How to connect and debug Bltouch/3D Touch

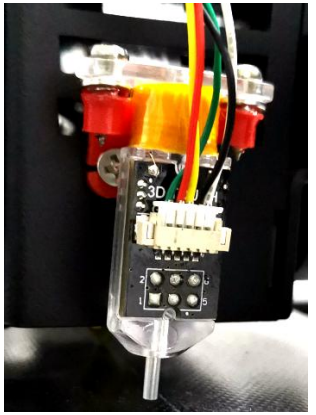
Ver: 4.0

Control Board: ZRIBV6/ZM3E4/ZM3E2

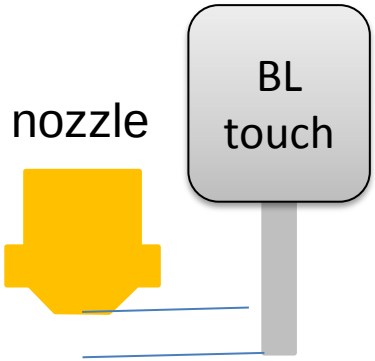
Install the Bltouch to the printer

Install the Bltouch module on the hotend housing, we make a print stl file, please print it and then install the Bltouch on the side of hotend.

Stl file name: BLtouch_Bracket

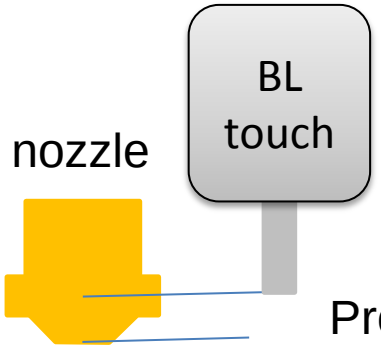


Please adjust the installation position of BLtouch, let its probe is lower than nozzle when Bloutch is deploy, and its probe is higher than nozzle when it is stow.



Probe is lower than nozzle

Deploy(DOWN)



Probe is higher than nozzle

Stow(UP)

Zonestar

Firmware upgrade

If the firmware of the control board supports the BLtouch (3DTouch) sensor, “BLtouch” will appear on the display menu. If the display menu of your machine does not have this item, you need to upgrade the control board firmware.

Upload firmware steps(for 8bit control boards ZRIBV6):

1. Unzip the firmware upgrade tool “Firmware Upload tools.zip” to your computer.
2. Copy the firmware “HEX “to the extracted directory.
3. Connect the printer to your PC, making sure the driver is properly installed.
4. Refer to the “AVRUpdateTools userGuide_ZRIB.pdf” instructions in the upgrade tool to upload the firmware to the control board.

Upload firmware steps(for 32bit control boards ZM3E4 and ZM3E2)

1. Download the bin file
2. Rename the bin file to firmware.bin
3. Copy firmware.bin to the root directoy of Micro-SD card
4. Power off the printer and plug the Micro-SD card into socket on control board
5. Power on the printer, wait about 30 seconds
6. Do the below step on LCD screen to initialize EEPROM after upload firmware:

MENU>>Configuration>>Advanced setting>>initialize EEPROM

Note: For the download link of the firmware and its tools, please note the instructions in the sales link or documentation. If you find a download link, please contact our sales staff.

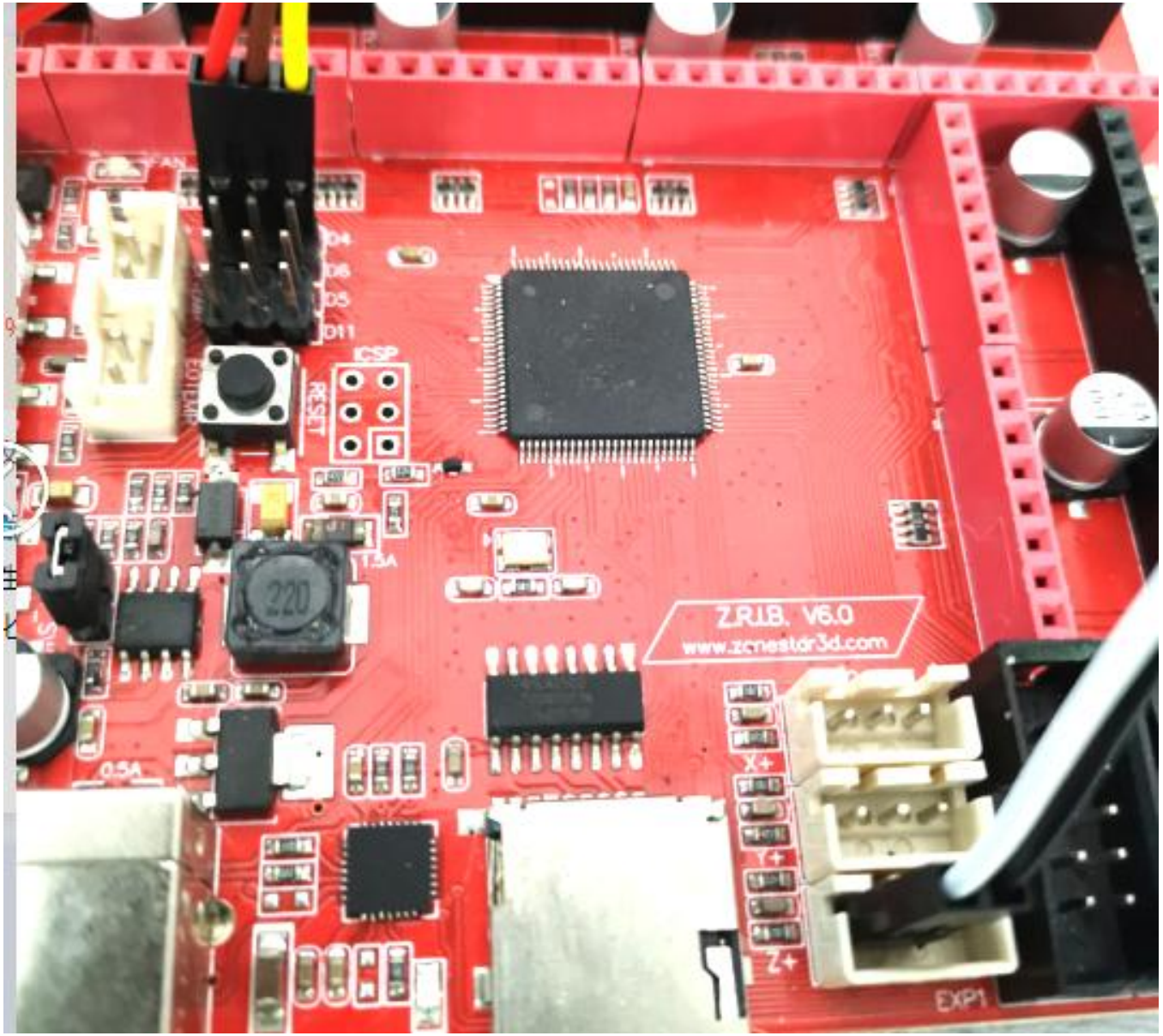
Zonestar

Wires of 3Dtouch

	Bltouch wire color	3D touch wire color	3D touch(V3.1) wire color	Signal
3 PIN connector for driver	Brown	Green	Brown	**GND**
	RED	RED	RED	**+5V**
	Yellow	Yellow	Yellow	**Drive (SERVO)**
2PIN connector for sensor	White	White	White	**Sensor**
	Black	Black	Black	**GND**

Zonestar

Wiring (ZRIB)

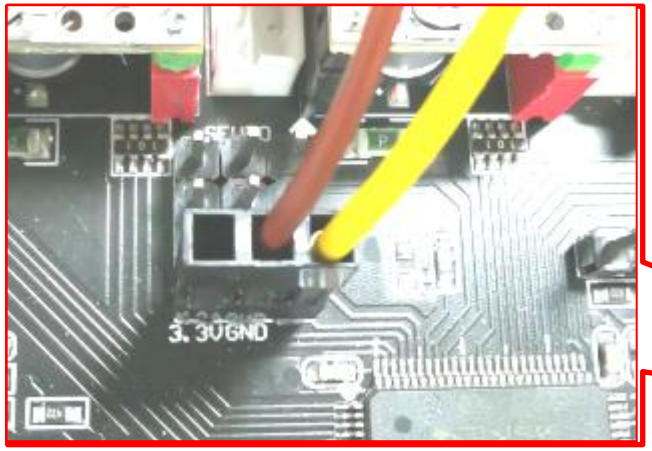


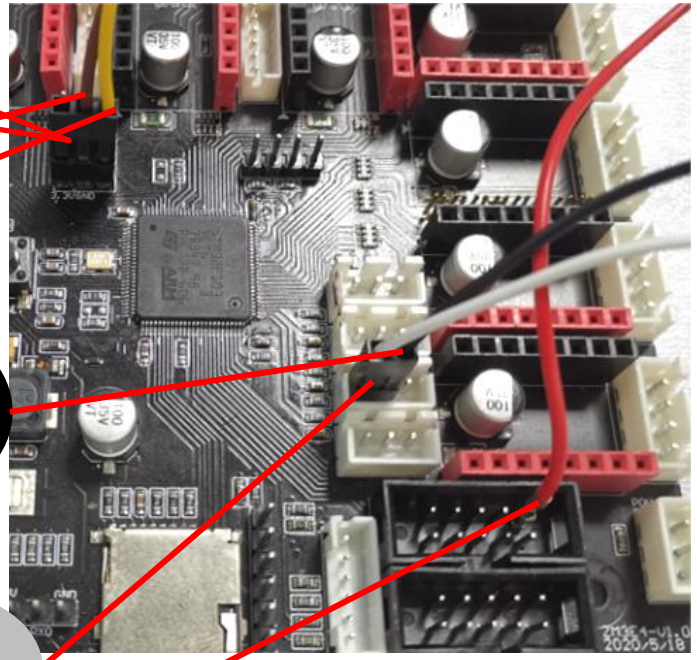
3 PIN
VDD ⇔ +
GND ⇔ -
SIG ⇔ D4

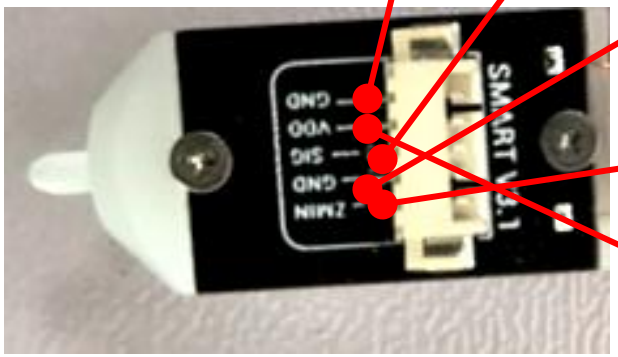
2 PIN
GND ⇔ -
MIN ⇔ S

Zonestar

Wiring (ZM3E4V1)







GND

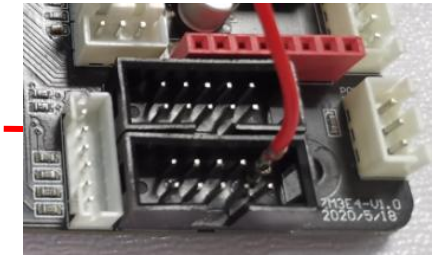
SIG

GND

ZMIN

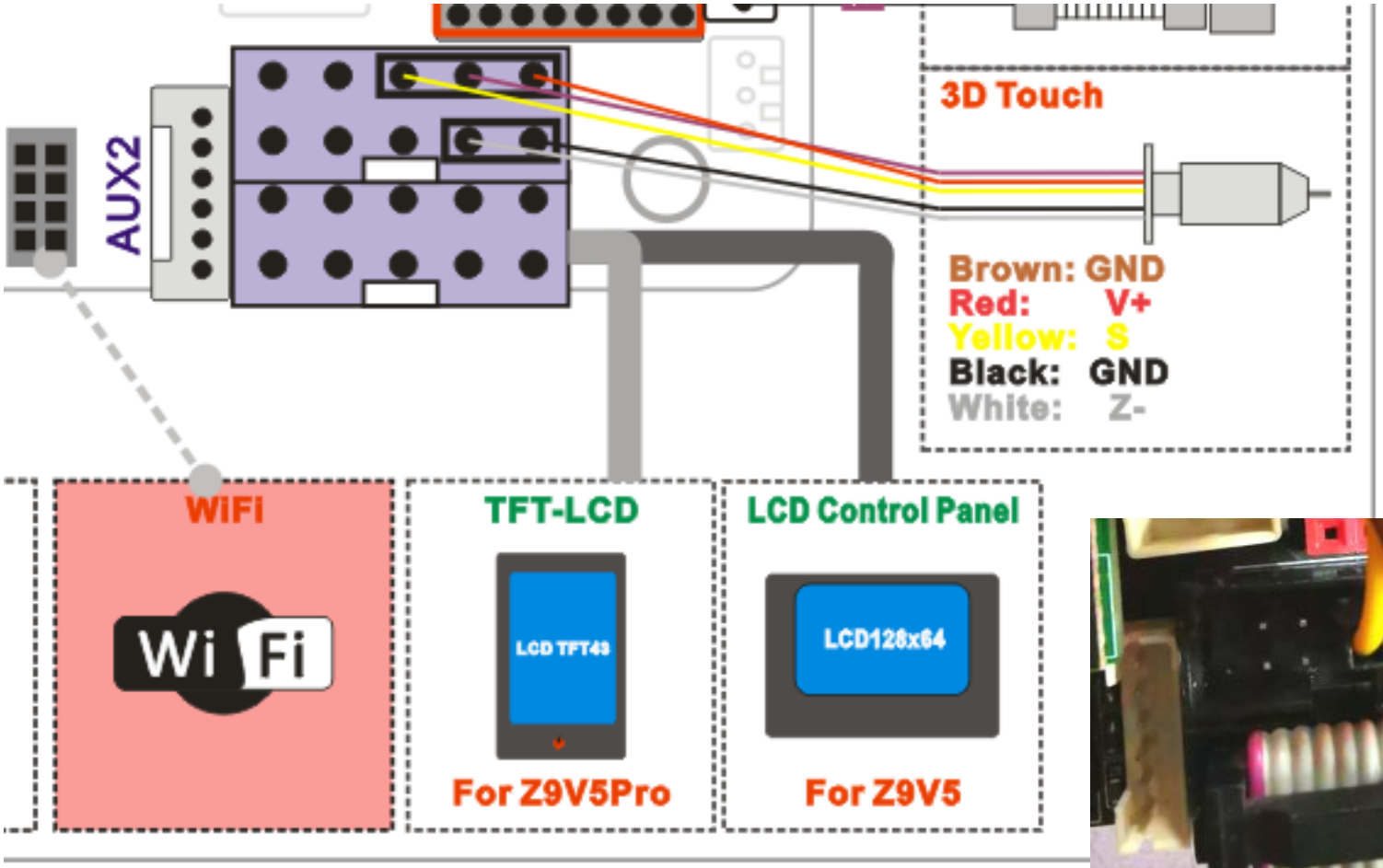
VDD
(+5V)

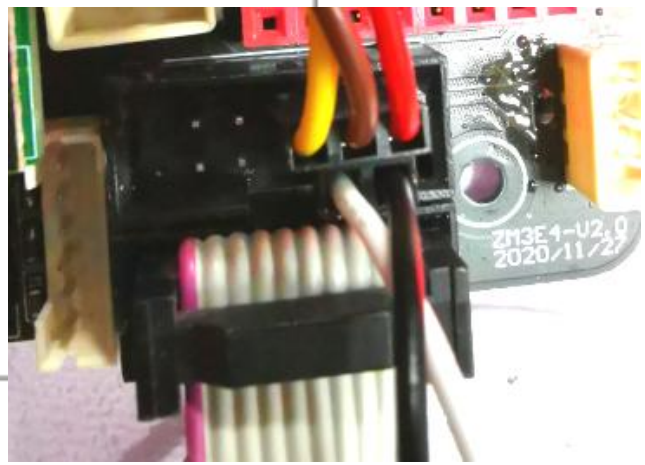
If EXP1 used



Zonestar

Wiring (Z9V5Pro)





Zonestar

Wiring (ZM3E2)

3 PIN	2 PIN
VDD ⇔ +	MIN⇔ SEN
GND ⇔ G	GND ⇔ -
SIG ⇔ PWM	

Zonestar

Verify installation and wiring

1. Power on the printer, and watch the LED on the Bltouch, LED will light if wires is connected well and Power on. Red LED will light if the probe is triggered.
2. Operate the control panel, *Configuration>>Bltouch>>Reset*.
3. Operate the control panel, *Configuration >>Bltouch>>Self-test*, Bltouch will deploy and stow automatically, you can check if the installation height of BLtouch is OK by using this function.
4. Operate the control panel, *Configuration >>Bltouch>>Reset*.
5. Operate the control panel, *Configuration >>Bltouch>>Deploy*, the probe will up;
6. Operate the control panel, *Configuration >>Bltouch>>Stow*, the probe will down;

Level Corners

1. Make sure the hotbed and nozzle are cool, clean the filament on the nozzle.
2. Turn on the 3d printer.
3. Do *Montion>> Bed Leveling>> Auto HOME(Fig1).*
4. Do *Montion>> Bed Leveling>> Level Corners(Fig2)..*
5. Adjust the screws under the hotend, let the nozzle almost to touch the hotend in the four corners (following the wizard)(Fig3).

Fig1

Fig2

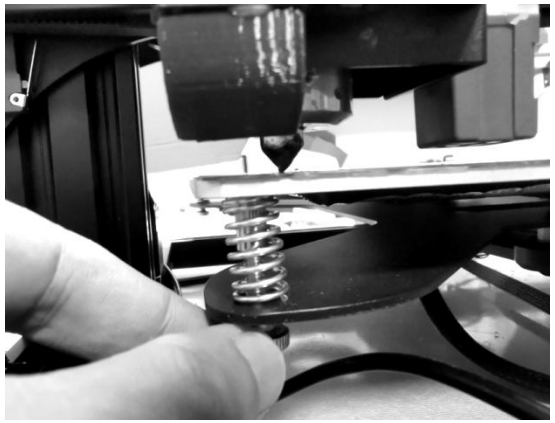


Fig3

NOTE: If you moved the position the nozzle or Z ENDSTOP, you need to do this step again.

Catch Z offset

1. Do *Montion>> Bed Leveling>> Auto HOME(Fig1).*
2. Do *Montion>> Bed Leveling>> Catch Z Offset(Fig2).*
3. Wait for the end of the test to see the Z offset value.

   Do *Montion>> Bed Leveling>> Probe Z Offset(Fig3).*
4. Save the Z offset value.

   Do *Montion>> Bed Leveling>> Store Settings(Fig4).*

Fig1

Fig2

Fig3

Fig4

NOTE: Due to the deformation of the hot bed under heating and unheated conditions, we suggest heating the hot bed to about 60 ° for this test.

Level Bed

1. Do *Montion>> Bed Leveling>> Auto HOME(Fig1).*
2. Do *Montion>> Bed Leveling>> Level Bed(Fig2).*
3. Wait for the end of the test.Operate Auto Home again, the auto leveling will change from off to on. It indicates that the automatic leveling feature is actived (Fig3).

Fig1

Fig2

Fig3

NOTE: It is also recommended that the hot bed be heated to about 60 ° for this test.

Updata Z Offset

When all the previous steps are complete, we provide a gcode file (level_YYY.gcode) to verify the auto leveling.

1. Copy "*level_YYY.gcode*" file (YYY is the size of hotbed, for example, Z9,Z8,Z5X is 300) to a SD card and insert the SD card to printer.
2. Print above leveling test gcode file from SD card.
3. 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**, remember this value (e.g.: 0.2mm). If you do not need to fine tune Z offset, you can stop printing directly.**
4. Open *Motion>> Bed Leveling>> Probe Z offset,* add above vale to ths Probe Z offset, for exmaple, before the *"**Probe Z offset**" is **-2.405**, new value is **-2.405-0.2 = -2.605mm.***
5. Do *Motion>> Bed Leveling>> Store settings.*
6. **Repeat step 4 and step 5**, to check whether the distance from nozzle to hotbed is appropriate when printing.

Apply auto leveling feature

Auto leveling feature will be disable automatically when the printer reset, you can turn it on manually or let it do automatically every time when printing from SD card.

- **The steps of appling it by manually:**

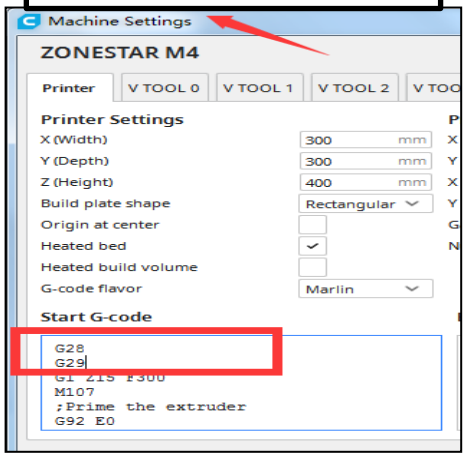
1. *Motion>> Bed Leveling>> Level Bed >>Auto Home*
2. *Motion>> Bed Leveling>> Level Bed >>bed leveling: OFF* →Change to *ON*

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

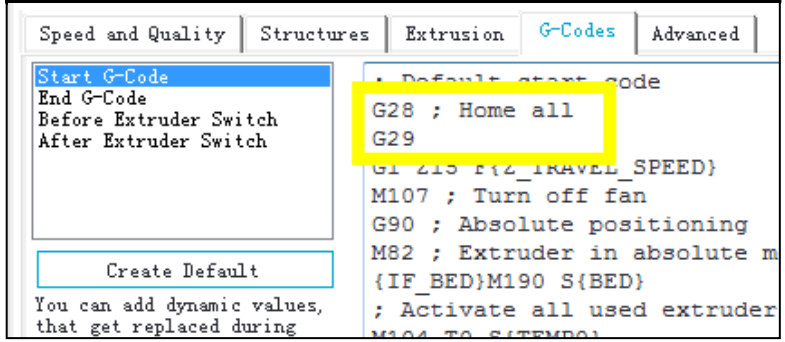
- ***Leveling the hotbed at each printing from SD card:***

Add a G29 command to the start gcode of slicing software

@ Cura



@ Repetier-host Cura Engine



@ Simplify3d