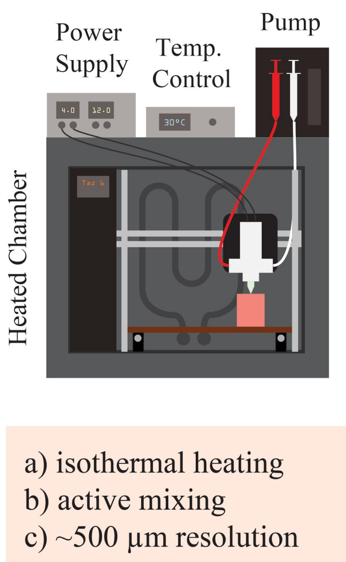
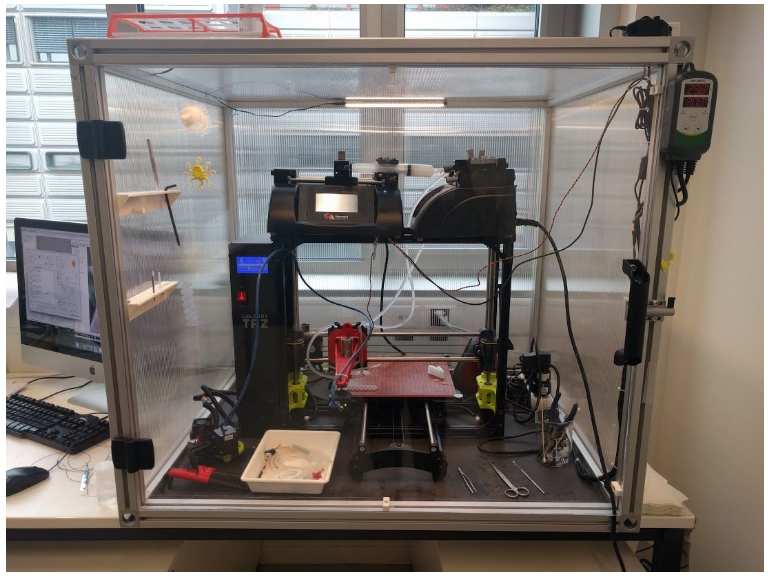
# How to Use the High Resolution Silicone 3D Printer (Full Version)

---



**Hello! Thank you for your interest in this printer and I hope you find this information valuable and informative!**

**Main contact: Steph Walker (walkerstephwork@gmail.com)**

**First, a disclaimer:**

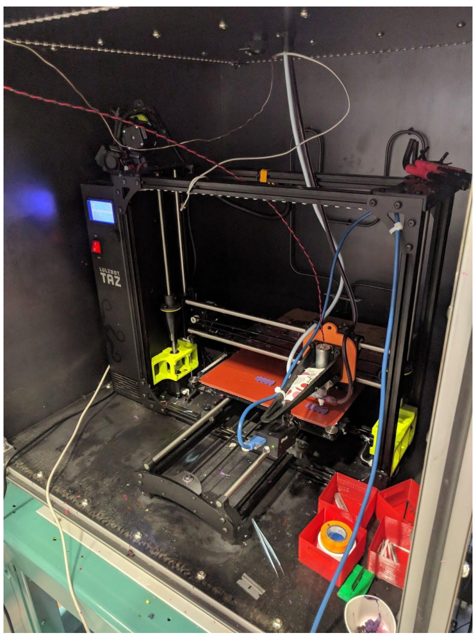
**Remember - this is a research machine and you will not be able to press print and walk away when you use this unless you have printed a few times and have your settings dialed-in. It takes some time to learn how the machine operates and where the errors occur. I have done my best to record the things that you will need to look for when printing, though there may be things I have accidentally omitted. If you notice any missing information, please feel free to submit it to the Github page for this printer so that others can benefit from your experience!**

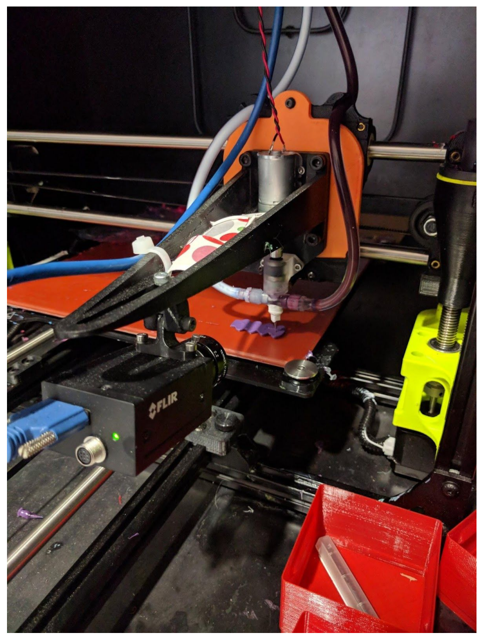
## Printer Setup V1 and V2

---

There are probably several ways to assemble this printer, but in general you need a way to make the inner chamber temperature controlled, power the active mixer, and run the gantry. The video camera is an added bonus and helps with troubleshooting of filament extrusion.

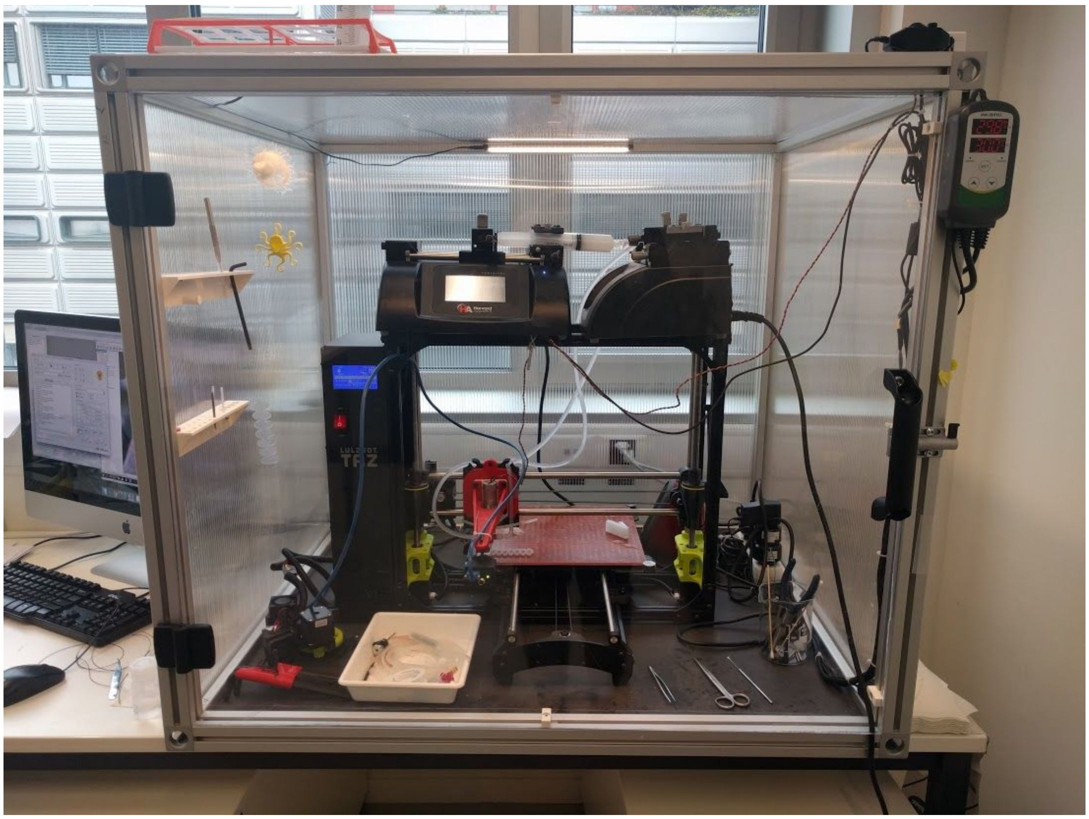
V1:

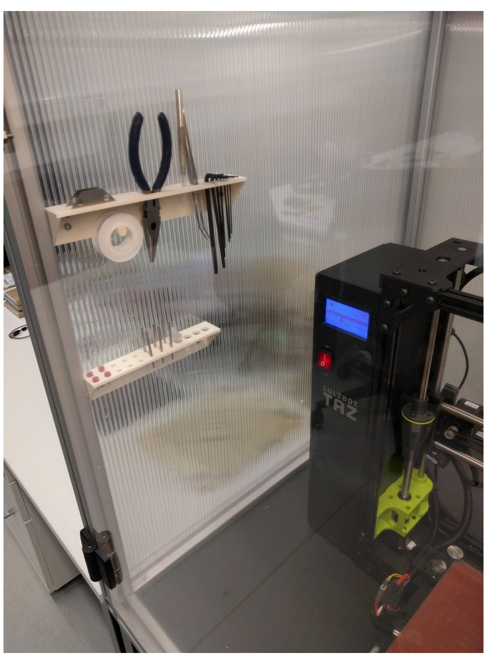


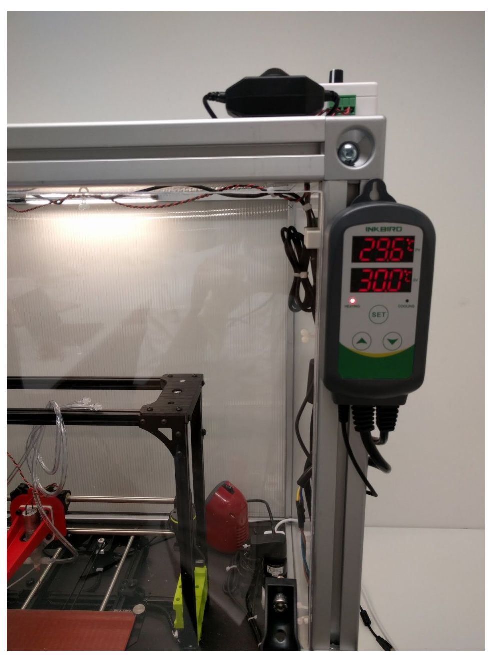




Printer V2 put the syringe pump inside the heated chamber but care is needed not to exceed the max operating temperature of the syringe pump. The chamber was made of lightweight plastic boards typically used for shower doors. The only temperature control is heating because the air-conditioned room served as a heat sink.

V2:







## Bill of Materials

| | | | | | | Grand Total: | $9,467.61 | |
|---|---|---|---|---|---|---|---|---|
| Category | Item / Description | Vendor | Link | Part Number | Qty | Cost ($) | Sub Total | Notes |
| Hardware | Harvard Apparatus PhD Ultra Syringe Pump | Harvard Instruments | [link] | 70-3007 | 1 | $4,303.00 | $4,303.00 | Can be replaced as long as force is sufficient, will require recalibration for flow |
| | Lulzbot Taz 6 3D Printer | Amazon | [link] | B01DLU3M3M | 1 | $2,500.00 | $2,500.00 | Can be any 3D printer gantry, but custom parts for this are designed for Taz 6 |
| | Precision Nozzles 27 Gauge | Production Automation Corporation | [link] | MT27-PBN MT | 10 | $16.61 | $166.10 | For other nozzle companies or cheaper ones, 3D printed mixer dimensions need adjusting |
| | Grasshopper3 3.2 MP Color USB3 Vision (Sony Pregius IMX252) | Point Grey | [link] | GS3-U3-32S4C-C | 1 | $975.00 | $975.00 | 9 week lead time |
| | Fresco FL1100, 4 Port, USB 3.1 Host Controller Card | Point Grey | [link] | ACC-01-1202 | 1 | $60.00 | $60.00 | 9 week lead time, not needed if computer already is capable of reading the video (see Grasshopper manual) |
| | USB 3.1, 3m, Type-A to Micro-B (Locking) Cable | Point Grey | [link] | ACC-01-2300 | 1 | $10.00 | $10.00 | 9 week lead time |

| | | | | | | | | |
|---|---|---|---|---|---|---|---|---|
| | Tamron M118FM16, 16mm, 1/1.8", C mount Lens | Point Grey | [link] | LENS-160 T4C | 1 | $180.00 | $180.0 0 | 9 week lead time |
| | 25D gearmotor | Pololu | [link] | 3252 | 1 | $19.95 | $19.95 | |
| | 25D gearmotor right angle bracket | Pololu | [link] | 2676 | 1 | $7.45 | $7.45 | |
| | M3 x 0.5 mm Thread, 8 mm Long, pack of 100 | McMaster-Carr | [link] | 91292A1 12 | 1 | $4.12 | $4.12 | |
| | Plastic Quick-Turn Tube Coupling, pack of 10 | McMaster-Carr | [link] | 51525K3 18 | 1 | $14.71 | $14.71 | |
| | Alloy Steel Cup-Point Set Screw, M3 x 0.5 mm Thread, 3 mm Long, pack of 100 | McMaster-Carr | [link] | 91390A0 97 | 1 | $4.22 | $4.22 | |
| | Ball Bearing, Double Sealed, for 1/8" Shaft Diameter, 3/8" OD | McMaster-Carr | [link] | 60355K8 51 | 2 | $3.12 | $6.24 | |
| | Steel Triple-Wave Washer, pack of 50 | McMaster-Carr | [link] | 90134A0 29 | 1 | $13.39 | $13.39 | |
| | Chemical-Resistant Shaft Seal | McMaster-Carr | [link] | 13125K6 3 | 1 | $19.69 | $19.69 | |
| | 18-8 Stainless Steel Socket Head Screw, M6 x 1 mm Thread, 16 mm Long, pack of 50 | McMaster-Carr | [link] | 91292A1 35 | 1 | $7.28 | $7.28 | |
| | Heat-Set Inserts for Plastics, M4 x 0.7 mm Thread Size, 4.7 mm Installed Length, pack of 100 | McMaster-Carr | [link] | 94180A3 51 | 1 | $14.25 | $14.25 | |
| | 18-8 Stainless Steel Hex Drive Flanged Rounded Head Screws, M4 x 0.7 mm Thread, 10 mm Long, pack of 10 | McMaster-Carr | [link] | 97654A3 73 | 1 | $6.63 | $6.63 | |
| | Plastic Quick-Turn Tube Coupling Sockets, for 5/32" Barbed Tube ID, Polypropylene, Packs of 10 | McMaster-Carr | [link] | 51525K2 94 | 2 | $5.33 | $10.66 | |
| | Low-Temp Tygon PVC Tubing for | McMaster-Carr | [link] | 8349T17 | 50 | $1.08 | $54.00 | |

| | | | | | | | | |
|---|---|---|---|---|---|---|---|---|
| | Chemicals 5/32" ID, 9/32" OD, 25 ft. Length | | | | | | | |
| | 18-8 Stainless Steel Hex Drive Rounded Head Screw, M4 x 0.7 mm Thread, 20 mm Long, pack of 50 | McMaster-Carr | [link] | 92095A196 | 1 | $6.57 | $6.57 | |
| Software | Simplify3D | Simplify3D | [link] | N/A | 1 | $149.00 | $149.00 | Useful for changing print settings based on layer number |
| Polymer Materials | Dragonskin 10 Very Fast Trial Kit | Smooth-On | [link] | 09301-005-000141 | 4 | $30.10 | $120.40 | |
| | Thi-Vex | Smooth-On | [link] | 09301-014-000001 | 1 | $31.01 | $31.01 | |
| | Silicone Thinner | Smooth-On | [link] | 09301-010-000001 | 1 | $17.90 | $17.90 | |
| | Silc Pig Dye (Red) | Smooth-On | [link] | 09301-015-000003 | 1 | $30.56 | $30.56 | |
| | Silc Pig Dye (Yellow) | Smooth-On | [link] | 09301-015-000004 | 1 | $30.56 | $30.56 | |
| | Silc Pig Dye (Blue) | Smooth-On | [link] | 09301-015-000006 | 1 | $30.56 | $30.56 | |
| | Silc Pig Dye (White) | Smooth-On | [link] | 09301-015-000002 | 1 | $30.56 | $30.56 | |
| | 60 or 50 mL Syringes Polypropylene | Any | | N/A | 1 | $100.00 | $100.00 | |
| Heat Box | Extrusion (McMaster) | McMaster-Carr | [link] | 5537T102 | 6 | $28.82 | $172.92 | |
| | Angle brackets | McMaster-Carr | [link] | 5537T291 | 8 | $11.18 | $89.44 | |
| | Hinges | McMaster-Carr | [link] | 5537T324 | 2 | $14.62 | $29.24 | |
| | Magnetic latches | McMaster-Carr | [link] | 5537T56 | 2 | $9.20 | $18.40 | |
| | Handle | McMaster-Carr | [link] | 5537T54 | 1 | $8.38 | $8.38 | |
| | Paneling (cut to fit your size needs): Plastic or metal sheeting | Any | | N/A | | $100.00 | $100.00 | Any material that is heat-resistant |
| | Led lighting and controller | Amazon | [link] | N/A | 1 | $26.99 | $26.99 | Optional |

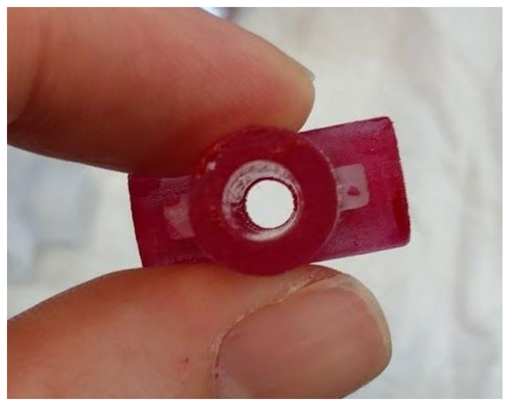
| | | | | | | | | |
|---|---|---|---|---|---|---|---|---|
| | Mixer speed controller | Amazon | [link] | N/A | 1 | $7.95 | $7.95 | Optional (can use power supply) |
| | Temperature controller | Amazon | [link] | N/A | 1 | $35.00 | $35.00 | Can be any model |
| | Heating element | Amazon | [link] | N/A | 1 | $27.99 | $27.99 | Can be any small heater |
| | Mixer power supply | Amazon | [link] | N/A | 1 | $11.99 | $11.99 | Can be any model |
| Custom 3D Prints | Adapter Plate for Taz 6 (PLA or ABS) | In-house | Github | N/A | 2 | | | |
| | Mixing Chamber Twistlock Rotary Seal (SLA/ High res print) | In-house | Github | N/A | 4 | | | |
| | Mixer Bearing Support Twistlock (SLA/ High res print) | In-house | Github | N/A | 4 | | | |
| | Mounting Plate (PLA or ABS) | In-house | Github | N/A | 2 | | | |
| | Camera Support (PLA or ABS) | In-house | Github | N/A | 1 | | | |
| | Misalignment Coupler (Stratasys Vero (outer), Tango (center)) | Send out | Github | N/A | 2 | | | |
| | Spiral Shaft Reamer (Print in steel) | Shapeways | [link] | N/A | 2 | $7.75 | $15.50 | Can try robust plastic also |

Notes about the build:

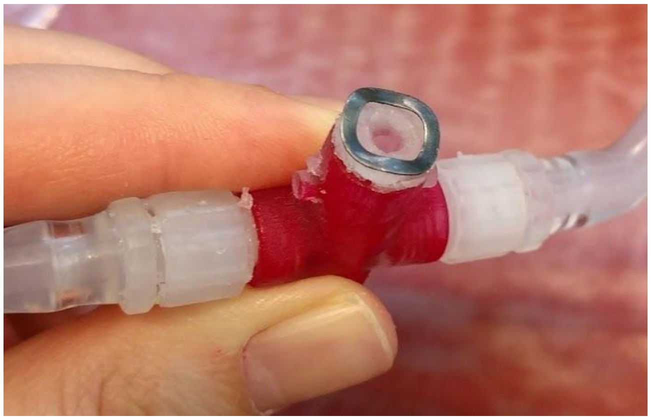
1. Lulzbot Taz 6 3D printer gantry (with thermoplastic extruder PLUGGED IN but unattached to the gantry). When we tried to work without the extruder plugged in it created issues when we tried to print, but this could be solved in the future.
2. PTFE Tape is needed to prevent leakage at the nozzle Luer lock
3. Colored nail polish can be used if lower mixer has too much silicone sticking after curing

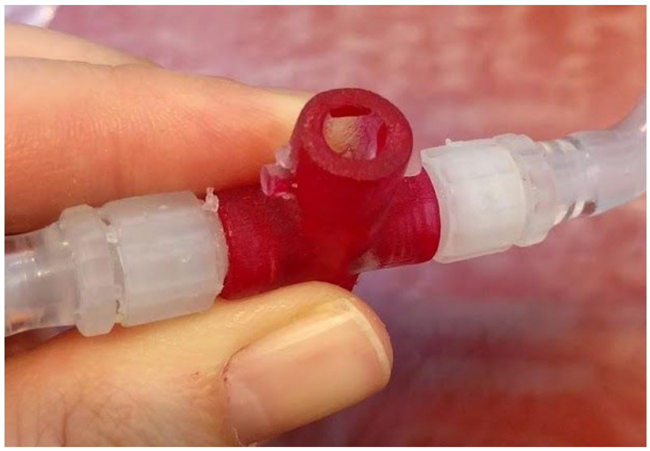
**Printer Startup Procedure:**

1. (Day before printing) Find a lower mixer and colored nail polish. Gently spread nail polish on the interior of the lower mixer in order to smooth out any porous surfaces that could serve as attachment points or nucleation sites for cured silicone clumps (this will also make it easier to clean the lower mixer later). I use colored nail polish so that I can see the coverage on the interior of the lower mixer. When we had 3D printed parts that were not cleaned as well, we didn't have to do this but we did have to deal with the first few prints potentially being inhibited by the extra PolyJet material.

2. Put in new filled syringes of Part A and Part B formulation into the pump and hook them into the proper line (DO NOT  PUT PART A INTO THE PART B LINE OR VICE VERSA – YOU WILL HAVE TO REPLACE ALL TUBING BECAUSE THE MIXING POLYMERS WILL CURE INSIDE OF THEM).
3. Turn on heating element (automatically controlled to 30 C by PID controller) and bed heater (to 30 C set on the printer interface) for 30 minutes before printing in order to warm up all polymers to the proper temperature.
4. After printer is warmed up, unscrew the fluid lines from the lower mixer and turn the lower mixer counter clockwise and pull down gently to remove it, taking care to find the upper silicone component gasket and the wave washer.

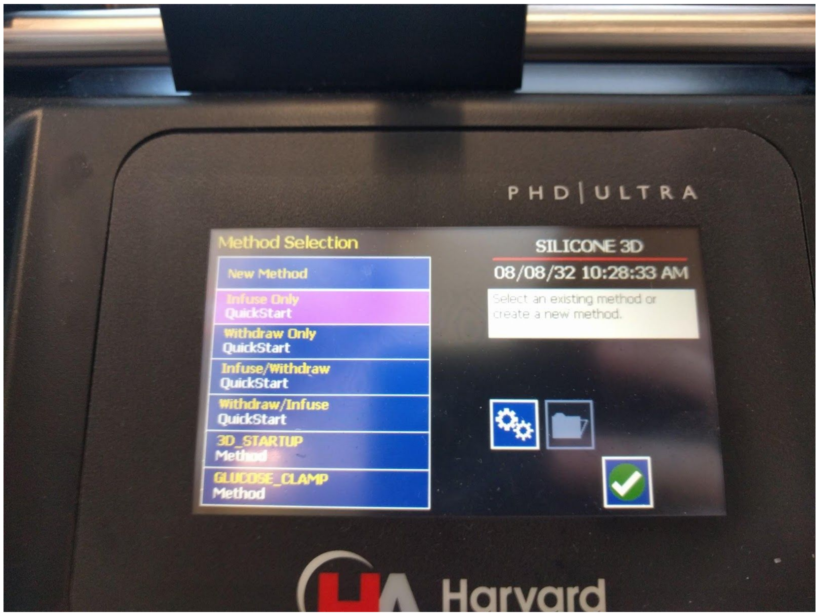


Then unscrew both fluid lines so you just have the mixer.

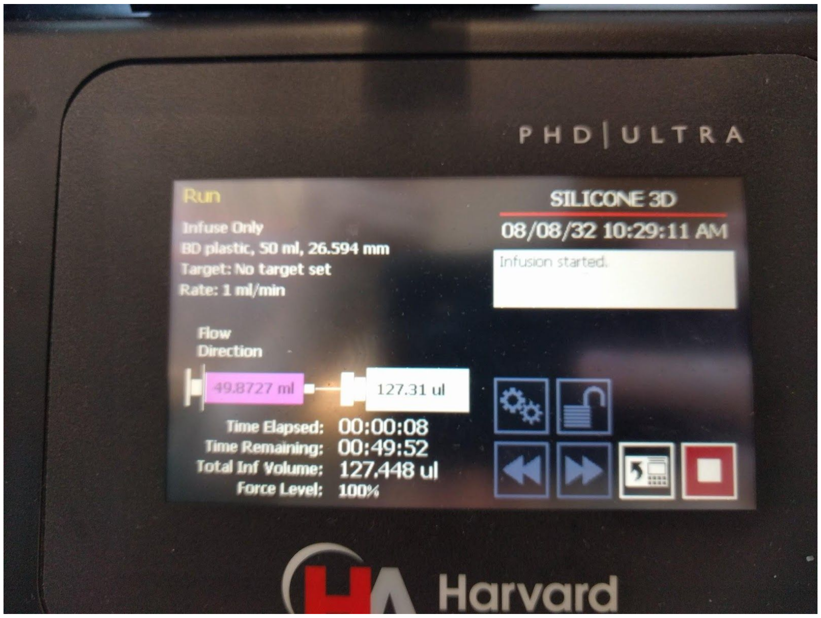
5. Turn the heating element OFF so that you don't overheat the system while working with the enclosure door open. Note: You don't have to turn the bed temperature down. Also, If I leave the door open while the heater is running, the chamber overheats because usually the sensor for the PID temperature controller will never reach 30 C, so it just keeps pumping in heat. Your system will clog if there is too much residual heat when you start printing, so be careful to work with the heater off or close the door as much as you can.
6. Gently squeeze the end of  each fluid line to double check that there is no cured polymer at the tips.
7. Clean out ALL cured polymer from the lower mixer. There should be a cured section of polymer in the top of the mixer that serves as a gasket so that the polymer doesn't leak out of the top during printing (the one mentioned earlier to keep with the wave washer). Keep that portion of cured silicone gasket but cut off any dangling parts of the gasket that might rip off during printing. The correct height of the gasket should be about 1 cm. Check all inlets and outlets for cured polymer by scraping gently with tweezers or a small metal spatula. Remove excess uncured polymer by pushing the wooden part of a medical cotton swab through the inlets and outlets. Wipe as much polymer out of the

lower mixer as you can with the cotton side of the swab. The lower mixer doesn't have to be 100% clean, but free of any cured silicone besides the gasket. Note: The silicone gasket can be pre-formed by mixing equal parts A and B before printing and injecting a few mL of the curing polymer into the top of the lower mixer. By then locking the lower mixer into place with reamer and upper mixer installed, and letting it cure without the reamer spinning, you will get the appropriate shaped gasket to prevent leakage. You may have to cut off excess polymer after you carefully remove the lower mixer. This gasket forms naturally over time if you choose to print without it at first, but there will be leakage problems the first few days until there is enough cured silicone buildup. Once the gasket is formed, it can last for a long time.

8. After the lower mixer is clean, put the gasket back into the top of the lower mixer and the wave washer on top of the lower mixer. Put the lower mixer back into the inset in the upper mixer with the tabs parallel to your line of sight and turn counter-clockwise to lock into place. DO NOT FORCE THE PARTS – THEY WILL BREAK.
9. Turn each fluid line "lefty loosey" 3 times and then tighten each fluid line finger-tight into the side of the mixer. It does not matter which side each line goes on. DO NOT OVERTIGHTEN OR USE PLIERS – THE PARTS WILL BREAK. Note: you do not have to clean the upper mixer out even if there is some silicone between the linear bearings. It does not typically cause problems in this system. You may get material moving into that area if the silicone gasket is not installed into the lower mixer. If the material starts to come out of the top of the upper mixer, you can gently wipe the polymer away with a cotton swab, though there shouldn't be too much of this if your lower gasket is installed properly.
10. Turn on the reamer motor to a level just fast enough to mix the polymer but not so fast as to potentially create local heating inside the mixer. You can determine this value by making your polymers with contrasting color dyes and adjusting the mixer (when the nozzle is installed) so that your colors blend properly and you can't see any of the original colors making stripe patterns in the extruded filament, then marking it on your mixer dial or controller.
11. Turn on the syringe pump to a constant feed of 1 mL/min (INFUSE ONLY) until about 3 mL of polymer comes out of the lower mixer (flushing the system). Note: Sometimes I have been able to print with the reamer not turning and the polymer not being fully mixed. I believe that this comes from parts A and B being so close together when they are squeezed out of the nozzle that the proximity of the polymers alone allows them to cure at the interface. These interfaces are close together and so it can appear that the print is solid and fully cured, but it is likely that there are still uncured portions of the polymer or just irregular curing in general. So, I don't recommend this method and have only printed in this way for fun to see if it would work when the motor on the mixer was broken. An interesting behavior to take into account.


PHD | ULTRA
Method Selection
SILICONE 3D
08/08/32 10:28:33 AM
Select an existing method or create a new method.
New Method
Infuse Only
QuickStart
Withdraw Only
QuickStart
Infuse/Withdraw
QuickStart
Withdraw/Infuse
QuickStart
3D_STARTUP
Method
GLUCOSE_CLAMP
Method
Harvard

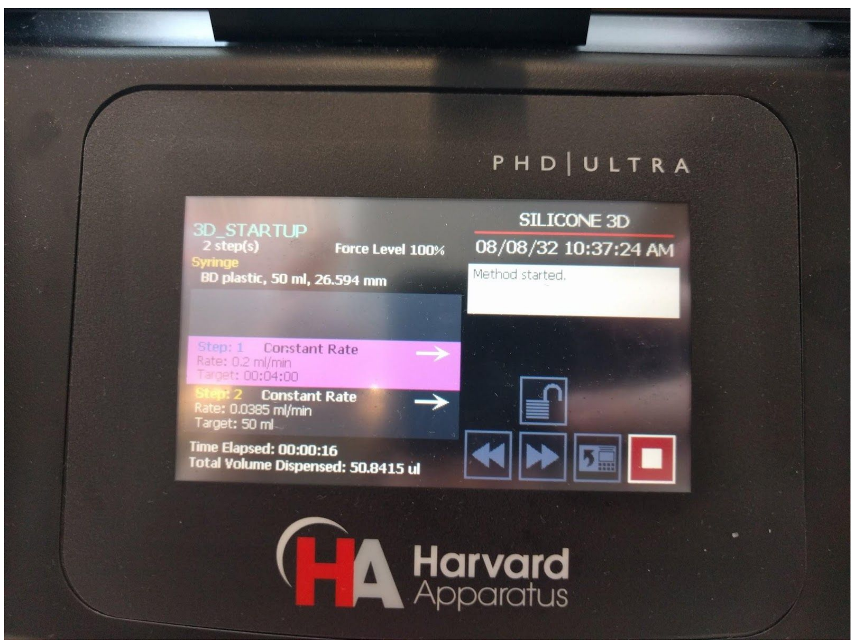

PHD | ULTRA
Quick Start
SILICONE 3D
08/08/32 10:29:01 AM
Enter the settings. Make sure to select the right syringe.
Method Select
Quick Start: Infuse Only
Syringe Select
BD plastic, 50 ml, 26.594 mm
Infuse Rate Select
1 ml/min
Target Volume/Time Select
No target set
Harvard
Apparatus




12. Wrap the precision nozzle with about 10 rotations of PTFE tape to serve as a seal. Note: The current files for the lower mixer have a large tolerance for the precision luer lock nozzles from Production Automation Corporation. This tolerance can be optimized for your system, however just be aware that depending on which 3D printer is printing your parts these tolerances may change. For us it was easier to work with something slightly larger and have to pad the nozzle with PTFE tape than to create an exact fit and risk the nozzle not fitting properly. Also, nozzles like the ones you can get from the chem shop or on Amazon are unlikely to fit in this system due to their imprecise luer lock shape. You can edit the luer lock shape in the SolidWorks source files you can download, but just be aware that the current files are optimized for precision nozzles. I believe that the luer lock fit on this print will work with all of the sizes of nozzles from Production Automation Corporation, but have not tried them yet. Also, be sure to double check that your tip of your nozzle has a cylindrical end and not a conical end! The conical end means that the cylindrical part has broken off and this will change the settings requirements, so if you are tuning your system it will be very frustrating to find out that you tuned with a broken nozzle!

(Left OK, right broken)

13. After about 3 mL of silicone drop out, gently wipe the bottom of the mixer with a paper towel and install the wrapped precision nozzle by turning it about 2 rotations into the luer lock threads. DO NOT OVERTIGHTEN-  THE MIXER WILL BREAK. Note: it may be a design flaw of the lower mixer but I have noticed that when I tighten the lower nozzle too tightly I get clogs sooner. I am guessing that the reamer is slightly too long for the lower mixer and is pressing or holding some of the polymer in the tip for too long, creating clogging. The lower mixer file should probably be edited to have about 2 more mm of length above the luer lock threads to prevent this. I also notice that if the lower mixer is misaligned and the reamer preferentially stays inside one side of the nozzle for too long, that the same clogging issue happens. This has not been tested but is anectodal.
14. If the nozzle is prepared and installed properly, there should be no leaks.
15. Wait 20 seconds at the 1 mL flow rate after your put on the nozzle. Then start the 3D STARTUP procedure on the syringe pump that consists of 2 steps: (1) 0.2 mL/min for 8 minutes and then (2) 0.0385 mL/min for the rest of printing. The procedure is supposed to stop at 50 mL of volume, but you will never reach that limit because there is always less than 50 mL in the syringes after startup. If the syringe pump stalls before you reach Step 2, wait 1 minute with the pump off and then program the Infuse Only setting on the pump to run at 0.0385 mL/min.


16. Turn the PID controlled heater back on and let the 3D STARTUP procedure run until it reaches the second 0.0385 mL/min step. At this point, you will want to wait about 30 minutes for the flow rate to even out. Most clogs will occur during this time if they are going to happen at all. Once the flow rate is running smoothly there is very little chance of clogging unless you have one of the specific situations below:
    a. Note: PRINTER WILL CLOG WHEN:
        i. You do not follow the proper flow rate step-down procedure
        ii. You work with the door open and the heater running while working inside the printer enclosure

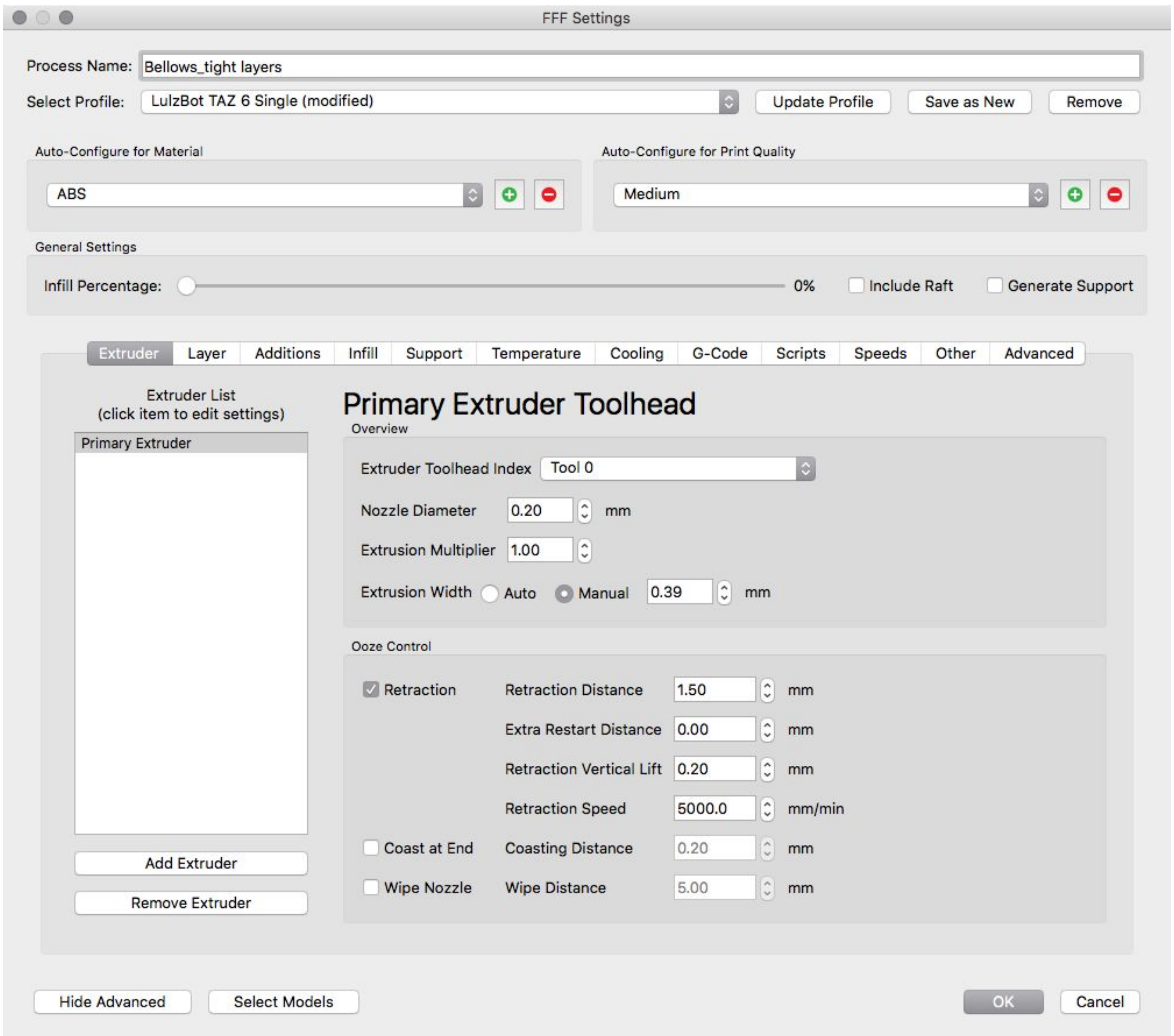
iii. There is excess cured silicone attached to the interior mixer, the ends of the lines, the reamer, or the inside of the nozzle.
iv. You scrape the bottom of the silicone bottle when making your syringes
v. There is loose/scraggly silicone attached to the lower mixer gasket that was not cut off
vi. The flow rate from the syringe pump is stopped for too long
vii. If the nozzle is off-center with respect to the reamer or the reamer is too close to the nozzle outlet
viii. If you scrape the inside of the mixer when it has a nail polish layer and nail polish flakes come loose
ix. If the reamer rotation speed is too high (possible local heating)
x. If your nozzle is digging into previous layers of the print during printing or your layer height is too small (not enough polymer can come out)

17. At this point you should plug in your camera of choice (ours is a Grasshopper3 from FLIR) and look at the flow coming out of the nozzle. It should be coming out smoothly without bumps or irregular filament. If the filament bends slightly to one side, that could mean that there is a small piece of silicone or some other element obstructing the nozzle from the inside, so run your fingers along the nozzle tip and pinch the filament while pulling down to try to pull out anything that might be hanging out. There also might be some extra silicone coating the nozzle tip and so pinching the nozzle tip with your fingers to clean it should resolve the problem. Once the flow looks relatively straight and smooth, you may notice that the filament diameter coming out may be slightly larger than the inner nozzle diameter. This is normal and is called “die swell” . It is just a property of an elastomeric fluid being squeezed through a small opening.
18. Once the silicone is flowing nicely and your camera is running, you can start setting up your model for printing while you wait for the flow to even out.

**Setting Up Models for Printing Using Simplify3D**

1. Open up Simplify3D and press either File>Import Models or press the Import button near the middle left hand side of the screen.
2. The model will be placed on the virtual bed. Press the Center and Arrange button to make sure none of your print is glitching through the bottom of the bed by mistake.
3. Press the letter W or the four-arrow button to move your model to where you would like to print it. This is especially important because sometimes the leveling on one side of the bed can be slightly different than the other side, and since we are printing at high resolutions it is good to calibrate first (see the Taz 6 manual) and then print in the same area of the bed for repeatability. I have had instances where one of the stepper motors moves the gantry a little higher in the z direction on one side than the other, which produces a height error after bed calibration. I have corrected this by hand when the printer was off by turning the lead screw and leveling the horizontal part of the gantry.

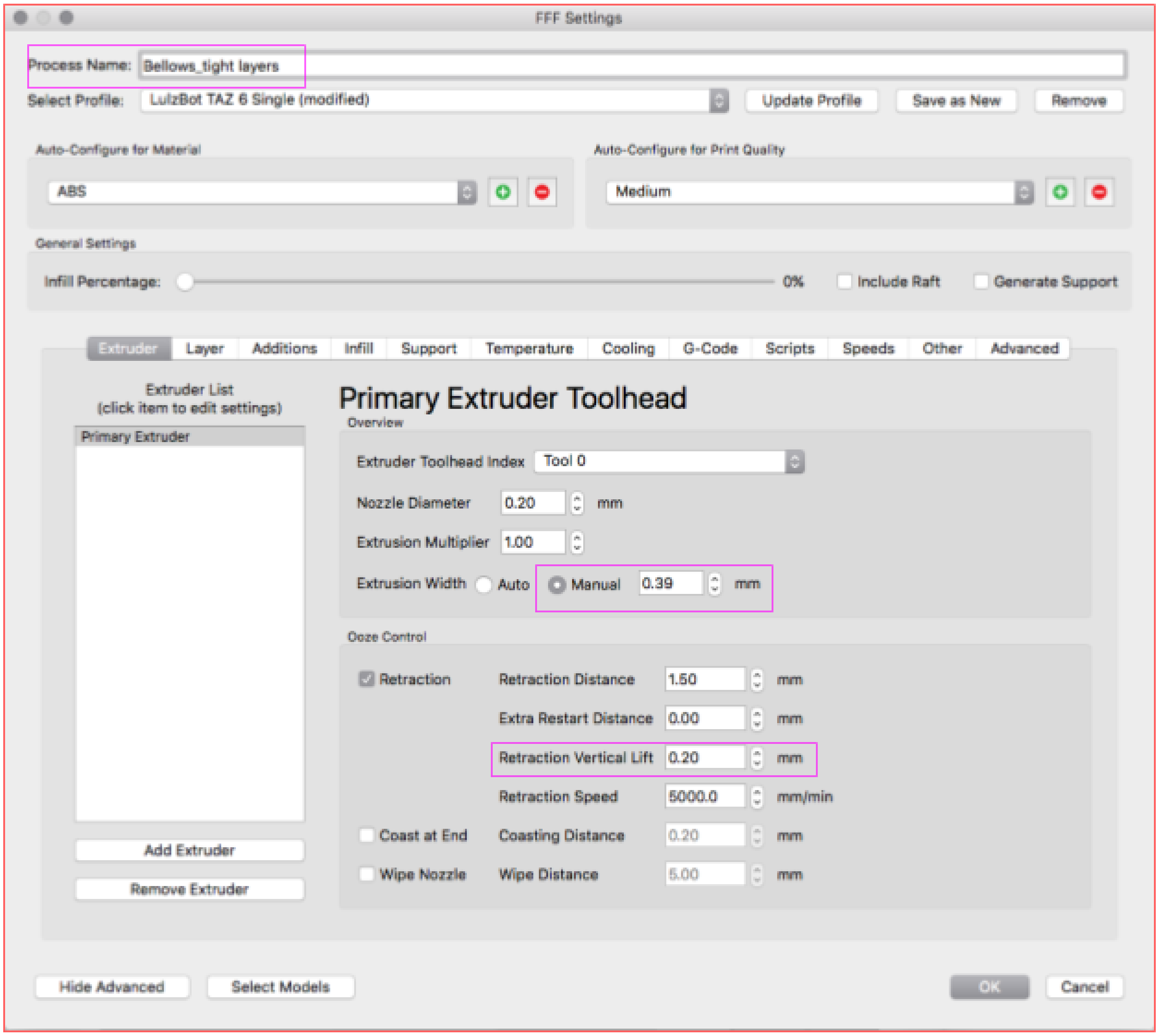
4. If you double click on the model itself, you can see options for changing position via coordinate entry, changing scaling of the model via mm or %, and changing rotation in 3 axes.
5. After you have placed your model in the area you like best, look at the Processes window in the left hand side of the software work area and double-click on an existing process name to edit, or press the Add button to create a new procedure.
6. A new dialog box will open and you will see the following set of tabs:




I will now go through each one and say which settings we use. It looks like a lot but there are only a few important ones we need. If you would like to go into expert mode on this and improve the process please do!

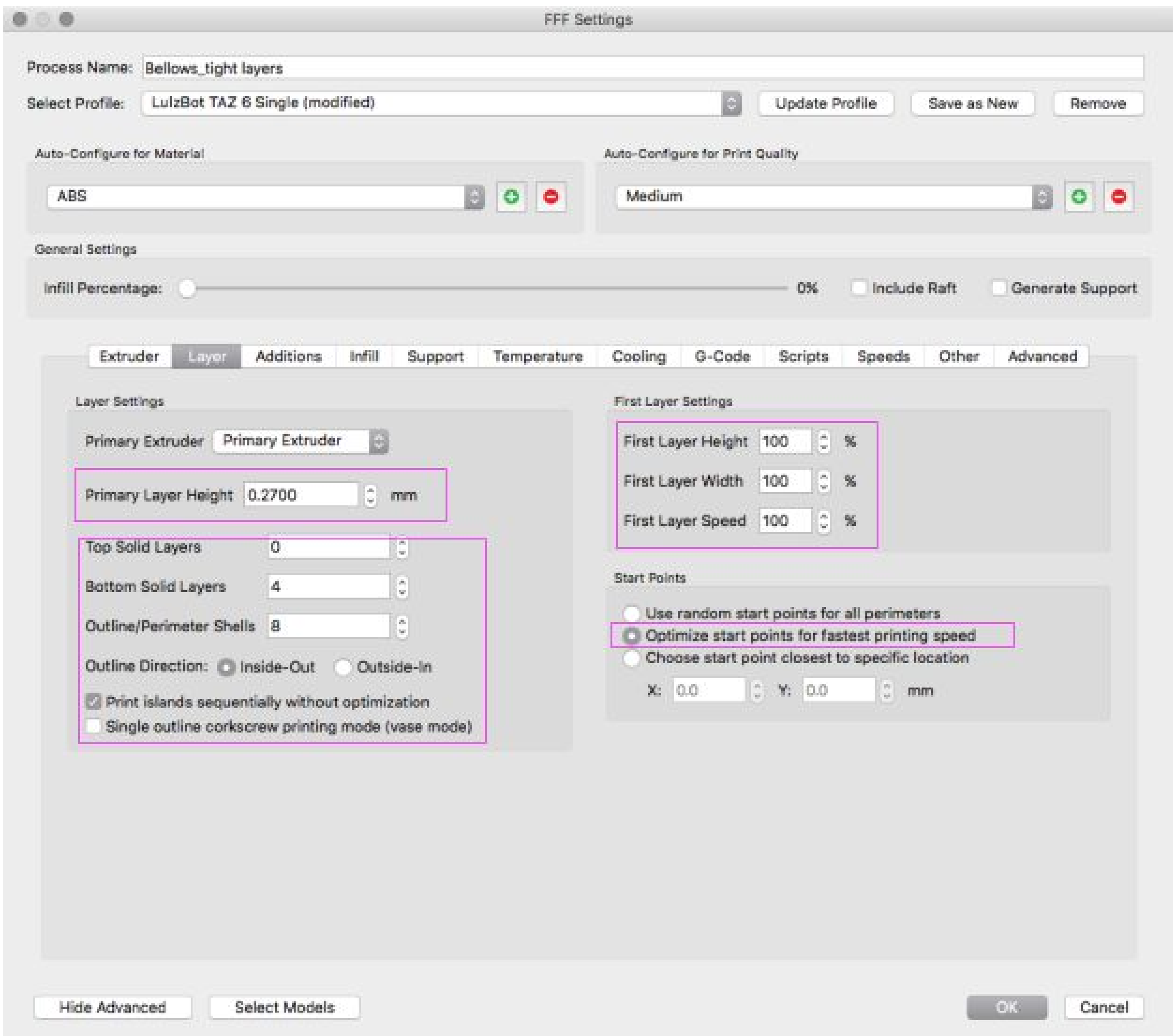
EXTRUDER TAB

First, name your process so you can find it later. Second, set the extrusion width to manual so you can specify a width without calculation by the software. Third, set the retraction vertical lift to a small number so you don't drag through your print if the G-code makes the nozzle move all over the place. The retraction vertical lift is the amount the nozzle will be raised when traveling across separate parts of the layer.



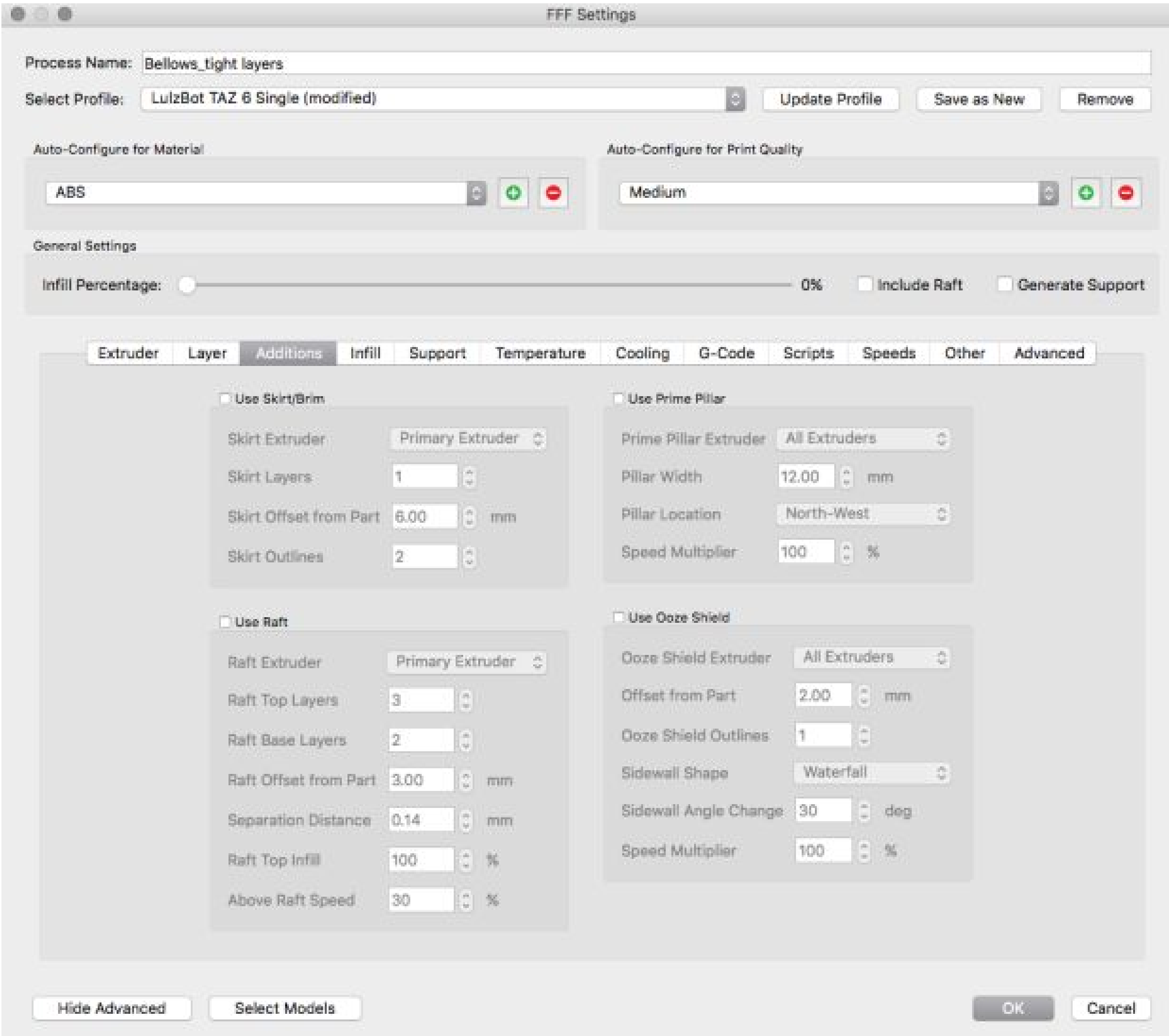

LAYER TAB:

Set the primary layer height to something that makes sense when compared with your silicone line height measurements. Set the number of top and bottom filled layers if that makes sense for your print. The print direction we most often use is "Inside-out" which will print from the middle of the layer outward (Note- this does not mean that if you print an annulus it will print from the inside of the whole print outwards, it means that it will print from the center of the annulus itself AKA the ring between the inner circle and outer circle). This print direction per layer is helpful with difficult geometries because it will print on top of where there is likely to be the most supporting lower layers. When inside-out is chosen, you can print tight layers and by the time you get to the overhanging part of the print (aka the “out”) there will be some previous polymer for it to stick to in the same plane. Printing islands sequentially without optimization I am pretty sure means that if you are printing multiple separate cylinders, for example, that each layer in each cylinder will finish before moving to the next one. I am not sure why you would need it to function the other way. The first layer settings I leave as the standard 100% but you may need to change these if your bed is slightly off-level or there is something weird going on with the zero height of your z axis. I choose to optimize start points for the fastest printing speed so that I don’t get large travel paths across the print.



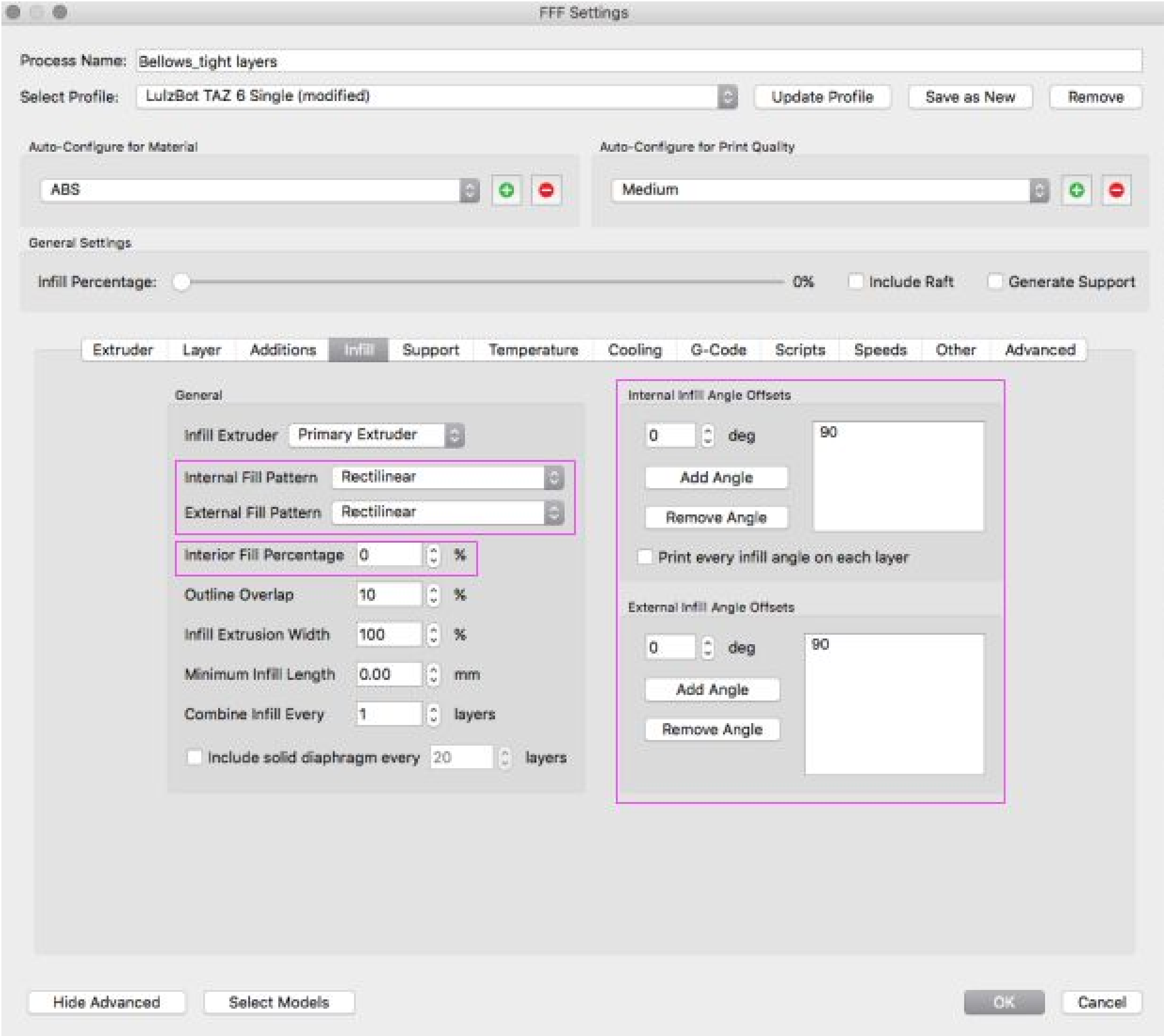
ADDITIONS TAB

I do nothing with this section :), though that does not mean that you shouldn't experiment with how these work for this system.



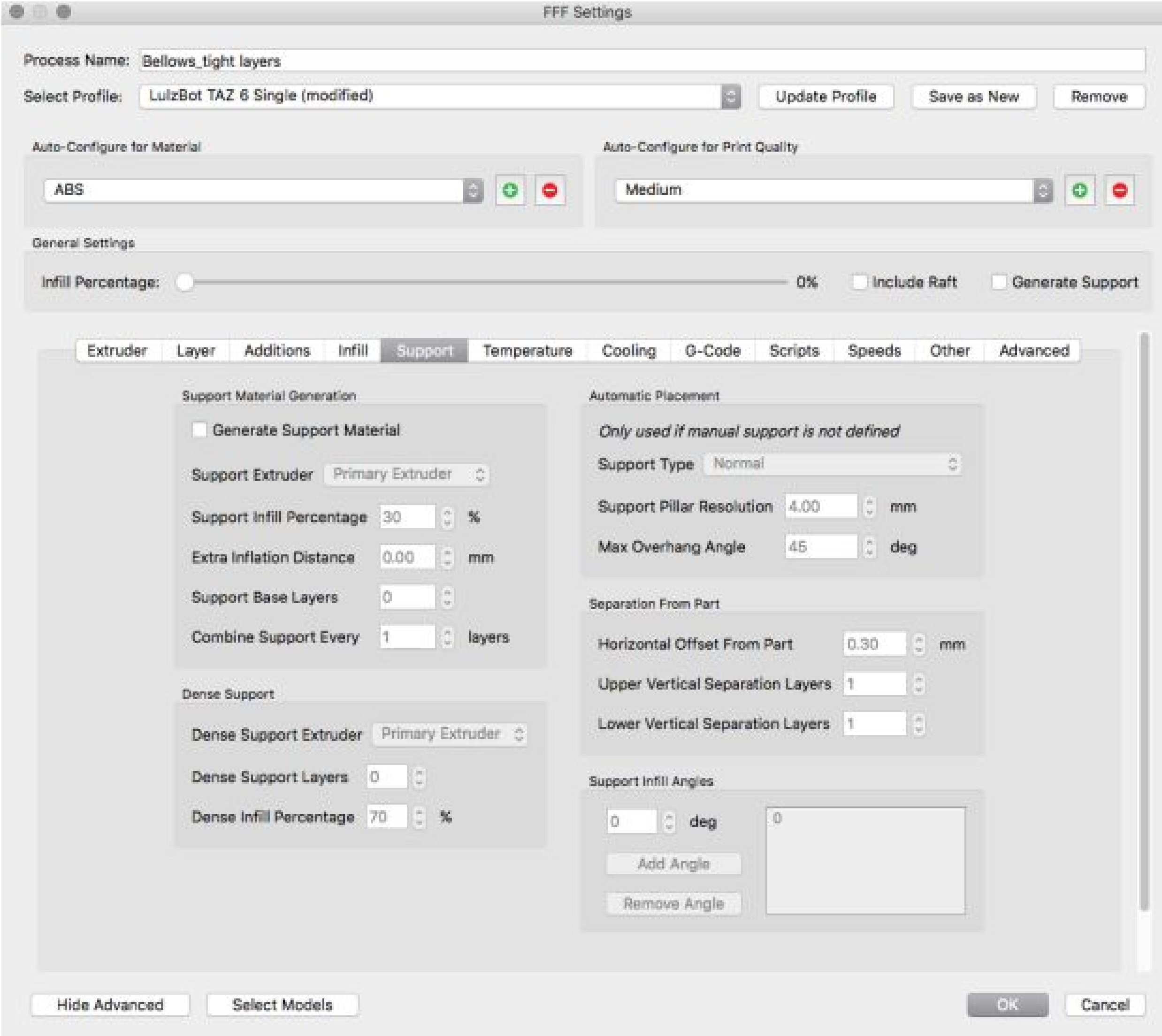

INFILL TAB

Usually I do not use any infill in prints where I can choose to have multiple concentric perimeters (0% interior fill percentage). But, rectilinear infill has worked for me in cases where I am embedding a fabric and need to control how the spanning filament is connecting to the side walls. I find 0 and 90 degree angle fill to be the easiest to control and the most predictable in terms of G-code paths, and try to make sure that the spanning filament length is the shortest it can be so I don't get too much drooping.



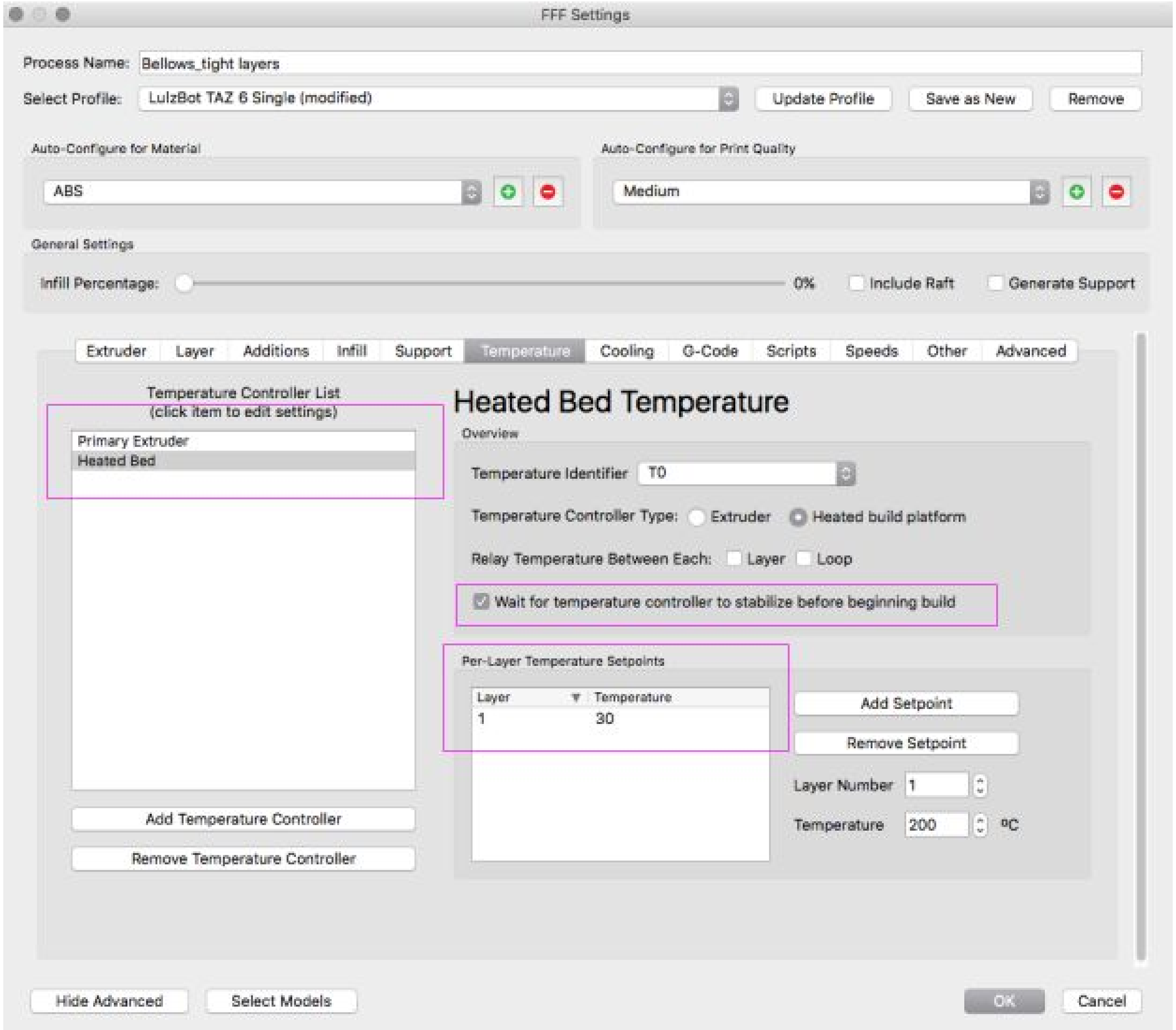

SUPPORT TAB

I normally do not use this, but if you want to play around with removable support settings for soft materials I would be very interested in your results!



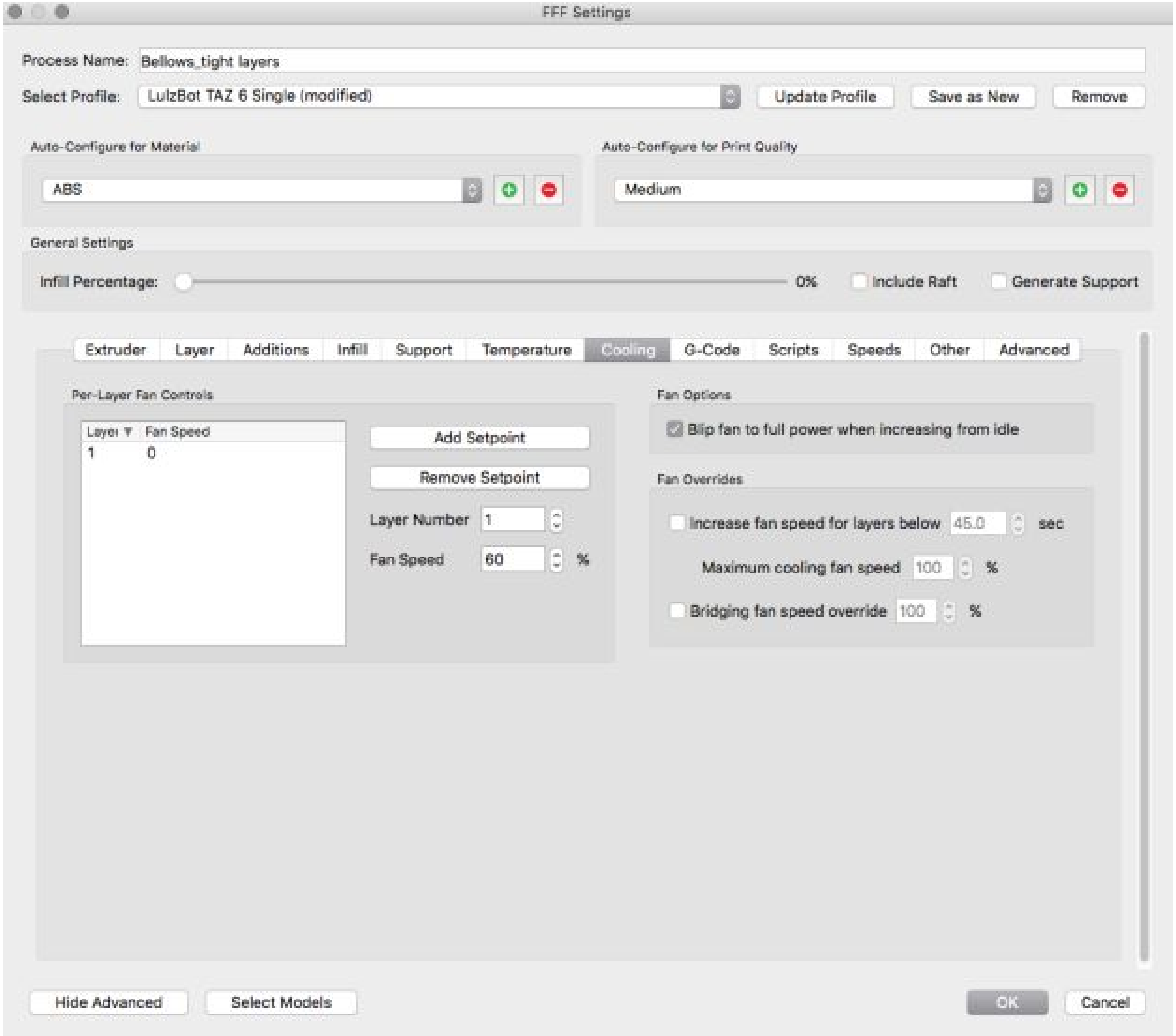

TEMPERATURE TAB

Because the original thermoplastic extruder is still connected, I set the Primary Extruder temperature to 0 so we don't have a hot extruder sitting inside the printer box. I set the Heated Bed temperature to 30 and also wait for the temperature to stabilize before printing.



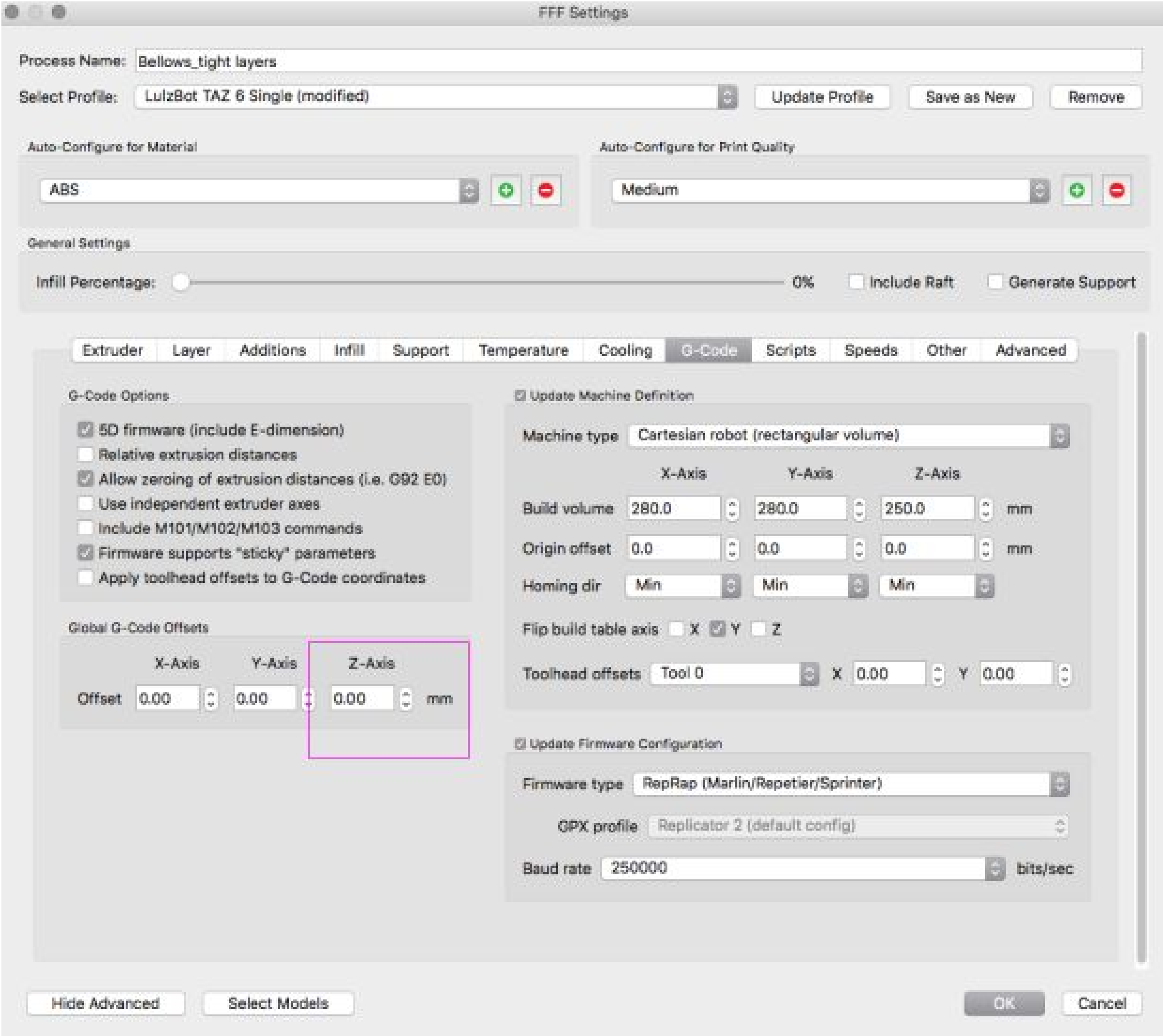

COOLING TAB

I never touch these settings so these are the standard.




G-CODE TAB

Simplify3D has the ability to start printing a model from a specific height (aka if you want to test printing just the middle and up of the model). If you would like to do this, you need to set your G-Code offset to be a negative number in the Z axis. For example, if you would like to start printing the middle of your model at 2 cm, you need to set your Z-Axis global offset at -1.8 cm in order to get the middle of your model to start printing on the build plate where Z = 0. I don't do this very often, but I have had to set the global Z offset to a slightly smaller negative number to make sure it doesn't scrape the build plate. I am not sure why this happens, so using this method will take some experimentation to print at the right height.




SCRIPTS TAB

We had to write our own starting and ending scripts because of our custom extruder size and the need to pull away from the print after completion so that polymer wouldn't be dropping on top of it. There are also opportunities to create other scripts such as layer change but we have not gotten into the complexities of that yet.

Our starting script is:

G26 ; clear probe fail condition

G28 XY ; home X and Y

G1 X-19 Y270 F5000 ; move to good position

G1 Z10 F1500 ; move down quickly

G1 X-19 Y245 F5000 ; move to safe homing position

G28 Z ; home Z

G92 E0 ; zero extruder

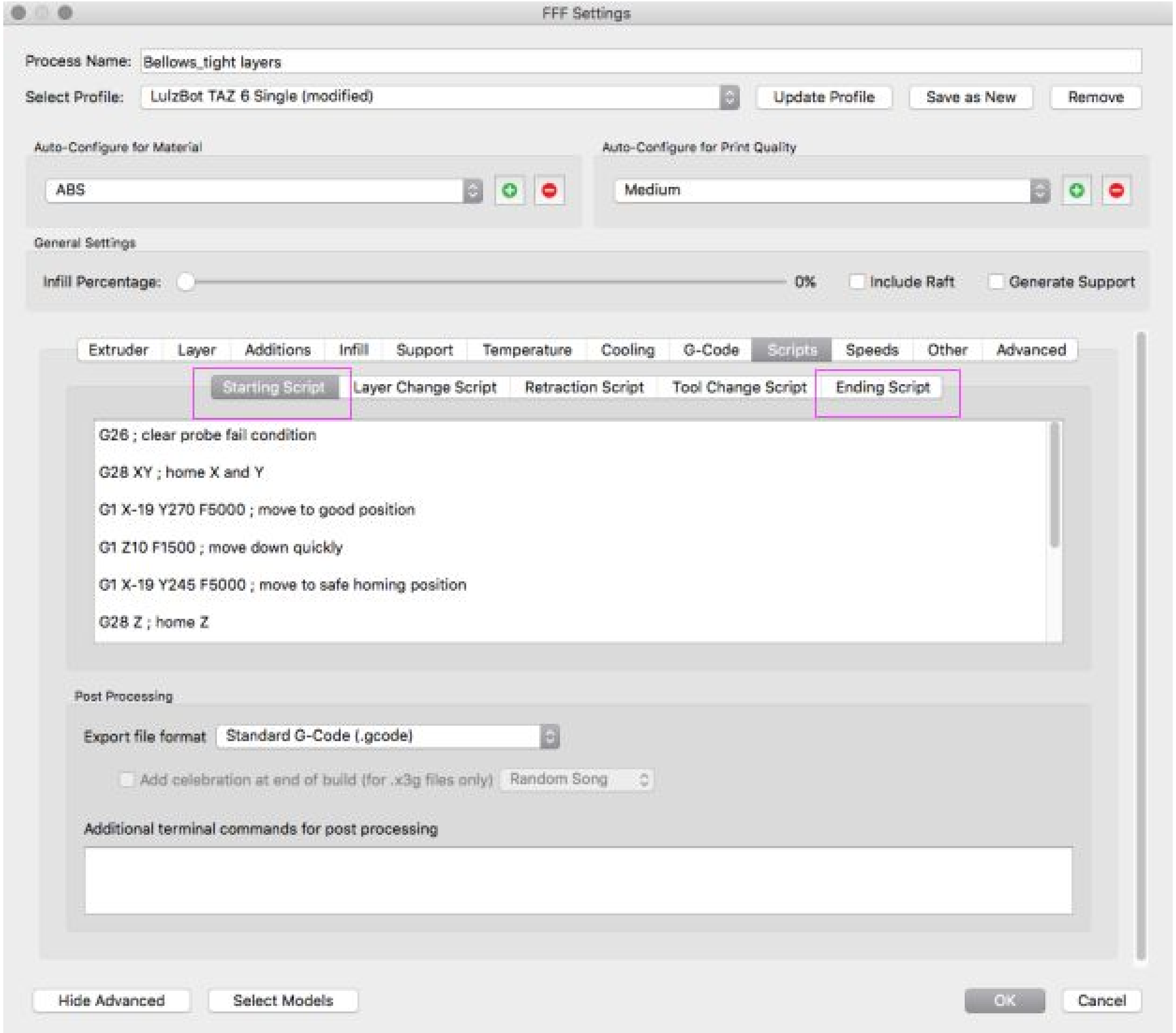
G1 X0 Y0 Z15 F5000 ; get out of the way

M400 ; clear buffer

M117 Silicone Printing...

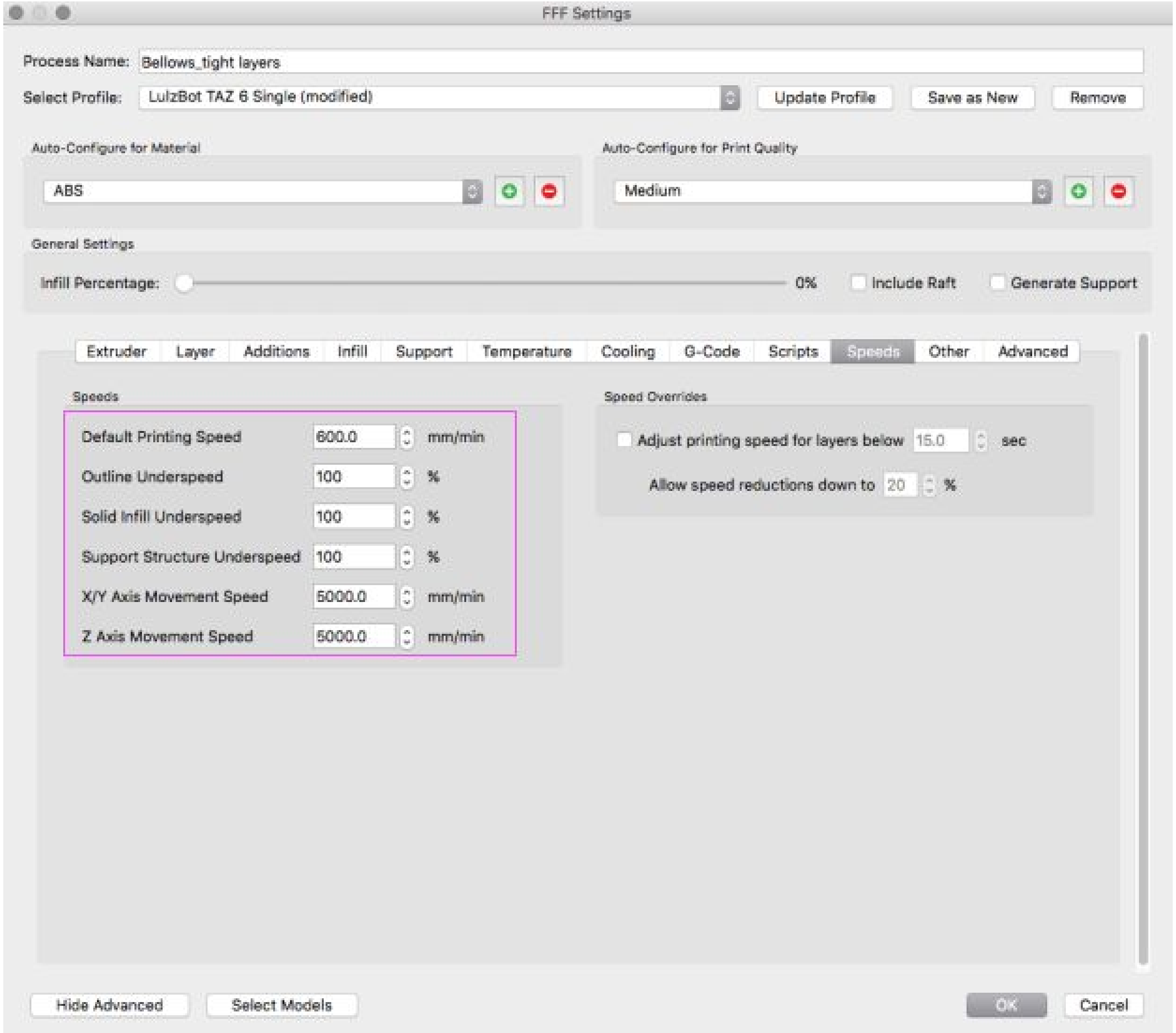
Our ending script is:

G1 X100 Y200 Z100 ; move to good position



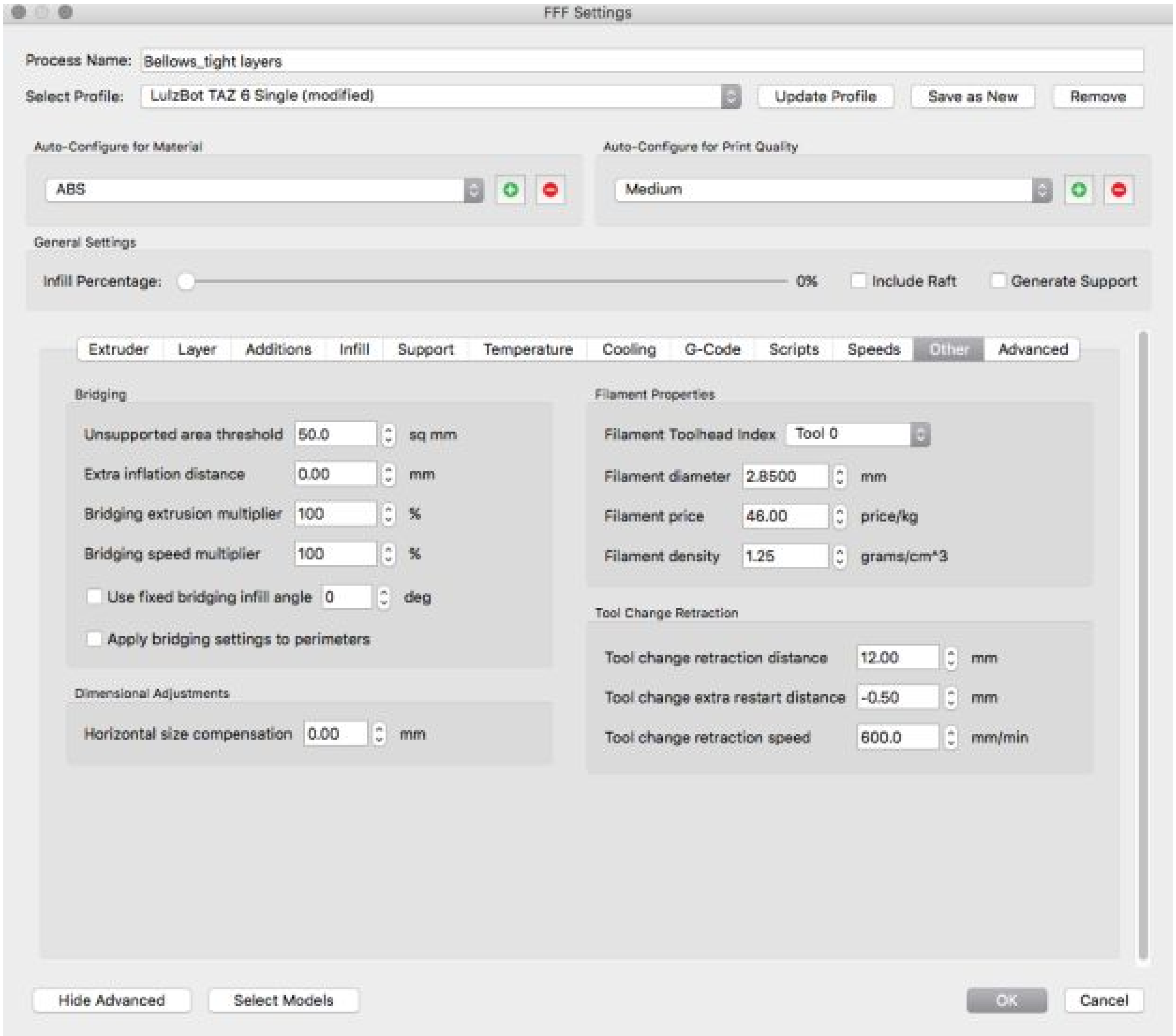

SPEEDS TAB

Our default printing speeds have ranged from 340 - 600 mm/min depending on print type (with simultaneous adjustment of filament width and height). I generally like to keep the infill and outline speeds 100% of that to simplify the print analysis. However, I bump up the X/Y and Z axis movement speed (the travel or non-deposition speed) to a very high number so that I get the maximum travel speed during printing in order to avoid extra material deposition in the wrong place as much as possible.



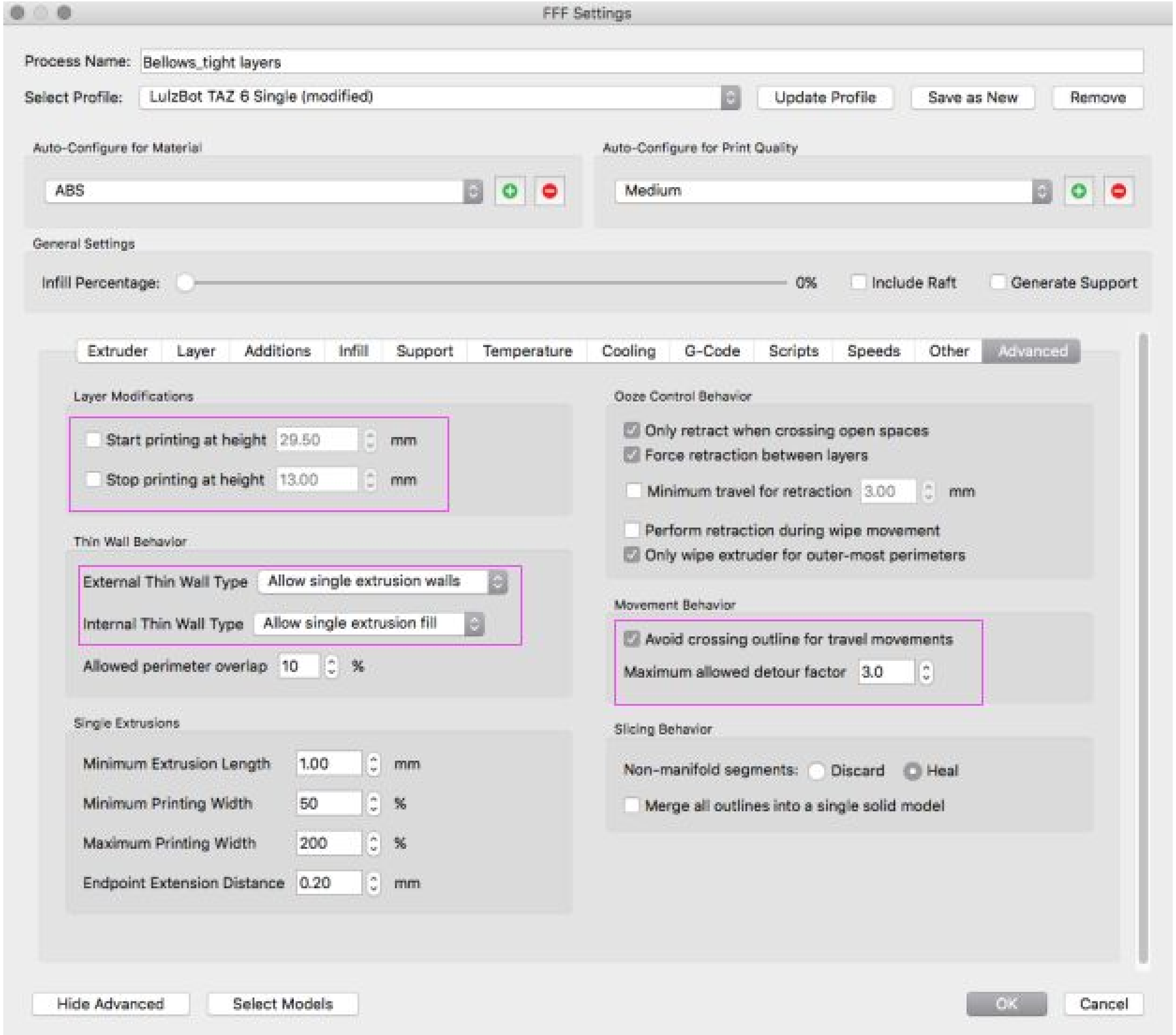

OTHER TAB

I don’t change anything in this section.

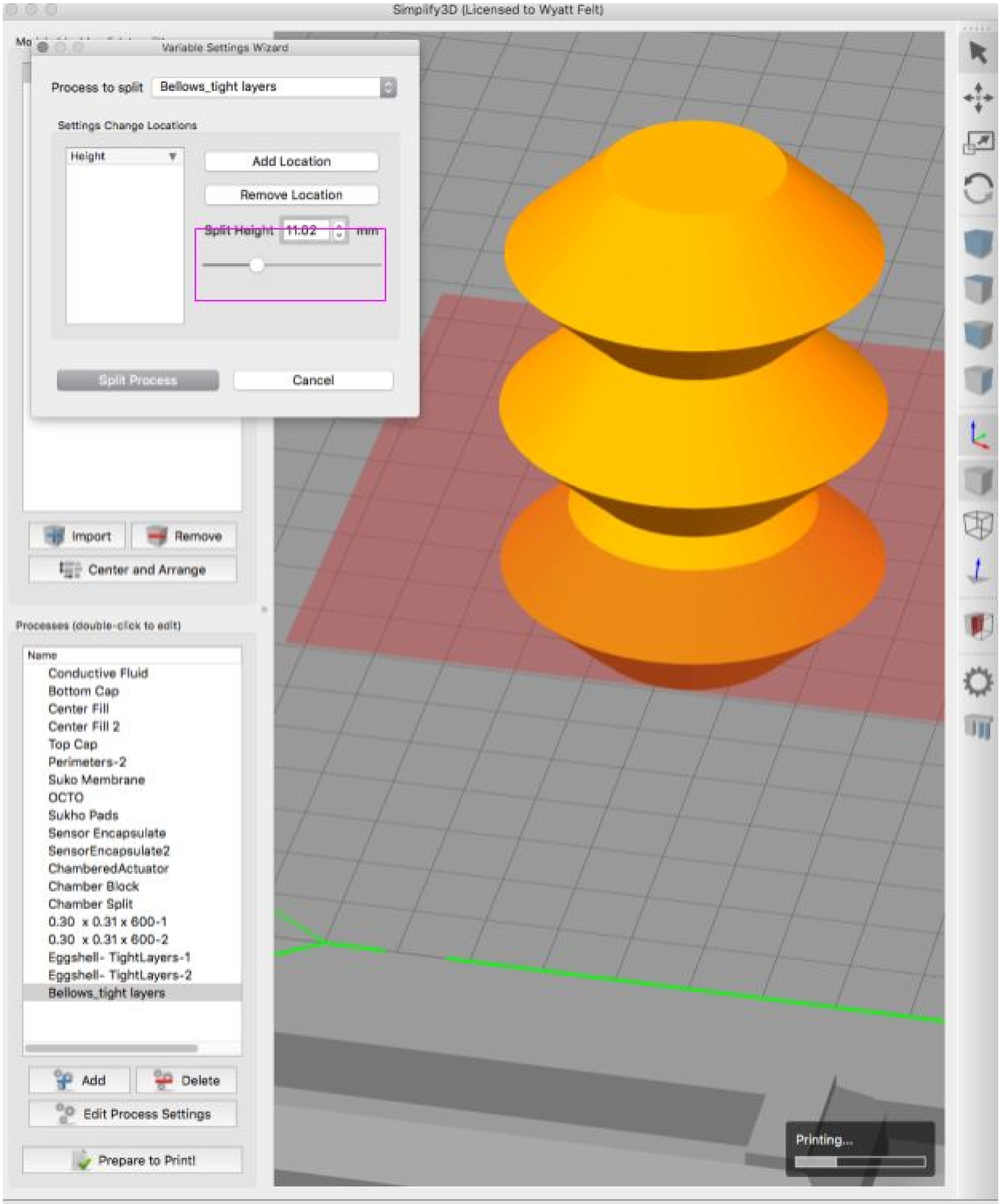


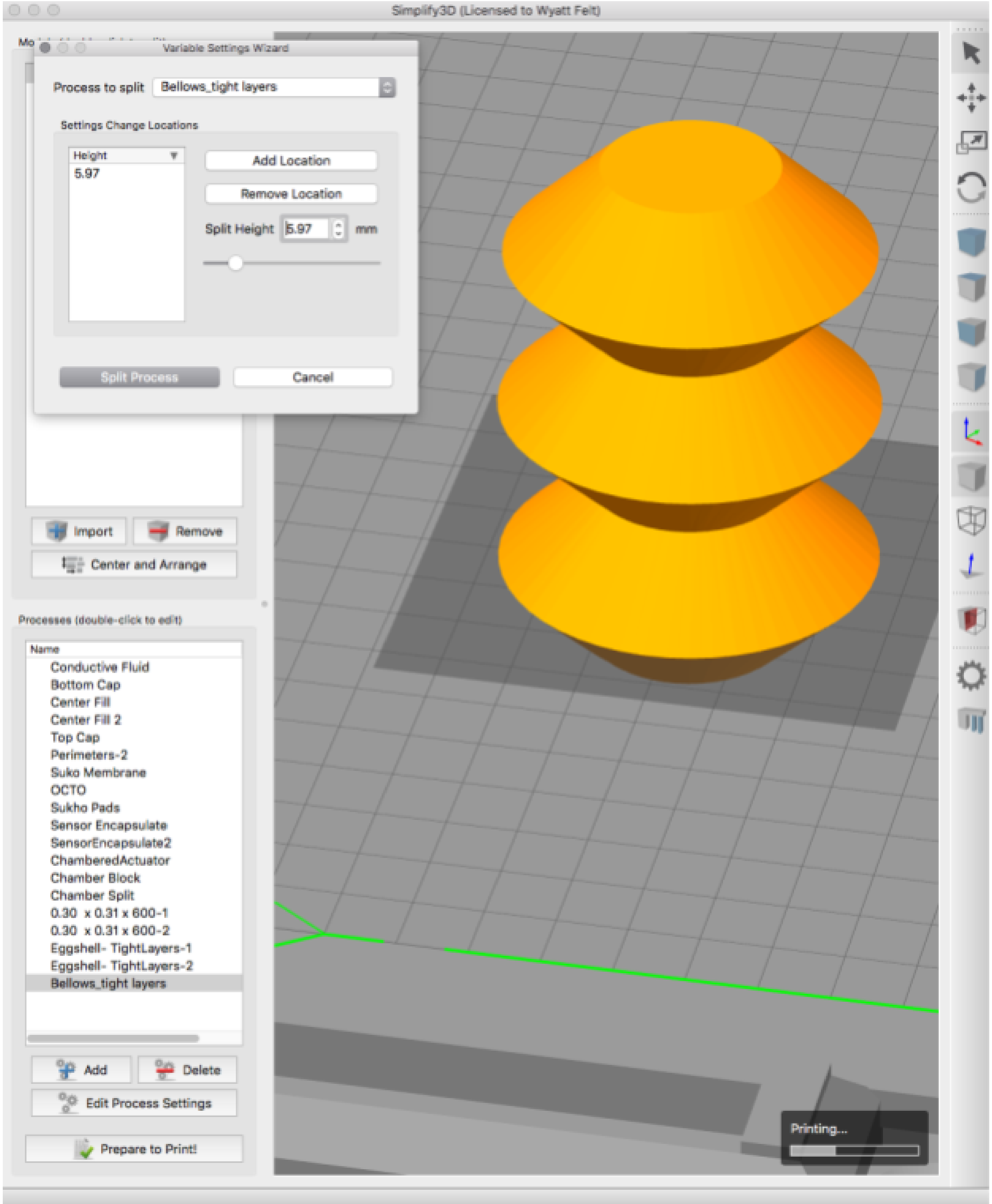

## ADVANCED TAB

This tab allows you to start and stop printing at specific heights in the print so you can use different settings for different parts of the print. Click on the button and set your height to activate this property. Usually, I also allow single extrusion walls and single extrusion fill. The reason for this is that if your layer width is not an integer multiple of your extrusion width, you will get gaps in your layer. Because silicone printing in this system relies heavily on attachment to existing printed structure, having gaps in your print can create collapsing elements that propagate throughout the print and ruin the structure. I also avoid traveling over the outline for aesthetic purposes, but have not experimented with the maximum allowed detour factor to determine how changing the number actually affects the prints.




I will clarify now now how the start and stop print at height works. First, if you have your print on the print bed and you have a starting setting profile you would like to print with selected, click Tools > Variable Settings Wizard. After you do this, you will see a red plane that you can move up and down the print model in the z axis. Move the scroller in the dialog box to move the plane up and down, and then click Add Location to add break points in the print profile setting. After you click Add Location you will see the plane turn grey and then you can add more locations in the print if you need. Once you have split up as much as you like via adding locations, click Split Process. Once you do this, you will notice that your settings profile you had originally clicked is now separated into many versions with a number on the end of the name corresponding to the order in the z-axis. The only thing this is doing is changing the start and stop printing at height settings in the image above. So. you can do this by hand by clicking the Add button in the processed dialog box (lower left hand corner of the main software area) and then changing the name and these z axis limits by hand, but the Wizard makes this process much faster.

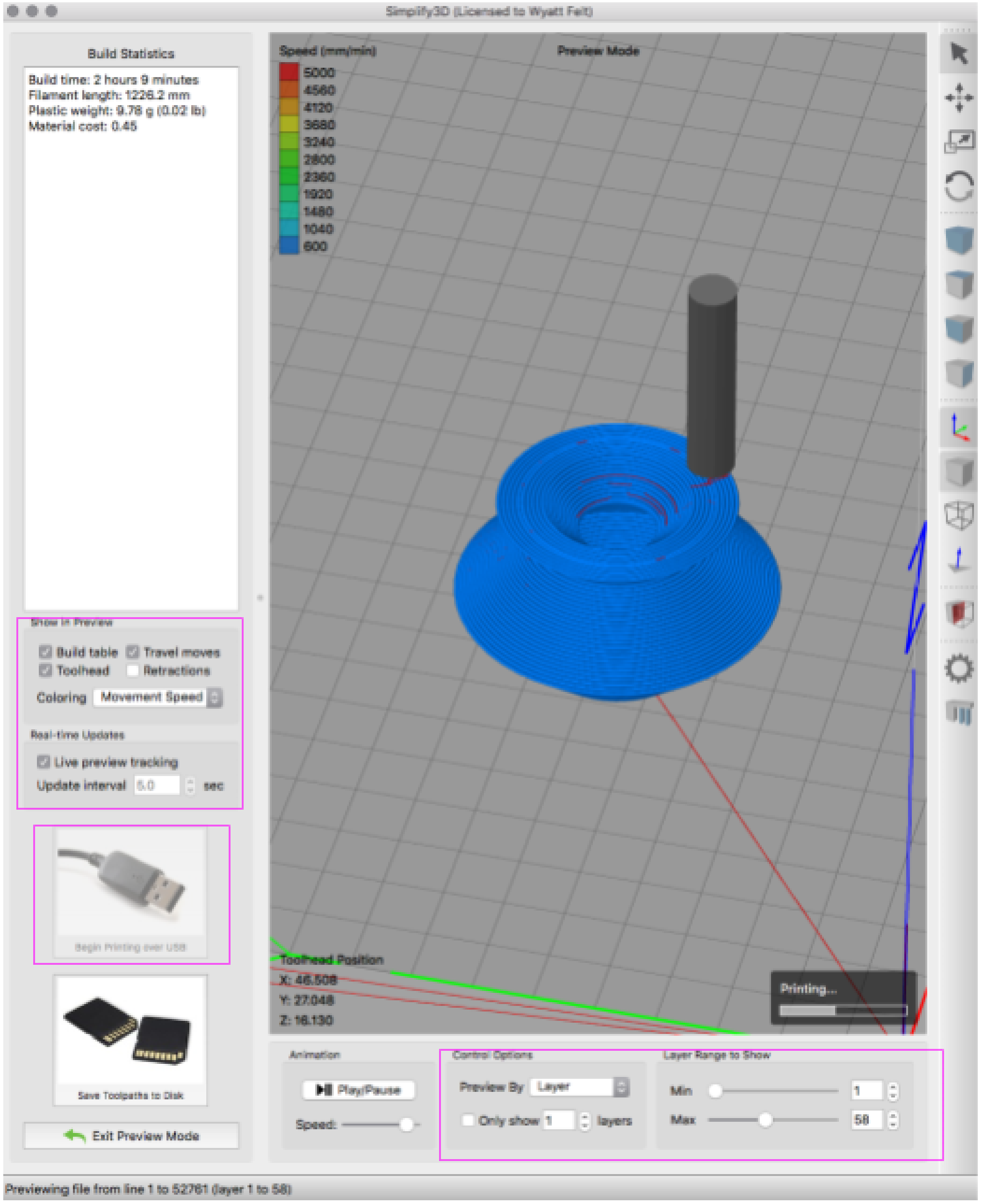

Simplify3D (Licensed to Wyatt Felt)
Variable Settings Wizard
Process to split
Bellows_tight layers
Settings Change Locations
Height
Add Location
Remove Location
Split Height
11.02
mm
Split Process
Cancel
Import
Remove
Center and Arrange
Processes (double-click to edit)
Name
Conductive Fluid
Bottom Cap
Center Fill
Center Fill 2
Top Cap
Perimeters-2
Suko Membrane
OCTO
Sukho Pads
Sensor Encapsulate
SensorEncapsulate2
ChamberedActuator
Chamber Block
Chamber Split
0.30 x 0.31 x 600-1
0.30 x 0.31 x 600-2
Eggshell- TightLayers-1
Eggshell- TightLayers-2
Bellows_tight layers
Add
Delete
Edit Process Settings
Prepare to Print!
Printing...




So, after you have prepared all of your settings and your flow is running smoothly, you are ready to print and experiment!

DURING PRINTING

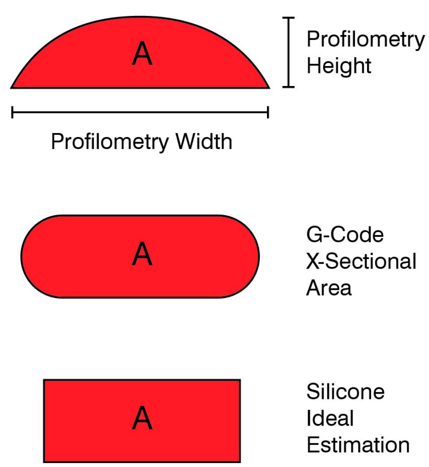
After you have your settings saved, press the Prepare to Print button in the bottom left of the window. It will change the window to a print preview. It is important to analyze your print preview and try to determine how many excess travel paths (pink lines on the image below) and if you can reduce them in either the model design or the print parameters it will make printing much easier. You can click on Live Preview Tracking in order to have a live view of how the print layer will look with time as you are watching. You can preview it layer by layer or line to line in the lower right hand area of the window.



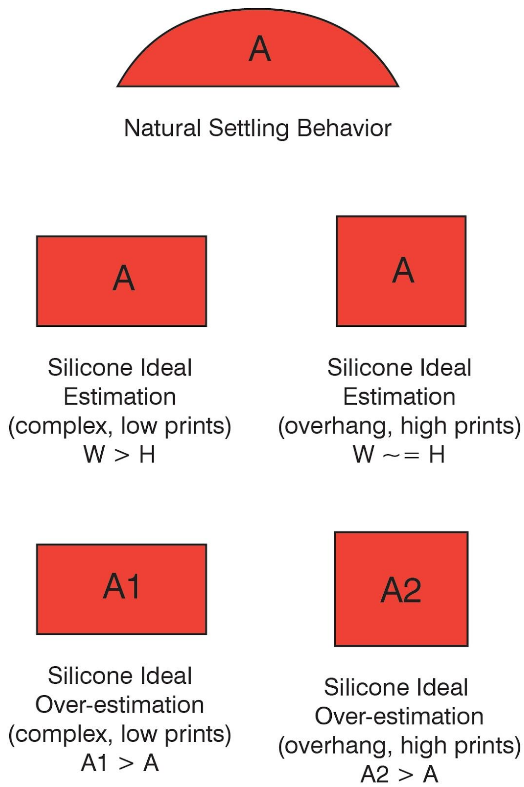

In order to print, you press on the (very literal interpretation of a USB print button) - "Begin Printing over USB" button. After you press this, your extruder will home and the nozzle will press lightly on the z homing button 2 times. To prevent silicone from getting into the button, we cover it with a very thin layer of plastic. If you turn your printer off and then on, the homing process will make the nozzle hover right over the homing button, therefore depositing a lot of silicone. In order to prevent the nozzle from being buried in silicone when homing, I move the plastic slightly when the nozzle is almost touching the button to give it a clean area to press. Because the silicone is continuously extruding, it is helpful to gently wipe the nozzle with the tips of your fingers to remove any excess that has accumulated during homing, but in general as long as I follow the process above for the first homing, I do not have to clean the nozzle for any prints after that. The homing process becomes a series of 2 motions, one where it sits right behind the button in the y direction while it is moving downwards, and then another motion at about Z  = 1 cm where the nozzle moves from just behind the button to right over the homing button in order to home the nozzle for a few seconds. After the nozzle homes on the Z button, it will move over to the lower left corner of the bed and slowly lower itself to the first layer height. During this process, I can tell if the silicone flow is moving properly because as the nozzle moves down to the first layer height I should see about 5-6 loops of polymer fall on the bed. After it reaches this point, it will travel to the first G-Code point in your model and start to print! Hooray!

ANALYZING YOUR PRINT DURING PRINTING, STRATEGIES FOR THE BEST QUALITY

After your print begins, and you have not printed this model before, it is useful to watch how the filament deposits on each successive layer to get an idea of how accurate your settings are representing the way that the filament is going to flow and settle within the print. This is a difficult task, because different prints can require slightly different settings. For example, I will change the settings based on whether or not I do a single-wall versus a multiple layer wall. First, having line profilometry performed on single filaments at the speed you would like to print at is useful for knowing how to specify your filament width and height. Remember, the profilometry data for width can include the tail end of a half-oval where not much material resides. So, you should plan to come in from your profilometry measurements slightly on width to ensure that you are approximating a cross-section of your deposited filament that is closer to a rectangular shape than a half oval. I do this by calculating the cross-sectional area of the profilometry result and then turn that into a simplified ideal rectangle. The G-Code interpretation of the cross sectional area is more like 2 half circles attached to the sides of a rectangle, but because I haven't dug deeply into the way the silicone behaves in terms of cross-sectional area shape, I assume a rectangle for now and work around that. In the future, I would love to do or have some help with a full characterization of how the filaments actually sit within the layer.



It is useful for me to design the filament profile based on how much overhang I am expecting. Interestingly, the silicone can print horizontally at small scales without support if it sticks properly to the side of the previous concentric layer (because it is so light). This is part of the reason that this printing method allows for the printing of models without support. However, there are some things to take into account in terms of how the polymer naturally wants to settle in order to try and tune the printing settings. As a reminder, the process I describe below is only a preliminary method to help achieve specific kinds of prints. It has yet to be optimized or truly characterized for accurate fidelity.




**When you are done printing**

When you are done printing, the extruder head will pull away and sit near the back of the build plate (this is done on purpose via the custom code for the end of a print described in the previous section). Wait for at least 10 minutes for the just-printed model to fully cure before touching or moving it. It should not be tacky at all (tackiness either means not fully cured yet or bad mixing). If the print stays tacky, there was a clog in the line or problem with the ratio of A to B that is preventing the full cure. In my case, I have had prints eventually fully cure even when the ratio was slightly off, but this is not how you want to normally operate.

If you are done printing for the day, all you need to do is to turn all of the equipment off (especially turn off the heater and unplug the camera because it gets hot) and unscrew the nozzle from the bottom of the extruder. The nozzle may be a little tight because there may be cured silicone holding it in place. You may need pliers to remove the nozzle, but you should do this GENTLY so as not to break the 3d printed parts. After this point, do not remove the silicone from either the inside of the nozzle or the inside of the extruder. Once the silicone cures it will be much easier to remove. There may also be parts of the silicone that will never cure because they are not mixed inside the extruder. This is normal.

**GOOD LUCK! Remember that this is a research machine and has not been fully optimized for you to press print and walk away. You will need to watch and analyze your prints (at first) until you get things running smoothly without error. Nevertheless, it is a fun and gratifying experience to make your own printed objects, and I look forward to others incorporating their own materials and ideas into this machine! If you have updates and ideas that you are willing to share (aka those which you are not going to necessarily publish) please add to the Github repository of information so others (including myself) can benefit from your experience.**