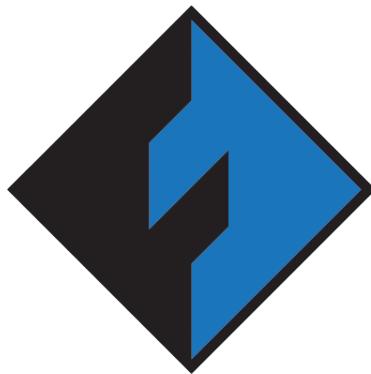


## Converting Your 3D Model into A Printer File (Slicing)

**Purpose:** This document will walk you through how to convert your 3D model into the file format needed by the 3D printer through a process known as slicing. This is done with FlashPrint, a slicing software developed by the manufacturer of our 3D printers, FlashForge.

### Step 1:

Open FlashPrint on the iMac machine located next to the 3D printers. The icon looks like this:

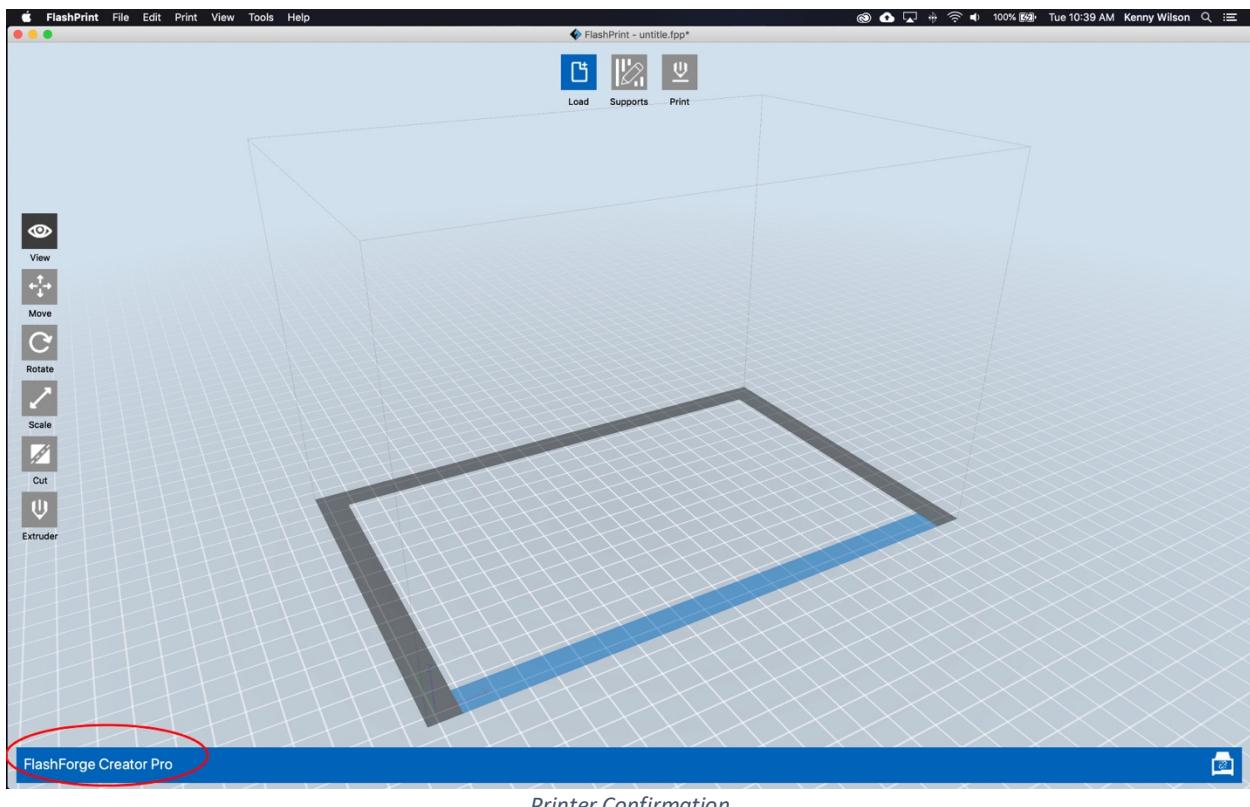


*FlashPrint Icon*

### Step 2:

Confirm the printer is set to ‘FlashForge Creator Pro’ in the bottom left corner. If it is not defaulted on ‘FlashForge Creator Pro’, you can change it to Creator Pro by going to:

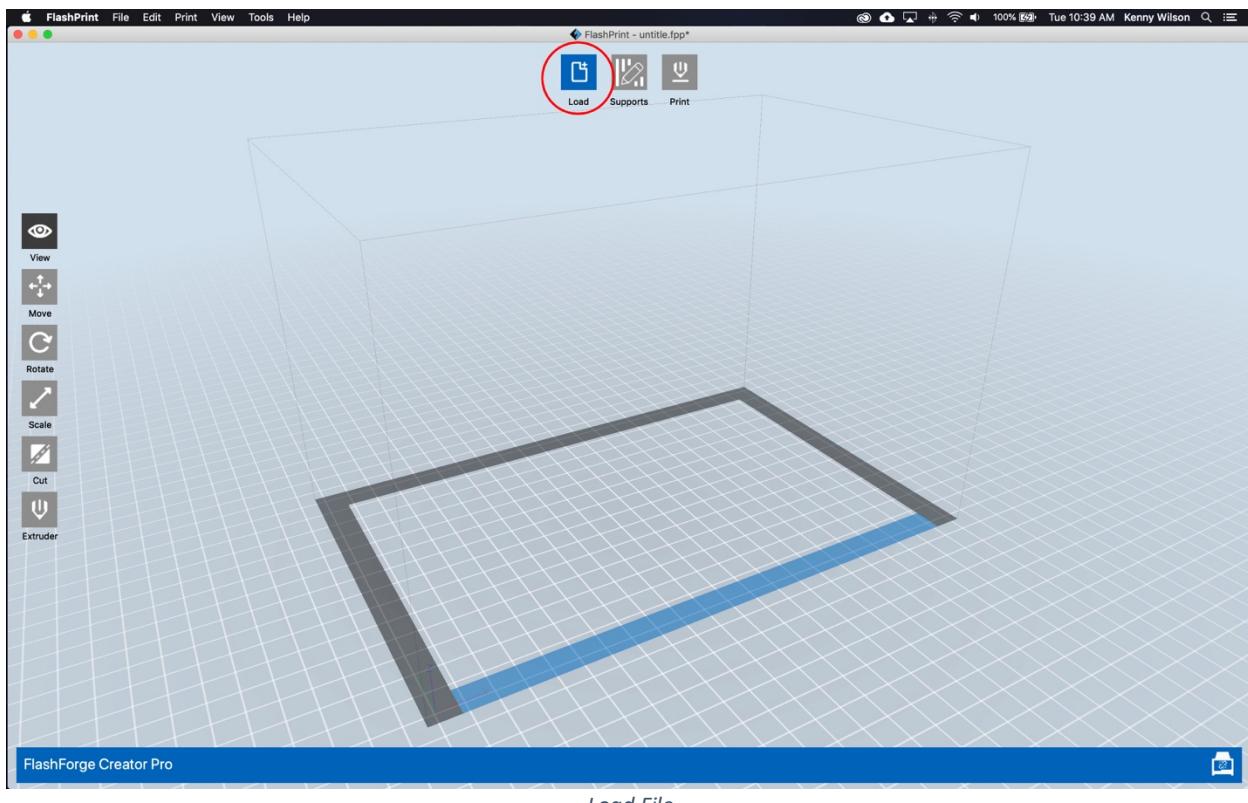
*Print -> Machine Type -> Flashforge Creator Pro*



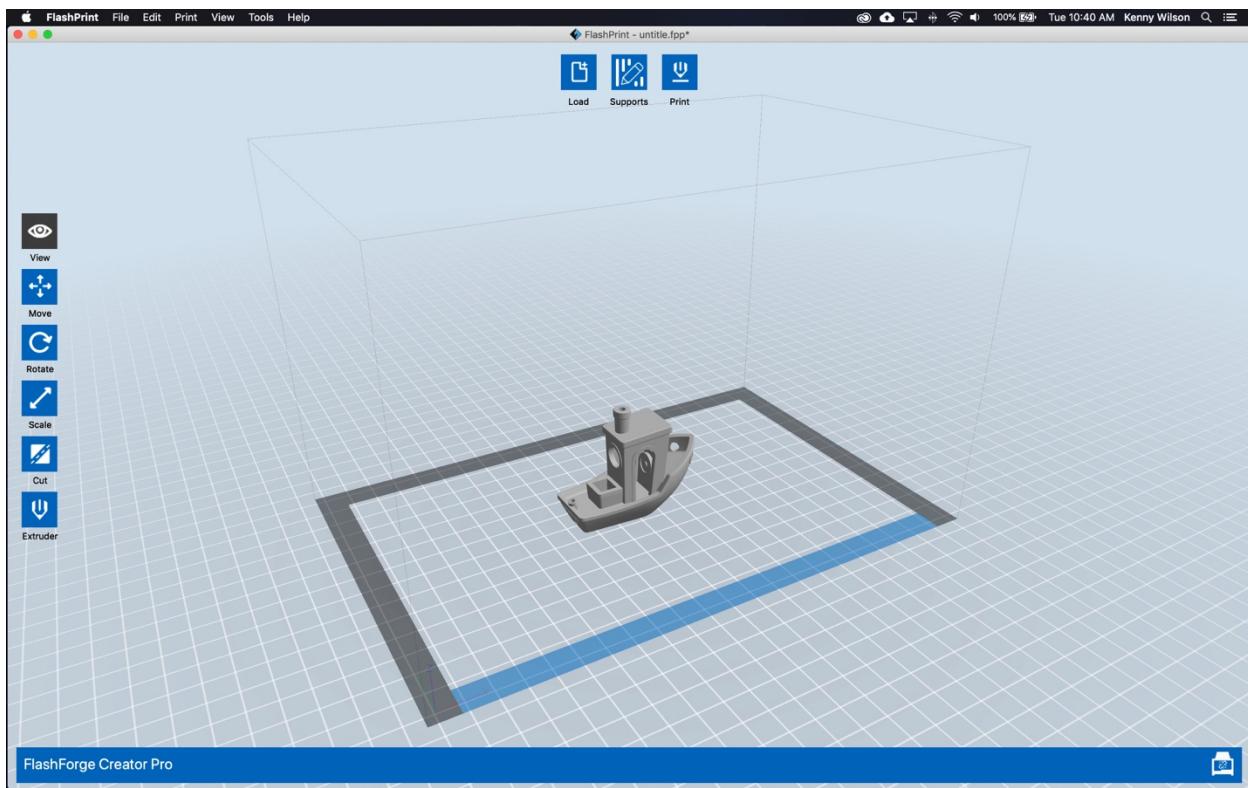
*Printer Confirmation*

### **Step 3:**

From the home screen, click on the button labeled ‘load’ located in the upper center of the window.



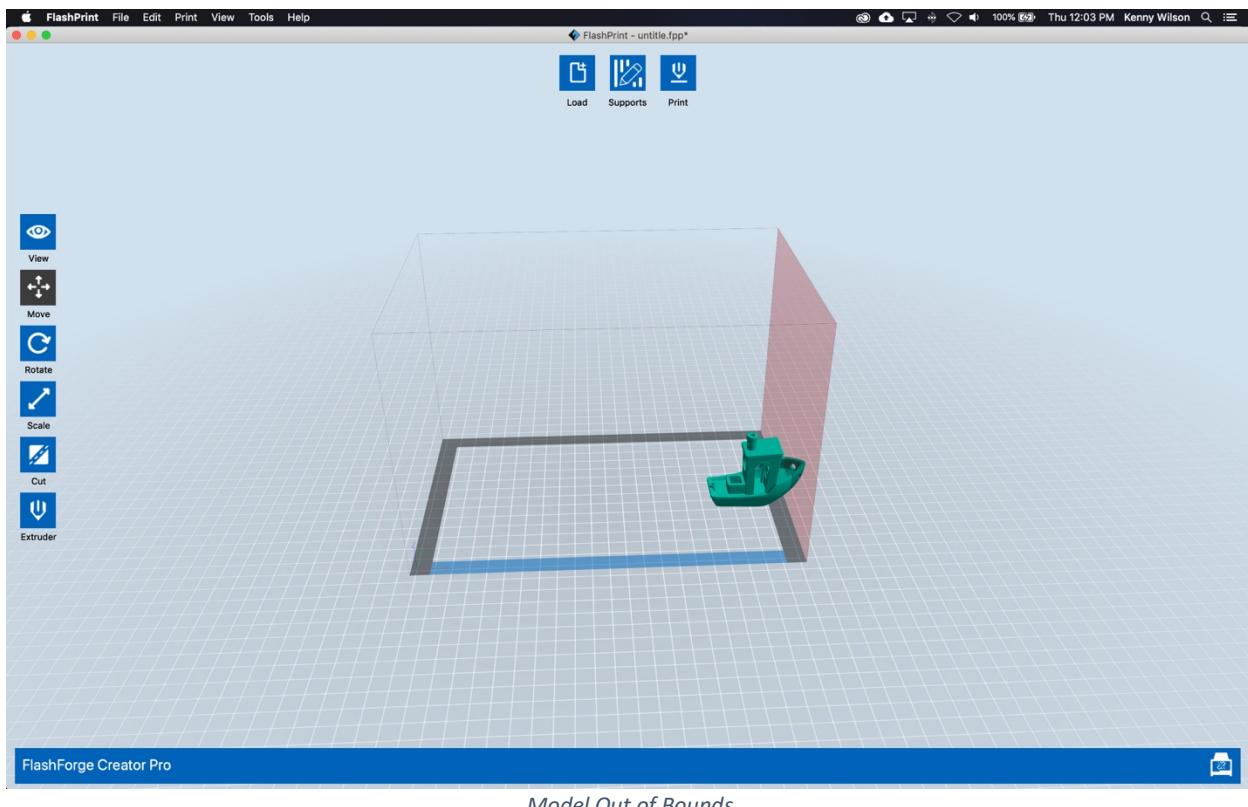
After you've imported a compatible 3D file, you should see your model on the print bed. The grid pattern represents the base of the print bed and the solid blue line indicates the front of the print space. The thin, grey lined box represents the total space, or volume, of the 3D printer.



**You cannot have models outside of this box.** If you do have a model outside of this space, you'll see a red wall on the side of the box you are protruding out from. You will receive a warning if you attempt to continue to save the print file while having a model outside of this bounding box. **Please do not attempt to continue printing when you have an item outside of the bounding box as it will cause damage to the 3D printer.** A way to help correct this issue, especially when you have multiple models on the print bed, is to go to:

*Edit -> Auto Layout All*

You can select 'OK' when prompted with the spacing. If your model(s) are still protruding out of the bounding box after performing this, then you may have too many models packed into one print. Or your models may simply be too large and you'll need to scale them down to fit.



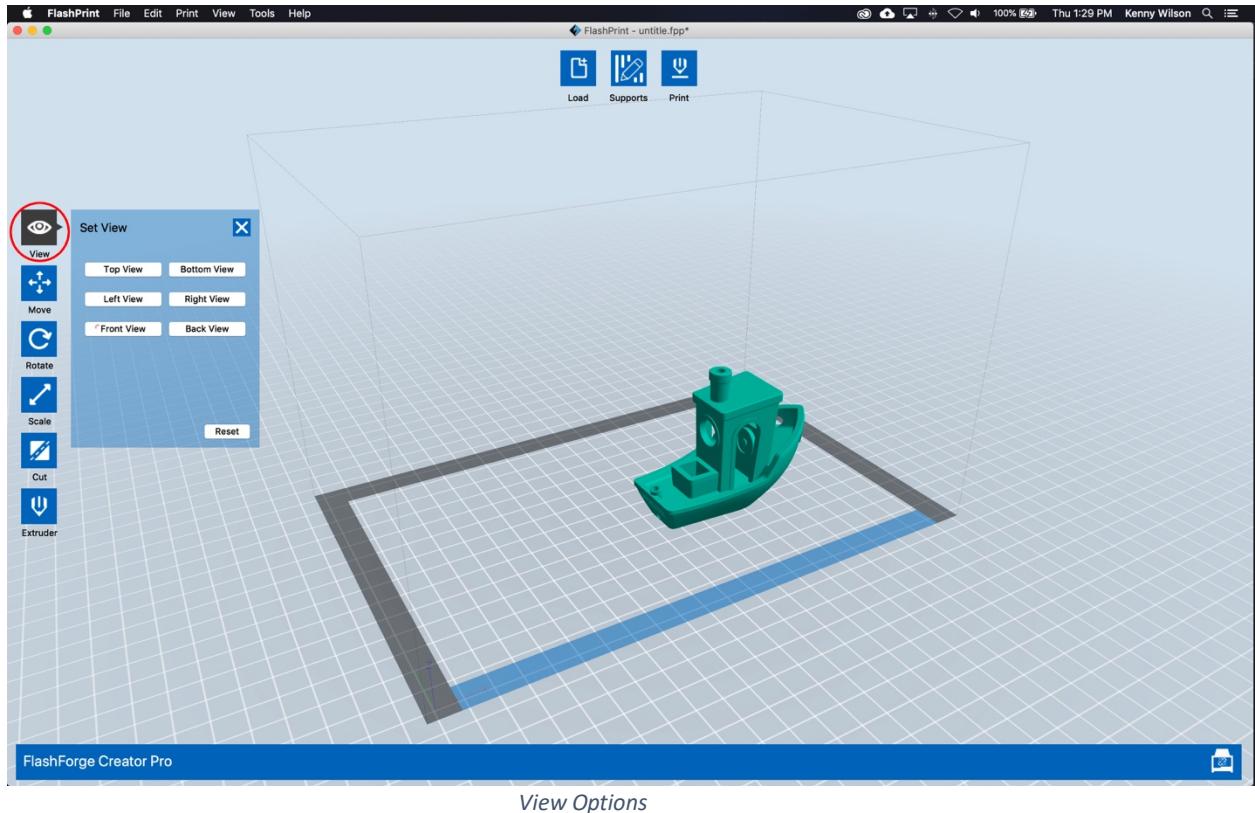
#### Step 4:

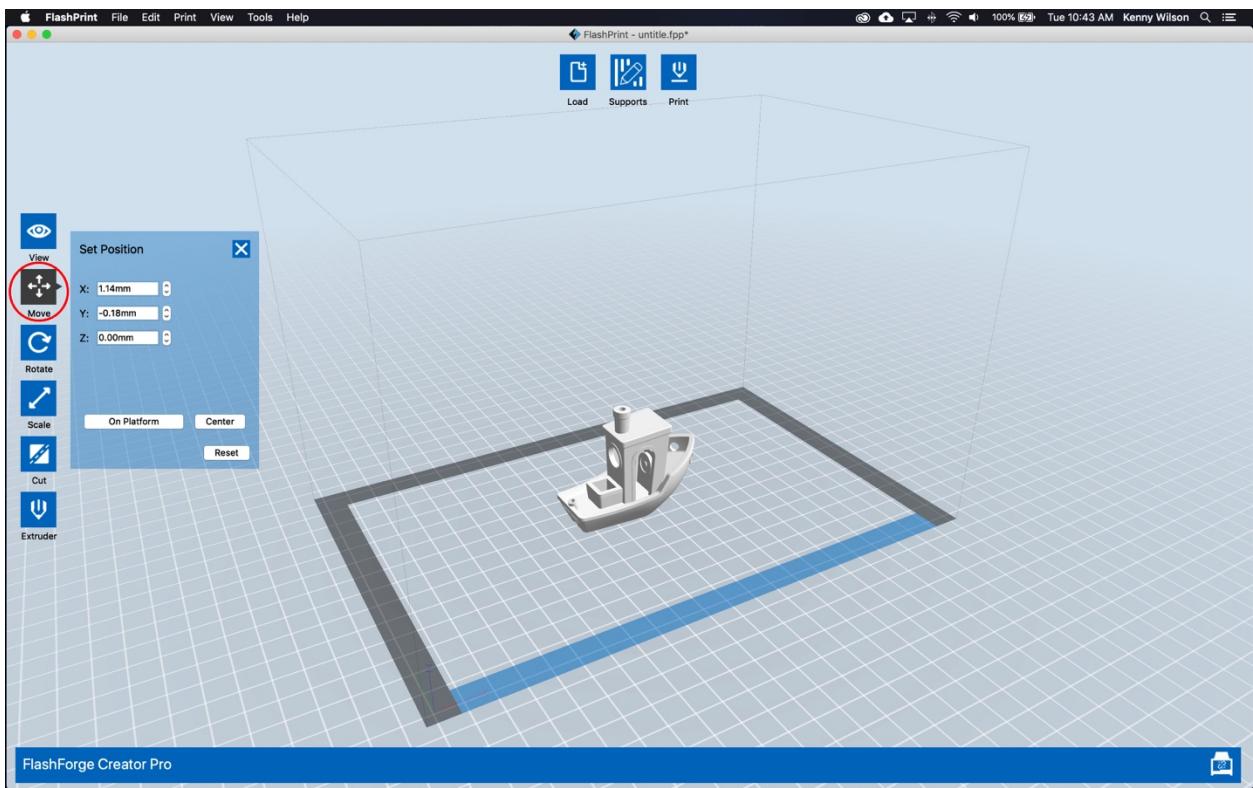
You can adjust the model if needed. Typically, the way your model is automatically placed into the print bed will be fine for printing, but some of the adjustments you can make include:

- View – View allows you to change the view, or perspective, you’re viewing the model and bounding box from. Additionally, pressing the right mouse button and dragging changes the view in free-form and pressing the left mouse button and dragging moves the ground plane up or down.
- Move – Select to move the model to different locations on the print bed. Click on the model and move in the X and Y plane. To move along the z-axis, click the arrow next to the Z input box or enter a value
- Rotate – This allows you to rotate your model amongst the X, Y, and Z planes. Once clicked you should see red, blue, and green circles that indicate the three planes. If you do not see this, make sure the check box next to ‘Surface to Platform’ is not checked. Click on one of the three circles and move your mouse to rotate the model. Additionally, you can enter values into the input boxes.
- Scale – This allows you to adjust the scale, or size, of your model. Typically, you’ll want to keep the ‘Uniform Scaling’ box selected. Click on the model and move your mouse to scale the model.
- Cut – Cut allows you to cut your model into two or more pieces to fit within the bounding box. We advise not having a model size that require this. This is an advanced

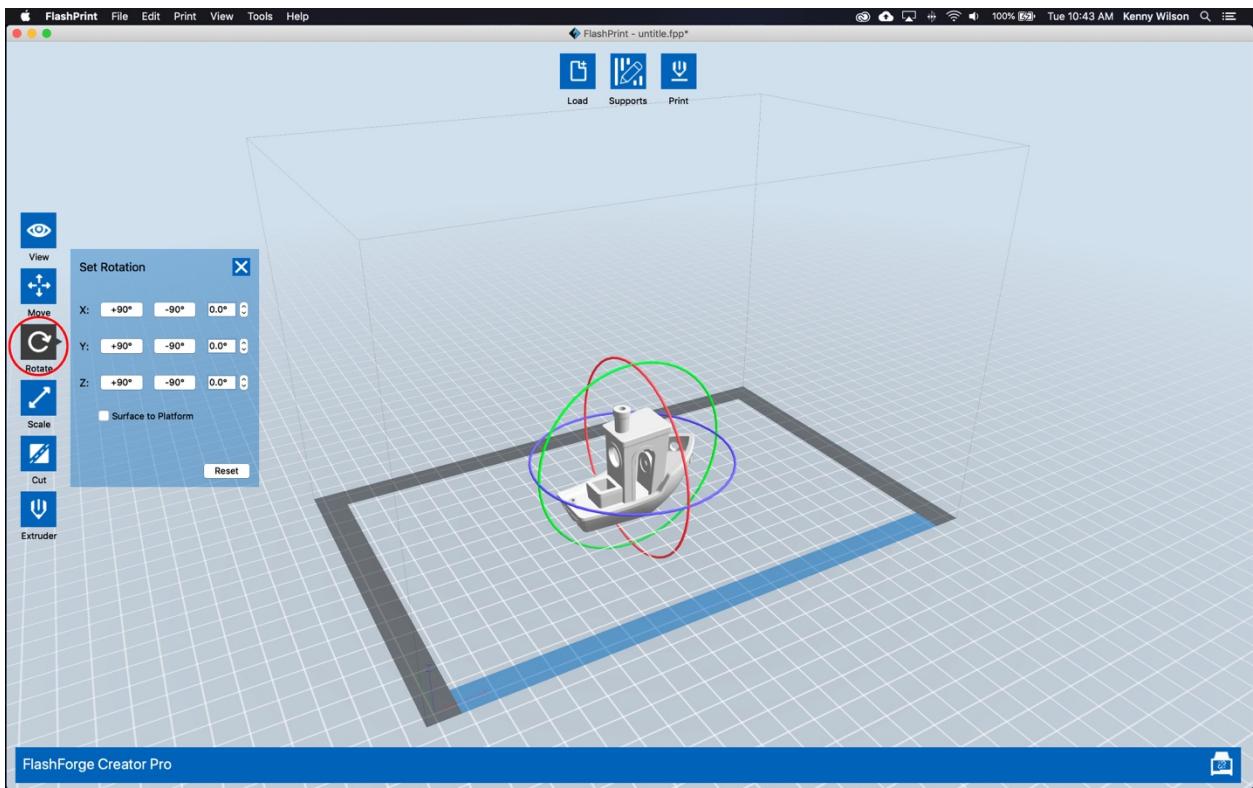
feature. For more information about this, refer to the FlashPrint manual linked in the previous page.

- Extruder – This allows you to choose which extruder, and filament, you want to use. It's defaulted on the right extruder. You can look at the color of filament loaded into the printer and decide which one you'd like by choosing left or right extruder.

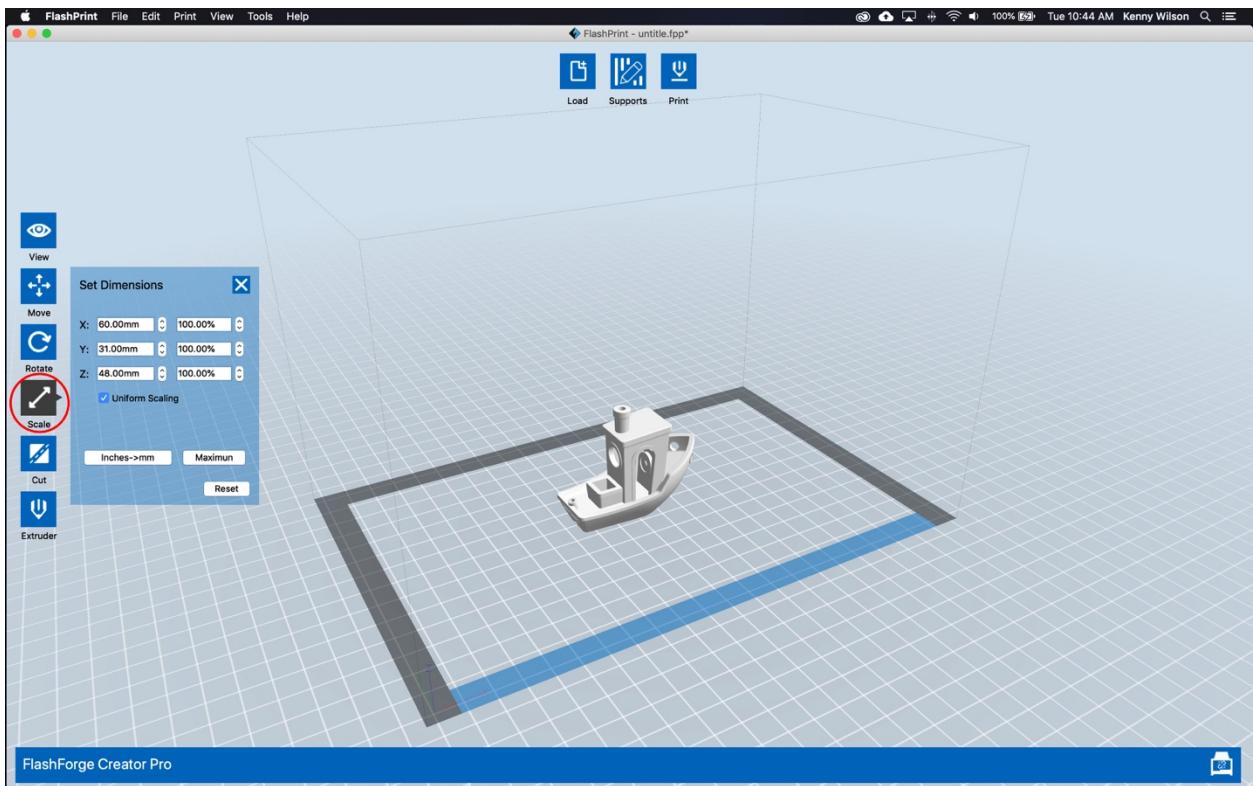




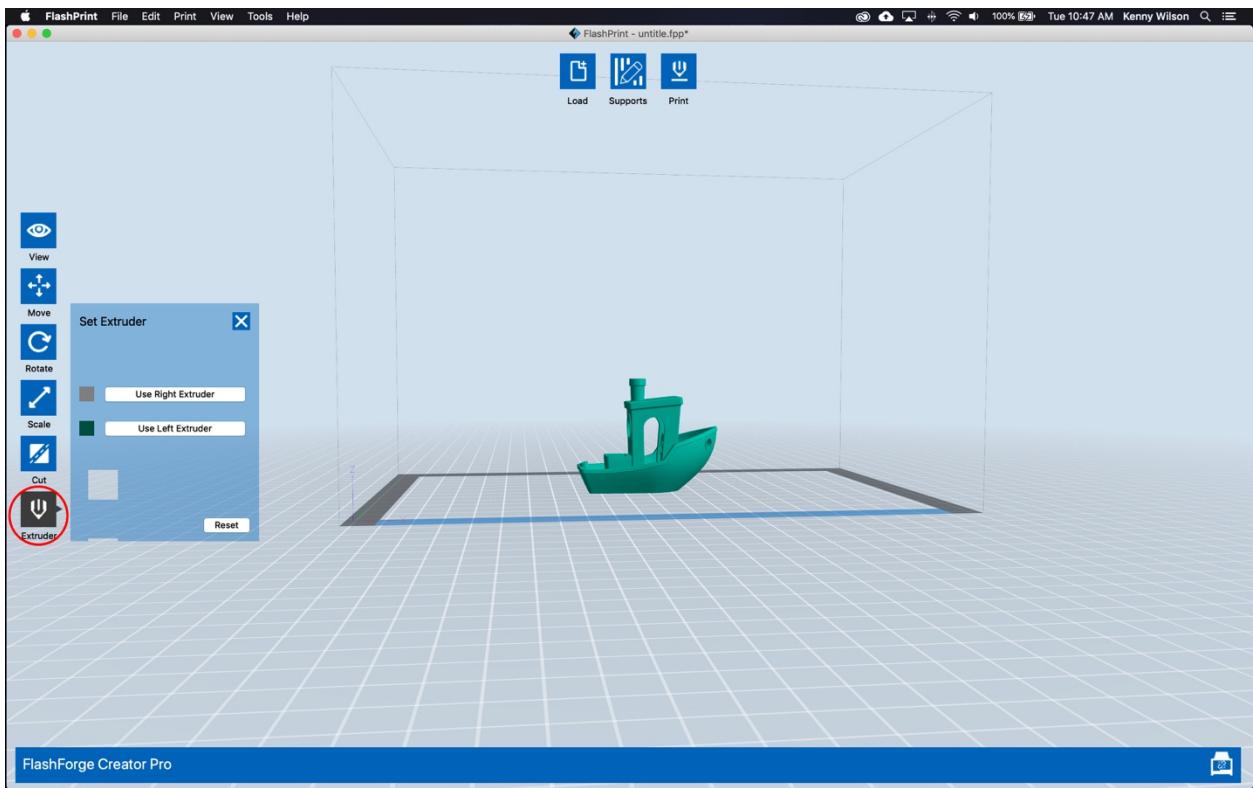
Move Options



Rotate Options



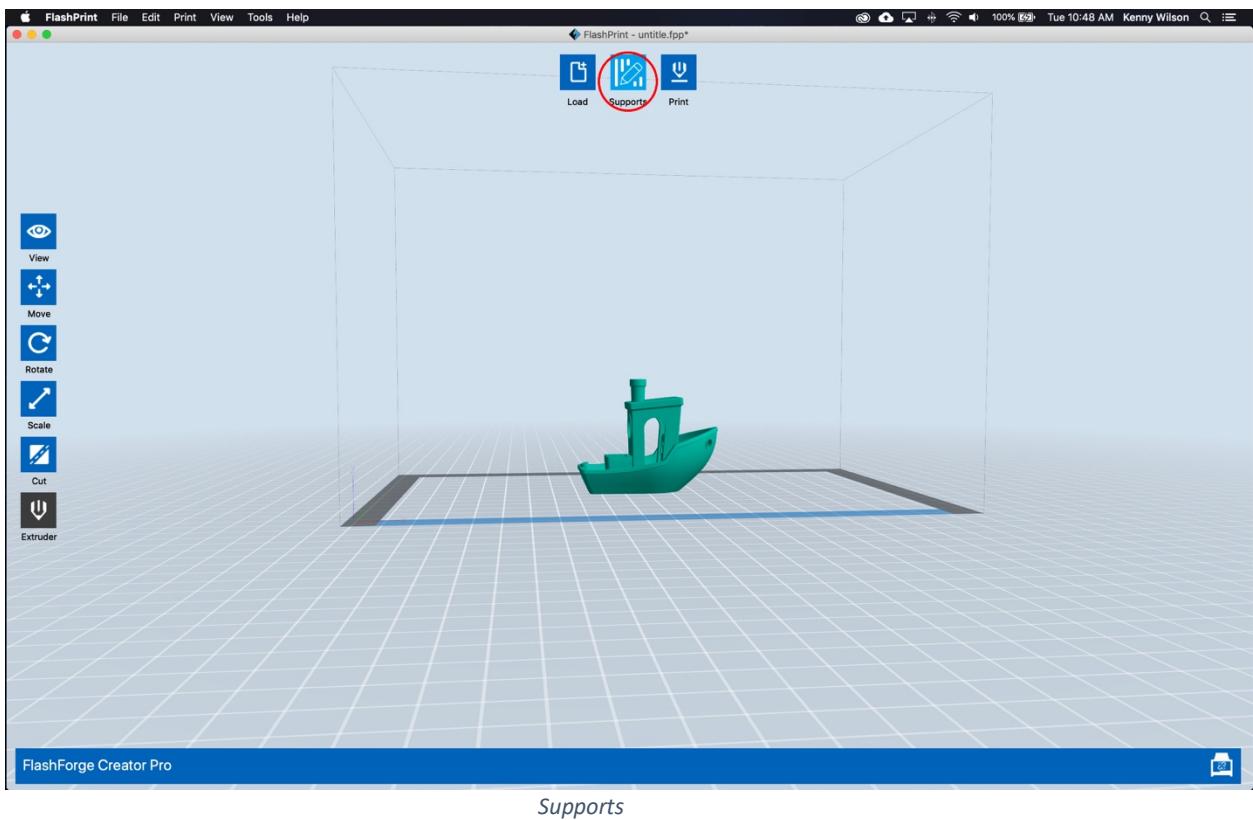
Scale Options



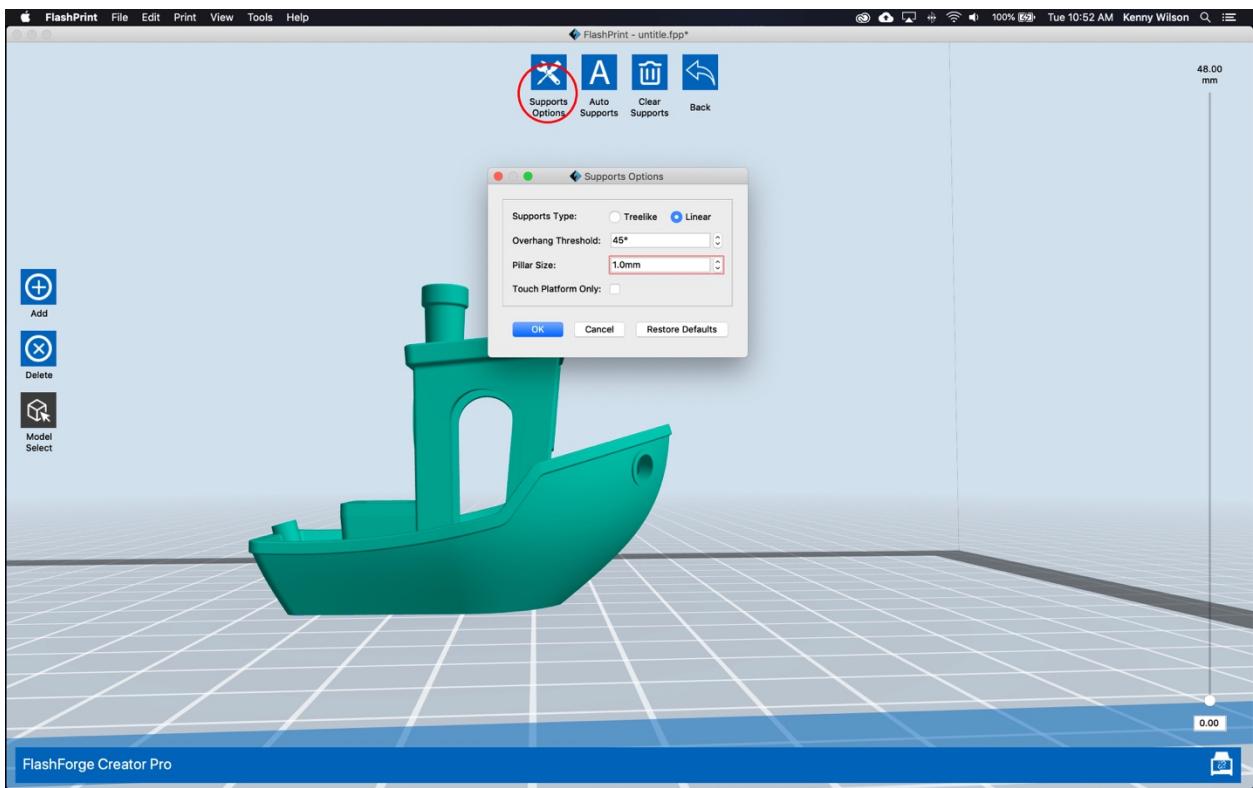
Choose Extruder

## **Step 5:**

Next, you want to select the ‘Supports’ button in the upper center of the window.

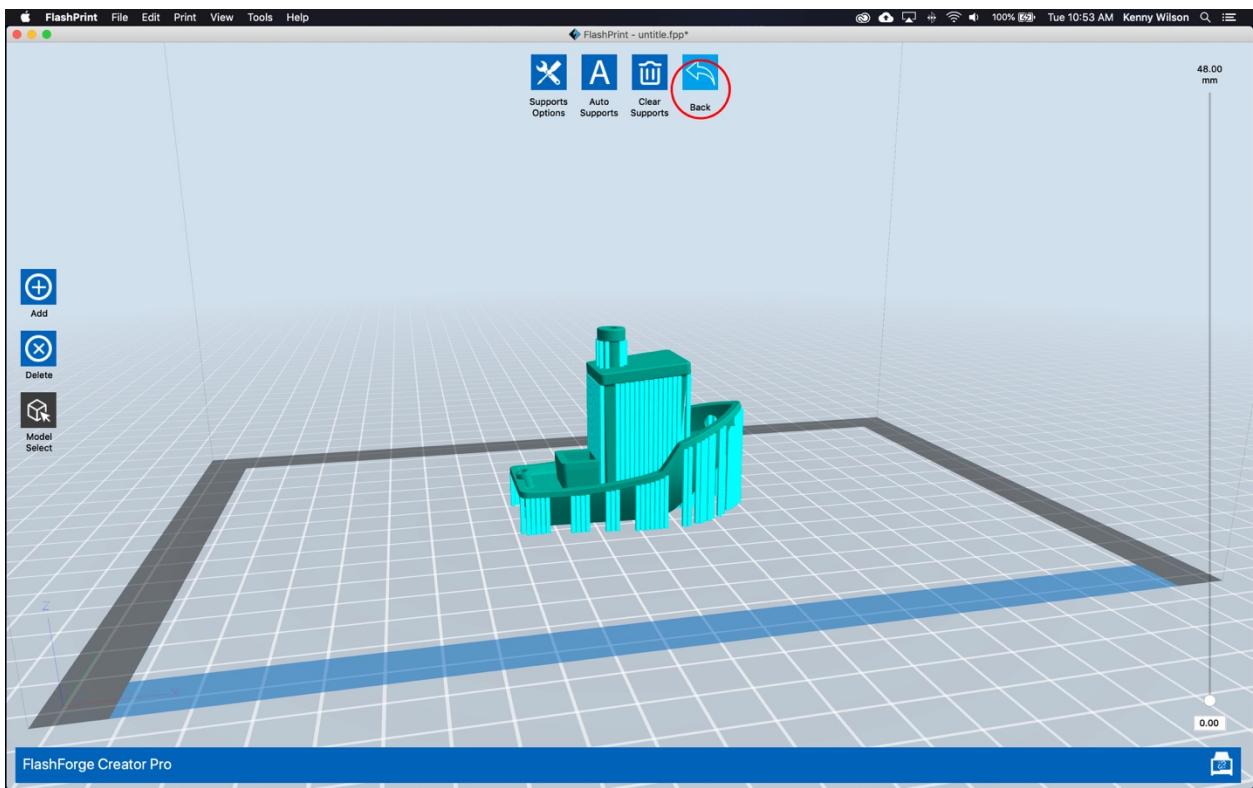


Then click ‘Support Options’. You’ll be presented with a dialog box that has a number of options to choose from. We recommend selecting the ‘Linear’ support type. ‘Overhang Threshold’ refers to the angle at which supports will be added to the model. A typical range for this would be 20 to 45 degrees. More often than not, you can go with the defaults and then experiment with other options as you gain more experience. Click ‘OK’ when done.



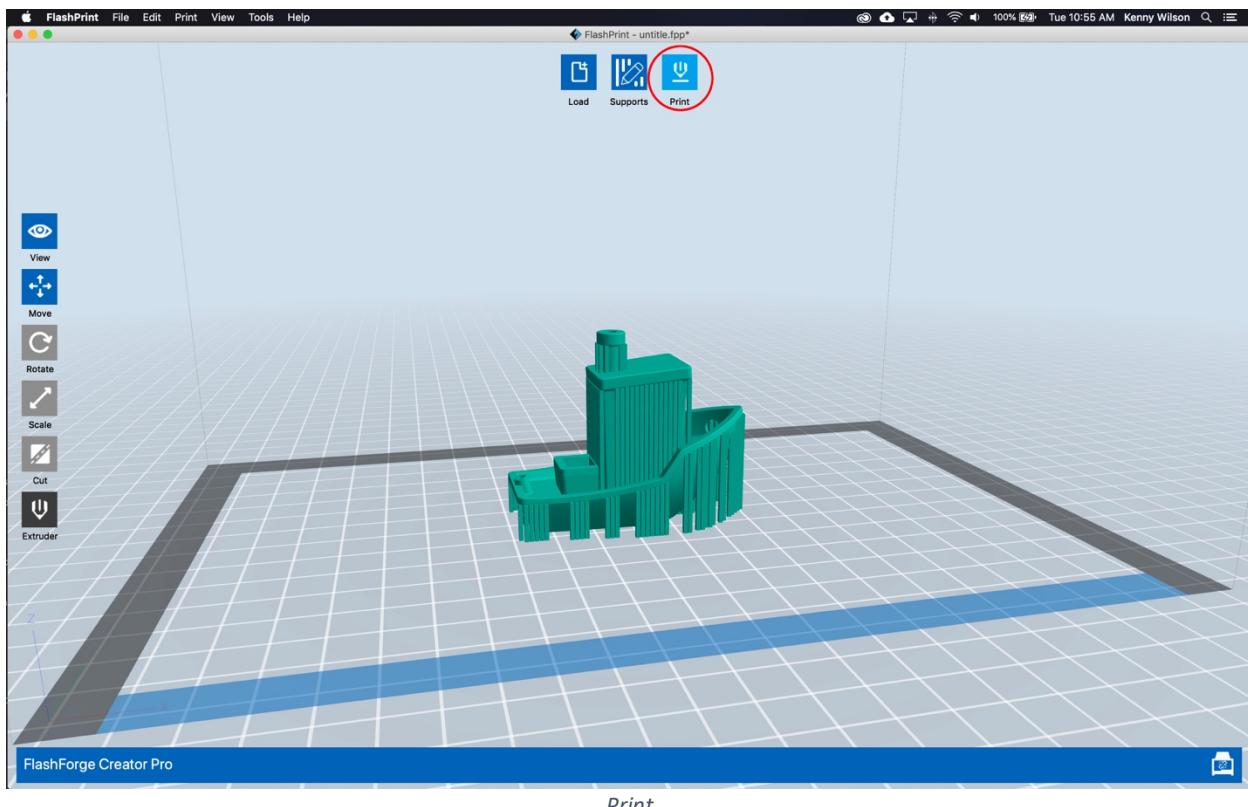
*Support Options*

Next, click 'Auto Supports'. The software will automatically generate supports according to the options you selected. Finally, when you are happy with the support structure created, click on the 'Back' button to return to the previous FlashPrint options. You will be asked if you'd like to save the supports to the model at this point. You can click 'No'.



## Step 6:

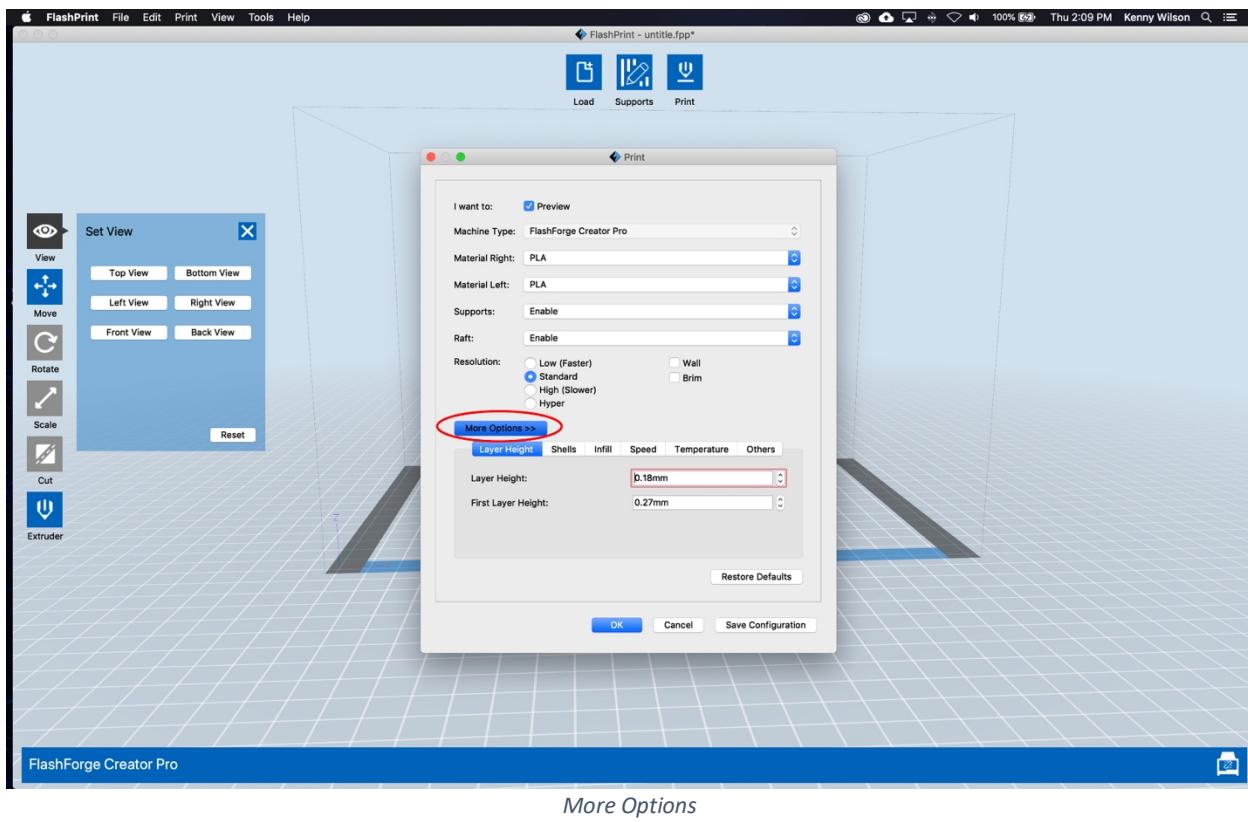
The next step is putting the final touches on the print file. Select 'Print' from the buttons in the upper center of the screen.



There are again a number of very important options within this step so **please pay close attention to this step**. More than any other previous step, this step can be the difference between a successful print and a big disaster! Choose the following options in the top half of the dialog box:

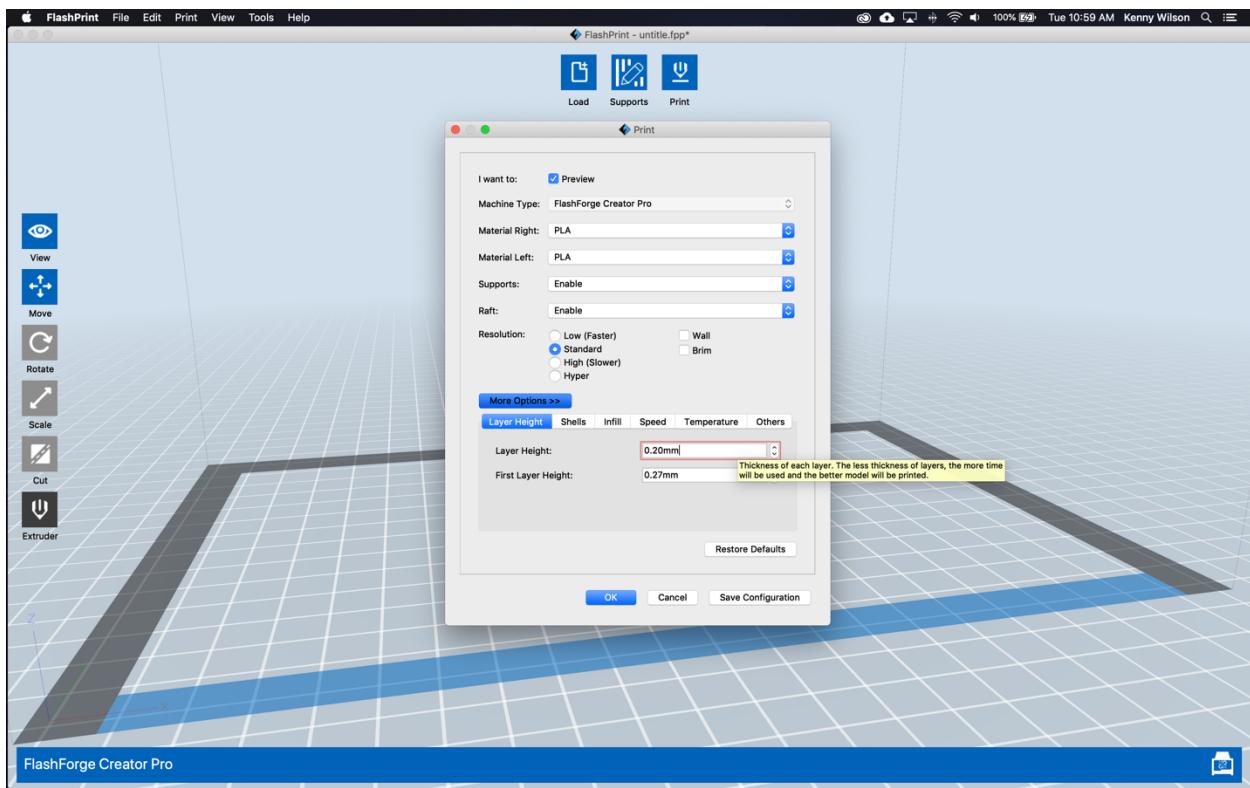
- Machine Type – FlashForge Creator Pro
- Material Right – PLA
- Material Left – PLA
- Supports – Enable
- Raft – Enable
- Resolution – Standard

Once those options have been selected, click on the ‘More Options’ button.

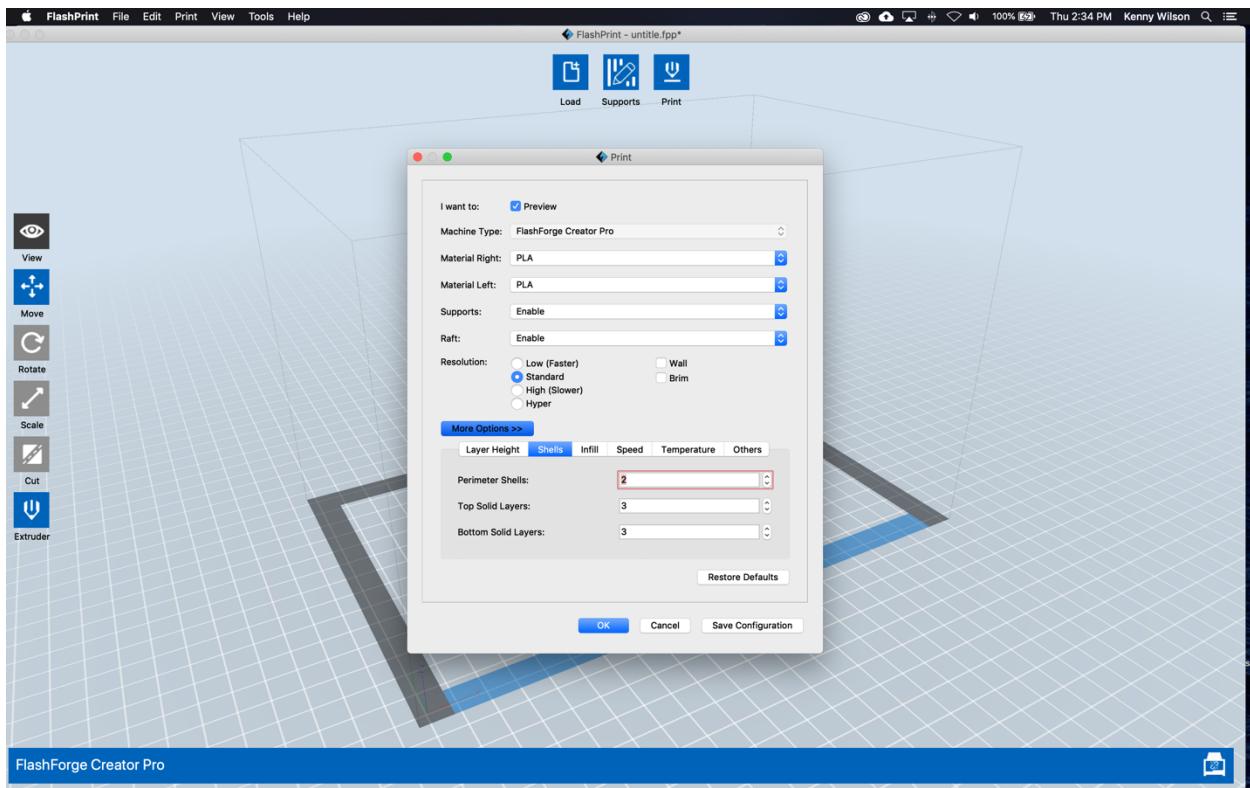


You again have a number of options to carefully select here in the different tabs. Moving from the left tab to the right tab we have:

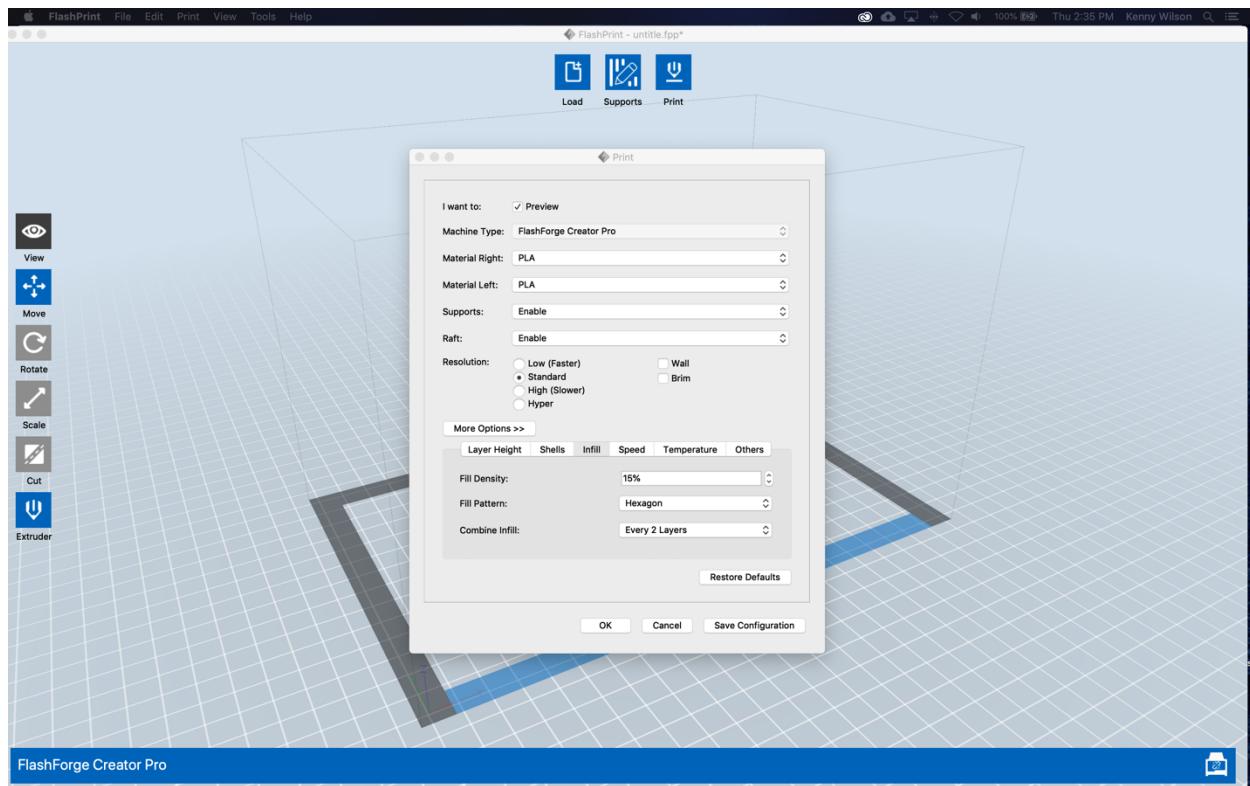
- Layer Height – Change Layer Height to 0.20 mm, leave First Layer Height as is
- Shells – You can leave this at the default value which is determined by your resolution selection
- Infill – Infill determines the amount of filament used to fill-in enclosed spaces in your model. The more the infill, the stronger the model and the more material it has. This setting can change with different models, needs, and applications; however, we recommend a fill density of 15-20%, a fill pattern of hexagon, and a combined infill every 2 layers as a standard choice
- Speed – You can leave this at the default value which is determined by your
- Temperature – Set the extruder temperature to 220 and the platform to 60
- Others – You can leave this alone



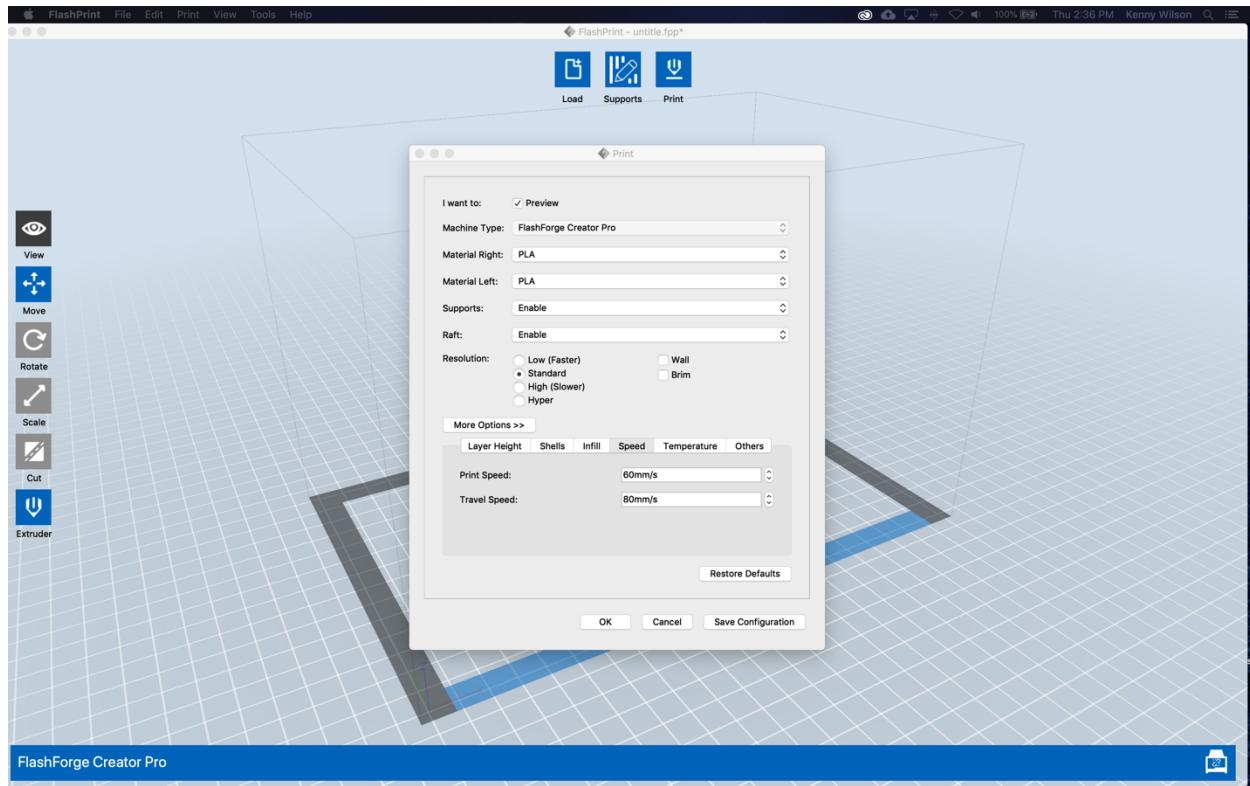
Layer Height



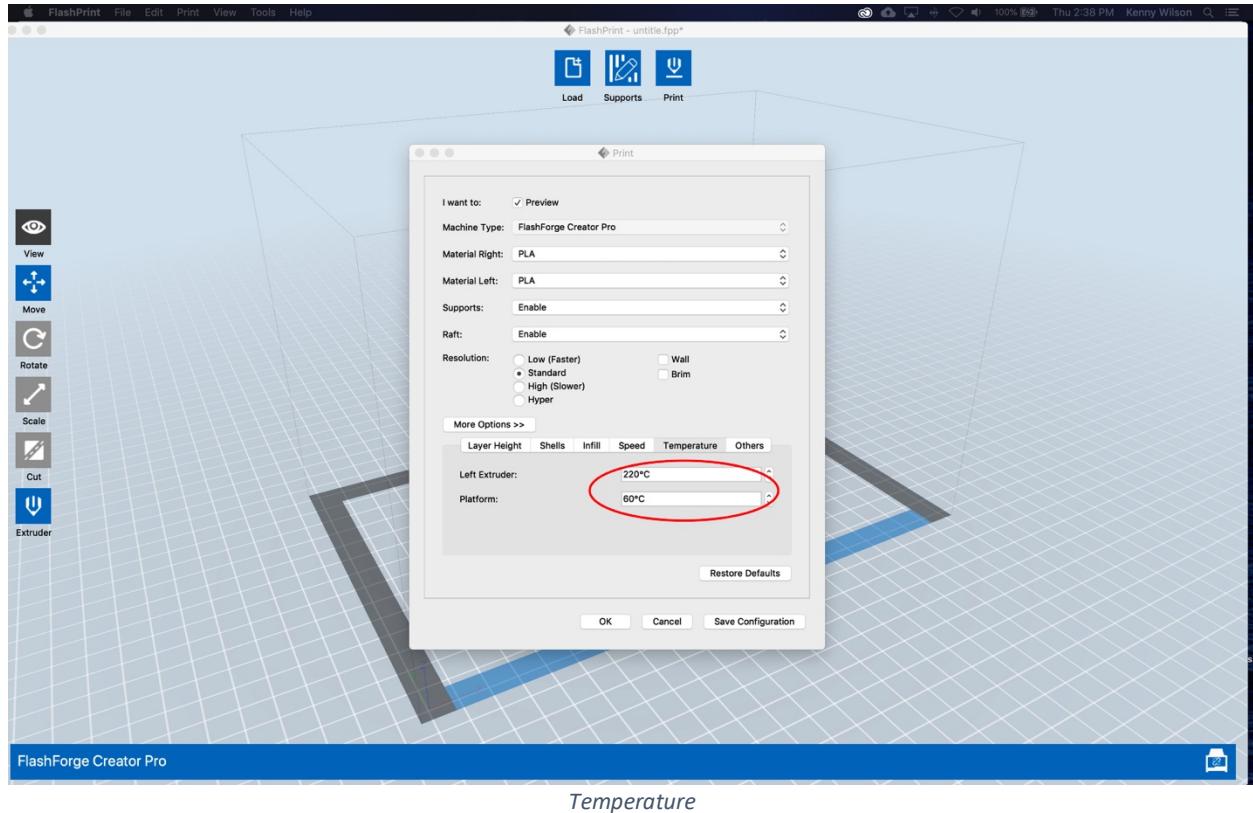
Shells



Infill

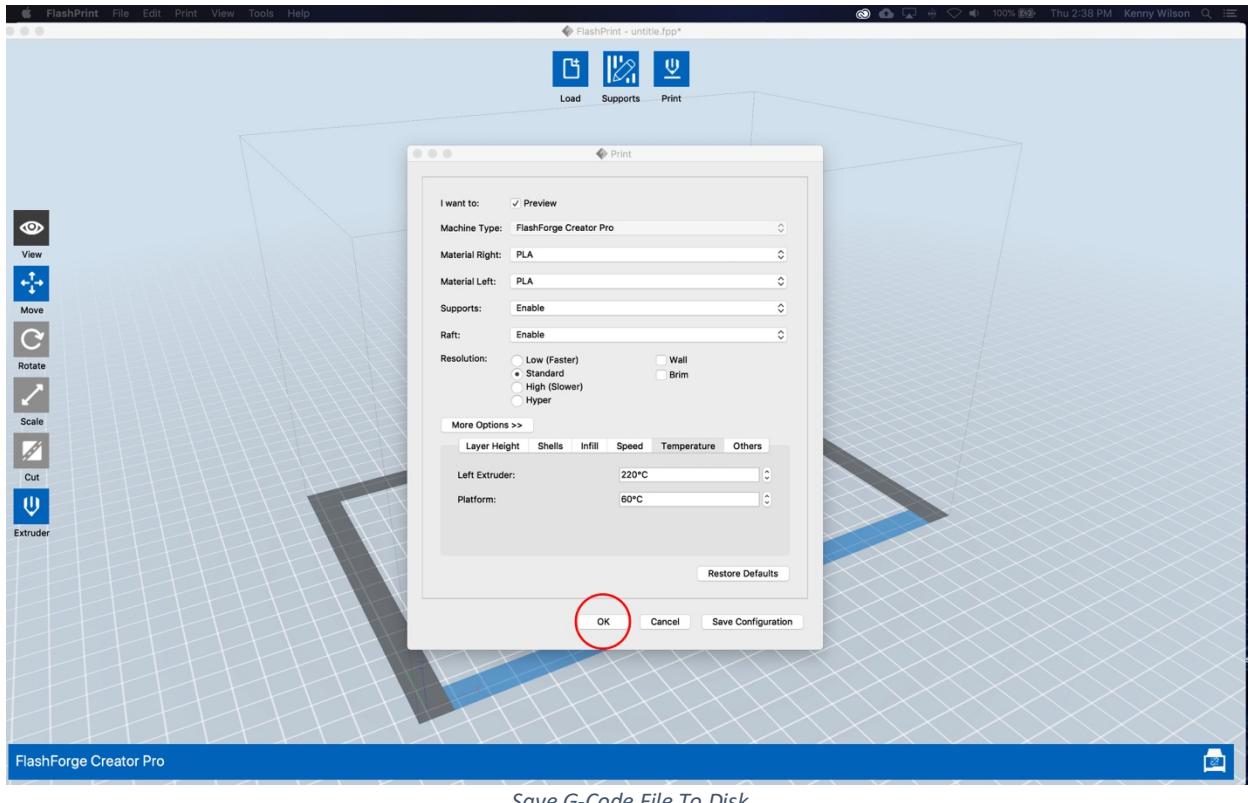


Speed



### Step 7:

The final step is to save the g-code file to the SD card. After making all your changes to the options, select the 'OK' button in the bottom of the dialog box.



Another box will pop up and this is where you save your file. Make sure the SD card from the 3D printer is inserted into the iMac machine and that you save your file to this SD card. You can rename if you want but it is not necessary. Finally, click 'Save'. FlashPrint will take a few moments to generate your file and when finished, you'll have a file named 'YourFileName'.x3g on the SD card. Eject the SD card from the iMac and insert it back into the 3D printer.

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