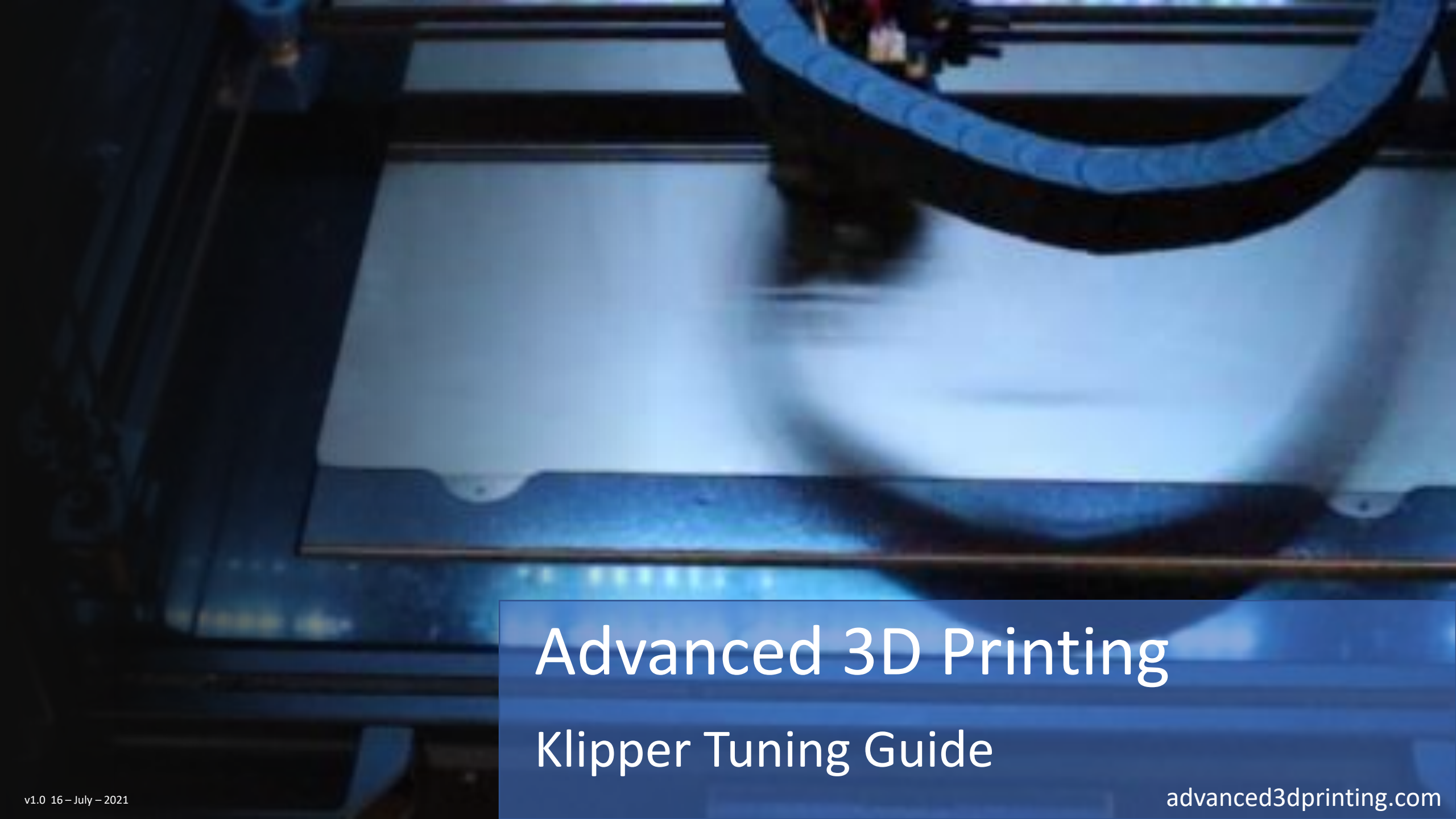


Advanced 3D Printing

Klipper Tuning Guide

v1.0 16 – July – 2021

advanced3dprinting.com

Tuning process overview

Introduction.
This guide covers additional steps for the tuning of your Troodon 300 or 400 printer upgraded with Avance3d 3D Printing Klipper Kit (please refer to the *Klipper Kit Installation Guide* for details).

The guide covers:
1. Z Offset calibration.
2. Pressure Advance calibration.
3. Input Shaper auto-calibration.

What you need.
The main prerequisite to this guide is the successful installation of Klipper as described in the *Klipper Kit Installation Guide*. Please ensure that the printer is operational and responds to commands from the Klipper web interface (Mainsail) and (optionally) Panel Due. You should be able to do homing, perform gantry levelling, heat up the hotend and heatbed, etc. You also need be able to load filament (for Pressure Advance calibration) and ensure that BL Touch is working.

It is also useful to set up a slicer profile at this point. While it is not a requirement for calibration and tuning described in this guide, Pressure Advance value that will be obtained in Step 2 is filament-specific. As a best practice, you will be entering this value into your slicer profile.

Please ensure that you have installed Accelerometer Harness and you have access to the accelerometer (e.g. it is left inside the printer enclosure - you many need to open the enclosure and route the accelerometer wires out if this is the case).

We also recommend to back up your printer.cfg file, please refer to Step 1.a for details.

Do not attempt any test prints before successfully performing Step 1 (Z Offset Calibration). It may damage the bed surface or even the hotend.

Install Klipper Kit

Step 1.
Calibrate Z Offset

a. Check printer.cfg file
b. Calibrate Z Offset (paper test).
c. Perform test print and adjust Z Offset.

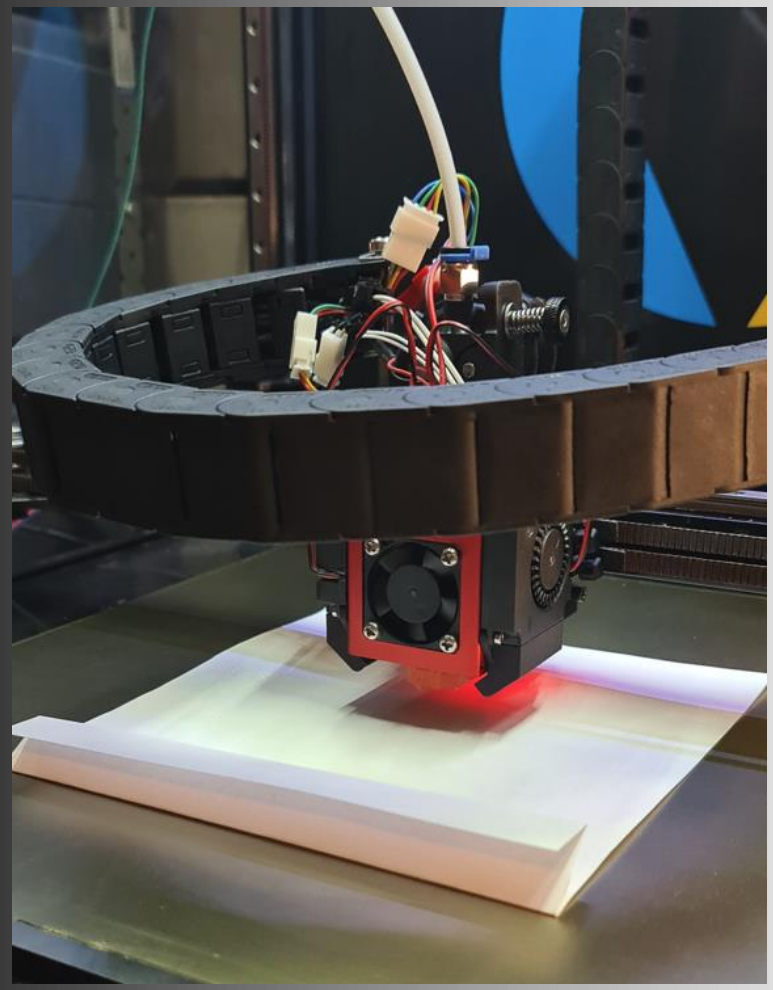
Step 2.
Calibrate Pressure Advance

a. Generate calibration G-code.
b. Print calibration pattern.
c. Interpret results.
d. Update slicer profile.

Step 3.
Calibrate Input Shaper

a. Connect and mount accelerometer.
b. Perform input shaper calibration.

Enjoy advanced 3d printing!



Step 1.

Calibrate Z Offset

1.a: Check printer.cfg

a. Check printer.cfg file

Use Klipper web interface (Mainsail) to access the “Settings tab”. Click on “Machine Settings” and locate the **printer.cfg** file in the “Config Files” section (Diagram 1.1).

Click on **printer.cfg** and scroll down to the end of the file until you see the “`#*# <---------- SAVE_CONFIG ---------->`” section. Please note that the content and structure of the file may vary depending on the version of the Klipper image you use, however this section will always be at the end of the file.

Locate the `#*# [bltouch]` entry in the SAVE_CONFIG section and note the `z_offset` value – this is the current Z Offset that will be adjusted to match your printer (Diagram 1.2).

In some cases, `z_offset` value in the **printer.cfg** file may be set to a negative value, which will cause an error in Klipper. In that case, please update the value manually to 0.5mm and perform “Save and Restart” as described in Step 1.c.5.

We suggest to create a back up of the prnter.cfg file at this stage. The easiest way do to that is to click on any of the values in the file and then Select All (Ctrl+A on Windows) – please check that you have the file content and not the line numbers selected (Diagram 1.3). Press Copy (Ctrl+C on Windows) , open the text editor of your choice (e.g. Notepad) and save the file to your computer. You will then be able to restore the file using the same technique if something goes wrong.

You can (and probably should) make periodic full backups of your Klipper configuration (all files stored in ‘/klipper_config/ directory on your Raspberry PI) and/or the whole Raspberry PI SD Card image. Please consult Raspberry PI documentation.

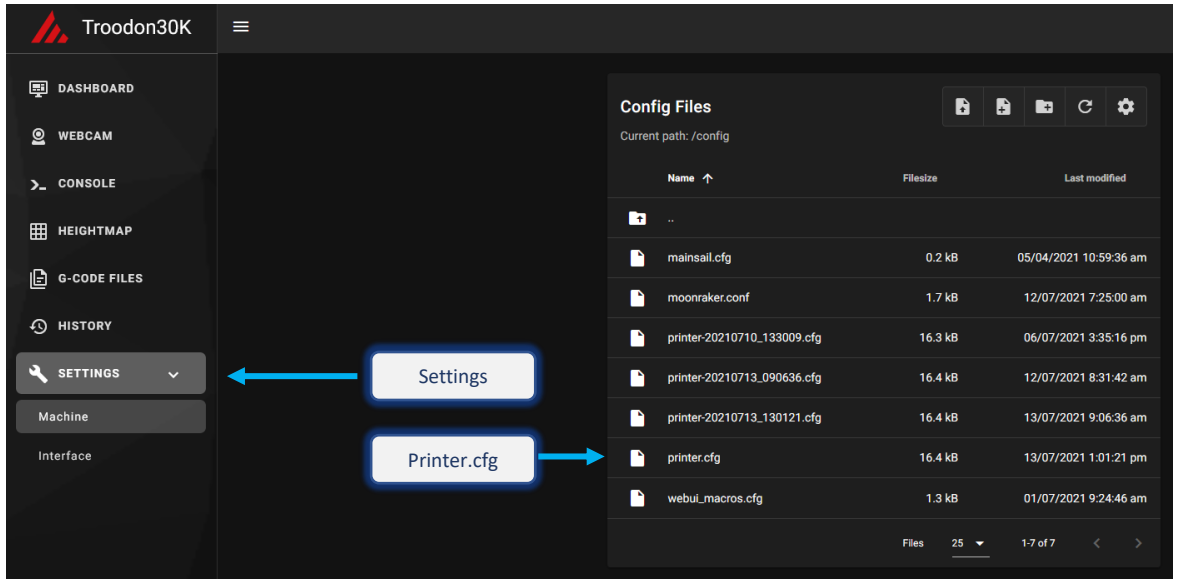


Diagram 1.1

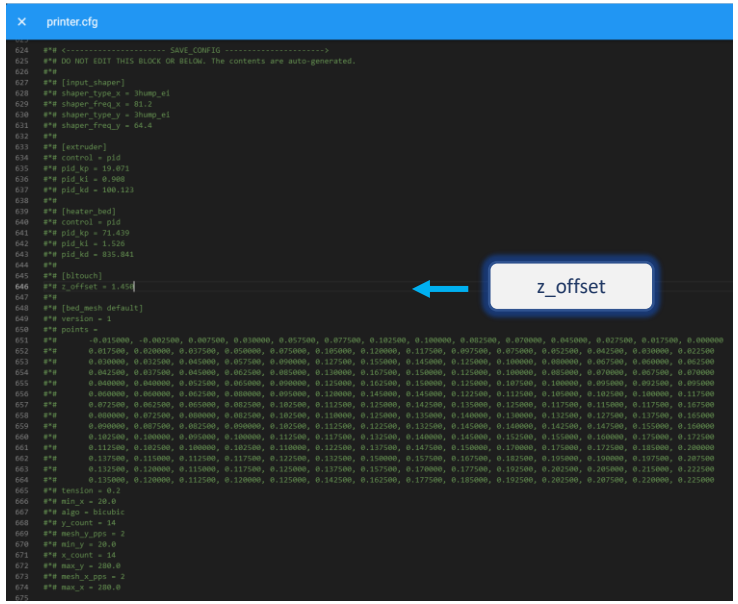


Diagram 1.2

Diagram 1.3

1.b: Calibrate Z Offset

b. Calibrate Z Offset (paper test)

Turn on the printer, home all axis and perform Quad Gantry Level aka QGL (Diagram 1.1). Check that Current Offset value in "Z Baby Stepping" panel is 0.

Please note that Z Offest calibration is performed at room temperature, with heatbed and hotend heaters off (set to 0). Ensure that there is no filament on the nozzle before calibrating Z Offset. You may want to heat the hotend, clear the nozzle and/or unload the filament and let it cool down before proceeding.

Place regular piece of copy machine paper between the printer's bed and nozzle (Diagram 1.2). Note that there should be a gap of 5-6 millimeters between the nozzle and the paper at this stage.

Go to the "Console" panel, type **PROBE_CALIBRATE** command and press "Send" button (Diagram 1.3). Responses like "Starting manual Z probe…" identify that you are in manual (Z Offset) probe calibration mode now.

Now start lowering the nozzle, until it is just touching the paper. To do that you will typing a series of TESTZ Z=[nnn] commands into the Console (pressing the "Send" button after each one), replacing [nnn] with a positive value in mm when you need to raise the nozzle or negative value in mm when you need to lower the nozzle.

You can use Copy & Past to avoid manual typing or press the Up Arrow key on your keyboard while in the Console to recall previously entered command.

Useful examples of TESTZ command are:

TESTZ Z=-1	- lower the nozzle by 1 mm
TESTZ Z=-0.1	- lower the nozzle by 0.1 mm
TESTZ Z=-0.01	- lower the nozzle by 0.01 mm
TESTZ Z=1	- raise the nozzle by 1 mm
TESTZ Z=0.1	- raise the nozzle by 0.1 mm
TESTZ Z=0.01	- raise the nozzle by 0.01 mm
TESTZ Z=-	- lower the nozzle by half of the previous position.
TESTZ Z=+	- raise the nozzle by half of the previous position.
TESTZ Z=-	- lower the nozzle by half of the previous position.
TESTZ Z=+	- raise the nozzle by half of the previous position.
TESTZ Z=--	- return the nozzle to the previous (lower) position.
TESTZ Z=++	- return the nozzle to the previous (lower) position.

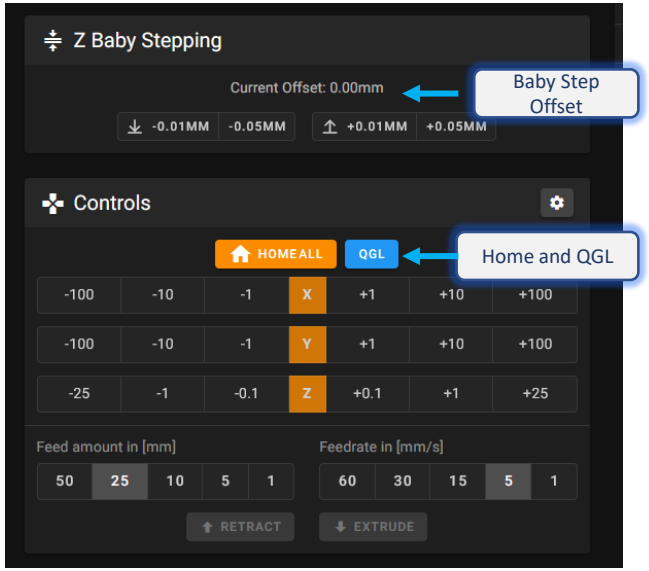


Diagram 1.1

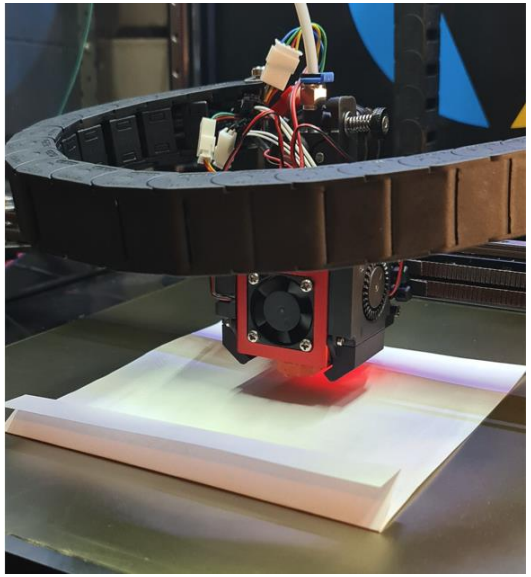


Diagram 1.2

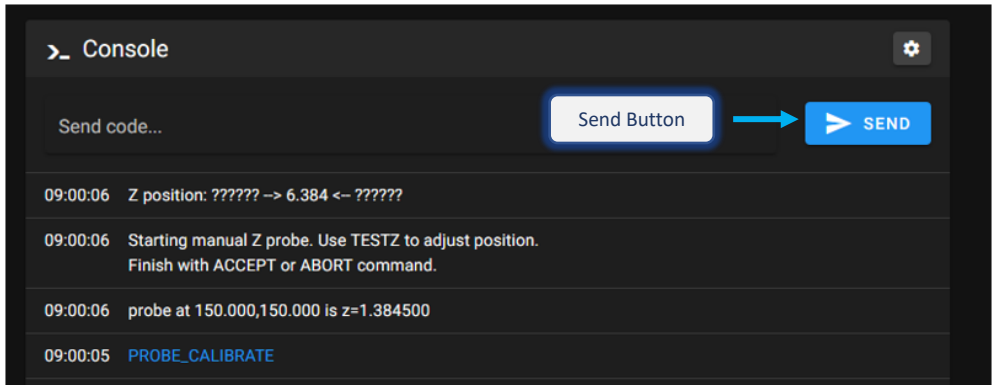


Diagram 1.3

1.b – 1c: Calibrate Z Offset and perform test print

Using sTESTZ commands in logical sequence, adjust the height of the nozzle by moving it up and down until it barely touches the paper. Push the paper back and forth under the nozzle – you must feel just a small amount of friction. Use TESTZ Z=- and TESTZ Z=+ commands once the nozzle starts touching the paper to get the right amount of friction.

Please be careful when lowering the nozzle – if you misjudge the distance or accidentally type 10 instead of 1, the nozzle will crash into the heatbed. It is always safer to work with smaller moves.

Once you are satisfied with the distance between the nozzle and the heatbed (the paper) type **ACCEPT** in the Console and press "Send" (Diagram 1.4).

You will be prompted to use the **SAVE_CONFIG** command to save new Z Offset in the **printer.cfg** file (Diagram 1.5). You can use the Console to enter the **SAVE_CONFIG** command or press the "Save Config" button that will appear in the top right corner of the Mainsail window (Diagram 1.5) . After doing that, you can check the **printer.cfg** file to ensure that new Z Offset value is saved (Ref. Step 1a).

c. Perform test print and adjust Z Offset

Z Offset that has been just calibrated may need further adjustments – to compensate for thermal expansion when the nozzle and heatbed are heated - and "a small amount of friction" is subjective anyway. To do that:

1. Perform test print of an object with reasonably large flat 1st layer (a box or a cube).
2. While the 1st layer being printed, use "Z Baby Stepping" panel buttons to adjust the distance between the nozzle and the heatbed until you are satisfied with the 1st layer quality (Diagram 1.6).
3. Note the baby step value ("Current Offset" in the "Z Baby Stepping" panel) that gives you the best result.
4. Open your **printer.cfg** file and navigate to the the `#*# [bltouch]` entry in the "SAVE_CONFIG" section as described in Step 1a.
5. Adjust the value of `z_offset` by the baby step value, then click "Save & Restart" button (Diagram 1.7)
6. Please note that the bay step value need to be **subtracted** from the current Z Offset value. Examples:
   - Current value is **1.458** and the nozzle needed to be lowered by further **0.15** mm (baby step is negative, e.g. **-0.15**). Adjusted value is **1.458 – (-0.15) = 1.608**
   - Current value is **1.458** and the nozzle needed to be raised by further **0.15** mm (baby step is positive, e.g. **-0.15**). Adjusted value is **1.458 – 0.15 = 1.308**
   - In other words, Z Offset needs to be **increased** if the nozzle needs to be **closer** to the heatbed and **decreased** of the nozzle needs to be **higher above** the bed.

Diagram 1.4

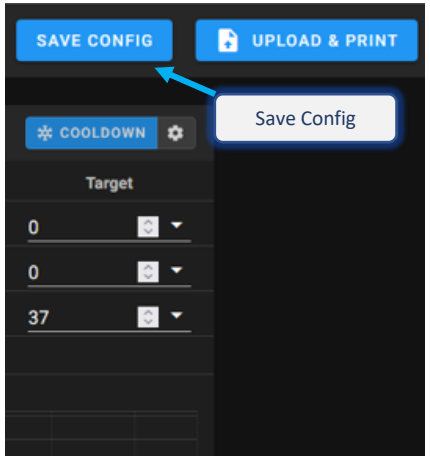


Diagram 1.5

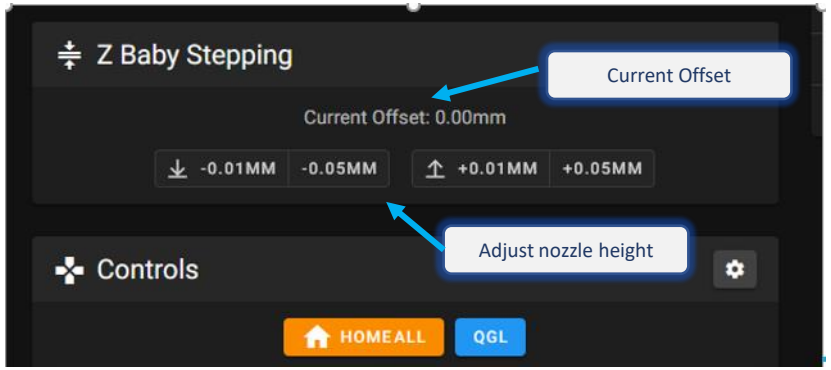


Diagram 1.6

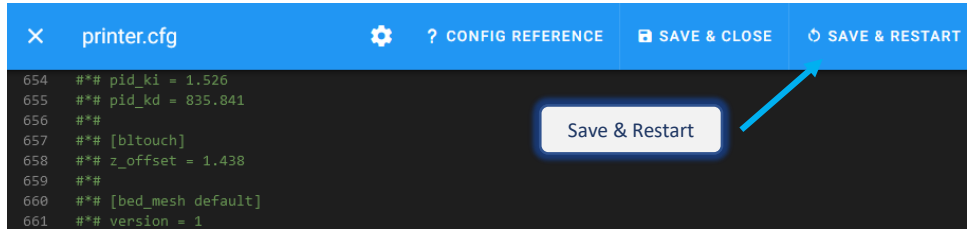
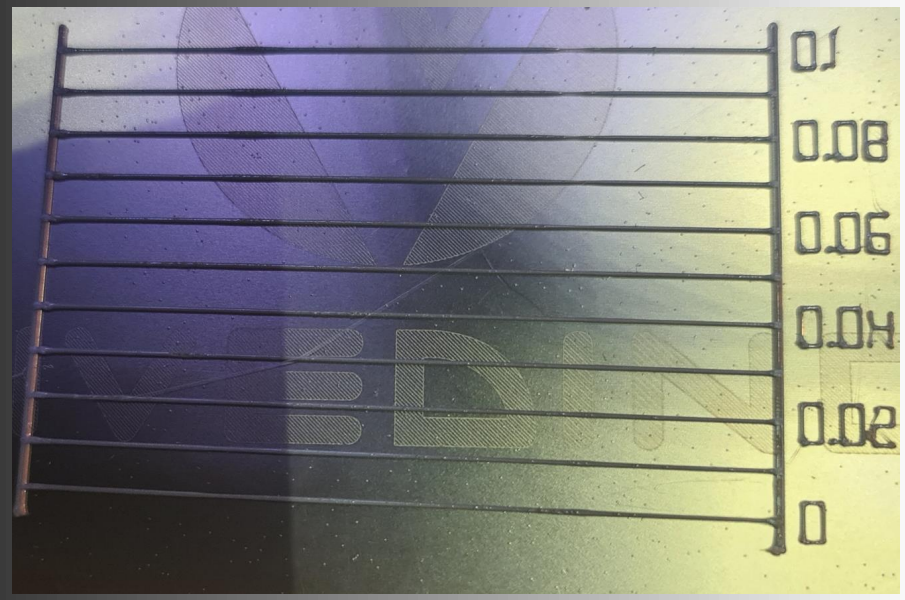


Diagram 1.7



Step 2.

Calibrate Pressure Advance

2.a: Generate calibration G-code

a. Generate calibration G-code

Pressure Advance calibration method discussed in this section is based on the use of online tool provided by Advanced 3D Printing. To access the tool please navigate to

https://advanced3dprinting.com/linear-advance-tool-klipper/

There is an online form, where some details about the printer and filament need to be entered (Diagram 2.1). Please check the instructions at the bottom of the page (Diagram 2.2). Key settings are:

Nozzle Temperature and **Bed Temperature**	Enter values that suit the filament used.
Retraction Distance	Typically 0.4-0.8mm for DDE set up and 3-4mm for Bowden set up.
Bed Size X and **Bed Size Y**	300 x 300 for Troodon 300 and 400 x 400 Troodon 400.
Retract Speed and **Unretract Speed**	Depend on retraction settings, 55 and 50 mm/s are good staring points.
Starting Value for K	Test will start with this value for Pressure Advance (K factor). 0 is a good staring point.
Ending Value for K	Test will end with this value for Pressure Advance (K factor). 0.1 for DDE set up and 1 for Bowden are good staring points.
Pressure Advance Stepping	Start wit 0.01 for DDE and 0.1 for Bowden.
Filename	Enter the desired name for the g-code file.

Press "Generate G-Code" button, then, once the box on the right is populated with G code, press the "Download as file" button and save the file to your computer (Diagram 2.3).

Some fields on the online form and web page refer to "Linear Advance" or "K Factor". In the context of this Step, it should be considered the same as "Pressure Advance".

Please note this calibration needs to be repeated for each new type of filament and/or nozzle size used.

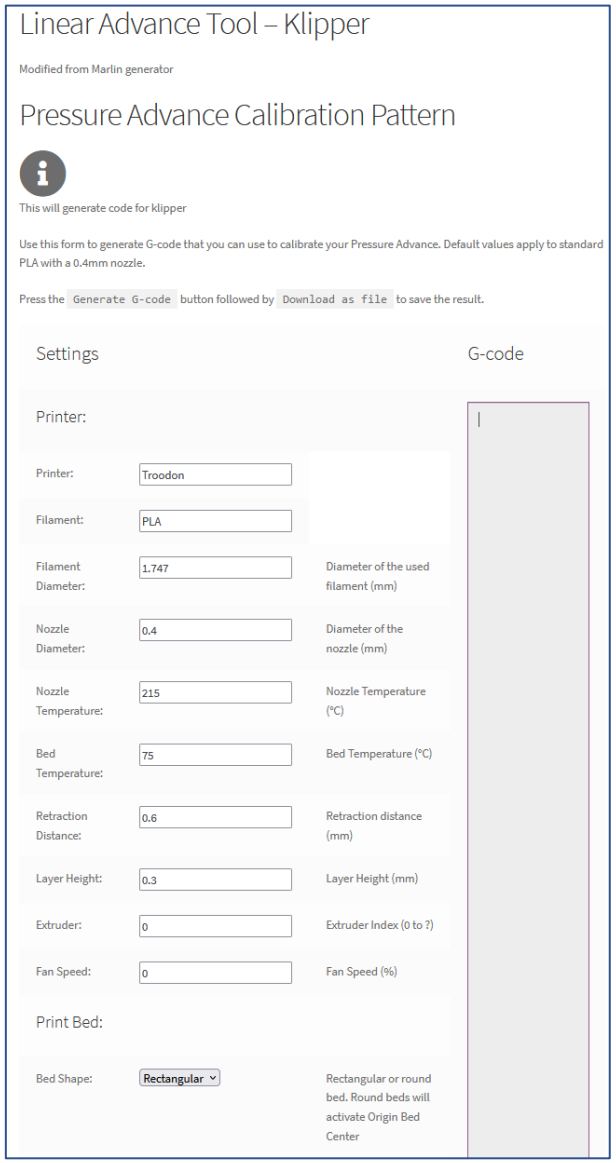


Diagram 2.1

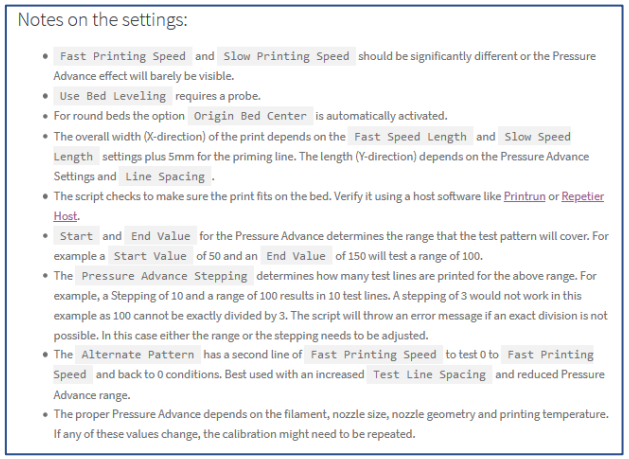


Diagram 2.2

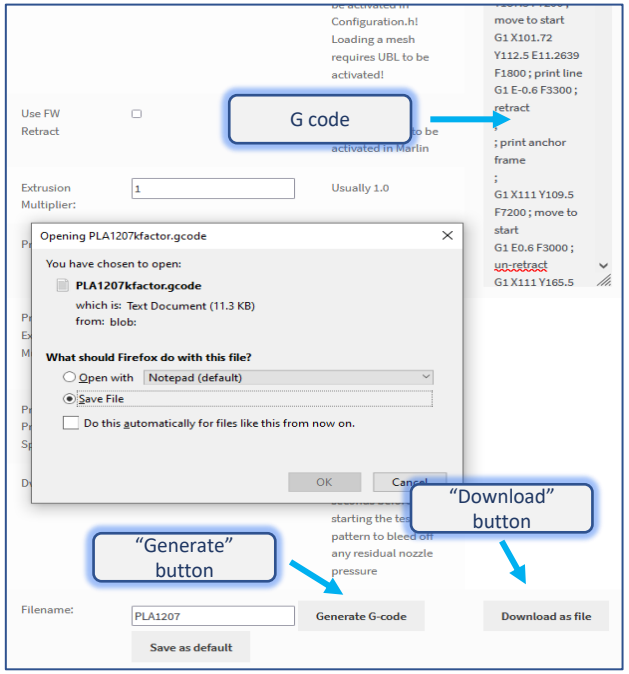


Diagram 2.3

2.b: Print calibration pattern

b. Print calibration pattern

Using Klipper web interface (Mainsail), upload and print the G code file that was generated in Step 2a. by using "UPLOAD & PRINT" Button (Diagram 2.4).

If the first layer (this is a single layer print) does not stick well to the bed you can go back to Step 2a and adjust bed temperature or Z offset (enter negative value, e.g. -0.5 to bring the nozzle closer to the bed by 0.5mm) in the online form. Alternatively, you can use Z baby Stepping controls in Mainsail (Diagram 2.5) to temporarily adjust the distance between the nozzle and the bed (bring the nozzle closer to the bed) while the calibration pattern is printing.

Temporarily changes to nozzle height described in this step will not affect Z Offset value stored in **printer.cfg** file (Ref. Step 1).

Once you have a print, similar to the one depicted in Diagram 2.6, you may proceed to the next step.

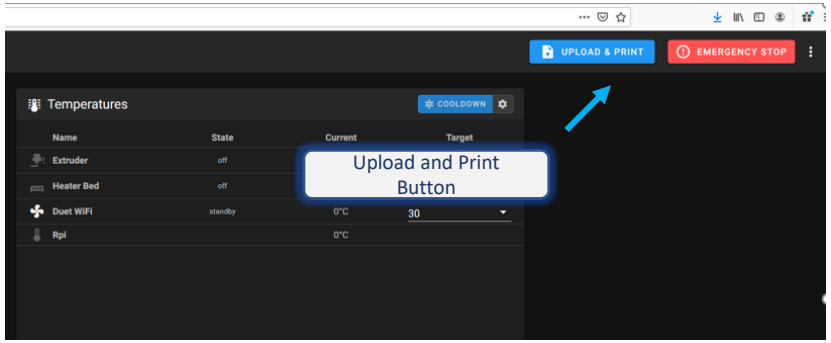


Diagram 2.4

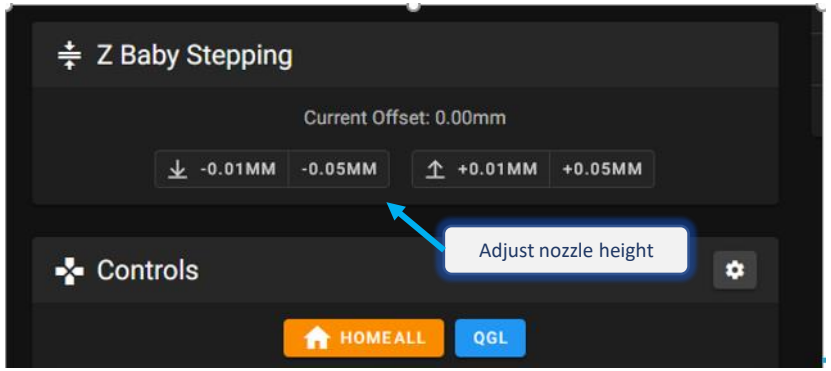


Diagram 2.5

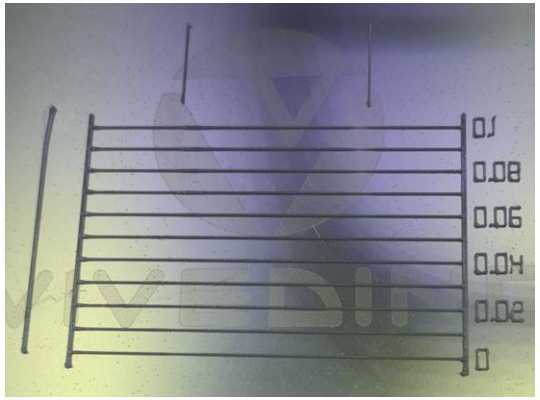


Diagram 2.6

2.c-2.d: Interpret results and update slicer profile

c. Interpret results

Remove the build plate from the heatbed and have a close look at the print (Diagram 2.7).

Pick up the line, which is has most consistency, especially when transitioning from slow to fast and fast to slow. The number on the right represents optimal Pressure Advance value (also called K factor) for the given filament.

In this case the proposed value is 0.03 (highlighted by the red frame). If the numbers on the right are not readable, the right Pressure Advance value can be calculated by counting the lines from the bottom/Starting K factor value and adding Pressure Advance Stepping value for each subsequent line.

d. Update slicer profiles

Klipper command for setting Pressure Advance is:

SET_PRESSURE_ADVANCE ADVANCE=[value]

In this case

SET_PRESSURE_ADVANCE ADVANCE=0.03

Pressure Advance value determined in previous steps is dependent on the filament used, nozzle diameter and to a lesser degree on filament temperature. It is therefore needs to be adjusted when you change the filament type (PLA, PETG, …) and sometimes brand. The most convenient way to achieve that is to place the above command into the filament settings section of your slicer profile. In SuperSlicer for example, the command needs to placed into the "Custom G-code" section of the "Filament Settings" tab (Diagram 2.8).

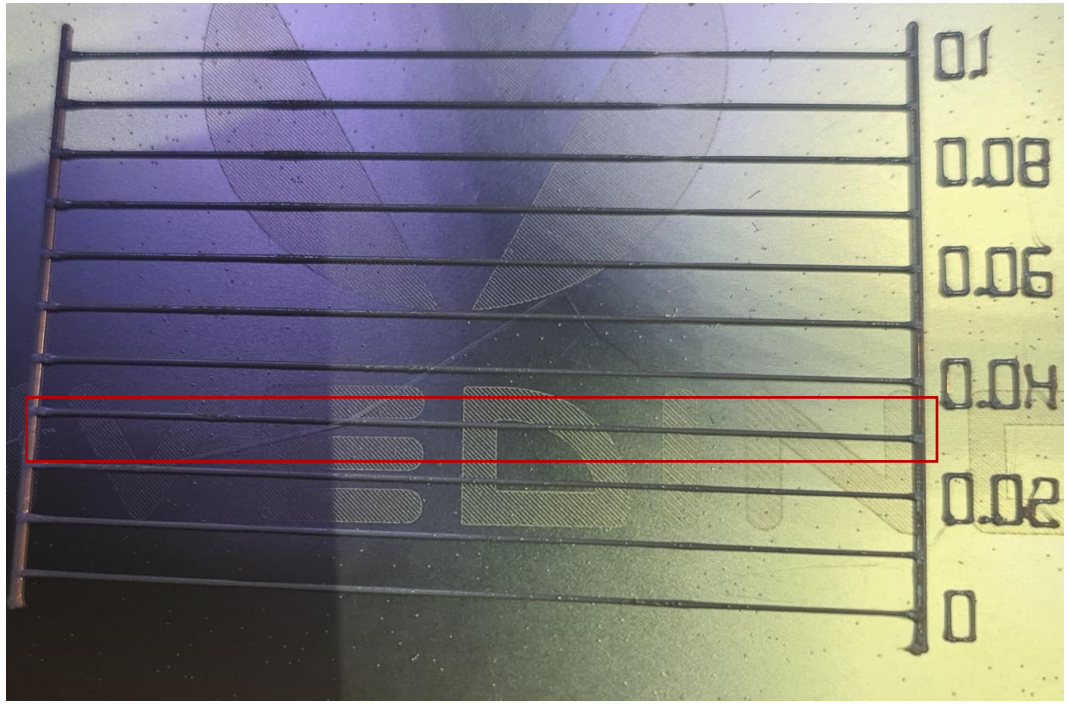


Diagram 2.7

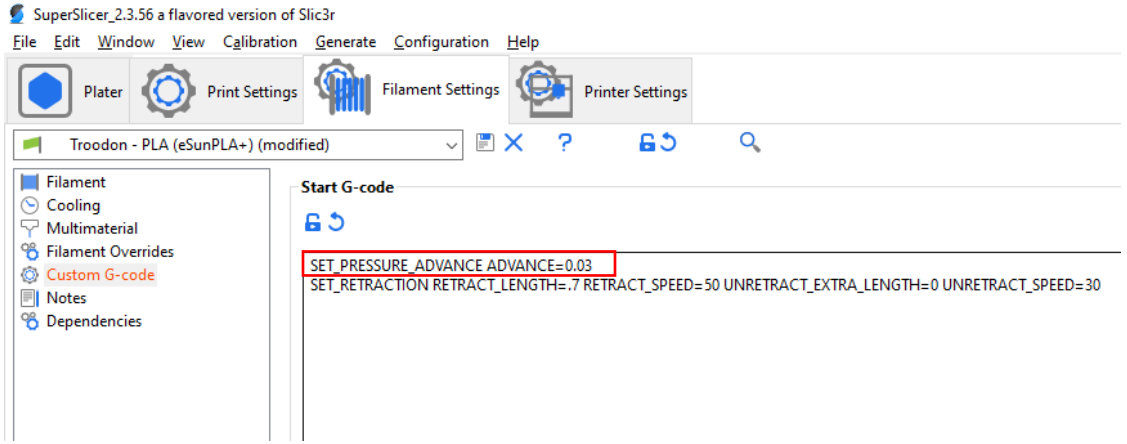
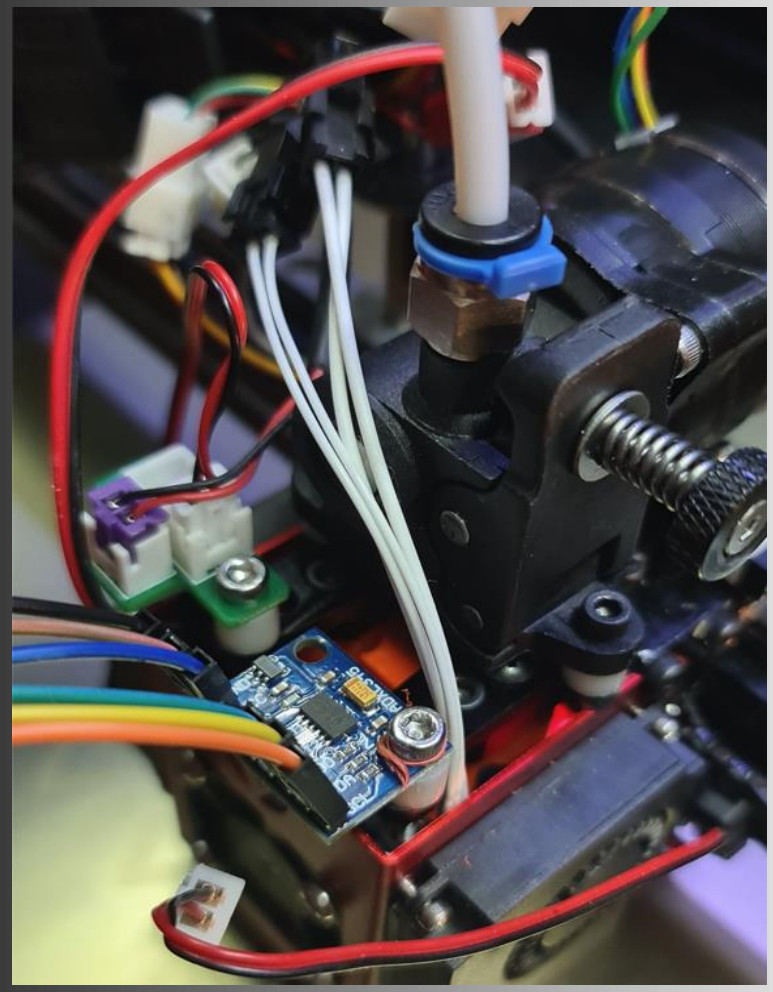


Diagram 2.8



Step 3.

Calibrate Input Shaper

3.a: Connect and mount accelerometer

a. Connect and mount accelerometer

Turn the printer off and check the Accelerometer Harness and its connection to the Raspberry PI and the accelerometer. Ensure that Accelerometer Harness are long enough to reach the printhead assembly, otherwise you may need to temporarily move Raspberry PI out of the enclosure.

Mount the accelerometer on the printhead, ensuring that the axes of the accelerometer align with the printer's axes. The easiest way to do that is to temporarily use the mount for the part cooling fan splitter (Diagram 3.1):

- Unscrew the splitter mount - you need to undo only one bolt, then you may just turn the splitter 180° to move it out of the way.
- Mount the accelerometer as shown, re-using the white plastic spacer – this is critical to prevent the short circuit between the accelerometer and the mounting plate.
- Check the accelerometer board is parallel to the printer's axes and tighten up the bolt just enough for the accelerometer to be secured.

An incorrect wiring or short circuit can permanently damage the accelerometer or Raspberry PI. Please double check the wiring is correct (Ref. *Klipper Kit Installation Guide*) and that the accelerometer does not touch the mounting plate or the bolt. If possible, use the nylon bolt to mount the accelerometer or put a short piece of the heat shrink tube between the bolt and the accelerometer board as shown. Mount the accelerometer only with the printer turned off.

To check that the accelerometer is operational, turn on the printer, go to the "Console" panel of Kilpper web interface (Mainsail), enter the ACCELEROMETER_QUERY command and press "Send" button. You should see the response from the accelerometer (Diagram 3.2).

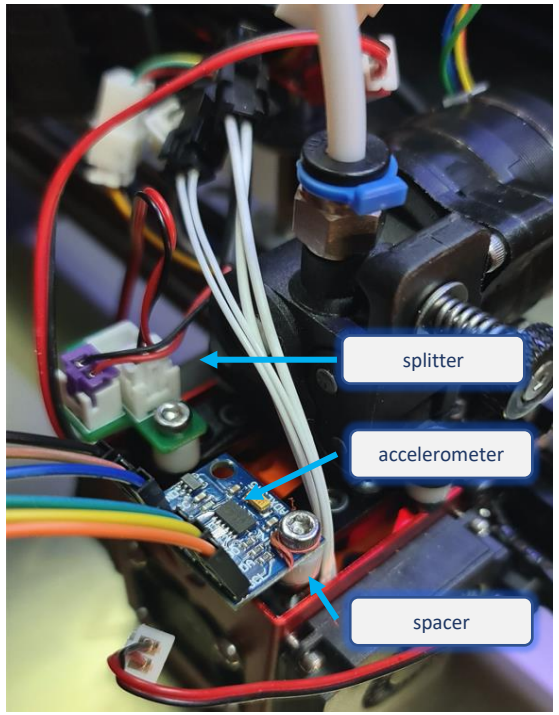


Diagram 3.1

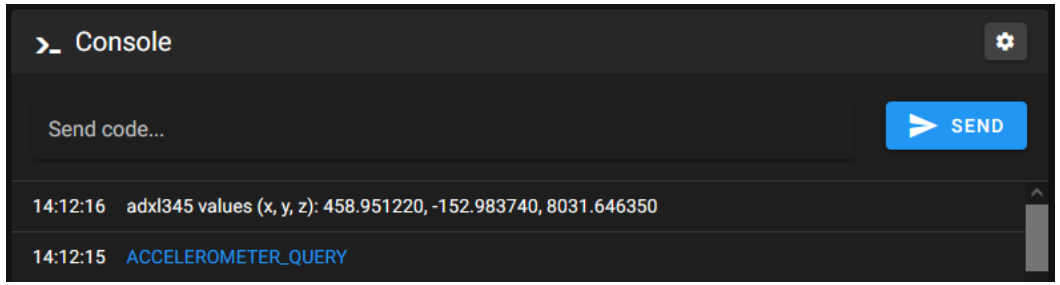


Diagram 3.2

3.b: Perform input shaper calibration

b. Perform input shaper calibration

In the “Console” panel enter the SHAPER_CALIBRATE command and press “Send” button (Diagram 3.1). The printer will vibrate while going through various stages of calibration. When the calibration process is finished, you will see suggested shaper and acceleration values as well as the prompt to use the SAVE_CONFIG command to save new input shaper parameters in **printer.cfg** file (Diagram 3.4).

You can use the Console to enter the **SAVE_CONFIG** command or press the “Save Config” button that will appear in the top right corner of the Mainsail window (Diagram 3.5) .

Input Shaper calibration method described in this guide does not cover acceleration tuning – a more advanced manual process.
Please refer to *https://www.klipper3d.org/Measuring_Resonances.html#measuring-resonances* for more details.

You should not perform Input Shaper calibration too frequently to avoid excessive wear of the printer components.

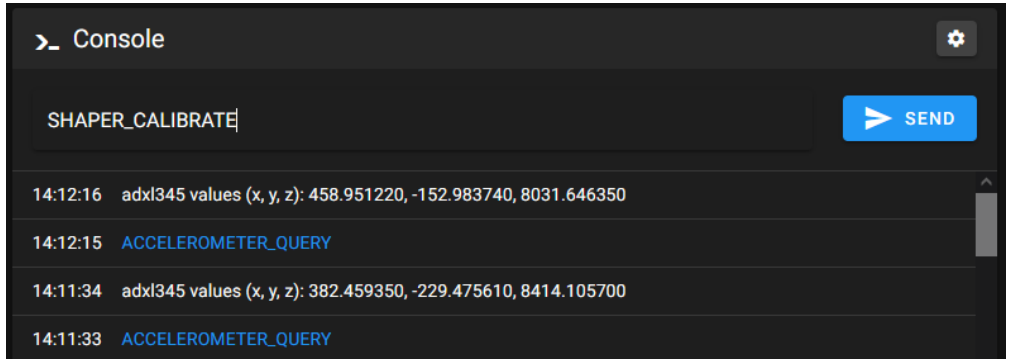


Diagram 3.3

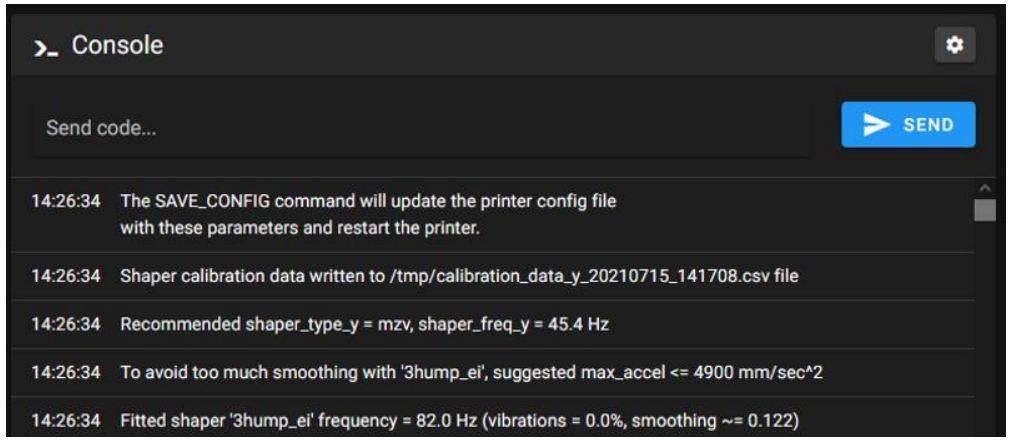


Diagram 3.4

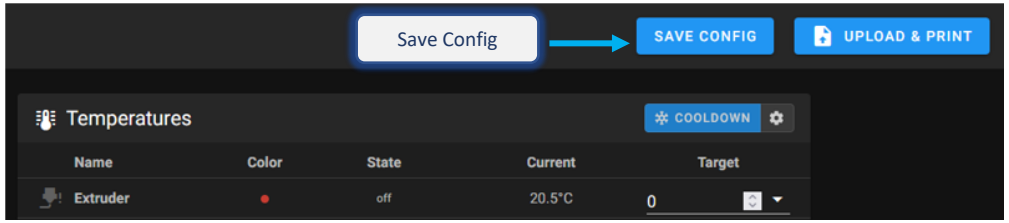


Diagram 3.5