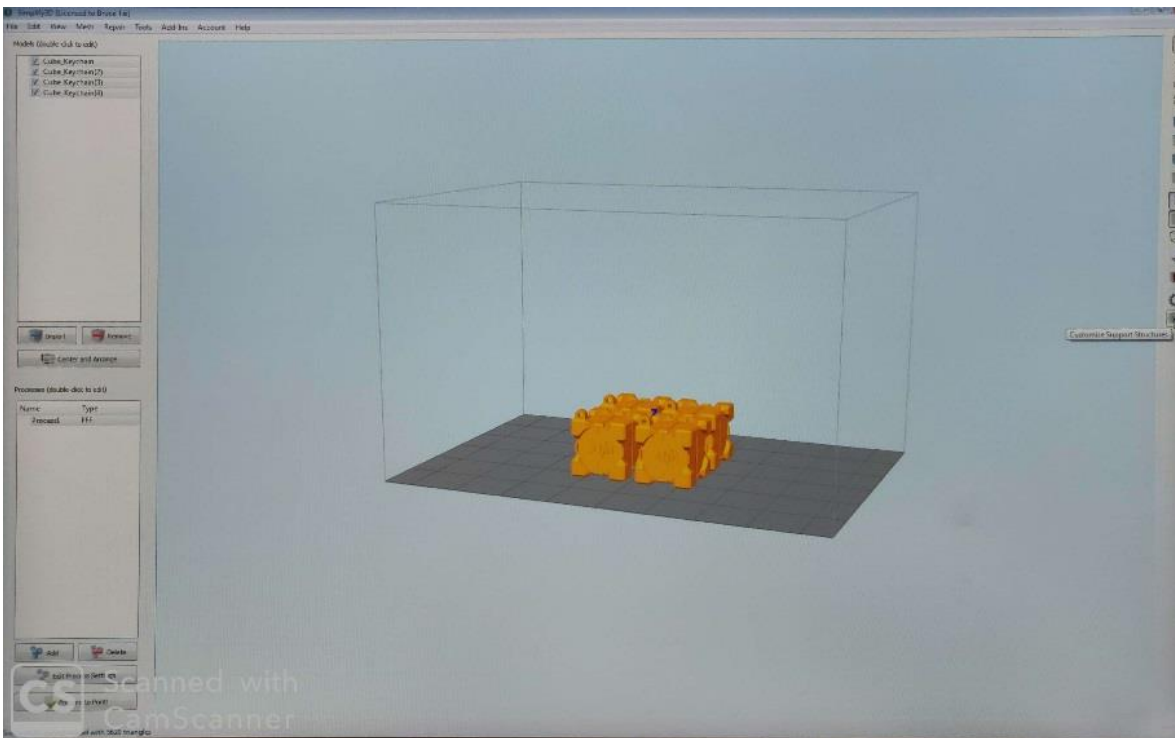


- 8) If multiple instances of the same need to be printed, simply **“ctrl+c”** and **“ctrl+v”** the part as many times as you want. Follow this up with a **“Center and Arrange”** command to ensure the right alignment.

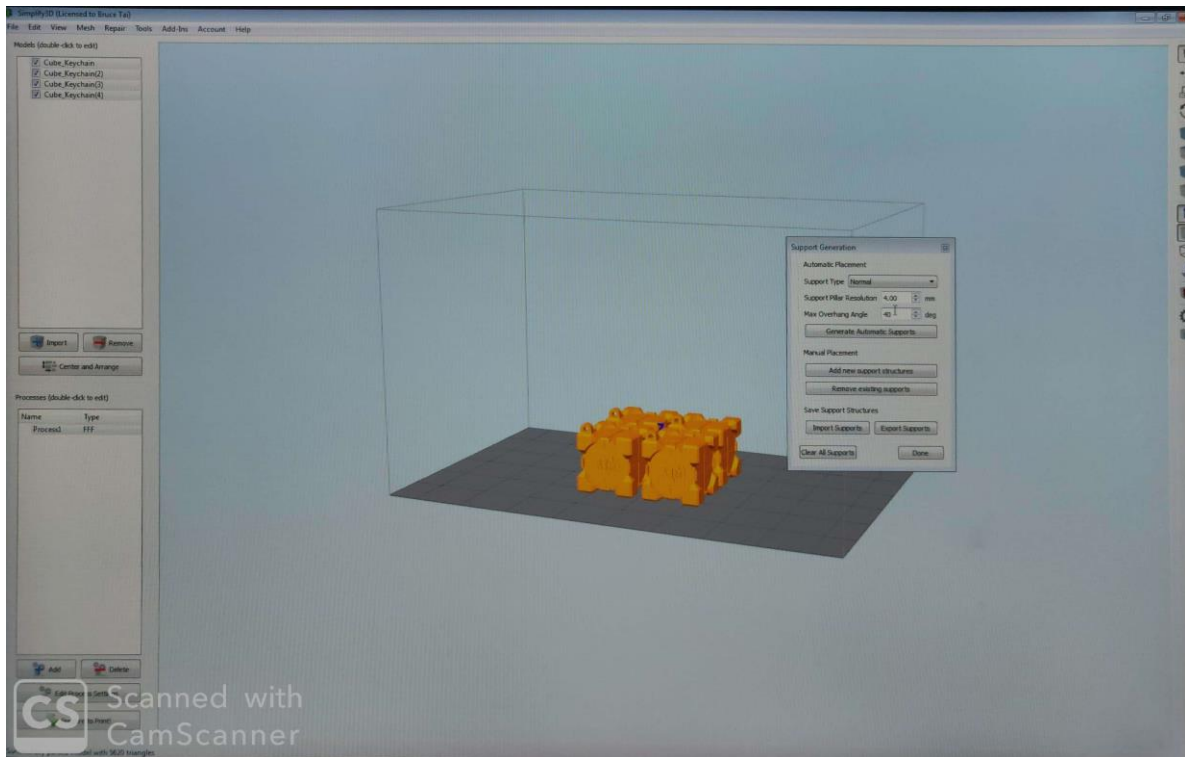




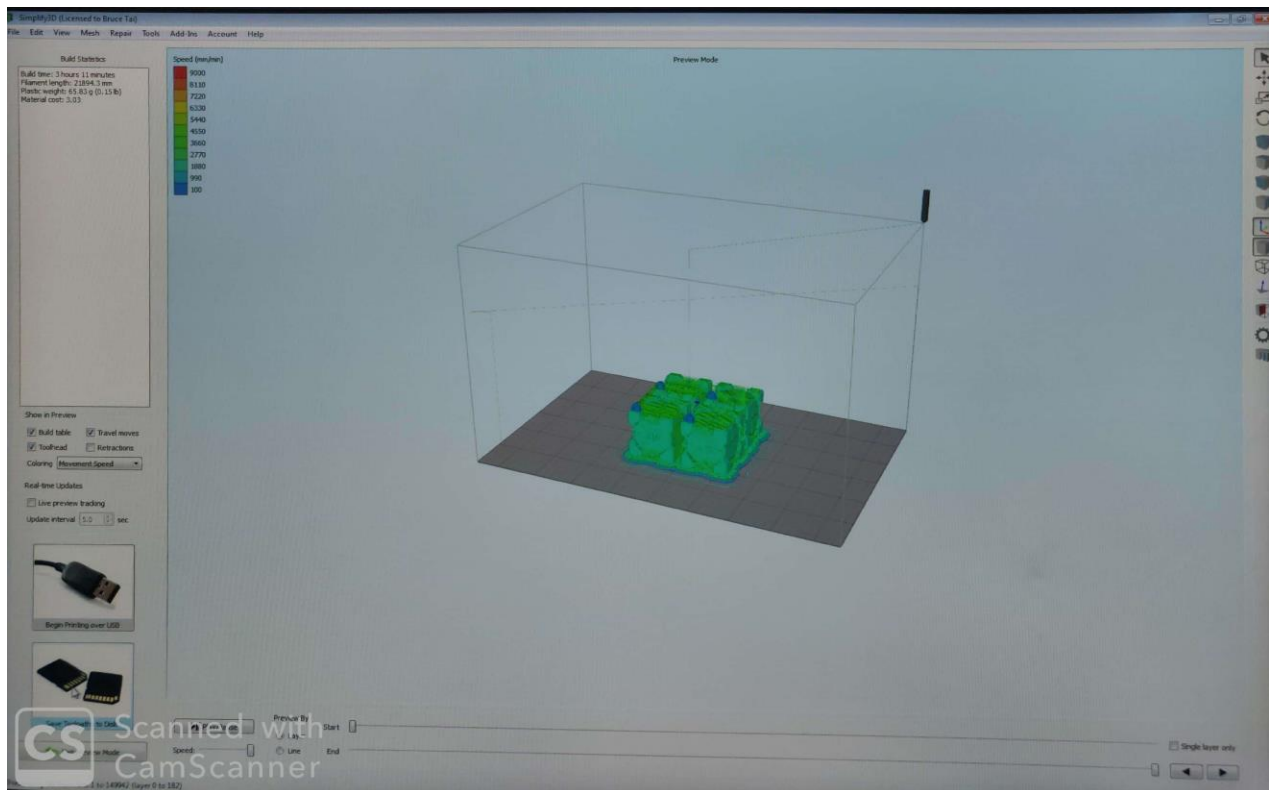
- 9) In order to ensure good part and print quality, support structures are absolutely necessary. To add supports, click the button that's called **"Support Generation"** (it's the very last icon on the right hand side). When clicking **"Generate Automatic Supports"**, Simplify3D will place supports everywhere it detects they are needed. By increasing or decreasing the **"Max Overhang Angle"** you can specify the criteria for when supports are added. Generally, this number is left at 40°. You can also increase and decrease the size of the supports depending on the part. The following picture shows the part with 4 mm supports.







10) Now that the supports have been added, the part is ready for printing. Click on **“Ready to Print”** in the bottom left corner of the screen. Insert an SD card into the computer and click **“Save Toolpath to Disk”**. Once saved, make sure to **“safely remove”** the SD card before taking it out of the computer. You can now put the SD card directly into the printer and begin printing.

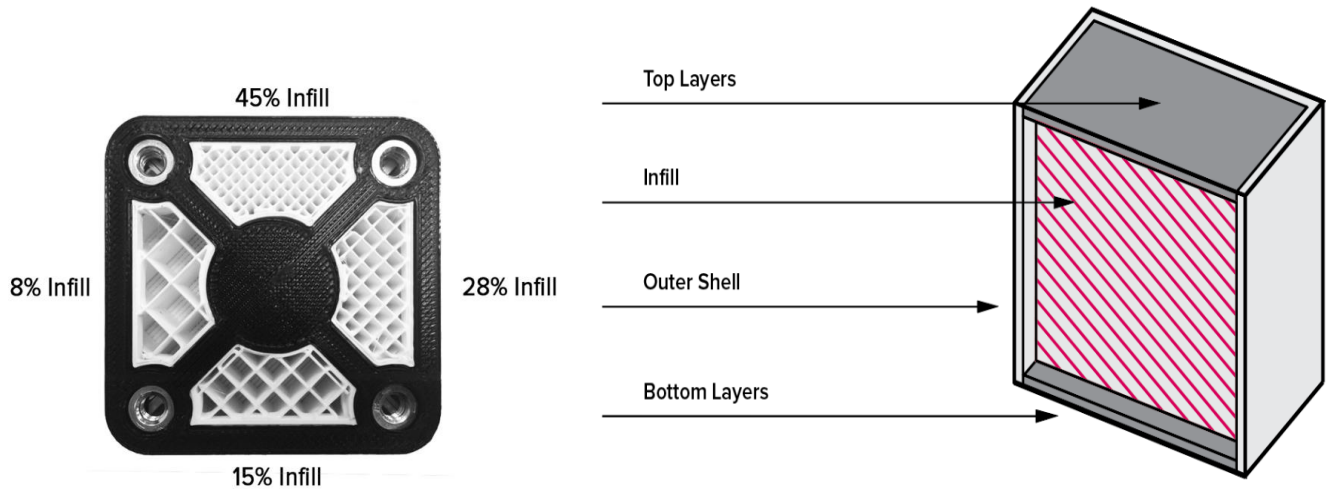


## Process Editing:

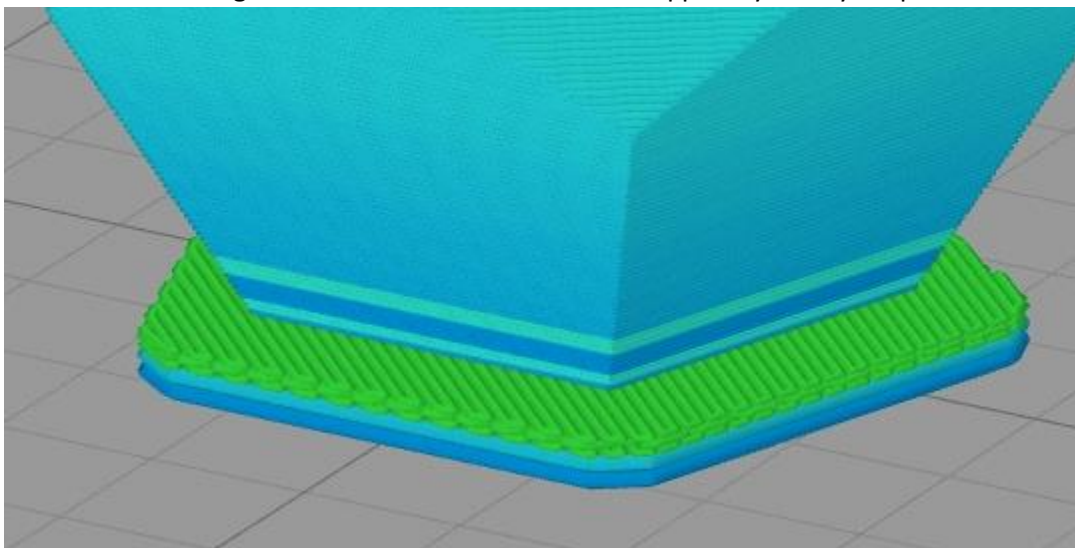
Under the option for “Edit process settings”, there are several parameters that can be tuned to improve the quality of the final print. Some of the most relevant parameters are discussed below

### General:

- 1) **Material Selection:** The material can be selected from a drop down menu in the settings dialog box.
- 2) **Infill Percentage:** It determines how much material is filled in the solid section of the part between layers. It's a percentage of the part solid volume. Generally speaking, the higher the infill percentage, the stronger the print, but the longer it takes to print. In most cases, 5% – 15% infill is sufficient. 100% infill is very rarely used.



- 3) **Rafts:** A Raft is a horizontal latticework of filament that is located underneath your part. Your 3D printed part will be printed on top of this raft, instead of directly on the build platform surface. Rafts are primarily used with ABS to help with warping and bed adhesion, but they can also be used to help stabilize models with small footprints, or to create a strong foundation on which to build the upper layers of your part.



### Under the “Extruder” tab of the dialog box:

- 1) **Nozzle Diameter:** Input the size of the nozzle you intend to use for the printing process

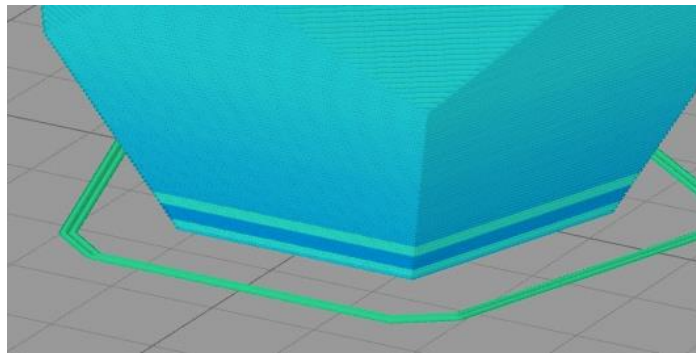
- 2) **Extrusion Multiplier:** Controls the amount of material flowing through the nozzle. The higher the multiplier the higher the flow rate

**Under the “Layer” tab of the dialog box:**

- 1) **Primary Layer Height:** Determines the thickness of each layer. The lower the thickness, the higher the resolution of the part (it will take longer though).
- 2) **Top and Bottom Solid Layers:** Number of layers to be printed at 100% infill in order to ensure a strong foundation for the part

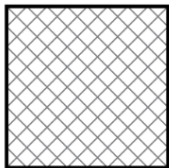
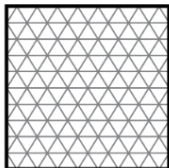

**Under the “Additions” tab of the dialog box:**

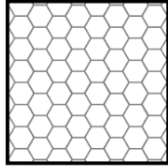
- 1) **Use Skirt/Brim:** A skirt is an outline that surrounds your part but does not touch the part. The skirt is extruded on the print bed before starting to print your model. Skirts serve a useful purpose because they help prime your extruder and establish a smooth flow of filament.



**Under the “Infill” tab of the dialog box:**

- 1) **Infill Pattern:** Determines the filling pattern within the shell of the print. Some commonly used Infill patterns are given below

Infill geometry	Description
	<b>Rectangular</b> - Standard infill pattern for FDM prints. Has strength in all directions and is reasonably fast to print. Requires the printer to do the least amount of bridging across the infill pattern.
	<b>Triangular or diagonal</b> - Used when strength is needed in the direction of the walls. Triangles take a little longer to print.
	<b>Wiggle</b> - Allows the model to be soft, to twist, or to compress. Can be a good choice particularly with a soft rubbery material or softer nylon.



**Honeycomb** - Popular infill. It is quick to print and is very strong, providing strength in all directions.

### Under the “Temperature” tab of the dialog box:

- 1) Setting the printing temperature is **extremely important** to ensure good print quality. The temperature can be set to affect the a) Nozzle temperature and b) the Bed temperature. In order to change the temperature, select the extruder or the bed from the box on the left and input the temperature value. This is primarily material dependent and the recommended temperatures for PLA and ABS are given below:

**ABS (very temperature sensitive):**

**Bed temperature** - 100 °C to 105 °C

**Nozzle Temperature** - 230 °C to 250 °C

**PLA:**

**Bed Temperature** - 60 °C - 65 °C

**Nozzle Temperature** - 205 °C to 215 °C

