Deterrence Dispensed recommends PLA+ for most firearm related projects. PLA+ is cheap, tough enough for most applications, and prints with ease. Nothing is perfect, however, and ease of use comes at the cost of low heat tolerance, and for some applications, not enough strength. Hobbyists today have a plethora of materials to chose from: ABS, PETG, Nylon, even exotic blends impregnated with carbon fibers. While all have their own advantages and disadvantages, the undisputed king of materials for engineering applications, heat tolerance, and strength is Polycarbonate (PC).

If PC is so much better for firearms applications, then why does DD not recommend it? Basically, the difficulty threshold for printing PC is much higher than PLA, and equipment is generally more expensive. This guide aims to change that. Using the ubiquitous and inexpensive Ender 3 that you probably already have, with minimal modifications, PC prints are perfectly attainable.

# Equipment:

- Ender 3

Any modifications you've done to improve your current Ender 3 will also help in your PC endeavors. As stated in the "Complete Getting Started Guide," a new print bed and upgraded bed leveling springs will ensure there is no warp in your bed and helps it stay level. These are practically mandatory upgrades for even PLA printing. I prefer a borosillicate glass bed, and before printing lay down a nice even layer of glue stick on the glass to aid in adhesion. PEI has been shown to be exceptional for PC as well.

- Micro-Swiss all metal hot end

PC requires printing at a higher temperature than PLA, and extended high temps over 250* can damage the PTFE tube. On the standard hot end, the PTFE tube extends all the way to the nozzle, causing degradation over time. An all metal hot end moves the PTFE tube above a heat break, allowing it to stay cooler. There are slightly cheaper options than the Micro-Swiss, but trust me, spend the little extra: they're proven, and their customer service is top notch.

- Wiring harness extension

You'll be printing inside a heated enclosure, so it's a good idea to get the sensitive electronics away from the heat. They might be fine for a while, but the general consensus is to extend the wiring and get the motherboard and power supply out of the heat. You can DIY the wiring with a soldering iron or butt connectors, heat shrink, a roll of wire, and patience. The alternative is to buy a complete harness extension. I've also used the PrinterMods plug and play kit and it works great.

- Heated enclosure

PC needs to be printed in a heated enclosure to avoid warping and bed adhesion issues. The enclosure can be as complex or as simple as you want. Searching "Ender 3 heated enclosure" will bring up a myriad of choices. The key is to have as few air gaps as possible, and some insulative properties wouldn't hurt. Some like to make enclosures out of an Ikea LACK table. Files are on Thingiverse for hinges, and brackets to bolt on plexiglass sheeting. I made my enclosure out of simple, cheap plywood screwed together. 24"x24"x24" gave plenty of room for the printer and the heaters. A couple of small hinges, magnetic cabinet latches, plexiglass sheet for a door, and an under-cabinet light finished it off, and it works great. The heat comes from two small personal office heaters, and is controlled by a simple plug and play Inkbird temperature controller.

Equipment List:

Micro-Swiss all metal hot-end: $64

Capricorn PTFE Bowden tube: $12

PrinterMods wire harness extension: $30 or DIY

Enclosure: $ Depends on what you make

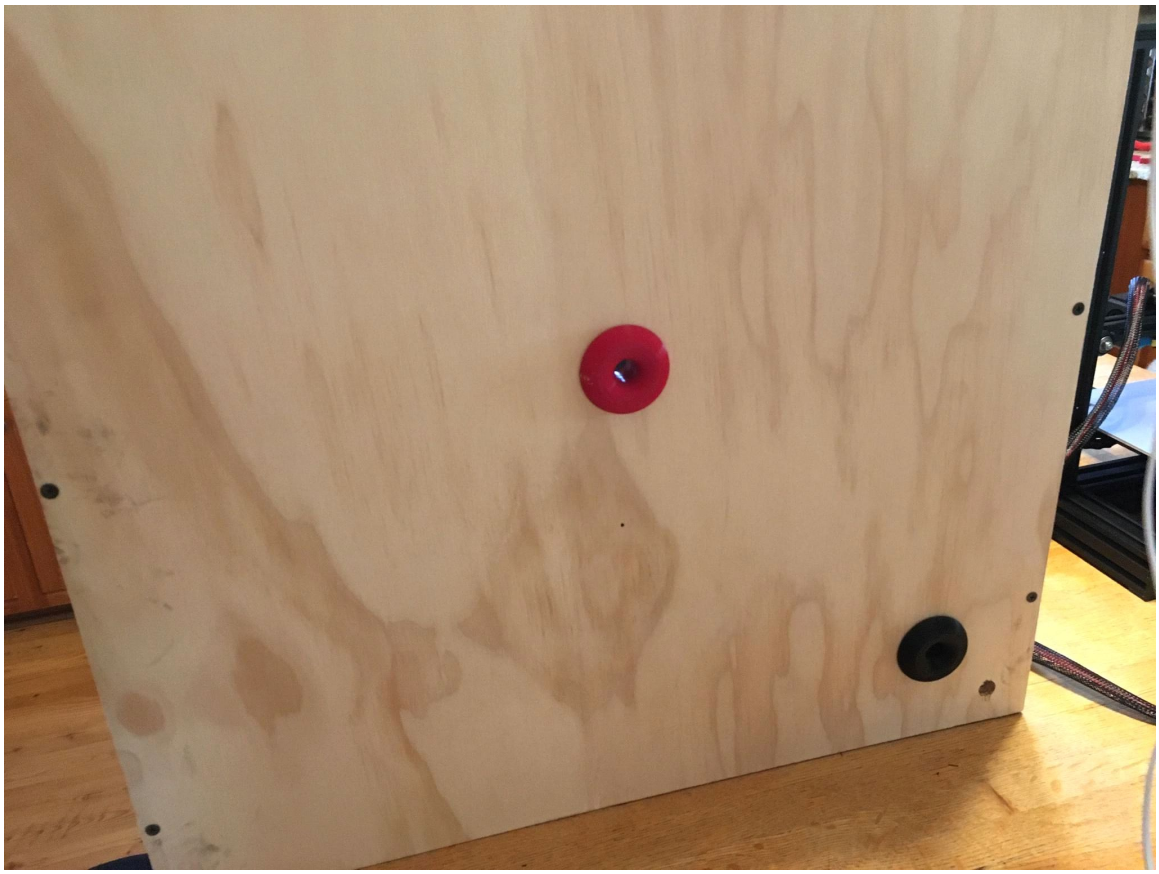
Honeywell ceramic personal heater: $18 (x2)

Inkbird ITC-308 temperature controller: $28

# Set-up Process:

Make an enclosure of your choice. I make a simple 24"x24"x24" 5 sided plywood cube. Most hardware stores will carry plywood, plexiglass, hinges, and magnetic cabinet latches. Screw

5 plywood panels together to make the cube. A 24"x24" plexiglass panel will serve as the door so you can see the magic happen. I added an LED under-cabinet light to the top to better see my prints in action. Drill holes to allow the wires for the printer, heater, and one for the filament. I made some bulkhead grommets that worked well with my plywood. These grommets should stop stress points where the wires go through the housing. I screwed the stock filament holder on the side to allow it to feed straight into the extruder. We are going to be heating a tiny box with cheap heaters made in China, and using a cheap printer made in China, so you may want to add some fire protection. Stovetop Firestop makes canisters that, once exposed to flame, will pop open and dump fire extinguishing dust all over the area. For $50/pair, it's cheap insurance.

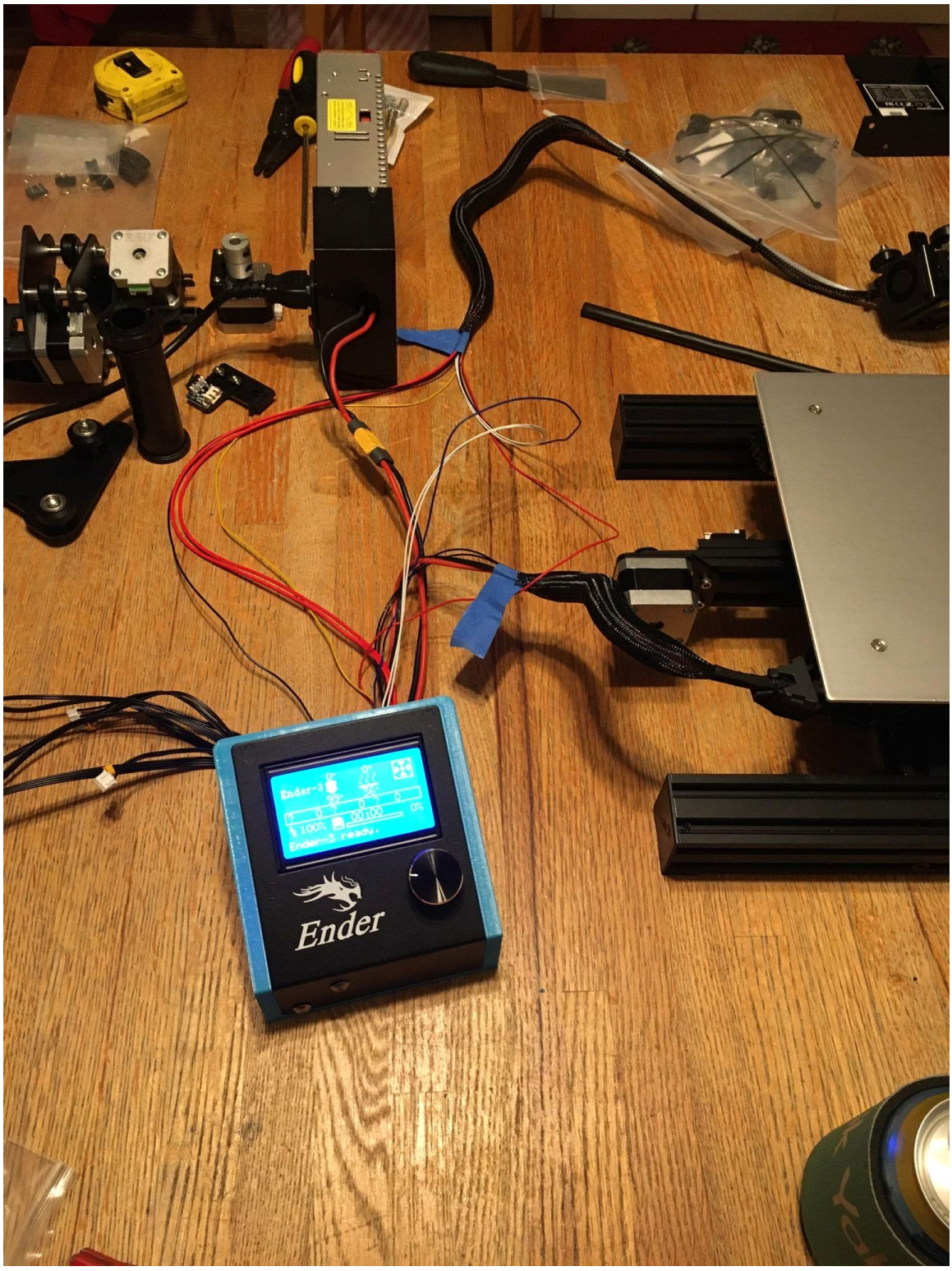


Install the hot end and bowden tube according to the manufacturer's instructions.

They're both pretty much a direct replacement of the originals.

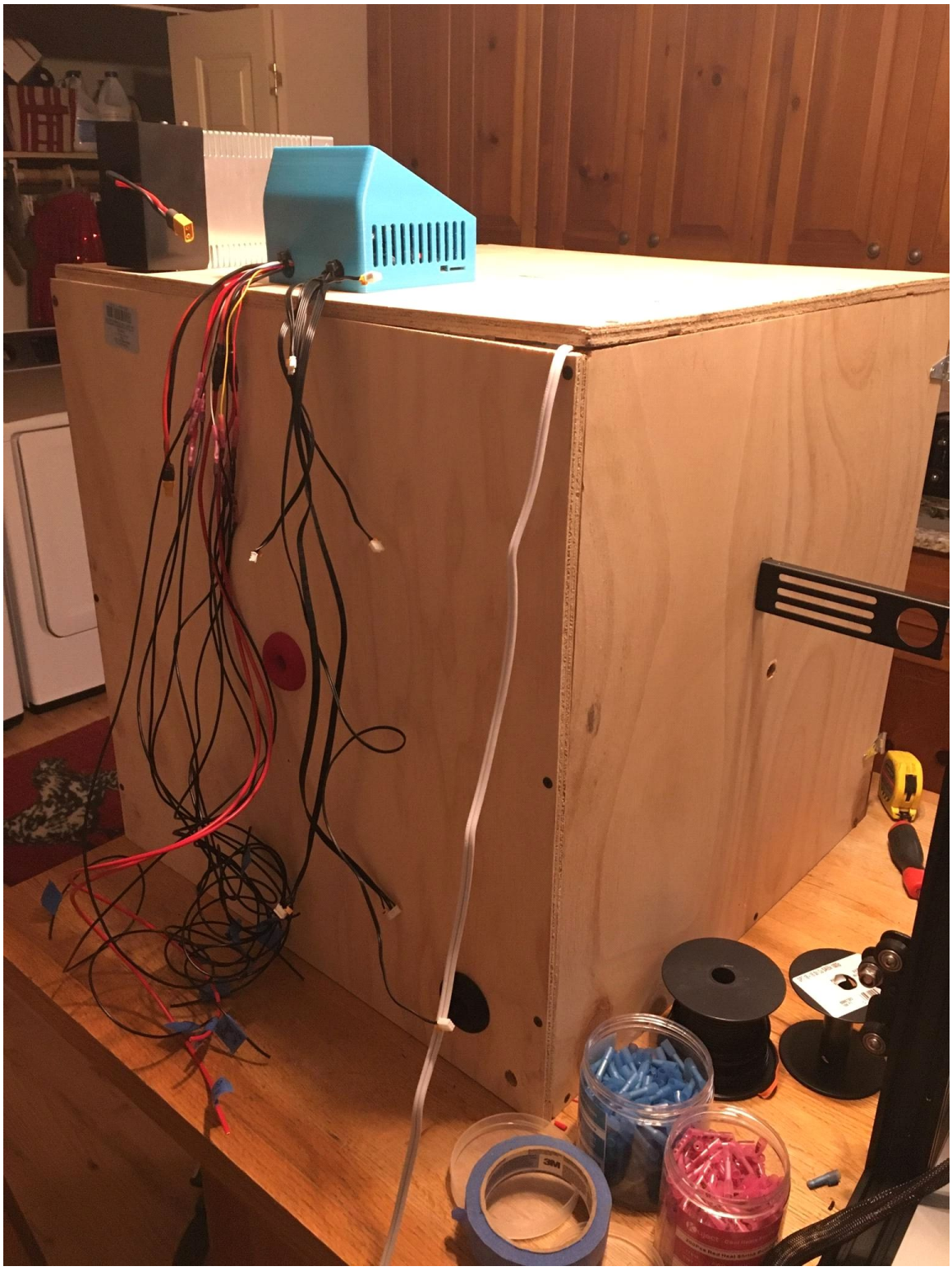
Now comes the fun part of extending the wiring. You can either install a pre-made kit or DIY it. I'll be going over the DIY method here. I used an enclosure off of Thingiverse and modified it to suit our purposes better. This will house the motherboard and screen in an attractive package and stop it from getting broken. (Thing # 3631413) You may choose to use the one included in this guide or use one of the original versions. I placed the enclosure and power supply on top of a little stand on top of the enclosure, and then lengthened the wires to reach the printer inside the enclosure. I used one hole for the wires on the hot end/extruder, and another hole for the wires on the bottom printer and bed wires.

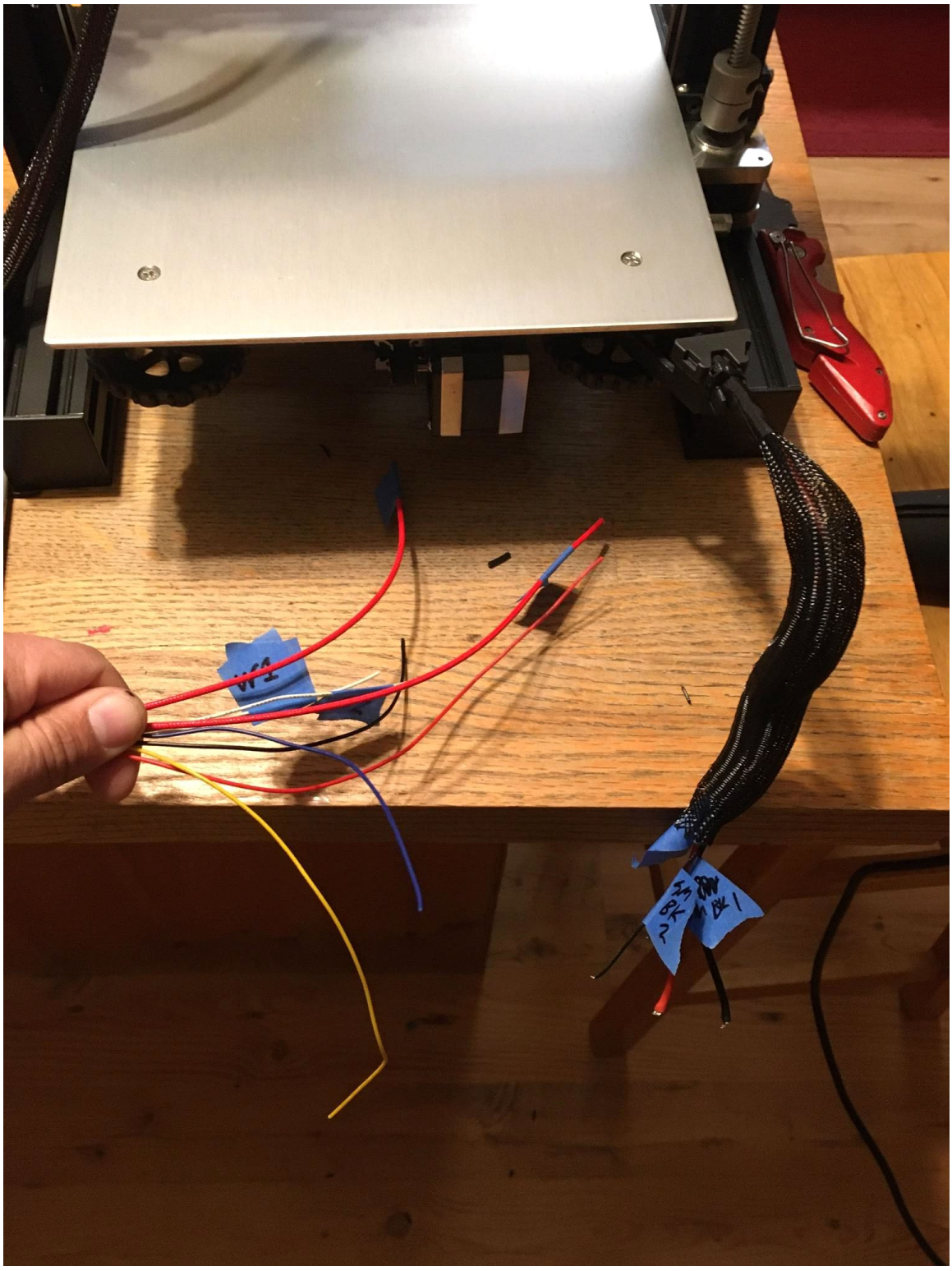
First put the electronics in the housing and plug everything up to ensure it works properly.

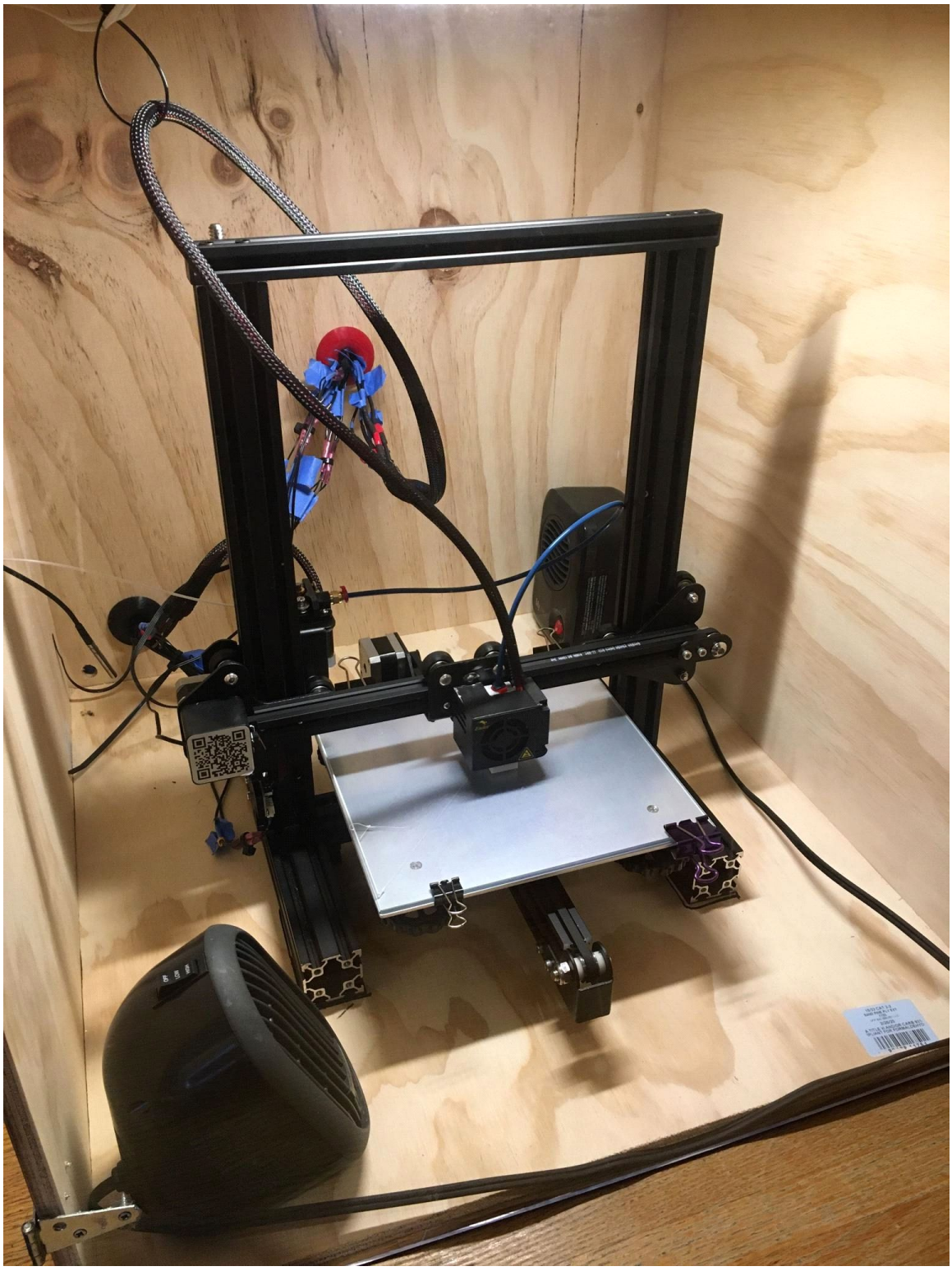



Now take the plunge and start extending wires! For the small wires you can use 18-20g

wire, for the large power wires use 12-14g wire. You can either solder the wires, or use heat shrink butt connectors. For the wires that have a removable plug, I used butt connectors. For the wires that are hard wired to the printer, such as the bed heater wires, I used male and female spade connectors to facilitate easy removal later on. Do one wire at a time, and label both sides with some masking tape. I used simple labels like "XS1 and XS2" for the two wires of the X stop switch, for example. I lengthened all wires by 36" and it worked well.

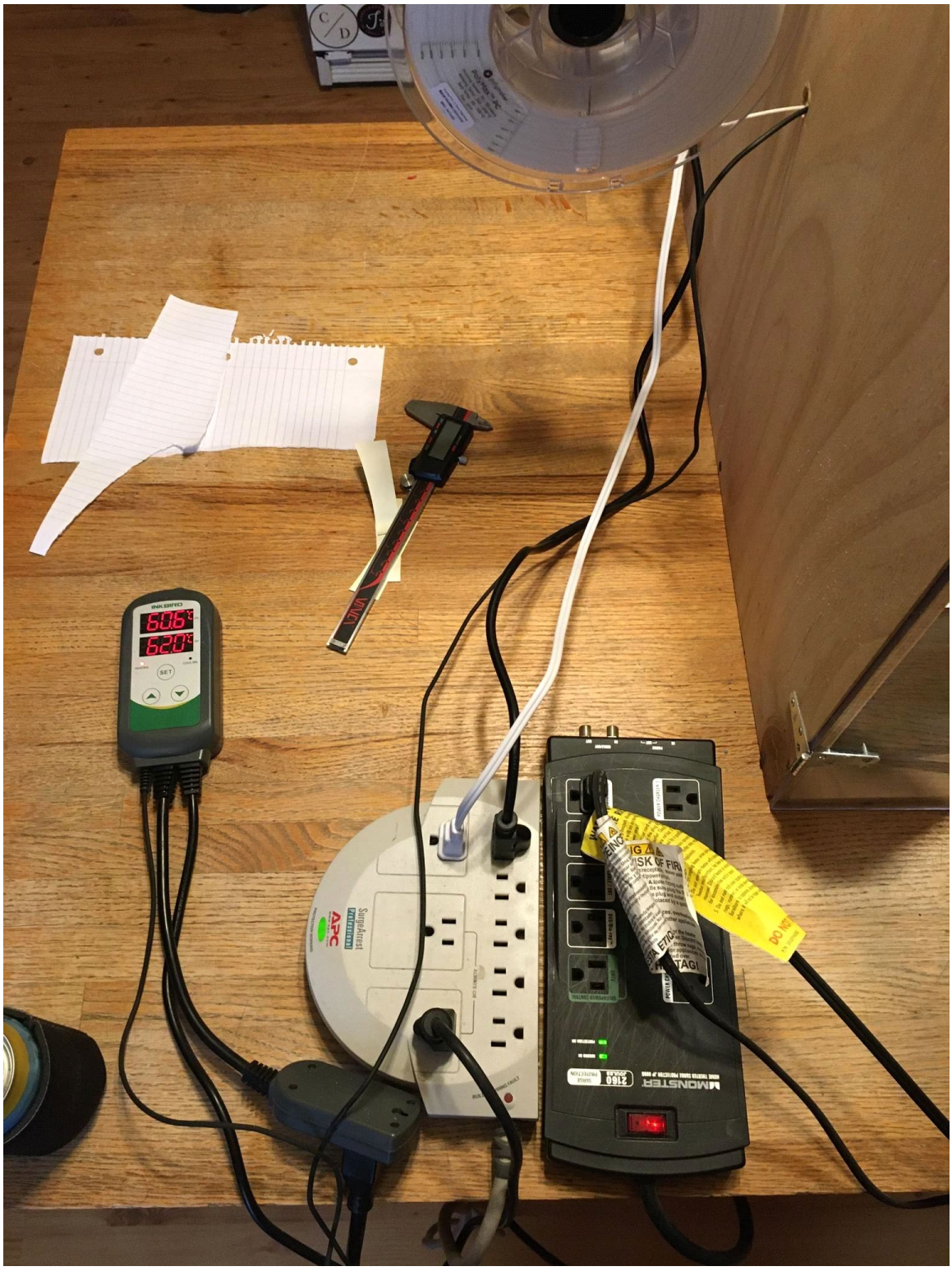






Put the personal heater(s) inside the cabinet. I aimed them along the sides of the

enclosure so it wouldn't blow directly on the print. Plug the heater(s) into the temperature controller, and controller into household power. Put the temperature probe through a small hole near the middle of the printer. I used the filament hole. You can use a small bracket to hold the temperature probe, or just let it dangle so long as the metal portion is away from drafts and not touching anything. Thingiverse has one specifically for the ITC-308. (Thing # 4030364) There are also various brackets to hang the plugs and controller itself if so desired. I originally tried using one heater with 250 watts of power and it wasn't enough to reach the desired temps. Two heaters of this size did the trick just fine though, so make sure the heater you purchase is around 500 watts, or use two 250 watt heaters in opposite corners to "swirl" the air. If using one heater, plug it into the "heat" outlet on the ITC-308, if using two, plug them into a power strip, and plug the power strip into the "heat" outlet.




Programming the ITC-308 is easy, just follow the instructions. Here are the parameters I

used:

TS - 62

HD - 1

Cd - 10

AH - 75

AL - -40

PT - 0

CA - 0.0

CF - C

You may have to adjust your temperature setting depending on the efficiency of your enclosure. When the heat reaches the desired temperature, the heaters will cut off, and the heat will begin to drop. When it reaches 1 degree below the desired temperature, (HD - 1) they will cut back on. It takes a few seconds for the heaters to catch back up and the temperature will continue to fall. See how low the temperature usually drops between cycles, and adjust the TS value accordingly. You just don't want the temperature to fall below 60* C.

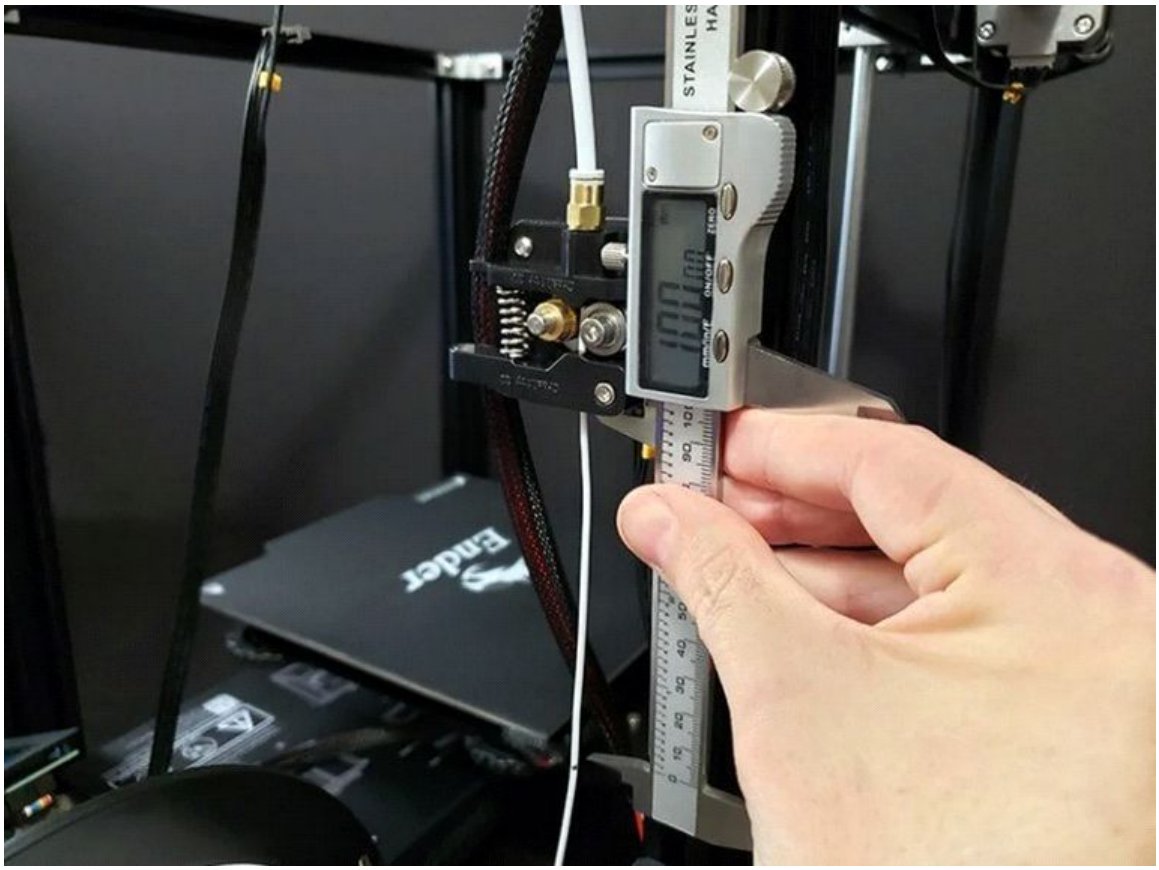
The stock Ender 3 software will only allow a hot end temperature of up to 260*. This is still too low for most PC filaments, but not all. PolyMax PC and PolyLite PC by PolyMaker are designed for a print range of 250-270*, perfect for your Ender 3. This guide will focus on PolyMax, but PolyLite offers a more economical approach if money is an issue. PolyLite has a lower impact resistance than PolyMax. The maximum bed temperature of the Ender 3 (110*) also worked perfect for all my tests. You will have to remove the bed, clean it with hot soapy water, and reapply glue stick in between prints. I used the normal purple glue stick that turns clear as it dries. It washes off nicely with hot water. If you print a calibration cube, take it off and then immediately print another, you'll have adhesion issues. Maybe if you move the model around in Cura you can get away with not redoing the bed every time.

# Calibrate the printer for your new filament:

1) Level your bed. If you're trying to print advanced filaments such as PC, you should be familiar with this process already. Getting PC to stick means a level bed is a must. I like using the bed level Gcode included in this package. Just click the scroll wheel and it moves to the next point.

2) Adjust your E-steps. PC might feed differently than your normal filament. Before starting, make sure that you're able to navigate to "Control" > "Motion" on whichever Ender 3 you have. Some people have noted that this option is missing, perhaps due to the status or version of their firmware.

- Measure 100mm from a set point on your extruder and mark it. Mark an extra 10mm in front of, and behind this 100mm marking to make measuring easier later on.




- Heat up your hot end and extrude 100 mm. (Put it all the way to 260* to ensure free flow.) You can do this by clicking "Prepare" > "Move axis" > "Move 10 mm" and then slowly rotating the knob until you arrive at 100 mm.

- Wait until the printer has finished extruding and measure how far from the 100-mm mark the filament stopped. If the mark has passed the extruder, then your extruder is over-extruding, and if it hasn't reached the mark, then it's under-extruding.

- Use the marks above and under the 100mm mark in order to estimate the amount of filament extruded.

- Now calculate the correct E-steps by multiplying 100 with the current E-steps, then divide this by how much your extruder extruded. The resulting value will be your new, correct E-step value.

- Click “Control” > “Motion”, then scroll down to “E-steps/mm” and change it to your new E-step value.

- Click "store settings" or else when you power off it will revert to the old settings. You should hear a confirmation beep.

3) Slicer settings: I've included my Cura profile in this package, (cfg file) but you will obviously have to change it slightly to match your setup. I purchased a bone stock Ender 3 for this write-up and set the profile up to work with that, so it's a good starting point. The main points are:

- Cooling fan off. A slight amount of fan can make the parts look much better and improve overhangs considerably, but even as little fan as 15% speed can reduce layer adhesion by up to 30%!

- Minimum layer time: 20 seconds

- Print speed: no more than 50 mm/s

- Temps: Max that Ender out! 260* hot end, 110* bed temp.

- Bed adhesion: I liked using a 5mm brim to ensure edges wouldn't start peeling up. Depending on the model, you may want to use a raft or nothing at all. For example, the calibration cubes will do fine without anything, but when I printed AR style magazines, I used a brim.

4) Calculate your flow rate. Your slicer has a flow percentage setting that is probably at 100% unless you've changed it before. I like to use a simple cube that takes about 15-20 min to print. This also gives you a good opportunity to see how your bed adhesion and temperatures are working. Select the file that coincides with your nozzle size. Each cube size is named based on your nozzle size, and the wall thickness will be 2x the nozzle. Print a cube and then perform these steps:

- Let the part cool down fully! I initially made mistakes on my flow rate because I measured it hot. It changed quite a bit once fully cool. It doesn't take long due to the small size. Give it a few minutes or blow on it, just don't put it in the fridge or get it below room temperature.

- Use calipers to measure and record 2 values from all 4 sides, for a total of 8 measurements.

- Add them together and divide by eight. Use this average for next step.

- Divide .8 by the average that was previously established.

- Multiply that by 100 to give your new flow rate or extrusion multiplier.

- Print another cube and see if the measurements are spot on or closer. Don't spend all day trying to chase that .01mm, it's never going to happen.

5) Calibrate your individual axis'. Now that your flow rate and e-steps are correct, the parts you print should be dimensionally accurate, right? Probably. You're extruding the right amount of filament, but is the printer moving that filament around properly? Now we will calibrate the X, Y, and Z axis steps/mm.

- Print the 20mm XYZ calibration cube

- Measure across the desired axis and apply this formula:

if:

e= expected dimension (20mm)

o = observed dimension

s = current number of steps per mm

then:

(e/o) x s = your new number of steps per mm

- Apply the new steps/mm in the same menu you did the E-steps, then store settings, making sure you get the audible beep.

- Print another cube. Again, don't spend all day chasing that final hundredth, it'll never happen.

Now you should be ready to go! Enjoy printing with one of the strongest filaments around.

Honeywell