

Auto Bed Leveling and Z-offset on a Tronxy XY-2 pro (the right way)

As written by Jeff Victor

Setup

This will be a step by step walk through of how to properly setup your XY-2 Pro, and how to avoid Z offset issues. Some of these steps may seem complicated, but I assure you it is much easier than you think.

Before we get started, I highly recommend you download Notepad ++. It's a great text editor that won't leave your config files corrupt, like Word or Wordpad can.

<https://notepad-plus-plus.org/downloads/>

Step 1

Download my "Savesettings.gcode" file from the following link.

<https://www.facebook.com/groups/884226342431100/permalink/884234792430255/>

(updated link)

This file, when loaded and run from your SD card, will save a copy of your machines current configuration, to a file on your SD called "currentconfig.gcode".

This will "Run" and not stop.

You will have to manually stop this after 25-30 seconds, as by this time the write to the card should have finished.

Save this file somewhere safe, this is your backup in case something goes wrong.

Step 2

Download and run my XY-2 ABL Tweak file

<https://www.facebook.com/groups/884226342431100/permalink/884234932430241/>

(updated link)

Tronxy has a horrible tendency to send their machines out with a bad firmware configuration. This tweak file recalibrates the location settings for the Auto Bed Leveling mesh, and sets up the machine to record the actual nozzle locations.

In the original config, the sensor will probe a location of the bed that is roughly 50mm away from where the machine thinks the nozzle will be. This, in my opinion, is horrible and wildly inaccurate. If anyone would like me to make a detailed post about why/how/who/what/where/when it works.. I will gladly do so. otherwise, just believe in the Magic!

Reboot your machine at this point, otherwise homing may fail.

Step 3

From your touch screen, Go into the Tools Menu and select Manual.

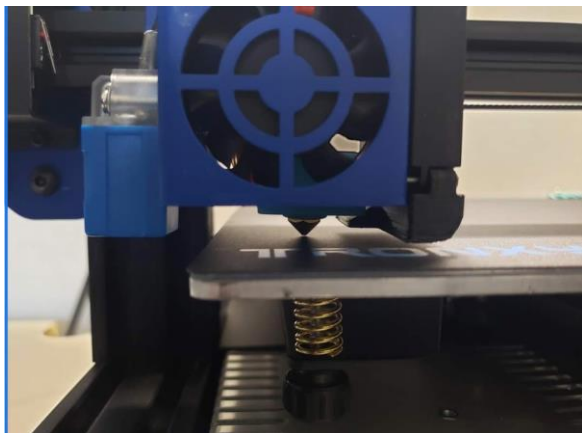
Click on the "Home" button and allow the machine to run its "Auto-Home" process. At this point we are not terribly concerned about how level your bed is. Attempting even to do a manual leveling at this point will just make your situation worse than it needs to be.

Once the machine has gone to "home" you will see that the sensor itself is about 40mm away from the left edge of the bed. You want to use the manual move buttons to set the print head to X0 Y0. This will be where the head and bed are both touching the end stops.

Now move the print head through the manual menu, to X40 Y40. This is not, or should not, be the same position you were in after homing.

Once you are here, back out of the manual movement menu, back to the tools menu, and hit the STOP button. This will disable your steppers.

Manually rotate the Z axis lead screw until the nozzle nearly touches the print bed. You can use a piece of paper here to feel for "drag" while setting the Z height.



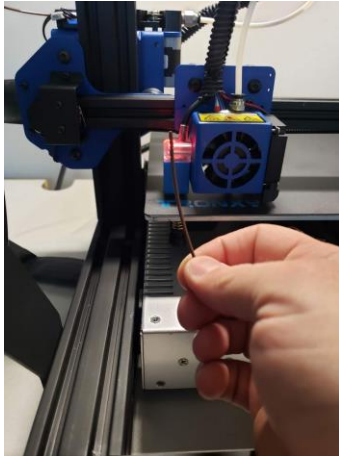
Step 4

Go back into the Manual menu and move the head to X80 Y45. You can verify this location in one of the information menus, although its not critical to be perfectly accurate at this point. You nozzle should still be clearing the bed. If it is not, tighten the screw on the right front of the bed to prevent damage to the print surface, but not too much, as to create a gap below the nozzle.

You will be setting your sensor position in this step.

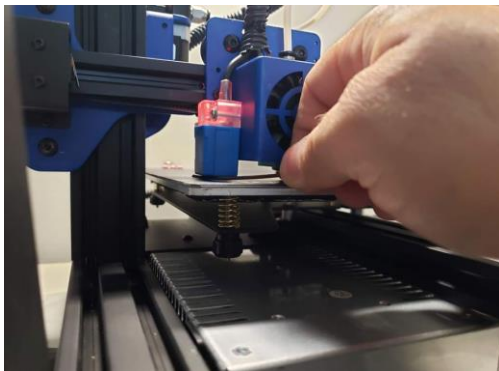
Loosen the two screws on your bed leveling sensor.

Using a piece of filament as a gauge, move the bed leveling sensor until it rests on top of the piece of filament.



Step 5

Once you have the sensor height set, tighten the screws holding it in place. Be sure to keep the sensor as level as possible with relation to the bed. Before you move on from this step, ensure the red LED is still illuminated on the sensor.



Step 6

From the "Tools" menu, press the Z offset button. Allow the machine to home and then move to the center of the bed. Once the head stops, DO NOT MOVE THE HEAD!!!!

Simply press the button to set Z to zero. We want the Z axis offset to have no value at this point.

After you have done this. Run the "save settings.gcode" file as described in the first comment and its link.

Step 7

Move your SD card to your PC and open currentconfig.gcode in notepad/notepad++.

Scroll down to M8084 and look for the entry that shows a Z value.
My example here reads M8084 Z2.194946
Change the value to Z0.0
Save this file.

What we have done here is ensured that when you start your first test print, the nozzle will not bury itself into the bed. I would much rather have you deal with some filament spaghetti, than destroying your print surface.

Put the SD card back in your machine and "print" currentconfig.gcode.
Once the file completes, press OK and close out to the main menu.

REBOOT YOUR MACHINE!!! This is very important. Do not proceed unless you have turned off the power to the machine, and turned it back in.

Step 8

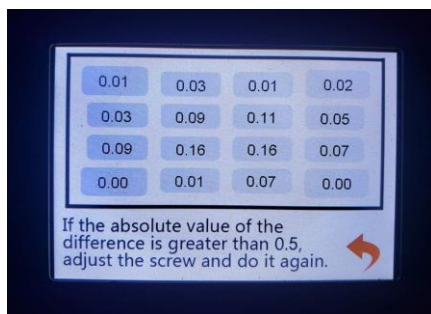
Now we can begin to calibrate the bed level of the machine.

Once you have powered the machine on, pre-heat the print bed to the temperature you normally print at. For PLA I like to be between 45-60°C.

Once the bed has reached its temperature, run the auto bed leveling function.

Once the machine has completed its passes, **ONLY** look at the four corner values. Ignore everything else but the corners. Your left front corner will always be zero. Adjust the other four corners until you have them relatively close to the value of the left front. I personally try to have mine closer than 0.2mm.

Every time you adjust the knobs, you must run the auto bed leveling function again and allow it to complete.



Step 9

Once you have your bed leveled, its time for a test print.

This will likely be the longest part of the process, but once you get the Z offset adjusted properly, you wont have to touch it again unless you dismantle the print head.

In order to get the most accurate zero possible, I like to print a test cube.

I prefer : <https://www.thingiverse.com/thing:1278865>

Open the cube and bring it into your slicer program.

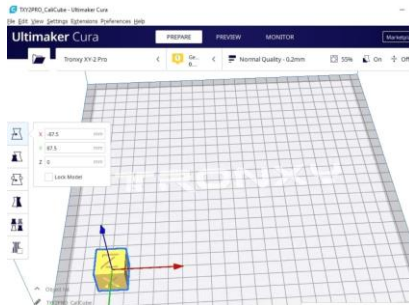
Set the cube's location to print at X40 Y40. This will center the test cube right over your

machines homing point and zero reference.

I use Cura. In order to get the cube to print at X40 Y40, you will need to set the location in your slicer to

X-87.5 Y87.5

Slice the file and load it to your SD card.



Step 10

During the initial setup, I will usually stop this print after the first layer finishes. At this point we are not concerned with the print quality, we just want to dial in the nozzle offset so we can get a perfect first layer.

I have included an infographic here from Billie Ruben's DIY 3D Printing blog

<http://diy3dprinting.blogspot.com/2019/07/bed-leveling-guide-by-billie-ruben.html>

Analyze your first layer by referencing the infographic.

You are looking for the quality of the first layer. From here, there are two different adjustments that we can make to the machine. If your first layer looks to be "too close" on this initial print, you will need to lower your bed leveling sensor and start over again.

Hopefully that is not the case. Your print SHOULD be too high off the bed, and look like pasta, almost not even sticking to the bed at this point.

Put your SD card back into your PC and change the M8084 Z value in increments of 0.15.

Your first correction should be 0.15 then 0.30, 0.45 etc. I prefer to use a .15mm adjustment because it is smaller than the "standard" initial layer height. This will reduce the likelihood of your nozzle crashing into the bed.

Continue to adjust the M8084 Z value until the printer puts down a proper first layer. Once you have the "large" adjustments down, begin changing the value by 0.05 until you are happy with the result.

Please note that the larger you make the Z offset value, the closer the nozzle will move to the bed. A very high value can cause your machine to crash the nozzle into the surface. Try to avoid this at all costs!

If your machine crashes the nozzle, do not hit "pause" or "stop". Just shut off the power to the machine. Hitting pause or stop will command the machine to move to X0 Y0, but Z won't move. So you will have a really cool scratch that runs clear across your print surface, or worse.

First layer analysis image

BED LEVELING

BY BILLIE RUBEN

Bed leveling is, quite literally, the foundation of any successful print, and one of the biggest hurdles newbies must overcome in order to avoid the dreaded plastic spaghetti.

But never fear! I've put together this handy guide to help you! ♥

INCORRECT BED LEVELING CAN CAUSE:

POOR ADHESION Making first layers hard to lay down	FAILED PRINTS Resulting in plastic spaghetti	UGLY FIRST LAYERS That have gaps or weird patterns	WARPING Ruining the accuracy of your prints
--	--	--	---

It occurs when the nozzle is not aligned properly to the bed. This is more accurately called 'tramming' (as nothing is being leveled to the ground), but 'leveling' is the common verbiage. As beds are usually leveled at 3 or 4 points, you may see any combination of the below in your first layer.

X MUCH TOO CLOSE Filament extrudes then or not at all in some places, often bulging out in others. Extruder motor might slip/click	X A LITTLE TOO CLOSE Filament bulges out of the sides of the nozzle which squashes into other lines, sometimes causing a pattern	✓ PERFECT Filament is flattened, adheres well to nearby lines, no gaps or peaks between lines	X A LITTLE TOO FAR Filament appears rounded, connection between lines is loose, gaps may appear	X MUCH TOO FAR Filament is totally rounded, little to no connection between lines, doesn't stick to bed
--	--	---	---	---

HOW TO FIX IT: LIVE LEVELING

- 1 ROUGH LEVEL, VISUALLY**
Move nozzle to lower left corner of bed, adjust leveling knobs until nozzle & bed touch, then turn it back just a bit little until a gap barely appears. Repeat for each knob on your bed.
- 2 SLICE A LEVELING HELPER**
In your slicer, resize a rectangular prism so the base is almost as large as your print bed. Give it a nice wide brim, and set the bottom layer pattern to concentric. First layer should look like a square spiral.
- 3 LIVE LEVEL**
Print it. As it's laying down the first layer adjust the leveling knobs so that the lines it's extruding look like the perfect example above, then cancel the print.
- 4 MAINTAIN IT**
Your bed is now level. To maintain the level, print all future models with a skirt (or brim) and watch it lay the brim down, adjusting leveling as required as it prints the skirt.

FURTHER HELP:

If you can't follow the instructions above it could be one of these issues:

- If you have leveled the edges but the middle is too high or low, your bed is warped, seek a replacement
- Your bed surface may need to be cleaned, or you may need to add some glue to the surface (esp. if cold)
- Your temperature may be wrong (bed usually should be around 60°C and nozzle 200 °C for PLA)
- The axis your nozzle travels on may not be level, seek instructions specific to your machine
- Your z (up/down) axis end stop may be too high or low, try to move it, or print an adapter
- Your speed may be too high. Seek a slicer settings profile from a trusted source (or use the slicer defaults)

To solve these issues you might need some further help from other printer folk. I help mod the largest 3D printing communities on discord (discord.gg/B4tp8MH) and reddit (reddit.com/r/3Dprinting). We'd be happy to help you there! ♥ Billie Ruben

Step 11

Now at this point, you have probably asked... "But Jeff, Tronxy gives us a Z Offset feature... why do you suggest not to use it?"

Well, the answer is both simple, and very complex. The bottom line is that due to a "fault" in the firmware, the Z axis offset isn't ever actually measured from a true zero point. When you hit "Home" on the machine, the print head will go to X0, Y0, and then move to X80 Y45 (when using my ABL tweak), and this places the center of the sensor at X40 Y40. When you choose the Z Offset function, the machine will read this same Z0 location, but then move to the center of the print bed. If you recall back to your bed leveling readout, is this point still 0mm... or does it vary? On my own machine, the exact center of my bed is actually about 0.12mm higher than where the initial Z0 reference was taken. So now my entire print will be inaccurate unless I try to compensate for that measurement.

Also, to add on from there, the manual moves of the Z axis while using the Z offset function

are wildly inaccurate. I have one Tronxy machine that requires 2.65mm of Z offset. I have found that if I command the machine to make two 1mm moves, six 0.1mm moves, and five 0.01mm moves, I will end up with a stored Z offset value of somewhere between 2.32mm and 2.8mm. It is almost never repeatable.

The bright side is that once you have setup the Z offset in your config, you never need to mess with it again unless you take the print head apart!

Step 12

On to my last point, and I hope this helps everyone.

STOP SETTING YOUR Z OFFSET BEFORE EVERY PRINT. Seriously, I see people doing this constantly, then blaming their machine for having a bad print. If you run the auto bed level, then change your Z offset, the EEPROM deletes the currently stored ABL mesh. So, stop it.

To print:

Pre-heat your bed to the temp you intend to print at.

Run auto bed leveling and adjust the bed if it is WAY out, again, only looking at the four outside corners.

start your print.

For more advanced users who use a removable flex bed, your level should stay fairly consistent. You can change your slicers start code to run the auto leveling for you, so you can just hit print and walk away.

Setting your start code to the following will pre-heat the bed, then run auto leveling before heating the nozzle and beginning to print.

G21; set units to mm

G90; set absolute positioning

M82; set extrusion to absolute

M107 T0; turn off fans tool 0

M140 S{material_bed_temperature}; set bed temperature

M190 S{material_bed_temperature}; wait for bed temperature

G28; auto home

G29; auto bed leveling

M104 S{material_print_temperature} T0; set nozzle temperature

M109 S{material_print_temperature} T0; wait for nozzle temperature

G92 E0; set extruder position to zero