Simplify3D Menu Information and other tips

Before setting off with the tips there are some items regarding filament control to be considered by Baron Williams from the S3D forum:

Extrusion Width - Auto: All it does it take your nozzle setting and multiply that by 1.2. Nothing magic here. For best print results, it is often recommended that your extrusion width be at least 1.2 times your nozzle width. This sets the width of the plastic printed for your model to be 0.48 mm for a 0.40 mm nozzle, sets it at 0.60 mm for a 0.50 mm nozzle, at 1.20 mm for a 1.00 mm nozzle, etc. So that's what this setting does. It doesn't affect anything other than the width of the plastic lines the printer extrudes.

Extrusion Width - Manual: This is the settings I always use. I never use Auto. I almost always use either 0.50 mm or 1.00 mm, depending on what I'm printing. I like numbers that easily divide by 1 or 0.5 and so I design my models that way. You get best results normally when your model's dimensions line up well with your printer's dimensions. For example, if I design a wall that's exactly 1 mm thick, this will print well with an extrusion with of 0.5 mm, but won't print well with an extrusion width of 0.48 mm, because you can't evenly divide 1 by 0.48, there will be a remainder, and that remainder will be difficult to print.

Extrusion Width is just like **Layer Height**, only it deals with width and not height. **Extrusion Width** only tells the printer how wide to make lines. It doesn't affect anything else except print speed. A wider line means more plastic per mm, so you'll need to lower your print speed. For example, if you double your width, you need to half your print speed, because twice as much plastic is printed per mm. The **Extrusion Width** should always be at least 1.2 times larger value than **Layer Height**. At slicing time Simplify3D will warn you if it's not at least 1.2 times larger. The reason for this is that the plastic will adhere better if lines are printed so that they are wider than they are tall. If the width and height are equal, then the amount of space that touches between each layer is too small, and results in very poor prints. By making the width bigger than the height, the lines are smashed together more, so they adhere better.

Extrusion Width should not be confused with **Extrusion Multiplier**, **Nozzle Diameter**, or **Filament Diameter**, because those affect how much plastic to extrude, and shouldn't be used to change the width of your print line. Any adjustments made to those can dramatically screw up your print. Only adjust those when you're having problems with prints not extruding properly. For example, if too much plastic is extruded and walls are overlapping, or not enough plastic is extruded and walls have gaps between them, that's when you start playing with these other adjustments. Don't use these to make your print lines thicker. That's not what they are for. They control how much plastic is printed per mm according to your **Extrusion Width** and **Layer Height** settings, not how wide or high to print your line. Only **Extrusion Width** tells the software how wide to print your lines. Only **Layer Height** settings tell the software how high to print each line.

It's true that adjustments made to **Extrusion Multiplier**, **Nozzle Diameter**, and **Filament Diameter** end up producing the same results, either more or less plastic being extruded per mm. They are different ways to tell the software how to adjust how much plastic to extrude, that's all. These affect the width of the line being printed, but DO NOT affect the horizontal spacing between the centre of each extruded line. If you make a line very thick using these settings, then lines will overlap each other horizontally because the space between the centre of each line doesn't increase when you adjust these, only the width increases, causing overlaps if too thick. **Extrusion Width** and **Layer Height** don't work like that. **Extrusion Width** tells

the printer to print lines with X amount of distance between the centre of each line horizontally, and **Layer Height** tells the printer to print lines with X amount of distance between the centre of each line vertically. Width and height are automatically adjusted so lines touch each other, normally. If they don't touch properly then you need to adjust **Extrusion Multiplier**, **Nozzle Diameter**, and **Filament Diameter**.

To clarify that further, think of **Extrusion Width** and **Layer Height** as the print RESOLUTION. If you want higher resolution, you use smaller values for these. If you want lower resolution, you use higher values for these. Don't use these to fix extrusion issues. They are for changing print resolution only.

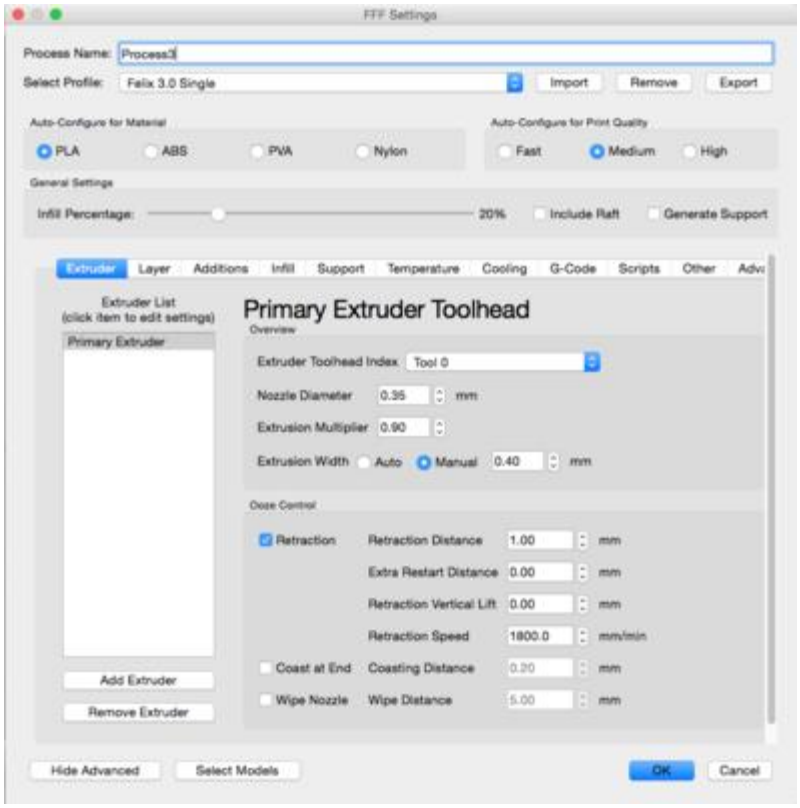
If lines are printed so thin that there's a gap when their shouldn't be, adjusting either **Extrusion Width** or **Layer Height** will NOT fix the problem. For example, if you have a 20% gap between walls that should be solid without a gap, and you make your **Extrusion Width** larger, your gap will still be 20%, but will now be a larger gap because the lines are now larger and further apart from each other (the print resolution is lower). You didn't get rid of the gap, you just enlarged the lines, and the line spacing, so the 20% gap appears wider, but is still there at 20%.

Likewise, If lines are printed too thick and are overlapping each other, adjusting either **Extrusion Width** or **Layer Height** will NOT fix the problem. For example, if you have a 20% overlap between walls, and you make your **Extrusion Width** smaller, your overlap will still be 20%, but will now be a smaller overlap because your lines are now smaller and closer together (the print resolution is higher). You didn't get rid of the overlap, you just shrunk the lines, and the line spacing, so the 20% overlap appears thinner, but is still there at 20%.

You adjust **Extrusion Multiplier**, **Nozzle Diameter**, and **Filament Diameter** when you need to address gaps or overlaps in printed lines. For ease of adjustment, **Nozzle Diameter**, and **Filament Diameter** should match your real nozzle and real filament diameter. Only change **Nozzle Diameter** to match the actual nozzle diameter in use. Don't use other values. Likewise, only change **Filament Diameter** to match the actual filament diameter in use. In most cases you'll keep the value of **Nozzle Diameter** the same unless you change to a different nozzle size. It's possible, that over time, your nozzle diameter can increase from use. That's the only time you might want to change the **Nozzle Diameter** value if you haven't changed nozzles. **Filament Diameter** can change from spool to spool. Some spools, although stated at 1.75mm, can be thicker or thinner than stated. In such cases, changing the **Filament Diameter** to match the current spool's real filament diameter is necessary to get good print results. This setting should NEVER be off from the actual filament diameter in use. If it is off, it will throw off extruder calculations. **Extrusion Multiplier** is the best setting to alter if the other two already match what's in use but prints still have either lines that overlap or lines with gaps.

Put another way, **Extrusion Width** and **Layer Height** adjust print resolution. **Extrusion Multiplier**, **Nozzle Diameter**, and **Filament Diameter** adjust print extrusion ratios according to print resolution, but have no effect on print resolution. The **Extrusion Multiplier** setting in particular is similar to an inkjet paper printer's ink level setting. On an inkjet printer if you set the ink level higher, the print darkens and the dots in the print start to bleed into each other because more ink is used per dot; if set lower, the print lightens because less ink is used per dot. The ink level doesn't effect the print resolution though. The **Extrusion Multiplier** is used like that, for fine tuning how much plastic is used per line, while not affecting print resolution per line.

Extruder tab



Overview Section

Extruder Tool head Index: This is the Identifier your associated with your extruder head that is decided in your firmware. The general convention is your first tool will be T0, if you have multiple extruders general convention also is that T0 will be the right extruder and T1 will be the left extruder.

Nozzle Diameter: This is the diameter of the tip of your nozzle where plastic is extruded from. Traditional values range from .3 mm all the way up to 1 mm for some of the more customized printers. A smaller nozzle is great for thin walls, for printing small miniatures and other things that require fine precise details. Large 1 mm nozzles are great for rapid prototyping, creating large parts quickly and efficiently by pushing out lots of plastic.

Extrusion Multiplier: The extrusion multiplier will multiply the amount of filament extruded for your entire print. This includes skirts, rafts, supports, perimeter and infill extrudes. The default values in the software are .90 for PLA and 1.0 for ABS. If you notice that your Top solid layers are not as filled-in as you'd like them to be, I would recommend increasing the # of top solid layers, and if you still think there's an issue then this is the setting I'd recommend changing.

Extrusion Width: The slicing engine doesn't use your nozzle diameter, but instead uses your Extrusion Width setting. For most cases, keeping this setting on Auto (1.2x your nozzle diameter) is the best route, however for certain cases where you have extremely thin nozzles, you may find it's best to go to Manual and play with lowering the Extrusion Width setting.

Ooze Control

Retraction Distance: How much filament will be retracted when the software does a retract. For standard Direct Drive, usually .5 to 2.5 mm will work well. For Bowden extruders, 5 to 8.5 mm seems to work well.

Extra Restart Distance: When the software does a prime (after retracting), it will prime the same amount as retracted. However, you can modify this by placing an Extra Restart Distance, if you notice that your prints have minor blobs at the starting points, you may find that placing a small negative Extra Restart Distance can help, such as -.2 mm.

Retraction Vertical Lift: The nozzle will move up in the Z-axis whenever you do a retract. This is particularly helpful with Delta printers and prints that have a lot of islands, since the delta printers can be prone to running into parts when doing rapids.

Retraction Speed: The speed at which the filament will be retracted/primed. I think 1800 mm/min or 30 mm/sec is about as slow as I'd want to go for standard retracts, that going quicker will help for print quality in most cases.

Coasting Distance: Setting a coasting value can be good if you want to empty out your nozzle before doing a retract. Let us say you're printing a single line that is 100 mm long. If you set a coasting distance of 5 mm, the extruder will be pushing out plastic for the first 95 mm of traveling, but then stop extruding and while it will move over the rest of the line, it will not actually extrude anything for the last 5 mm. It will instead depend on the filaments momentum, and gravity to let the rest of the filament ooze out and fill in the region for the last 5 mm of the line.

Wipe Distance: Wiping happens after you've printed your outer most outline (Advanced tab, there's an option that's usually enabled for this). When you print your outer most outline, when it's time to do a retract, there's a good chance that not all of the filament in the extruder nozzle head is going to rise up, that a blob of molten liquid plastic is going to be at the tip of the nozzle still. For this reason, instead of rapidly moving to the next spot right away, you can use the Wipe function, which will wipe over the perimeter and let that filament ooze out, similar to Coasting. However, Coasting is not extruding over areas that need filament (you risk under-extrusion voids if coasting value is too high), Wiping is not extruding over areas that have already been printed on(much lower risk of part quality being negatively affected).

There's a lot of terms in there, but here's the order they go in for an outer perimeter:

1) Coasting starts *'m'* mm before outer most perimeter is done
2) Finishing printing outer most perimeter
3) Retract *'n'* mm at *'s'* Retraction Speed
4) Wipe *'w'* mm over the outer most perimeter
5) Vertical lift up *'u'* mm
6) Rapid move at Rapid Move Speed *(see Other Tab)*
7) Vertical lift down *'d'* mm
8) Prime filament. Amount primed is *Retraction Distance + Extra Restart Distance*

Layer Tab

Primary Extruder: Which extruder you'd like to have the Perimeters printed with.

Primary Layer Height: How thick you want each layer to be on the Z-axis. Smaller means finer resolution and better print quality, but also means many more layers you'll have to print, which can dramatically increase print time.

Most profiles use the following:
Fast prints: 0.3 mm
Medium: 0.2 mm
High Quality: 0.1 mm

Top Solid Layers: The number of 100% infill layers that will be placed at the top of your part. If you notice that there's some incomplete filling your top layers, I would recommend raising this value from 3 to 5 and increasing the extrusion multiplier slightly under the Extruder tab if needed.

Bottom Solid Layers: The number of 100% infill layers that will be placed at the bottom of your part.

Outline/Perimeter Shells: Outline shells will trace the outline of your part, and extrude at your extrusion thickness. The printer will print the outline shells, then print Infill afterwards. I traditionally print with 2 Outline Shells.

Outline Direction; Inside-Out: It will print your perimeter shells from the inner shell to the outer most shell. This is very beneficial when printing overhangs, as the print is branching out in the X-Y direction for each layer.

Outline Direction; Outside-In; It will print your perimeter shells from the outer most shell to the inner most shell. This is better for surface quality finish usually. For instance, if printing a cube, this may be the better route.

Print Islands Sequentially without optimization: Let us say you are printing the part below. If you went from pillar to pillar in the most efficient order, you may get to the next layer so quickly there may be issues with heat-build up and the previous layer not being solidified. Therefore, you can turn off optimization, and then it will print the pillars in a random-order therefore helping you prevent from heat-build up.

Single outline corkscrew printing mode (vase mode): The extruder will print with one outline/perimeter shell and won't make any retracts. This means that it will slowly move up in the Z as it prints, imagine spiralling upwards instead of printing a static layer, then moving up to do another layer. Traditionally with vases, the best settings I've found are Zero top solid layers, 3 Bottom Solid Layers and under the *Advanced* tab enable "*Merge all outlines into a single solid model*".

First Layer Settings (These are applied to where the Bed touches your model. If you have a raft, that means the First Layer Settings will apply to your raft).

First Layer Height: This % value will take that percentage of your Primary Layer Height. If you are printing with a small *Primary Layer Height* like 0.1mm I would recommend a *First Layer Height* of 250%, resulting in a 0.25 mm first layer height

If your *First Layer Height* is below 100%, the extrusion amount will remain the same, only the Z-will change, but if you increase above 100%, the extrusion amount will scale accordingly.

First Layer Width: the extrusion width of your first layer, you may find that your first layer sticks better with a thicker extrusion (>100%). I don't have too much experience with this, but I think 125% or 150% would be good starting points.

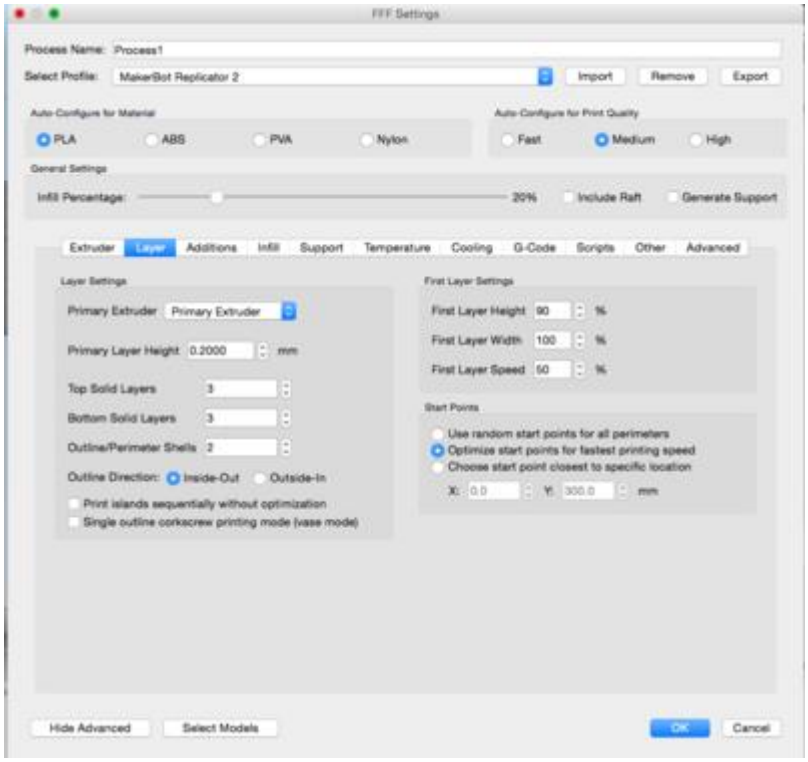
First Layer Speed: Slows down the First Layer Speed to a percentage of your Default Printing Speed *(Other tab)*

Start Points: Certain printers produce very small voids at the starting points. This can create a seam on the print. Start points options can help for both by controlling where these voids (the seam) goes and may also help print time.

Use Random Start Points for all perimeters: Randomizes the starting points

Optimize Start points for fastest printing speed: Optimizes starting points by finding the closest location for the next starting point. When generating G-Code you can see the travel moves; traditionally this option really minimizes the amount of red lines that signify travel moves that you will see on the G-Code previewer

Choose start points closest to specific location: This can be useful if you want to line-up your starting points on a certain part of your print, so the seam isn't obvious or is hidden.



Skirt Settings

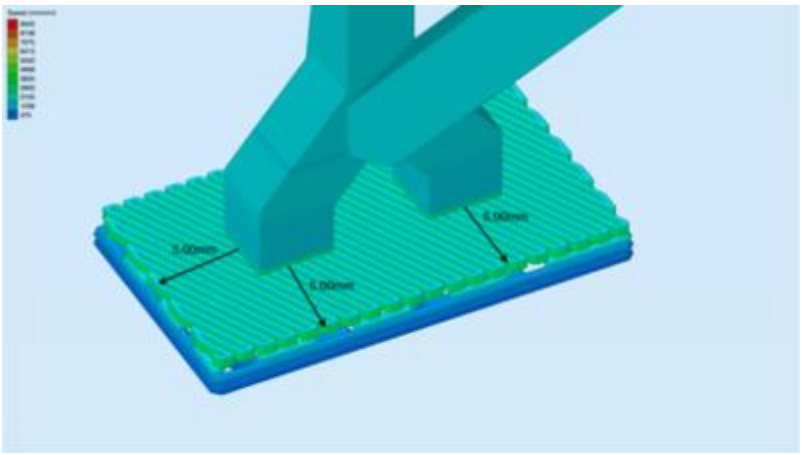
Skirts are really useful for priming your extruder prior to printing. I generally print with a skirt offset of 4 mm and 2 outlines. However, changing the offset to 0 mm, offers the ability to create a "brim" that will connect to your part and help create surface adhesion. For more information on skirts, brims and rafts, this tutorial is very helpful:
http://www.simplify3d.com/support/tutorials/rafts-skirts-and-brims/

Skirt Extruder: Selects the extruder to be used for printing your Skirt
Skirt Layers: How tall your skirt will be. If you want the skirt to be as tall as your part, you can just place 99999 in this field.
Skirt Offset: How far away your skirt will be from your part.
Skirt Outlines: The number of perimeter outlines for your skirt.

Raft Settings

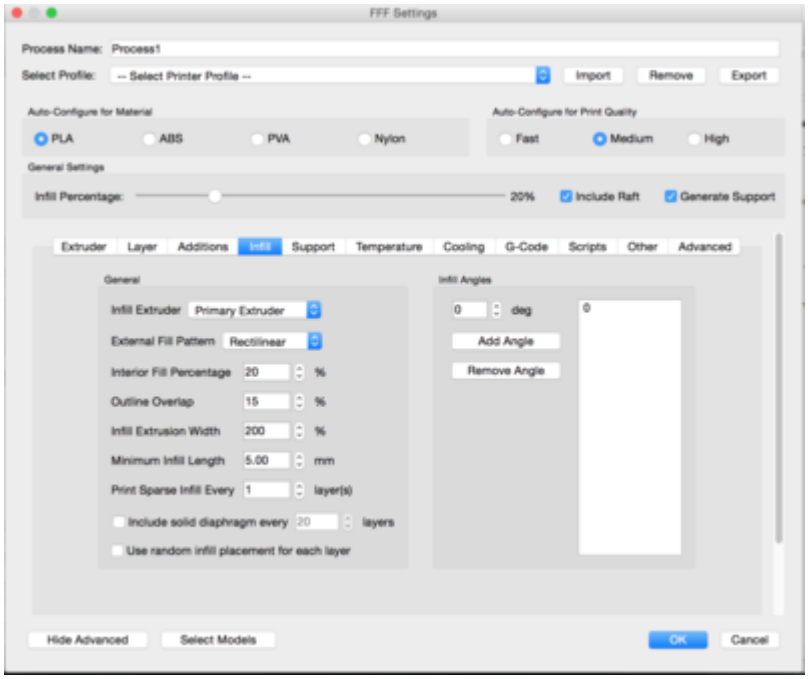
A raft offers a base under-structure that will contour to the outline of your print, increasing the surface area to the bed. This in-turn will offer more adhesion to the print bed. However, because rafts take a long time to print and use a lot of material, it is only recommended to use rafts for prints that have a low surface area with the print bed or that generally don't have a good base will often benefit from a raft. Also, whether due to the design of the printer or calibration, those users with beds that are slightly uneven have often found that rafts can provide a good foundation.

Raft Extruder: Selects the extruder to be used for printing your Raft
Raft Layers: How many layers your raft will be.
Raft Offset: How far away your part the raft will extend out to.



Separation: How much extra distance on the Z-axis from the raft to the bottom of your part.
Raft Infill %: This will typically be set around 80%, set the top layer of your rafts infill. For rafts made from dissolvable filament, upping this to 100% may be helpful.
Disable Base Layers: Disables the thicker layers at the start of the raft, disabling this can be helpful if your rafts stick to the bed too much.

Infill tab:



Infill Extruder: Pick which extruder you'd like to extrude the infill of your part with.

External Infill Pattern: The top and bottom solid layers will be printed with this choice. Either *Rectilinear* or *Concentric*. Concentric infill will mimic your perimeter outlines, coming inwards. For a cylinder, it would make smaller and smaller circles.

Rectilinear is the more common choice, it will fill in using straight lines.

Infill Fill Percentage: What % your part you'd like to be filled in.

Outline Overlap This value will dictate how far into the perimeter the infill goes. If this value was 0, the infill would start at the perimeter and not overlap at all. At 100%, you would have complete overlapping. Calibrating this value, I would say the average value I find that works well is 30% for the printers I've used, but it does vary.

Infill Extrusion Width: This % value will modify your extrusion width under the extruder tab for the Infill. If you have a 0.5 mm extrusion width under the Extruder tab and set a 200% *Infill Extrusion Width*, your Infill will be 1 mm thick.

Minimum Infill Length: This is default set to 5 mm, if there are small sections in the G-Code previewer that aren't filling in, you may want to set this to a lower value.

Print Sparse Infill Every *'x'* layer(s): If you'd like to only print Infill once every 2 or every 3 layers, then you'd place that number in the input. My experience has been that most printers can print infill at 0.3 mm just fine. Therefore, if I'm at 0.1 mm layer height, I'll use 3 sparse layers, and if at 0.15 mm I'll use 2 sparse layers.

Include Solid diaphragm every *'y'* layers: A solid diaphragm is a layer of infill at 100%. For certain parts, this can be helpful, if you want to keep the infill percentage low, but also adds the extra

structural rigidity a 100% layer of infill can provide.

Use random Infill placement for each layer: When this option is enabled, each layer of infill will start at a random spot. This means that the lines of infill won't line up.

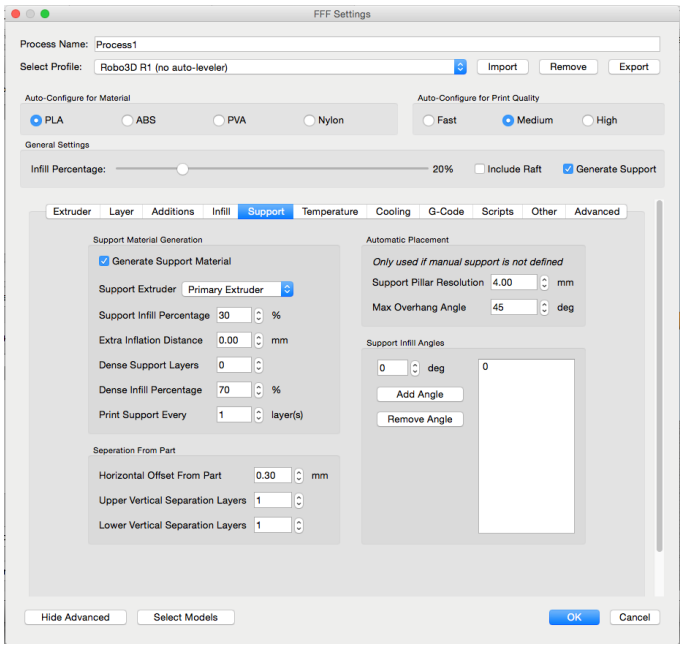
I personally am a fan of keeping this option off, stacking the layers of infill on-top of each other seems to provide stronger parts.

Infill Angles: Standard angles are 45˚/-45˚. That means that layer infill angles and external infill will print at alternating 45 degree angles. These values traditionally don't need to be adjusted, but I'm also a fan of using 0˚, 45˚, -45˚or 90˚ as the angle orientations.

Support tab

One of the best features in Simplify3D is the management of support material. The option of where to put it, in addition to how easily it breaks off makes it a key feature.

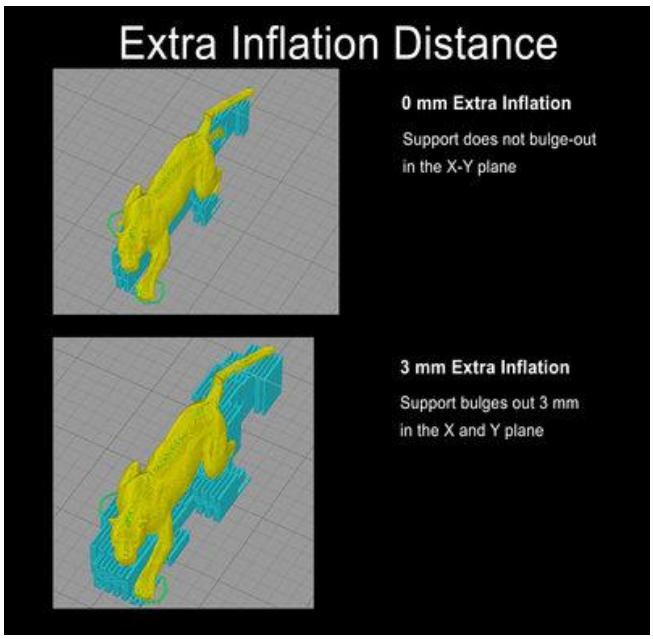
https://www.simplify3d.com/support/tutorials/adding-and-modifying-support-structures/



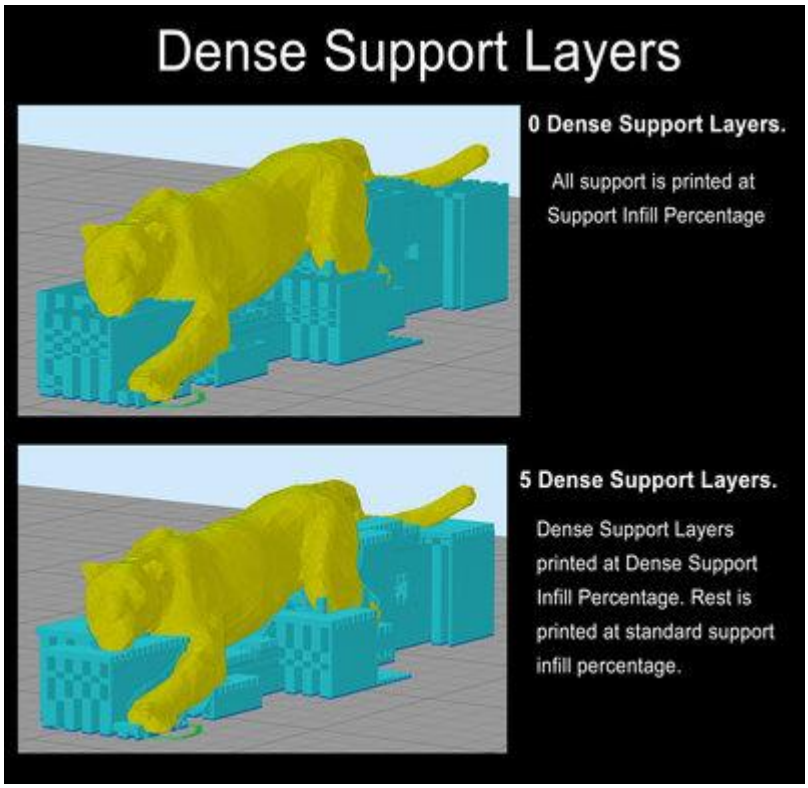
Support Material Generation

Support Extruder: Which extruder your support material will be printed with.
Support Infill Percentage: The Infill percentage that the support will be printed with. For PLA/ABS anywhere from 20 to 40% usually works really well, I've typically left this on the default value.

Extra Inflation Distance: This will increase the amount of support generated in the X-Y plane.



Dense Support Layers: How many of the layers closest to your part will be filled with the dense infill percentage
Dense Infill Percentage: The infill percentage for the top layers closest to your part. If you use PVA/HIPS filament, this feature is definitely a must-have, so you can print the bulk of your support at 20-30% support, then bump it up to 60-100% for the layers that will have contact with your part.



Print Support Every *'g'* Layers: Similar to spare infill, sparse support means that it will extrude for your support, but only print the support every *'g'* layers. For instance, if you have 0.2mm layer height, and print supported every 2 layers, it will print your support at a 0.4mm layer height.

Separation from Part

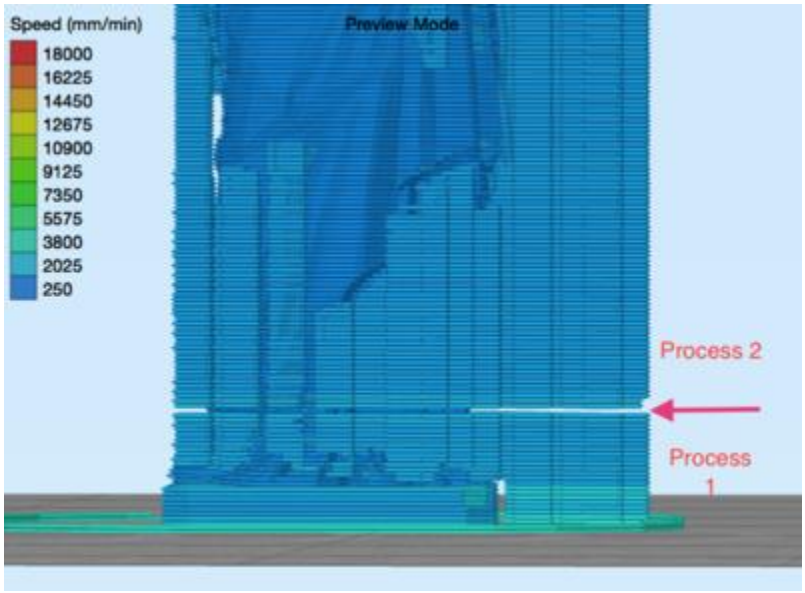
Horizontal Offset From Part: How far away on the X-Y plane the support will be generated from your part.
Upper Vertical Separation Layers: How far off in the Z-plane the support will be set from your part. Generally, you can keep this set to 1, unless you're using HIPS/PVA, you may want to set it to zero.
Lower Vertical Separation Layers:

Support Gap in between Processes

This is something will happen if you set up a part with two processes that has support. The default settings for support material configuration will set it up, so that you will have a gap of 2 layers in the middle of your support that is very-much unwanted.

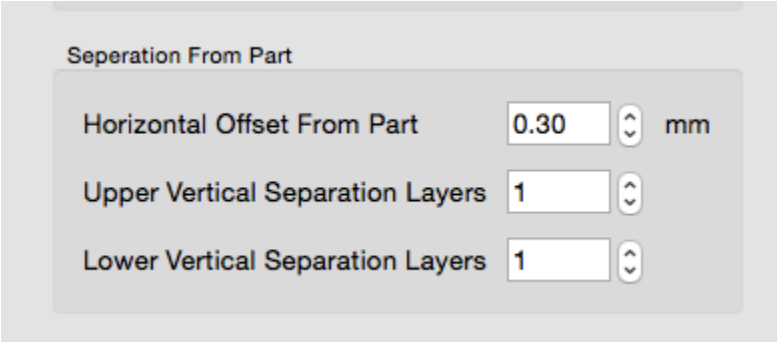
The screenshot below is for a print of the Statue of Liberty, default supports and two processes.
Process 1- From 0 to 5 mm
Process 2- From 5 mm onwards

You can clearly see that there's missing support layers where the arrow points, where Process 1 ends, and Process 2 starts.



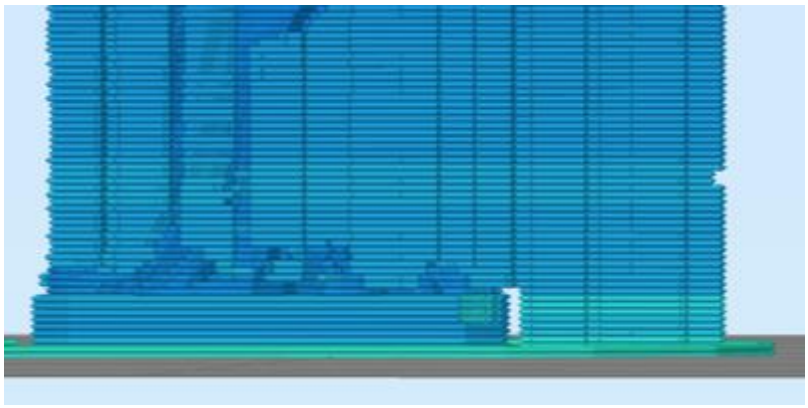
To fix this, we need to change settings in the Support Tab for each process.

In the Support tab settings, there's the option for Upper Vertical Separation Layers and Lower Vertical Separation Layers. In order to get rid of the Gap in the support in the picture above, you would need to change these settings.



Process 1- From 0 to 5 mm: Upper Vertical Separation Layers set to Zero
Process 2- From 5 mm onwards: Lower Vertical Separation Layers set to Zero

After those changes are made, the file will slice without any gap between the two processes.



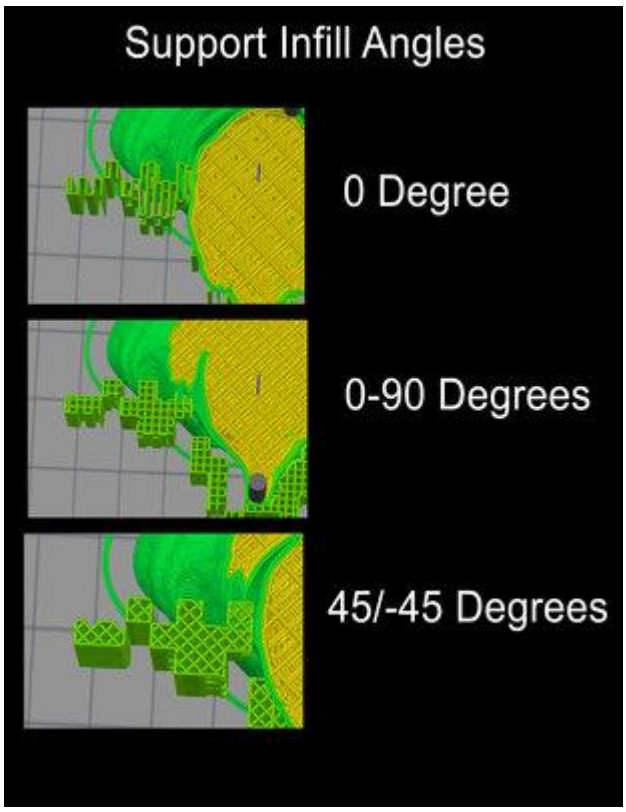
If you're new to printing one part with different settings for different regions, the tutorial below is very helpful in explaining the process.

http://www.simplify3d.com/support/tutorials/different-settings-for-different-regions-of-a-model/

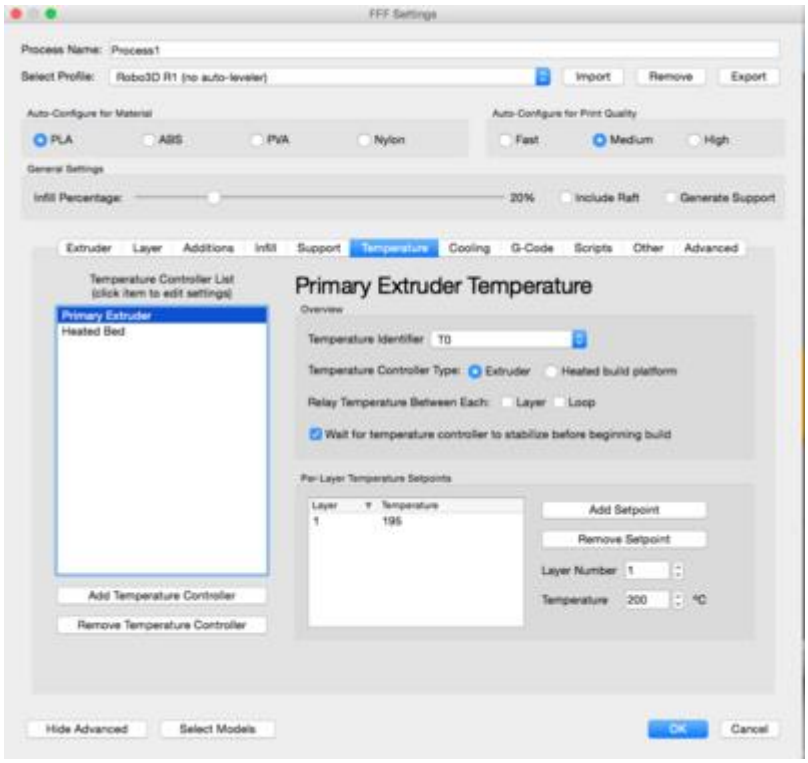
Automatic Placement

Support Pillar Resolution: The size of the pillars generated. This is default at 4 mm, for smaller parts you'll most likely need to lower this to one or two mm to get the support pillars you want.
Max Overhang Angle: The max overhang angle that will be allowed. I recommend this calibration test piece for figuring out how well your printer works with support generally: http://www.thingiverse.com/thing:40382

Support Infill Angles



Temperature tab



Temperature Controller List: This is where you'll have your extruders and heated bed listed. If you need to add an extruder/heated bed you would click the button for "Add Temperature Controller"

Temperature Identifier: The firmware identifier for this tool head. Most printers use the notation for T0 for the Right Extruder and T1 for the Left Extruder. If you aren't 100% sure,

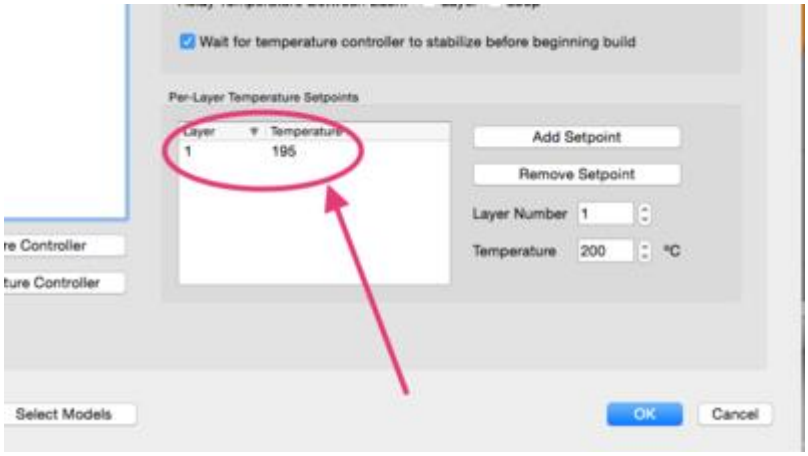
you can open the Machine Control Panel and use the temperature identifier selector in that screen to find out what tool head is which.

Temperature Controller Type: Sets whether your element is an extruder or heated bed. This makes a difference in the G-Code used to control it. For instance, the RepRap commands for an extruder are M104 and M109 whereas for a heated bed it's M140 and M190.

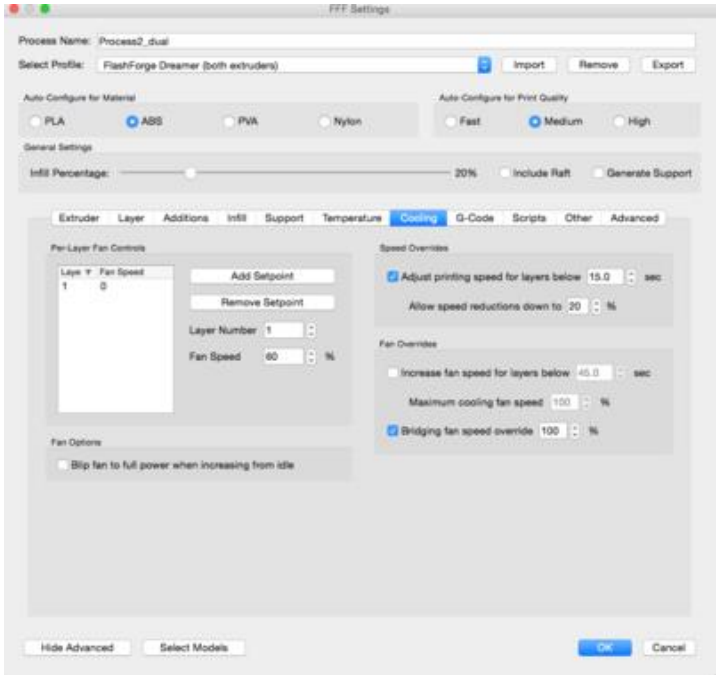
Relay Temperature between each: This will set how often to relay the temperature for USB prints.

Per-Layer Temperature Settings: Allows you to set temperature on a per layer basis. The temperatures that are set are read to the left. The adjustable numbers on the right do not affect your print, only when you click *"Add Setpoint"* are the temperature commands entered in.

Also, if printing with multiple processes, the layer # chosen is relative to the process. For instance, if this process applied from 5 mm to 10 mm on the model. The temperature for layer 1 would take place at the first layer after 5 mm.



Cooling tab



Per-Layer Fan Controls: You can edit the fan speed on a per layer basis. This is very similar to how the temperature tab works*. The fan speed will only change when there's a command to have it change. Therefore, if you set it to 60% at layer 1, the entire print will run at 60%. The most common recommends for PLA/ABS are as follows:

PLA:
Layer 1- Fan off
Layer 2- Fan On 100%

ABS:
Layer 1- Fan off

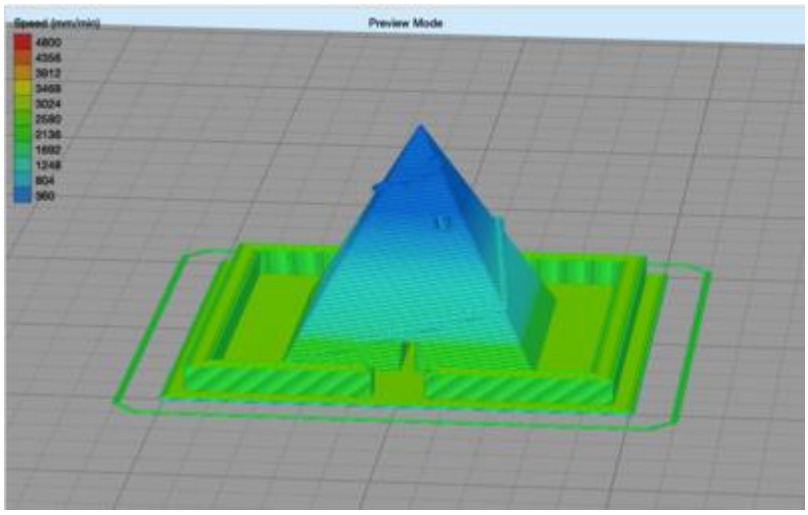
Blip fan to full power when increasing from idle: With small fans, there's very little torque. Meaning the fans can sometimes not generate enough force to get moving, (once they're moving they're fine, but getting started can be tough), this setting can help blip the voltage to the fan and get it running. For instance, if you have Blip fan on and your PLA settings are:

PLA:
Layer 1- Fan off
Layer 2- Fan On 60%

The G-Code would be written to turn the fan on at 100% first (higher torque blip), then turn the fan to 60%.

;layer 2
Fan on 100%
Delay .5 seconds
Fan on 60%

Adjust Printing Speed for Layers below *'t'* Seconds, Allow speed reductions to %: This setting takes effect for layers that would otherwise print too quickly. For instance, if you printing a pyramid as seen below, you can see that the print speed lowers as the layers get smaller. You can see from the legend on the side, that the dark blue is the slowest speed, it's because the Cooling Speed Overrides are slowing this layer down dramatically because it would take far less than 15 seconds per layer.

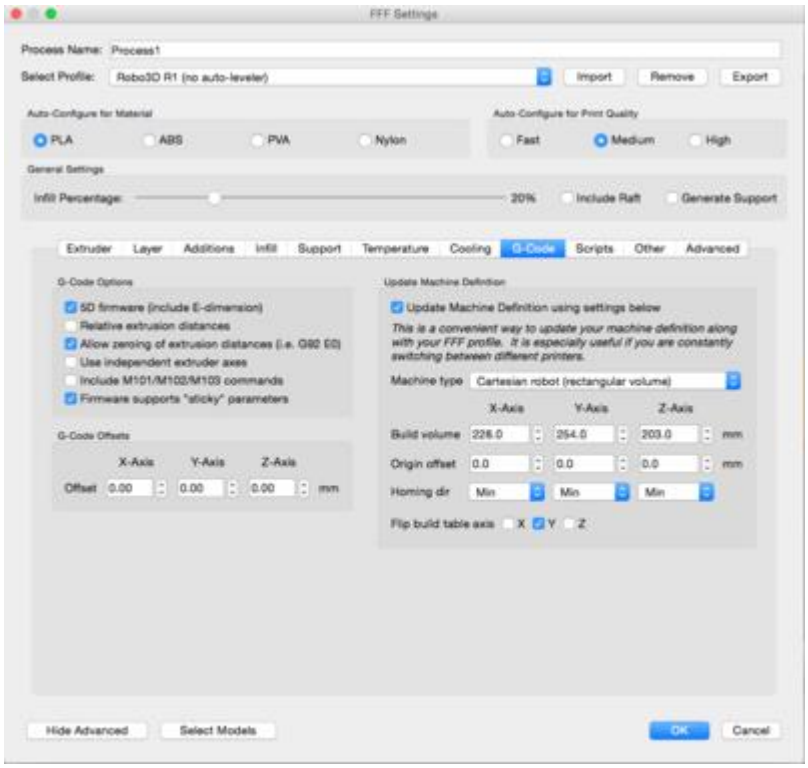


Increase fan speed for layers below ___ Seconds, Maximum Cooling fan speed ___% This is similar to the speed overrides above, but you would just be controlling fan speed. I personally have not used this feature, as I typically like to print with either 0 or 100 % fan speed.

Bridging fan speed override: For bridging it's important to get the material to solidify as fast as possible, setting a fan speed for bridging only can help accomplish this. For the rest of the bridging settings, see the Other tab.

**When printing with multiple processes, the layer referenced is relative to where the process starts. For instance, if this process started at 5 mm (Advanced tab setting), then the layer 1 command would take place at the first layer after 5 mm.*

G-Code tab



5D firmware (include E-Dimensions): 5D firmware means that your printer supports X,Y,Z, Speed and Extrusion amounts. Almost every printer currently on the market does, so this should remain checked for most of printers.

Relative Extrusion Distances: The options for G-Code creation are absolute or relative extrusion distances. For relative extrusion distances, each line would have an E-value that only applies for that line, whereas for absolute extrusion distances the E-values stack up.

Relative Extrusion:
G1 X## Y## E2.5
G1 X## Y## E2.5
G1 X## Y## E2.5
G1 X## Y## E2.5

Absolute Extrusion:
G1 X## Y## E2.5
G1 X## Y## E5
G1 X## Y## E7.5
G1 X## Y## E10

Allow zeroing of extrusion distances (i.e. G92 E0): When using Absolute extrusion mode, the printer keeps a running virtual position of the extruder. Using the command G92 E0 can reset this virtual position to zero, this command is in most the starting scripts for RepRap machines prior to and after priming. Certain printers have firmware compatibility issues with this command and certain printers (relative extrusion machines) don't need G92 E0 commands, so this command will vary from printer to printer.

Use independent extruder axes: Certain firmware (MakerBot/Sailfish) will track the extrusion rates of printers individually, whereas RepRap printers will use a single running tally (not independent) for extrusion rates.

Include M101/M102/M103 Commands: The tool-tip states that these are legacy commands and not used by modern printers. In looking up these commands this was what I found they used to do (in case you're interested):

M101 Extruder on, forward
M102 Extruder on, reverse
M103 Extruder off

Firmware supports "sticky" parameters: Sticky commands are a great way of preventing much larger G-code files. The way to think of "sticky" commands is something won't change unless it's told to change.

If sticky commands were not supported, changing the X value and extruding	**With Sticky Commands:**
G1 X10 Y10 E1 F100;	G1 X10 Y10 E1 F100;
G1 X20 Y10 E2 F100;	G1 X20 E2;
G1 X30 Y10 E3 F100;	G1 X30 E3;
G1 X40 Y10 E4 F100;	G1 X40 E4;

G-Code Offsets: If your prints are off-centred or too high off your build-plate the G-Code offsets are a great way to fix that. For instance, if your prints are 2 mm too high off your build plate, apply -2 mm in the Z-axis. If your prints are 10 mm to the right, place -10 mm in the X-axis. You will see these shifts in the G-Code previewer, but since these offsets are what works for your machine then that shouldn't be an issue!

Update Machine Definition using settings below: There are two ways to set the Machine Definition in Simplify3D. Under Tools-->Options (Windows) or Simplify3D--->Preferences (Mac) and under the FFF Settings window. If you set the Machine Definition under the Options window and then find that it's being overwritten every time you click on your process, this would mean that this option is enabled and overwriting your build volume with different values.

Build Volume: This is pretty straight forward, the only time that it gets a bit tricky is with Delta style printers. For a Delta printer, take your diameter and multiply it by .707 (Since Simpliy3D measures the largest square that will fit in your build-plate), you will still get to use your entire build volume when printing, it just uses a different number when setting the build volume.

Origin Offset: If your origin is at the bottom left of your printer (RepRap), it would be 0,0,0.

If your origin is in the centre of your machine (MakerBot/Sailfish and Delta printers) you take your build volume # and divide by two to place the origin in the centre of your build plate.

Homing Dir: This setting doesn't change the print instructions, but just changes the G-Code previewer. You can set where your homing endstops are relative to your axis that way when Simplify3D has a G28 command *(home all axis)* it will know where the extruder gantry is.

Flip Build Axis: This is for the build area in Simplify3D, generally can leave this as default. If you need to flip one of the axis relative to your origin for your printers configurations you can do so from here.

Scripts tab, G-Code

The scripts tab is a really powerful tool in the FFF settings window. This allows you to set Starting Scripts, set G-Code for every Tool Change, set G-Code for every Layer Change and set G-Code for the end of your print!

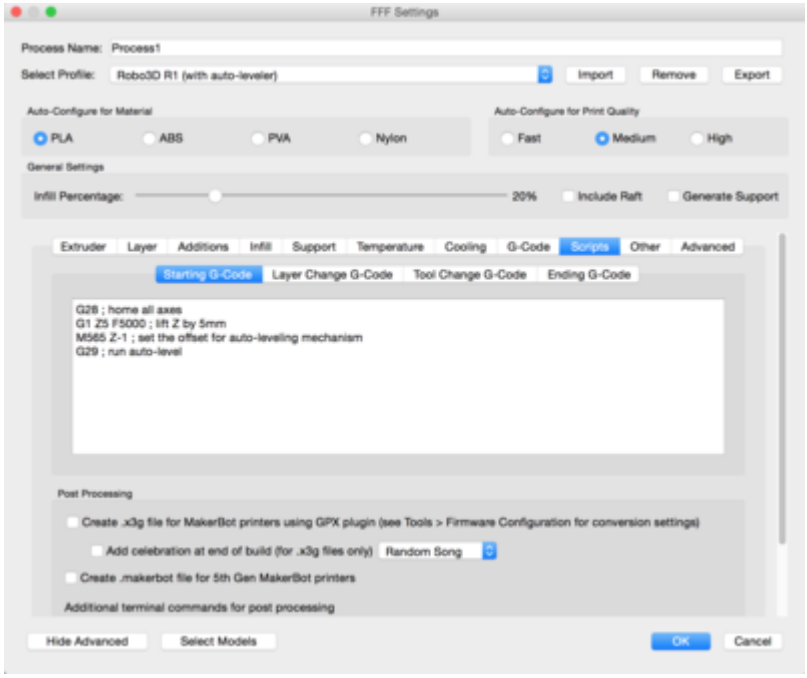
If you're intimidated by G-Code, it's actually a lot easier than it looks, a great way of learning and testing is to use the Communications tab over USB connection and send your printer a line of G-Code at a time to see the responses. One thing to keep in mind is that different printers will accept different commands to accomplish the same task. This is just something you should keep in mind if you're wondering why a certain G-Code command does not work with your printer.

Basics of G-Code:

G-Code is read line by line, the printer will accomplish one line of G-Code, then move on to the next. Always capitalize the letters in G-Code, spacing is important, and anything that comes after a semi-colon will not be read on that line. The two lines below are for home-all, and then move to 100, 100, 100 at a certain speed. The printer will home all 3 axis, then once that is done, move to the next line of G-Code.

G28; home all axis
G1 X100 Y100 Z100 F3600; Move to X100, Y100, Z100 at 3600 mm/min
(G-Code is natively in mm/min)
The link below is extremely useful, a glossary of G-Code commands for RepRap style printers
http://reprap.org/wiki/G-code

Scripts Tab:



Starting G-Code:

This is the G-Code that will be ran after heating up but before your print. Traditionally, this will consist of homing the printer, potentially purging the nozzle, wiping the nozzle and running an auto-levelling function if the printer supports it. In addition, you can add PID values, or set tool-head offsets here if you'd like to.

If you're not a fan of the fact that Simplify3D heats up the bed/extruder, than runs your starting-script, you can customize this, by using the commands below. If you have the commands for [bed#_temperature] or [extruder#_temperature] in your starting G-Code script, the software won't add in heating commands, it will just run off of the commands in your starting script.

[fan_speed_percentage]
[fan_speed_pwm] - for example, M106 S[fan_speed_pwm]
[extruder0_temperature] - for example, M104 S[extruder0_temperature] T0; this will take the layer 1 temperature for extruder 0
[extruder1_temperature]
...
[bed0_temperature] - for example, M140 S[bed0_temperature] T0; this will take the layer 1 temperature for bed 0
[bed1_temperature]

Layer Change G-Code:

I personally haven't had to use this, but I'm sure that there are some excellent reasons/ideas to use for this. If you'd like for a G-Code script to be inserted in-between each layer, than you can simply place it in this tab. One interesting use of this, is for the FlashForge Dreamer, to have the lights blink in between each layer, however that can be a bit too much at times! The placeholders that are available for this tab are below:

[previous_Z_position]
[current_Z_position]

Tool Change G-Code:

This tab will insert G-Code on tool change commands. These commands can help do a ton of things for multiple extruder prints, especially if using the IF command functionality. For instance, if you wanted to change the temperature of your extruders (T0, and T1) for each tool change switch:

G28 X0 Y0; homes X-Y axis
{IF NEWTOOL=0}M104 S165 T1; set T1, inactive extruder to 165 C
{IF NEWTOOL=0}M109 S220 T0; Set T0, new active extruder to 220 and wait for it to reach temperature before proceeding.
{IF NEWTOOL=1}M109 S220 T1; set T1, Heat T1 to 220
{IF NEWTOOL=1}M104 S165 T0; Cool T0 to 165

If you were using one process (printed part with one extruder, support with the other), you would only need this script in your one process. However, if using multiple processes, you would need to copy and paste this into each process.

{IF OLDTOOL=0}G1 E-10 F1800 - this will only include the line that precedes the {IF} brackets if the old tool (the one that was active prior to the tool change) is tool 0. You could use scripts like this to have different tool change retract distances for different tools
{IF NEWTOOL=0}G1 E10 F1800 - similar to the above command, however it checks the new tool (the one that will be active after the tool change)
[old_tool]
[new_tool]
[current_Z_position]

Ending G-Code:

Usually the Ending G-Code's purpose will be to get the nozzle off of the part and then turn off the heaters/motors. The Ending G-Code below is taken from the RoBo 3D profile:

M104 S0 ; turn off extruder
M140 S0 ; turn off bed
G28 X0 ; home X axis
M84 ; disable motors

The placeholders available for the Ending G-Code:
[current_Z_position]

Post Processing:

Create X3G file for MakerBot printers using GPX plugin: This is enabled for all of the MakerBot(Not 5th Gen MakerBot)/Sailfish style printers. The conversion will take your Machine Profile from the X3G tab of the Firmware Configuration and use that to convert your G-Code file to X3G. When you click Save Toolpaths to disk after slicing, the X3G file is automatically created when you save a .Gcode file.

Add Celebration to end of build: Add song to end of build, using Random always keeps

things new when printing

Create .MakerBot file for 5th generation printers: Converts the G-Code file into .MakerBot file. Similar to the X3G, when you click Save Toolpaths to disk after slicing, the .MakerBot file is automatically created when you save a .Gcode file.

Additional Terminal Commands for Post Processing:

If you are using a custom GPX ini file, this a very important section, as you will need to reference GPX in this section rather than use the built-in functionality. There's also a ton of post-processing commands below that help for those who are working with custom firmwares, or doing projects with their printers that require more functionality.

Using the Post Processing box would be if using a MakerBot/Sailfish style printer that doesn't have traditional steps-per-mm values, you would need to do the following:

1) Use the Machine ini example to set a text document ini with your machine information: https://github.com/whpthomas/GPX/blob/master/examples/example-machine.ini
2)Save the file as GPX.ini, place it in the Simplify3D installation folder
3)Place the following in the post-processing box: GPX [output_filepath]

Below are the post-processing tools that also work in the post-processing box:

{REPLACE "E" "A"} - search and replace for the text within quotes, in this example every "E" character would be replaced with an "A" character
{PREPEND "G92 E0\n"} - prepends the specified text at the very beginning of the gcode file, note that the \n is converted into a true newline character, not two separate "\" and "n" characters
{APPEND "G28 X0 Y0\n"} - appends the specified text to the very end of the gcode file
{DELETE "M82\n"} - deletes every occurrence of the specified text from the gcode file, note that it will not automatically delete a line if it is suddenly empty after the deletion, so that is why you might want to include the \n at the end (so that the empty line is also removed)
{STRIP ";"} - completely deletes every line in the gcode file that begins with the specified text
{TOOL0REPLACE "E" "A"} and {TOOL1REPLACE "E" "B"} - these special TOOL#REPLACE commands will do a search and replace, very similar to the {REPLACE} command, however, the replace only occurs if the specified tool is active. For example, when using TOOL1REPLACE, the replacement will only occur if tool 1 was currently active at that line of the gcode file

"I've been working a lot with dual head prints and I am trying the tweak in the settings... With regard to the tool change g-code and reducing temperatures of the inactive extruder, is it not possible to reference the extruder temperature variable and use it in the tool change g-code?

For example I mean this:

From:
G1 X0 Y40 F4000 ; move to wait for temperatures
{IF NEWTOOL=0}M104 S165 T1; set T1, inactive extruder to 165 C
{IF NEWTOOL=0}M109 ***S205*** *T0; Set T0, new active extruder to T0 Temperature and wait for it to reach temperature before proceeding.*

{IF NEWTOOL=1}M104 S165 T0; Cool T0 to 165
{IF NEWTOOL=1}M109 ***S205*** *T1; set T1, Heat T1 to T1 Temperature*

To:
G1 X0 Y40 F4000 ; move to wait for temperatures
{IF NEWTOOL=0}M104 S165 T1; set T1, inactive extruder to 165 C
{IF NEWTOOL=0}M109 ***S[extruder0_temperature]*** *T0; Set T0, new active extruder to T0 Temperature and wait for it to reach temperature before proceeding.*
{IF NEWTOOL=1}M104 S165 T0; Cool T0 to 165
{IF NEWTOOL=1}M109 ***S[extruder1_temperature]*** *T1; set T1, Heat T1 to T1 Temperature*

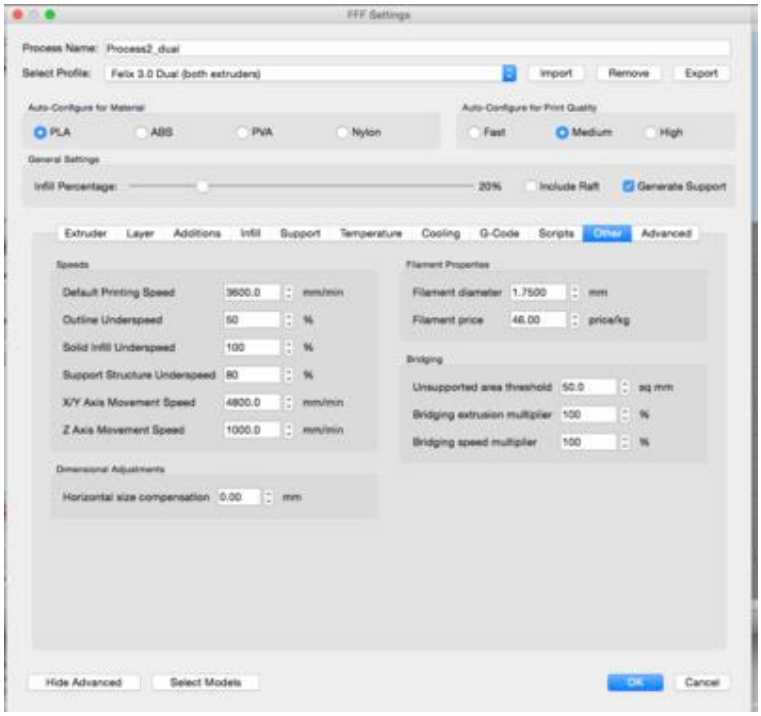
The reason for doing this is to have different temperatures at different layers that are controlled by the temperature tab in the settings... It didnt seem to work for me when i tried it so i was curious if there is another way to reference the temperature/layer tables defined?

Also, there seems to be a slight pause when switching tools prior to the move "G1" code in the tool change script. is there a way to minimize this/eliminate this? Normally its not so bad but if you have a small feature you are trying to print the dwell tends to screw up the finish.

thanks for the input. Hopefully someone figured this out already.

Advice for those that want to use this functionality, you may want to change the order from the original post of what tools are activated and when. if you command it to wait for temperature (M109) first and then set the inactive extruder to a lower temperature it will operate in that order. Its more often best to always command the inactive extruder to the lower temp first and then the active extruder to the target temp/stabilize. This prevents extra ooze from the soon to be inactive extruder while the new tool head is warming up."

Other tab



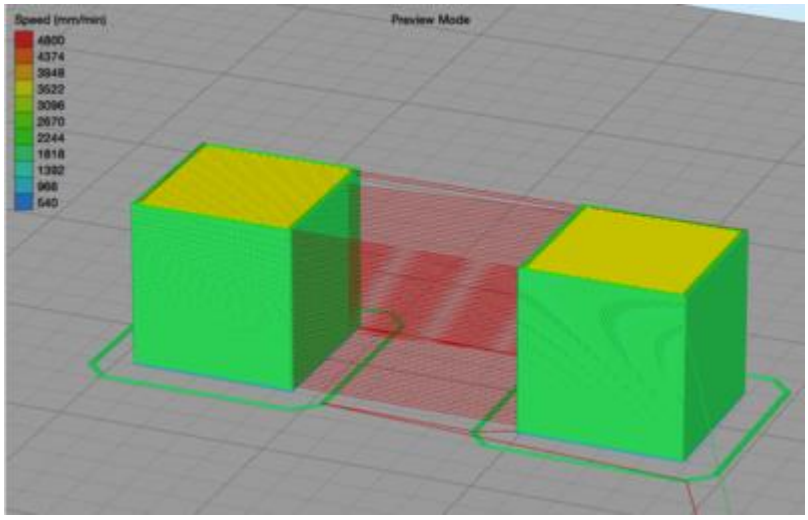
Default Printing Speed: This is the speed at which the Infill of your part will be printed, assuming the Cooling Overrides don't adjust it and that it's not a 100% infill layer.

Outline Underspeed: For your perimeters, you print at a % of your Default Printing Speed, this is the %. For instance, if you want to print your Perimeters at 75% of your Default Print Speed.

Solid Infill Underspeed: For your any layer that is 100% infill, including Bottom Solid Layers, Top Solid Layers and if you have 100% infill set in your part, you print at a % of your Default Printing Speed.

Support Structure Underspeed: The support material will be printed at a percentage of your default printing speed.

X/Y Axis Movement Speed: This is the speed your printer will move when your printer is not printing in the X-Y axis. When looking at the G-Code previewer, you can see these lines as the Travel Moves



Z Axis Movement Speed: The speed at which your Z-axis will move.

Filament Diameter: The diameter of filament your printer uses. This is very important for extrusion values. For instance, if you use 3 mm filament but have 1.75 mm filament in your setting, the software will be turning your extruder motor FAR too much since it will think you are using only 1.75 mm diameter filament. Small tweaks to this would be the same as small tweaks to your extrusion multiplier.

Filament Price: Price that you paid for your filament per kg, so that when you slice your file in the G-Code preview you can see the estimated cost.

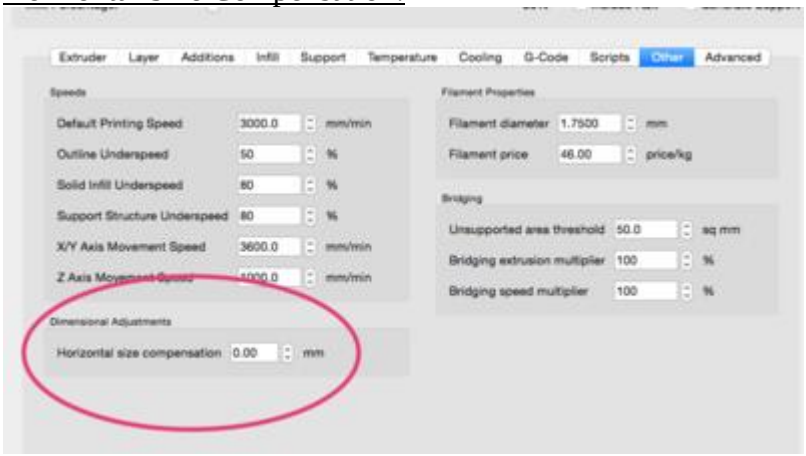
Unsupported area threshold: Any area larger than this that is unsupported will have the bridging multipliers applied to the infill in the bridging.

Bridging extrusion multiplier: If you find that extruding a lot more/less material helps with bridging, then you would set that here.

Bridging Speed multiplier: If you find that printing faster/slower helps with bridging, then

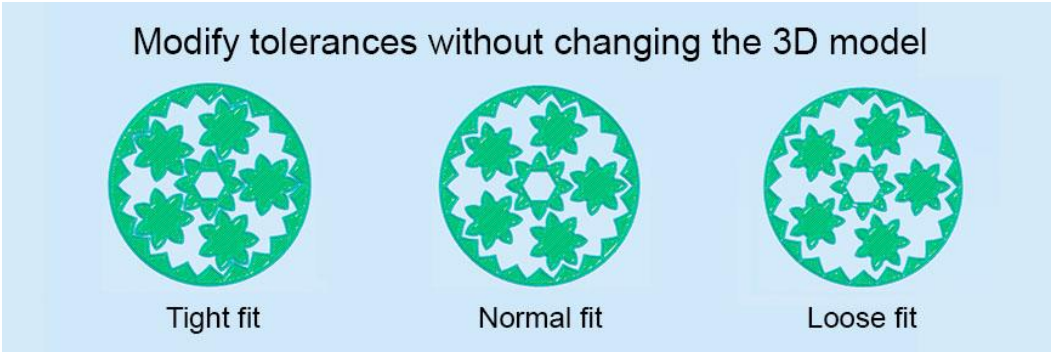
you would set that here. One trick I use occasionally, is I'll temporarily set the bridging speed multiplier to 999, then slice my file, that way the areas in which the bridging multiplier is taking place are bright red and stand out in the G-Code previewer, that way I have a better idea of where the bridging settings are occurring.

Horizontal Size Compensation:



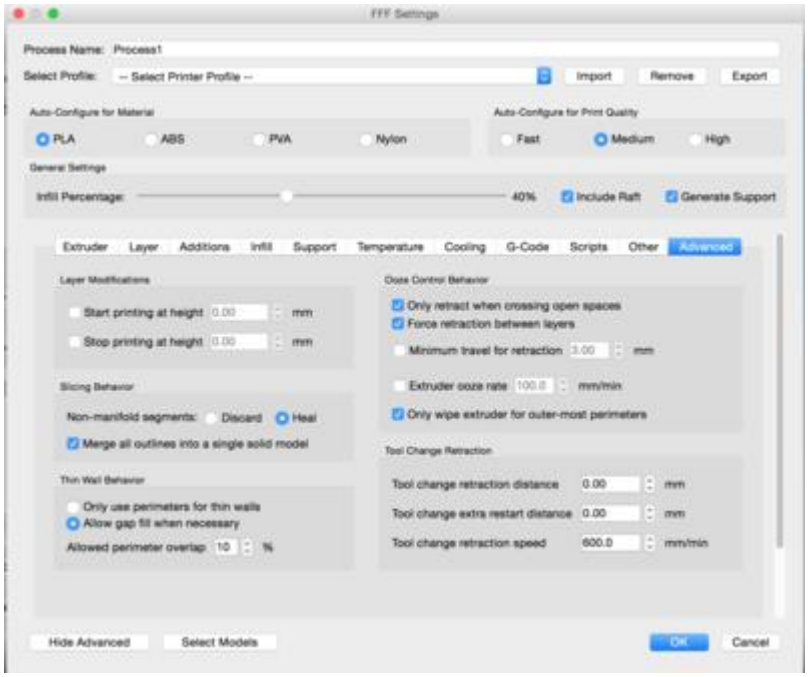
The horizontal size compensation is a new feature in version 2.2.2, if you don't see this option under the Other tab, this may mean you are using version 2.2.1 or earlier. There are many, many usage case scenarios for which you could use this feature. Taken from the blog post, the gear bearings by Emmet. If you print these out, and the plastic has moulded together, you could place a value of -.05 to increase the gap by 1.0 mm, or -0.1 mm gap to increase the gap by 0.2 mm (half the average nozzle diameter for perspective). the gap is doubled, since both perimeters are compensating away from one another, so the gap is twice the value of the input.

http://www.thingiverse.com/thing:53451



That's one use case scenario, but there's also the use case for a standard print. For instance, if you print a 25 mm cube and notice that it's 24.8 mm in the X-Y axis, then you print a 50 mm cube and it's 49.8 mm, you could add .1 mm to the size compensation to quickly remedy the size of your prints.

Advanced Tab



Layer Modifications Section

Start Printing at Height: All models in this process will start printing at *'x'* Height (mm)

Stop Printing at Height: All models in this process will stop printing at *'y'* Height (mm)

These two options are used primarily if you want to print with different settings in one part. For instance, if printing the Statue of Liberty below, you may want to print the base of the statue at a much thicker layer height then the statue itself.

Two Process Statue of Liberty Example:
The first process would stop at 50 mm (assuming 50 mm is where the base ends) and the second process would have a starting point of 50 mm so that the second process print settings would be applied to everything after 50 mm.

http://www.simplify3d.com/support/tutorials/different-settings-for-different-regions-of-a-model/- Tutorial on printing different settings for different regions

Slicing Behaviour

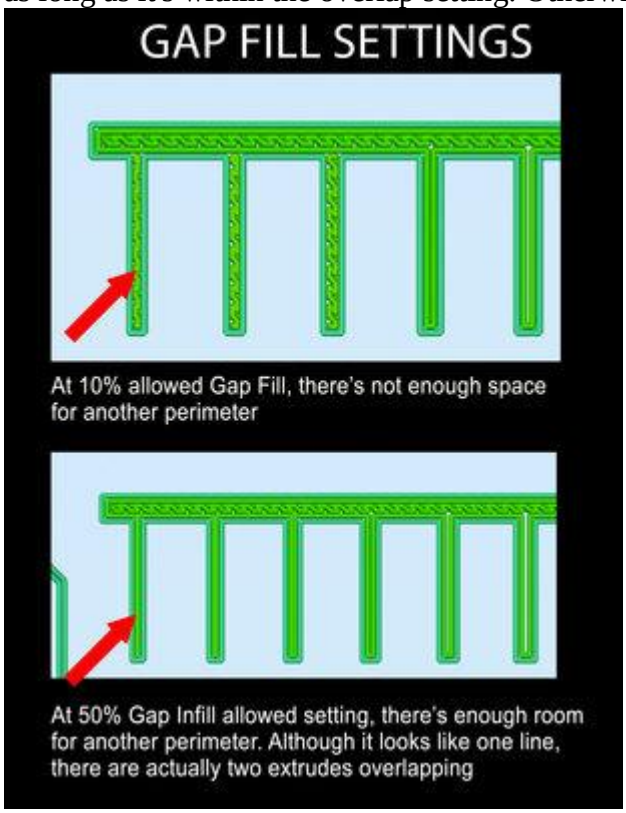
Non-manifold segments: Non-Manifold segments are errors in the mesh. These are relatively common; the best behaviour traditionally is to use Heal to attempt to patch up these missing areas. I personally have not needed to use the Discard option.

Merge all outlines into a single solid model: This feature in the software will slice the model taking all of the outer most perimeter outlines and slice with those assuming a solid model as the interior. For instance, if you had a model that was a block with screw-holes in it, but had "Merge all outlines into a single solid model" on, the holes would not show up. Using "Merge all outlines" is generally really useful for models that have errors in the mesh or if using Vase Mode.

Thin Wall Behaviour

Only use perimeters for thin walls: If printing a very thin wall, this means that the software will not add in Infill into a very small crevices. If there's room to fit multiple perimeters, it will print in multiple perimeters, but otherwise you'd be left with a small gap.

Allow gap fill when necessary: This setting will work to add in another perimeter by allowing as long as it's within the overlap setting. Otherwise, it will fill the area with Infill.



Ooze Control Behaviour:

Only retract when crossing open spaces: This is enabled by default for most profiles, since you really only need to do a retraction if you are crossing over an open space, since you would be crossing perimeters and be at risk for stringing.

Force retraction between layers: Forces a retraction from layer to layer. This is recommended.

Minimum travel for retraction: If a movement is below this threshold, it won't retract for that movement. This would be if your printer has a lot of problems with retractions, you may need to tweak this, to limit the number of retractions your print has.

Extruder ooze rate: The features tool-tip states this is an experimental option, so I personally have not experimented much with it. However, the idea would be to heat up your extruder, prime it, then wipe the filament away and measure how many mm's of filament will slowly extrude out of the nozzle over a minute, then using that data you would set that in your Extruder ooze rate setting.

Only wipe extruder for outer-most perimeters: This will ensure that you only use the Wipe

feature (Extruder tab) when needed, when it would affect the appearance of your model. Therefore, to save time and be efficient it will only wipe the extruder for outer-most perimeters. It may not be helpful to wipe the extruder after doing Infill, however if you wanted the extruder to wipe after Infill you could do that by unchecking this box.

Tool Change Retraction:

Tool change retraction distance: When changing tools, a retract of *'x'* mm will occur. The idea is that instead of a standard small retract, you can retract the filament far up the extruder to prevent any oozing while the other tool is printing.
Tool change extra restart distance: When priming the extruder that was previously retracted, this extra restart distance will be added to that value. For instance, if you have a tool change retraction of 12 mm, and an extra restart of -0.5 mm, the following will occur:

Print with Right extruder
Tool Change Retract Right Extruder (12 mm)
Prime Left Extruder (12 mm + (-0.5 mm))= 11.5 mm prime
Print with Left Extruder

Tool change retraction speed: The speed at which the retraction and primes will occur for the tool change retract/primes. I've found since you are moving much more filament, moving slower is very important. Trying to retract 12 mm of filament at F4800 may cause a lot of skipping which leads to other issues.

General Troubleshooting Simplify3D

These are some common things/subjects that I feel come up, or things that I've experienced and wanted to share.

Printer won't connect to Simplify3D

1) Check Baud Rate to be sure. Generally, 115200 or 250000 for most printers. Some printers use other Baud Rates, this information should come with your printer.

2) Ensure USB drivers are installed (If Simplify3D doesn't show your USB port for your printer, chances are there's no driver so your computers not able to recognize the USB detection. Key test, can you connect to the printer using a different software program?

RepRap style Arduino/Ramps board: install Arduino IDE that will install the proper drivers (Mendel's, Prusa's, RoBo 3D, anything with an Arduino Mega on it)
LeapFrog 3D printers: FTDI drivers http://www.lpfrg.com/support/installation
Ultimaker FTDI drivers: http://wiki.ultimaker.com/FTDI_Driver_installation
Rambo Board- http://reprap.org/wiki/Rambo#USB_Driver (SeeMeCNC Orion/Delta MAX, Lulzbot Taz, Airwolf printers, MakerGear, new Mendel's)
MakerBot/Sailfish- I would recommend installing ReplicatorG as that will include drivers as needed
MakerBot 5th Generation- Windows 7 and 8 may already have the necessary drivers, but if they don't installing MakerBot Desktop should fix this.

3) (If MakerBot/Sailfish style machine) Open MakerBot Desktop, go to Services and select Stop Background Services. Background Services will connect to the printer even if MakerBot Desktop isn't open, so if you try to connect with Simplify3D, Simplify3D will get an error

since the printer's already connected to another piece of software!

Printing over USB, Firmware Unresponsive error in machine control panel/print stops randomly

There are a few reasons this could be happening. However, I personally avoid USB printing at all costs as I find that the added risks of a failed print from USB cable transmission/PC issues is not worth it.

CNC machines are extremely similar to 3D printers and even run off of G-Code, however if you were to use a USB cable to stream G-Code to your CNC machine, that would be ridiculed as terrible design. The CNC machines I've worked with use either a flash drive or SD card for data transmission. With 3D printing, the stakes are a lot lower, if you lose a print due to streaming USB transmission failure from your PC, it's potentially only a few dollars in plastic and few hours wasted. With CNC machines, it could be a block of titanium worth 1,000 dollars being cut, so the CNC designers definitely would not have the added risks and add chance of failure in transmission.

Reasons why computer to USB transmission should be a secondary choice to SD card printing:

- Main reason: Data cables (e.g. USB) are prone to EM interference- If too many transmissions get dropped in the '*hand-shaking'* process the firmware will go unresponsive.
- Computer's sleep settings may be "on"
- Laptop battery could run out of charge
- Automatic system updates restarting the computer
- If your processor lags (playing a video game that requires high performance)- transmission could be interrupted
- Accidentally closing the software
- Computer freezing due to another program

However, if you're crashing while using a SD card, one possibility is that your microprocessor is overheating. I've had this on a Ramps Board occur, which was remedied by adding a small fan to cool the processor..

Upgraded to new version of the software, new profiles aren't showing up

Installing the new version, doesn't uninstall the old version, so it can be an easy to accidentally open a previous version if the shortcut on your desktop is linked to the old version or if your file association your computer is associated with the old version.

However, if you are opening the new version of the software, this can be fixed by going to Help-->Reset All Settings

(RepRap/Marlin style printers)

You added an extruder to the printer and the tool-head offsets are incorrect:

The firmware controls the tool-head offsets, however you can also adjust the value at the start of each print under the Scripts tab. There's the Starting G-Code where you can add tool head offset G-Code commands. For instance, if there's a 33 mm offset in the X direction between your right and left extruder, place these two commands at the beginning of your starting G-Code.

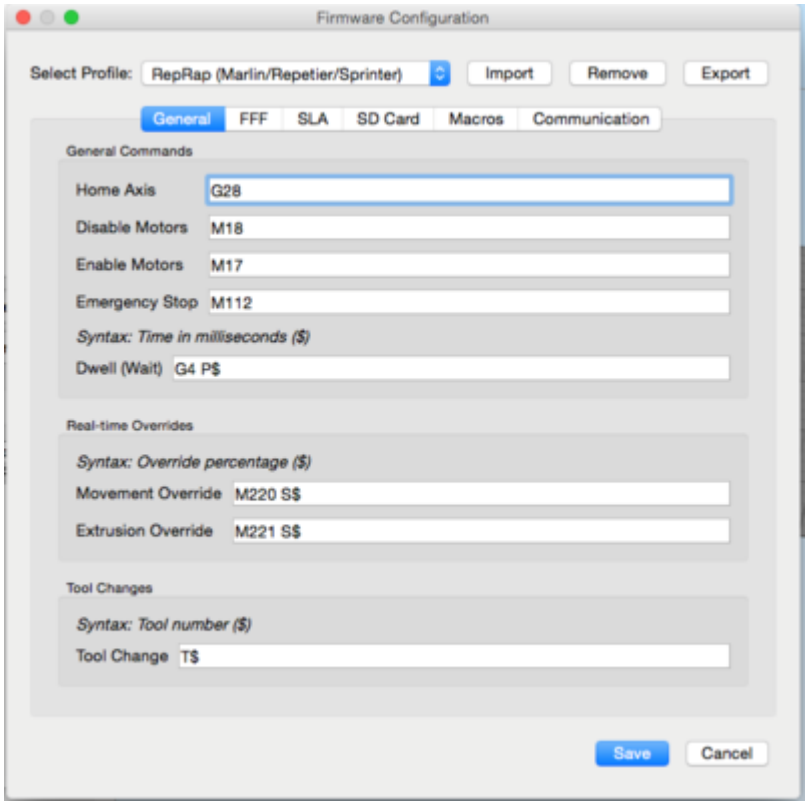
M218 T0 X0 Y0;
M218 T1 X33 Y0;

(MakerBot/Sailfish) Left Extruder/Heated Bed is not working. Or print is not to size:

Firmware Configuration is very important, it basically tells Simplify3D what "flavour" of G-Code to make and what kind of X3G file to make (if necessary). This is set automatically by the software through the Configuration Assistant. However, when you have multiple printers you switch back-and-forth from, you may not be going through the Configuration Assistant every time you switch, which means you should probably ensure your firmware configuration is right.

For RepRap style machines

As long as your properly set to RepRap style for a RepRap printer, you shouldn't have an issue. If you are set to the wrong Firmware Configuration you might notice your bed not waiting to heat up, fans not turning on, and tool-head switches not going through.

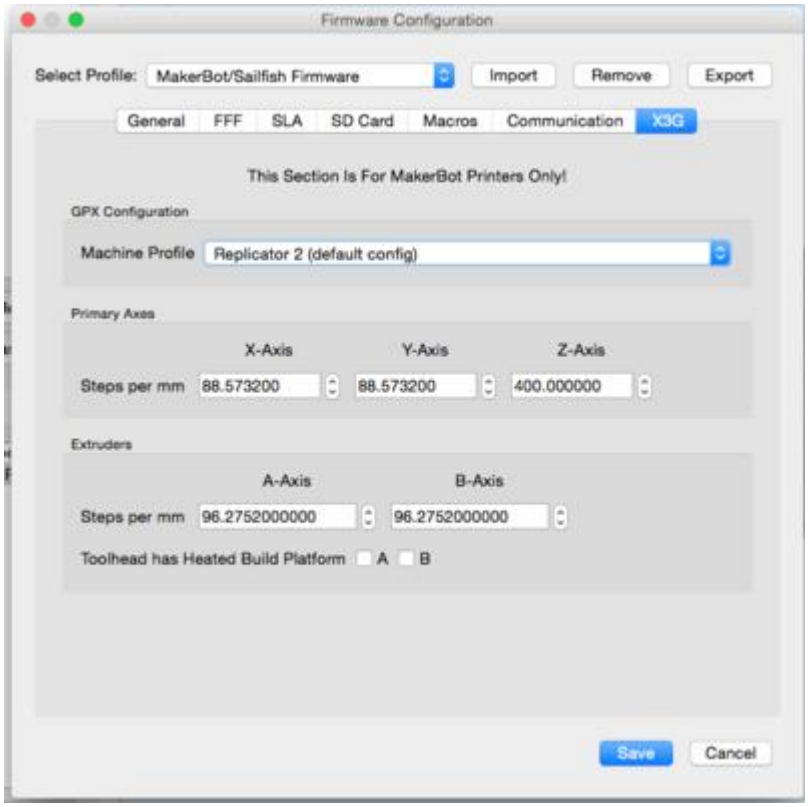


MakerBot/Sailfish style machines (*5th Gen MakerBots use RepRap Firmware Configuration*)

Having the correct profile is important, but with MakerBot/Sailfish firmware configuration, the X3G setting the Machine Profile is really important!

For instance a common occurrence of wrong Machine Profile for MakerBot style printers, If you're using a Replicator 1-dual style printer, (FF Creator/CreatorPro, Wanhao Duplicator, PowerSpec, etc.), but have it set on the default Replicator 2 profile you'll have the following symptoms:

Your heated bed won't work. (Replicator 2 doesn't have a heated bed)
You won't be able to use your left extruder. (Replicator 2 doesn't have a left extruder)
Your parts will be about 7% too small. (Replicator 2 uses different Steps-Per-MM values)



Example of practical use

For instance, if you use a FlashForge Creator Pro printer and a custom delta machine:

FlashForge Creator Pro-- MakerBot/Sailfish firmware with Replicator 1-dual extruder selected under the X3G tab

Custom Delta Machine-- RepRap firmware

When switching process settings from one to the other, it's very important to also go in to your Firmware Configuration and set that as well, otherwise you'll have a whole host of problems.

Conclusion

This generally applies to people with more than one printer, since single printer users can traditionally use the Configuration Assistant on the printer they have the first time through the software, and that will set the Firmware Configuration. However, if you're constantly switching printers, and are a printing enthusiast then it's vital to know this!

RepRap Prints are not to size:

Steps Per MM, tolerances and more!

Printing with tolerances can be tough. There are settings in the software as well as settings in your printer's firmware that may control these tolerances. If you have a prototype that needs to fit with other parts, or you want to make something fits with a certain part, it can be frustrating when your part comes out skewed. There are two different options: "Off by a large margin" and "Off by a small margin". If you're printing 20 mm calibration cubes and finding that your measurements are 20.5 mm or larger in the X-Y or Z direction, I would definitely recommend starting off in the "Off by a large margin" category first!

Off by a large margin:

If your print is off by a large margin, in the X-Y or even Z- axis, it points to one thing. Steps-per-mm values. When the G-Code Simplify3D makes is sent to your printer's firmware, your printer uses your Steps-per-mm values to decide how much to turn the stepper motor to move one mm. If the Steps-per-mm value is off, than you will get parts that are not to scale.

If you are using a RepRap style printer, you can open the Configuration.H file of the firmware in Arduino and find the steps-per-mm values there. For MakerBot/Sailfish style printers, you will need to use a custom GPX.ini file in your Simplify3D folder and reference it using the post processing script option in Simplify3D which is a bit more involved.

My recommendation for calibrating Steps per mm if necessary is to do the following:

1) Print a 20 mm cube
2) Measure the cube with digital callipers
3) Use this formula for correction for the axis you'd like to calibrate. This formula is simply for a 20 mm cube, using cross multiplication to find the Corrected Steps per mm value
CorrectedStepsPerMMValue = (*20mm x CurrentStepsPerMMValue*)/(*MeasuredSide*)

4) Use new *StepsPerMM* values and reprint the 20 mm cube to validate results!

Quick and Dirty Fix: If you don't want to change your printers firmware or use a custom GPX file, and you need correct tolerances NOW for a deadline. Just use the formula below and then double click your part and scale it accordingly in the X-Y and Z-axis. *(Not recommended, but it would most likely be a good enough temporary solution)*

Scaling= CorrectSideLength/PrintedSideLength

Off by a small margin

Small margins can be tough! Since any number of factors may be at play.

- If your extrusion multiplier is too high, your outer perimeters could be about too thick and lead to tolerance issues due to that
- Warping due to too much heat could cause your outer perimeters to lose their shape
- Print speed may be too high

Ensuring your extrusion multiplier isn't too high and also remembering not to print at a speed higher than your printer can handle are great starting points for small margins.

Also, if shifting is causing your parts to be off:
- If your belts are too loose, the motor could be turning and not turning the belt with it
- If your belts are too tight, the motor could be turning and once again, not turning the belt with it
- The voltage to the stepper motors may be off

To Conclude

I think changing and using the formula for Steps-per-mm values and working with those is very important. For small margins, you're really just changing general printing settings, that would affect many aspects of your print, not just size. If your extrusion multiplier is off, you'll probably see it in more spots than just the tolerances of your part. Really calibrating and setting your Steps-per-mm value absolutely right and as close to as possible as your callipers will let you can provide amazing results and really take your printing to the next level.

From a CAD/design stand-point, Make Magazine has a good article with a section designated for this exact topic below.

http://makezine.com/2013/12/11/top-ten-tips-designing-models-for-3d-printing/

FIT TOLERANCES FOR INTERLOCKING PARTS

For objects with multiple interlocking parts, design in your fit tolerance. Getting tolerances correct can be difficult. Kacie’s tips for creating correct tolerances: use a 0.2mm offset for a tight fit (press fit parts, connecters) and use a 0.4mm offset for a loose fit (hinges, box lids). You will have to test it yourself with your particular model to determine what is the right tolerance for the thing you are creating.