

Converting Your 3D Model into A Printer File (Slicing)

Purpose: This document will walk you through how to convert your 3D model file into the file format needed by the 3D printer (.X3G). This process is known as slicing. This is done with the Ultimaker Cura software package.

Step 1:

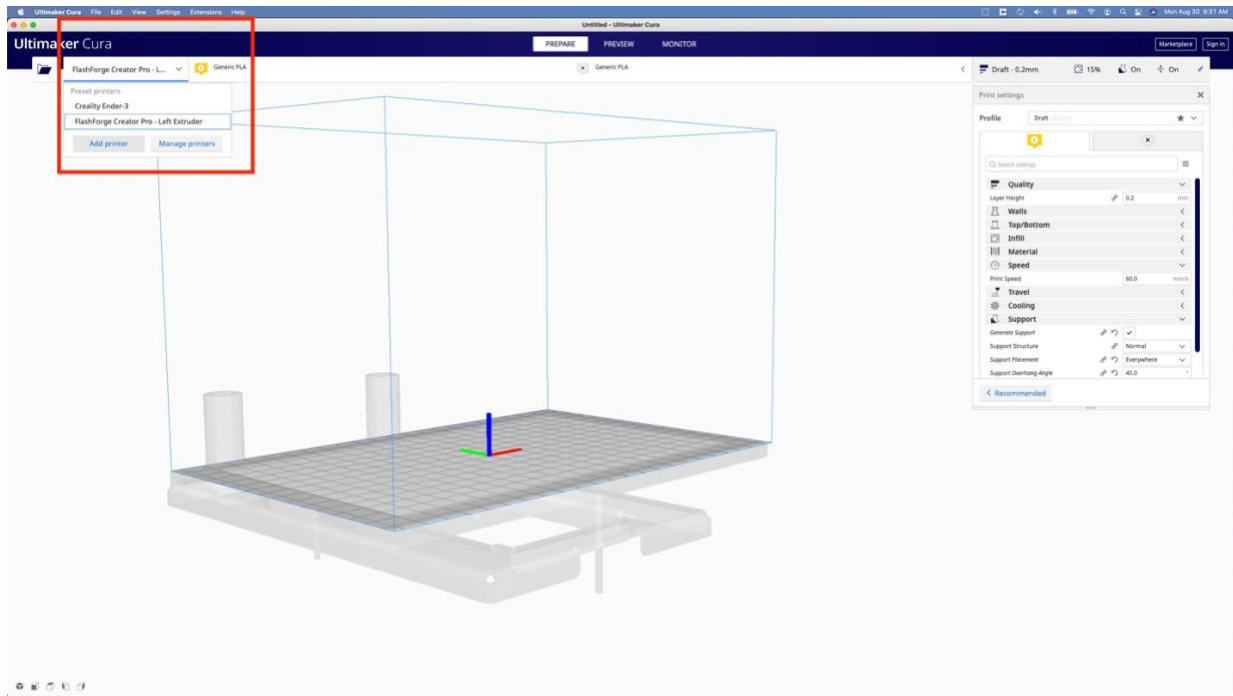
Open Ultimaker Cura on the iMac machine located next to the 3D Printer. The icon looks like this:



Step 2:

Confirm that the printer profile is set to '*FlashForge Creator Pro – Left Extruder*' or '*FlashForge Creator Pro – Right Extruder*' depending on your desired filament. The Flashforge Creator Pro is not a built-in printer profile in Cura, so it must be added to Cura by placing files into Cura's directory. If you are wanting to add FlashForge Creator Pro to Cura on your own machine, follow the instructions [here](https://www.thingiverse.com/thing:3569113):

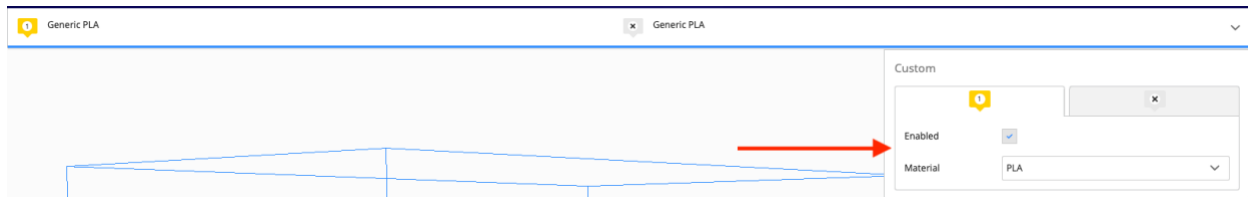
<https://www.thingiverse.com/thing:3569113>



Selecting FlashForge Creator Pro Profile

Step 3:

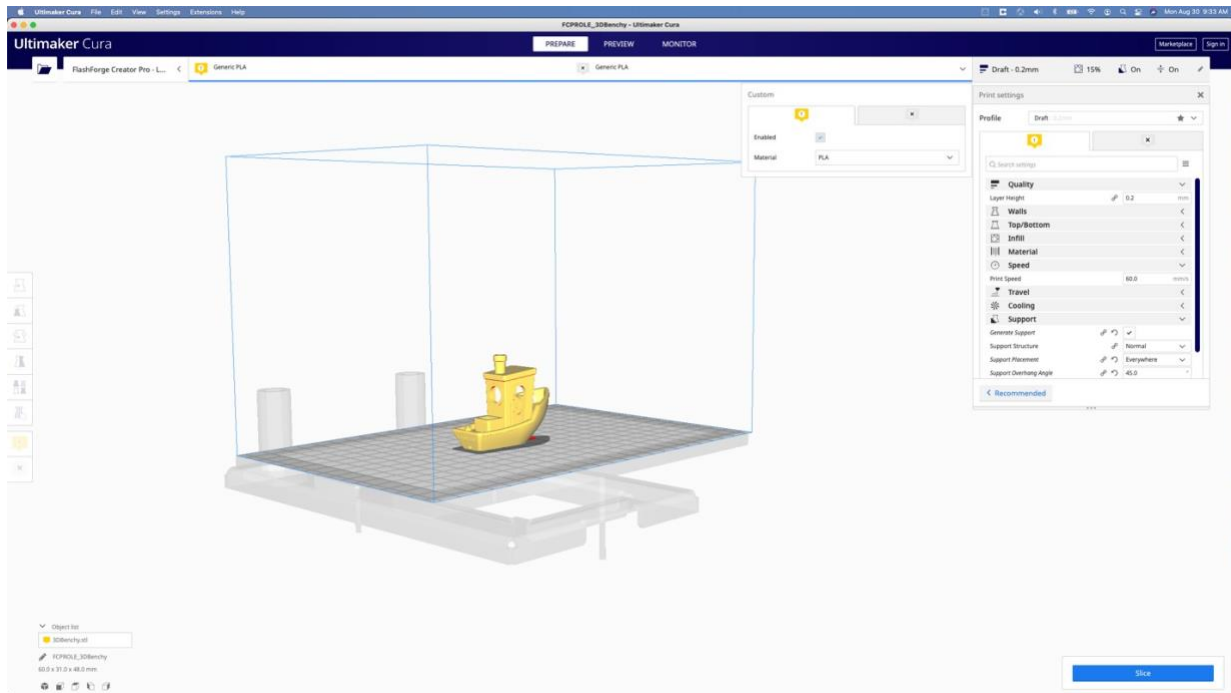
Make sure that only the correct extruder (nozzle) is active, and that PLA is set as the material. PLA is the only plastic allowed for use in the library as it is relatively non- toxic.



Extruder and nozzle selections

Step 4:

Add your model file by clicking the folder icon in the upper left, or simply dragging and dropping your model into Cura.



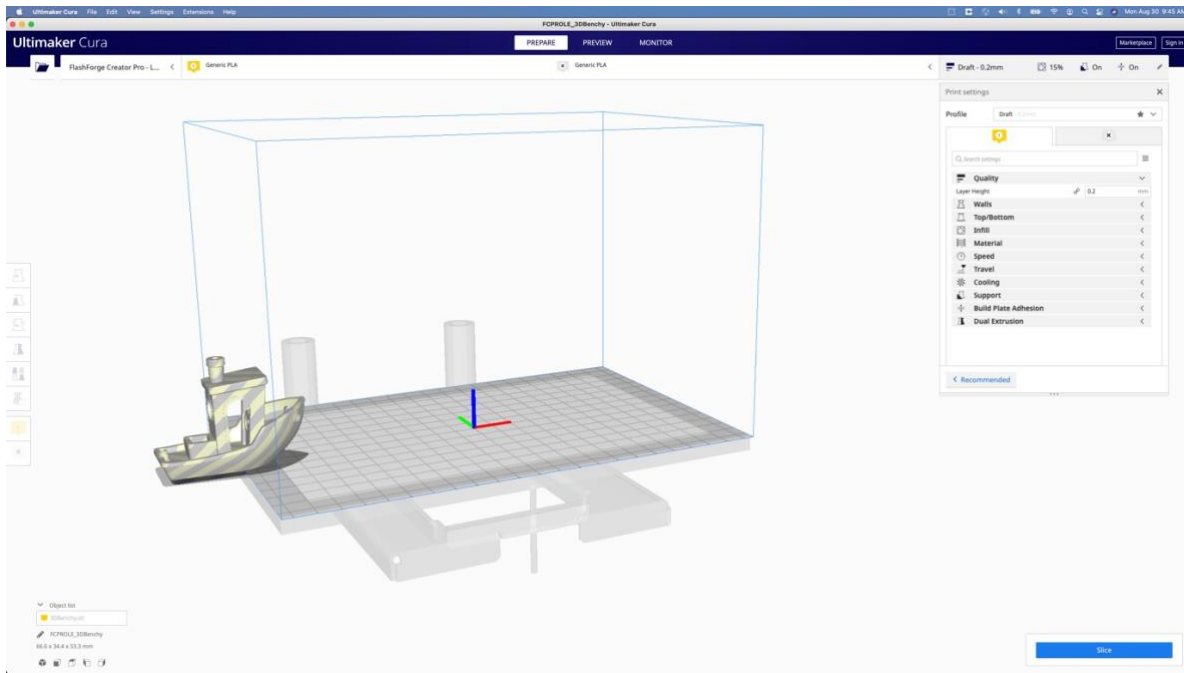
Cura with model added

After you've imported a compatible 3D file, you should see your model on the print bed. The grid pattern represents the base of the print bed and the end opposite the 2 tubes represents the front of the printer. The thin, blue lined box represents the build volume of the printer.

You cannot have models outside of this box. If you do have a model outside of this space, you'll see the model represented with yellow and gray lines. You will receive a warning if you try to slice a model while not being within the build volume of the printer. **Please do not attempt to continue printing when you have an item outside of the build volume.** Cura should prevent this from happening, but please do not attempt to force it. A way to help correct this issue, especially when you have multiple models on the print bed is to go to:

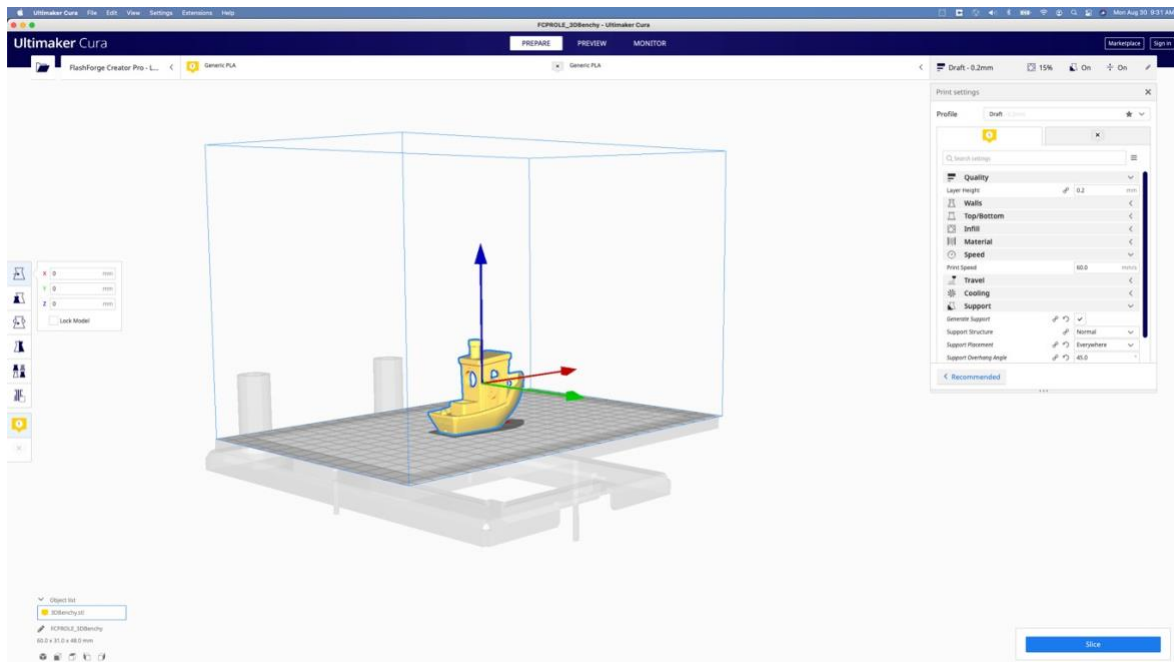
Edit -> Arrange All Models

If your model(s) are still protruding out of the bounding box after performing this, then you have too many models packed into one print, or your model file is too large to work without scaling it down.

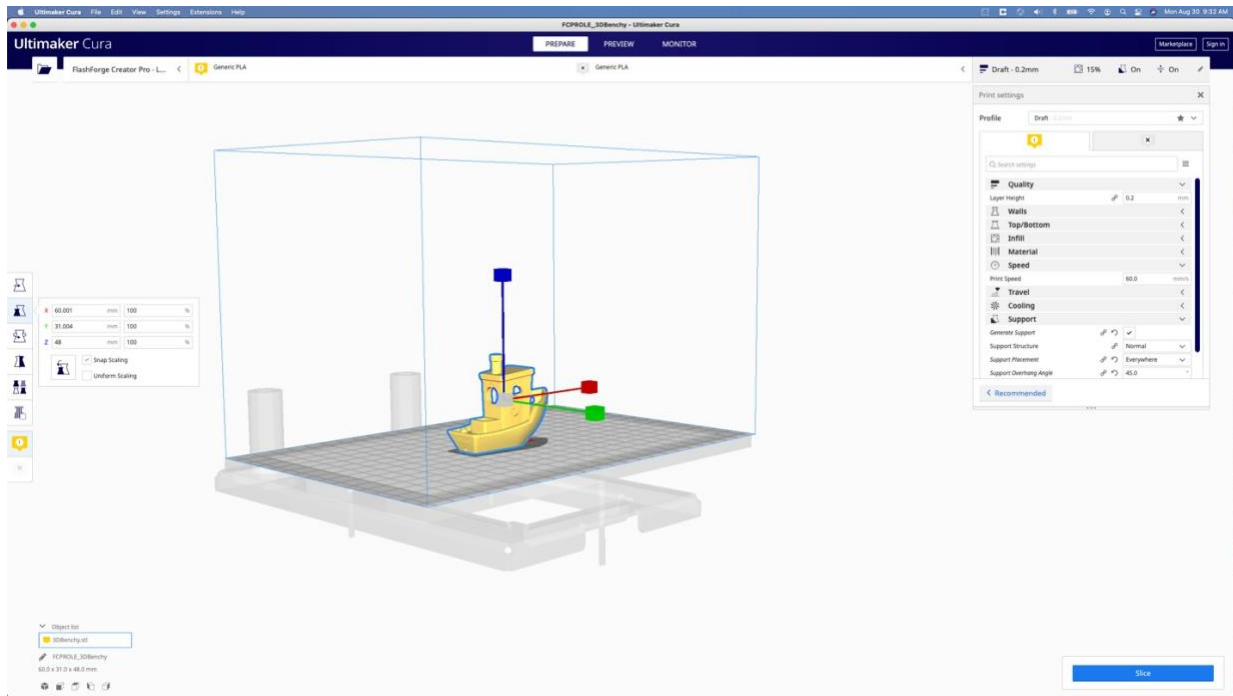


A model that is outside of the build volume of the printer

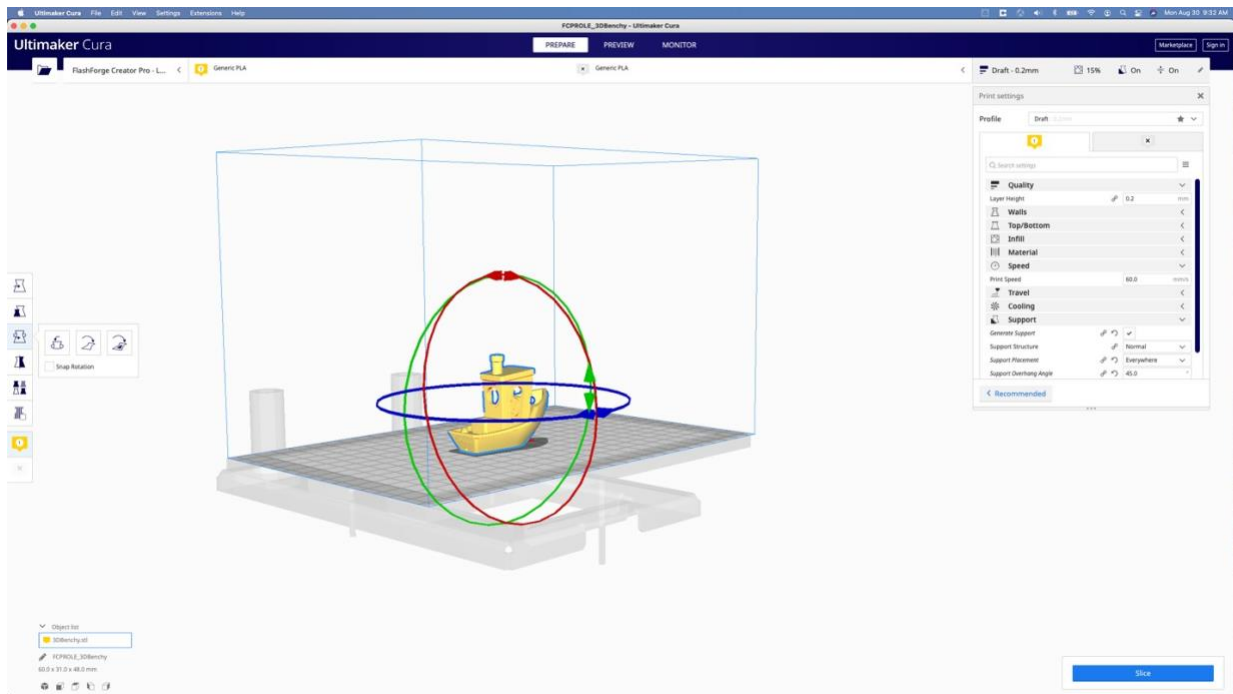
There are several tools that help you move, scale, rotate, reflect, and more. These tools are located along the left vertical menu. Some of these choices have sub-menu choices for more precise manipulation of the model.



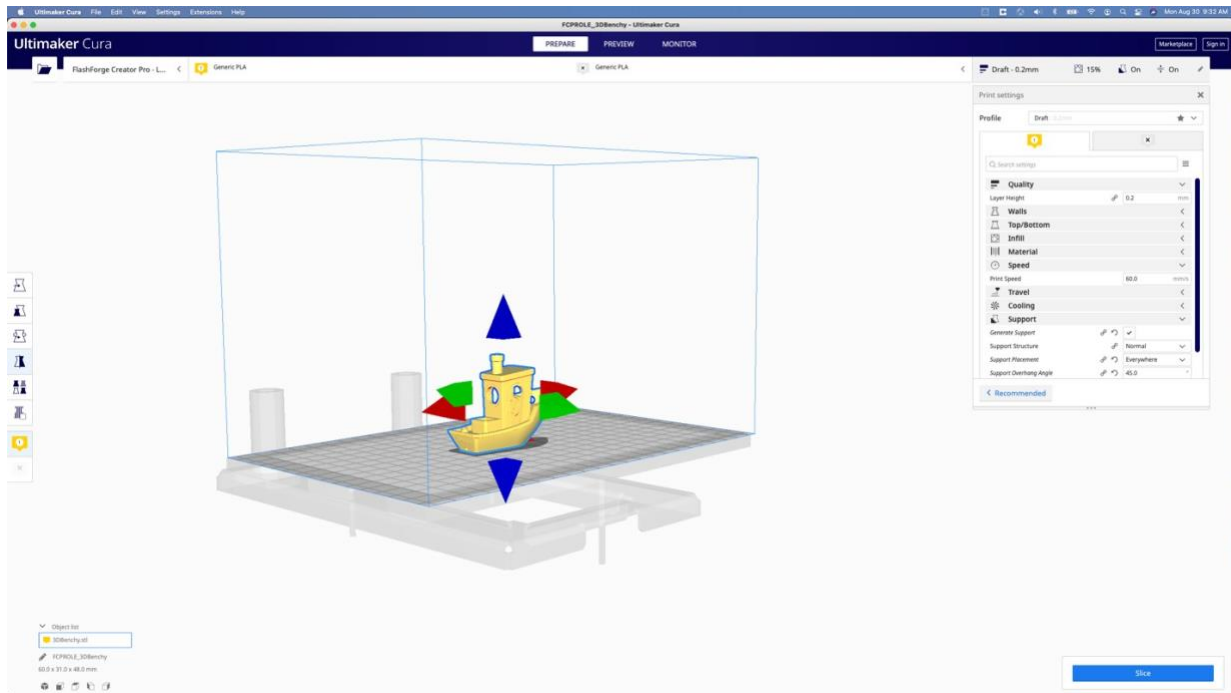
Move a model



Scale a model



Rotate a model

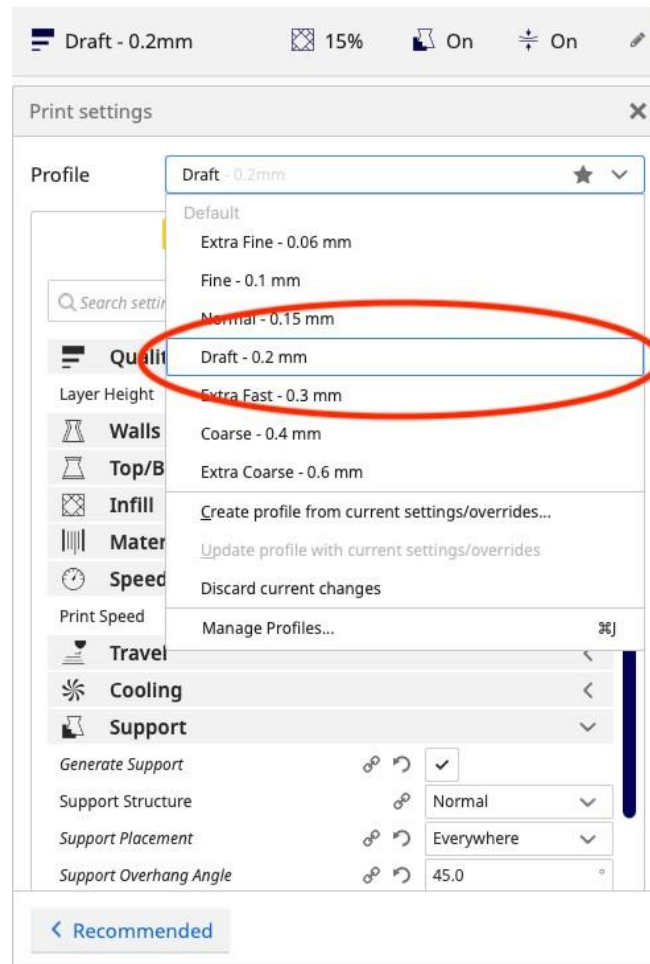


Mirror a model along an axis

Step 5:

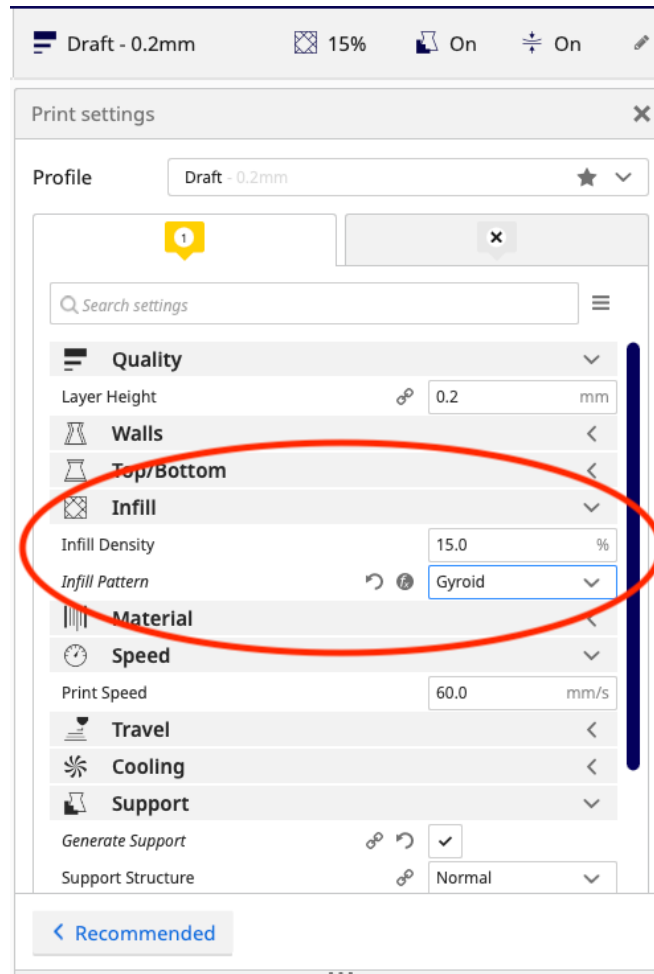
With your model imported and properly placed, you will move into the dropdown menu on the righthand side to set some specific values for temperatures, infill, supports, etc.

First, you can choose the quality preset. Typically, 'Draft – 0.2mm' is more than adequate. This is a standard layer height for many printers. Occasionally, you may wish to choose 'Normal' or 'Fine'. The tradeoff is time. A larger print line means quicker printer but lower resolution. A finer print line means longer printing but higher resolution. Prototype with larger print line size for quicker printing.



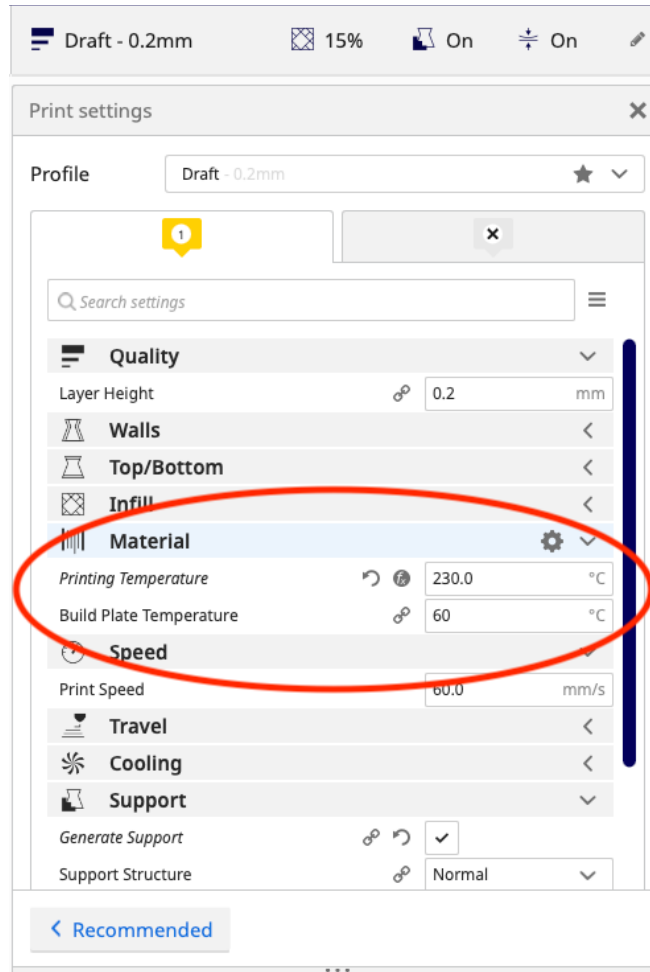
Layer height choice

Next, you'll make your choices for infill percentage and type. Infill is a support structure that provides structural integrity to the inside of a model without having to create a fully solid plastic object. Typically, anywhere from 15% to 25% is a large enough infill, however, this value can change depending on your intentions for the final object. Gyroids and hexagons are sturdy yet lightweight structures and are recommended.



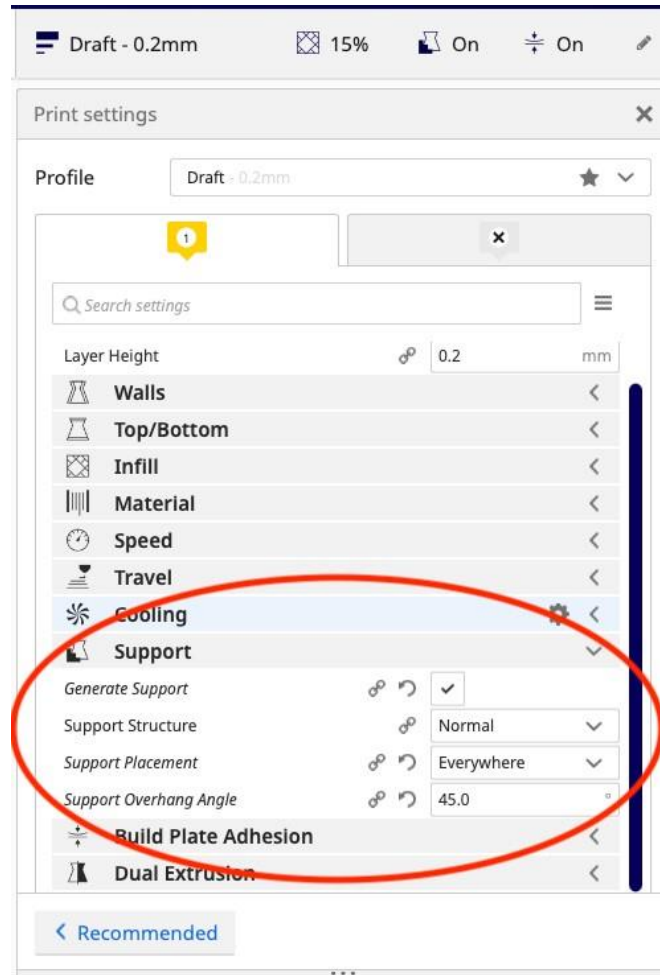
Infill options

Next you need set the temperature for both the extruder and the print bed. This is found under the 'Material' drop down. For the extruder, set the temperature to 220°. For the bed, set the temperature to 60°. These degrees are in Celsius and have been the temperatures we have seen the most success with on these printers.



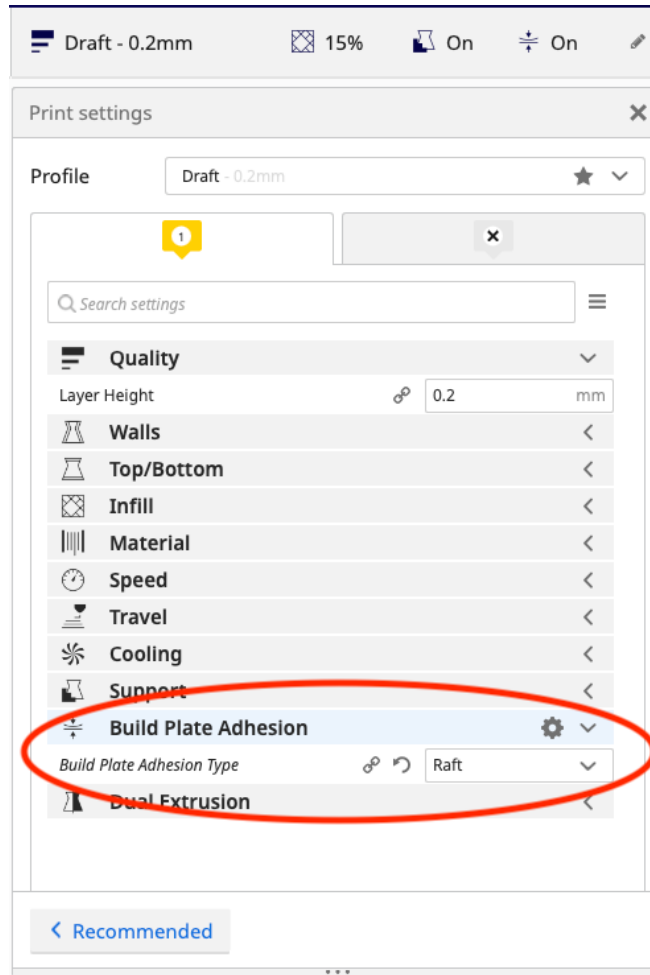
Temperature Options

Next, you will choose to utilize supports or not. Supports are added-in structures that support your model while it is being printed. Typically, the rule of thumb is to add support to anything that is overhanging by 45° unsupported. Cura offers “normal” or “tree” support structures, however, “normal” is the standard choice. The support material is removed from the object when the print is complete. Feel free to play with supported or unsupported prints, normal or tree, and even the overhang value. You will gain a better sense of this with practice.



Support Options

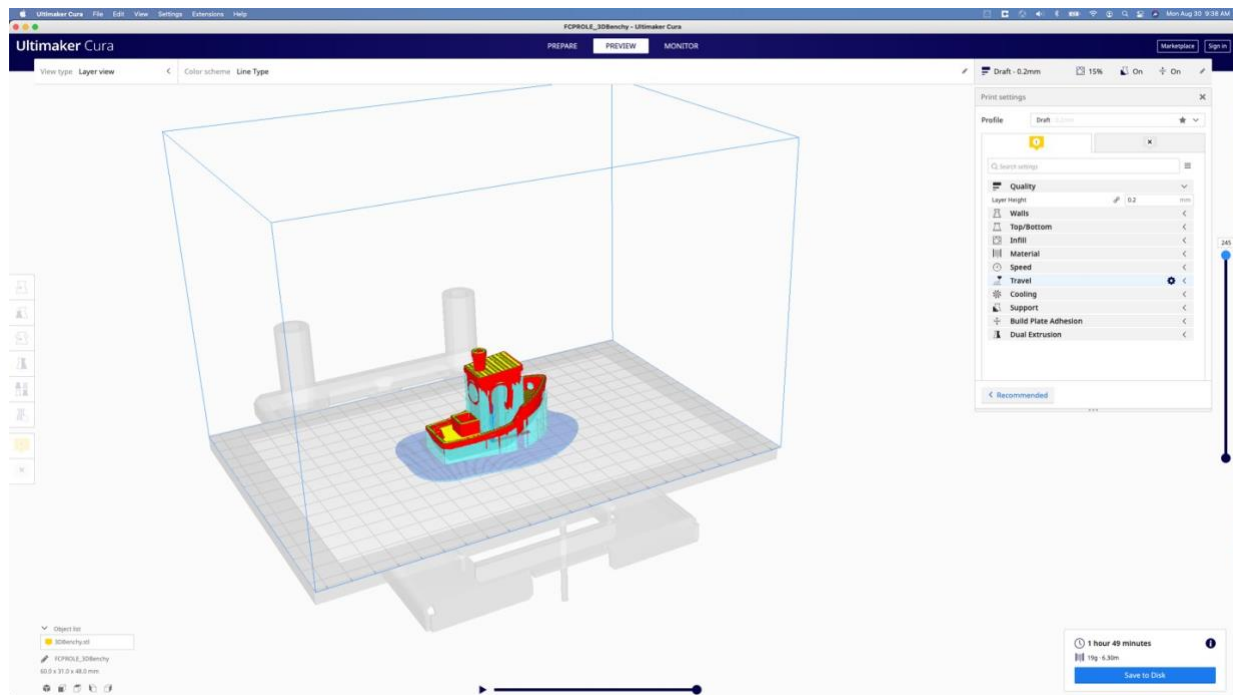
Finally, you will want to choose a build plate adhesion type. These are options that prime the nozzle or help to ensure your print will adhere to the print bed, increasing the chances of a successful print. For users that are new to 3D printing, it is recommended to use a raft. A raft is a plastic platform that will be created on which your model will then be printed on top of. This plastic platform is then separated from the model when the print is completed.



Build Plate Adhesion Options

Step 6:

After making all your choices for print settings, you are ready to “slice” your model and save the file. Choose the ‘Slice’ button in the bottom right corner. Cura will take moment to process the print. After this completes, you can choose to preview the model, or save the file. Selecting preview will update the window to show you your print, the support, and any other aspects of the finished print.



Preview of your print

Finally, you'll want to take the SD card out of the printer you will be using and insert it into the iMac computer. Click "Save to Disk" and save your sliced file to the SD card. Most printers work with g-code files, however, FlashForge uses the .X3G file format. So, you'll be saving an .X3G file to this disc. For example: bench.x3g. After you have saved this file to the SD card, you can eject it from the iMac, load it back into the 3D printer, and launch the file.

Please ask the DLMC Media Mentors if you have questions or need help.