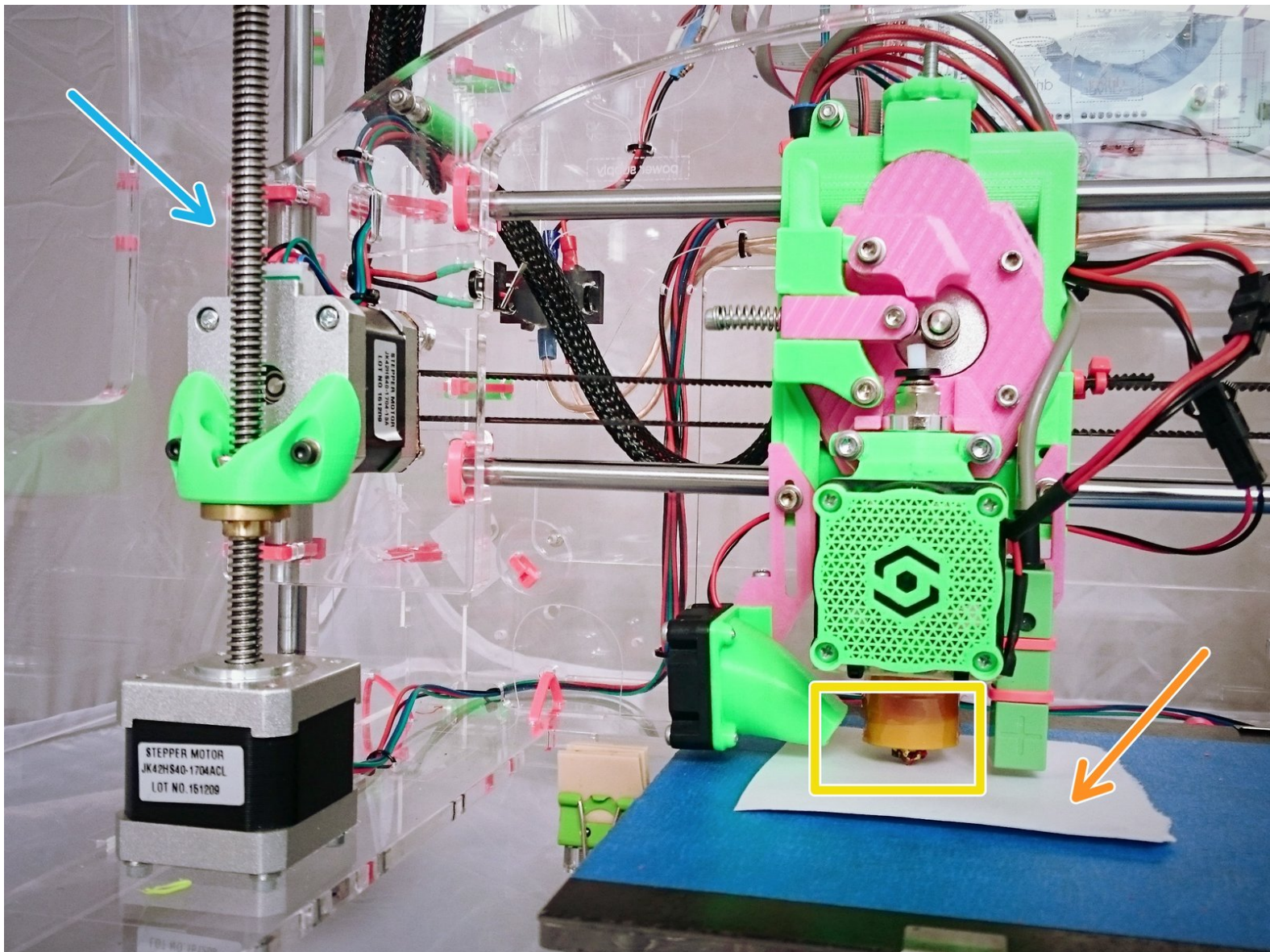




Calibrate Z probe offset

Every hotend and every extruder are slightly different. This guide shows you how to calibrate the distance



INTRODUCTION

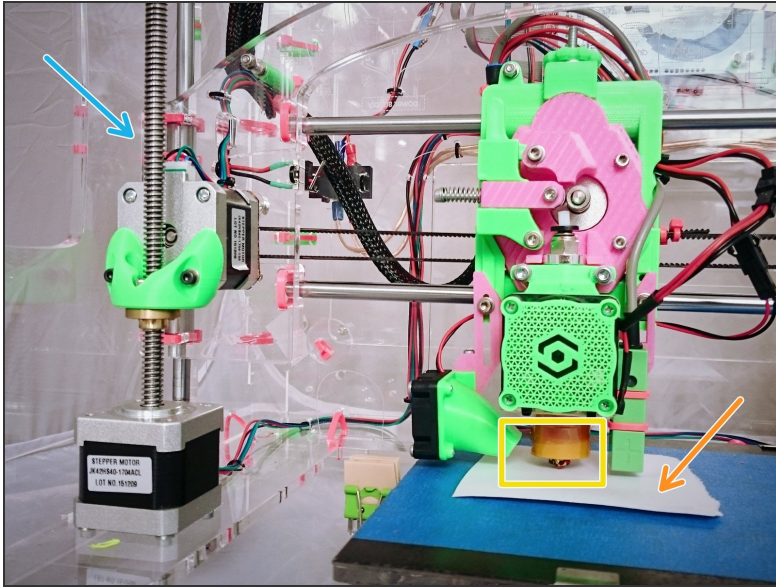
ALERT (optional): NEW PROCEDURE that is simpler and does not require computer at <http://bit.ly/Jellybox-OneNote-Guides>. (It's unpolished, but correct.)



TOOLS:

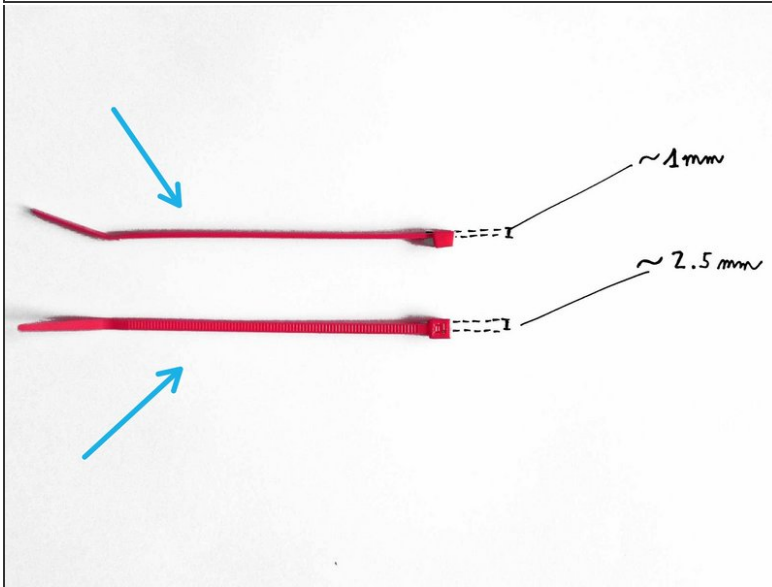
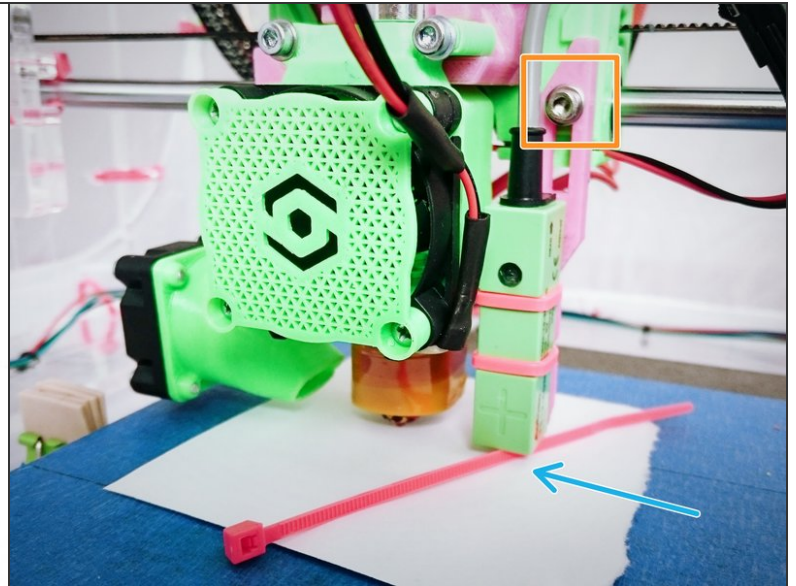
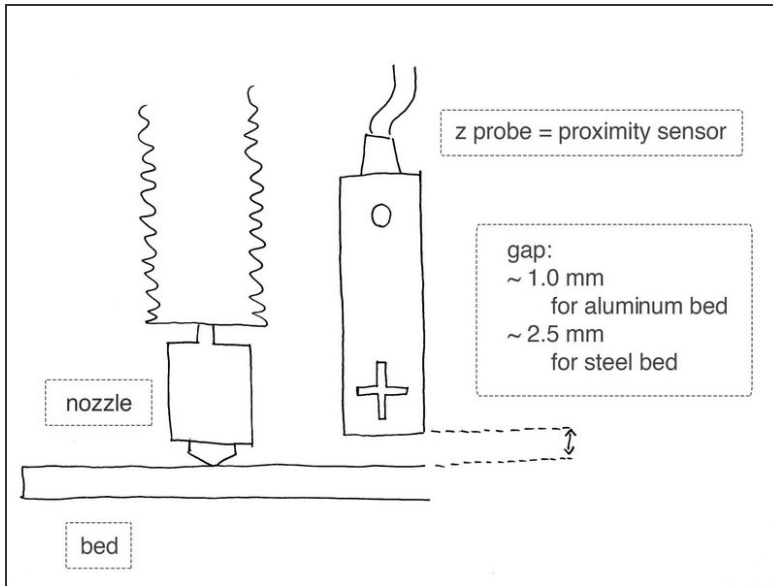
- [sheet of a common office paper](#) (1)
 - [tweezers](#) (1)
 - [filament shears](#) (1)
 - [M3 hex \(allen\) key](#) (1)
 - [USB Cable](#) (1)
 - [computer with Printron installed](#) (1)
-

Step 1 — Position the printhead



- Remove any filament from the nozzle.
- Put a sheet of a common office paper on the bed and keep lowering the nozzle until you can just barely move the paper.
 - This puts the nozzle just shy of touching the printbed. That's our goal.
- *PRO TIP: you can simply rotate the Z leadscrew with your hand to lower the printhead. No need to turn on the printer yet.*

Step 2 — Position the z probe



- We want to position the probe so that it is about 2.5mm away from the bed if you're using a steel bed or about 1mm away if you're using an aluminum bed.
- First, loosen up the z probe mount with your M3 hex key (2.5 mm)
- Then, use a small IMAD3D zip tie as a spacer to get the right gap with ease. The width of the zip tie is ~2.5 mm, and the thickness is ~1mm!

Step 3 — ALERT: NEW PROCEDURE

on this Page (Cmd ⌘)
ELCOME/README
RITTEN POST-ASSEMBLY G
D Printing 101: Flow, STLs ar
re-flight check
oad unload filament
eeder spring tension
Z probe height set up
nozzle fan height set up
st layer tweaking
Compatible gcodes and SC ca
Set up Cura 2+
Slice STL in Cura 2+
Calibrate the extruder
PID tuning - Calibrate the heat
Upgrading firmware (flashing A

Z probe height set up

Tuesday, July 12, 2016 11:19 PM

What up with Z probe and 1st layer height

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.

There's two parts to the process of getting the 1st layer right

1. **This guide** shows you how to physically move the proximity sensor - the Z probe - into a position in which it can actually sense the electromagnetic field of the aluminum built plate.
2. When you're finished here, follow the "1st layer tweaking" guide to fine tune the sensing distance offset in software.

You're going to need:

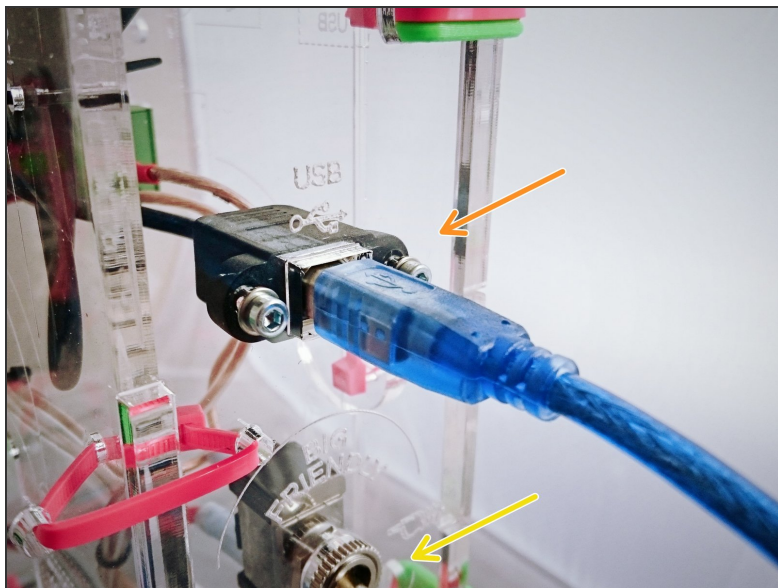
- 2.5 mm M3 hex key
- one miniature zip tie or a leftover from a miniature zip tie
- and things that you normally use for printing that is
 - some filament, snips, some blue tape, and an SD card with some gcode sliced as you normally slice models.

Steps

1. Put some blue tape on the bed.
2. Loosen the M3 screw holding the z probe in place.
3. Turn the z lead screw with your bare hand until the nozzle is lightly touching the build plate. The easiest way to do this is by touch. Slide a piece of a common office paper underneath the nozzle and when you can't move the paper freely back-and-forth that's when the nozzle is touching the plate.
4. We need the gap between the tip of the nozzle and lower end of the z probe to be about 1 mm, which is about the thickness of a miniature zip tie. So rest the probe on a miniature zip tie and tighten the screw to

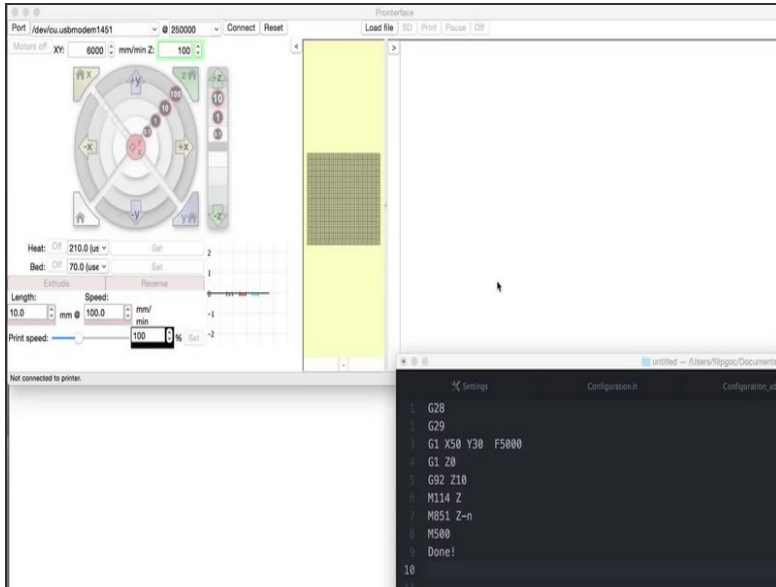
- While this guide *can* be used as-is, there is a simpler procedure that does not involve using computer at all.
- You can find draft of instructions (unpolished but correct) at <http://bit.ly/Jellybox-OneNote-Guides>

Step 4 — Connect the Jellybox to the computer



- Use a USB cable to connect the Jellybox and the Computer.
- Turn the printer on.
- Open Printron application and connect the printer.
- 📌 Our [Connect to a computer with Printron via USB](#) shows step by step how to acquire Printron and connect your Jellybox.

Step 5 — Find the offset part I

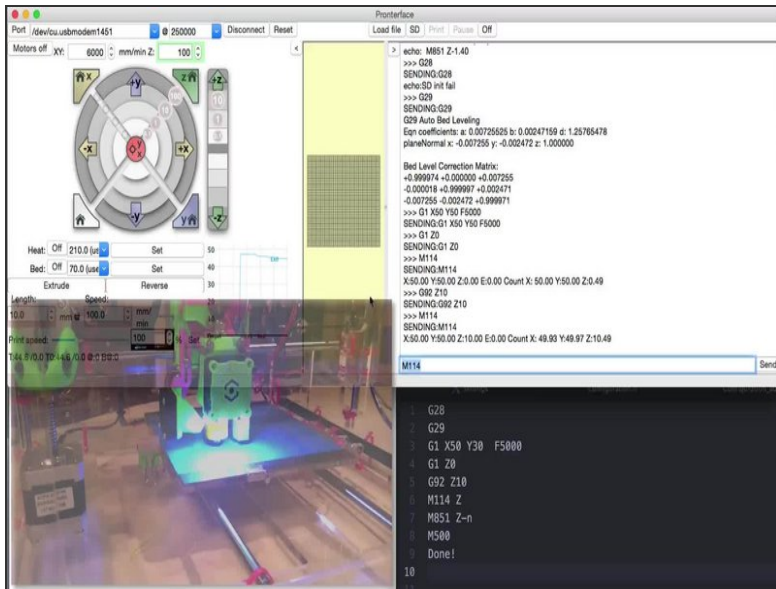


- Follow the video walkthrough to find and save the correct z probe offset. **Gcodes used in the video follow:**

⚠ M851 Z0; Set Z probe offset to 0 (this is missing in the video; sorry)

- G28; home the axes
- G29; level (tram) the bed
- G1 X50 Y30 F5000; position nozzle for easy access
- G1 Z0.0; drive nozzle to Z origin
- G92 Z10; make printer think it's 10mm above origin
- M114 Z; read the value *a* and computer the offset = $10 - a + 0.1$

Step 6 — Find the offset part II



- move the nozzle towards bed until a sheet of paper just barely slides underneath
 - M114 Z; read the value *a* and computer the $offset = 10 - a + 0.1$
 - M851 Z-offset; set the offset as a negative value (!) of the result of previous step
 - M500; save to EEPROM so as the value is not wiped out on power cycle
 - DONE!
- i* This process only gets you into the right neighborhood. There is variance in thickness of paper you're using and in the subjective feel of the paper sliding. *You may need to further tweak the z probe offset with M851 while you watch the printer print until you get the perfect first layer.*

Step 7 — Print something

- Do it!
 - Why else to go through this guide?
- (obviously waiting for 1st print or howto print from computer guide....)