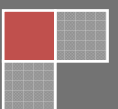


2015

Enhanced Calibration of a Fused Deposition Modeling (FDM) 3D Printer

Abstract:

This document discusses a procedure for enhanced calibration of a FDM 3D printer. While these machines are well within hobbyist accuracy limits, it is desirable to squeeze out every bit of capability the machine has. To do this some preliminary calibration steps are required followed by some more advanced calibration procedures specific to machines running Sailfish firmware. At the end are some extra steps that can be done to further improve accuracy, but this involves material specific settings that are less relevant to the project.



Keywords:

Fused Deposition Modeling (FDM): The 3D printing process of printing layer by layer, using lines or beads of extruded plastic filament

Build Platform/Build Plate: The platform that the plastic is extruded onto

Heated Build Plate (HBP): A heated variety of build plate, usually implemented with traces in a PCB that current is fed into which resides under a simple aluminum build plate

Extruder Head: The stepper motor, drive gear, and nozzle that the plastic is pushed through make up the Extruder Head

Extruder Carriage: This is the part that contains the extruder head(s) and is moved around by the other stepper motors to extrude plastic such that the desired part is created

Glass Transition Temperature (T_g): For all intents and purposes this is the temperature at which a plastic becomes more glass-like and is not as flexible. For 3D printing, this means the plastic shouldn't deform or move from where it was placed once it falls below this temperature.

Gcode: The syntax of commands used to instruct the firmware on the printer how the motors should be driven to accomplish the result of the input file.

Layer Thickness (height): This describes how thick each layer is that makes up the printed part. A smaller layer height produces a finer resolution print.

Flow Rate: A parameter to describe how fast the extruder motor will be turning

Feed Rate: How fast an axis of the machine moves (mm/s)

Width over Thickness (aka width to thickness) ratio: How wide vs. how tall the bead is. There are both infill and perimeter variants to control this setting for the infill of the part, as well as the shell around the perimeter. This is found in the slicer settings in ReplicatorG -- perimeter in the Carve plugin, and infill in the Fill plugin.

ReplicatorG: The software used to generate Gcode for the printer in this tutorial. It is adaptive, open-source, and has many plugins already installed to adjust parameters of generated Gcode. It also has an option to adjust firmware settings on the printer's EEPROM when plugged in via USB using the "Onboard Preferences" dialogue found under the "Machine" menu.

Sailfish: A specialized firmware offering advanced features developed for 3D printers with MakerBot Mighty Boards, ATmega chip based control boards, and a few others.

Preliminary Steps:

A FDM printer's build procedure is composed of drawing out a bead of extruded plastic. This extruder head is guided around the build platform, also known as a build plate. For certain materials such as ABS, the build plate must be of the heated variety to even out the temperature gradient and allow the material to stick to the build platform. For ABS, this temperature is usually around 90-110 C. Plastics also can shrink upon cooling, and with ABS it is recommended that the printer be enclosed to slow the rate of cooling to avoid warping. This combined with the HBP provides the ABS a nice and thermally stable environment to be drawn together and sit while the part then cools below the glass transition temperature (T_g) which is about 105 C for ABS. It is of utmost importance to understand the material being used as well as its properties during printing to improve the performance. For ease of use, PLA is desired, however ABS examples are used to demonstrate the necessity of knowing your material.

The next most important thing is to understand the physical properties of what is going on. For the plastic (or metal) to adhere to the build platform, it should be level with the XY plane that the extruder head moves in. The height of certain areas of the HBP can be adjusted by turning the screws on the bottom. There are four of them, and by repositioning the head near all four attachment points, an accurate setting can be achieved. There is also a script built into the firmware for calibrating the build plate, however it can be more trouble than it's worth. It is recommended if you are new to working with this type of machine. The bottom line is that the extruder head must be approximately the same distance from the HBP for any position in the XY plane.

If a dual extruder printer is used, such as in our project, the nozzle position with respect to one another must be an input to the Gcode generator software so it knows how far to move to position the second extruder head correctly. This is done by printing a test line and measuring the distance with calipers, or attempting to measure the distance between the nozzles directly with calipers.

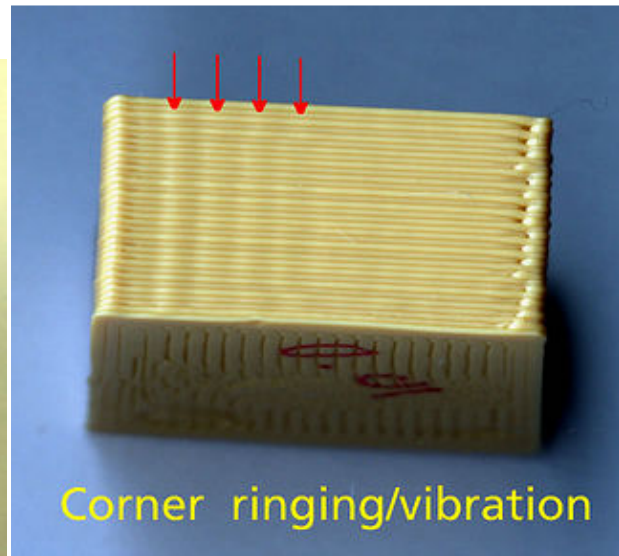
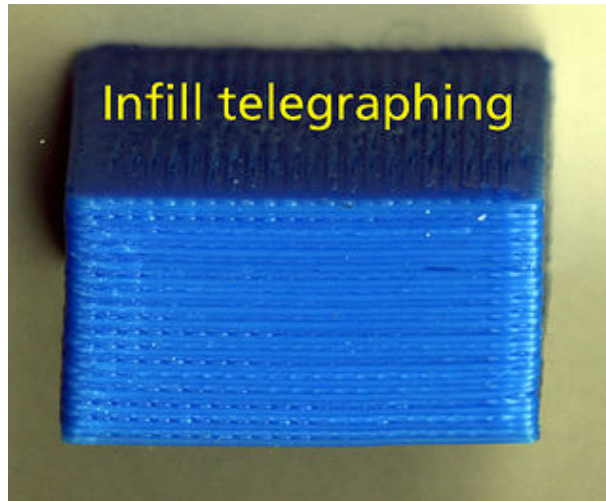
Another hassle of a dual extruder printer is that both nozzles must be aligned in the z direction as well. This is adjusted by a set screw in the back of the aluminum mount block on the printer's extruder carriage. The set screw is loosened, and the extruder nozzle is re-positioned to line up with the other nozzle. If this step is avoided, only one of the nozzles will have the correct height for optimal extrusion onto the HBP, and if the other nozzle is not used (i.e. a single extruder print) it can drag across the top of the object, and since it will be cold, will knock the object from the HBP (unlike when being used and it is hot and will push its way through the plastic, deforming it but not causing the print to fail). This will cause the print to fail in a mess of plastic web if left unattended.

Calibration Procedure:

Now that the basics are covered, it is time to calibrate the electronics and software. This is meaningless if the part will not stick, so the preliminary calibration steps are highly recommended. The first step is to print and analyze the printer's output when generating Gcode for a 20mm calibration box. This file comes with MakerBot's MakerWare, and also can be easily downloaded from Thingiverse. It is necessary for this testing procedure.

The goal of printing the 20mm calibration box is to see if the proper amount of filament is extruded for a controlled volume. The sharp corners of the box also provide a good test for the printer's ability to stop or change directions accurately.

While analyzing the output part, it is important to know what errors in the setup the defects point to. The 20mm box should be printed at 100% infill, 1 shell, and whatever layer height desired. Below are some pictures:



http://farm9.staticflickr.com/8015/6970439452_9f2827423f_n.jpg

http://farm8.staticflickr.com/7038/6970439344_eecbd925b8_n.jpg

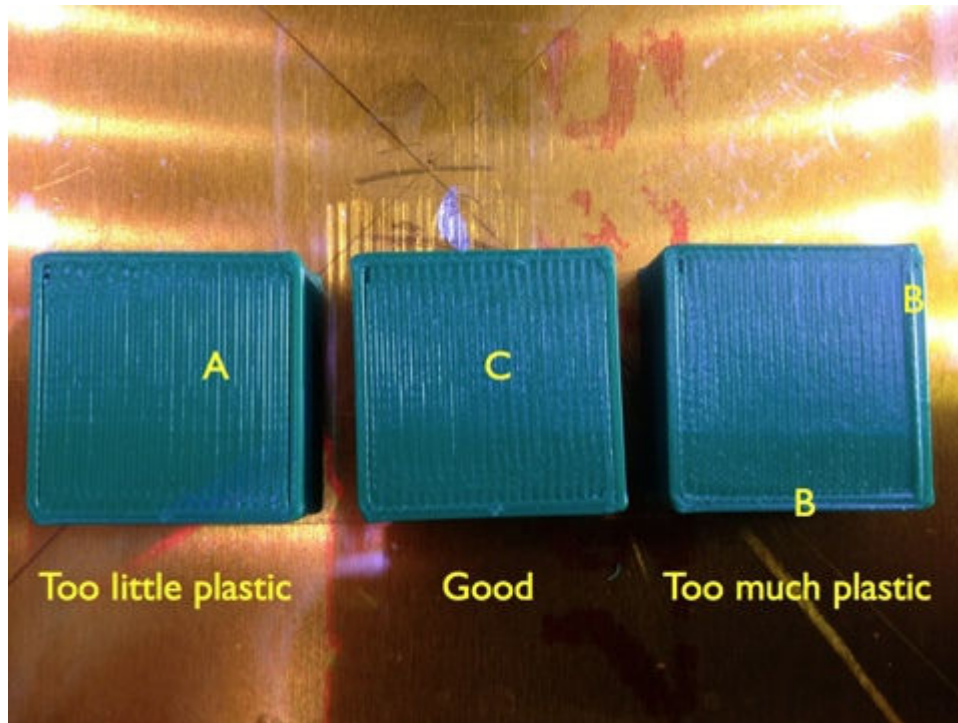
The first picture is an example of "Infill Telegraphing", which basically means the infill pattern is exposed through the perimeter paths. While typically more of a problem printing with one shell, it shows a problem and should be fixed. This can be remedied by adjusting the perimeter width to thickness ratio as well as cleaning the nozzle face, as a dirty nozzle can push the layers around and cause this to appear. This is a software parameter found in the slicer settings of ReplicatorG. It is part of the carve plugin and is known as "Edge Width over Height (ratio)".

Sometimes a dirty nozzle will also push the infill out past the perimeter because the plastic blobs displace some of the freshly laid (and warm/pliable) layers towards the outside. It is good to rule out things that are not a problem with the setup. If somehow a smooth result cannot be achieved, printing parts with two shells can cover up the issue, however it may become a problem later on.

The second image shows corner ringing. As the print head approaches a corner, the pulse train used to drive the stepper motors decreases in frequency, sometimes hitting the resonant frequency of the extruder heads/carriage amplifying the vibrations of the motor enough that it becomes visible in the print. Because the machine does the same thing on each layer of the box (except top and bottom-most), this is visible throughout all similar layers. To remedy, the maximum X and Y speed change values and/or maximum X and Y acceleration values must be changed in the machine onboard preferences

which can be accomplished by going to "Machine > Onboard Preferences" while the printer is connected to the computer via USB. This writes data to the machine's EEPROM, so do not unplug it or remove power from the board during this process.

To test the proper fill of the cube, acceleration should be turned off on the printer's menu, and the feed rate should be set for 30mm/s in the software. The box should again be printed at 100% infill, and 1 shell. This is true for all of these procedure steps unless stated otherwise. Below are some examples of boxes with various results:



http://farm8.staticflickr.com/7113/6948525864_cb926ca751_d.jpg

Examining print A, there are dips in between the beads along the top due to too little plastic being deposited. In extreme cases the plastic may even sag and a visible concave depression will be present. If too much plastic is used, the dips/gaps in between fill paths will be filled in, however the plastic can be messy as the extra is pushed around on the surface. It tends to build up near the end of the path.

The Infill Width over Thickness ratio should be adjusted by .05 increments. If there is not enough plastic, it should be turned down, likewise for too much plastic, it should be turned up. Once it is close, smaller increments can be used. The feedrate should be held at a constant 30mm/s to be slow enough to not require acceleration. At this point, the basic enhanced calibration steps are complete, and the additional steps section will refer to continuing on here, before the acceleration settings are dialed in.

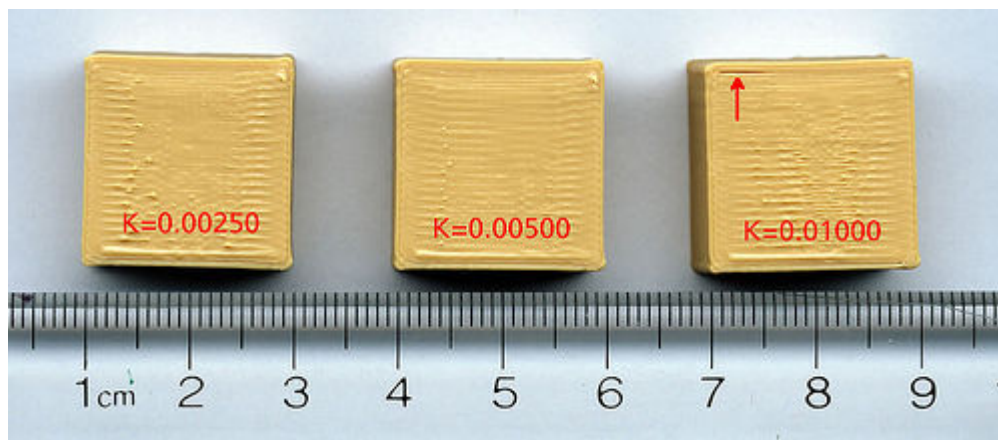
Accelerated Printing Calibration:

With the current adjustments, the printer should produce significantly better prints. The remaining steps concern printing at faster speeds, which for the metal plasma deposition head won't be used

anyway. It is desirable for plastic prints, however, and will help make this rapid prototyping solution more rapid.

The primary reason for the development of the Sailfish firmware was to print faster while maintaining accuracy. This is accomplished by carefully monitoring the acceleration and deceleration of the print head. While the printer is moving in this enhanced nature, the extrusion feed must also be adjusted or it will result in blobbing at low speed near corners, and small gaps at higher speed areas.

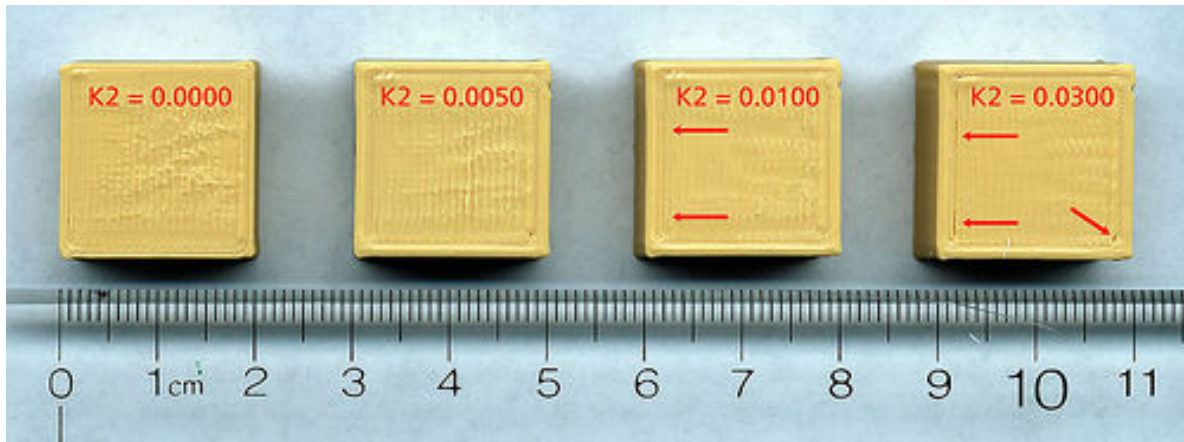
The creators of the firmware known as "Jetty", Bernhard Kubicek, and Dan Newman chose to name these extra parameters JKN Advance K, and JKN Advance K2. These are onboard firmware parameters. The K parameter deals with the plastic deficit and surplus while accelerating and decelerating quicker than normal. While setting the K2 parameter to zero, the maximum X and Y acceleration values to 500 mm/s² or less, as well as a feedrate that the printer can handle (30mm/s is appropriate unless a quicker value has been proven to work), the 20mm calibration box is printed with the default K value of 0.0025. Another should be printed at 0.0075 as well as 0 for a control. The idea is to reduce the amount of surplus plastic at the corners, but not too far which will result in corners that looked beveled or gaps in the top surface (similar to under fill while setting the width to thickness ratio). Below is an example:



http://farm8.staticflickr.com/7185/6967541502_7ae110794c.jpg

The left cube could have a better top surface because there is excess plastic near the edges and corners (increase K). The middle cube is much better, but still has extra plastic in the corners. The right cube has a gap near one of the edges, which indicates that the plastic deposition during deceleration has been reduced too much, so K should be decreased. This tells us that a good K value is somewhere in between 0.005 and 0.010 for this setup.

The K2 parameter accounts for the asymmetrical pressure differences in the extruder when accelerating or decelerating. They are not identical, and this parameter effectively fine tunes K. Leaving the previous K value found in the last step, the calibration box should be printed with varying values of K2.



http://farm8.staticflickr.com/7265/6979991360_9a378946a4.jpg

Above are some examples. With K2 set to zero, the corners protrude more than they should. With K2 set to 0.0050, slight improvement is observed with the upper two corners not projecting as much. The box printed with K2=0.0100 has nicer corners, but the left side's perimeter is pulling away from the infill (not quite visible in picture). To emphasize this effect, the box with K2=0.0300 shows clearly visible separation on the left side, as well as separation in the bottom right corner. A good value for this setup would be between 0.0050 and 0.0100. It should be noted that these minute differences are easier to see with real calibration boxes instead of pictures.

If speed is not necessarily a priority, or some increased acceleration is desired, but quality is paramount, the "Altshell" plug-in can be activated in the slicer settings. This will slice the model with accelerated infill commands, but disable them for the perimeter. When using the plugin, be sure to check the "use M320/M321" checkbox for Sailfish firmwares (as opposed to the older Jetty firmware). The non-accelerated perimeters will be printed with the "maximum speed change" values.

Additional Steps:

In addition to the steps described above, there are additional calibration procedures that can help the machine print parts to fit with each other better, as well as have an even finer tuned perimeter finish. Before, the infill width to thickness ratio was modified to ensure proper fill, however there is also a perimeter width to thickness ratio, that was briefly mentioned to fix infill telegraphing.

Start by downloading the following STL files:

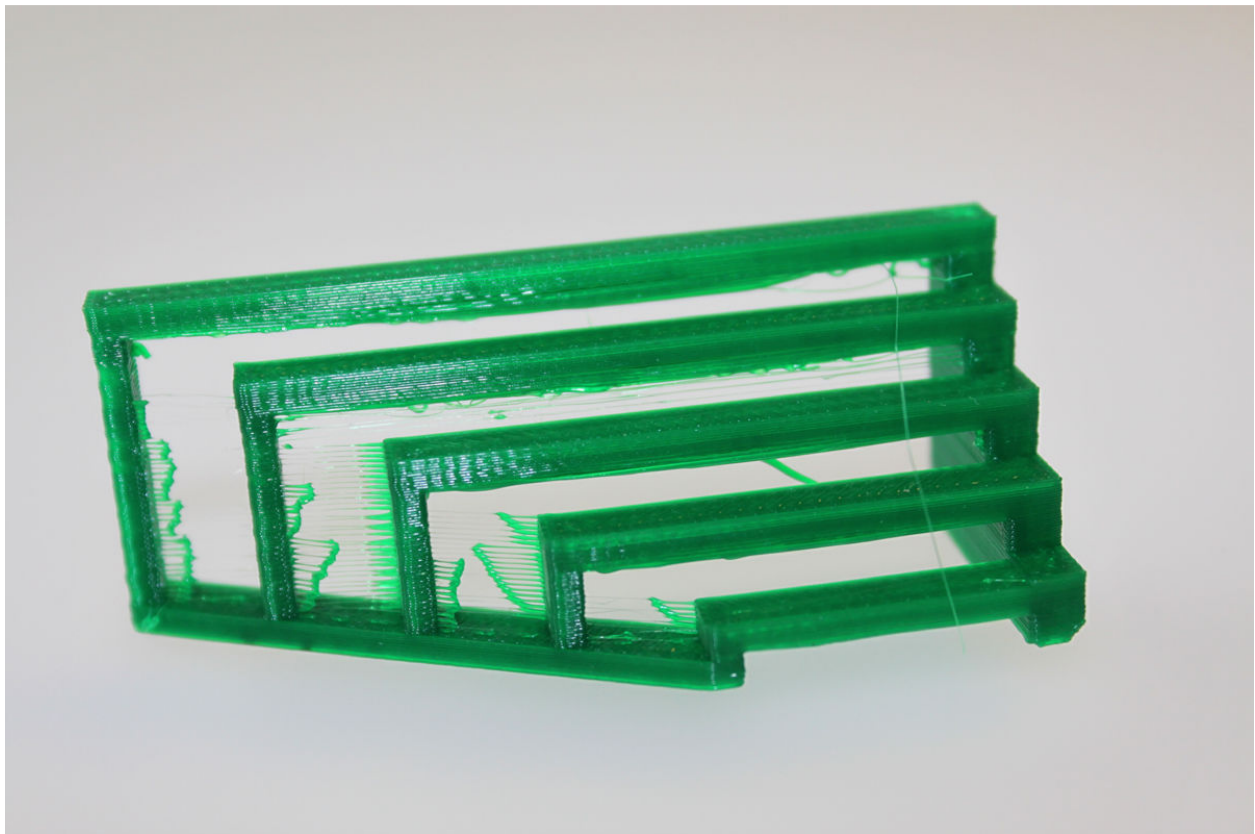
<http://www.thingiverse.com/thing:5573/>

Print "perimeter-wt.stl" using 100% infill and 1 shell. 200 micron layer height would be a good choice, but it can really be whatever is desired. It is important to make sure it is 100% infill, because this will be double checking the infill width to thickness ratio. The parts should fit snugly together, or at least partially fit in. If it doesn't, this means there is too much plastic, so the perimeter width over thickness must be increased, but stop if infill telegraphing becomes visible. In between this range lies the ideal

value. Like earlier, adjusting in 0.05 increments are a good place to start, and smaller increments can be used as the desired fitting is approached.

One last thing that can be fine tuned, is the printer's abilities to print what are known as bridges. Because the FDM process does not allow for overhangs of much more than 45 degrees unless support material is used, a bridge describes an overhang that connects to another part of a surface, hopefully not needing support. The printer can print this without support, provided it travels fast enough and puts out enough plastic per distance to make a bead attached to one side, string through the air, and then make contact with the other. There will always be a length limitation, but that limit can be pushed with the correct settings.

Below is an example of various bridge tests, note that as the length increases, more plastic is visible drooping below the bridge.



http://3dprintingforbeginners.com/wp-content/uploads/2014/11/Pirx_Review_482.jpg

Also included in Thingiverse Thing # 5573 is a hollow version of the 20mm box used previously. Although printing this will not allow you to see the bridge after it's done, the inside is completely hollow, so it will have to print the top of the box without any support underneath which effectively tests its ability to correctly print bridges.

Print the 20mm hollow box with the same settings as before, and observe if the top droops. With a box this size, a droop indicates the plastic isn't cooling quickly enough, or not enough plastic is being fed in

time, and the bottom layers are too weak. To remedy, go to the speed plugin and increase the bridge feedrate multiplier by .1 increments until the top stops drooping. There isn't a good way to check the upper limit of this number, so .1 increments must be used continuously until it looks good. There is also an overhang test included in the package, but the best way to get better overhang performance (as well as bridge) is to install an active cooling modification. Active cooling refers to actively blowing air at the plastic to make it cool faster, typically only used for PLA since it does not harden quickly and tends to have much worse overhang and bridge performance. The MakerBot Replicator 2 is one of the few printers that comes with this feature, however there are aftermarket solutions available on Thingiverse.

This concludes the procedures for enhanced calibration of a 3D printer running Sailfish firmware. To recap, your printer should now have a level build plate from leveling as described in the preliminary steps. The onboard preferences should be adjusted (if needed) to control acceleration levels in the firmware to avoid infill telegraphing and corner ringing/vibration. Next the infill width to thickness ratio was adjusted to get good performance with the maximum infill (which is the worst case scenario). After the baselines are covered, the accelerated parameters were adjusted by changing the JKN Advance K and K2 parameters. The Altshell plugin for ReplicatorG was also proposed as a solution to improve print quality if maximum speed is not desired. Lastly, additional steps to set the perimeter width to thickness ratio, as well as the bridge feedrate multiplier were covered. These steps will program the printer to deploy the exact amount of plastic required for your material of choice. Additionally, scaling can be set to accommodate for different shrink rates of different plastics, however that is beyond the scope of this procedure. The printer should now be printing at its maximum capability for its current motor and control board setup; note that if hardware is changed, this procedure will more than likely need to be redone to once again achieve prime accuracy from the printer.

References:

<http://www.makerbot.com/sailfish/setup/>

<http://www.makerbot.com/sailfish/troubleshooting/>

<http://www.makerbot.com/sailfish/tuning/>

<http://www.thingiverse.com/thing:5573/>

[file:///C:/Users/\[user\]/replicatorg-0040r22-Sailfish/skein_engines/skeinforge-50/documentation/skeinforge_application.skeinforge.html](file:///C:/Users/[user]/replicatorg-0040r22-Sailfish/skein_engines/skeinforge-50/documentation/skeinforge_application.skeinforge.html)

<http://www.sailfishfirmware.com/doc/parameters.html#x18-560011>

<https://davedurant.wordpress.com/2010/10/18/configuring-skeinforge-the-4-or-5-big-settings/>

<http://www.thingiverse.com/thing:52946/#instructions>