Fix First Layer Problems

To print the first layer successfully:

- I strongly advise to read the document "**How to level your printer**"! It is looong, but messing further around with your printer takes longer, promised.

Requirements

---

1. If your printer has v-wheels make very sure ALL your v-wheels are set correctly, AND your **bed** / **extruder** / **gantry** also have **no play** for other reasons.

2. You washed your build-plate with **soap** and **microfiber cloth**, and never touched it's surface a bit afterwards.

3. Calibrate your Rotation Distance! It is a must-do.
Here I have developed a nice little easy tool for reliable measurement:
https://www.printables.com/model/639746

Test Files

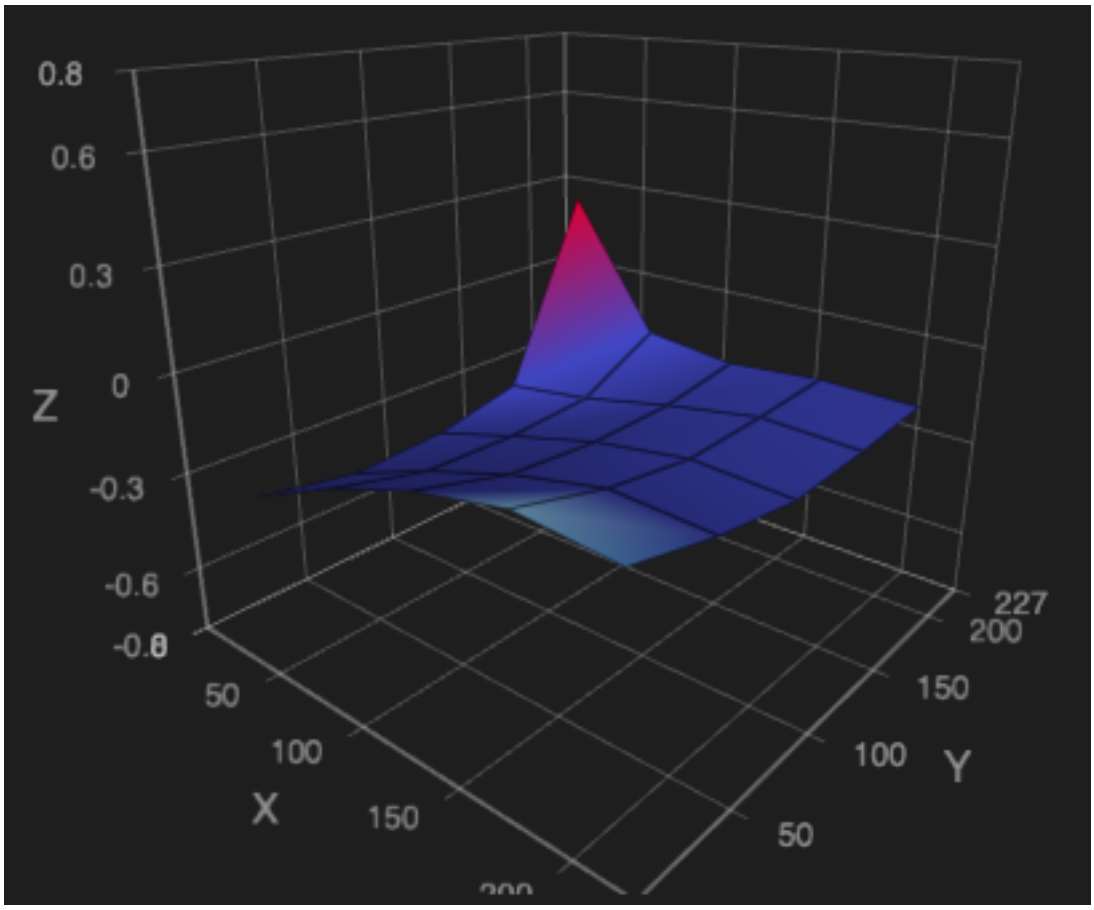
- In the download of the Printer Addition are **STL files** for your slicer included, to print a FULL first layer. It's the only method to show the whole picture, and to live-adjust the z-level.

I **strongly recommend PLA** filament for testing, as it is the easiest and most forgiving material, and allows more speed, so your testing goes faster.
Test with 100 mm/s speed.
If everything is perfectly set up, it is possible to print with PLA the full first layer test files with 200 mm/s or more.

Mesh

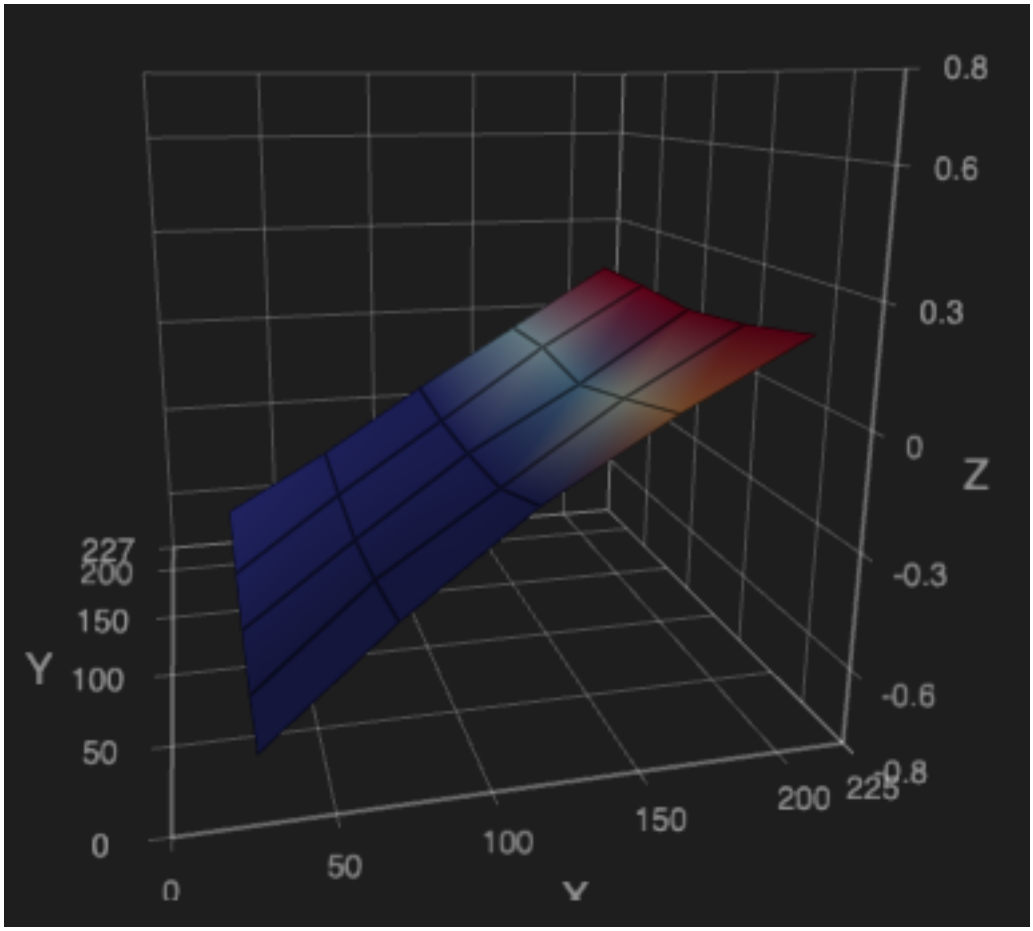
- Check if your mesh **looks normal**.

- This is **not normal** - it has an unusual spike and the mesh needs to be re-created:

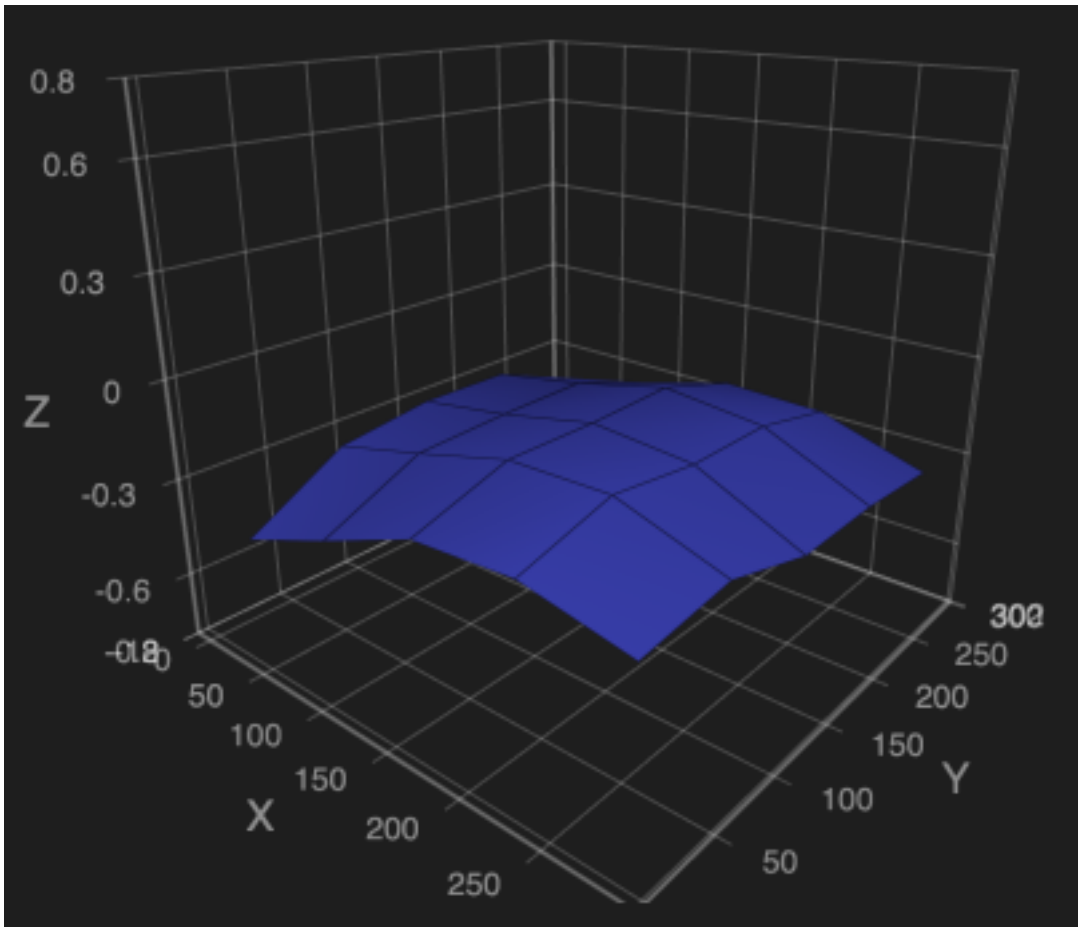


If it's not a probing-error, then check if there's **dirt** under the build plate.

- This is also not normal - the bed is not leveled against the gantry:



- This is not super-great, but more than enough to print successfully:



• While you print the test file, **live-adjust the z-level** in Mainsail / fluidd, so that it produces a good surface.

When you are happy with the new z-level, click the little blue "SAVE". If you use SAVE_CONFIG instead the new z-level will not be saved!

- **Z too high**: Non-sticking / holes / wavy hole pattern / sticking, but flaking lines.
- **Z too deep**: Crumbly, dry looking surface.

If you can't reach a 100% even result: It is then **better** to have the **z-level a bit too deep** at some areas of the bed. That will not affect the end result of a normal print, but everything will stick.

Z-Level changed then again?
If after some time your z-level doesn't seem to fit any longer, try re-creating the mesh first and see if it cures.
To do so, click the Button "**MESH CREATE FOR TEMPERATURE**".
Don't forget to click the "**SAVE CONFIG**" button in Mainsail, after you've re-created the desired mesh(es).

Still doesn't work?

- Make sure **Gantry Calibration** (Z-Tilt) is not turned off in the KPA.
  When you start a print, and you read "OFF: Gantry Calibration", turn it back on.

- If you often get "**Probe samples exceed tolerance. Retrying...**", then one possible cause is a print head that cannot move freely enough in the Z direction. Check that the **filament is not under tension** and can slide easily into the extruder.
  A source of error is a **cable holder** that exerts a pulling force on the extruder (e.g. a rubber band).
  Another is a **filament sensor** which may cause too much drag.

- Go **slower on the first layer**. Not every filament wants to print fast. Try 40 mm/s, to check if that makes a difference.

- **Raise bed temperature**. Try 20°C over the vendors recommendation.

- **Raise filament temperature** by 10°C over the vendors max recommendation. Only for the first layer.

- You did calibrate **Rotation Distance**?

- You did **calibrate filament flow**? This setting is usually found in the filament settings of the slicer. OrcaSlicer has a build-in test for this.

- Make sure **no other additional G-Code**, or Klipper-commands are in the slicer's start-G-code. Set it up according to the Slicer Setup Guides, contained in the download of the KPA.

- I'm currently experimenting with ridiculous high probe point counts. I used 32x32 probe points, and set **bicubic tension to 0**. That excludes interpolation errors. I printed it with my worst printing filament, and my result was very very good, a near perfect first layer, with only a few tiny issues at the front border (which heats not well).
  Try this in Multiple Mesh mode, as it takes obviously some time to create such a mesh, and the KPA will automatically re-use it this way.

Support

If my work supports you, please support me too:
- PayPal: https://www.paypal.com/paypalme/Morgennebel/

When you buy something, please use my Referral-Link, at no additional cost:
- AliExpress: https://s.click.aliexpress.com/e/_De0Vu1h
- Amazon: https://amzn.to/3L35OX2

Thank you!
Christian 🙂