



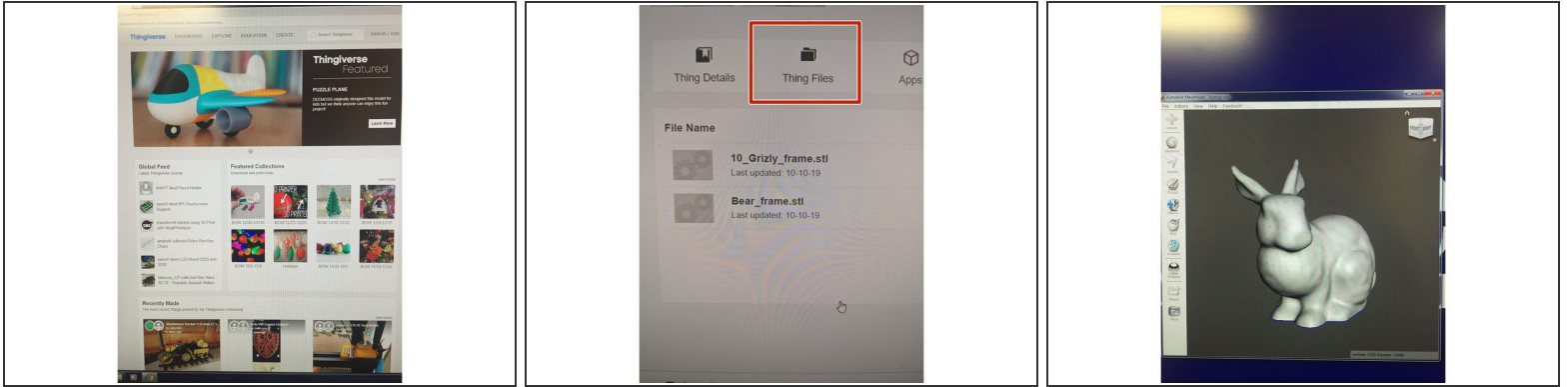
Lulzbot Mini

This tutorial is a basic tutorial on the Lulzbot Mini 3D printers, which can be accessed and managed wirelessly via RaspBerry Pi running OctoPrint.

Written By: Sabrina Shankar

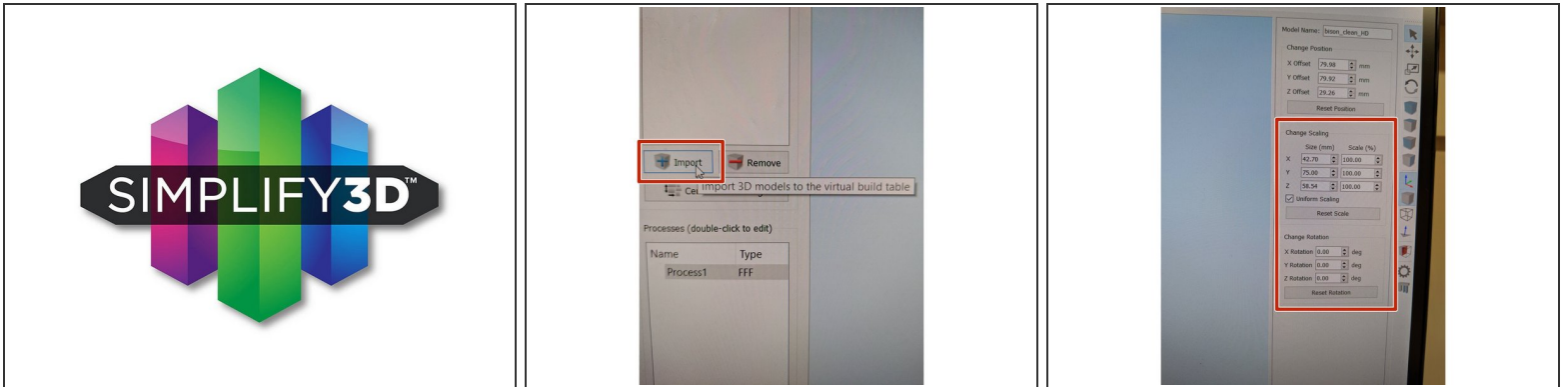


Step 1 — Saving your File



- In order to print on the Lulzbot Mini you must have an STL file.
- If you are creating your own 3D print file in SolidWorks, Rhino, SketchUp, etc. please save your file as a binary stl.
- If you are downloading your file from an online source such as thingiverse.com, the file should already download as an stl.
- ❗ When downloading from thingiverse (or a website like this), download the files individually - not as a folder. This will prevent you from having to extract the files later on.

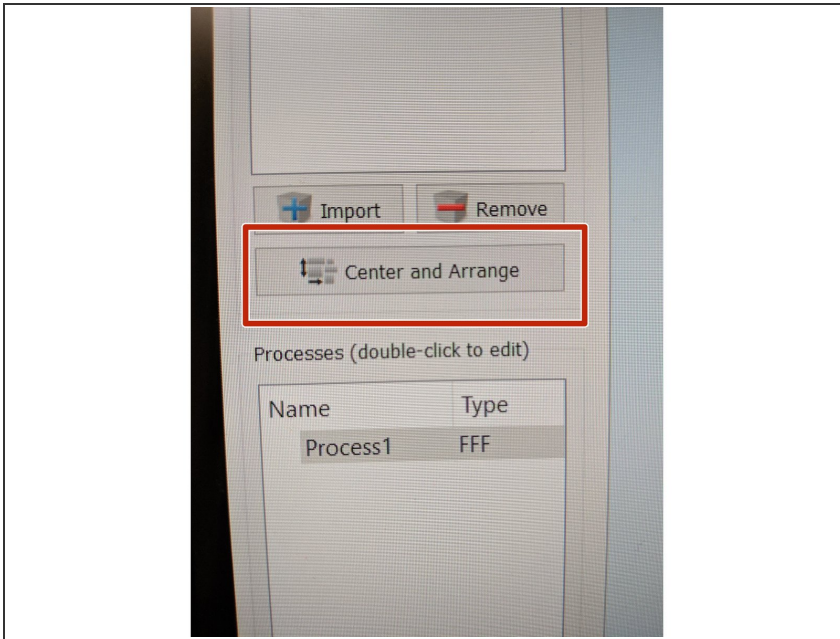
Step 2 — Simplify 3D



i In order to 3D print, you need to create a .gcode.

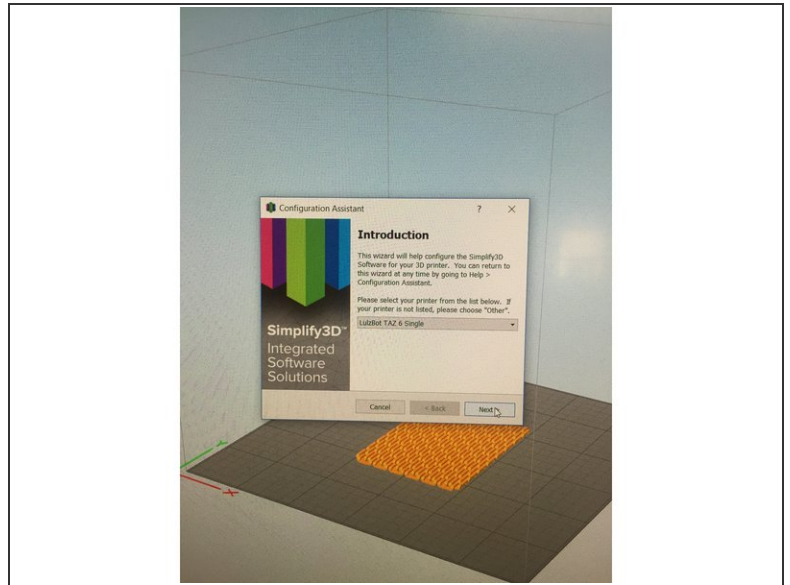
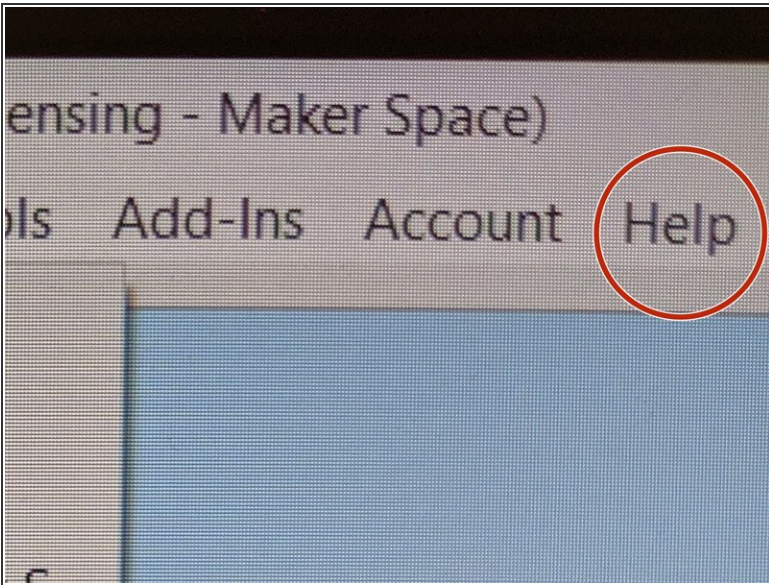
- Open Simplify 3D.
 - You may need to log in. Please use the username/password on the whiteboard or on Makerspace computer monitor. When prompted, select the printer type as Lulzbot Mini.
- ⚠ If you are 3D printing on Lulzbot Mini 4, select Lulzbot Mini 2 as the type of printer you are working with.**
- Import your model by clicking "Import" or pressing File -> Import Model.
- Check to make sure your file is oriented correctly - ideally you want it oriented in whatever position will yield the least amount of support material.
 - To change the orientation of your model you need to alter the x, y, or z positioning until you are in the most optimal position. (Typically the longest, flattest side should be flush against the bed).
- If you need to change the scaling, double click on the object and change the scaling on the right side. Make sure the "Uniform Scaling" box is checked if you want the object to scale uniformly.

Step 3 — Center and Arrange



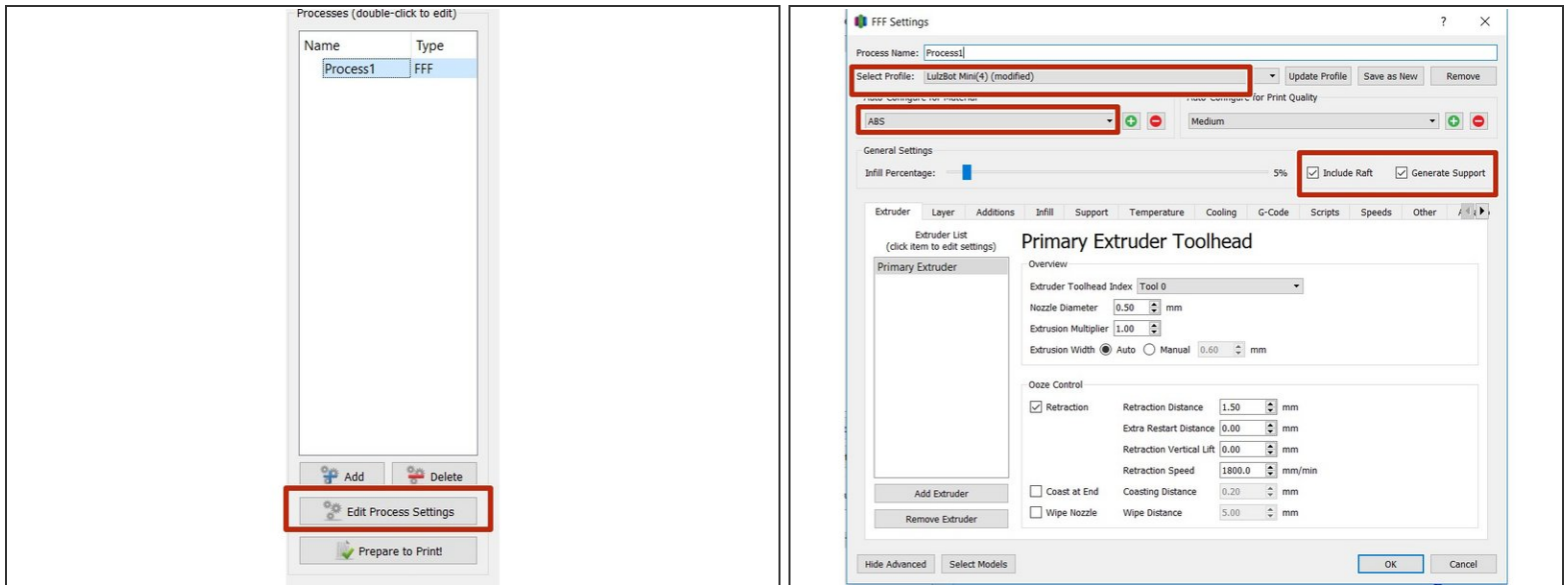
- Once you're satisfied with the way your print looks, press the "Center and Arrange" button on the left side of the screen.
- ① This will put your print in the most optimal position for the 3D printer to print.

Step 4 — Set up your FFF Profile



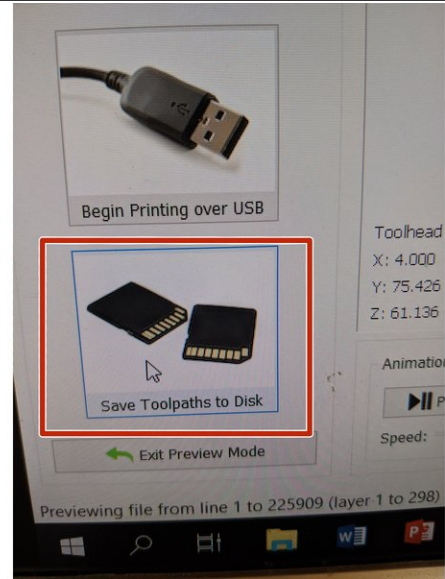
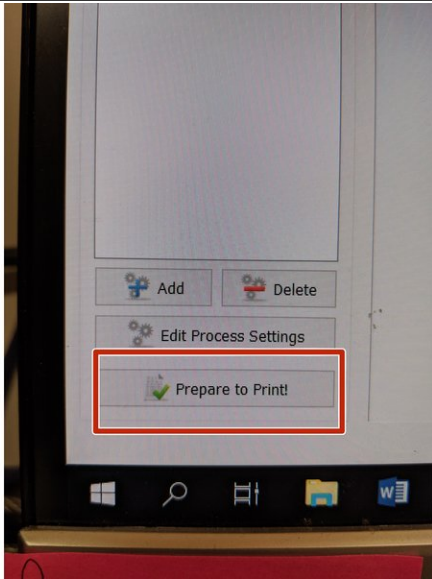
- Now we want to make sure our file is set up for the printer type we intend on using.
- In the top toolbar, click Help -> Configuration Assistant.
- A pop - up box will appear. Under "Select Printer" select Lulzbot Mini (unless you are printing with Lulzbot Mini number 4 then select Lulzbot Mini 2).
- Click "Finish".

Step 5 — Altering FFF Profiles



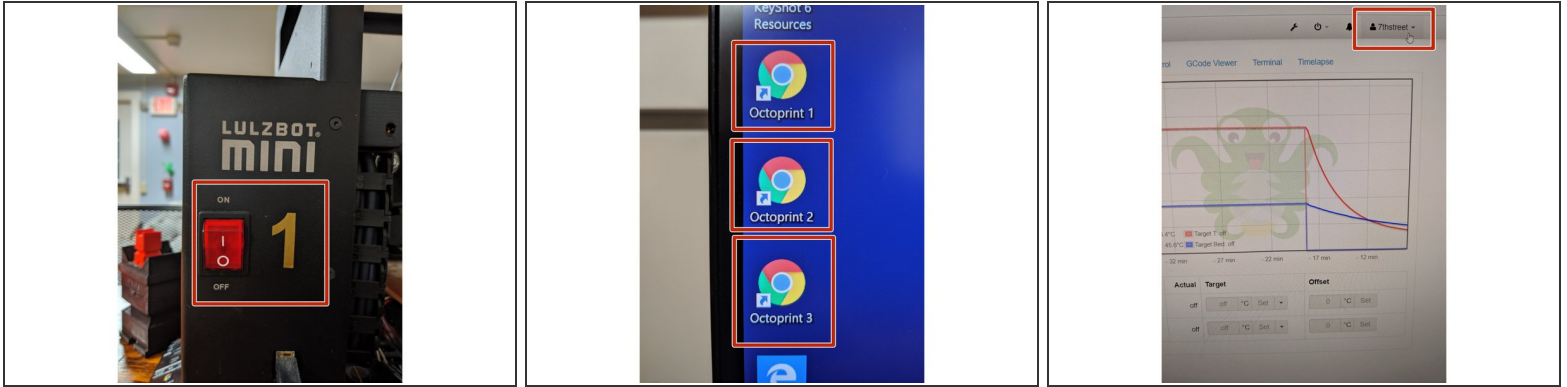
- Click on "Edit Process Settings" in the bottom left corner of the left toolbar. This will open a new window which you will use to edit the FFF settings.
- Make sure the Profile selected is the name of the profile that you just imported otherwise click the drop down arrow and select the correct profile.
- Then select the material type you are working with today "ABS" or "PLA". To know which material you are working with, look at the roll of vinyl on the printer you will use. It will say it on the roll.
- If the item you are printing has overhangs 45 degrees or greater, you will need to add supports. Check the box next to "Generate Supports".
- If you add supports or are printing an object with a curved bottom, you will want to include a raft at the bottom of your object. Check the box that says "Include Rafts".
- Click OK.

Step 6 — Prepare your File to Print



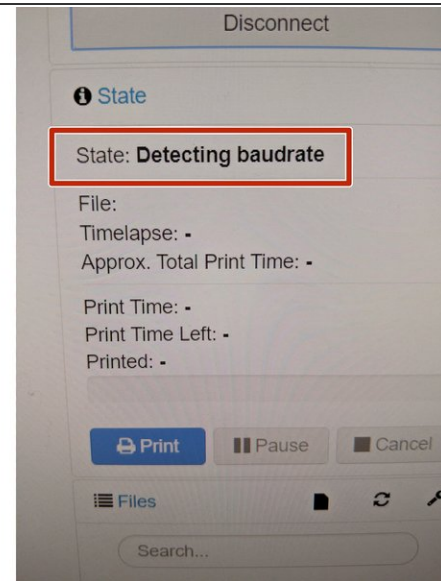
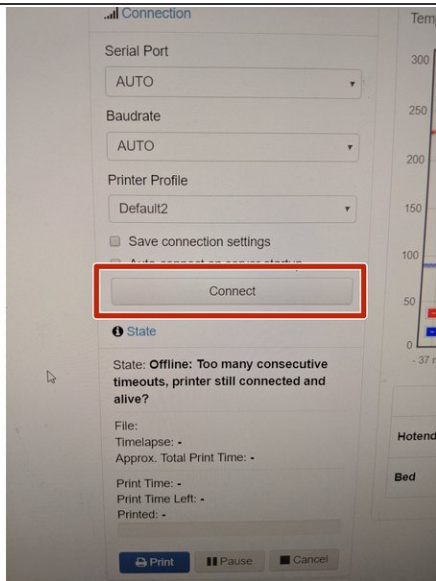
- Your file is now ready to print and ready to generate a .gcode.
- To do this, select "Prepare to Print" in the bottom left side.
- Then select "Save Toolpaths to Disk".
- Save the .gcode file in your documents.

Step 7 — Starting the 3D Printer



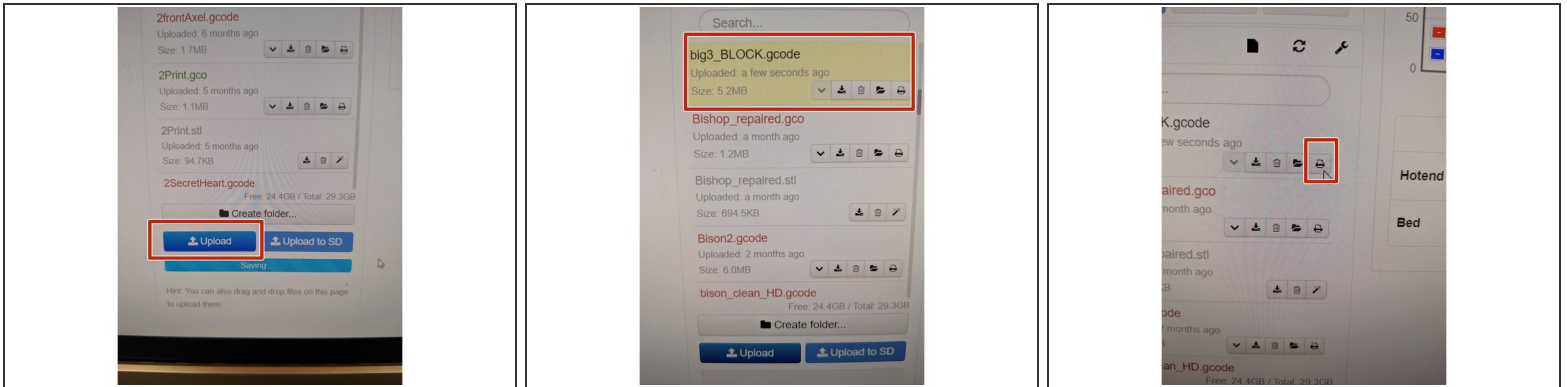
- Determine which Lulzbot Mini you will be using and look at the golden number on the machine. Turn on the printer.
- On the desktop of your computer, there are 4 different address links each connecting to a different 3D printer. Open the Octoprint number that corresponds to the 3D printer you are using.
- Login on the top right hand corner of the screen.
 - ① Username: makerspace
 - ① Password: octoprint

Step 8 — Connecting to the OctoPrint Server



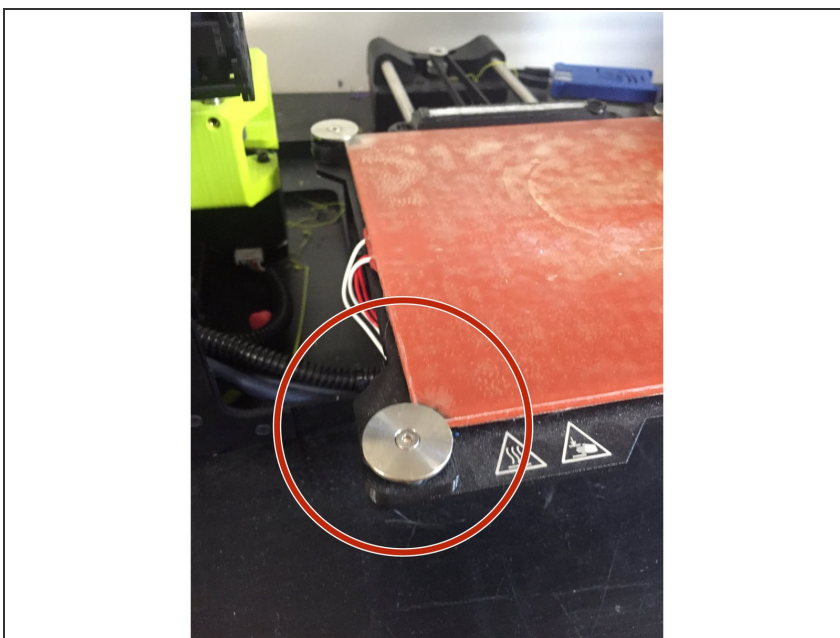
- Click the "Connect" button located on the left-hand side of the screen. This will connect our Octoprint to the 3D printer.
 - If your printer is not connecting, make sure it is turned on!
 - The state will now say "Detecting Baudrate".
 - Once the state says "Operational" you are ready to upload your file.
- i** If it does not say Operational but instead gives an error message. Try toggling the on/off switch of your machine and starting again. If this doesn't work, please talk to a student employee.

Step 9 — Uploading Your Model



- Upload your .gcode file by dragging your file onto the left side of the OctoPrint Screen or by pressing the "Upload" button on the bottom left corner.
- Navigate your file and open it.
- Your file is now saving to the queue. Once it is uploaded it will flash yellow in the queue.
- Press the printer icon next to your file now.
- The 3D printer will now begin printing your file.

Step 10 — Stay with your print!



- After starting your print, you must stay to watch the first 10 to 20 layers. This is really important because prints tend to fail within the first couple of layers. It is better to wait 15 minutes for the first layers to print then to come back hours later and see the print has failed.

- Once the printer is at the correct temperature, it will begin moving down and touch-pointing each of the

four metal circles in the corners.
This is how the machine levels its
bed!

Step 11 — Removing your print



- Once your print is finished and cooled, slowly remove the plastic model from the tempered glass, applying light pressure with a pry tool (if needed).
- Usually it will come off easily, but be patient and careful to avoid damaging the print bed.
- Remove the raft and supports (if used on your print).