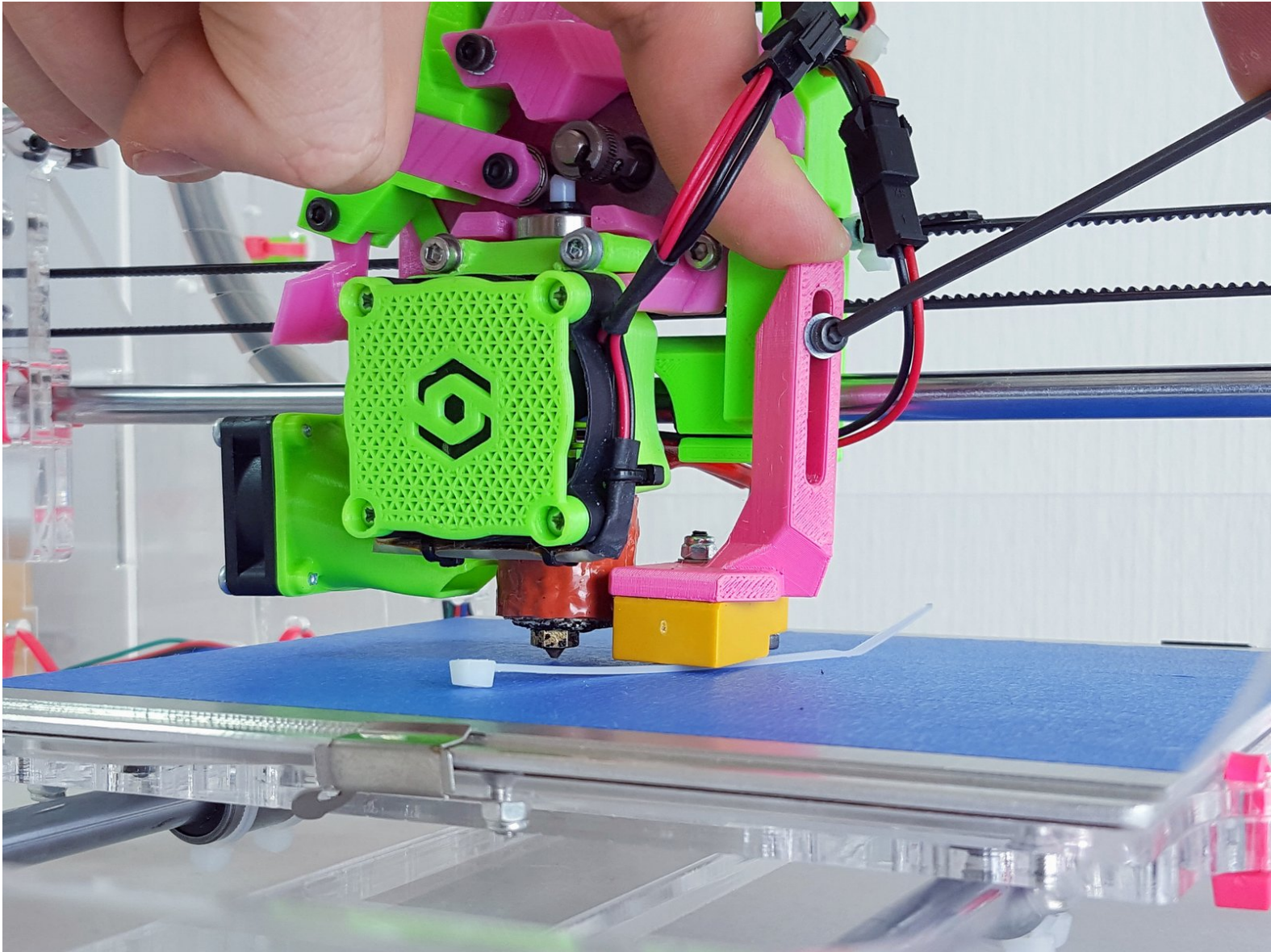




First Layer Height Setup



INTRODUCTION

This is a two part process

PART ONE: You will physically move the sensor - the Z probe - into a position in which it can sense the electromagnetic field of the aluminum built plate.

PART TWO: You will fine tune the 1st layer height by adjusting the "Z Probe Offset" via the LCD controller.

Background

When you print, it is imperative that the model - the object you're printing - is stuck to the build plate; only to be peeled off once the print is finished.

For this, the printer needs to be able to repeatedly and precisely position the nozzle very close to the build plate so that the filament gets stuck to it as it's being extruded.

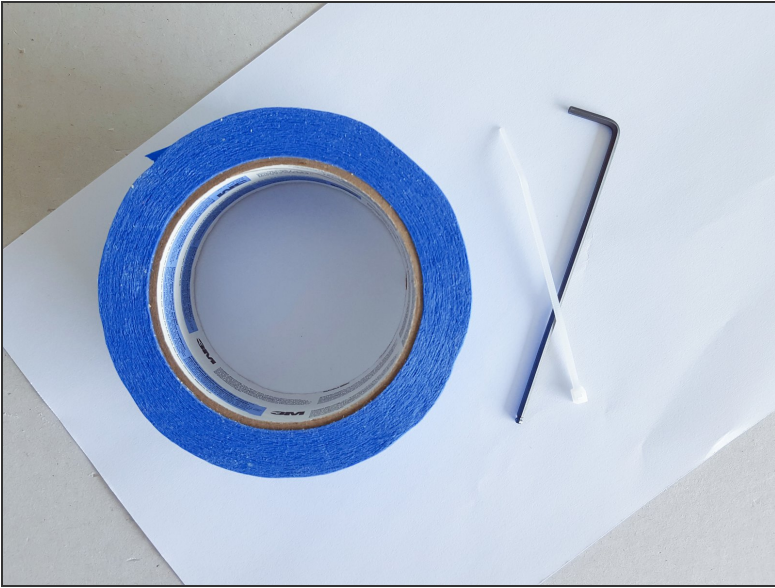
Proximity sensor is a switch that becomes engaged when it comes to a specific distance from a piece of metal.

In Jellybox, the proximity sensor, also known as Z-probe, plays the role of z endstop for Jellybox.

It senses the magnetic field of the build plate and thus determines the z homing position - just like simple mechanical switches determine the x and y homing positions.

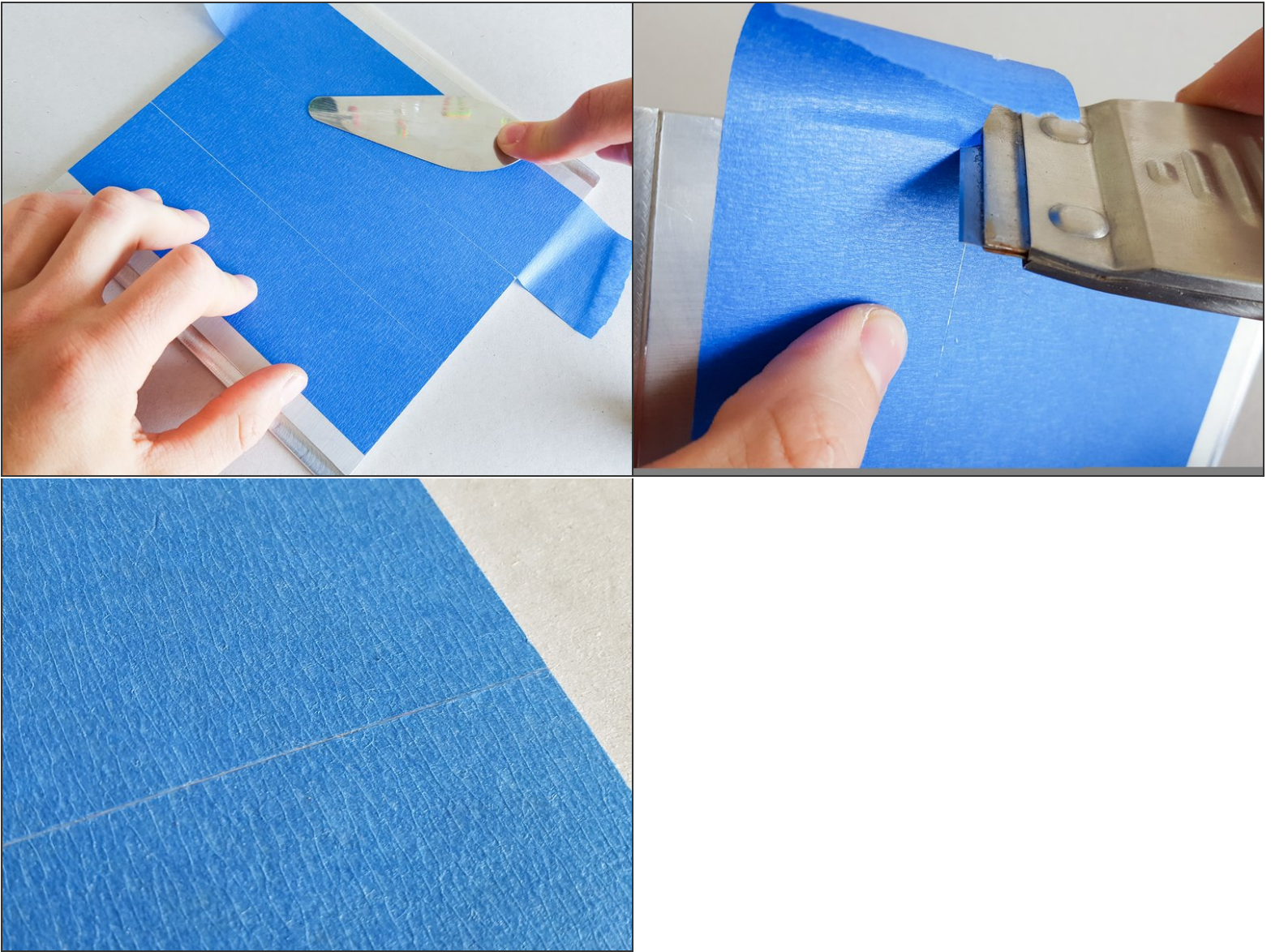
This means by adjusting the z probe, we can change the z homing position and thus also the first layer height.

Step 1 — ↪ Z Probe Mechanical Adjustment



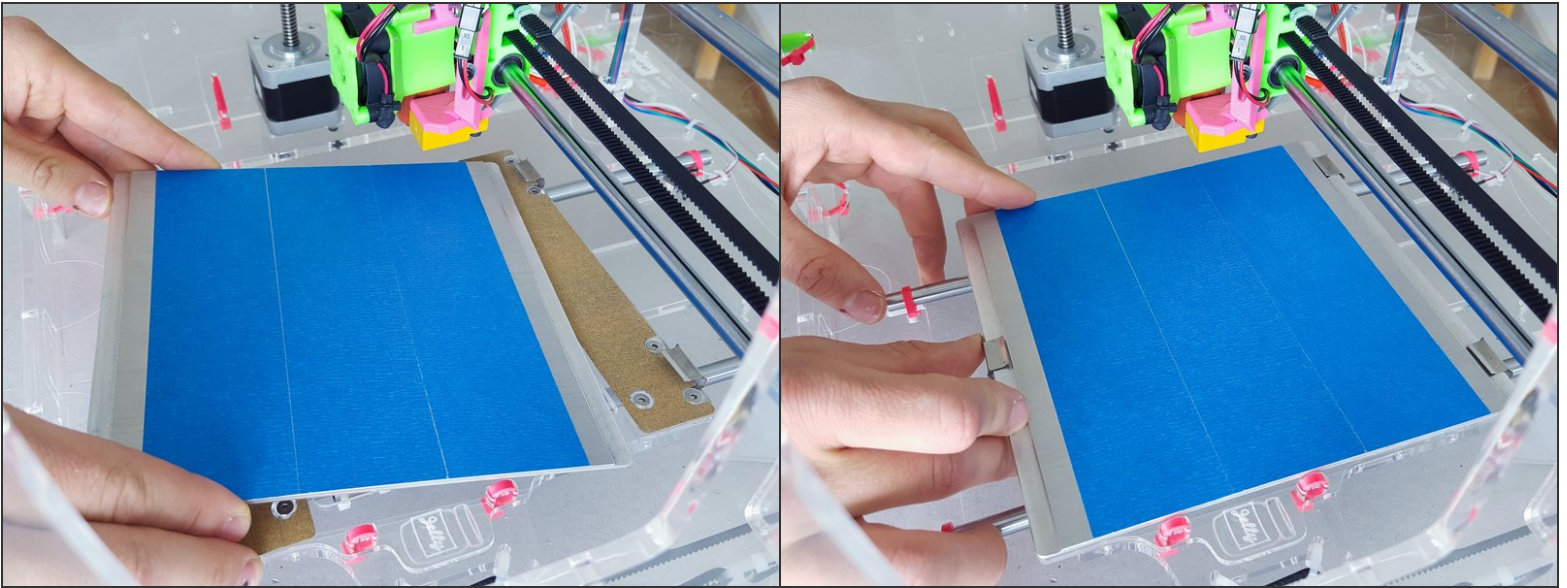
- 2.5 mm M3 hex key
- one miniature zip tie or a leftover from a miniature zip tie
- some blue tape
- a piece of an ordinary paper

Step 2



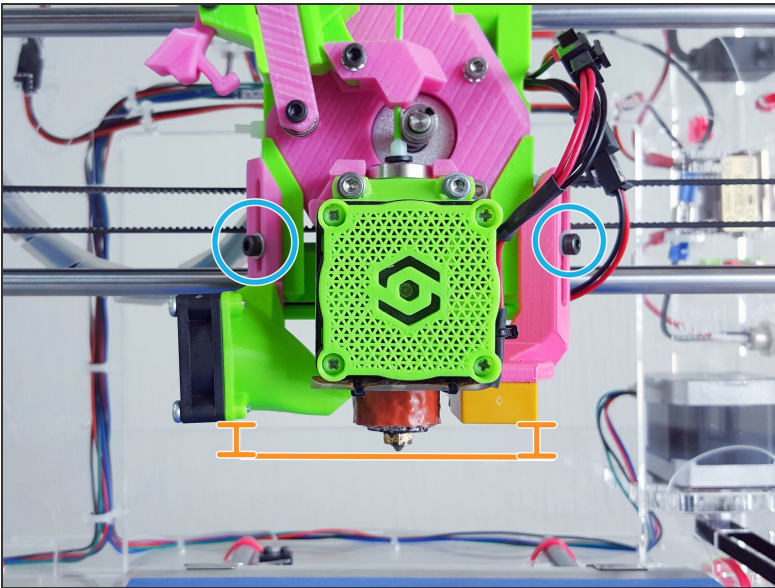
- Put some blue tape on the bed.
- ☑ It's better to have gaps between the tape strips than any overlap. Overlap is not good.

Step 3



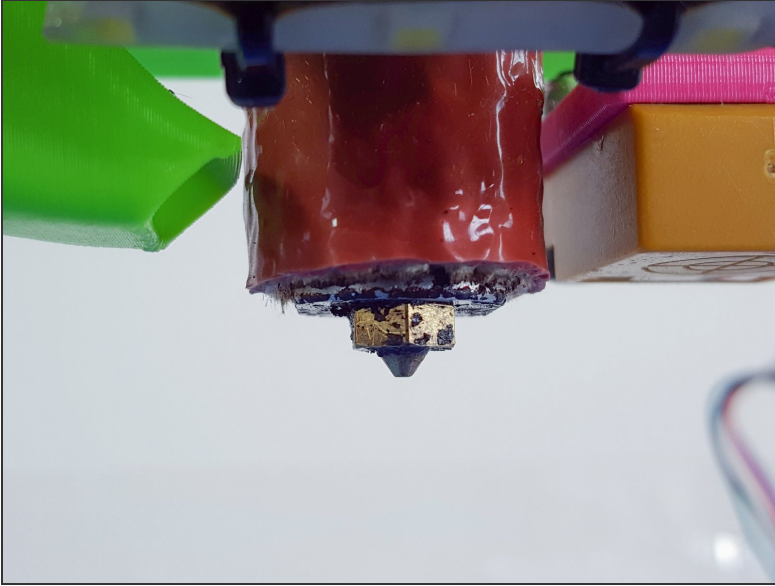
- (re)-insert the bed into the printer

Step 4



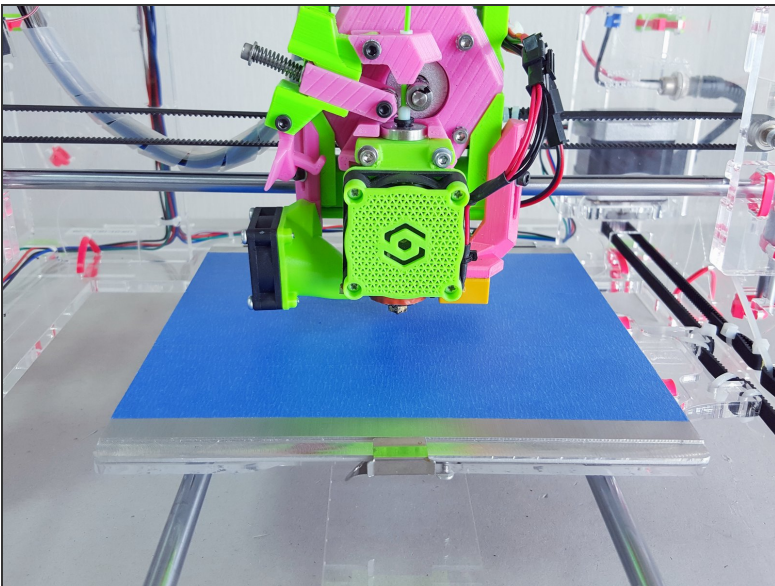
- M3 screws
- Move the filament fan and the Z probe above the nozzle.
- ☑ Do not over-tighten these screws. No need. (Many people tend to over-tighten these screws.)

Step 5



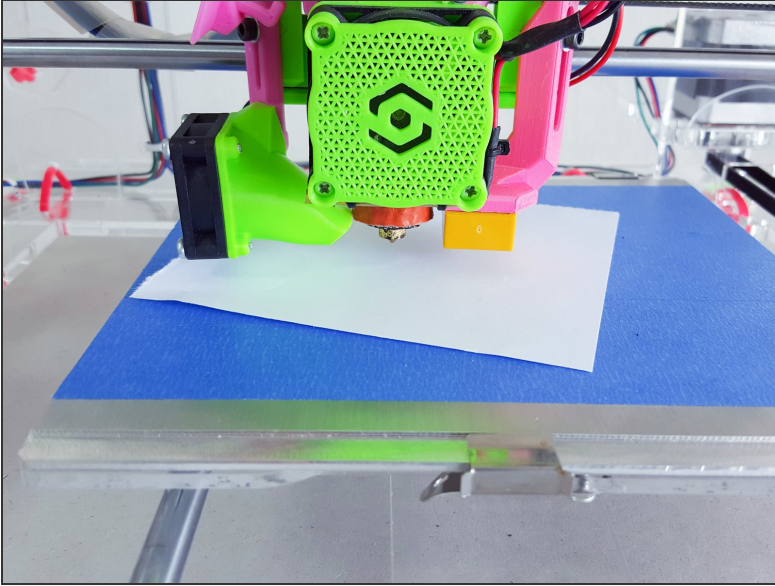
- Make sure the nozzle tip has no plastic build up.
- The nozzle can be dirty, but the tip needs to be clean.
- An easy way to clean the nozzle is to heat it up to a printing temperature, and use tweezers or a brass brush to remove the build up.

Step 6



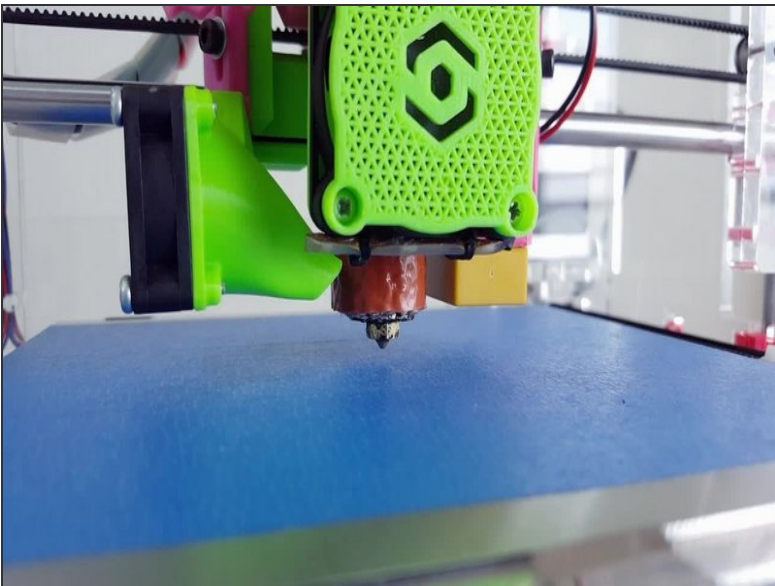
- Position the extruder around the middle of the build plate.

Step 7



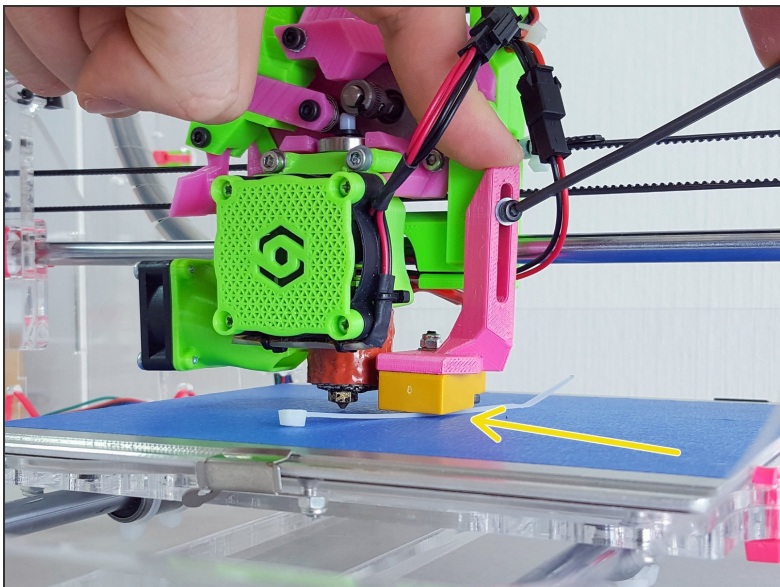
- Slide a piece of paper underneath the nozzle.
- A basic office paper works fine; that's about 0.1mm thick.

Step 8



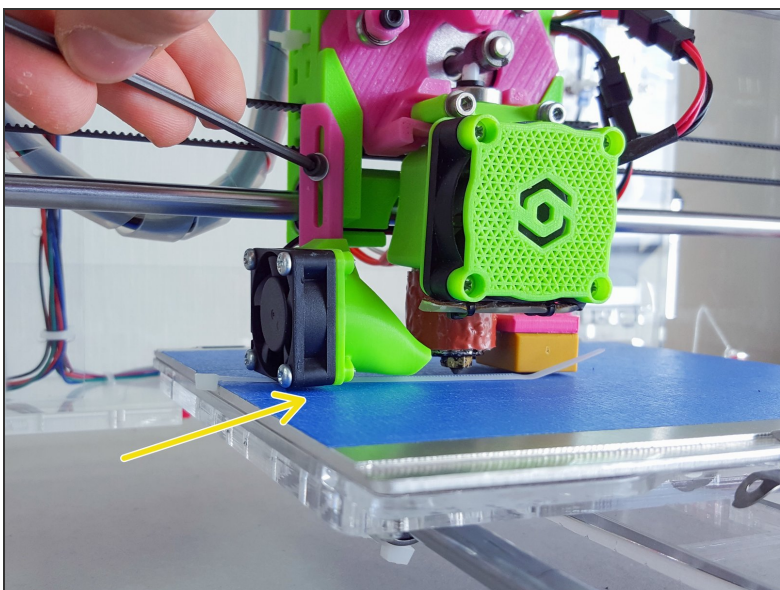
- Turn the z lead screw with your bare hand until the nozzle is lightly touching the build plate.
- A proven way to do this is by touch: when you can't move the paper freely back-and-forth that's when the nozzle is touching the plate.
- You don't have to worry about being crazy precise. The important thing is that you're consistent. Just get a feel for the paper. In a few calibration cycles, you'll be a master.

Step 9



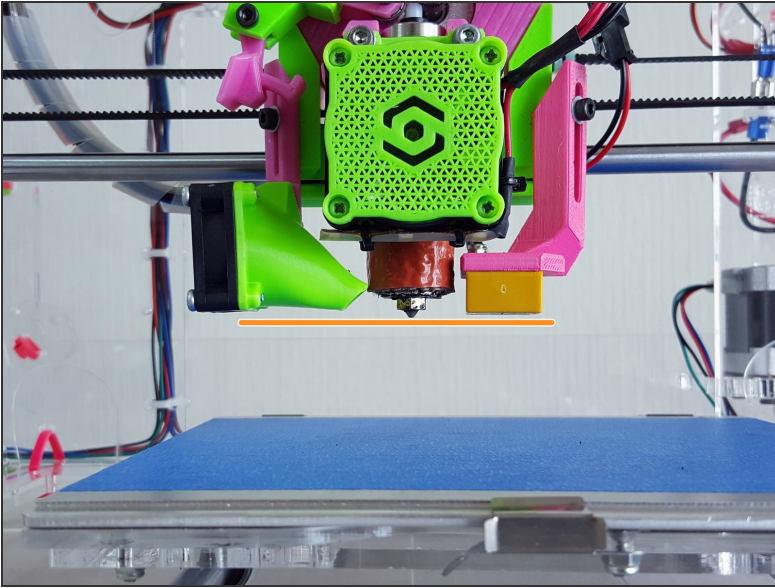
- Rest the probe on a 4" zip tie and tighten the screw to keep it in place.
- Since the thickness of a 4" zip tie is about 1mm, we now have a gap between the tip of the nozzle and lower end of the z probe about 1 mm.

Step 10



- Take this opportunity to adjust the filament fan height as well.
- Rest the filament fan on a 4" zip tie and tighten the screw to keep it in place.
- Good cooling is essential for quality prints from many materials. For PLA it's a must.

Step 11



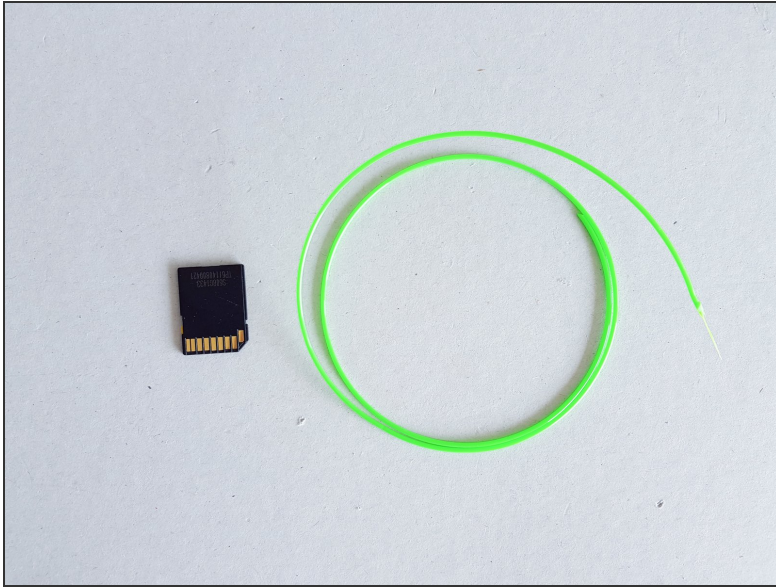
- Now there should be about 1mm gap between the tip of the nozzle and the Z probe.
- This concludes the mechanical part of the Z probe setup. When your JellyBOX probes the build plate, the probe now should be well within its sensing distance.

Step 12 — ↪ First Layer Tweaking



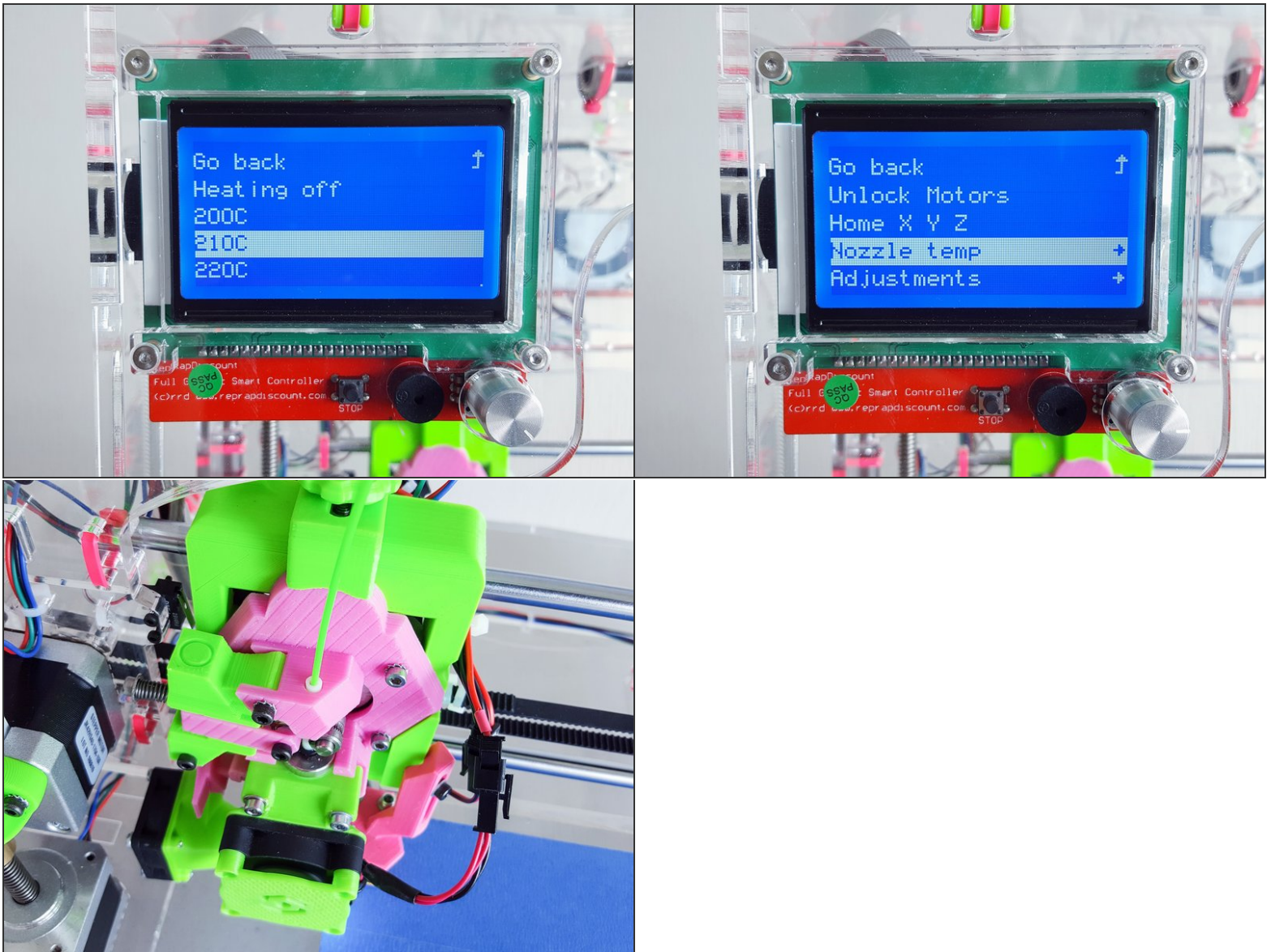
- Z Probe Offset?
- Z Probe Offset adjusts at what height the printer starts printing in reference to the Z homing position.
- The Z homing position is equivalent to the Z probe's sensing distance in our case :-)
- We will print something, analyze the 1st layer, and use the Z Probe Offset to digitally adjust the 1st layer height until the filament is nicely squished and sticking to the build platform.
- The general procedure is simply to start printing anything, and lower the nozzle if it's too high and raise the nozzle if it's too low. That's all there is to it.

Step 13



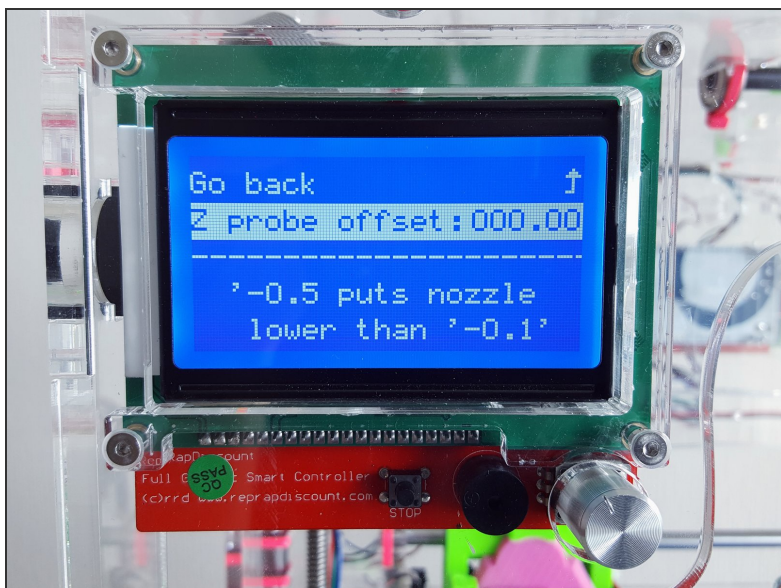
- Get ready the thing you'd normally use for printing; snips, some blue tape...
- some PLA filament
- SD card with a compatible GCODE.
 - JellyBOX comes with an SD card pre-loaded with compatible GCODEs as part of JellyBOX Essentials.
- You can download the GCODE we are using in this guide [here](#).

Step 14



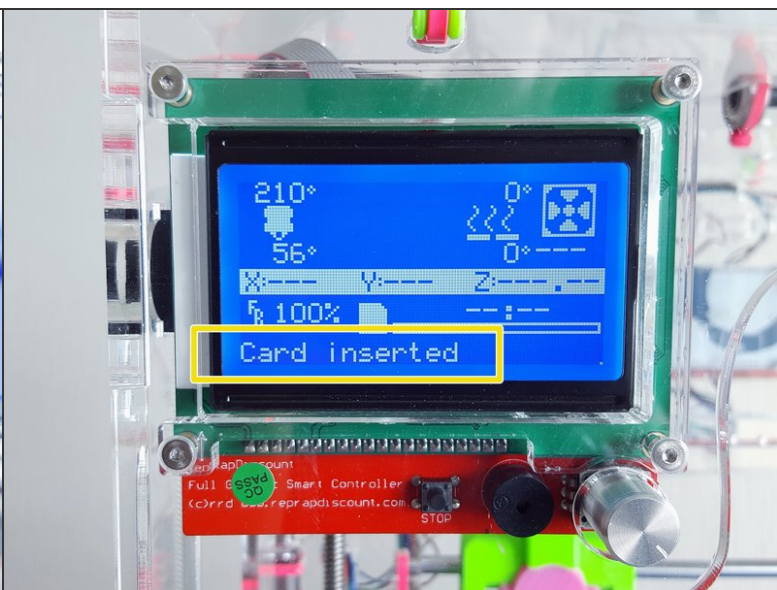
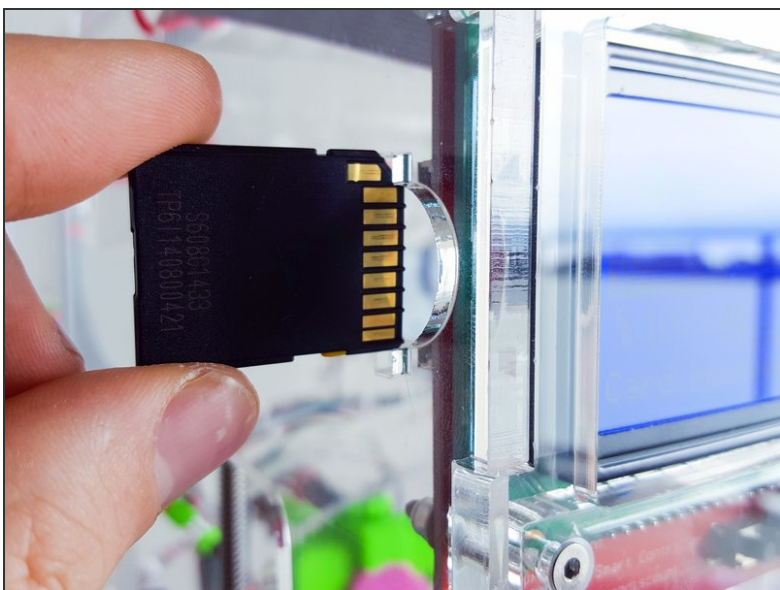
- Preheat the nozzle to 210 °C for PLA
- Load the filament
- Head over to the [Load and Unload Filament Guide](#) if you haven't done this before.

Step 15



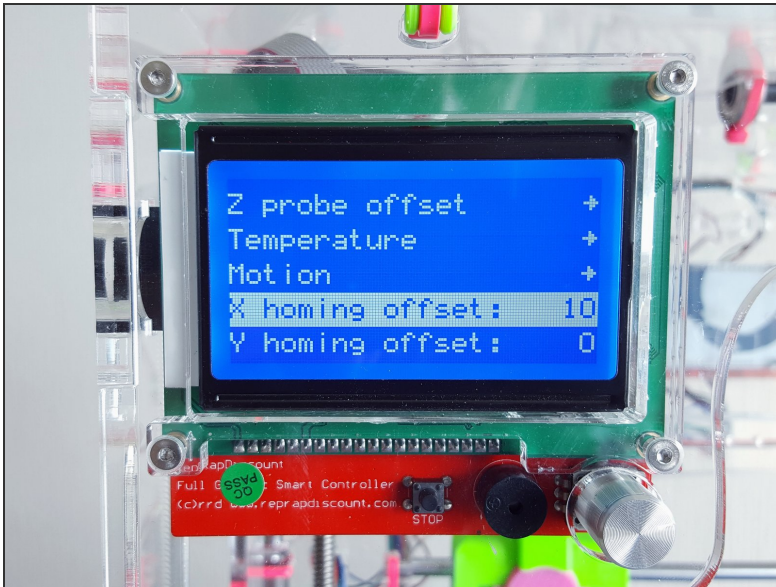
- Make sure the Z probe offset is set to 0.
- Settings > Z probe offset
- This ensures you will not drive your nozzle into the build plate.

Step 16



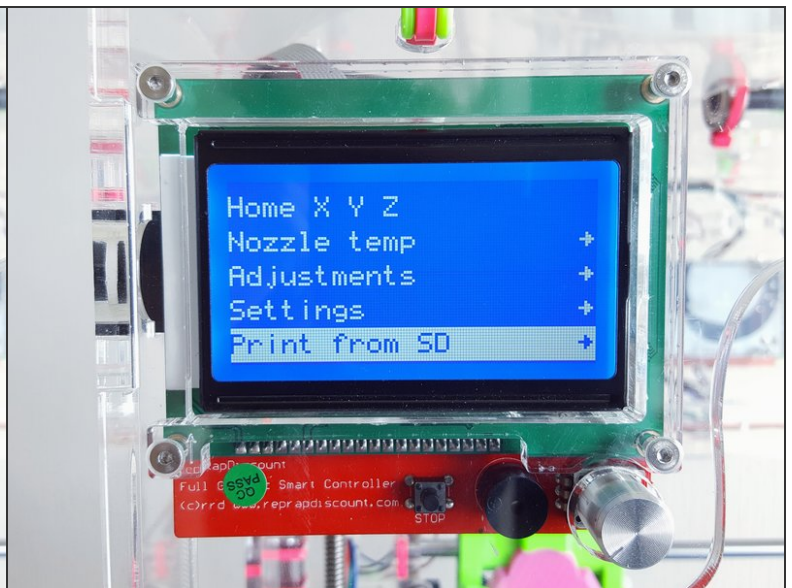
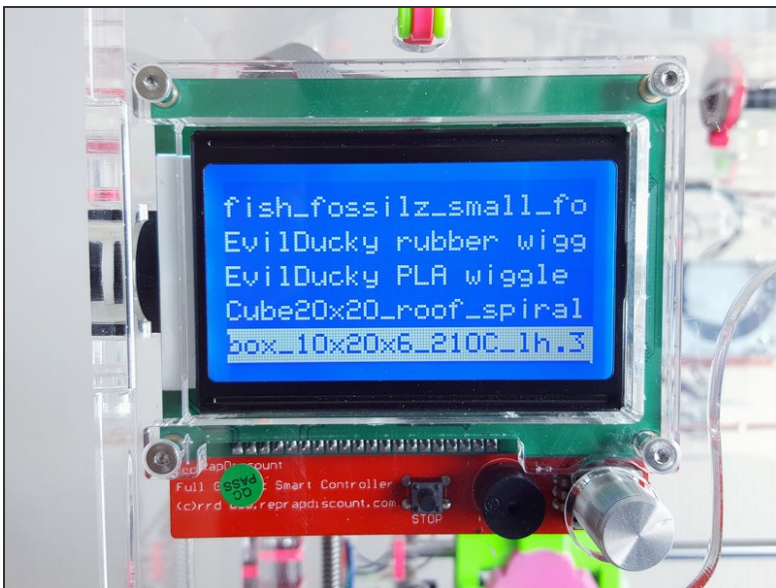
- Insert the card with a compatible GCODE into the reader on the left side of the LCD controller.

Step 17



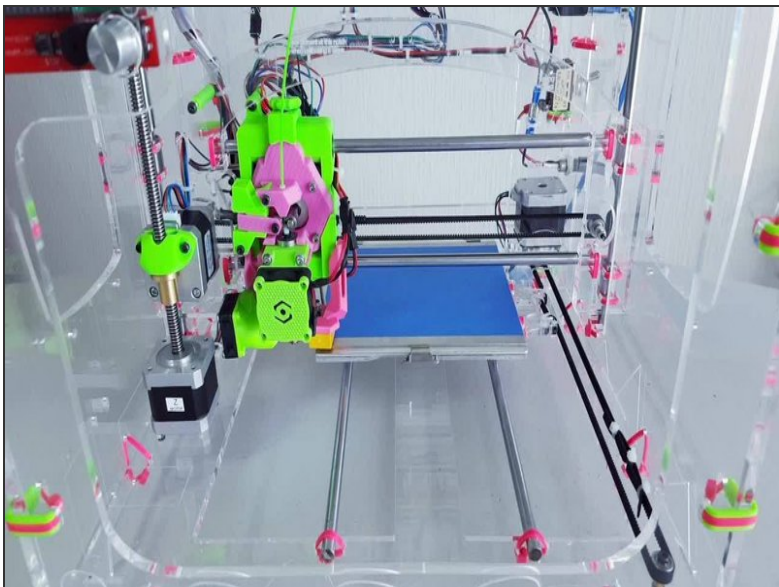
⚠ If you never set the **X Homing Offset**, please do it now.

Step 18



- Press the controller knob once, scroll all the way down > Print from SD > and select your gcode file.
- In our case, GCODES folder and select *calibration_box_30x30x1.gcode*

Step 19



- Jellybox should heat up to the printing temperature, perform "auto bed leveling" procedure, and start printing.
- Wait for the print to start.
- Observe the 1st layer. (Continue reading!)

Step 20

1st LAYER SQUISHINESS REFERENCE

relative additional
Z PROBE OFFSET

TOO
HIGH

0

-0.1

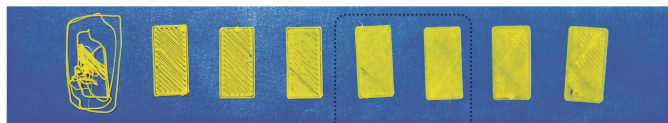
-0.2

-0.3

-0.4

-0.5

-0.6



sticks less

middle
ground

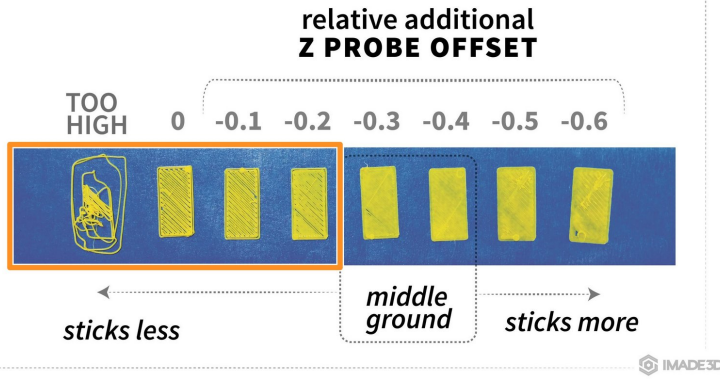
sticks more

IMADE3D

- Observe the 1st layer.
- Use the "1st Layer Squishiness Reference" to determine if the nozzle is too high or too low.
- You can download printable pdf of the 1st Layer Squishiness Reference [here](#).

Step 21

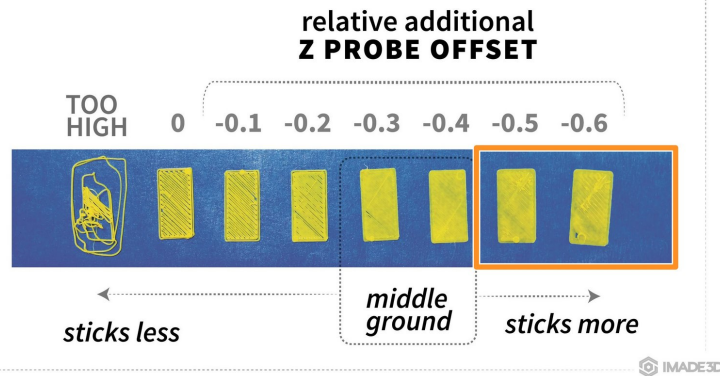
1st LAYER SQUISHINESS REFERENCE



- CASE ONE: The first layer is not squished enough; the print does not stick or you're getting spaghetti.
- The nozzle is too high.
- Therefore, **increase the Z probe offset**, making it 'more negative'.
- For example, "-0.6" becomes "-0.9"

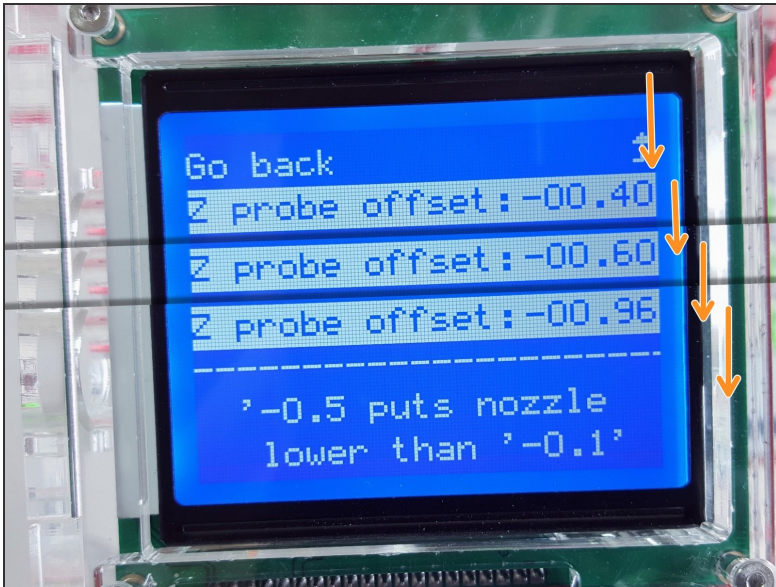
Step 22

1st LAYER SQUISHINESS REFERENCE



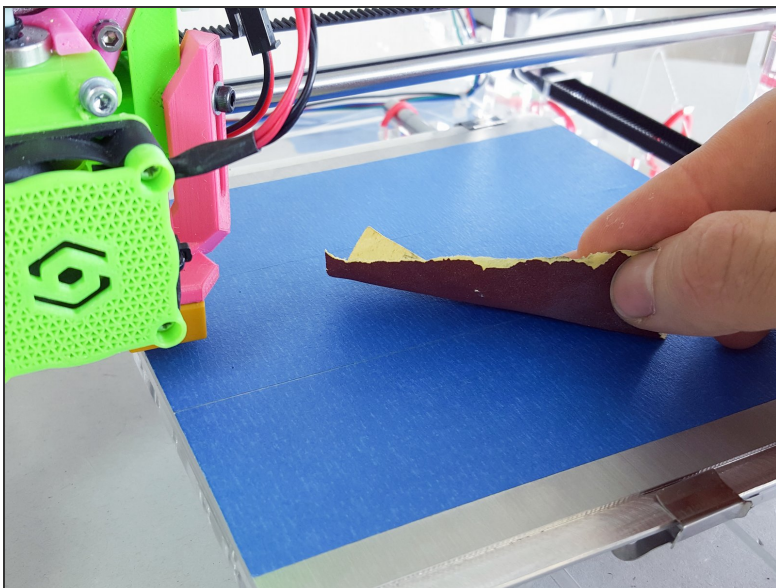
- CASE TWO: The first layer is too squished; you're getting overflow artifact or dragging the nozzle through the build plate.
- The nozzle is too low.
- Therefore, **decrease the Z probe offset**, making it 'less negative'.
- For example, "-0.6" becomes "-0.3"

Step 23



- Repeat the *print - observe - adjust Z probe offset - print* procedure as many times as necessary.
- What you're looking for is a moderately squished first layer with no visible gaps between parallel lines throughout the first layer.
- Squishing the first layer more will get you a better adhesion while making the first layer slightly oversized, and with visible artifacts.
- Squishing the first layer less will get you a more 'precise' print, but with a higher risk of the object lifting and warping.

Step 24



- Tip: Blue tape stops sticking after a few prints. Replace the affected area as needed.
- Tip: Buff the blue tape using some medium grit sandpaper. The sandpaper treatment does wonders - we've been able to even hold even ABS quite well. (Still we do NOT recommend printing ABS. Don't do it. The fumes are not good for your health.)
- Tip: Another way to get the prints to stick well to the tape is to clean the tape with denatured alcohol. Works wonders as suggested by Joel at <https://forum.imade3d.com/t/pre-print-ch...>