# **How to Calibrate Your 3D Printer**

Before you do anything get your current EEPROM settings before you flash new firmware to the controller board by doing the following:

Backup Firmware settings:

Issue a M503 command in a g-code terminal to report your current settings, write them down, flash the firmware then do a M501 command to load the saved settings from EEPROM, then a M503 so you can see the settings and compare them to what they were before you flashed the firmware, if all settings are correct do an M500 to save them (just in case you made changes).

URL on EEPROM settings:

<https://github.com/MarlinFirmware/Marlin/wiki/EEPROM>

NOTE: PLEASE, PLEASE do a M503 in a terminal window (via Octoprint or pronterface) so you can document your EEPROM settings (all your calibrations you have done up to this point). Sometimes when you flash new firmware you will get an EEPROM version mismatch error and to clear it you have to do "M502 and M500 then M501 is normally enough to clear the eeprom and write a new version of everything including the version number" {quote from <https://github.com/MarlinFirmware/Marlin/issues/12860>, which sets all your EEPROM values back to the Marlin default settings and you lose all your calibrations.

**VS code Editor**

Use VScode to make changes to the Marlin V2.0 firmware or the firmware preference of your choice.

Here is a great video on how to setup VScode so you can use Marlin (or your preferred firmware) on your Ender 3:

Link to YouTube videos to install VScode for Marlin firmware:

<https://youtu.be/W6zYvRgGr3Q>

<https://youtu.be/7Xr9Eb7o0rw>

URL: <https://youtu.be/U8_ldMckGDE>

and starting at 4:16 <https://youtu.be/GNGN2iSQ5j4>

USE VScode to make changes to firmware and compile to get your firmware.bin file.

You can use VScode to upload the firmware automatically by attaching a USB cable to the front of your 3D printer or you can use OctoPrint “Firmware Updater” to flash the firmware.bin file.

**OctoPrint Firmware Updater:**

Firmware Updater Plugin:

On the raspberry pi running OctoPrint, you must perform the following Linux commands:

$sudo apt-get install avrdude

$whereis avrdude

URL for OctoPrint Firmware Updater:

<https://github.com/OctoPrint/OctoPrint-FirmwareUpdater>

\*\*\*\*Note: if you turn on the printer and get a blank blue screen it means no firmware has been loaded.

\*\*\*\*\*\*\*\*Notes: if VScode has problems, try disconnecting the USB cable , exit VScode, install USB cable ,open VScode, wait for VScode to load , select Platformio from side Menu, select Project tasks, select Rebuild Intellisense Index” then try to recompile or upload again

**3D Printing Calibrations**

1. Calibrate your extruder: only needs to be done if something changes or you reload the firmware , follow [Teaching Tech](https://www.youtube.com/channel/UCbgBDBrwsikmtoLqtpc59Bw) video: start at 4:38 <https://youtu.be/VyDv0CnLWU0> or
   1. M503, write down M92 command’s E value, my E is 98.86
   2. M83
   3. G1 150
   4. G1 E100
2. Calibrate your X,Y,Z axis: only needs to be done if something changes or you reload the firmware, use the following URL to calibrate X,Y,Z axis or follow my steps below <https://all3dp.com/2/how-to-calibrate-a-3d-printer-simply-explained/>
   1. So, print a 20mmx20mm cube XYZ (<https://www.thingiverse.com/thing:1278865>) cube. Take your calipers and measure the X-Axis length (8 different times), take the average of the X-Axis length then 20mm\*(X-Axis Steps/mm from M92[do a M503])/average of X-Axis length that you measured = new X-Axis Steps/mm value. Now do an M92 Xii.ii in terminal window, followed by an M500 to save new value to EEPROM (ii.ii are your new values). Repeat until happy.
   2. So, from the print of the XYZ cube, take Y-Axis measurement 8 times. Take the average of the Y-Axis length. Now take 20mm\*(Y-Axis Steps/mm from M92)/ measured average of Y-Axis length = new Y-Axis Steps/mm value. Now do an M92 Yii.ii in terminal window, followed by an M500 to save new value to EEPROM. Repeat until happy.
   3. So, from the print of the XYZ cube, take Z-Axis measurement 8 times. Take the average of the Z-Axis length. Now take 20mm\*(Z-Axis Steps/mm from M92)/ measured average of Z-Axis length = new Z-Axis Steps/mm value. Now do an M92 Zii.ii in terminal window, followed by an M500 to save new value to EEPROM. Repeat until happy.
   4. So, now that you have done all axes, do an M503 to see if your new values have been saved to EEPROM.
3. Calibrate you filament, do this every print. Use a caliper, measure you filament diameter at several locations. Average out the measurements (at least 3) and enter that into your slicer under filament diameter.
4. Calibrate your z-height and first layer. Do this whenever something changes in your printer. First write down the First Layer Height percentage and First Layer Width percentage you're currently using in your slicer settings (so you can change them back after you finish this calibration), also do a M503 in terminal window (if you are using an ABL) so you can write down your M851 values for X, Y, and Z(this is your Z probe offset value when using ABL), if you do not use an ABL then you will have to manually move your Z end stop:
   1. Print a single layer (say 20\*20mm cube) with your first layer at 100% height and width.
   2. Using a caliper measure the print in several places (at least 8) and adjust your bed (no ABL) or adjust the z offset with G-code command (if you have Automatic Bed Leveling or ABL) by using M851 Z-xx. In the M851 the Z number should be a negative number. Use M500 to save and use M503 to see if the change occurred.
      1. You measure the height of the print in 8 different places in order to find out if your z height is set correctly. Take your print height (for me 0.2mm) - the average print height measured=adjustment for the Z offset or amount to manually move up the build bed. A negative z height means the nozzle is too far away from the build surface. If you get a negative z height move the nozzle closer to the bed by adjusting Probe Z offset if using auto bed leveling (use M851). If not using ABL just level your bed manually and bring the nozzle closer to the bed before setting your z end stop.
   3. Repeat until you are happy.
5. Calibrate your extrusion multiplier! Do this every print, to be a little lazy every roll, to be really lazy every brand of material.
   1. Print out a cube (20\*20mm)
   2. In vase mode (single outline, no infill, no top or bottom layers)
   3. Set your extrusion multiplier to 1
   4. Set your extrusion width to be equal to: In Simplify3D use “auto” which defaults to 0.48mm extrusion width; in other slicers use 0.5mm as your extrusion width.
      1. You do not want to use 0.4mm extrusion width for 0.4mm nozzle because the 3D printer slicer does not like that thin of a wall. Here is a quote from “[Maker’s Muse](https://www.youtube.com/user/TheMakersMuse) 50 3D Printing Tips 2017” (<https://gumroad.com/l/QWAh>) I quote “ The thinnest you can realistically reproduce on a single section in a hobby FDM 3D print is the thickness of a single extrusion width, which with a 0.4mm diameter nozzle will be about 0.48mm wide. However, in practice, these thin details will be VERY weak, as 3D Printing does not carry the same level of strength as injection or blow molded plastic.

* In practice, I would recommend the thinnest you go in your model is around 1.5mm, and any stand-alone detail such as vertical columns should be at least 2.5mm thick to avoid breaking and bending during the print.
* If you want to test your 3D Printer's capability in reproducing thin wall details, check out our video covering how using the Maker's Muse thin wall test file <https://youtu.be/uUcK2QiX2NE> “.
  1. Print the cube, Measure the walls with a caliper in several places (at least 8) and get an average. If you used Simplify3D then you are looking for measurements close to 0.48mm for other slicers you want 0.5mm..
  2. Change your extrusion multiplier: new multiplier = old multiplier x (extrusion width/ average measurement). For Simplify3D: new extrusion multiplier = 1.0\*(0.48/measured average wall width). For other slicers: new extrusion multiplier = 1.0\*(0.5/measured average wall width).
  3. Repeat until you are happy.

1. PID tune your hotend and your heated Bed (do this when... just like #4 item). Before and after you calibrate temperatures and any time you change a fan or move something or season changes. PID tune your hotend and bed to keep your temperature fluctuations to a minimum. To use M304 PID tune for Heated Bed you need to enable it in Marlin firmware (uncomment PIDTEMPBED):
   1. Note current PID values M503.
   2. M303 E (Extruder 0 for hotend, -1 for bed) C (#of cycles 3-8) S (Desired Temperature).
   3. M303 E0 C8 S210= PLA Hotend tune for 8 cycles.
   4. Enter new HOTEND values in terminal M301 Pxx.xx Ixx.xx Dxx.xx (Kp is new P, Ki is new I, Kd is new D. Use M301 for Hotend ...USE M304 for Heated Bed!!
   5. M303 E-1 C8 S60 = PLA heated BED.
   6. For HEATED bed new values enter in terminal M304 Px Ix Dx
   7. M500 save to EEPROM.
   8. M503 check values are saved.
2. Calibrate your temperatures: do this for every different filament (color, brand, material, etc.).
   1. Grab a temperature calibration tower off Thingiverse. <https://www.thingiverse.com/thing:915435>
   2. 2. Set the temperature range to the range listed on your filament or by manufacturer.
   3. 3. Print the calibration tower and choose the best temp.
3. Calibrate your fan speed. (I have a Delta with three powerful layer fans. If they all run at 100% the hotend loses temp)
   1. Print your calibration tower again but change the fan speed vs the extruder temp.
   2. Choose the best fan speed for the finish you desire.
4. PID tune your hotend again with the layer fan set to the ideal speed. As noted above if your fans in any way change the tempting your hotend (all layer fans do) you should PID tune the hotend with the fan set at the most common speed to keep temperature fluctuations to a minimum.
5. More calibration tests that can be run to tune your slicer settings: “Ender 3 Pro: Initial Setup and Recommended Prints”: URL: <https://www.instructables.com/id/Ender-3-Pro-Initial-Setup-and-Recommended-Prints/> and [Teaching Tech’s](https://www.youtube.com/channel/UCbgBDBrwsikmtoLqtpc59Bw) (speed test) <https://www.thingiverse.com/thing:3071464> and <https://youtu.be/3yIebnVjADM>.
6. Calibrating for retraction speed. I started with this profile from Angus’ Ender 3 profile. URL: <https://www.youtube.com/watch?v=_QRb54zVPfQ&t=24s>. From Anugus’ above video on YouTube channel [Maker’sMuse](https://www.youtube.com/user/TheMakersMuse):
   1. Angus’ new Simplify3D and ideaMaker profiles for the Ender 3 (will be updated as they are tweaked and improved - enter '0' to download for free):
   2. <https://www.youtube.com/redirect?v=_QRb54zVPfQ&redir_token=M3FGqLpxhsXYbVoJylEN30LvQHp8MTU3NTQwMDE3NUAxNTc1MzEzNzc1&event=video_description&q=https%3A%2F%2Fgum.co%2FZrffZ>
   3. I started with this profile and changed its Extrusion Multiplier (you calculated thus in STEP 5). I used his default print speed because my printer can print at 90mm/sec from step 10 ([Teaching Tech](https://www.youtube.com/channel/UCbgBDBrwsikmtoLqtpc59Bw). Speed test). So I ONLY changed my extrusion multiplier all other setting I used and to my pleasant surprise I ended up with NO STRINGS!!
   4. I reran the XYZ calibration cube just to ensure everything was correct.
   5. After this I considered my profile calibrated. I hope this helps others.