



HowTo Calibrate, Tune and Fine Tune your printer and filament

3d printers are quite dumb as far as electronics go in the sense that there is a lot going on during a print which is assumed by the electronics. 99% of the time the electronics are assuming incorrectly.

If you want better prints, tune your printer to your filament.

Ideally you would do most of these steps for every print or every individual roll of filament. If you use a single brand you could setup profiles for each material, if you use a single brand and single material you could do this once. You should do this for every spool of filament or every print.

Step 1: Calibrate your extruder (This only needs to be done when something changes)

- Disconnect your hotend from the extruder.
- Cut the filament flush with whatever fitting you have.
- Using whatever interface you use to control your printer extrude 100mm/10cm of filament.
 - Your firmware may have a safety temperature preventing cold extrusion. If this is the case you will have to bring your hotend up to temp in order to extrude filament.
- Cut the filament and repeat this procedure two more times.
- Measure the three pieces of filament.
- Get an average (add the three measurements and divide by 300).
- If your average is not close to 100 you need to change your extruder steps per mm. Take the current steps per mm and divide it by (your average/100).
- Repeat until you are happy.

Step 2: Calibrate your filament diameter, do this every print!

Using a caliper, measure your filament diameter at several locations. Average out the measurements, at least 3, and enter that into your slicer under filament diameter.

Step 3: calibrate your z height and first layer. Do this whenever something changes in your printer.

- Print a single layer (of say a 20*20mm cube) with your first layer at 100% height and width.
- Using a caliper measure the print in several places (at least 8) and adjust your bed or gcode z offset.
- Repeat until you are happy.

Step 4: calibrate your extrusion multiplier!

Do this every print, to be a little lazy every roll, to be really lazy every brand and material, if your slacker only once :(

- Print out a cube
 - (20mmx20mm would be nice)
 - in vase mode (single outline, no infill, no top or bottom layers)
 - set your extrusion multiplier to 1
 - set your extrusion width to equal the width of your nozzle.
- Measure the walls with a caliper in several places (at least 8) and get an average.
- Change your extrusion multiplier: new multiplier = old multiplier x (extrusion width / average measurement)
- Repeat until you are happy.

Step 4.5: PID tune your bed and hotend.

Before and after you calibrate temperatures and any time you change a fan or move something or a season changes. PID tune your hotend and bed to keep your temperature fluctuations to a minimum.

- M303 E(Extruder 0 for hotend 1 for bed) C(# of cycles 3-8) S(Desired Temperature)
- M303 E0 C5 S180 = PLA Hotend tune for 5 cycles

Step 5: calibrate your temperatures.

Do this for every different filament (color, brand, material, etc)

- Grab a temperature calibration tower off thingiverse: <https://www.thingiverse.com/thing:915435>
- Set the temperature range to the range listed on your filament or by the manufacturer.
- Print the calibration tower and choose the best temp.

Step 6: Calibrate your fan speed.

(I have a Delta with three really powerful layer fans. If they all three run at 100% the hotend loses temp)

- Print your calibration tower again but change the fan speed vs the extruder temp.
- Choose the best fan speed for the finish you desire.

Step 6.5: 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 temperature of 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.

I left a lot of details out and some things are vague. Let me know what should be added and I'll edit it. Also my formulas may be off.

SOC3D 2 years ago [/Print settings](#) [/How To Hub](#) [/Tutorial](#) [Reply](#)



Hi,

I have a Velleman Vertex K8400 printer with a 0.35 mm standard nozzle. Printing with 1.75 mm PLA without a heated bed.

Step 1: Estep/mm was calibrated from 200 to 140

Step 2: Measured the filament between 1.65 mm and 1.75 mm - On average about 1.73 mm

Step 3: Don't understand it. Am I to print only the first layer of a Cube, so rectangle of 20x20mm? what am I supposed to measure, and adjusting the bed - are you referring to the bed height? How should one use the measurements to adjust the bed or code z offset?

Step 4: Measured 9 times between 0.45 mm and 0.59 mm . avg. 0.50 mm = New multiplier 0.7

Step 4.5: Didn't understand this either

Step 5: Not yet tested

Step 6: Not yet tested

On step 4: The cube that I printed is under extruded at 1.0 extrusion multiplier... Step 4 recommend to use 0.7 - is this correct?

Christian_Simon 2 years ago [Reply](#)

Step 3 you should print a single layer of anything at 100% first layer height and 100% first layer width.

You then measure the height of the print in several places in order to find out if your z height is set correctly.

If your layer height is .2mm your part should measure .2 mm.

On Step 4 what was your line width set to? With a .35mm nozzle you should set your extrusion multiplier to 1.0 your line width to .35mm your variance should not be more than 100 micron really it should be within 20-40 micron.

If your extrusion is looking weird or under during this you may just have bad filament? If it's PLA try a new roll or try heating/vacuuming the moisture out of the filament.

Can you attach a picture of the cube you printed and tell me a little more about the filament, brand - how old is it - how is it stored.

SOC3D 2 years ago [Reply](#)

Hi,

Follow up question to Step 3: So if I measure 0.3 mm height when the layer height should be 0.2mm, does this indicate that my nozzle is 0.1 mm to high from the bed?

Step 4: (first picture, and print to the left on the second picture)

See pictures attached from the first 20mmx20mm cube test that had an average wall thickness of 0.5 mm at 1.0 extrusion multiplier. (first picture)

Step 4 - take 2: (second picture, and middle print)

Based on the first test I reduced the extrusion multiplier to 0.84 and did a new print. I printed this only 10 mm tall to see the difference between the two. Printed wall size was then measured to 0.40-0.43 mm and average of 0.42 mm

Step 4 - take 3: (second picture, print to the right)

Based on the second test I decided to use the new Beta version of Cura (2.1) rather than the Cura (15.04.5). I set the layer height to 0.06 mm, but kept the extrusion multiplier at 0.84 Printed wall size was then measured to 0.35-0.42 or an average of 0.38 mm (so getting fairly close now to the 0.35 mm target)

Regarding the question on the filament. I bought the Velleman own PLA 1KG 1.75 mm filament. It was opened 2 weeks ago. Since then it has been stored on the printer in room temperature of about 23 degree C. Do not see any info on production date.



Christian_Simon 2 years ago [Reply](#)

There is something wrong with your -- Filament or Hotend Temperature or Extruder -- the prints in those pictures are not due to calibration or tuning of extrusion.

Can you try printing a temperature tower off thingiverse? You should use the temperature range listed on your filament or if there is no range, 5 degree celsius steps from 180 degrees to 220 degrees.

You should also run PID tuning (M303) on your hotend to ensure you are getting a consistent temperature.

SOC3D 2 years ago [Reply](#)

Hi,

I will try to print the temperature tower tomorrow. I tried to print this today, and it turned out pretty well I think.



Christian_Simon 2 years ago [Reply](#)

Have you run the PID tuning on your hotend?

SOC3D 2 years ago [Reply](#)

thanks for the post!

3dprintingguy 2 years ago [Reply](#)

I'm similarly confused about step 3. My first layer averaged 0.33mm when I had a 0.2mm layer height. So does this indicate my nozzle is too close or too far from the bed? Do I set my z height to -0.13mm?

MitcheG1 about a year ago [Reply](#)

That indicates the nozzle is too far away. A negative number should move the nozzle closer to the bed. Should.

SOC3D about a year ago [Reply](#)

Not the answer you're looking for? Ask a new question to 10 [Print settings](#) experts active on Talk.

[Ask a question](#)

KEEP ON GOING, WE'VE GOT PLENTY TO TALK ABOUT

What's with all the Blue??

Just a quickie - but who decided to move from the neutral shades of Green and Red buttons to all buttons, all notifications and all links Purple-Blue??

CHUNKYSTEVE0 2 COMMENTS

fulfilled by 3d hubs program - less orders - unfair competition?

Hello All! Now 3D-hubs has introduced " FULFILLED BY 3D HUBS" - program I receive significant less orders. Not because there is a lot of competition, but because 50km away there is a company who has the funds to get this title.

MMARK 13 COMMENTS

Did you know there are 3D printing services near Saint Petersburg?

[Explore 3D Printers in Saint Petersburg](#)

Suspended hub by Offline printers.

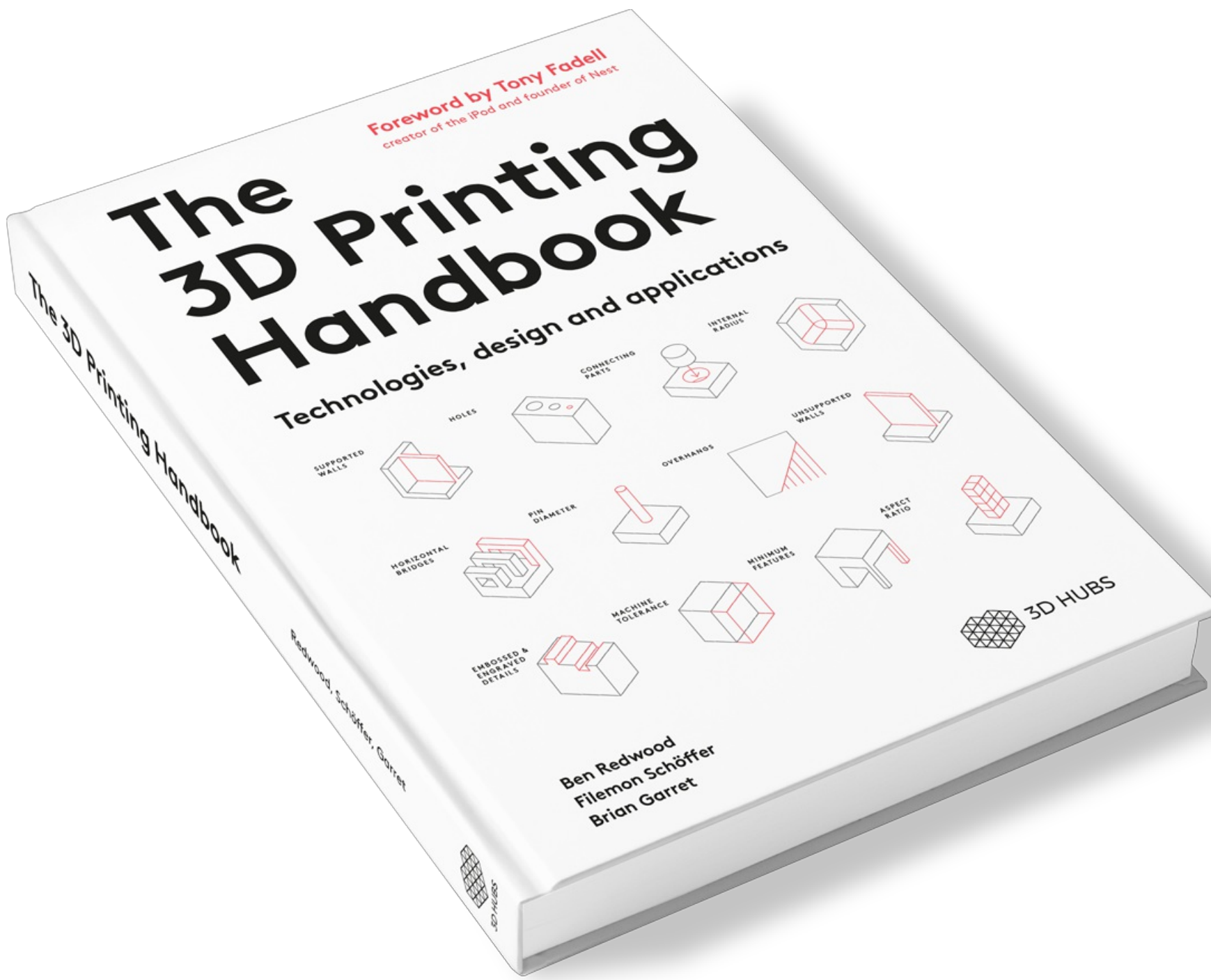
Hi, My hub just got suspended, cause my printers where offline for a while. Looks like your ranking is getting lower if you're not active. Its a pity to see this happen without any notice. After being on this platform since 2013 i feel deserted.

LEFT 4 COMMENTS

The 3D Printing Handbook is available now

If you're a professional looking to master the key aspects of 3D printing, this book is for you.

See what's inside



Company

- About 3D Hubs
- Work at 3D Hubs
- Team
- Press
- Terms / Privacy

Materials

- Prototyping plastics (FDM)
- High detail resin (SLA)
- SLS Nylon (SLS)
- Polyjet (Jetting)
- Industrial metals (DMLS)

Resources

- How 3D Hubs works
- How 3D printing works
- Knowledge base
- 3D Printer Guide
- Talk
- List your service

Services

- CNC
- Rapid prototyping
- Additive manufacturing
- 3D printing for education
- Managed services
- Bulk 3D Printing

Top Cities

- New York, NY
- London, UK
- Los Angeles, CA
- Chicago, IL
- Houston, TX

Contact

- Customer Support / FAQ
- Partnerships
- Press enquiries
- General Contact
- Submit ideas

in t f @

@2018 3D Hubs

We're Hiring!