Print: Lego: Print It! <mark>FLASHFORGE</mark>
Module F3D.6

Follow the instructions in this document to learn how to export a Tinkercad model as an .STL file, Import it into a slicer program, prepare the model for 3D printing, and finally send the file to be printed using a USB drive.
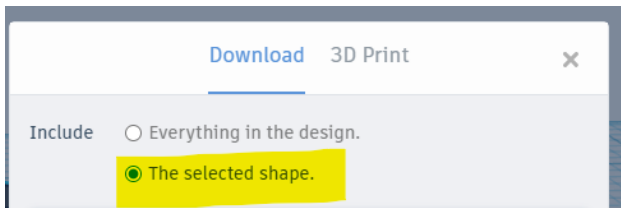
<mark>**The first time is the hardest... but once you know the process it is fast and easy!**</mark>
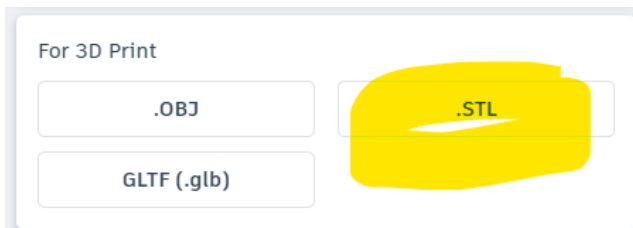
Step 1: Complete your design in Tinkercad.

Step 2: Select the thing you want to print. You should go through Steps 2 - 5 for each piece of your design individually so they are each separate files.
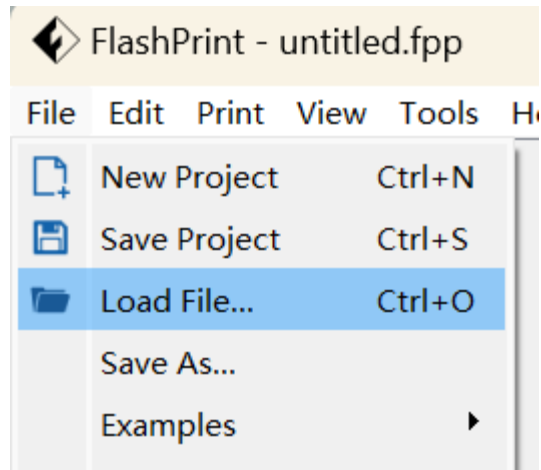
Step 3: Click Export in the upper right corner.

Step 4: In the Download window that pops up, make sure that "The selected Shape" is selected.

Step 5: Select .STL to download the 3D model as an STL file. <mark>Make sure you know where it is being saved!</mark>
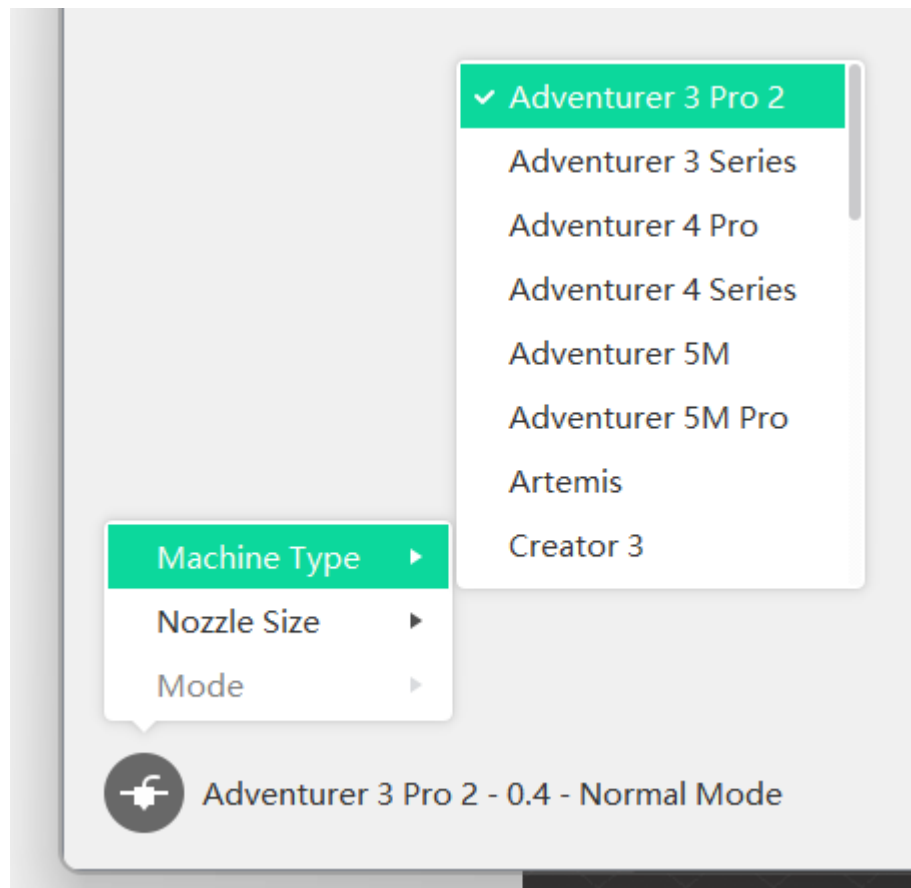

Step 6: On your computer, open the program called Flashprint

Step 7: In Flashprint, go to "File" and select "Load File".

Step 8: Choose your downloaded STL file that you just created in Tinkercad. Your 3D model should now appear on the print bed.
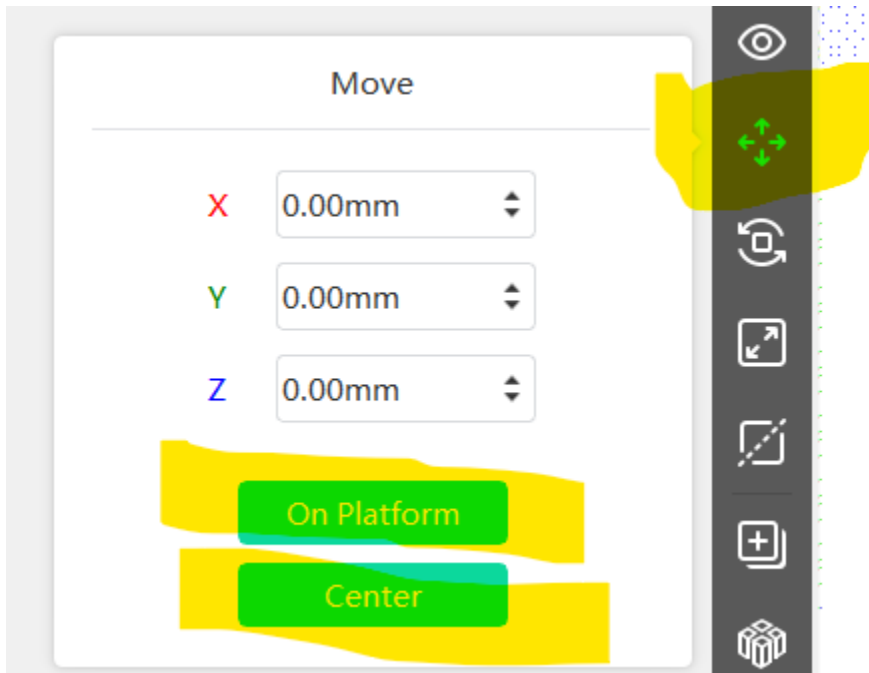
Step 9: In the lower left corner of the screen, make sure that the correct printer is selected: most likely you are
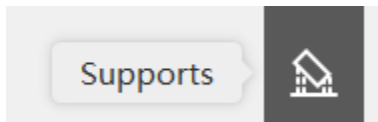

using the Adventurer 3 Pro 2

Step 10: Using the menu bar on the right edge, move, rotate, scale (expand/shrink), etc to change your print as you see best fit.

Step 11: Use the move menu option to confirm that your model is placed on the print bed by clicking the "On Platform" button. Use the "Center" button to make sure your print is in the exact middle of the print bed.

Step 13: If your print is complex with lots of overhangs, potentially floating parts, or roofs, select the Supports button in the right-side menu.

Step 14: In the Supports Options window, select the "Support Type" (hover over the 2 options and it will pop up a description to read) and then choose "Auto Supports"

Supports Options

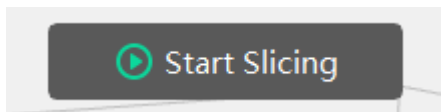|	
Supports Type	[tree icon] [lines icon]
Overhang Threshold	55°
Post Diameter	3.0mm
Base Diameter	6.0mm
Base Height	6.0mm

☐ Touch Platform Only
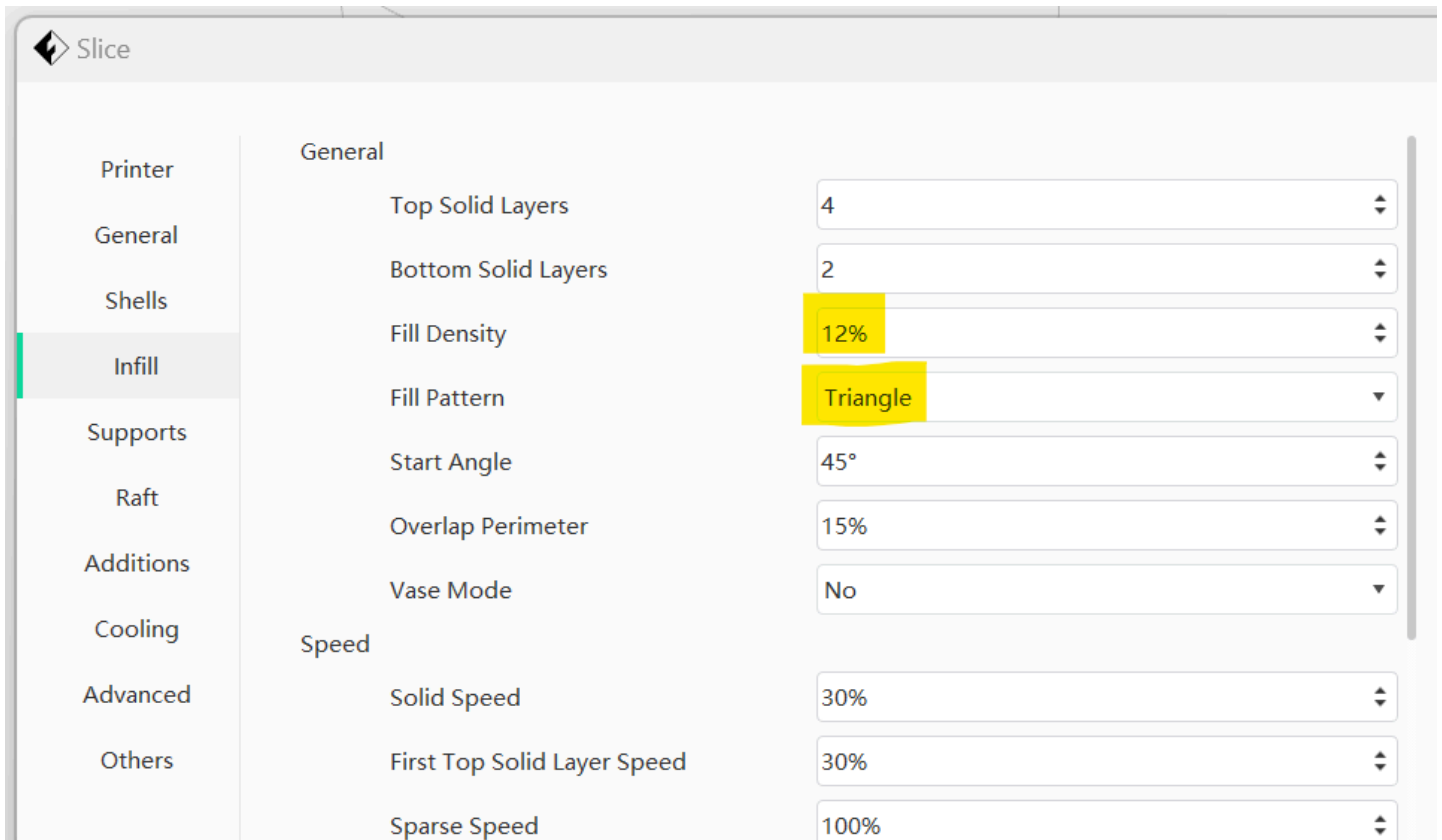
Manually ⊕ 🗑

Auto Supports

Clear Supports

Back

Step 15: Once you have prepared your print, you are ready to establish the Print Settings by clicking the Start Slicing button at the top, middle of the screen.
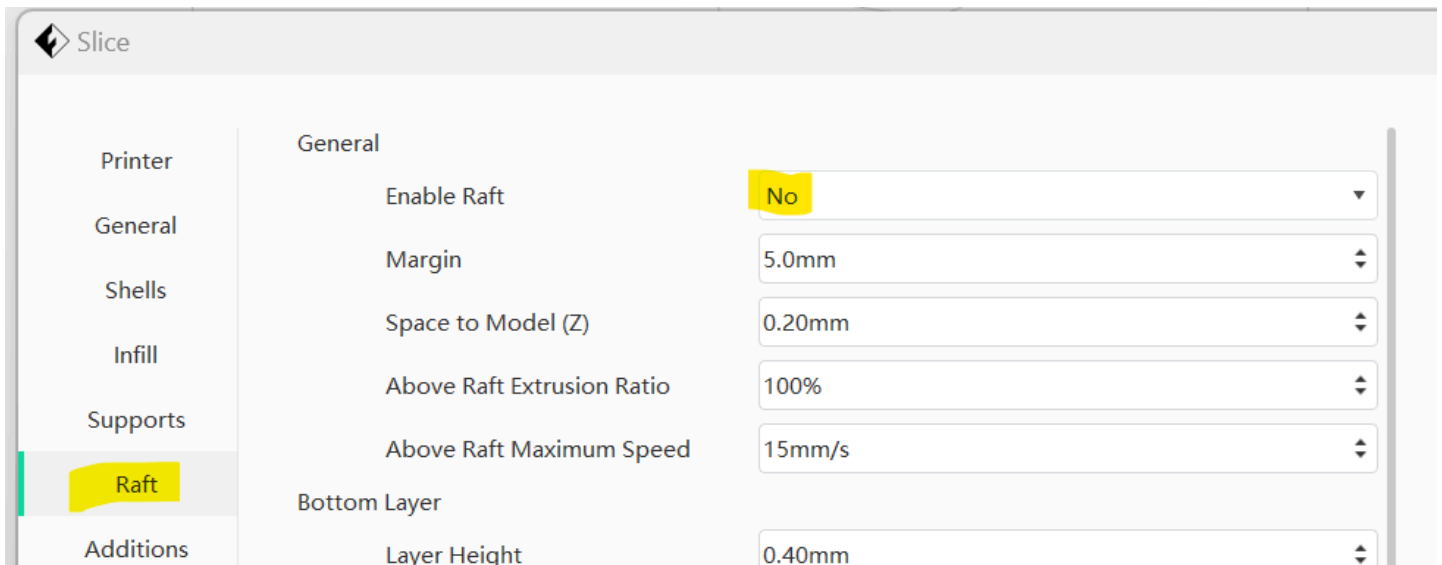
▶ Start Slicing

Step 16: In the Slice screen, choose the "Infill" menu. The 2 important values that you can alter are:
- **Fill Density:** The higher the %, the stronger the print will be- but also the more material it will require and longer it will take to print.
- **Fill Pattern:** There are many different patterns to choose from. Each has pluses and negatives- the only way to see what a pattern looks like is to click Slice and watch the preview that is shown.

Step 17: Next, choose the "Raft" menu and confirm whether or not you want to have a raft printed. These printers have been found to print best without a raft most of the time.
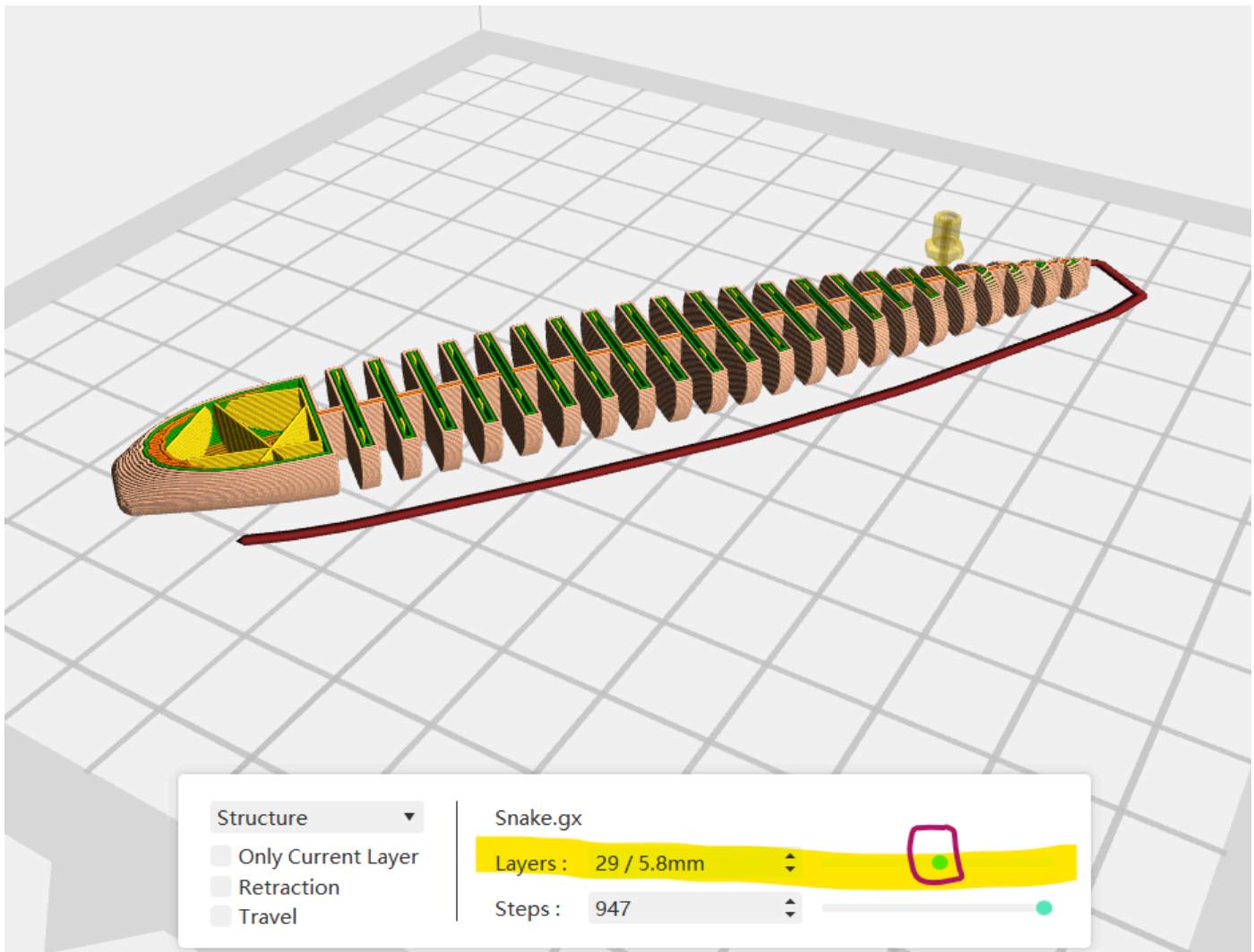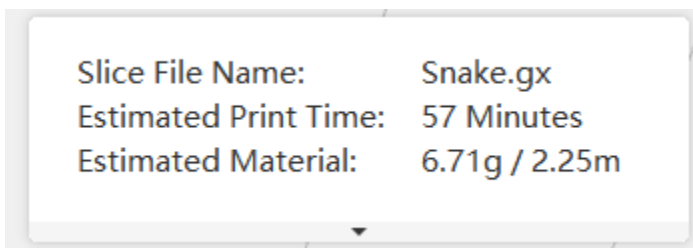

Step 18: Now you are ready to Slice.. Click it!

Step 18: Once the program has finished slicing the 3D model, click the "Slice Preview" button along the top menu bar.

Step 19: You should now be able to see a print preview view controls along the bottom of the screen. Move the layers slider back and forth to examine how the print will be fabricated.

Step 20: Confirm Time and Estimated Material in the upper right corner. 3D printing is slow AND expensive (approximately $0.10 per gram)

Slice File Name: Snake.gx
Estimated Print Time: 57 Minutes
Estimated Material: 6.71g / 2.25m

Step 21: Make sure your print is under the Module's set limit for # of grams used. If your print is using too much material, exit the Print Preview and use the Scale tool to change one of the % numbers to a smaller value, which will shrink the model. Run the Print Preview again after shrinking the model to confirm that you are under the required maximum # of grams used.
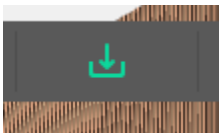

Step 22: Another way to change the amount of material used is to change the "Infill Density" in the Slicing Menu we looked at before..

Step 23: If your print is ready, it is time to get a "3D Print Que Slip" and a USB Jump Drive from the 3D printer rack in the Makerspace.

Step 24: Fill out the Print Que Slip and get it signed by your teacher. Any 3D printer that is running but doesn't have one of these slips attached will be automatically turned off.
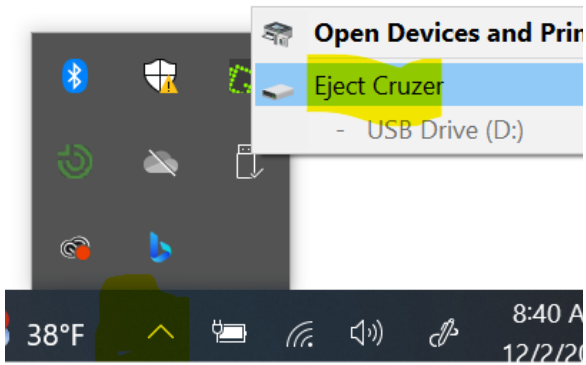
Step 25: Plug the USB Jump Drive into your computer.

Step 26: Now that you have run a Print Preview and KNOW that everything is set to print successfully, you are ready to EXPORT the file out of FlashPrint by clicking the "Save to Local" button.

Step 27: Select the jump drive and save your file directly onto the jump drive.

Step 28: Safely eject the USB Jump Drive by clicking the up carrot along the bottom menu of your computer screen and choosing Eject. ==DO NOT UNPLUG THE JUMP DRIVE UNTIL YOU ARE NOTIFIED THAT IT IS SAFE TO DO SO.==

Step 29: Bring the Print Que Slip and your jump drive over to the 3D printers and prep the printer to run the file. <mark>**If this is your first time, you need to participate in an orientation with your teacher.**</mark>