#### **Introduction**

This document is designed to be a supplement to class instruction. Contained in the document are all steps and safety needed to use the 3D printers located in the BBHHS Fab Lab. Machines covered are Lulzbot Taz 5, Lulzbot Taz 6, and Ender 3 pro.

\*Must have approval/supervision from BBHHS staff

### Step 1: Model

before using machines.







#### Option 2.2

Now you will need an <u>.STL</u> file of the part you want to print.

Example: Fusion 360

1. On the tools Tab select MAKE then click on 3D Print.



#### Option 2.2

- Now use the 3D Print window that pops up to select the model to be printed (click on your 3D model to highlight it blue).
- 3. Select the level of detail (most times medium or high is best).
- 4. Output: Either press OK to save the .STL to a storage location or check the box next to Send to 3D Print Utility to have Fusion send it directly to the slicer program for the printer (we use Cura).





## Step 2: Slice & Prepare Model

**1.** Once you have your file downloaded and saved you will have to set it up on Cura, the software we use for all of our 3D printers.

Open up Cura on your computer and then open up your file by clicking the open file icon on the top left. Your file will appear on the virtual build plate on the screen.



**2.** After opening the file, you need to orient it to achieve the best print. Select the model and use the rotate tool to rotate it such that its flat side is facing down and it has the least amount of overhangs (structures with no support on the bottom). If you are unsure which is the best orientation, please consult a teacher.



**3.** On the top right of the Cura window you will see the printer settings. First make sure the top green box matches the printer you are using or atleast the nozzle size is correct (ask the teacher if you have a question). Ensure that "Material" is set correctly to the material you are using..

If you wish to print in a material that is not PLA, you must use Ultimaker 1 labeled "Experimental Material." **No other printer can be used with materials that are not PLA.** Any print using a material that is not PLA must be approved by a Student Technician. They will help you through the process.



### ulzBot TAZ 5 | SE | 0.5 mm

All

i i -

Category

Material

Profile

Standard - 0.250mm

PLA (Verbatim)



**4.** Under "Print Setup," make sure the "**Recommended**" tab is selected. Options for Layer Height/Print Speed, Infill, Support, and Build Plate Adhesion should be available. Refer to the picture for the recommended settings (Note: the "Generate Support" setting is dependent entirely on your model. If you are unsure whether or not your object needs support, consult a Student Technician.) While the recommended settings are good for most prints, some adjustments may need made for others. The settings are described below.

**Layer Height/Build Speed:** This is the thickness of the material being deposited on the build plate. The smaller the number, the thinner the layer, and thinner layers result in higher quality prints. Thinner layers also result in longer build times.

**Infill:** This is how solid your object is. 100 percent infill would be completely solid while 0 percent infill would be completely hollow. More infill leads to a higher build time and a more expensive part. (20% is standard)

**Generate Support:** This creates support for any overhanging geometry to keep it from drooping while printing. The support is easily removed by peeling it away once the print is completed. It is generally recommended to print with the least amount of support.

**Build Plate Adhesion:** This creates extra layers around the base to help the print stick to the bed. This should be used in nearly every print. (**Brim** is Recommended)

Pictured here are the Custom settings. Please only use these if you know what you are doing or need something special and have asked someone knowledgeable about the settings you are adjusting.





**5.** After all the settings have been adjusted appropriately, select "Prepare" and Cura will automatically slice your part. Make note of the **time and material usage estimate** for the print on the bottom right. Be aware that the time estimate generated on the computer will differ from the estimate generated by the machine - the computer's estimate is correct.

Now, click "Save to File" and save the model to your H Drive then to a USB or SD Card.

Take the SD card or USB out and choose an available 3D Printer to print your model!

## Step 3: Use the Printer

- 1. Load Filament (If needed)
  - 1. Heat the printer to the temperature needed for printing.
  - 2. Once the extruder is hot, pull out old filament.
  - 3. Load filament spool on holder, feed the strand through the tube and insert the filament into the extruder.
  - 4. Either hold the extruder head and push filament through the head until the color is what you are looking for then lock the filament in with the tensioner. The other option is to insert the filament into the head then lock the tensioner. Once the tensioner is installed use the computer or the lcd controller to extrude filament until the old filament is gone.
  - 5. Return previous filament to air tight storage.









Modified from Case Western Think[box] Document.

assistance.