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.
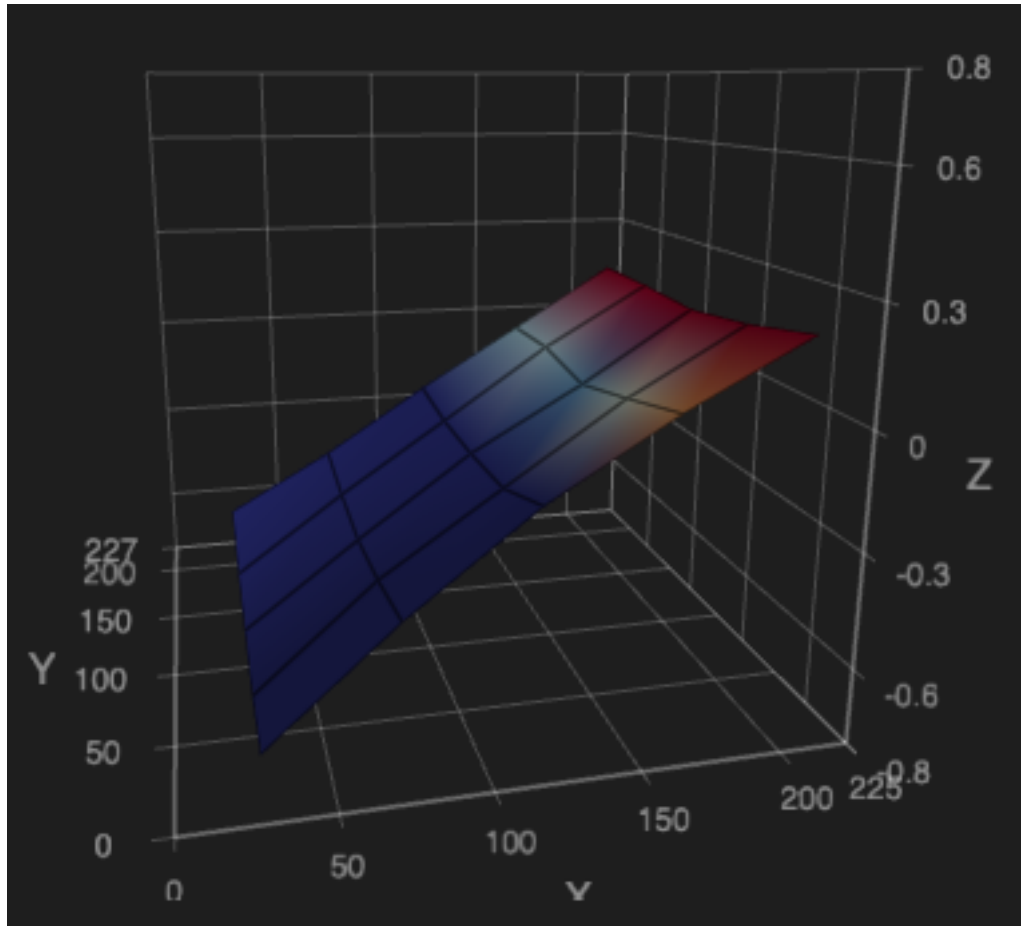
Mesh

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

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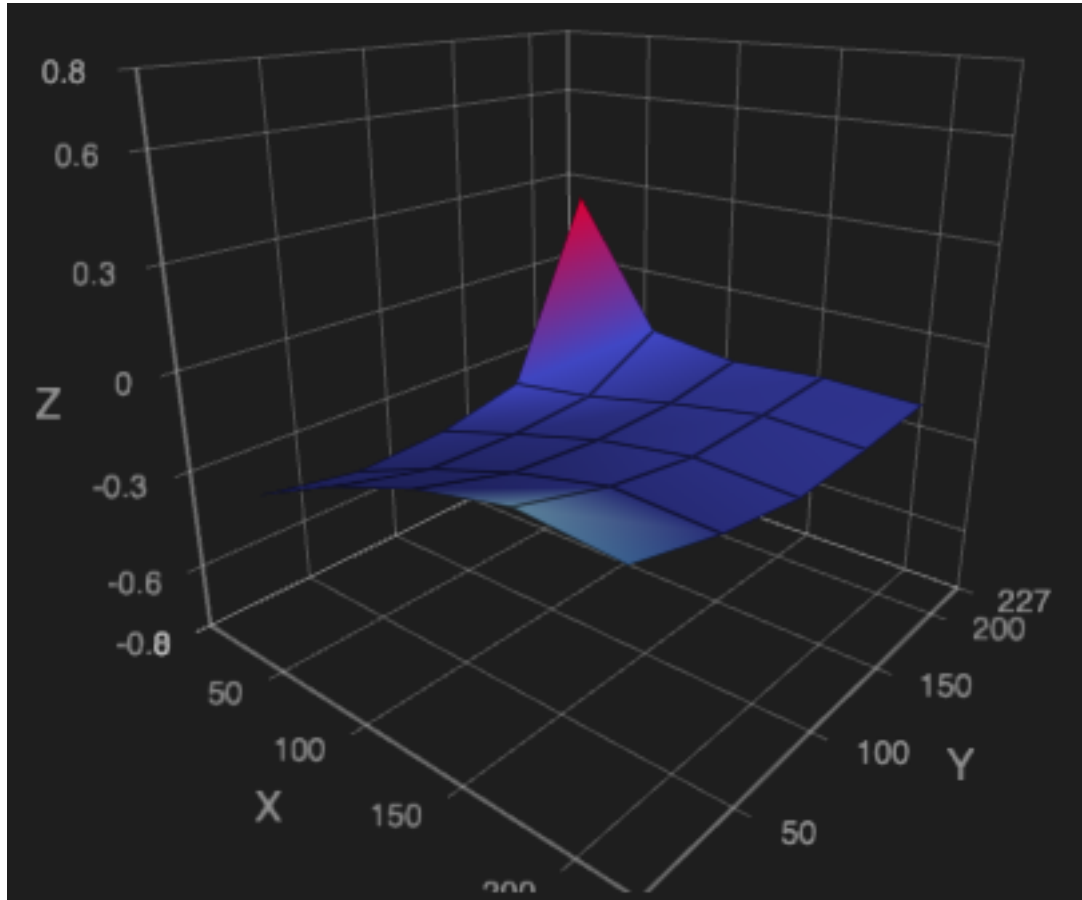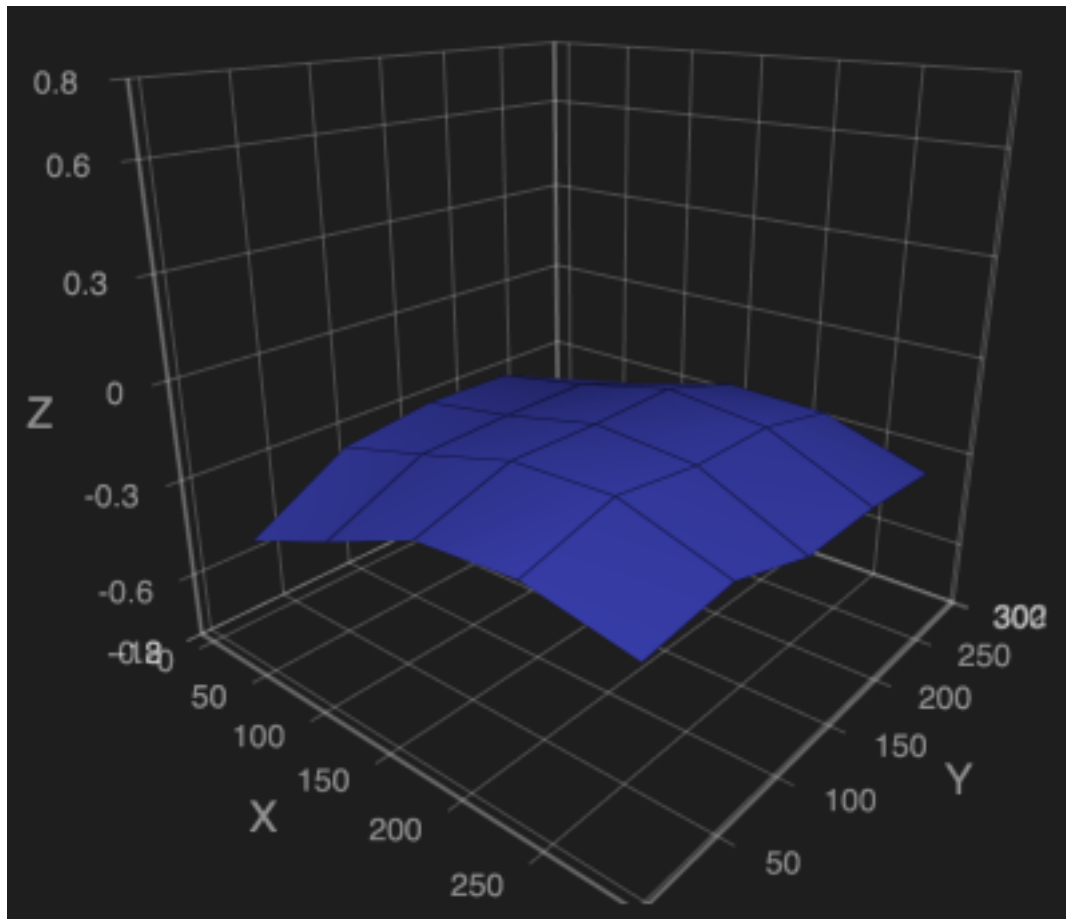

If it's not a probing-error, then check if there's **dirt** under the build plate.

• 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).

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 🙂