

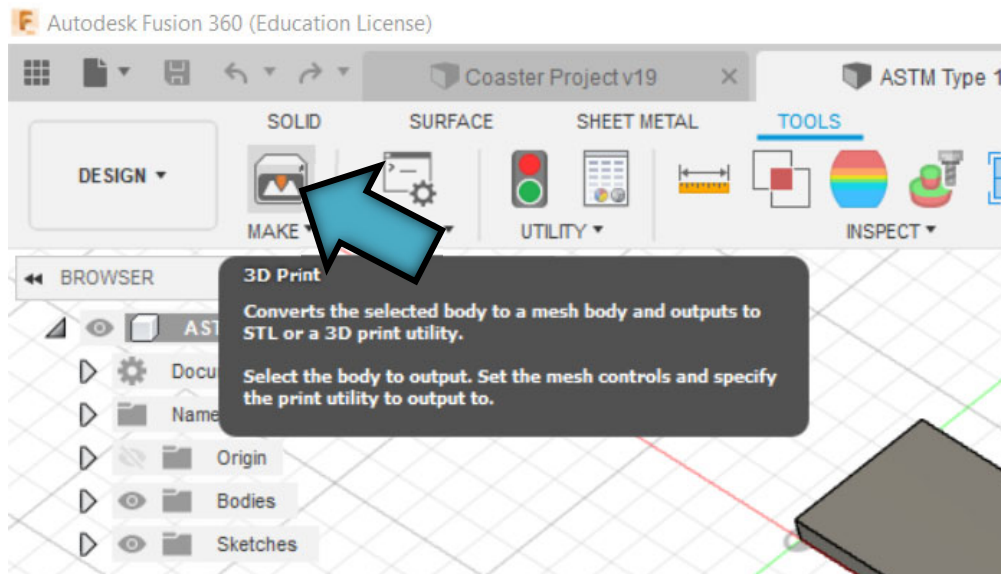
How To 3D Print using Cura (Recommended Settings)

Overall Steps

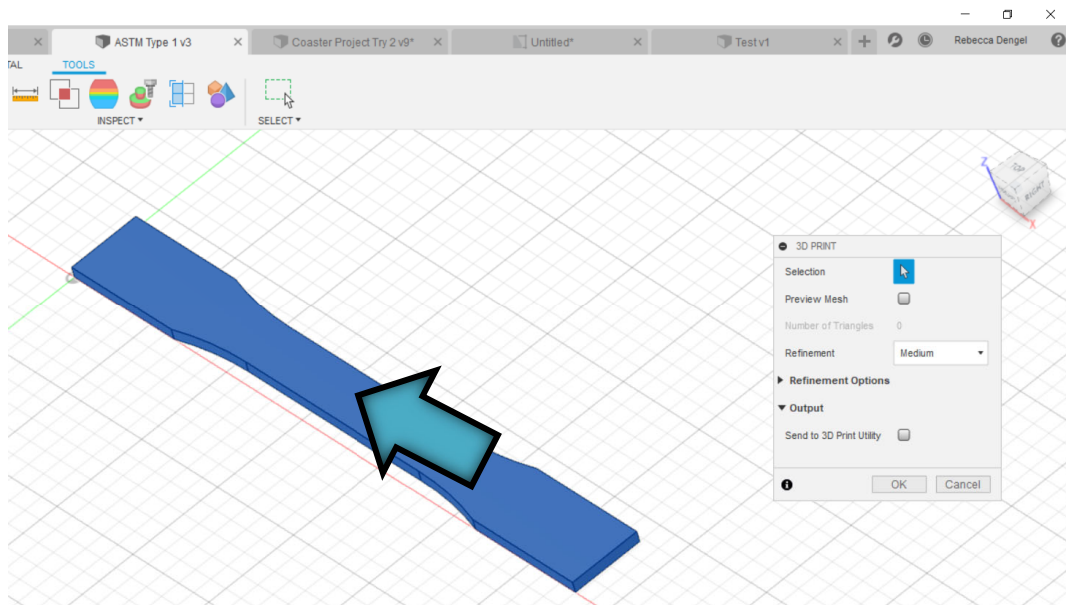
Step 1: Save file in Fusion 360 as a STL File	1
Step 2: Open Ultimaker Cura	3
Step 3: Add a printer: If this is your first time opening Cura you will need to add a printer	3
Step 4: Open file in Cura that you are 3D printing	5
Step 5: Update the Settings for the printer	5
Step 6: Click Slice	9
Step 7: Click Preview	9
Step 8: Save to removable disk	9
Step 9: Take G-Code file to printer	9

Step 1: Save file in Fusion 360 as a STL File

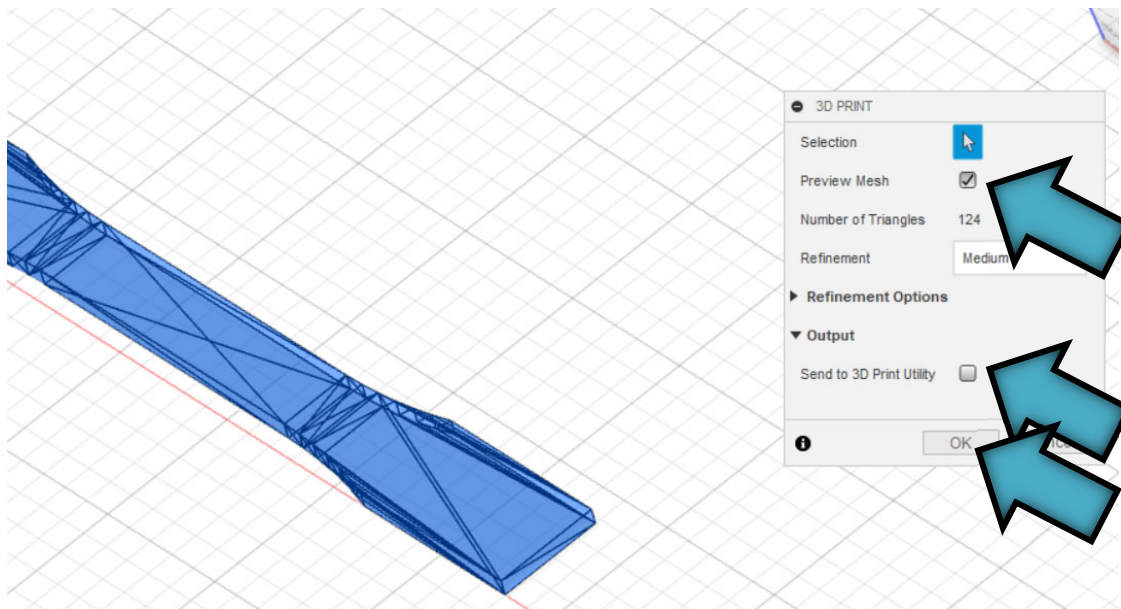
- 1) Click the image for 3D Print (top left corner)



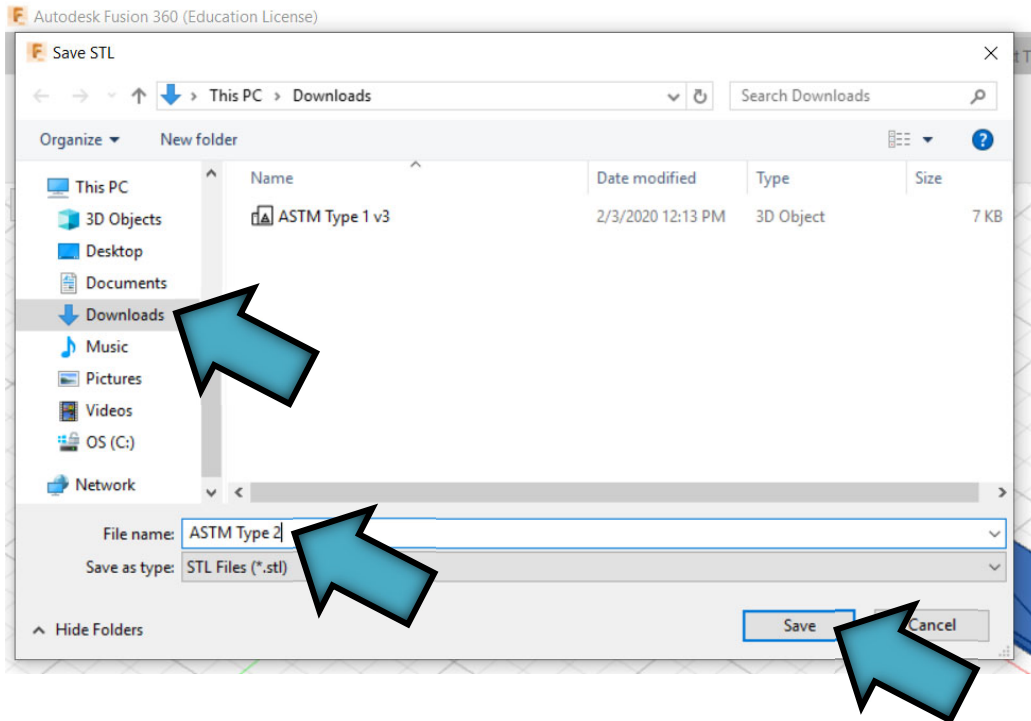
- 2) Select the body you want to export by clicking it



3) Check “Preview Mesh” and **uncheck** “Send to 3D Print Utility” then click “OK”



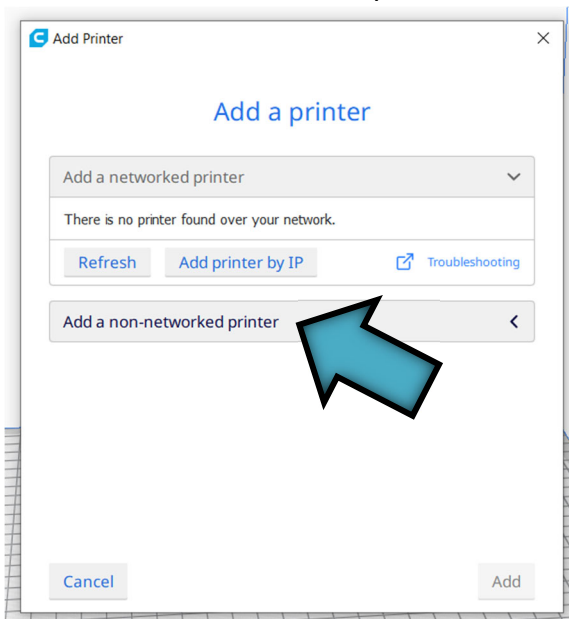
4) Name the file and choose a folder to put it in then click “Save”



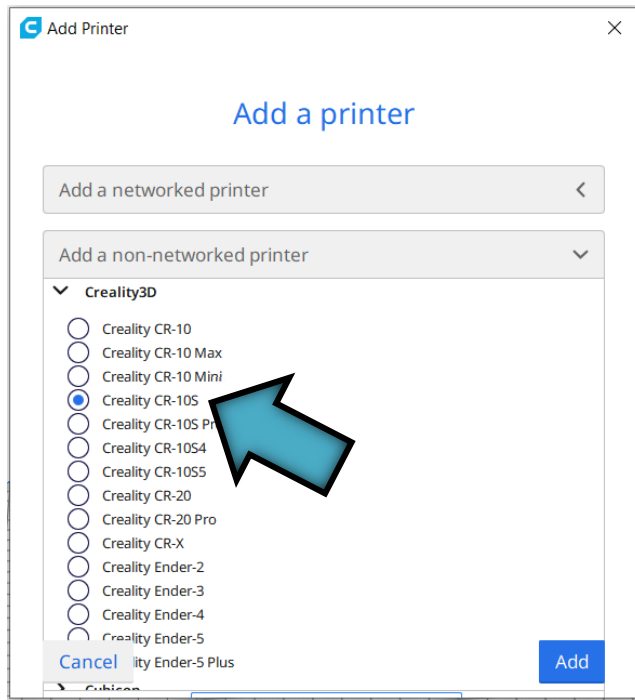
Step 2: Open Ultimaker Cura

Step 3: Add a printer: If this is your first time opening Cura you will need to add a printer

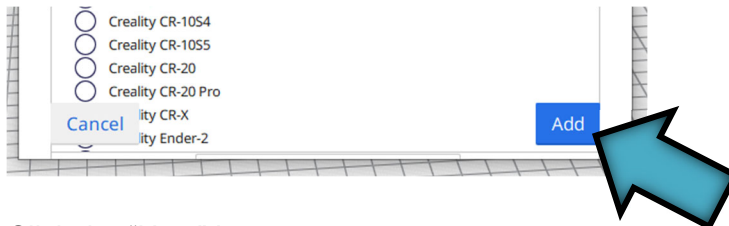
- 1) If the add a printer pop up does not automatically appear follow these steps to get to it:
 - a) Click “Settings” in the top left corner
 - b) Click “Printer” from the drop-down menu
 - c) Click “Add Printer...” from the next drop-down menu
- 2) Click “Add a non-networked printer”



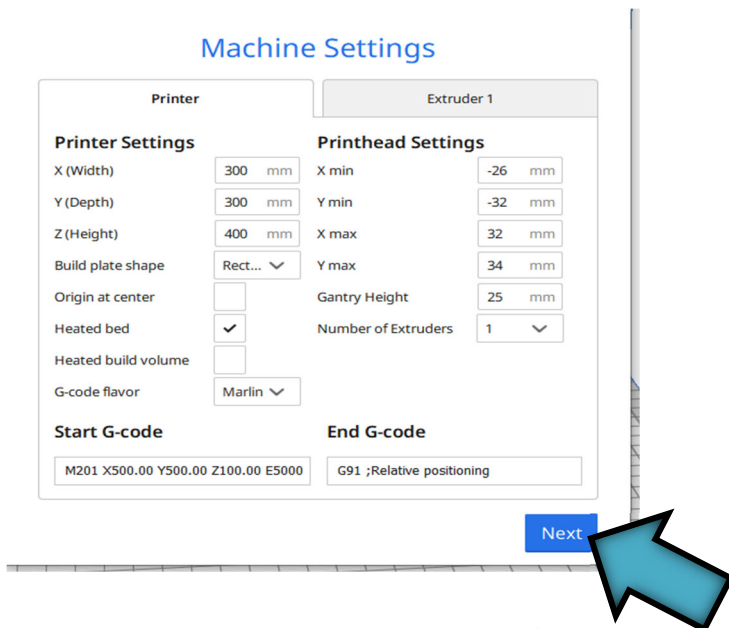
- 3) Choose 'Creality 3D' from the dropdown list then choose "Creality CR-10S" from the next dropdown menu



- 4) Click the "Add" button

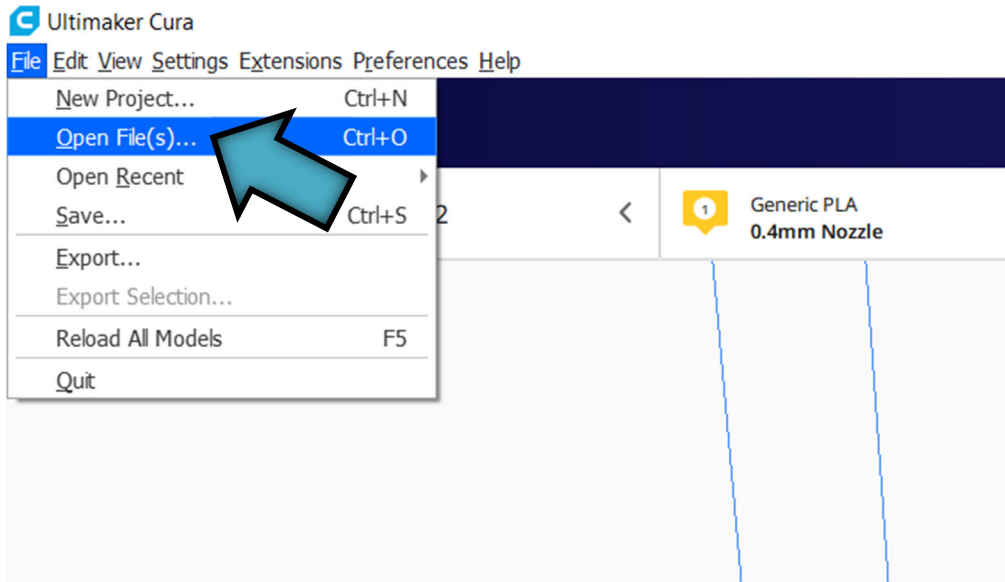


- 5) Click the "Next" button

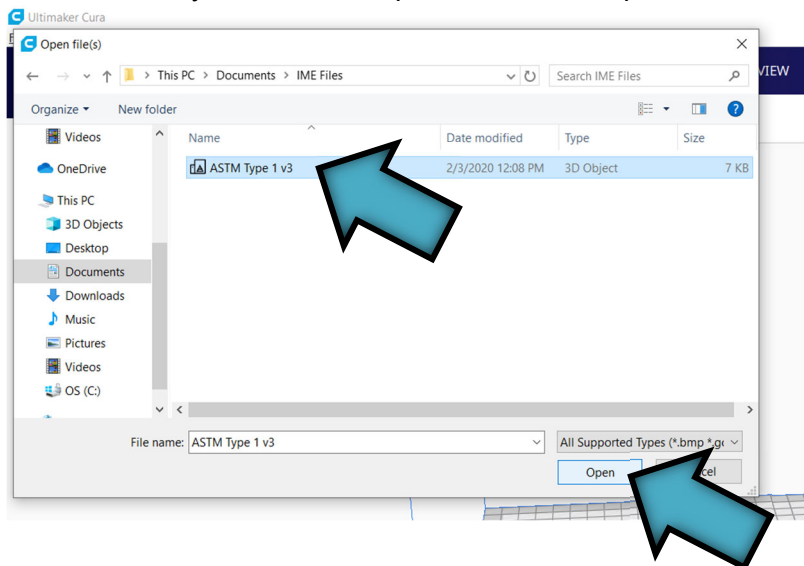


Step 4: Open file in Cura that you are 3D printing

- 1) Click “File” and then “Open File(s) in the dropdown menu under File (top left corner)

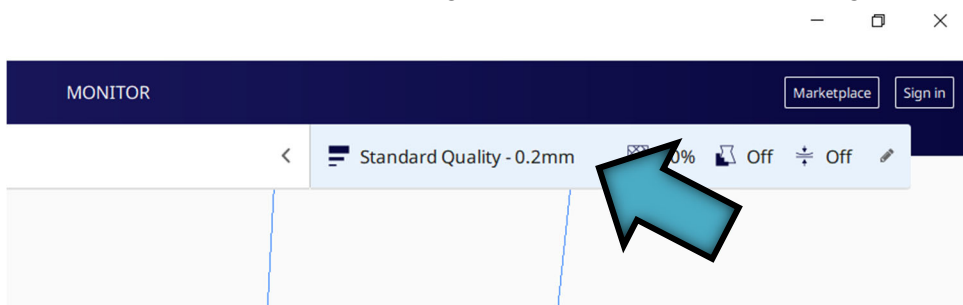


- 2) Select the file you want to 3D print then click “Open”

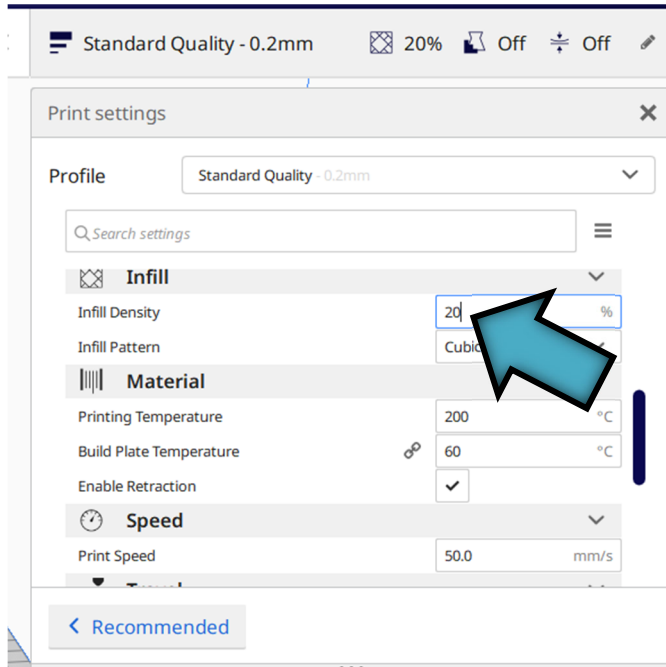


Step 5: Update the Settings for the printer

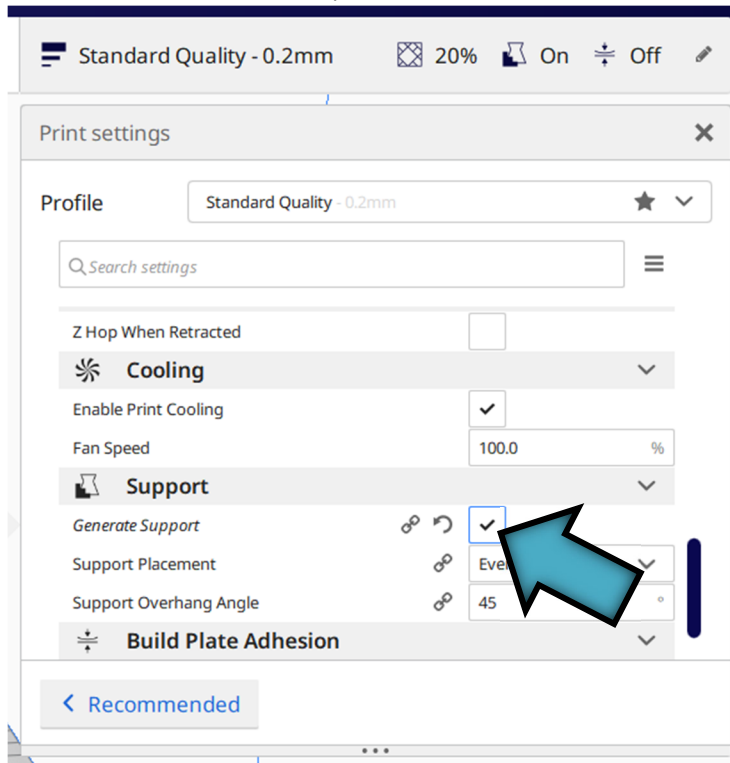
- 1) Click on the bar towards the top right of the screen to open settings



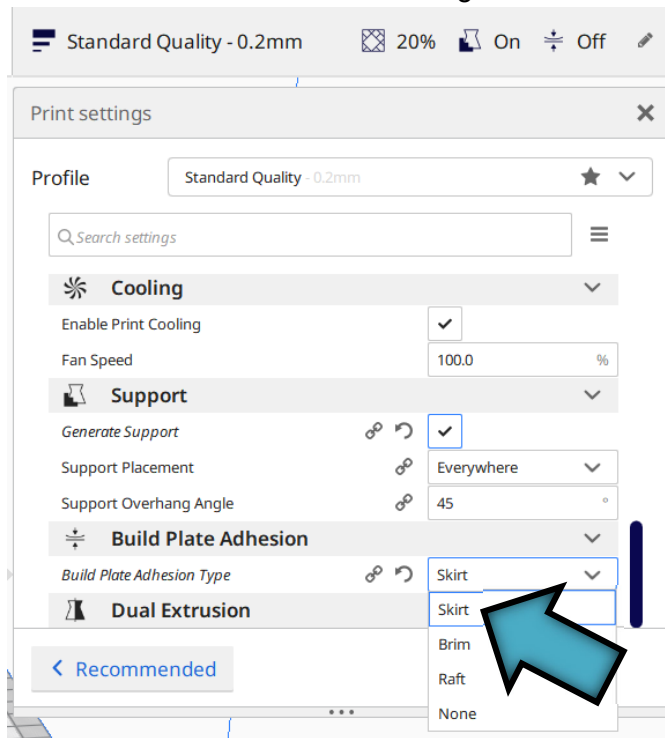
- 2) Scroll down to “Infill” and set “Infill Density” to anywhere between 20-60% (the higher the infill the stronger the part will be but 20% is generally good)



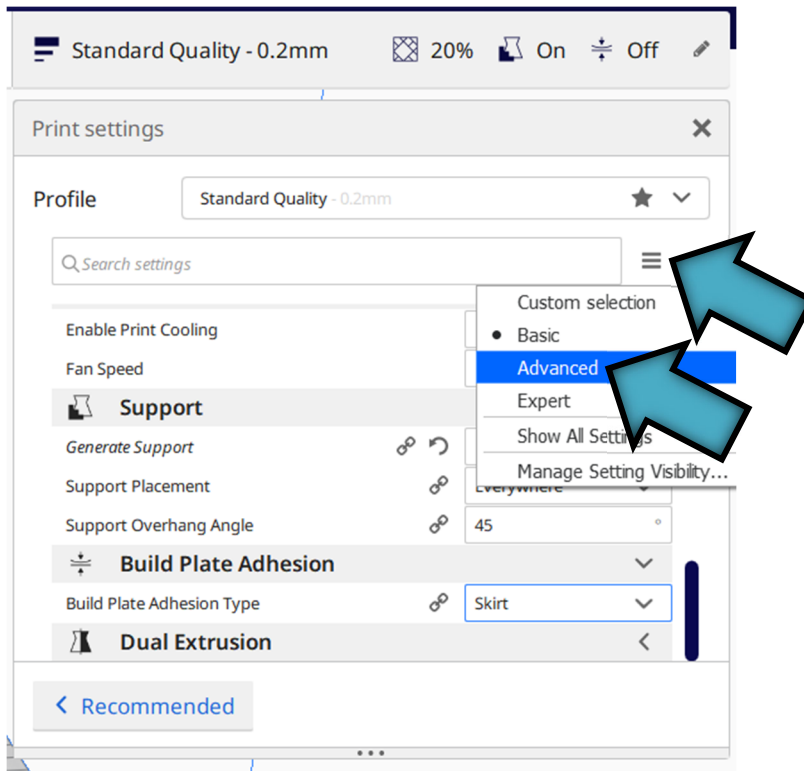
- 3) Scroll down to the bottom and under the category “Support” check “Generate Support” if your part requires support (it will need support if any part of it has material with empty space below the material)



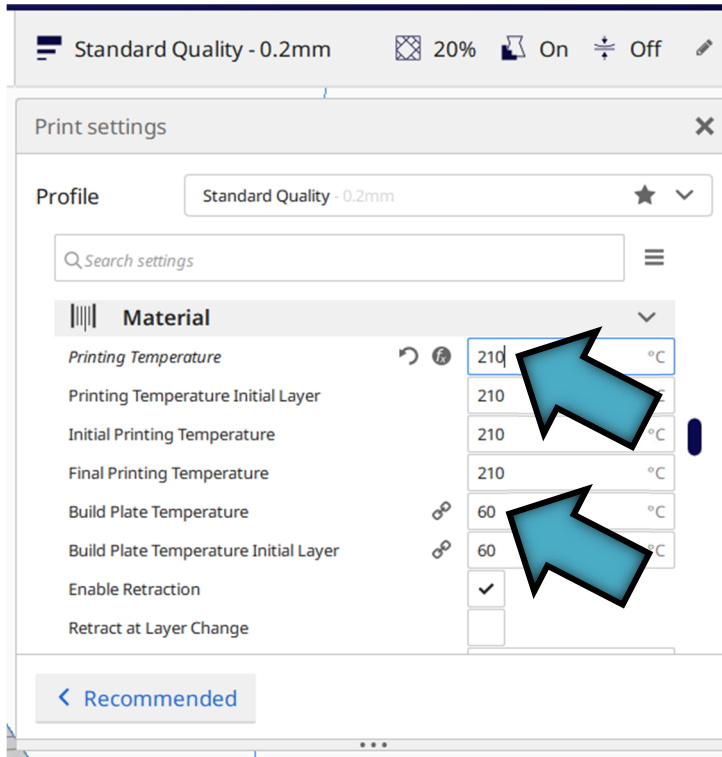
- 4) Under “Build Plate Adhesion” change “Build Plate Adhesion Type” to “Skirt”



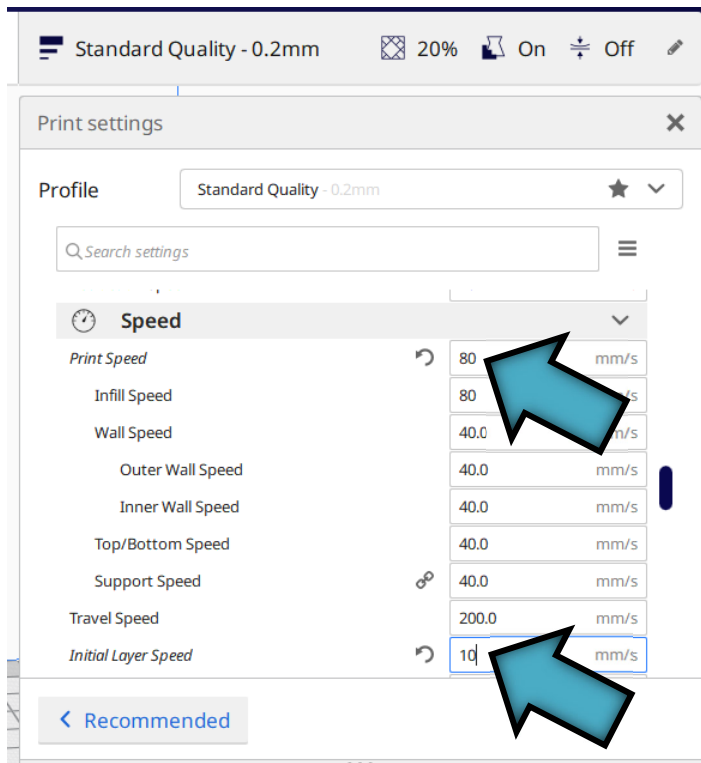
- 5) Next, click the hamburger to open more levels of settings available and choose “Advanced”



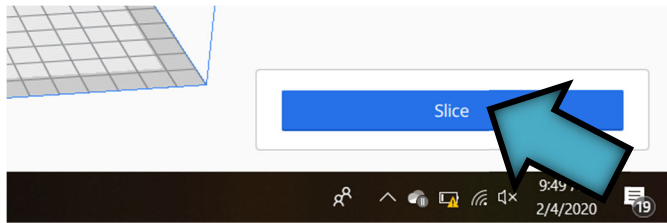
- 6) Scroll to the “Material” dropdown and set “Printing Temperature” to 210°C and the “Build Plate Temperature” to 60°C



- 7) Scroll to the “Speed” dropdown and set “Printing Speed” to 80 and “Initial Layer Speed” to 10

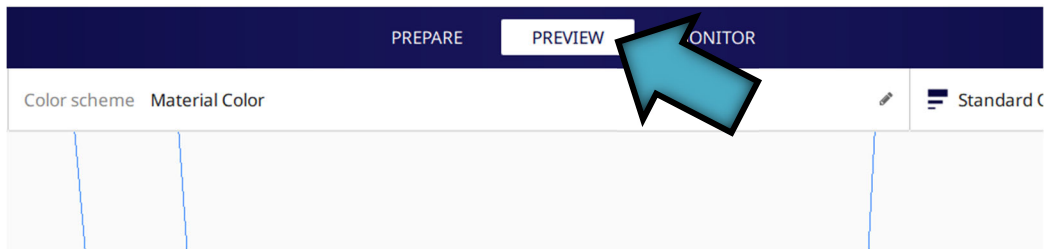


Step 6: Click Slice (bottom left hand corner)



Step 7: Click Preview (top center)

Note: Bar on left can be used to show the different layers being printed



Step 8: Save to removable disk

Step 9: Take G-Code file to printer

Note: Printer only uses micro SD