ZONESTAR

# How to upgrade and debug bed auto leveling

**Bed Leveling Sensor: PL-08N Proximity Sensor**

**Ver: 3.0**

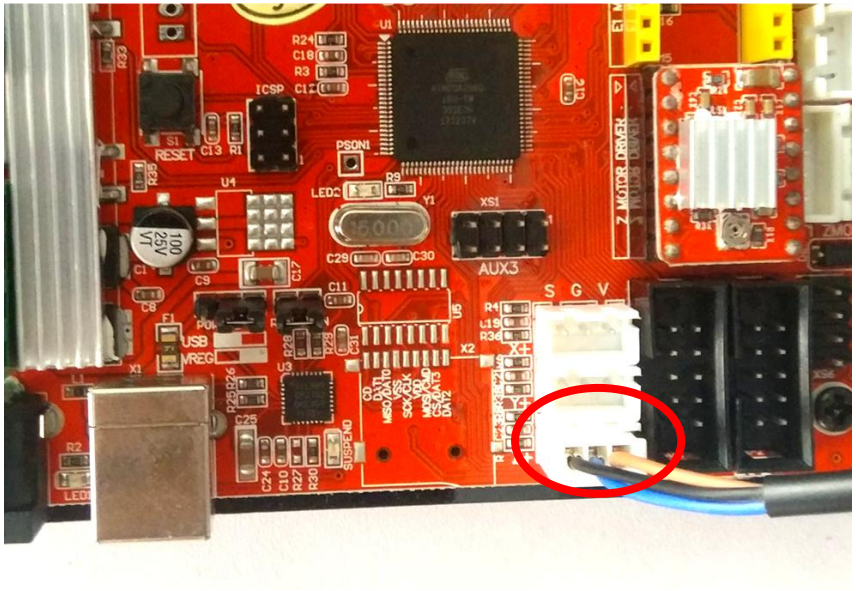
**For Mainlin 2.0.x firmware only**

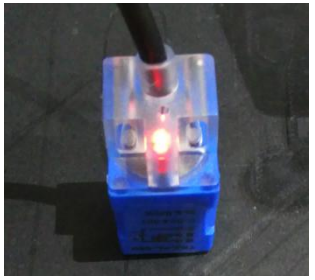
**Note:**

1. **The print platform (hotbed) should be metal.**
2. **Before upgrading , you need to level hot bed manual .**
3. **It can only corrects small irregularities deformation of hot bed, usually error should be less than 0.5 mm.**

# Step 1: Wiring

1. Connect the Proximity Sensor to the Z+ connector on the control board.
2. Power on the control board.
3. When Proximity Sensor is close to the hot bed, the LED will light up and the LED is off when it is far away from the hot bed.



Near, light on

Far, light off

ZONESTAR

# Step 2: Install the proximity sensor

1. Move HOTEND to the middle of hot bed and adjust height of HOTEND or print platform, let the nozzle almost touched hot bed.

2. Install the Proximity sensor on the side of HOTEND. The bottom of the sensor should be approximately 2-4 mm above the nozzle (Fig 1).

3. Adjust the position of Z-axis limit switch (the printer which Z-axis limit switches is removable, such as Z5, Z6, Z8S, etc.) or adjust the "Z height adjustment screw" (the printer which Z-axis limit switches is fixed but there is a screw to adjust the Z height, such as P802N, Z9, etc.), and let their position to meet the following conditions (Fig 2).

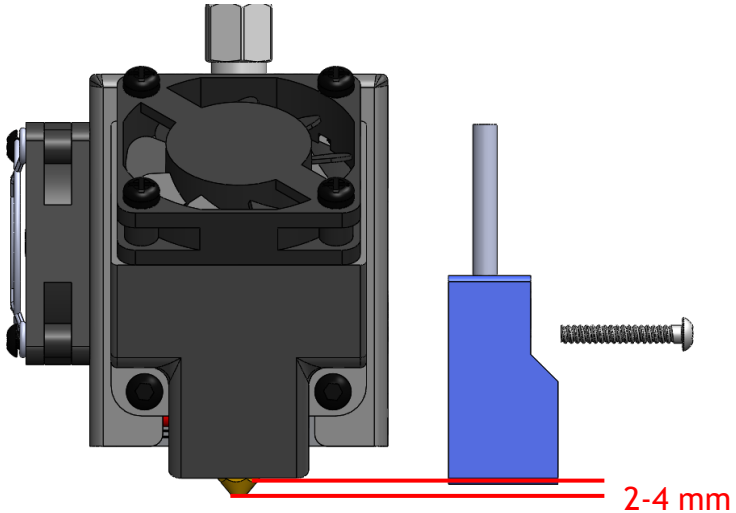



Fig1

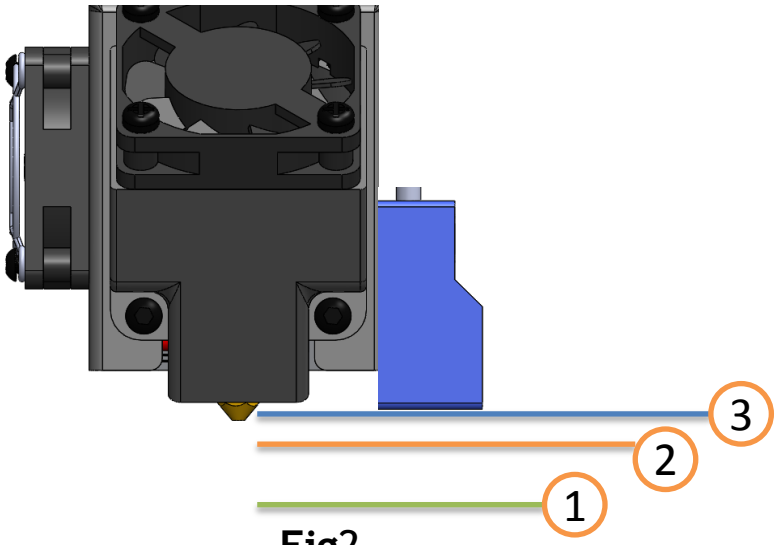



Fig2

① If the hotbed is at this height, the proimity sensor is released (LED turn off).
② If the hotbed is at this height, the proimity sensor is triggered (LED light up).
③ If the hotbed is at this height, Z ENDSTOP is triggered.

# Step 3: Level Corners

**NOTE: Make sure the hotbed and nozzle are cool, clean the filament on the nozzle if there is.**

1. Turn on the 3d printer.
2. Do *MENU>>Montion>>Auto HOME* (Fig1)
3. Do *MENU>>Montion>>Level Corner*(Fig2)
4. Adjust the screws under the hotend, let the nozzle almost to touch the hotbed (the distance from nozzle to hotbed is less than 0.1mm). (Fig3)


Fig1


Fig2

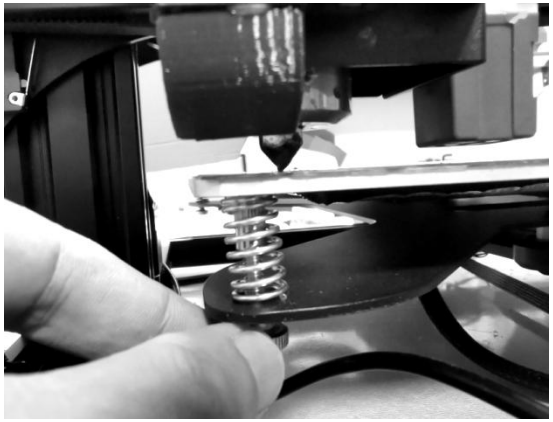


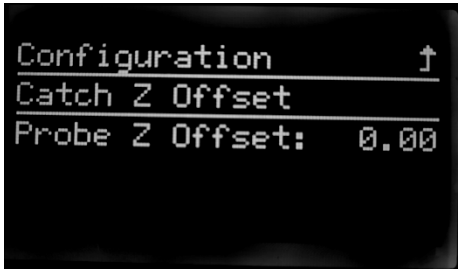
Fig3

# Step 4: Catch Probe Z offset and do bed leveling

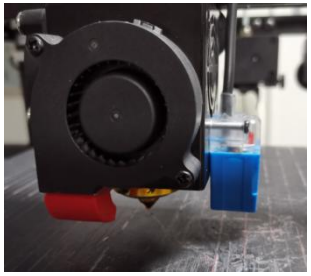
**NOTE1: Make sure the hotbed and nozzle are cool.**

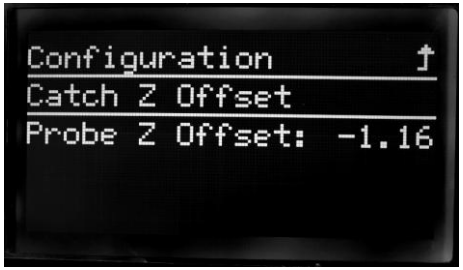
**NOTE2: If you moved the position the Nozzle or Z ENDSTOP or proximity sensor , you may need to do this step again.**

1. Do *MENU>>Configuration>>Probe Z offset>>Catch Z offset* (fig1)
2. After done “Catch Probe Z offset”, the printer will probing 3 times at the center of the hotbed (fig2), to measure the “Probe Z offset” automatically.
3. Once it success finished, the Probe Z offset will be stored into the NVRAM (fig3).
4. You can also modify the “Probe Z offset” on LCD screen if need, do it in *MENU>>Configuration>>Probe Z offset>> Probe Z offset*, remember to do “*MENU>>Configuration>>Store Settings*” after modified it.
5. Now you can do *MENU>>Motion>>Level Bed* to start a auto bed leveling measure.



(fig1)


(fig2)



(fig3)

(fig4)

ZONESTAR

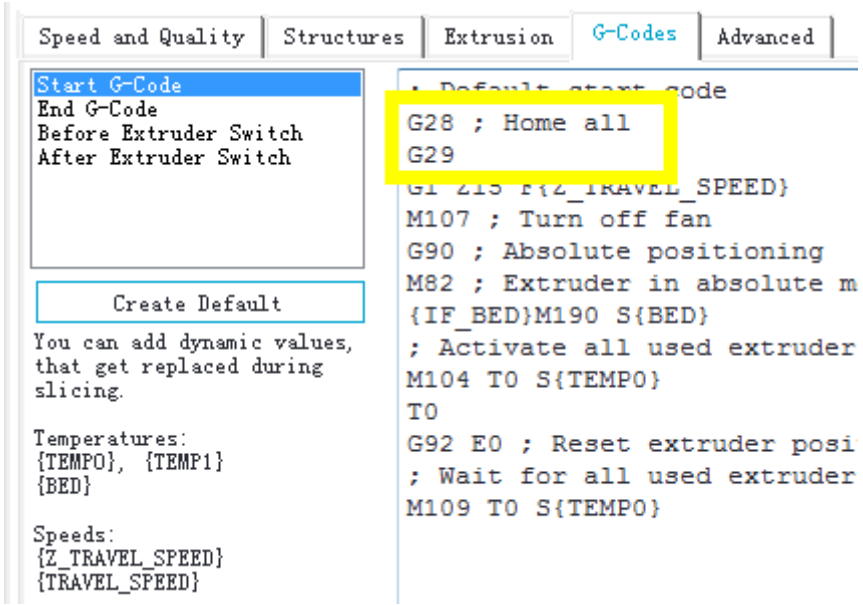
# Verify Z OFFSET

1. Copy "*Bed auto leveling test (YYY).gcode*" file to a SD card and insert it to printer, print this gcode file (YYY is the size of hotbed, for example, Z9 is 300).
2. After starting to print the first layer, double-click the knob to open *the " babystep Z "* menu, then rotate the knob to fine tune the nozzle to appropriate height (refer to the below picture) **, remember this value (e.g.: -0.25mm).**
3. Open *MENU>>Configuration>>Probe Z offset>> Probe Z offset,* add above to ths Probe Z offset, for exmaple, before the "***Probe Z offset***" *is -1.16, New value is -1.16-0.25 = -1.41mm.*

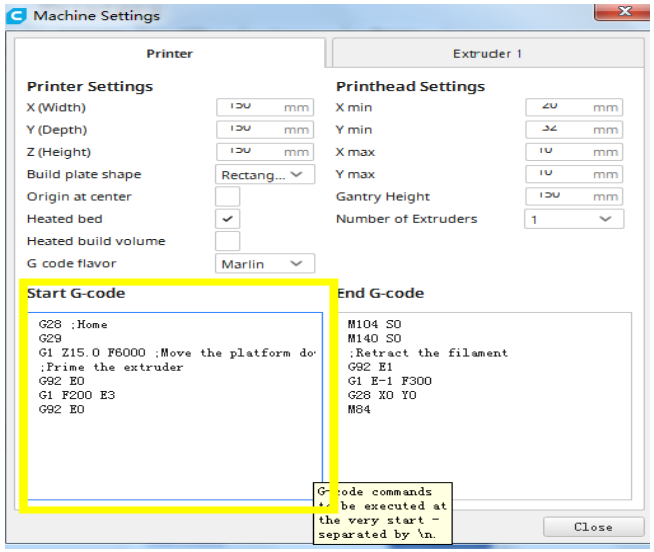
# Enable

*In order to enable the bed auto leveling, you need to add a G29 command behind the G28 command. You can add it to the start code of slicing software.*

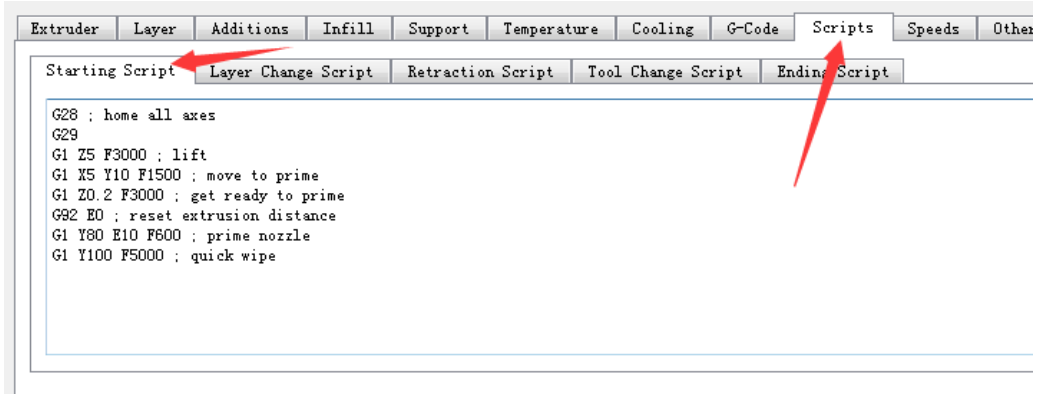
## @ Repetier-host Cura Engine




## @ Cura




## @ Simplify3d




ZONESTAR