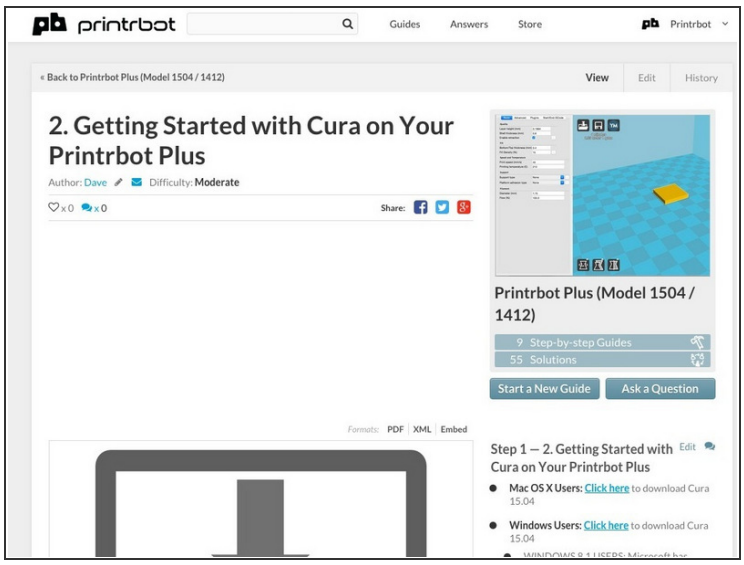
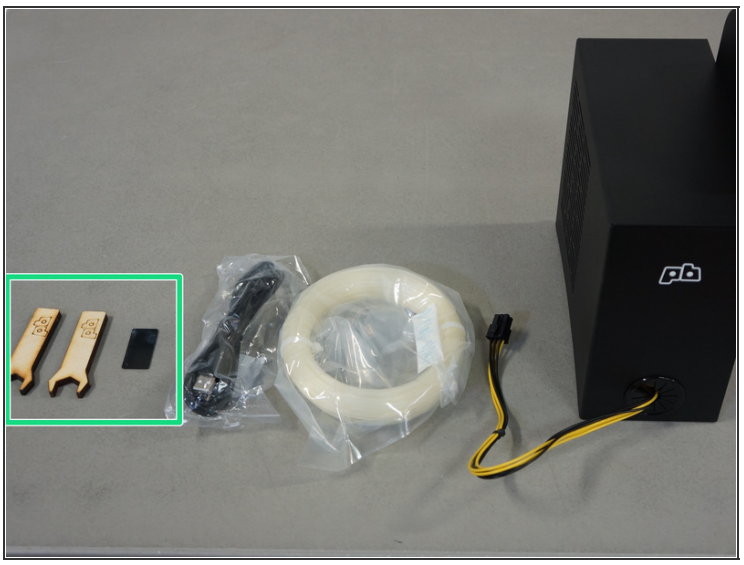
# 3. Using Cura to Set Up Your Auto-Leveling Probe and Create Your First Print with Your Printrbot Plus

Written By: Dave

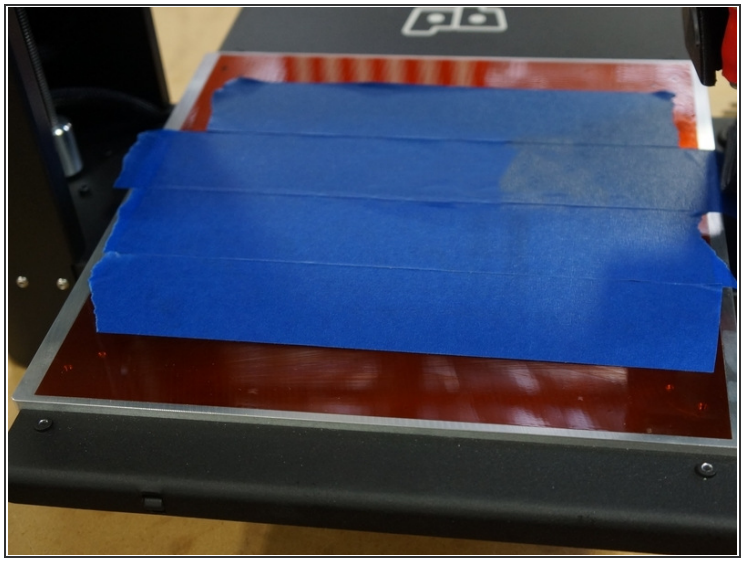
## Step 1 — PRE-CALIBRATION

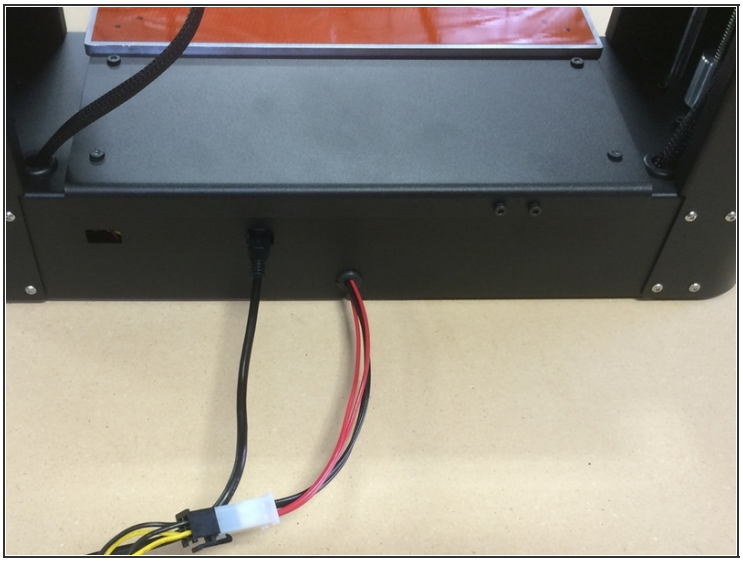


- If you have an assembled Printrbot, it was already calibrated and tested before leaving our shop. However, the Auto-Leveling Probe may need some adjustment from time to time.
- This calibration process is absolutely necessary for kit builders.
- **REQUIREMENTS:**
  - **Click here** to review "Getting Started With Cura on Your Printrbot Plus".
  - **Click here** to download the 3mm box STL . We will be using this file as the test print throughout the guide.
  - Wooden test wrenches and delrin spacer (supplied)

## Step 2 — PRE-PRINTING





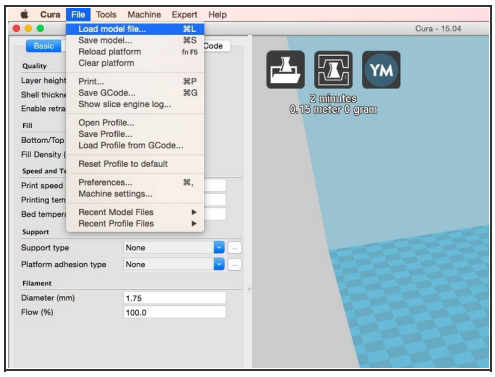
- Put down some basic painters tape to improve adhesion for the first layer of your print.

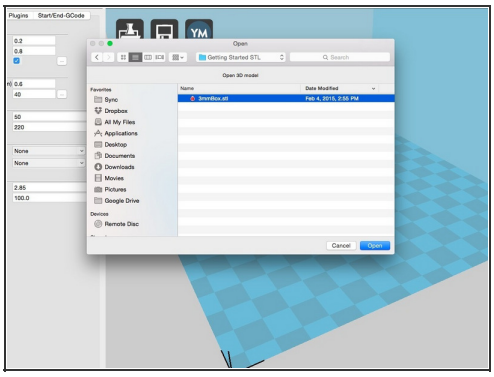
  *TIP: Most common types of painters tape will work, but if you would like to use what we use at PBHQ, click here.*

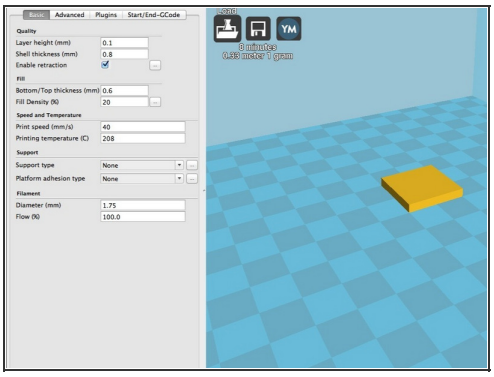
- Insert the ATX power supply and micro-USB cables to the appropriate connectors on your Plus.

- Connect the USB cable to your computer and power on your ATX supply.

## Step 3



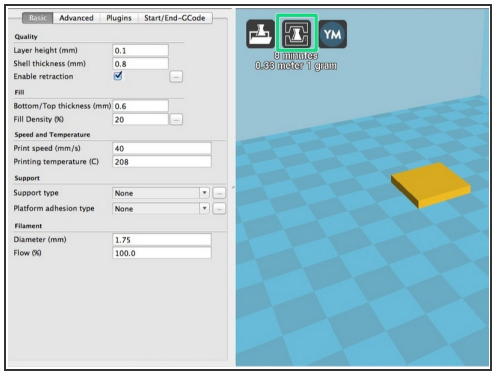



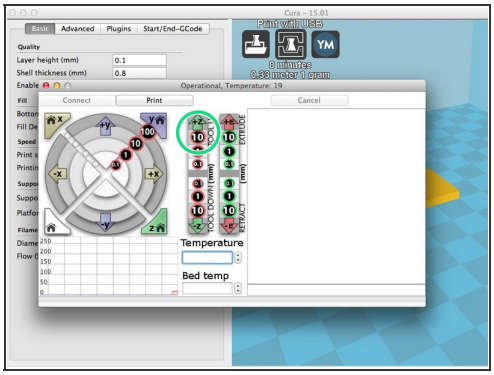



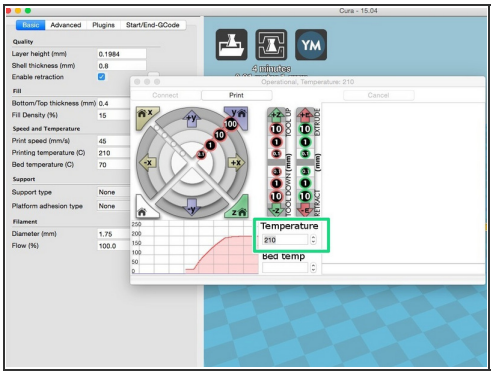

- Open Cura, and load the 3mm Box STL file by selecting "File", "Load Model File", and then opening the 3mm Box file from your hard drive.
  - The 3mm Box STL file can be found at the Printrbot Knowledge Base under "Downloads".

- The 3mm Box model will appear on the "platform" in Cura.

## Step 4 — PRONTERFACE UI (USER INTERFACE)







- To open the Pronterface UI (user interface), Click the "Print with USB" icon in the upper left hand corner of the screen.

  *Not seeing the "Print with USB" icon? Windows users, be sure that you have installed drivers for your Printrbot. See STEP 1 of the guide on "Getting Started with Cura on your Printrbot Plus" - click here. For more connection troubleshooting, click here.*

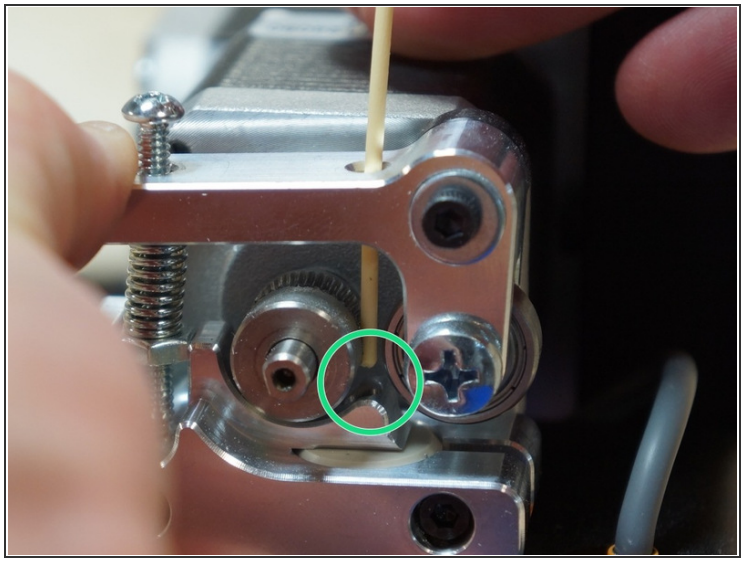
- Pronterface will appear and begin connecting to your Printrbot. The Pronterface UI will not appear unless an STL file is loaded onto the platform in Cura.

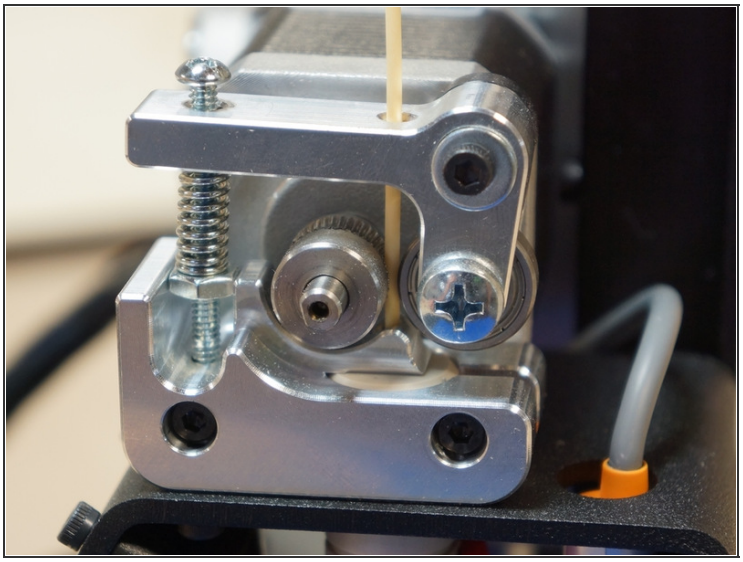
- Once connected, click "10" under the "+Z" icon three times. This will raise the Z axis of your Printrbot 30mm.

- Enter "210" in the "Temperature" command bar. Tab out of this field. This will begin to bring the hot end to 210 degrees celsius. You will see the graph reflect the rise in temperature.

  *Don't worry if you are having trouble reading the lines in Pronterface, we will zoom in on them throughout the rest of the guide. If you would like to be able to zoom in on your own screen, you may be interested in the "Zoom It" app. Apple users click here for more info. Windows users, click here.*

## Step 5 — LOADING FILAMENT

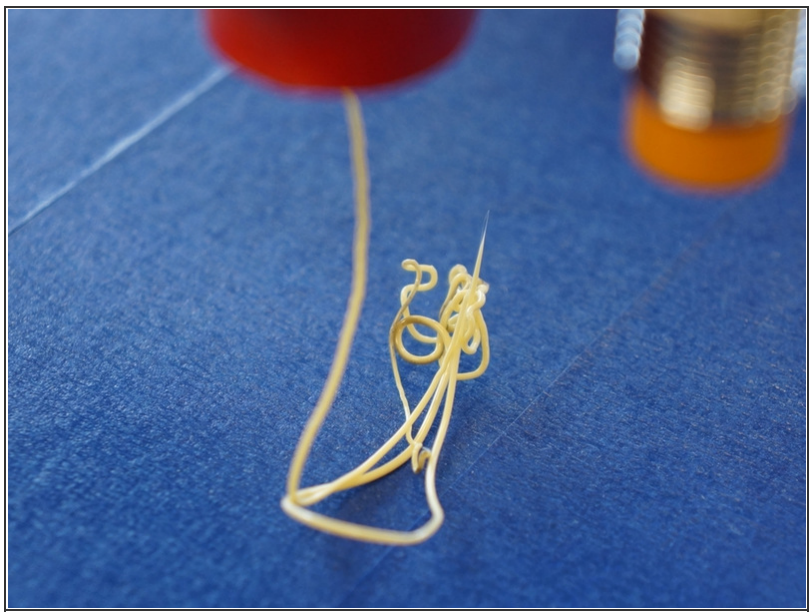




- Once the hot end is "at temp" (210C), insert the filament.
  - Compress the extruder spring, by pushing down on the extruder arm.
  - Guide the filament into the top of the extruder, past the drive gear and down the guide into the hot end.
  - Be sure that the filament sits in the teeth of the drive gear and is feeding straight into the hot end.

⚠ **BE CAREFUL - THE HOT END WILL BE HOT!**

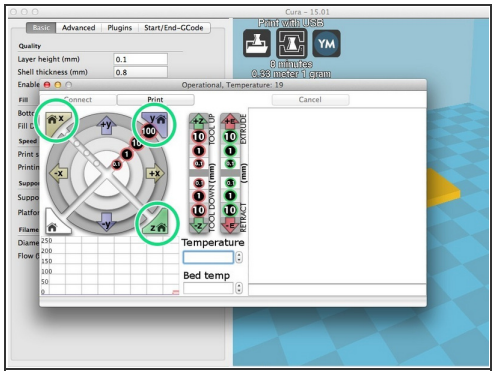
## Step 6

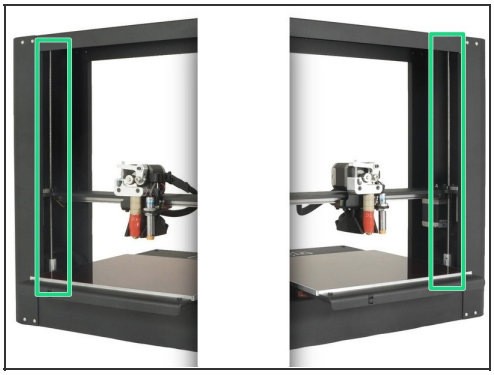
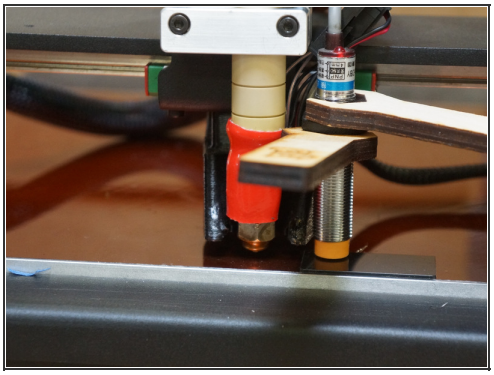


- Manually feed the filament through the hot end by compressing the extruder spring and pushing the filament down further into the extruder. The filament should feel a little "mushy" as it is fed through the hot end.

  *TIP: Do not be alarmed if the filament coming out of the hot end is a different color than what you are pushing in. We test and calibrate each assembled model. There will probably be left-over filament in the nozzle from testing at PBHQ.*

## Step 7



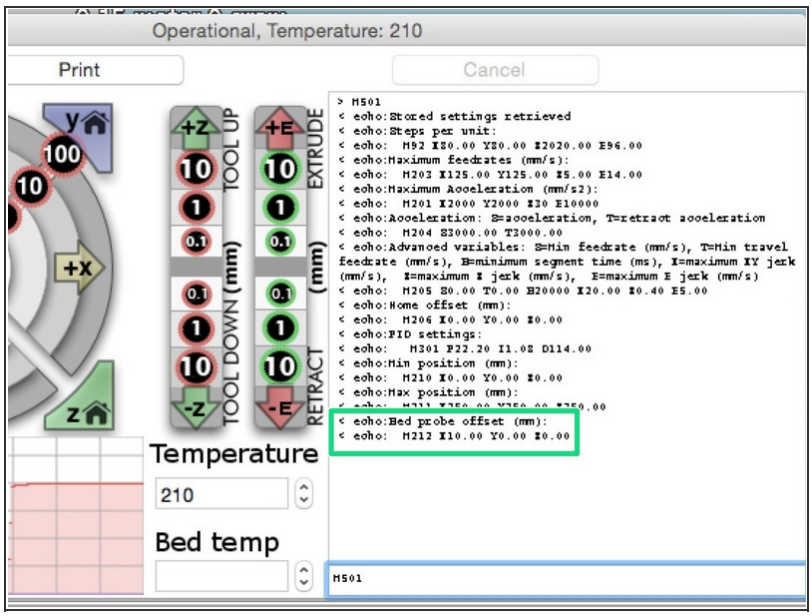
- To "home" your extruder, click the "Home X", "Home Y", and "Home Z" icons, in that order.
- Turn both threaded rods or "lead screws" (Z axis) counter clockwise until the hot end tip touches the print bed.
- Slide your delrin plastic spacer under the Auto Leveling Probe. It should fit "snug" under the probe. You may notice that the LED on the probe lights up as it nears the print bed. The light indicates that the probe is sensing metal.

⚠ **BE CAREFUL - THE HOT END WILL BE HOT**

- If you ordered an assembled unit, you will probably not need to adjust the height of your probe.
- If the spacer is not snug under the probe, you can adjust the height of the probe by turning both nuts with your wooden wrenches.

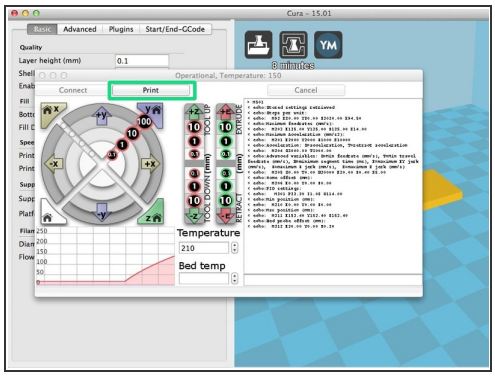
ⓘ *TIP: To raise the sensor, turn the lower nut clockwise, then turn the upper nut clockwise. To lower the sensor, turn the upper nut counter-clockwise, then turn the lower nut counter-clockwise.*

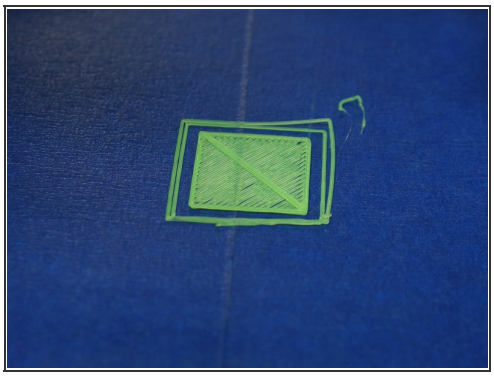
## Step 8

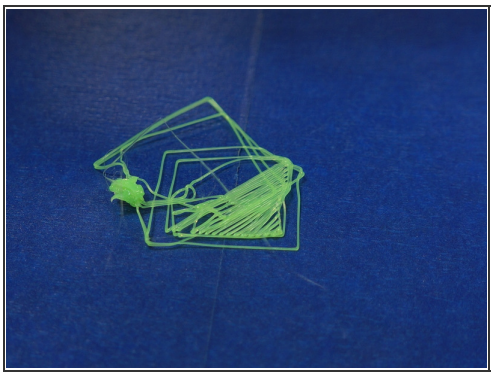



- Enter "M501" (case sensitive) into the command line. This will display the settings on your Printrbot.
- To calibrate the Auto-Leveling Probe, the "Z" value of the "M212" line will have to be adjusted. In this example, the Z value is zero.

## Step 9 — PRINT





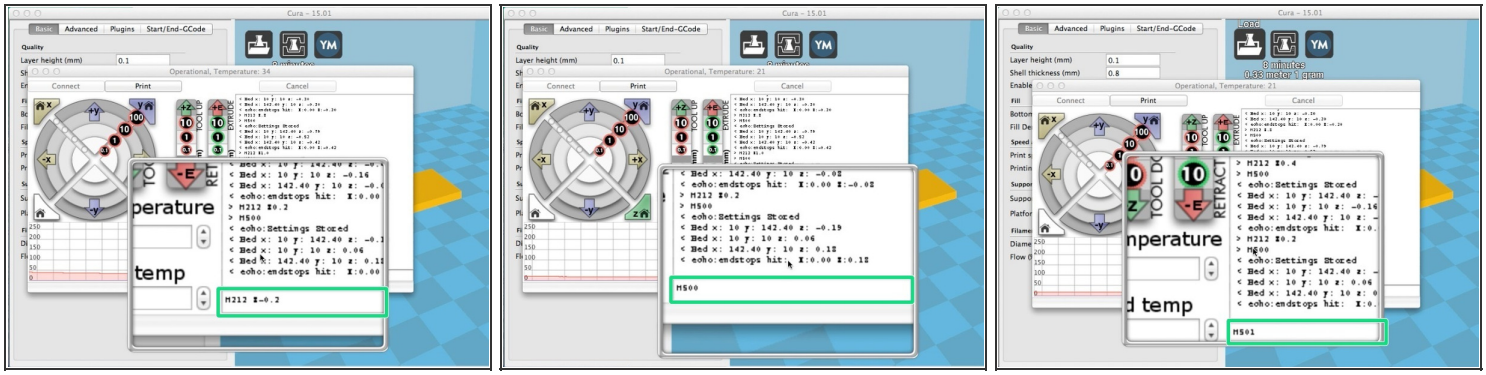
- Click "Print" and monitor your Printrbot.

⚠ **CAUTION: DO NOT LEAVE YOUR BOT RUNNING UNATTENDED**. Stay near by and monitor the machine while it is printing.

ⓘ *TIP: To reduce the risk of scraping your print bed, be ready to kill the power on your Printrbot as the print is starting.*

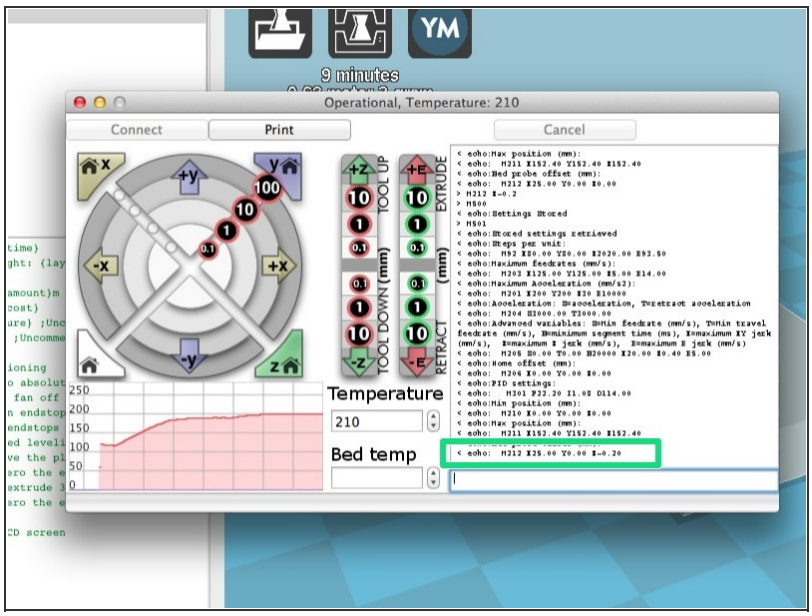
- Do not be discouraged if your first attempt goes poorly. The print will likely be stringy.

## Step 10



- Enter the new M212 values to adjust your probe offset.
  - **M212 Z-0.2**
    - *This G-code command sets the Z axis home position at negative 0.2mm.*
  - **M500**
    - *This G code command saves the new value to the Printrbot.*
  - **M501**
  - *This G code commands displays the current settings of the Printrbot.*

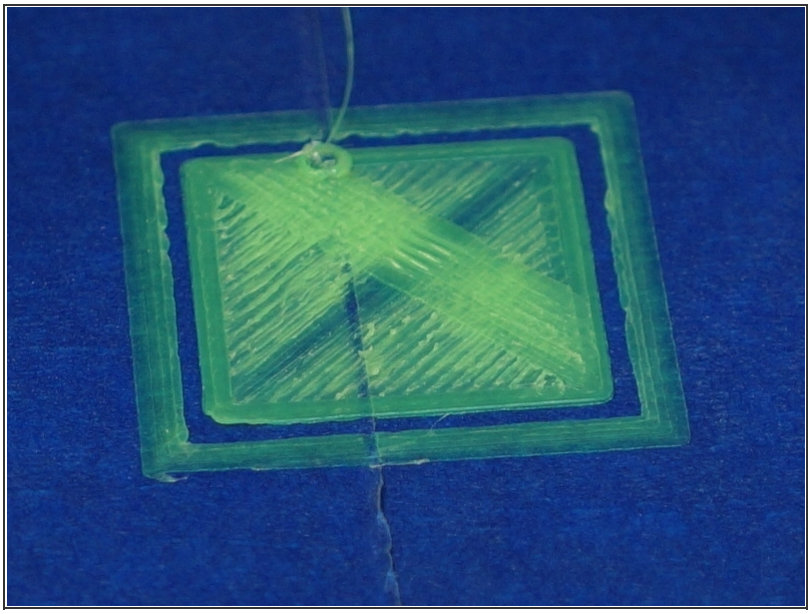
## Step 11




- M501 will display your current settings, which should be "M212 X25.00 Y0.00 Z-0.20".
- The more that you decrement the negative value, the lower the extruder will go towards the print bed before beginning the print.
  - For example, if you want the Z axis to home lower, try a value of -0.4. If you want the Z axis to home higher, try a Z value of 0.0, or a positive value of 0.2.
  - If the M212 Z value is set at 0.0 and the extruder is still printing too high above the print bed. Revisit the steps explaining how to physically adjust the probe using the delrin spacer. Your probe is not close enough in relation to the print bed.

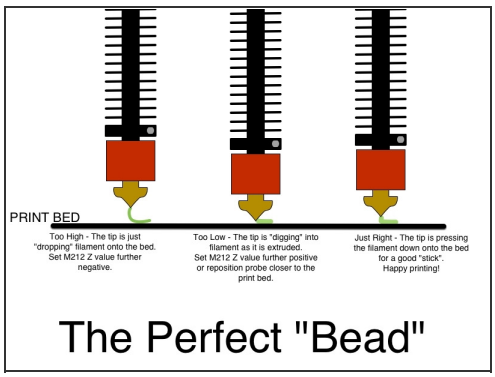
⚠ **WARNING: ADJUST THE M212 VALUE IN 0.2mm INTERVALS.** - Adjusting the M212 Z value too dramatically could result in gouging your print bed.

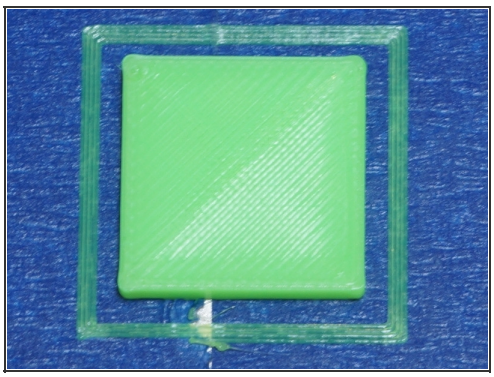
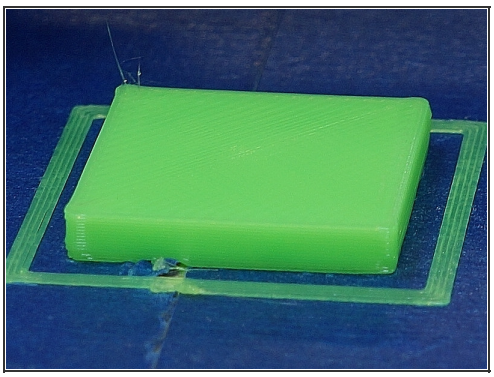
## Step 12



- If the Z offset value (M212) is set too far negative, the print layers will look flat or smashed.
- If this is the case, work the other way by giving a value closer to zero. Please note that the Z adjustments are not cumulative. If Z-0.1 does not work, try Z-0.3 etc.
- If your M212 value is set at Z0 and the layers still appear smashed, try using the wood wrenches to adjust the leveling probe closer to the bed.

## Step 13



- You are looking for the perfect "bead" as the filament extrudes. The hot end tip should just press the filament onto the bed as it extrudes. Keep experimenting with the negative Z value (M212 command) until you find the correct setting for your printer.
- It can seem like a tedious process, but be patient. The greater care you take now, the better your results in the long run.
- Great job! To remove the print, some users prefer using a putty knife to pry the print up from the bed.
- **NEXT STEPS:**
  - Visit YouMagine for more printable files available for free download.
  - Happy printing!

This document was last generated on 2016-03-10 07:40:19 PM.