

Auto leveling feaure user guide (Z9V5Pro)

Bed Leveling Sensor: **ZL-Sensor**

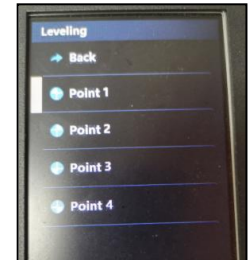
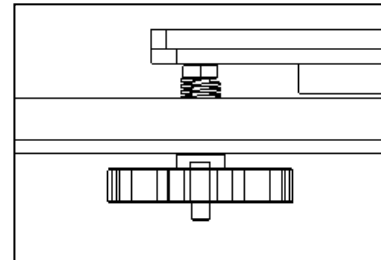
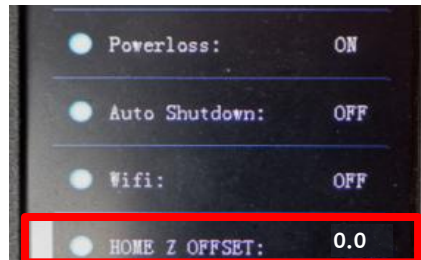
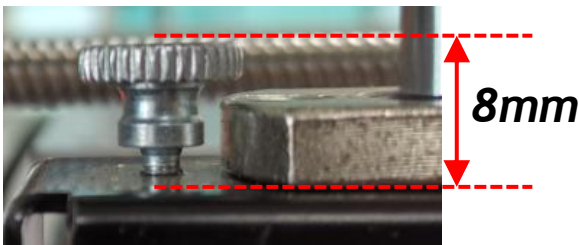
Firmware Version V2.0.0 or last

Note:

If the hot bed is enough level without auto leveling, don't use bed auto leveling feature, Bed automatic leveling will slightly reduce the printing quality and increase the printing time.

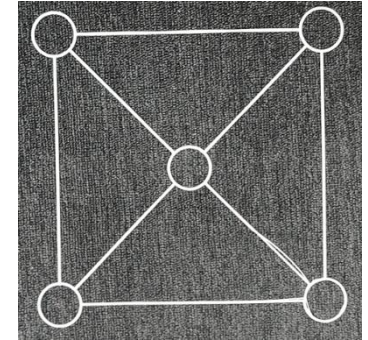
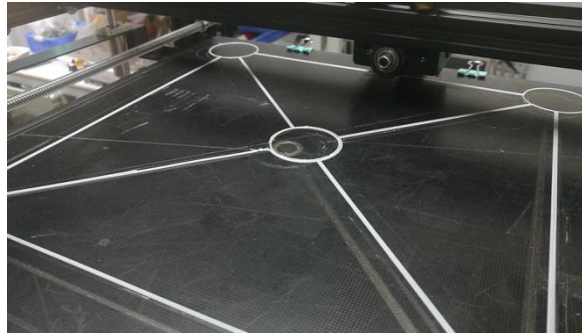
Prepare

1. Adjust the hotbed adjusting screws on the Z carrier, height is about 10mm
2. Set **MENU>>Config>>Configure>>HOME Z Offset: 0.0**
3. Make sure the hotbed and nozzle are cool, clean the filament on the nozzle.
4. Turn on the 3d printer.
5. Do **Prepare>> Bed Leveling>> Point1~4.**
6. Adjust the screws under the hotend, let the nozzle almost to touch the hotend in the four corners.

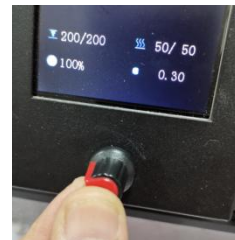


Print test file

After level coners, please try to print "**level_test_310.gcode**" from SD card, and check if the hotbed is enough level.



Press the knob twice within one second to open the babysteps menu, and then rotate the knob to fine tune the distance from nozzle to hotbed.



NOTICE: if the hot bed is enough level, don't need to use bed auto leveling feature. You can ignore the below steps and start to print

Auto Leveling >>> Catch Z offset

Step 1. Do *Control>> Configure>> Auto Leveling ON* to turn on “Auto leveling” menu



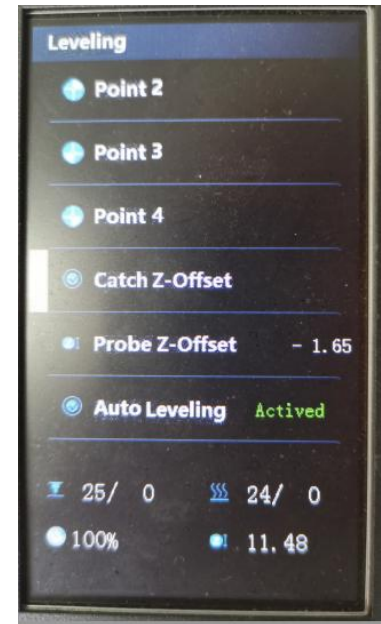
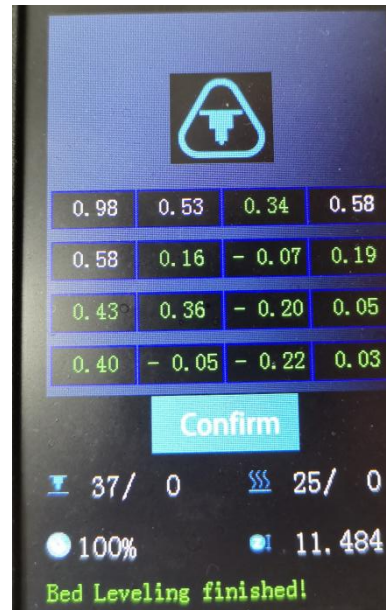
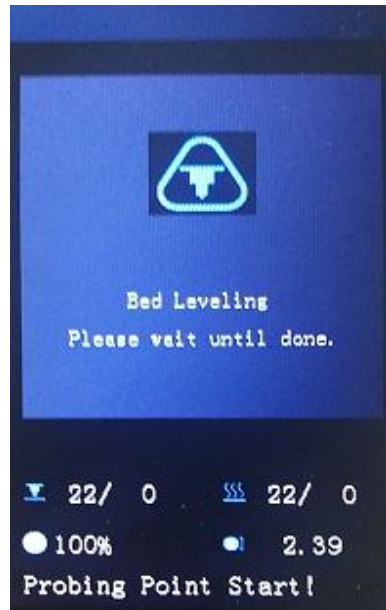
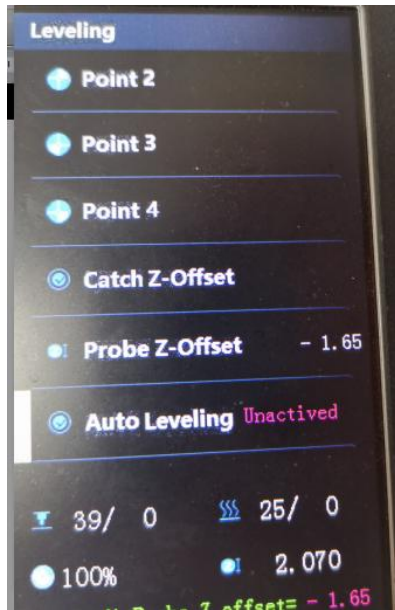
Step 2. Do *Prepare>> Bed Leveling>> Catch Z-Offset*.



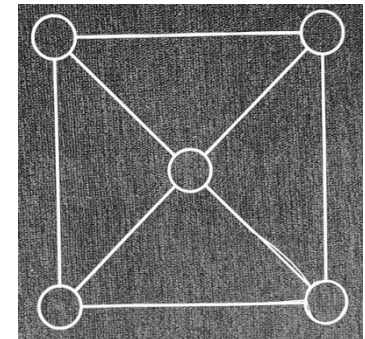
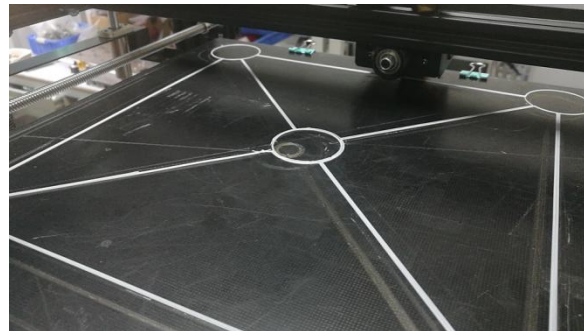
Auto Leveling >>> Leveling measure

Step 3. Do *Prepare>> Bed Leveling>>Auto Leveling*.

After measure done, the state of Auto leveling on Leveling menu will change from **Unactivated** to **Activated**.



Step 4. Print the test file again refer to the page 4



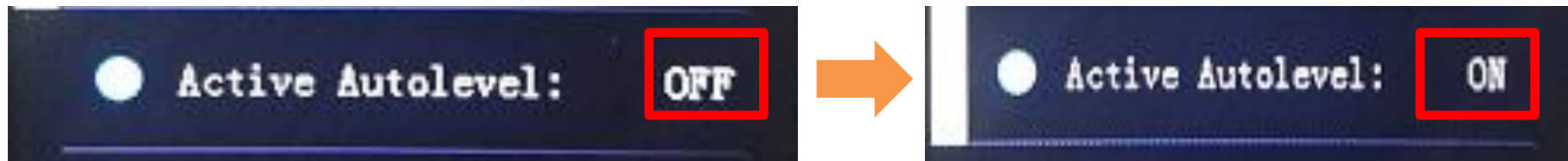
Active auto leveling after printer reset

Auto leveling feature will be disabled automatically when the printer resets, you can turn it on manually from the LCD screen.

Step 1. *Menu>>Prepare>>Auto Home*

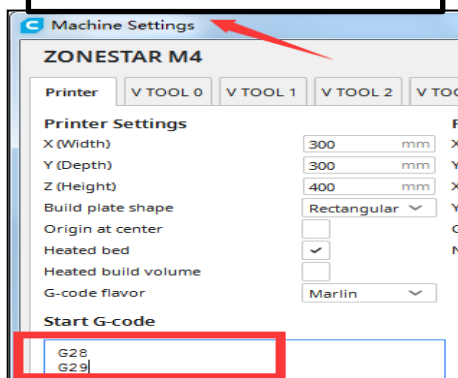
Step 2. *Motion>>Control>>Configure>>Active autolevel: ON*

NOTE: After doing these 2 steps, the printer will apply stored leveling correction parameters in the last "bed level".

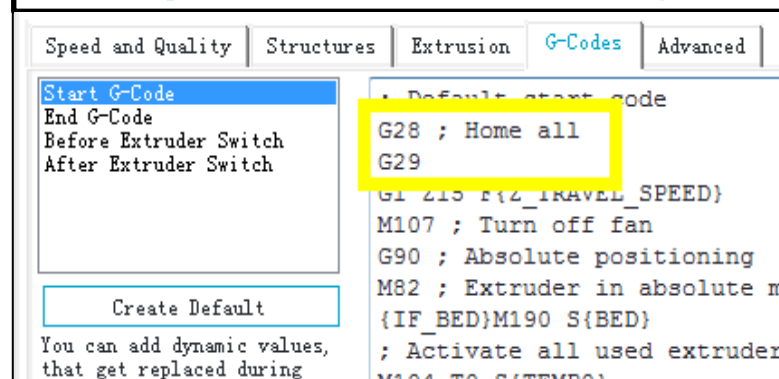


Also, you can let the printer do bed auto leveling for each printing; you only need to add a "G29" command in the "start gcode" of printer settings of the slicing software.

@ Cura



@ Repetier-host Cura Engine



@ Simplify3d

