



# How to upgrade and debug bed auto leveling

Bed Leveling Sensor: PL-08N Proximity Sensor

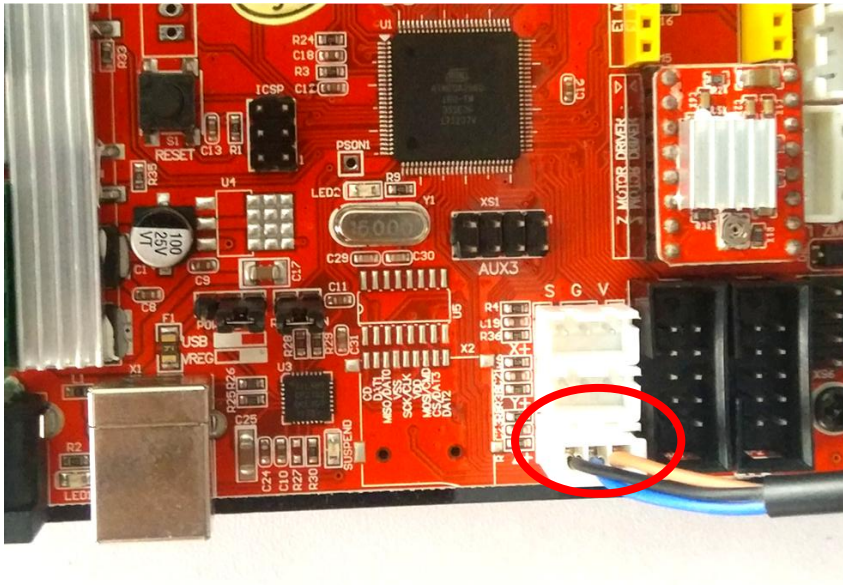
Ver: 2.0

**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.

# 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

# Installation

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).

- ① *If hot bed is at this height, the sensor is released (LED turn off).*
- ② *If hot bed is at this height, the sensor is triggered (LED light up).*
- ③ *If hot bed is at this height, Z ENDSTOP is triggered.*

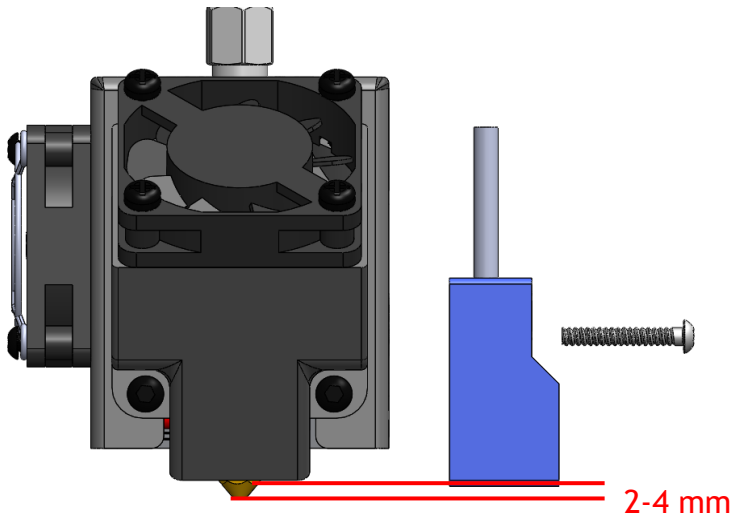


Fig1

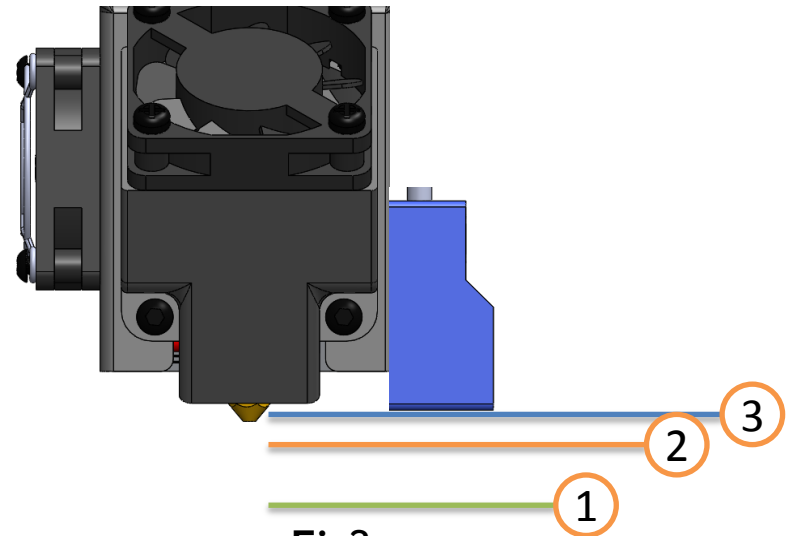


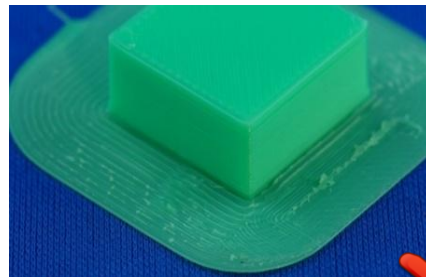
Fig2

# Verify Z OFFSET

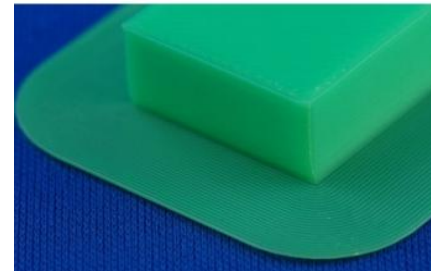
1. First, manually level the hot bed to ensure that the height error of each point of the hot bed is within 0.5mm.
2. Copy "*Bed auto leveling test.gcode*" file to a SD card and insert it to printer, print this gcode file.
3. After starting to print the first layer, double-click the knob to open the "*babystep Z*" menu, then rotate the knob to adjust the nozzle to appropriate height (refer to the below picture) , remember this value (e.g.: **-0.25mm**).



too far



too near



perfect



4. Open the slicing software what you use, add a "G29" command following the "G28 " command, and add "G92 Z**0.25** " following the G29 command.

**Go to the next page**

# Verify Z OFFSET

## Repetier-host with Cura Engine

The image shows two windows from the Cura software. The left window is the 'CuraEngine Settings' dialog, and the right window is the 'P802QR2' configuration window. Numbered callouts (1-5) highlight specific settings and actions:

- 1**: Points to the 'Slicer' dropdown menu in the P802QR2 window, which is set to 'CuraEngine'.
- 2**: Points to the 'Configuration' button (gear icon) in the P802QR2 window.
- 3**: Points to the 'G-Codes' tab in the CuraEngine Settings window.
- 4**: Points to the 'Start G-Code' button in the CuraEngine Settings window.
- 5**: Points to the 'G92 Z0.25' line in the G-Code editor within the CuraEngine Settings window.

**CuraEngine Settings (G-Codes tab):**

```
; Default start code
G28 ; Home extruder
G29 ;
G92 Z0.25
G1 Z15 F{Z_TRAVEL_SPEED}
M107 ; Turn off fan
G90 ; Absolute positioning
M82 ; Extruder in absolute mode
{IF_BED}M190 S{BED}
; Activate all used extruder
{IF_EXT0}M104 T0 S{TEMP0}
{IF_EXT1}M104 T1 S{TEMP1}
G92 E0 ; Reset extruder position
; Wait for all used extruders to reach temperature
{IF_EXT0}M109 T0 S{TEMP0}
{IF_EXT1}M109 T1 S{TEMP1}
```

**P802QR2 Configuration:**

- Slicer: CuraEngine
- Configuration: 802XR2
- Adhesion Type: None
- Quality: 0.2 mm
- Support Type: None
- Speed: Slow to Fast slider
- Print Speed: 50 mm/s
- Outer Perimeter Speed: 50 mm/s
- Infill Speed: 60 mm/s
- Infill Density: 30%
- ☒ Enable Cooling
- Filament Settings:
  - Extruder 1: PLA\_1\_75mm
  - Extruder 2: PLA\_1\_75mm

# Verify Z OFFSET

## Simplify3d

The screenshot displays the Simplify3d software interface. On the left sidebar, the 'Processes' section is visible, containing a table with one entry: 'Process1' of type 'FFF'. A red arrow labeled '1' points to this entry. The main window has a tabbed interface with 'Scripts' selected. Within the 'Scripts' tab, there are sub-tabs: 'Starting Script', 'Layer Change Script', 'Retraction Script', 'Tool Change Script', and 'Ending Script'. A red arrow labeled '2' points to the 'Scripts' tab itself. Another red arrow labeled '3' points to the 'Starting Script' sub-tab. The 'Starting Script' area contains a list of G-code commands. A red arrow labeled '4' points to the line 'G92 Z0.25' in this list. Below the script area, the 'Post Processing' section is visible, showing 'Export file format' set to 'Standard G-Code (.gcode)' and a checkbox for 'Add celebration at end of build (for .x3g files only)' which is currently unchecked. The 'Additional terminal commands for post processing' section is empty.

Processes (double-click to edit)

Name	Type
Process1	FFF

Buttons: Import, Remove, Center and Arrange, Add, Delete, Edit Process Settings

Scripts Tab: Starting Script, Layer Change Script, Retraction Script, Tool Change Script, Ending Script

G-code commands in Starting Script:

```
G28 ; home all axes
G29
G92 Z0.25
G1 Z5 F3000 ; lift
G1 X5 Y10 F1500 ; move to prime
G1 Z0.2 F3000 ; get ready to prime
G92 E0 ; reset extrusion distance
G1 Y80 E10 F600 ; prime nozzle
G1 Y100 F5000 ; quick wipe
```

Post Processing

Export file format: Standard G-Code (.gcode)

☐ Add celebration at end of build (for .x3g files only) Random Song

Additional terminal commands for post processing

# Verify Z OFFSET

Cura

