TAZ 3 Lulzbot 3D Printer

Location of Machine: Composites Lab, RFM 1218

Location of SOP and Machine Operating & Safety Manual: Composites Lab website under resources; Composites Lab TRACS site; and Hardcopy near machine.

Emergency Contact:

- Call 911
- Call EHS & Risk Management at 512-245-3616
- Call Head Lab Technician, Dr. Ray Cook (office 512-245-2050)
- Call Dr. Jitendra S Tate (office 512-245-4872)

Before using this machine:

- You must have permission from Dr. Tate.
- You must have received formal training from technician or, trained research student (designated by Dr. Tate) related to machine safety and operation.
- You must read and understand **SOP and Machine Operating & Manual.**
- You must use this machine under direct supervision of Dr. Tate or, Dr. Cook or, trained research student (designated by Dr. Tate).
- You must have signed "Lab Rules" document with Dr. Tate. This document must be signed every semester fall, spring, and summer (as applicable).
- If you do NOT follow above instructions you will be held responsible for your own safety and damages.

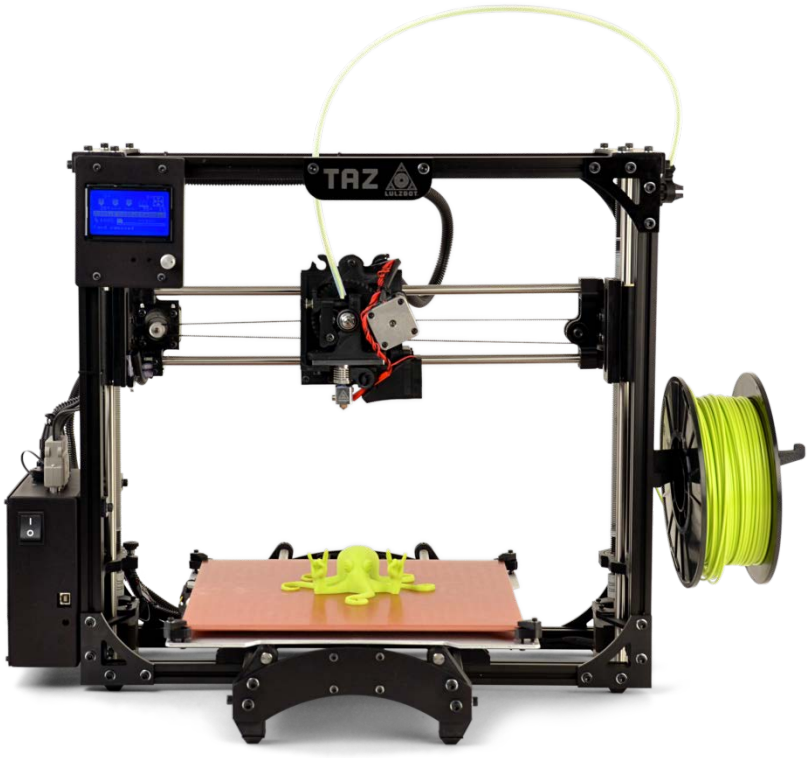
Safety Precautions:

Protective Equipment: Prior to performing this procedure, the following personal protective equipment must be obtained and ready for use: **Safety Goggles and lab coat**

Important Safeguards:

- Never touch the hot end or the bed with your fingers while the heat is activated. Use tongs, tweezers, or pliers to adjust hot components. Hot end must be heated before disassembly/reassembly.
- Deactivate printer motors before manually adjusting steppers. Failure to do so can cause back-EMF and burn out the motherboard.
- Ensure both the motherboard and power supply are off and unplugged before attempting to perform maintenance on electrical wires and components.

Accidental Spill: (In the event that a hazardous material spills during this procedure, be prepared to execute the following emergency procedure)
Clean with cleaner according to MSDS of material used



General information

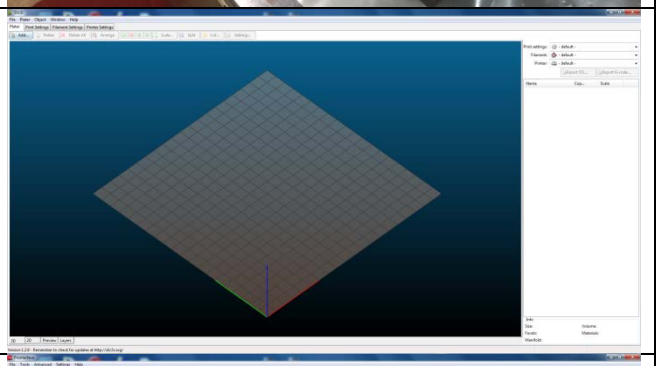
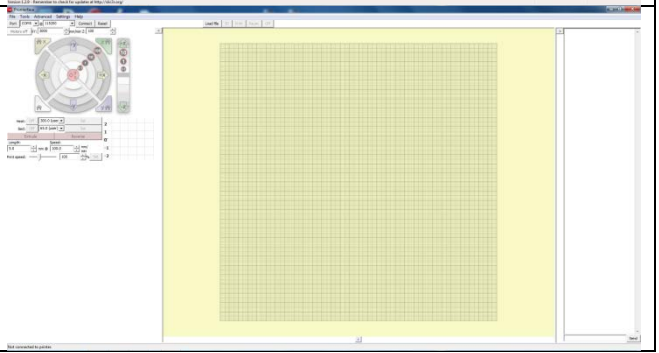
The evolution of desktop 3D printing continues with TAZ 3, LulzBot's top-of-the-line, highest quality printer to date. Merging technical expertise with design sensibilities, TAZ is for inventors, entrepreneurs, design engineers and prototypers — bring your ideas to life with TAZ.

Built for Performance TAZ is engineered for quality and tested for the long haul. As the most dependable desktop 3D printer on the market, TAZ requires minimal maintenance - helping you advance productivity and reduce costs. TAZ prints fine layer heights at speeds faster than most of the competition, is perfect for small manufacturing and comes with self-lubricating bushings. It's also the quietest printer on the market.

Wood, polystyrene, plastic? Fantastic. TAZ is one of the few desktop 3D printers that can print with more than just your run-of-the-mill plastic. Create with ABS, PLA, PVA, high-impact polystyrene, and wood filament. But don't stop there. With add-ons, you can print with nylon and more.

Specifications:

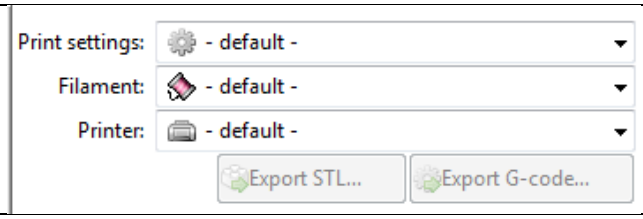
- Materials: PLA, ABS, Nylon, PVA, PS
- Temperature Ranges:
    - Hot end: 0 - 300°C
    - Bed: 0 - 120°C

TAZ 3 Lulzbot: Setup	
1. Turn on Printer Power Supply a) Plug the power cord into the outlet located underneath the printer table and flip switch. b) Ensure cooling fan is functional.	
2. Turn on Printer Motherboard a) Flip the switch on the motherboard on the left side of the printer. b) Printer should turn on and display home screen.	
3. Open Slic3r on desktop a) This program slices 3D printer files and sets parameters for printer.	
4. Open Pronterface on desktop a) This program controls printer movement and temperatures.	

5. Set Printing Parameters in Slic3r

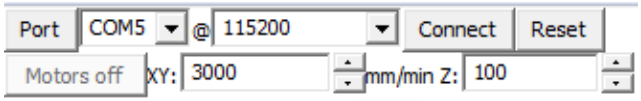
a) Select your **Print Settings**
b) Select your **Filament**
c) Select your **Printer**

*See **Slic3r Settings** for Specific Material Settings*

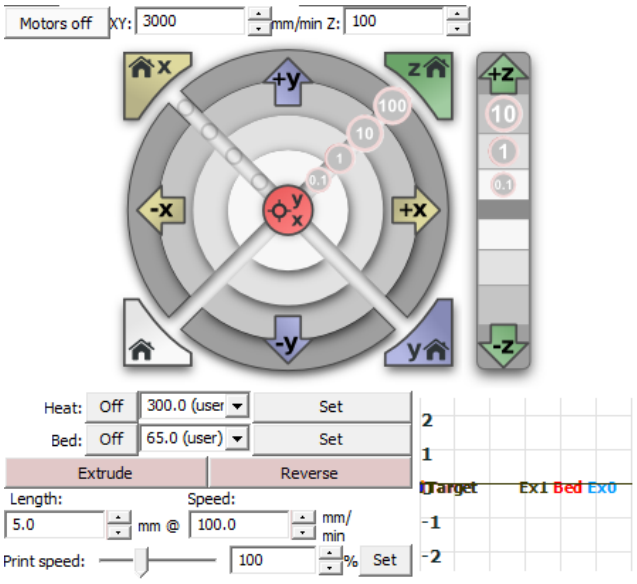


6. Set Printing Parameters in Pronterface

a) Connecting Printer
   a. Select **Port** or Choose from dropdown menu.
   b. Typically use **COM5 @115200**
   c. Select **Connect**
      i. Note: If problems connecting, see Troubleshooting.

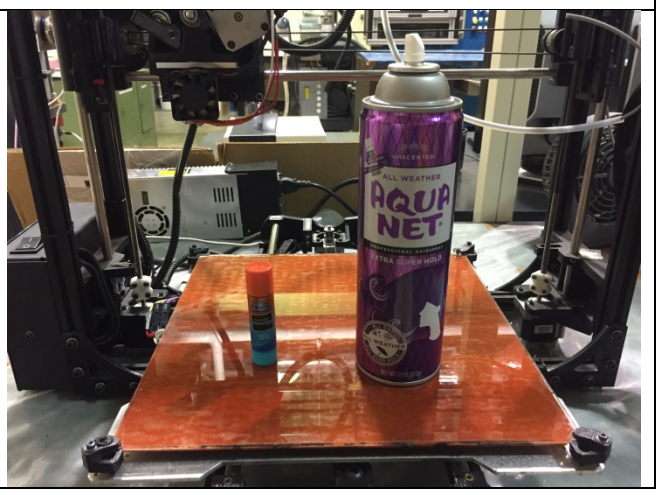


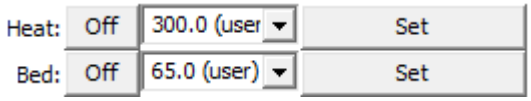
b) Homing Printer Head
   a. Colored home buttons home respective axis. White Home button homes all three.
   b. Numbers indicate head movement in mm relative to home position. (Example: Picture shows moving 10mm to the right in X-axis.)
   c. Note: Printer head must be re-homed after nozzle or bed removal/replacement. See troubleshooting.
   d. **Note: If positioning printer head or bed manually, select "Motors off" first. Otherwise, motherboard could burn out from back EMF.**



7. Prepare Printer Bed

a) Depending on the type of filament, prepare the bed for material setting.
b) Ensure the bed is clear of any debris or plastic from a previous print. Use water and a putty knife or razor blade to clear dried plastic.
c) For PLA and ABS, spray a generous amount of hairspray on the printing area and let dry for a few minutes. Note: Be careful not to get any hairspray on the rest of



the printer (print head, steppers, etc.) as this can cause seizing or slipping. d) For PLA, ABS, and Nylon, use a glue stick to apply a generous coating on the printing area.	
8. Set Temperatures a) Heater: a. PLA: 180°C – 200°C b. ABS: 230°C - 235°C c. Nylon: 230°C - 235°C b) Bed: a. PLA: 55°C - 70°C b. ABS: 85°C - 90°C c. Nylon: 85°C - 90°C c) Periodically check temperatures throughout print.	

TAZ 3 Lulzbot: Printing

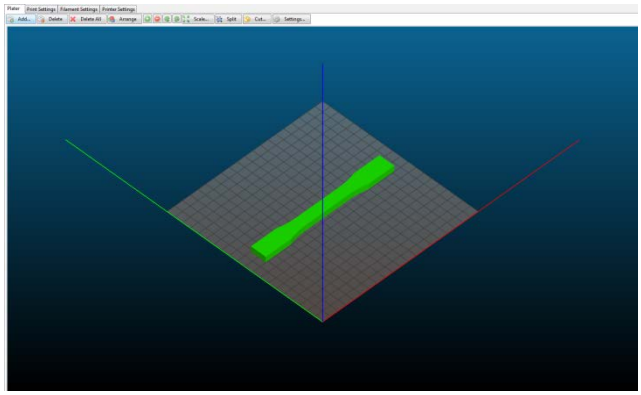
1. Choose a File to Print

a) Open an STL file in Slic3r from the Desktop by selecting **Add** in the **Plater** tab.

b) Files can be created using 3D modeling software such as AutoCAD, MasterCAM, Solidworks, etc. or imported from online.

c) You can add, delete, rotate, and scale parts in this window.

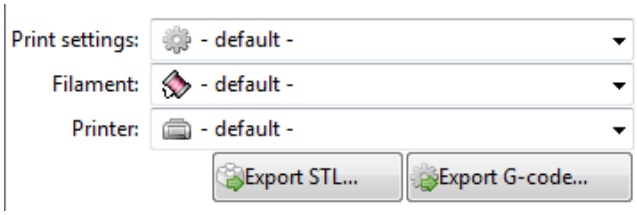
d) Use websites like myminifactory.com or thingiverse.com to download pre-made STL files. Tinkercad.com can be used to edit STL files (flipping, rotating, scaling, etc.)



2. Slicing the File

a) Once the part is in the desired position and orientation, and the printer settings are set, select **Export G-code** to convert the STL file to G-code.

b) Save the G-code file in the appropriate location.



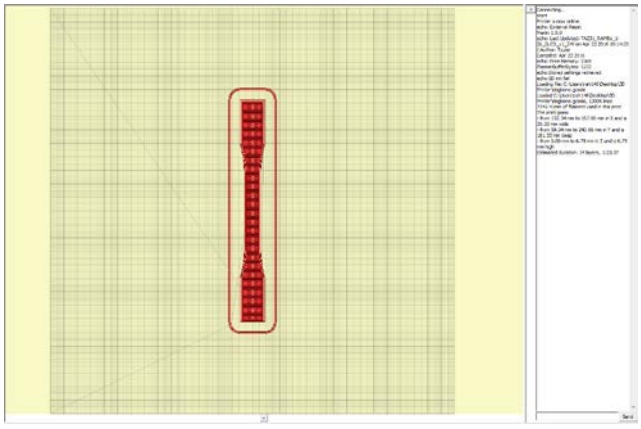
3. Importing into Pronterface

a) In Pronterface, click **Load File**, and select the G-code file you want to print.

b) Note: Selecting the STL file instead of the G-code file will result in an error.

c) You will see the first layer in the yellow grid. The part may go beyond the grid if the printer size is incorrect. <u>This is normally fine and will not impact the printing process.</u> You may want to check the **Printer Settings** under **Settings > Options** in Pronterface to ensure the bed size is correct.

d) On the right side, a series of text will be displayed showing the dimensions of the build and an estimated print time.



4. Printing

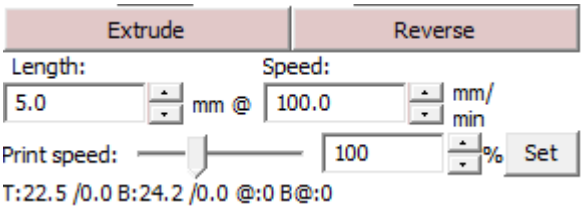
a) Home the printer

b) Set **Length** to approximately 5mm and click **Extrude** once or twice. This will extrude a small amount of filament to ensure there are no clogs.

c) If no problems, make sure the hot end and bed are up to temperature, then click **Print** and the printer will begin.

d) Monitor the printer to ensure the first few layers are lying down correctly. If you notice material not sticking to the bed, or material is not coming out at all, stop the print job and troubleshoot.

e) Let the printer run, but periodically monitor the print job to ensure no clogging occurs.

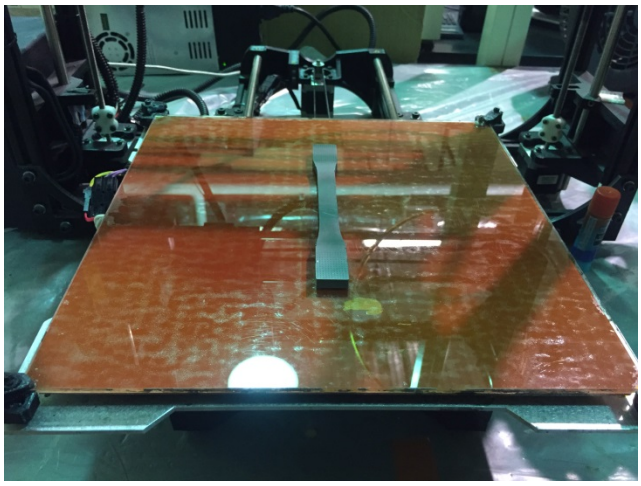


5. Part Removal

a) Once the part is complete, the print head will return to its home position, and the bed will extend out. The heater and bed will turn off and return to room temperature. Let the part and bed cool a bit before removal to avoid warping the part. At this point, you may remove the 3D printed part from the bed using a putty knife or your hands. Be careful not to break the part. A warm, not hot bed is ideal for easy part removal.

b) Clear the bed of any excess or support material.

c) Clean off the finished part by removing the support material with your hands or with a flat tool.



6. Shutting Down

a) Once you are finished with the printer, click **Disconnect** in Pronterface and close the program. You can also close Slic3r.

b) Cut the filament at the top of the print head to separate the spool. Keep the spool wound, and store it in the vacuum bags with a desiccant canister to keep out moisture.

c) Flip the switch on the Motherboard to turn off the printer.

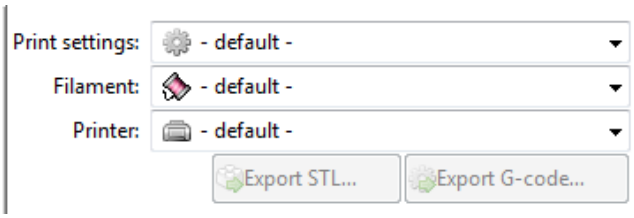
d) Flip the switch on the Power Supply to turn off the power.

e) Return any tools to their proper locations and clean up any debris around the printer.

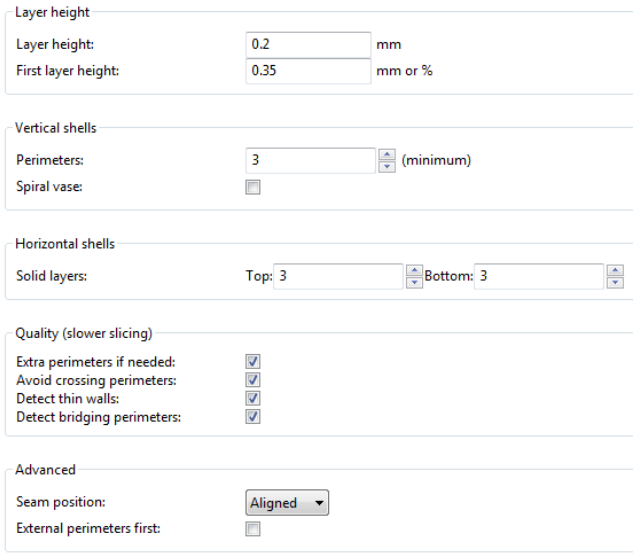
TAZ 3 Lulzbot: Slic3r Settings

1. Print Settings:

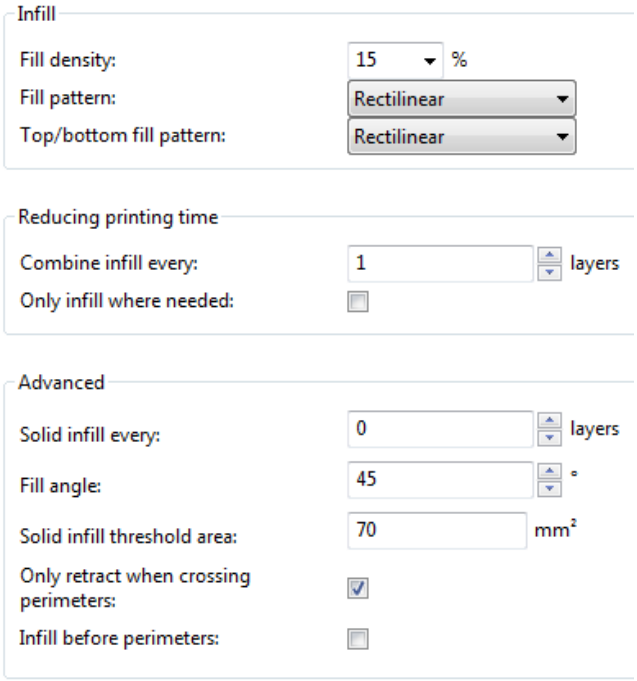
a) Select desired material from drop down menu



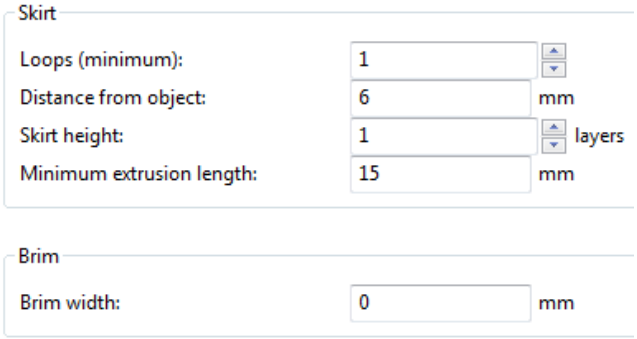
b) Layers and Perimeters:
   a. Layer Height:
      i. Layer Height: How tall each extruded layer will be.
      ii. First Layer Height: How tall the initial layer will be.
   b. Vertical Shells:
      i. Perimeters: Minimum number of shell perimeters per layer.
      ii. Spiral Vase: Gradually raises Z axis to eliminate seams.
   c. Horizontal Shells:
      i. Solid Layers: Number of horizontal shells on the top and bottom.
   d. Quality:
      i. Extra Perimeters if needed: Adds extra perimeters to avoid gaps in walls.
      ii. Avoid crossing perimeters: Minimizes oozing and dragging over internal perimeters.
      iii. Detect Bridging Perimeters: Adjusts flows for overhang
   e. Advanced:
      i. Seam Position: Position of perimeters starting point.
      ii. External Perimeters First: Prints external perimeters before internal perimeters.



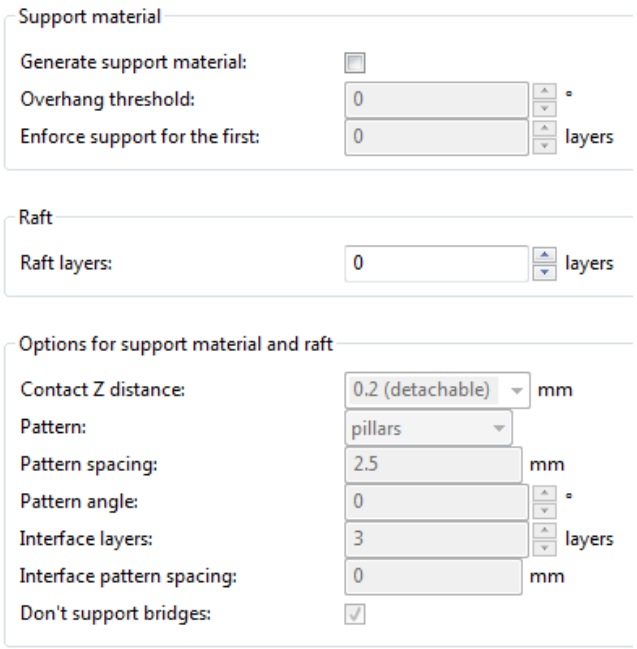
c) Infill:
   a. Infill
      i. Fill Density: Density of internal fill.
      ii. Fill Pattern: Pattern for infill
      iii. Top/Bottom Fill Pattern: Pattern infill of top and bottom layers.
   b. Reducing Printing Time:
      i. Combine infill every: Combines infill layers
      ii. Only infill w here needed: Infills only when necessary
   c. Advanced:
      i. Solid Infill Every: Forces a solid infill
      ii. Fill Angle: Angle of infill
      iii. Solid Infill Threshold Area: Forces solid infill in regions small than specified area
      iv. Only Retract When Crossing Perimeters: Will only retract when path and layer height will cross.
      v. Infill Before Perimeters: Switches print order of perimeters and infill.



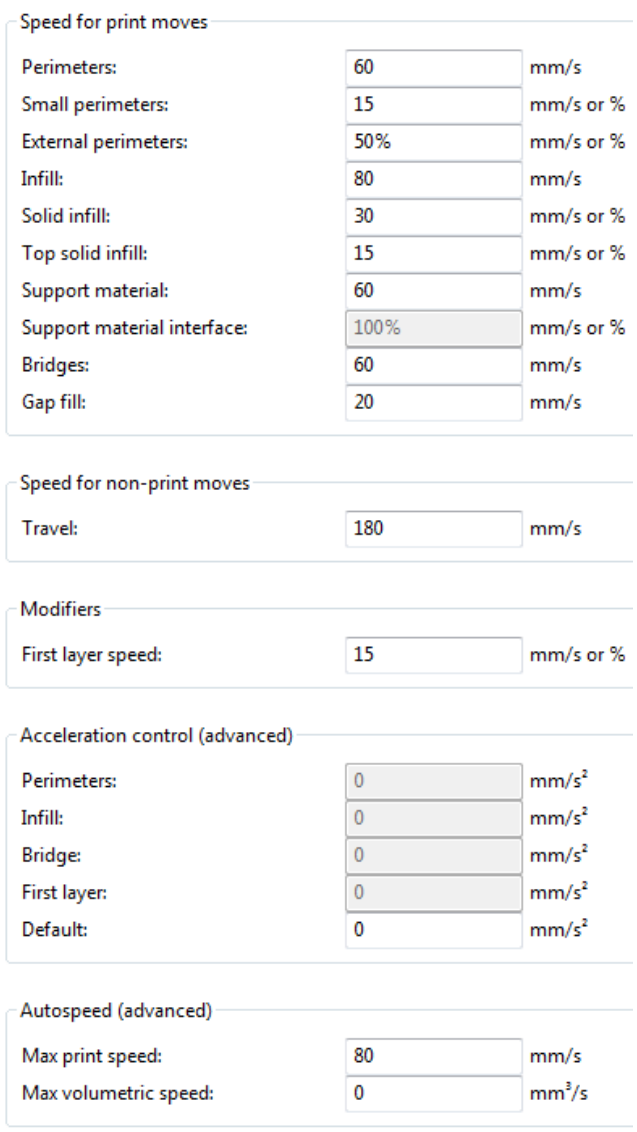
d) Skirt and Brim:
   a. Skirt:
      i. Loops: Minimum number of loops for skirt
      ii. Distance from Object: Distance skirt will be from part
      iii. Skirt Height: Number of layers for skirt
      iv. Minimum Extrusion Length: Minimum amount of material to extrude for skirt.
   b. Brim:
      i. Brim Width: Width of brim around part



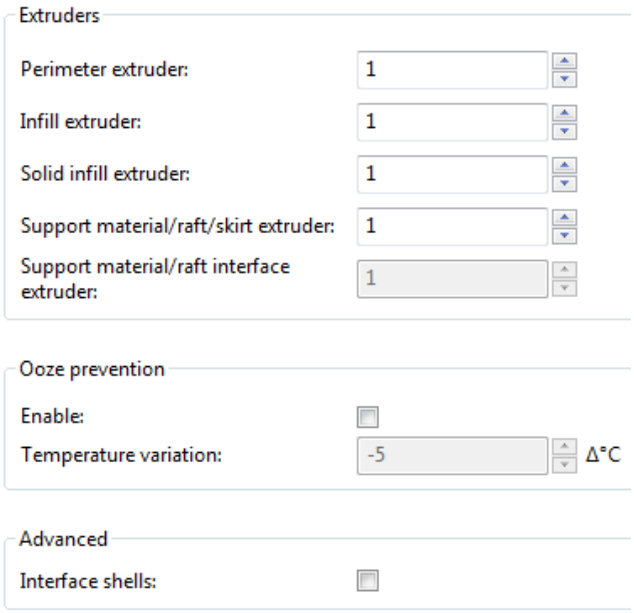
e) Support Material:
   a. Support Material:
      i. Generate Support Material: Choose to use support material.
      ii. Overhang Threshold: Angle of slope at which support material will begin to generate.
      iii. Enforce Support for the First: Forces support for first number of layers
   b. Raft:
      i. Raft Layers: Raises object by a number of layers and prints support underneath.
   c. Options for Support Material and Raft:
      i. Contact Z Distance: Distance between support material and object.
      ii. Pattern: Pattern type of support.
      iii. Pattern Spacing: Spacing between support lines.
      iv. Pattern Angle: Rotates support pattern horizontally.
      v. Interface Layers: Number of interface layers between support and object.
      vi. Interface pattern spacing: Spacing between interface lines.
      vii. Don't Support Bridges: Support will not generate under bridge areas.



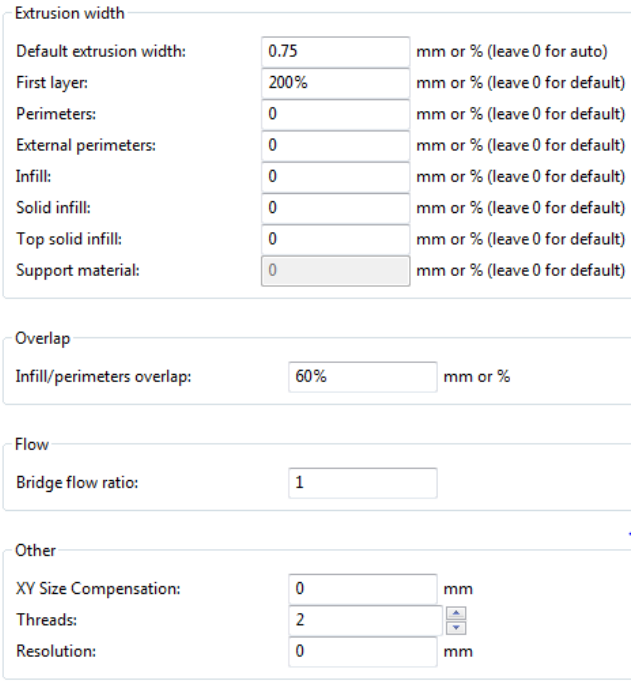
f) Speed:
   a. Speed for Print Moves:
      i. Perimeters: Speed for perimeters (shells).
      ii. Small perimeters: Speed for small radii (holes).
      iii. External Perimeters: Speed for external perimeters.
      iv. Infill: Speed for infill.
      v. Solid Infill: Speed for solid infill.
      vi. Top Solid Infill: Speed for infill of top layer.
      vii. Support Material: Speed for support material.
      viii. Support Material Interface: Speed for support material interface.
      ix. Bridges: Speed for bridges.
      x. Gap Fill: Speed for filling small gaps.
   b. Speed for Non-Print Moves:
      i. Travel: Print head travel speed between extrusions.
   c. Modifiers:
      i. First Layer Speed: Speed for first layer.
   d. Acceleration Control:
      i. Perimeters: Acceleration for Perimeters.
      ii. Infill: Acceleration for infill.
      iii. Bridge: Acceleration for Bridges.
      iv. First Layer: Acceleration for first layer.
      v. Default: Default acceleration.
   e. Autospeed:
      i. Max Print Speed: Max autocalculated speed.
      ii. Max Volumetric Speed: Max volumetric speed for supports.

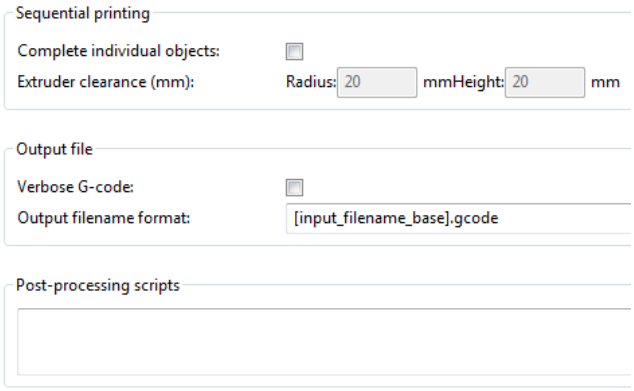
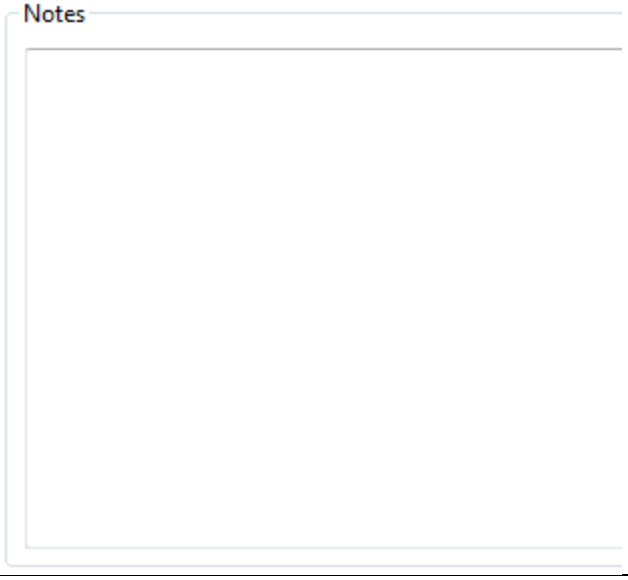


g) Multiple Extruders:
   a. Extruders:
      i. Perimeter Extruder: Select extruder for perimeters.
      ii. Infill Extruder: Select extruder for infill.
      iii. Solid Infill Extruder: Select extruder for solid infill.
      iv. Support Material/Raft/Skirt Extruder: Select extruder for support/raft/skirt.
      v. Support Material/Raft Interface Extruder: Select extruder for support/raft interface.
   b. Ooze Prevention:
      i. Enable: Drops temperature of inactive extruders to prevent oozing.
      ii. Temperature Variation: Change in temperature for inactive extruders.
   c. Advanced:
      i. Interface Shells: Forces shell generation between adjacent materials.



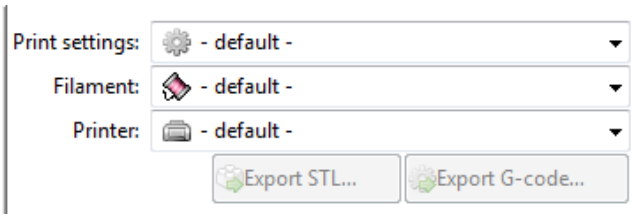
h) Advanced:
   a. Extrusion Width:
      i. Default Extrusion Width: Default extrusion width (autocalculated).
      ii. First Layer: Extrusion width of first layer.
      iii. Perimeters: Extrusion width of perimeters.
      iv. External Perimeters: Extrusion width of external perimeters.
      v. Infill: Extrusion width of infill.
      vi. Solid Infill: Extrusion width of solid infill.
      vii. Top Solid Infill: Extrusion width of top solid infill.
      viii. Support Material: Extrusion width of support material.
   b. Overlap:
      i. Infill/Perimeters Overlap: Overlap of infill and



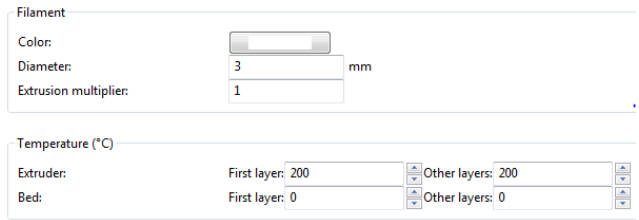
perimeters. c. Flow: i. Bridge Flow Ratio: Amount of plastic for bridging. d. Other: i. XY Size Compensation: Grows/shrinks object in XY plane for fine tuning. ii. Threads: Parallelizes long-running tasks. iii. Resolution: Minimum detail resolution.	
i) Output Options: a. Sequential Printing: i. Complete Individual Objects: Completes individual parts before proceeding when printing multiple objects or copies. ii. Extruder Clearance: 1. Radius: Clearance radius for extruder. 2. Height: Clearance height for extruder. b. Output File: i. Verbose G-code: Provides a descriptive G-code. ii. Output Filename Format: Format for G-code filename. iii. Post-Processing Scripts: Custom scripts.	
j) Notes: a. Notes: Personal Notes appended to the G-code.	

2. Filament Settings

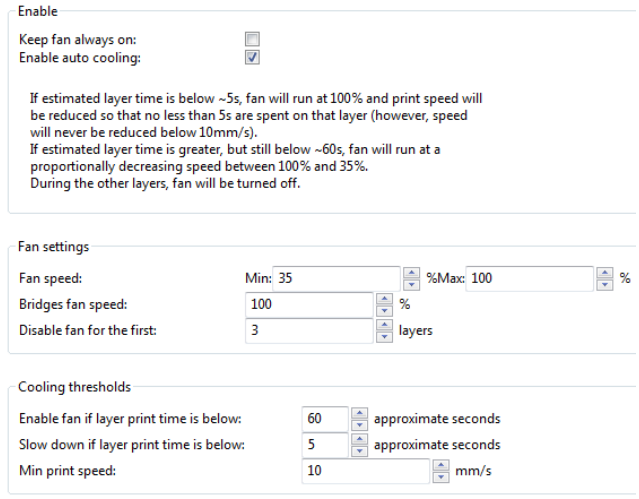
a) Select desired material from dropdown menu.



b) Filament:
   a. Filament:
      i. Color: Filament color.
      ii. Diameter: Filament diameter.
      iii. Extrusion Multiplier: Flow proportions.
   b. Temperature:
      i. Extruder:
         1. First Layer: Temperature for first layer.
         2. Other Layers: Temperature for subsequent layers.
      ii. Bed:
         1. First Layer: Temperature for first layer.
         2. Other Layers: Temperature for subsequent layers.

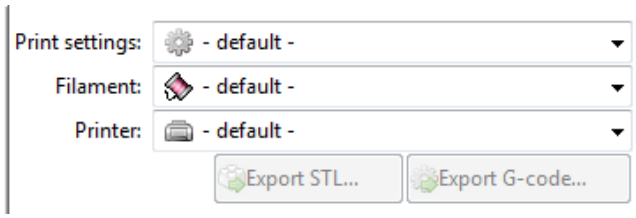


c) Cooling:
   a. Enable:
      i. Keep Fan Always On: Forces cooling fan to stay on.
      ii. Enable Auto Cooling: Automatically adjusts print speed and fan speed based on layer time.
   b. Fan Settings:
      i. Fan Speed:
         1. Min: Minimum fan speed.
         2. Max: Maximum fan speed.
      ii. Bridges Fan Speed: Fan speed for bridges.
      iii. Disable Fan for the First: Disables fan for first number of layers.

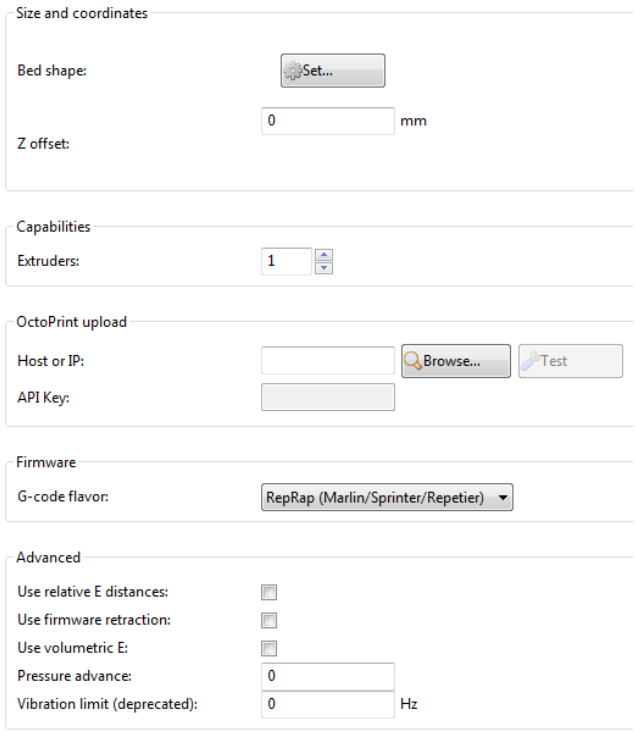


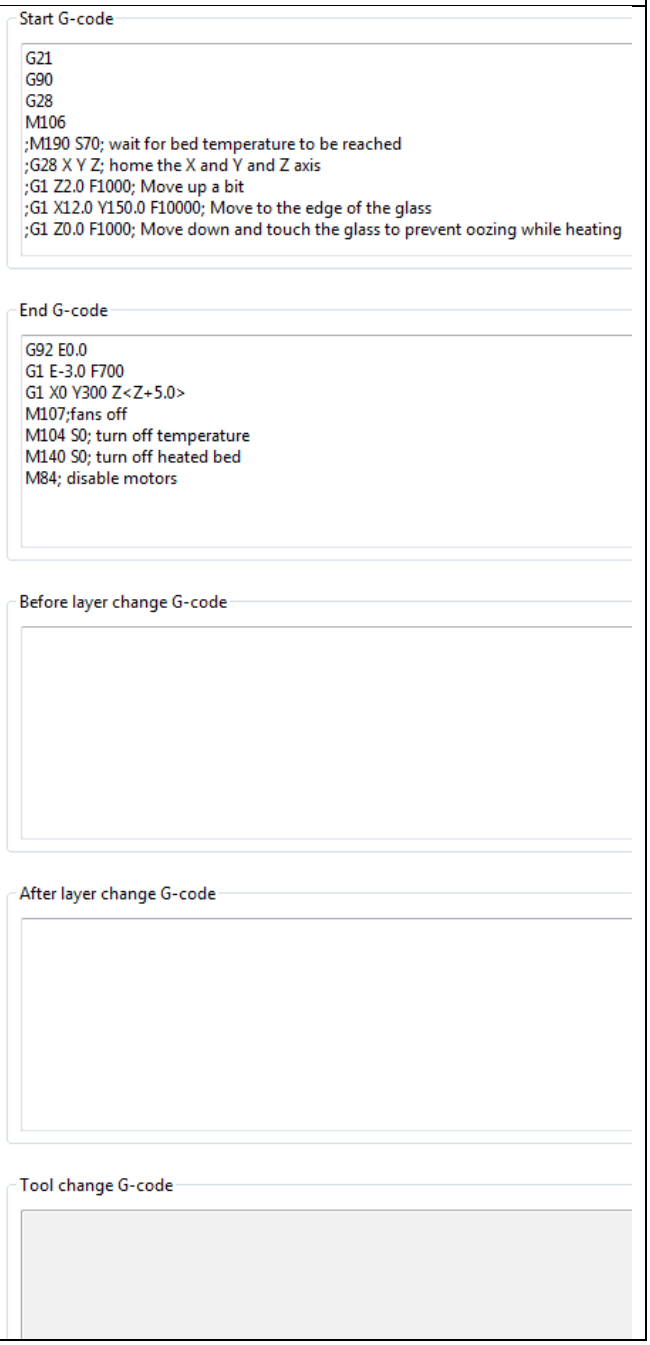
c. Cooling Thresholds:
   i. Enable Fan if Layer Print Time is Below: Enables fan is print time is under a set time.
   ii. Slow Down if Layer Print Time is Below: Slows down print speed if print time is under a set time.
   iii. Min Print Speed: Will not slow down print speed less than set value.

3. Printer Settings:
   a) Select desired material from dropdown menu.

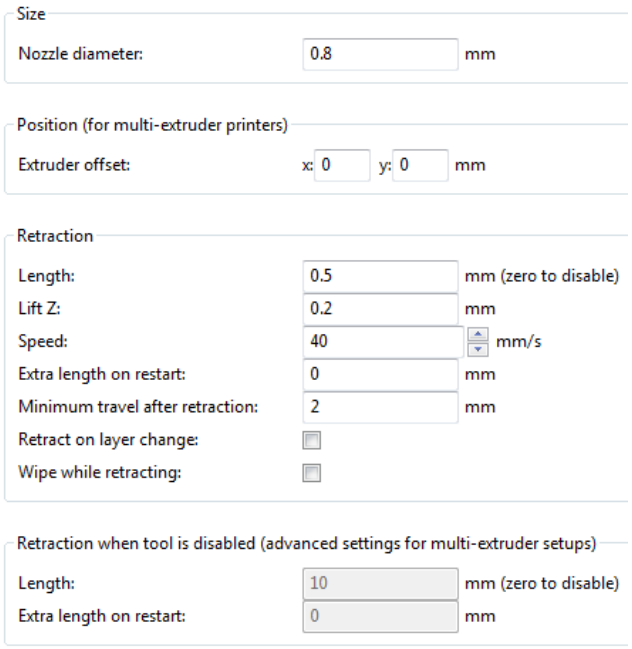


   b) General:
      a. Size and Coordinates:
         i. Bed Shape: Set bed shape and size.
         ii. Z offset: Offsets for Z axis relative to bed to account for bed roughness.
      b. Capabilities:
         i. Extruders: Number of extruders for printer.
      c. OctoPrint Upload:
         i. Host or IP: IP address to upload G-code to OctoPrint.
         ii. API Key: Authentication key.
      d. Firmware:
         i. G-Code Flavor: Type of G-code.
      e. Advanced:
         i. Use Relative E distances: Enables relative distances.
         ii. Use Firmware Retraction: Enables firmware to handle extruder retractions.
         iii. Use Volumetric E: Outputs E values in cubic centimeters.



iv. Pressure Advance: Enables pressure regulation. v. Vibration Limit: Minimize mechanical resonance by slowing down movements reaching set frequency.	
c) Custom G-code: a. Start G-code: G-code prior to print b. End G-code: G-code post print. c. Before Layer Change G-code: G-code inserted at layer change before Z move. d. After Layer Change G-code: G-code inserted at layer change after Z move. e. Tool Change G-code: G-code for changing tools.	

d) Extruder:
    a. Size:
        i. Nozzle Diameter: Diameter of extruder nozzle.
    b. Position:
        i. Extruder Offset: Extruder offset in XY plane.
    c. Retraction:
        i. Length: Retracts filament by set length when triggered.
        ii. Lift Z: Raises Z axis by set distance when triggered.
        iii. Speed: Retraction speed.
        iv. Extra Length on Restart: Pushes set length of material after retraction.
        v. Minimum Travel After Retraction: Minimum distance to move before printing resumes.
        vi. Retract on Layer Change: Forces retraction for each layer change.
        vii. Wipe While Retracting: Moves extruder while retracting to minimize oozing.
    d. Retraction When Tool is Disabled:
        i. Length: Retracts filament by set length for disabled tool.
        ii. Extra Length on Restart: Pushes set length of material when restarting disabled tool.

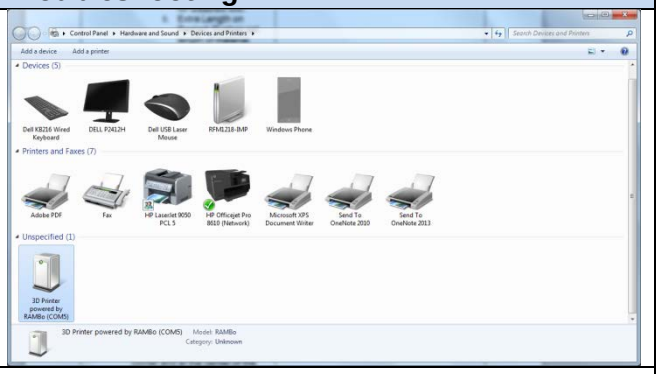


TAZ 3 Lulzbot: Troubleshooting

1. Connecting to Printer

a) Close Pronterface, unplug/replug grey cable from motherboard, then relaunch Pronterface.

b) If problem persists, close Pronterface, unplug/replug grey cable from motherboard, click Start > Devices and Printers > troubleshoot COM5, then relaunch Pronterface.



2. Re-Homing

a) Turn off Nozzle and Bed heat.

b) Raise the printer head in the Z-axis enough to have adequate working room.

c) Loosen the four springs on the corners of the bed using an allen wrench.

d) Tighten the springs with fingers equally.

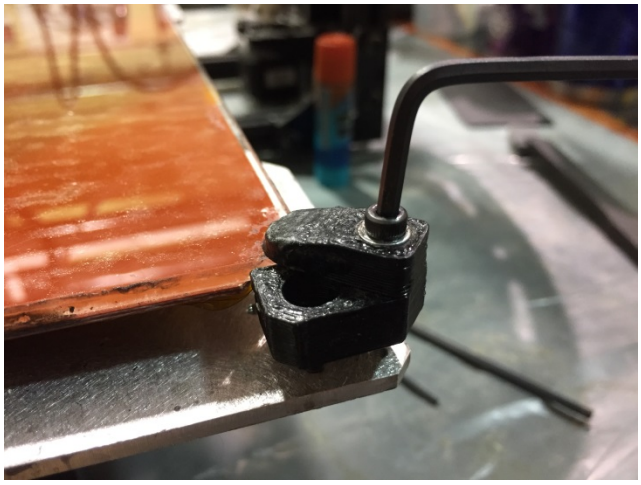
e) Lower the printer head to its current home position.

f) Select Motors Off

g) Check the distance between nozzle and bed with a thin piece of paper or feeler gauge at each corner and at the center of the bed. There should be a small amount of tugging from the nozzle on the paper.

h) Adjust the springs as needed.

i) Home the printer head on Pronterface and double-check distances.



3. Not Heating Up

a) If neither the hot end nor the bed start heating, check the fuses on the motherboard as they may have blown.