1. Turn on the two flip switches on the Lulzbot power supply and the box connected to the printer
2. On the computer, go to the start menu and look for Cura 19.12
3. Program will load and you will need to select settings
   1. select Taz 4 or 5 icon
   2. select stock Taz 5 (PET & v2) icon
   3. select 0.5mm nozzle diameter
   4. click finish
4. Cura should start up now
5. Delete the model on the screen by right clicking and selecting delete object
6. Go to the expert tab and select "switch to full settings"

\*\*NOTE: you may not have to do steps 3 – 6 if it already starts up this way\*\*

1. Click the basic settings tab
2. The red box indicates the fill density which is how dense the model will print
   1. 0 is hallow, 100 is solid
3. The Orange box indicates the temperature to print. \*\*IMPORTANT\*\*
   1. print speed should never exceed 125 mm/s
   2. printer temperature depends on the filament
      1. Example: For PLA, set it at 210
   3. bed temperature depends on the filament as well
      1. Example: For PLA, set it at 60
4. The green box indicates the supports needed for the model
   1. Generally, it is recommended to set it to "touching buildplate"
      1. Click the 3 dots next to the support type and change "fill amount" to 2-5 for easier supports to remove after printing
   2. Platform adhesion type. You may not have to use this if your model has a large flat portion in contact with the bed plate
      1. You may choose one of the options if you need to.
         1. Brim is a single layer print that will keep your object adhered to the bedplate.
         2. Raft is a multiple layer print that is commonly used when the model has little connecting surfaces on the bed.
5. The Blue box indicates the filament size.
   1. The numbers will only be 2.85mm or 3mm.
   2. If you are unsure, choose 3mm
6. Click the advanced tab
7. The Red box indicates the initial layer thickness -Change the number to 0.0
8. The Blue box indicates the speed -Change the travel speed to 100 mm/s
9. To Load a model onto Cura, Drag and drop your .STL or .OBJ file onto the Cura window or go to the file -load model file
10. You can click the model to change the rotation, scale, or mirror it.
    1. It is recommended to rotate the model so that surface that is the flattest and the biggest will be touching the bed plate
11. After all settings are configured, go to the control icon at the top of the window
12. clicking this icon will bring up another window
13. wait for the printer to connect
    1. you will know it is connected when the top of the window says "operational"
14. Next, click the print button and the printer will start the printing process
15. Once print is completed, you can remove your model. The model may be stuck to the bed plate very well and this may require the use of a spatula
    1. Be careful not to damage the bed plate or the machine
    2. Do not force the object off of the bed plate

NOTE: If the bed plate is chipped/damaged or any other part of the machine when using the printer, you are liable for any charges. (Glass plate $35.00, Full Heated Plate Kit $150.00, etc)

1. Follow steps 1-4 in "setting up a print" section
2. Next, go to control icon on the top of the Cura window
3. Go to the temperature box and type in 220 (for PLA) and click set
   1. This will set the hot end (nozzle) to 220 if using PLA and will allow for the removal of the filament
   2. It is important to wait till the nozzle reaches full temperature before moving on
   3. Once the nozzle reaches 220, go back to the additional window and navigate to the retract icons
4. Once filament is all unloaded, insert the new filament into the same hole that the previous filament was in
5. Next click the extrude arrow pointing down
   1. you may need to click the extrude arrow 8 or more times till the filament starts
6. coming out of the nozzle
   1. Make sure the old color filament is completely extruded out before moving on
   2. You may need to wiggle the filament around till the gears grip the strand
7. Once that is done, move to the "Setting up a print" section

**Settings for Filaments provided by Dospace**

|  |  |  |
| --- | --- | --- |
| **Filament** | **Extruder** | **Bed** |
| PLA |  |  |
| Esun | 220 | 60 |
| Meltink | 240 | 60 |
| ABS\* | 240 | 110 |

\*run ABS with no fan

Advanced Setup

I have created a collection of Print profile that can be used on the Taz and they can be found at https://github.com/Masterwolf2050/3dprofiles. You are free to fork this repo if you like. If you have any issues or suggestions, feel free to post them to the repo as well. You can also find more info on filament types by brand as well as how tos at https://devel.lulzbot.com/filament/