

Debug and Print

catalogue

Debug and Print.....	- 1 -
catalogue.....	- 1 -
1 Connect and Software.....	- 4 -
1 Software Introduction.....	- 4 -
2 Connect printer to PC.....	- 4 -
3 Repetier and control printer.....	- 5 -
1 Software Installation.....	- 5 -
2 Language Settings.....	- 6 -
3 Printer settings.....	- 7 -
4 Connect the printer.....	- 8 -
5 Panel Introduction.....	- 8 -
6 some important command.....	- 12 -
2 Leveling and printing.....	- 19 -
1,Calibration.....	- 19 -
2,Start printing.....	- 31 -
1 Feeder filament.....	- 31 -
2 Slice.....	- 37 -
3 slice setting.....	- 39 -
4 printing.....	- 43 -
3 Marlin.....	- 45 -
1 Arduino.....	- 45 -
2 Marlin.....	- 45 -
1 Open marlin.....	- 45 -
2 settings.....	- 46 -
3 Upload Firmware.....	- 47 -
4 Offline printing.....	- 49 -
1 load the model.....	- 49 -
2 slice.....	- 50 -
3 Export Gcode file.....	- 51 -
4 Copy.....	- 52 -
5 Printing.....	- 52 -
5 Heated bed.....	- 53 -
6 Problem and advice.....	- 55 -

thank you for your kind support to choose our product!

when you use the printer,meet some problems, pls contact us.

Amazon aftersale email 1:dapengqian@126.com

Other email:rose@flsund3d.com

Technical Support skype

1:wangbo5111 Technical Support

skype 2:oldjiawei Yin

whatsapp :008613183017642

1,If you are satisfied with kossel 3d printer kits,please leave a positive feedback to

us,thank you so much for your kindness.

2,If you are not satisfied for any reason,please contact us before leaving a negative

feedback(any negative review no longer

be changed again),we will work together with you to help you solve any problem until

you are satisfied.

Important Note

A,when use switch power supply,If your country AC voltage is 220V,pls choose 220V;if it's110V,pls choose 110V,there is one adjust button on the switch power supply.

B, when heating the nozzle or printing,for the fan which cooling heat sink,guarantee it working,if not,pls stop heating and printing.

C,the temperature is a important parameter, for PLA filament, the working temperature is 200°C-210°C, you could adjust it,and the temperature should not too high.too high temperature maybe lead to Nozzle clogging.

D,Before printing, please paste high temperature tape on the hot bed

E,If you have installed the touch screen ,pls choose PC or touch screen to make a Single control

F,If you need to upload marlin ,when you upload ,pls disconnect the touch screen with the motherboard

1 Connect and Software

1 Software Introduction

Need software as follows:

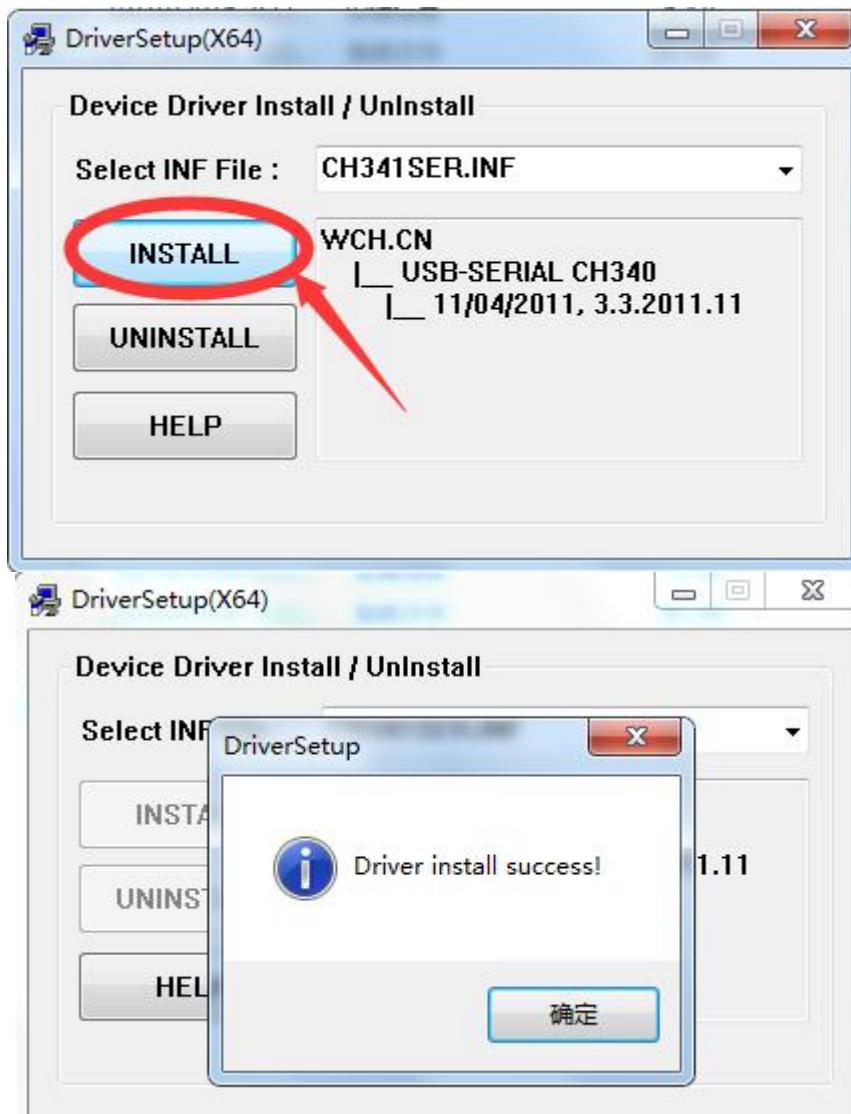
A:**arduino**,it's the firmware working environment.

B:**Marlin**,it's the firmware,it's program to run printer working,need flash firmware to motherboard.

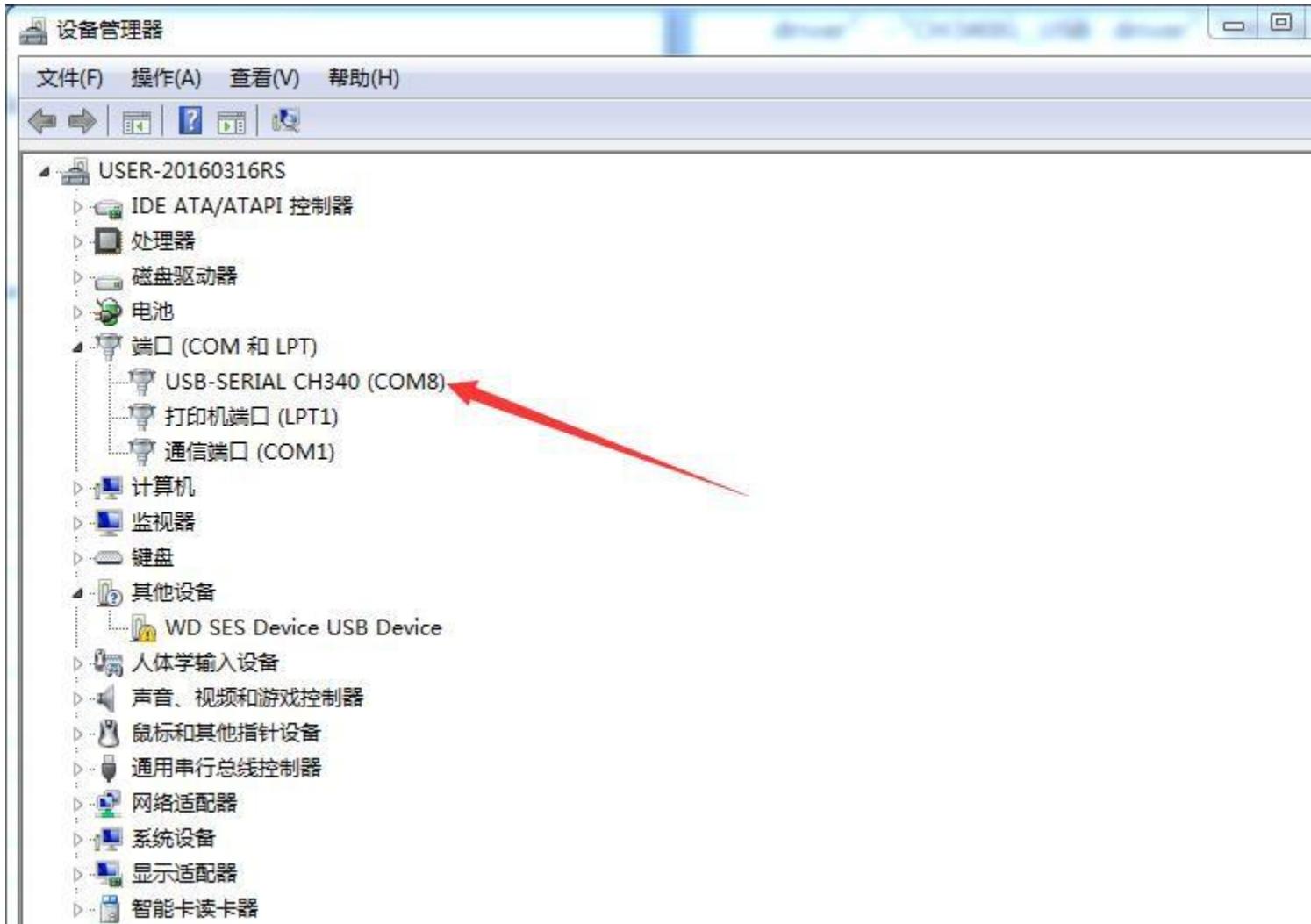
C:**RepetierHost**,it's also PC software,can control the printer,can slice the model, when printing,it's good to use.

2 Connect printer to PC

Install the driver. copy the file in the sd card to PC. open the file " CH340G_USB toTTL driver"-“CH340G_USB driver”-“DRVSETUP64”(64-bit system) or “SETUP”*(64-bit system).



Check Device Manager, will find the hardware



If the PC can not find the hardware.pls check out the file “Solve streamline Version Windows 7,Can not install d river issue”.do as the file show.

3 Repetier and control printer

1 Software Installation

The repetier is software to control the printer,so need install it first.

Install the “software,setupRepetierHost_1_0_6”, the software could be download on the link:<http://www.repetier.com/download/>

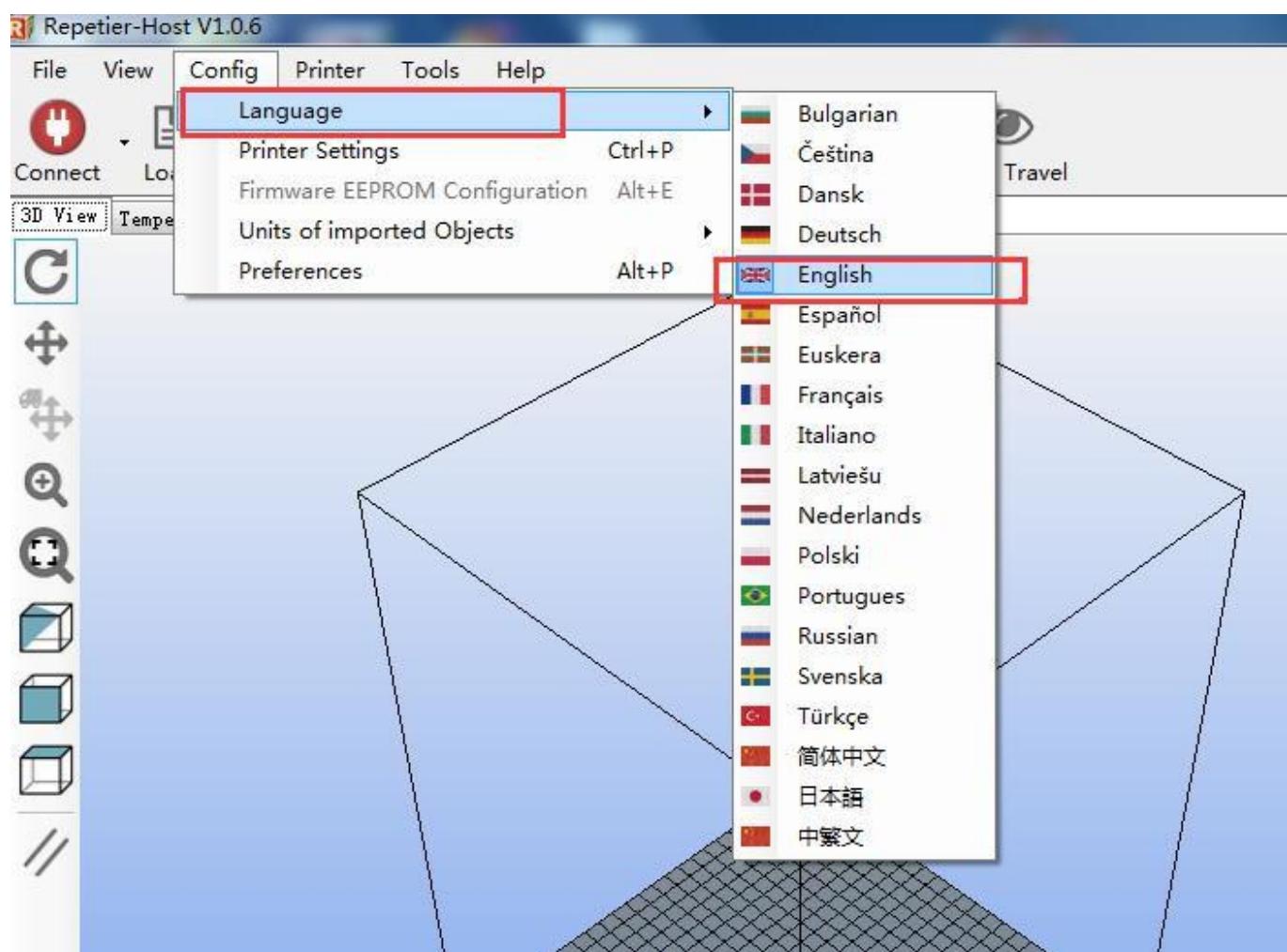
About the more info about the Repetier, pls go the the link :
<http://www.repetier.com/documentation/repetier-host/>

autolevel-i3-Marlin-1.1.2 -	2017\11\1
i3-Marlin-1.1.2 -	2017\11\1
arduino-1.8.1-macosx	2017\8\9
arduino-1.8.1-windows	2017\8\9
setupRepetierHost_1_0_6	2014\10\2

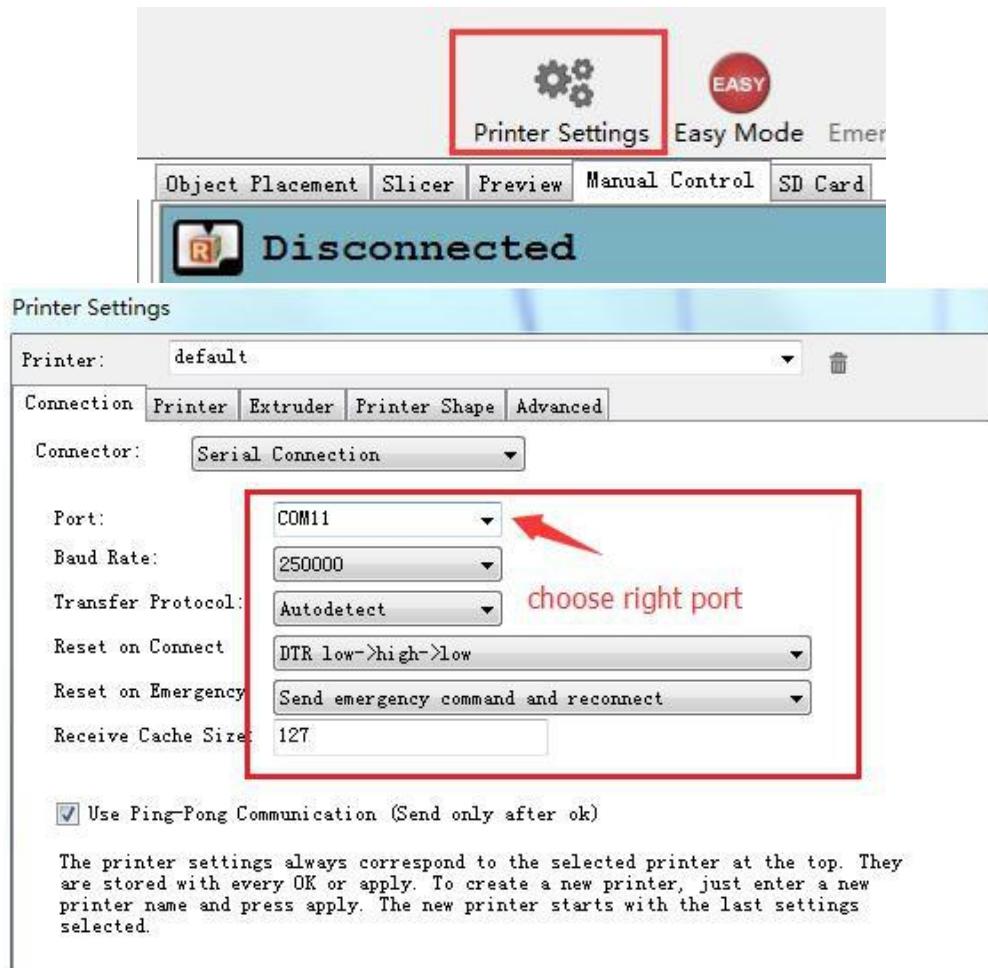
install the repetier

2 Language Settings

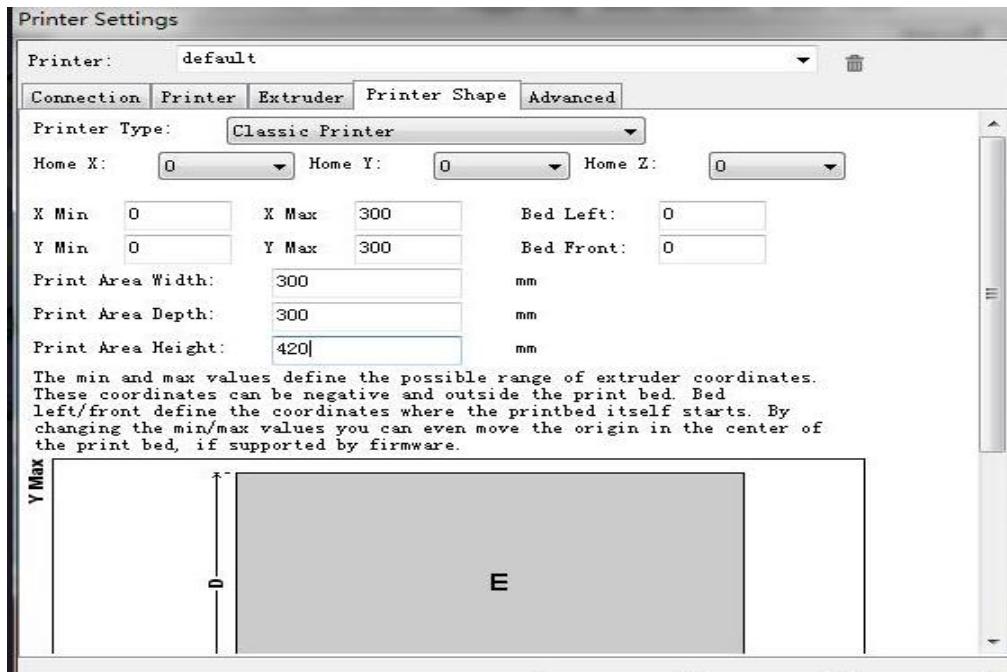
Set the language, Config-Language-choose the suit language.



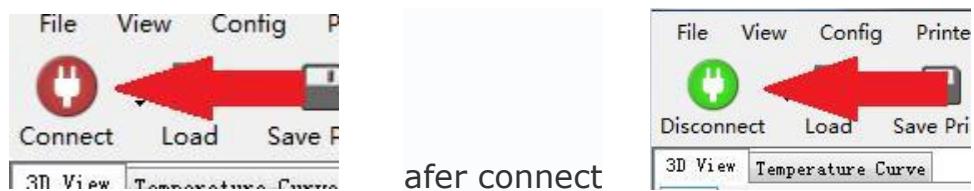
3 Printer settings



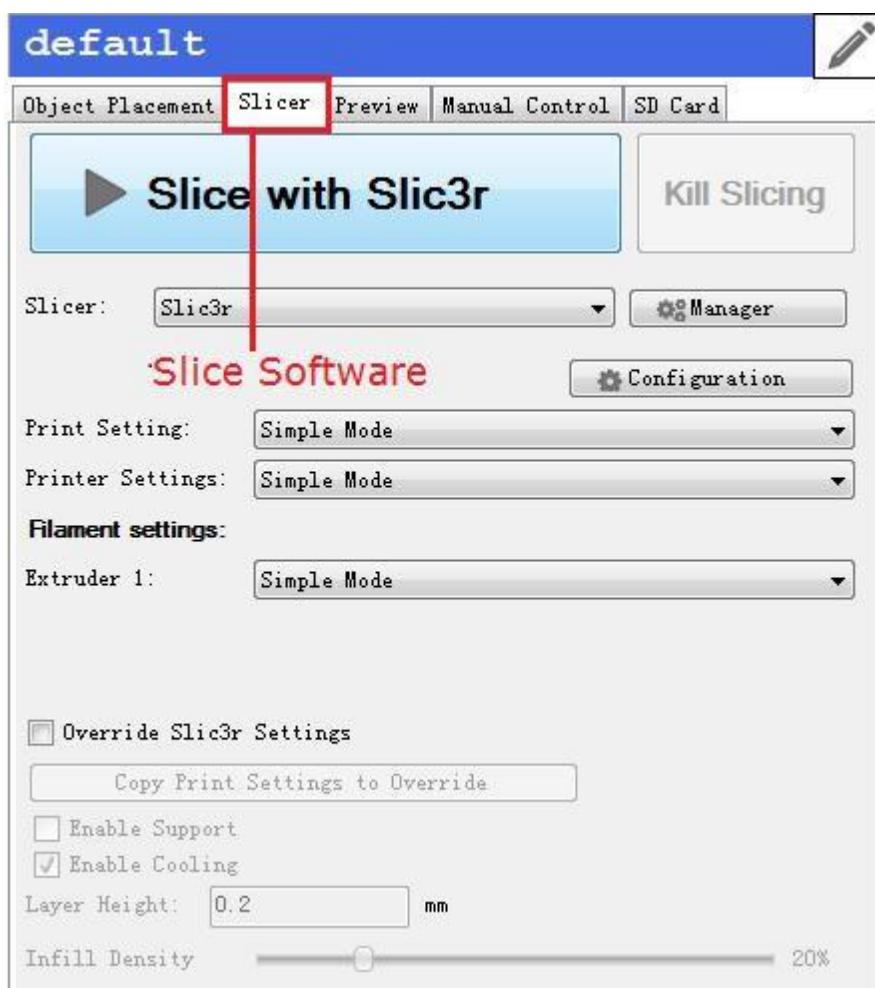
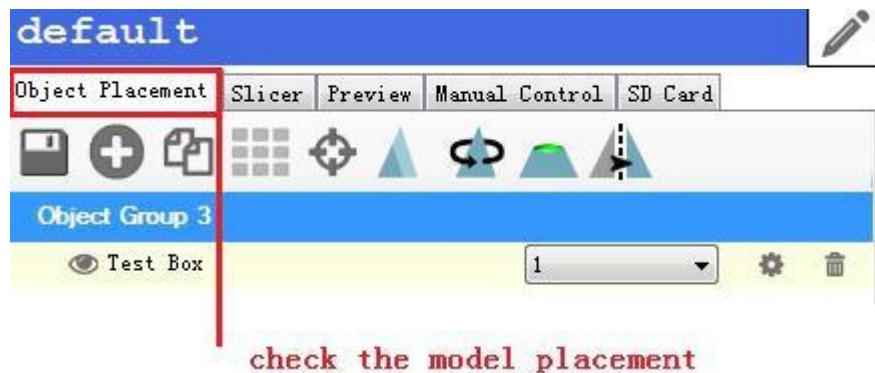
IN the printer shape,choose the Classic printer,the home X is 0,the home Y is 0,the home Z is 0.the setting as the follow pictue.

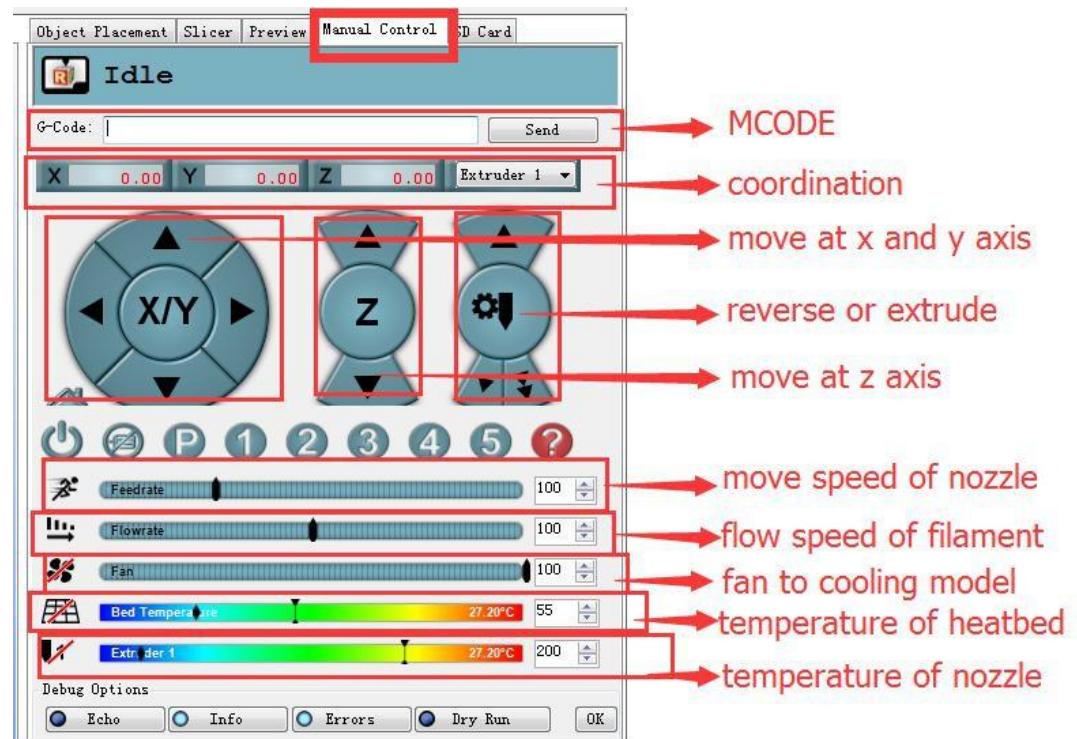
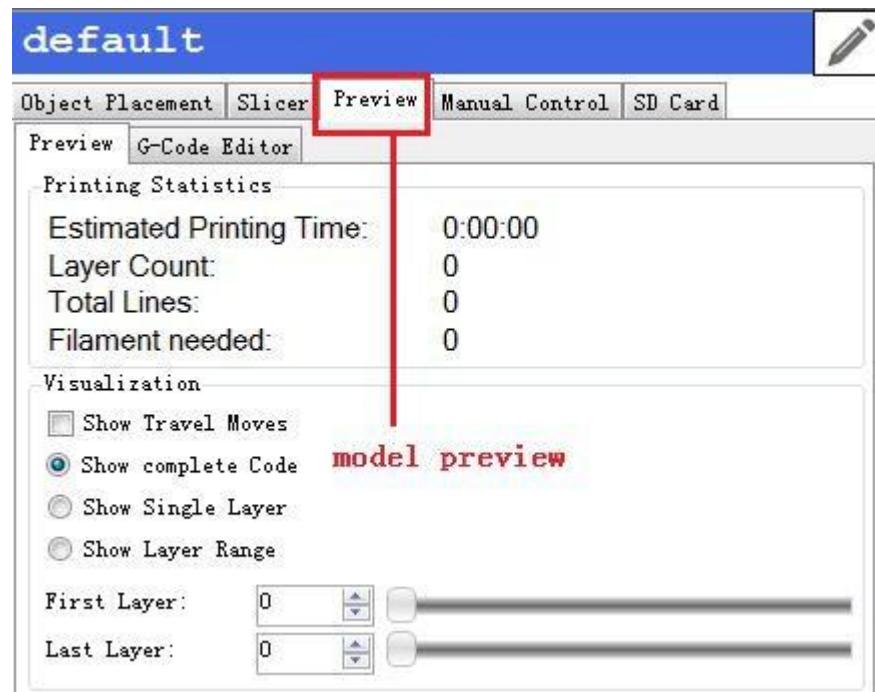


4 Connect the printer



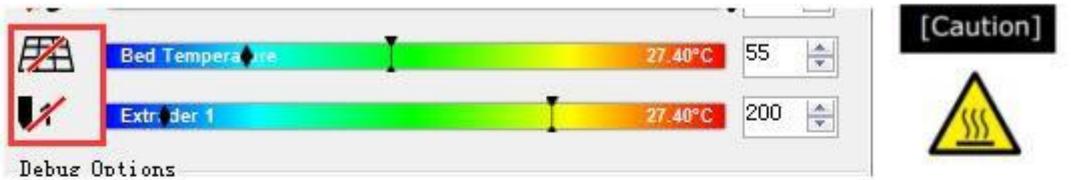
5 Panel Introduction



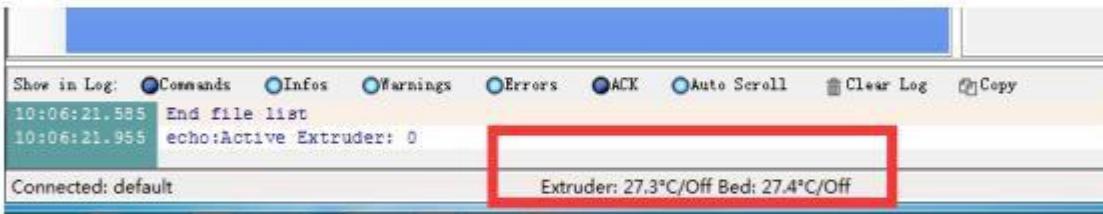


home function same to G28

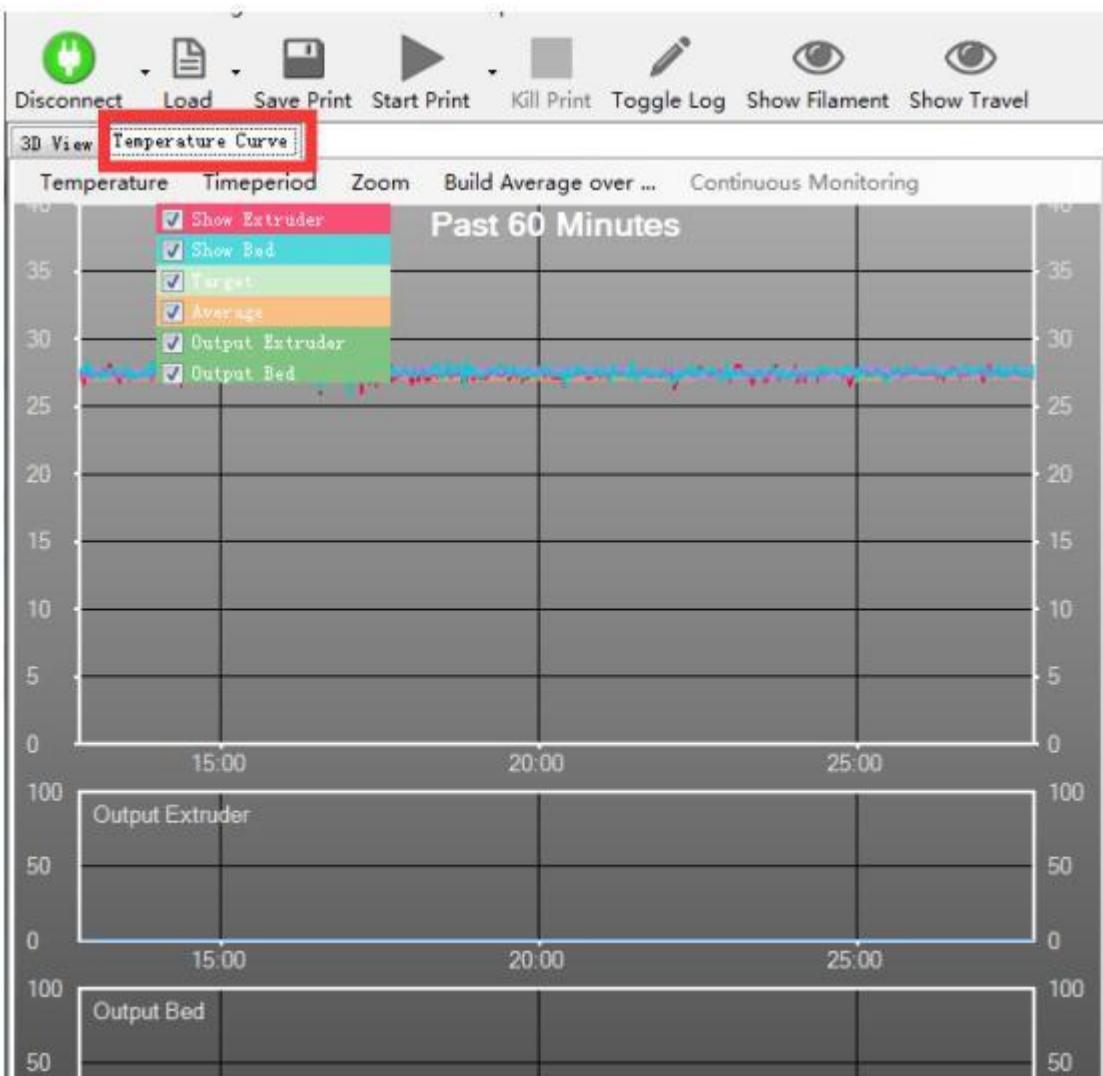
>> Preheat hotend and bed



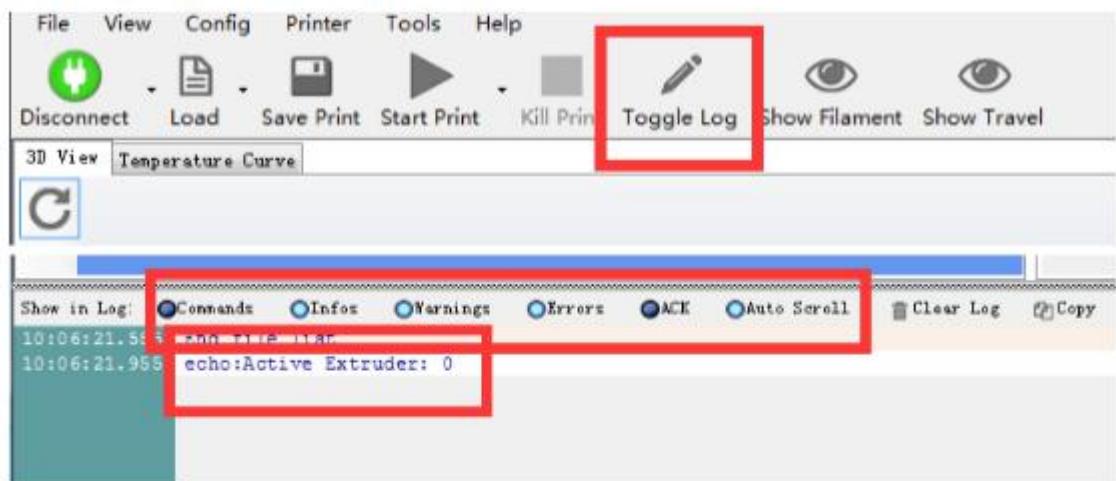
Check from there (bottom of repetier)



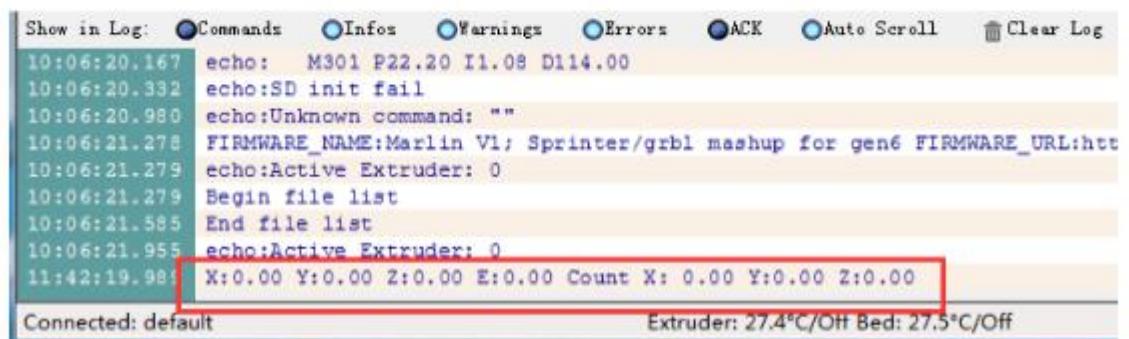
Temperature curve



>>Info of operation, Check at the bottom



>> Info of coordination ,send m114 then check at the bottom

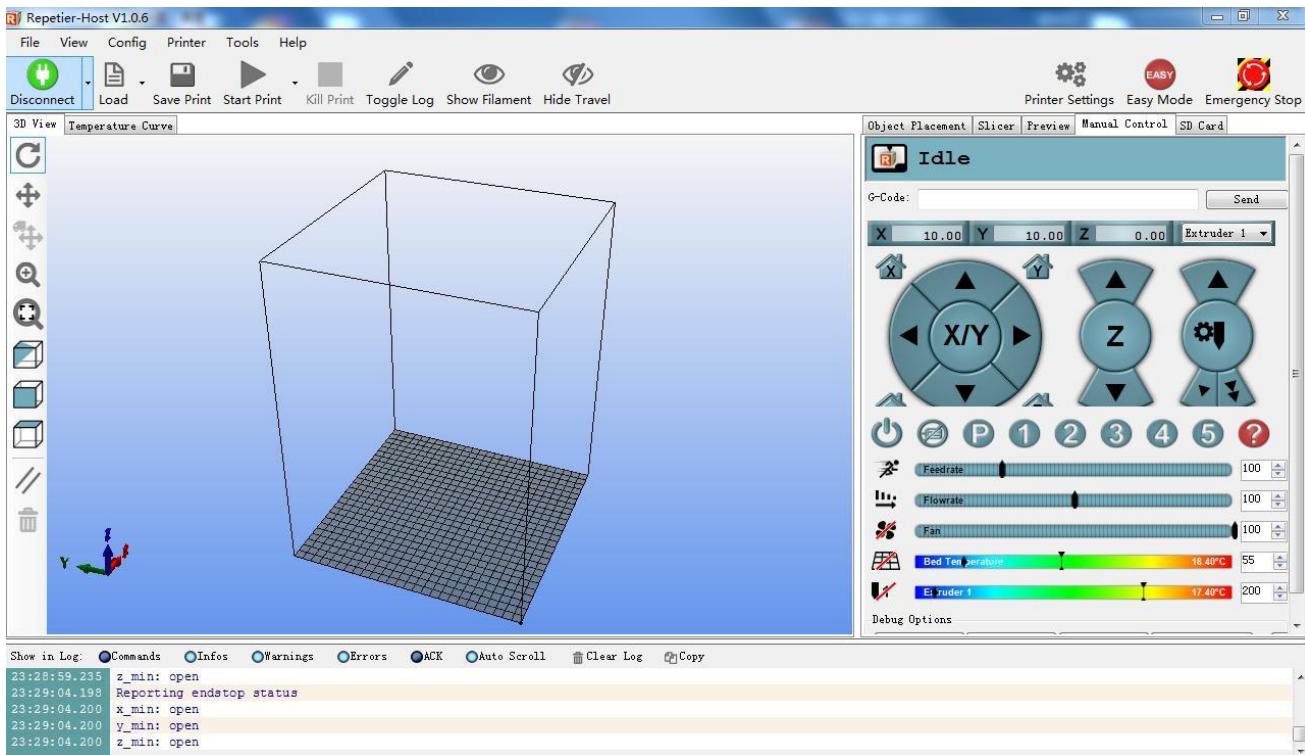


>>Info of limit switch, send m119 then check at the bottom

```
09:34:54.446 Reporting endstop status
09:34:54.448 x_min: open
09:34:54.448 y_min: open
09:34:54.448 z_min: open
09:34:54.448 z_max: closed
```

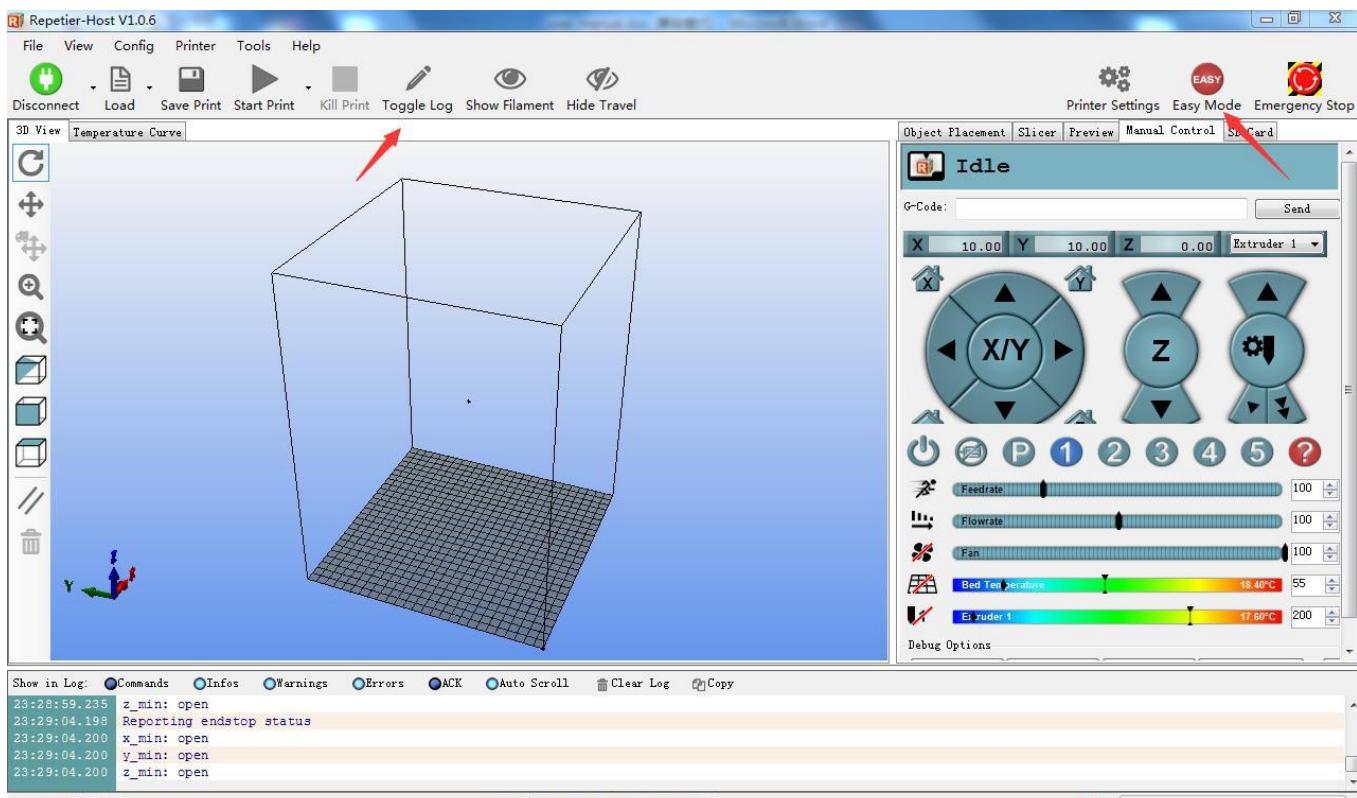
6 some important command

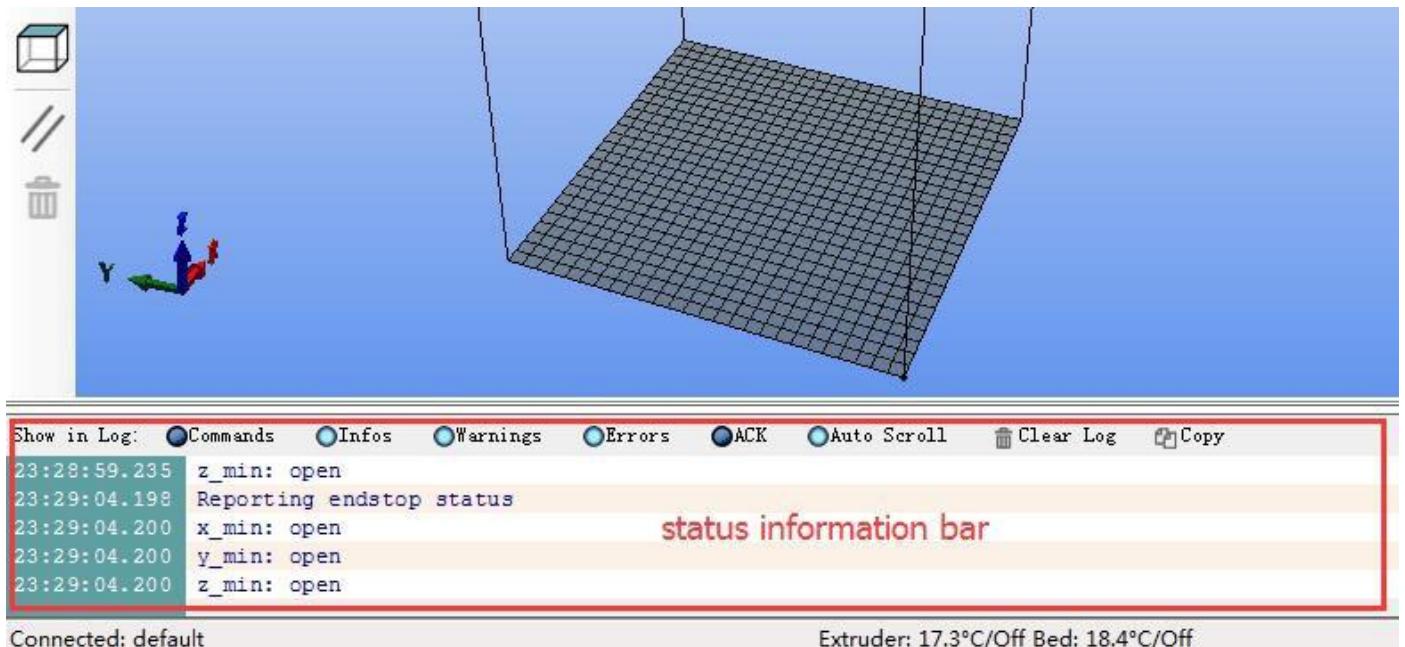
Controling the printer by repetier, after opening repetier, choose right port and click connect. The repetier interface is like this



open “command bar” where we can input command. **Click Easy Mode open command bar**, can input command in command bar.

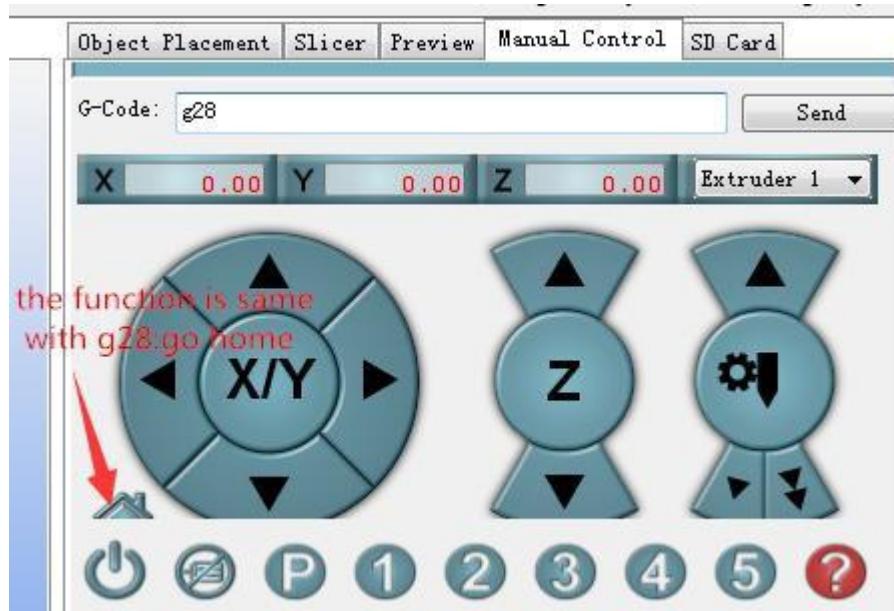
Then **click Toggle Log**, The information bar appears below the repetier interface. can read info about printer in information bar.





(1) g28----all axis go home

In the repetier letters are case-insensitive,G28 also can be used.but in simplify3D,only can use capital letters .The other command is the same



Send g28,in delta 3D printer all axis rise and hit limit switch ,G28 is same to this icon



(2) m114----check the current coordinates of the nozzle

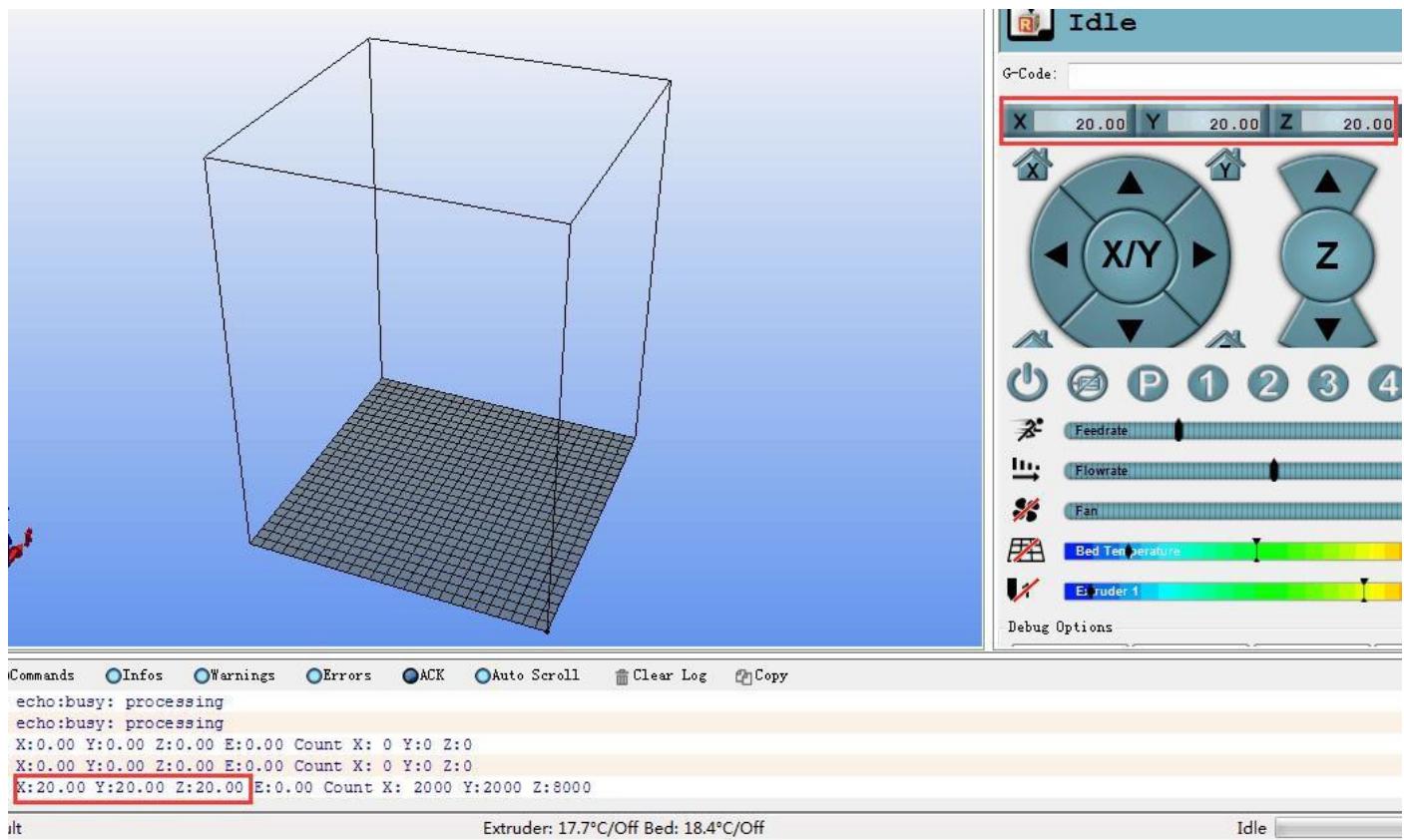
input m114 and send ,can see the current coordinates of the nozzle in information bar.

(3) g1 x_y_z----move nozzle to targeted coordinate

For example ,input g1 x20 y20 z20,the nozzle will go to coordinate x y z(20 20 20)



After we send this command ,we can view coordinate of nozzle now.



It's ok to move only one or two axis

Eg:

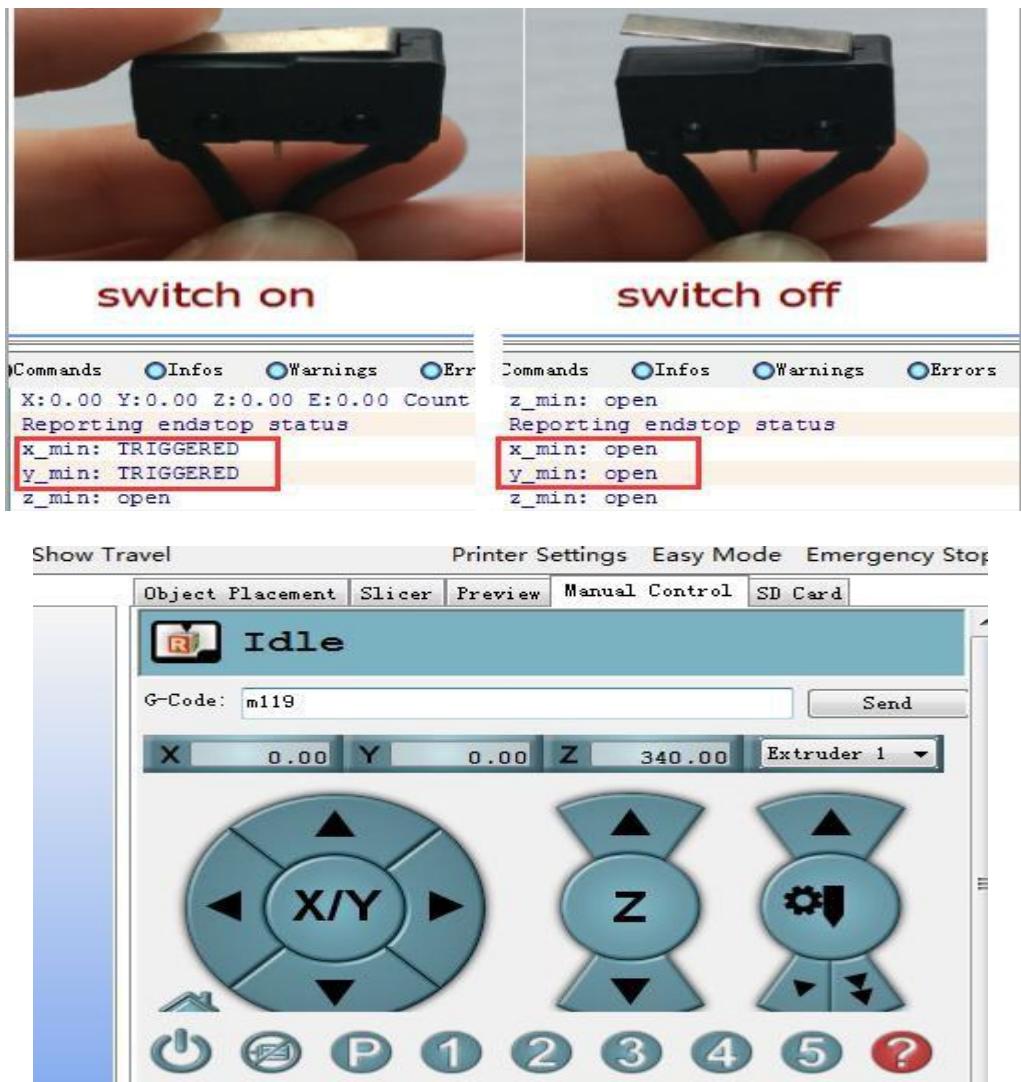
g1 z0—x and y keep previous coordinate,z coordinate arrive to x 0,this arrive the lowest bottom;
g1 x20—y and z keep previous coordinate,x coordinate will arrive to x 20;

g1 x20 y10—z keeps previous coordinate,x coordinate arrive to x 20;y coordinate arrive to y 10; g1 y15 z30—x keeps previous coordinate,y coordinate arrive to y15; z coordinate arrive to y 30.

(4) M119----check the status of the switch.

The switch status is important directive,when meeting the motor moving problem,can check the switch status to search problem.

When hit the switch,it shows TRIGGERED;if not touch, it shows open.

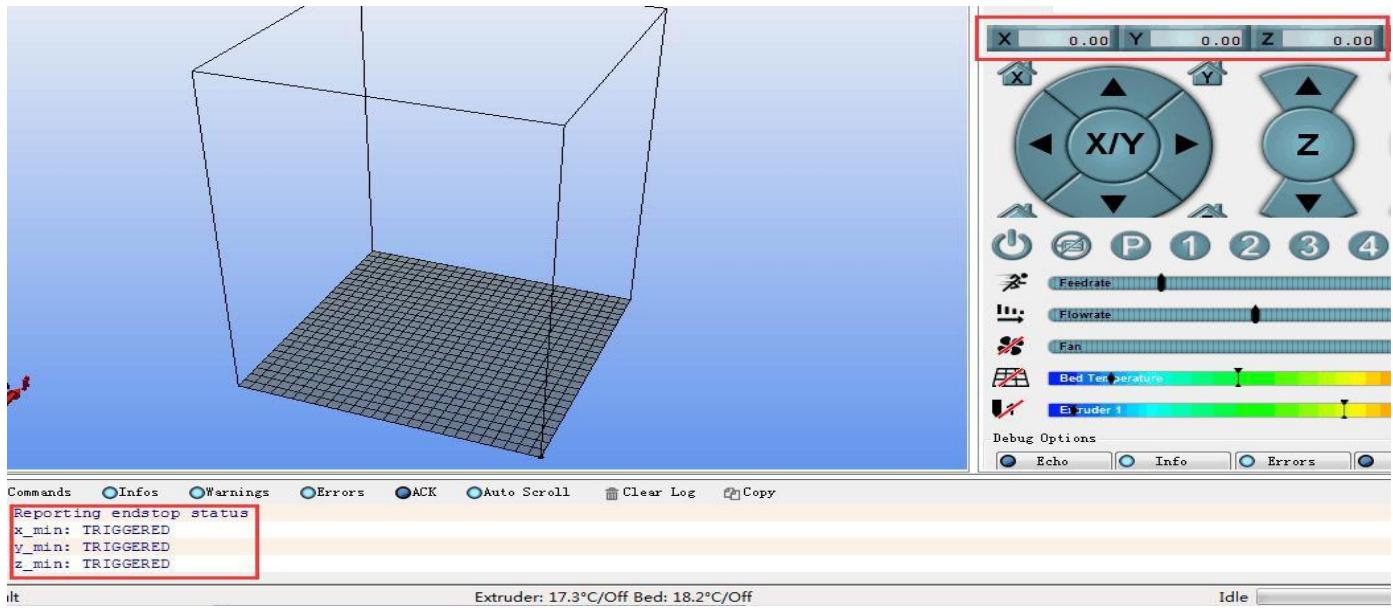


When send g28, the nozzle will go home, then send m119 It show:

x_min TRIGGERED

y_min TRIGGERED

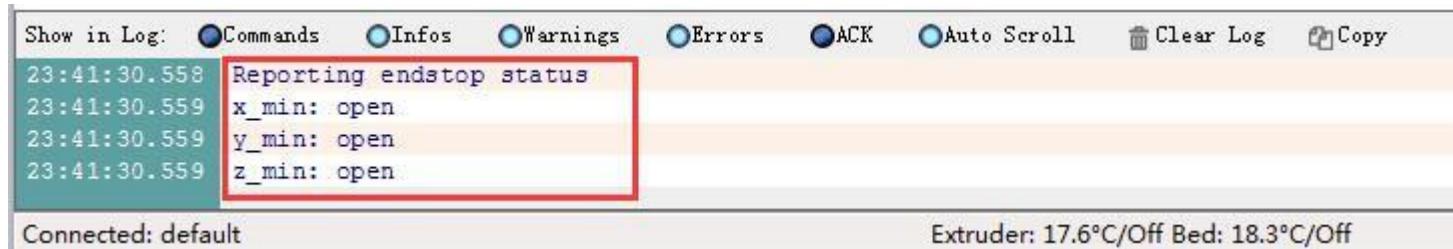
z_min TRIGGERED



If the nozzle is at the middle air, not hit any limit switch, all four switch show open.

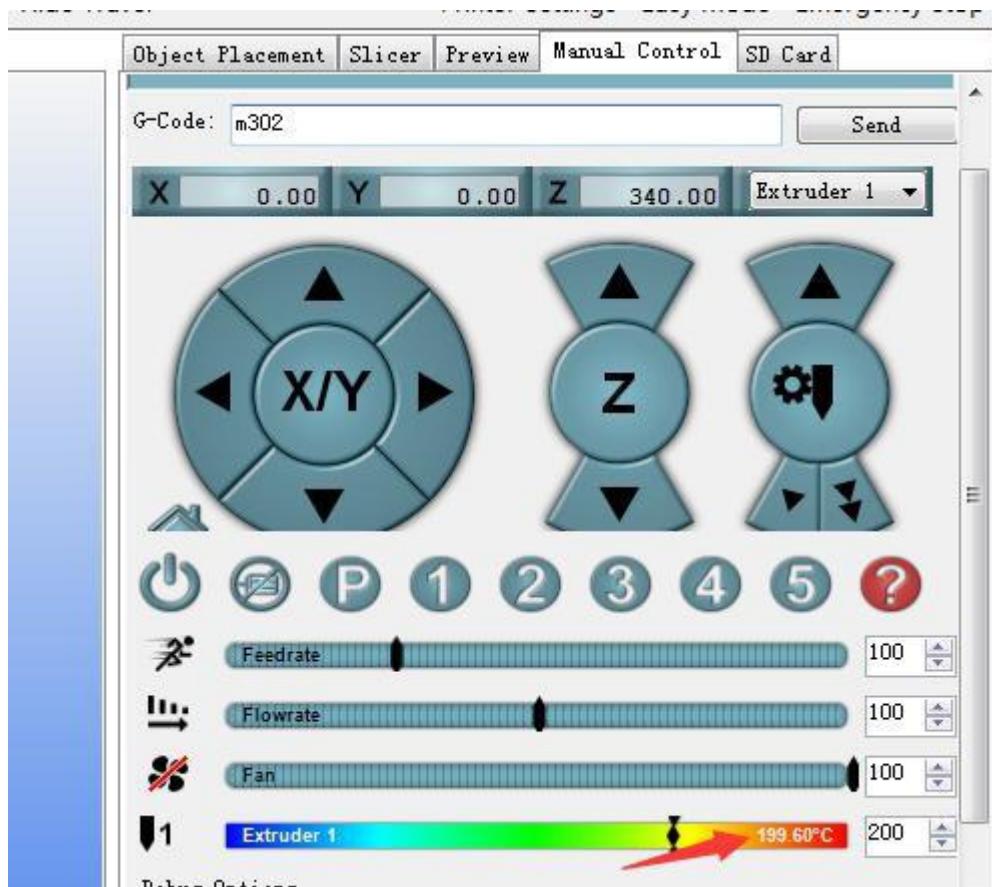
It show:

x_min open
y_min open
z_min open

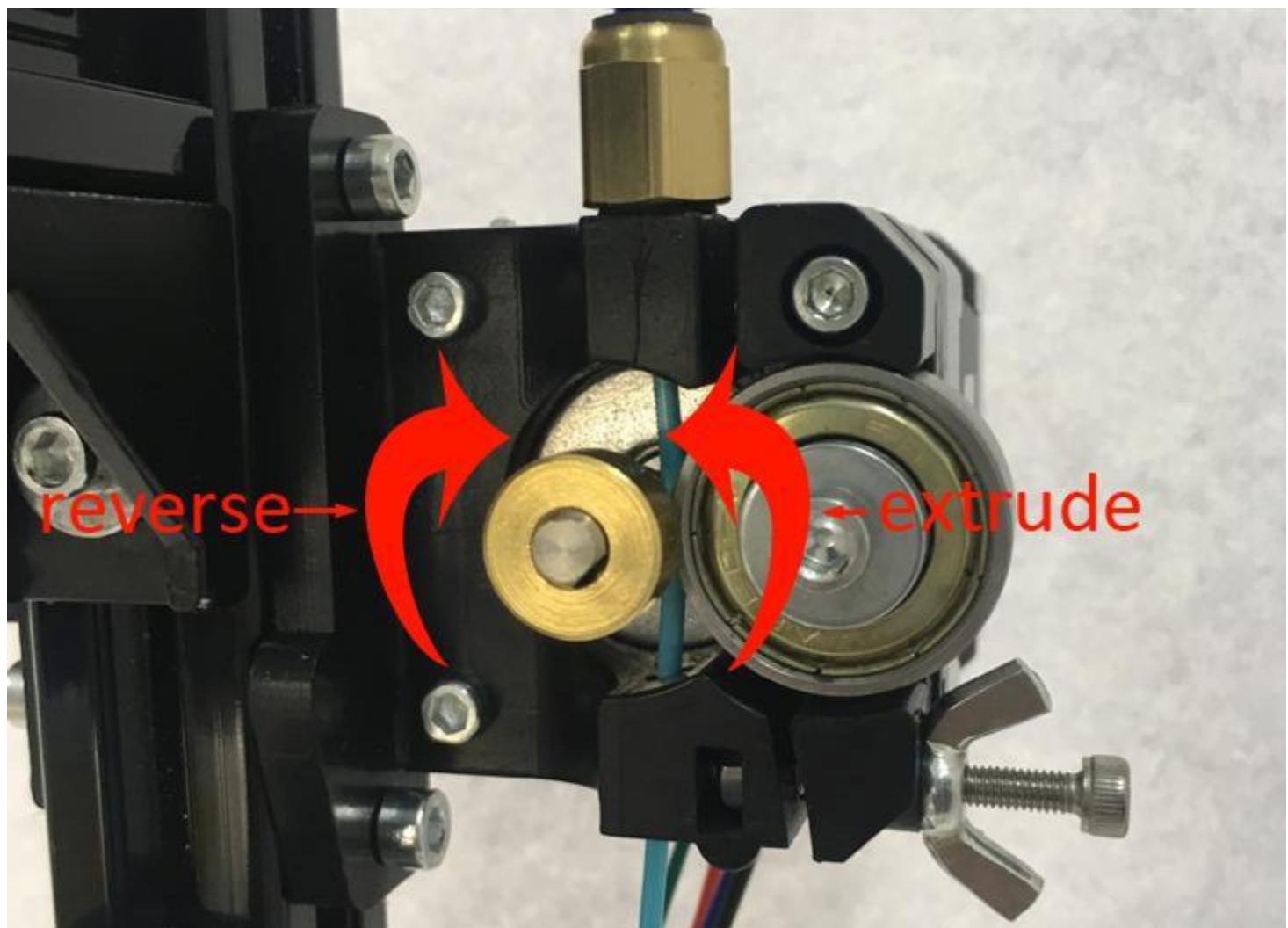
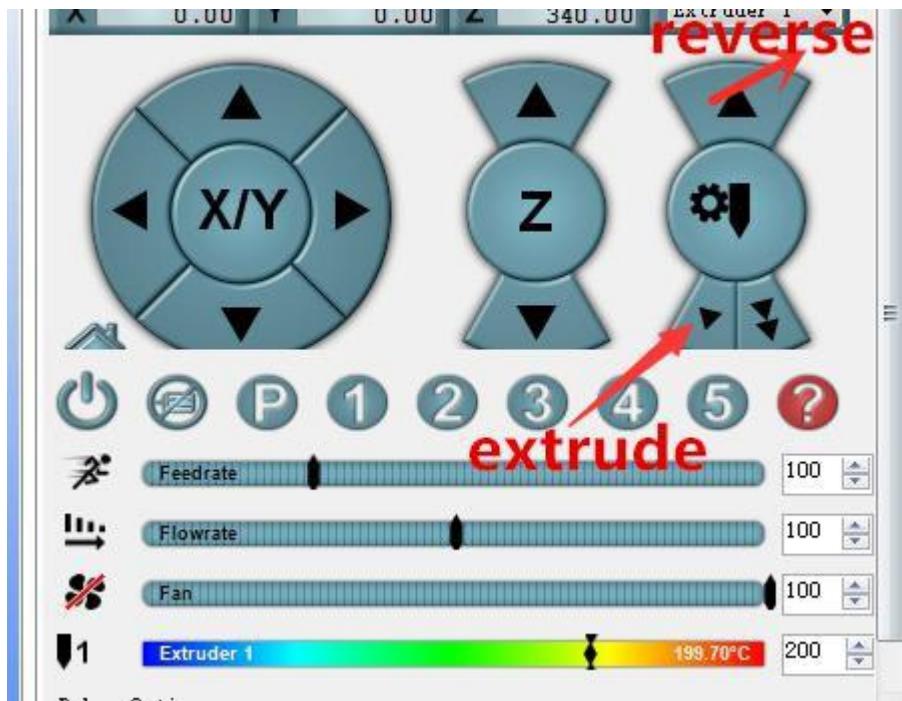


(5) m302----manual control extruder motor

Before manual control extruder motor , need heat nozzle to targeted temperature and can melt filament. this step is to prevent the extruder run at cold status, then input 302.

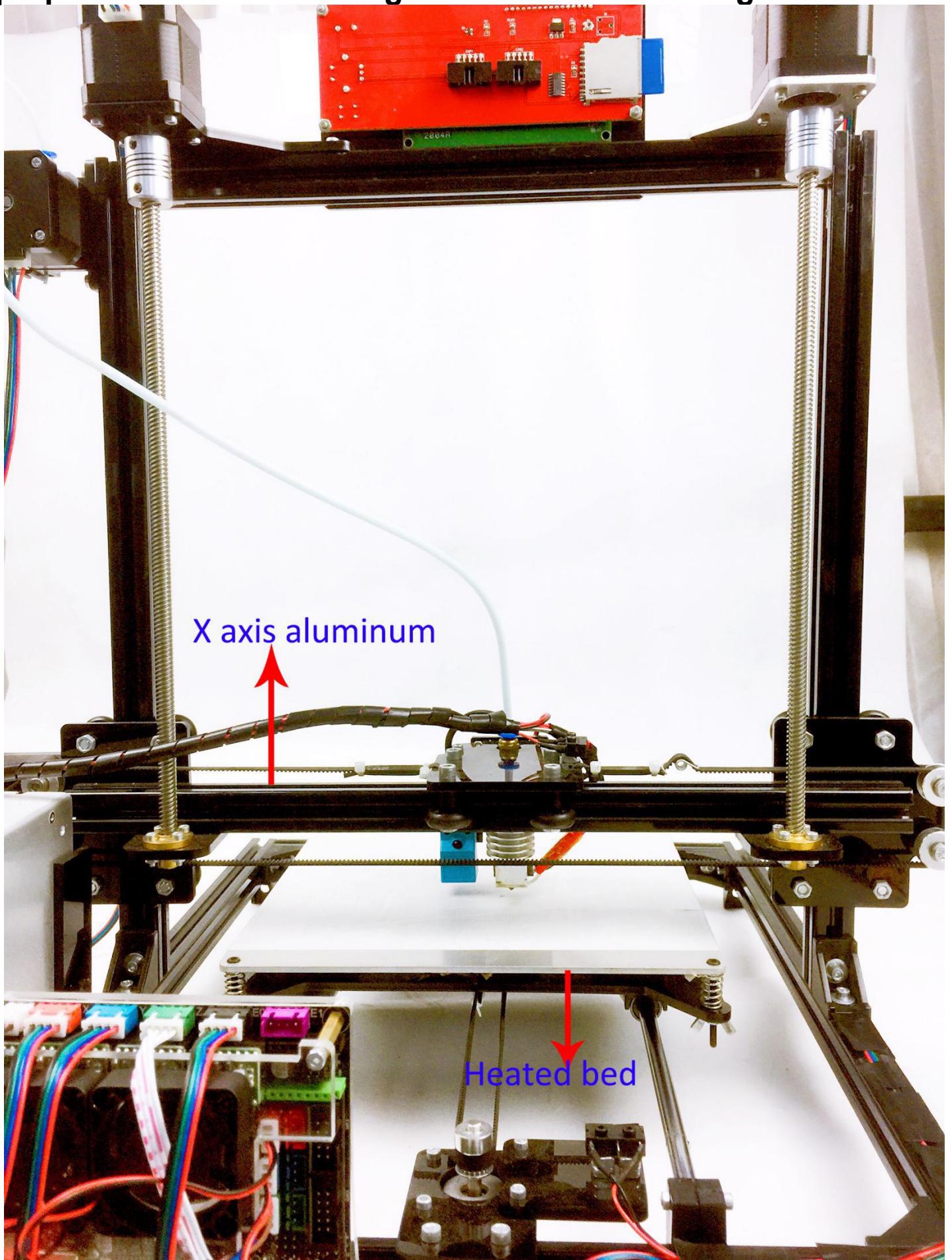


Important note: if use M302 code, must guarantee the nozzle temp reaach 175°C.
If can not reach 175°C, the extruder can not run. !!!



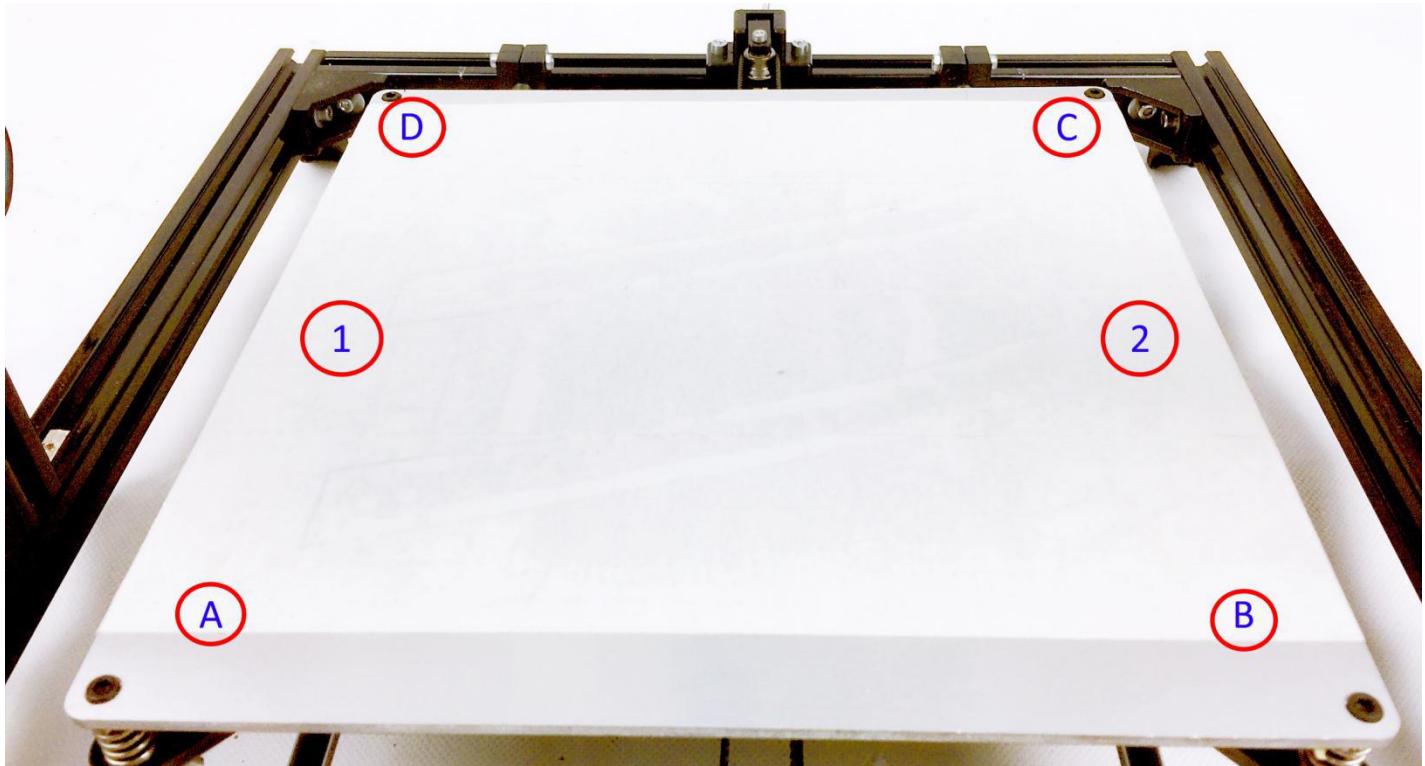
When heating the nozzle ,first input m302 and send ,and then click extrude,we can see extrude motor run ,and filament come out from nozzle(if you have installed filament)

2 preparation before leveling and automatic leveling



Before automatic leveling, the heated bed and the X axis aluminum should be leveling

We can through ①② two points, adjust the aluminum , through the A B C D four points, adjust the hot bed



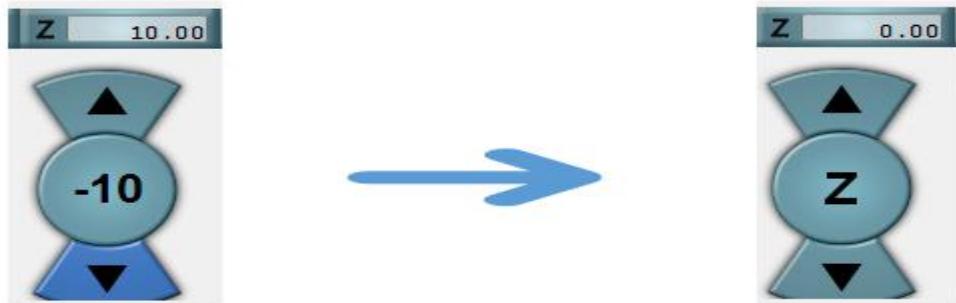
First enter the G28 command, so that the machine is in the middle position.

Second in the command bar enter: g1 z10 (The purpose of doing so is to prevent the scratches when the nozzle goes down to the next coordinate)

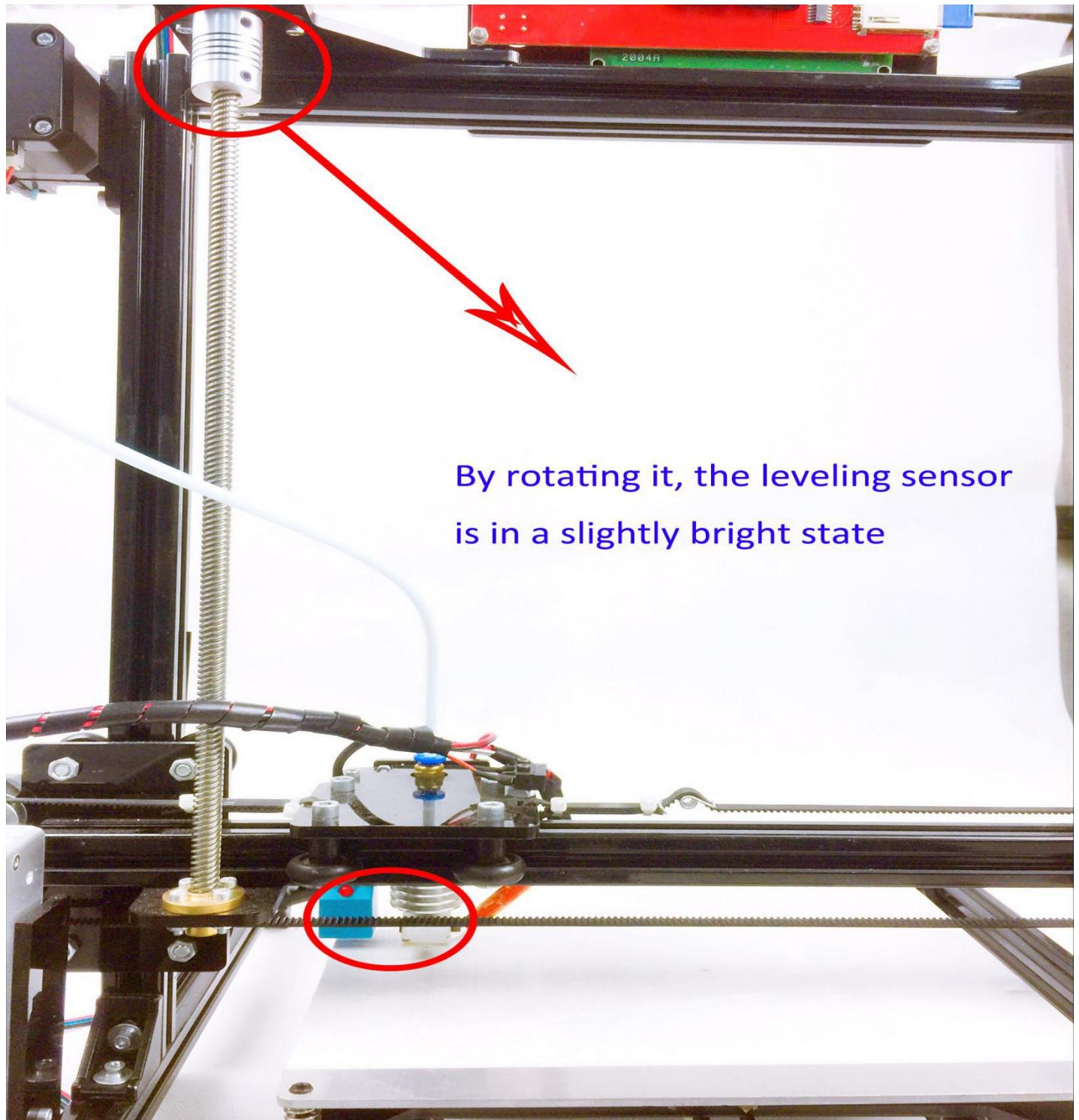


Then enter: g1 x180 y90 ,The nozzle will go to ① coordinate

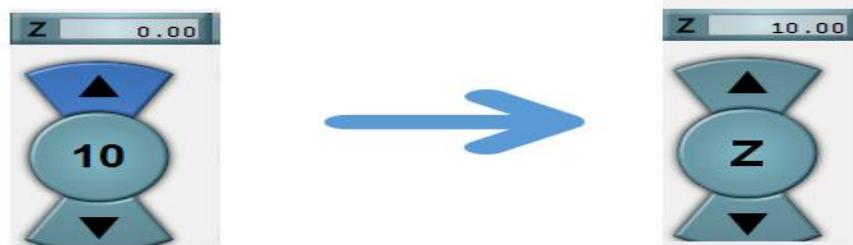
Then



Move the Z axis to zero



By rotating it, the leveling sensor
is in a slightly bright state

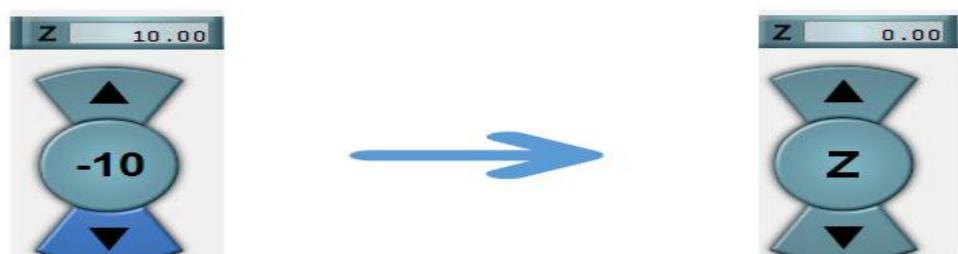


Move the Z axis to 10

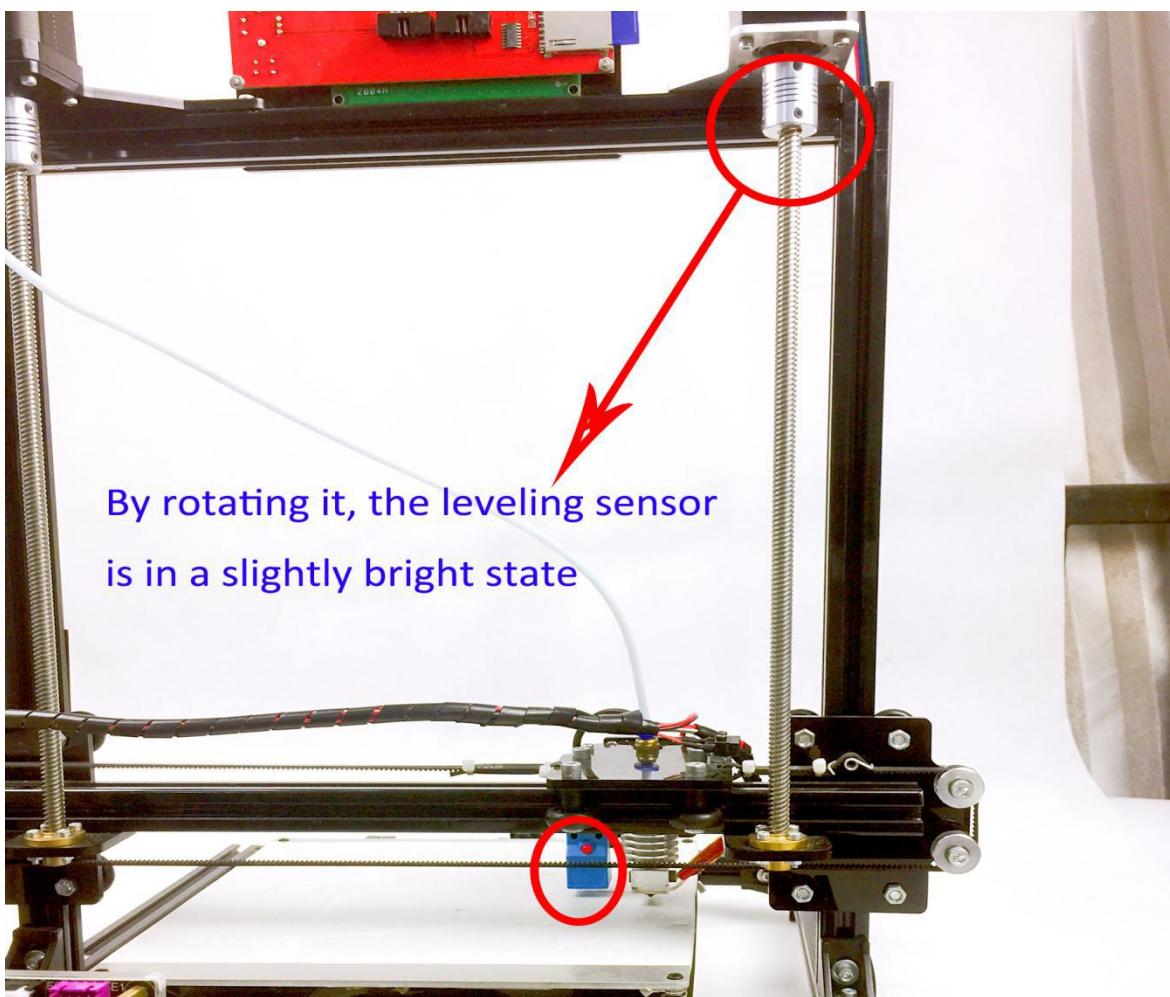
(The purpose of doing so is to prevent the scratches when the nozzle goes down to the next coordinate)

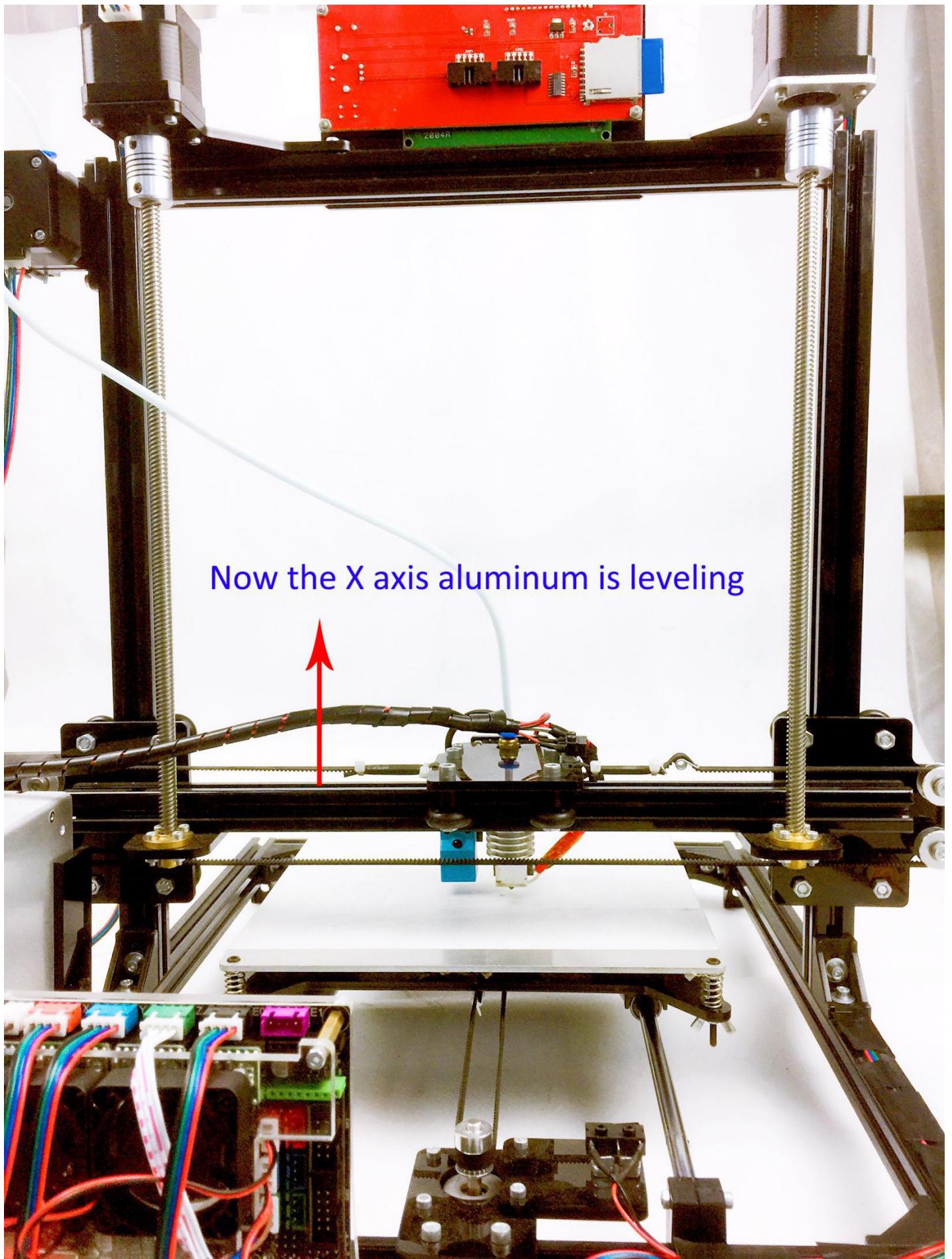
Then enter: g1 x10 y90 ,The nozzle will go to ② coordinate

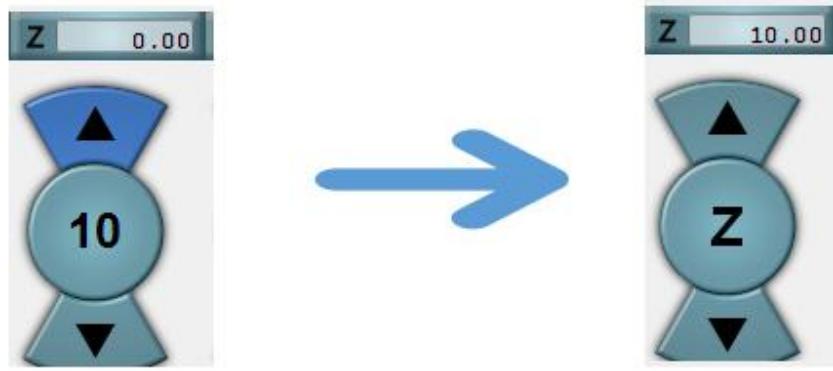
Then



Move the Z axis to zero







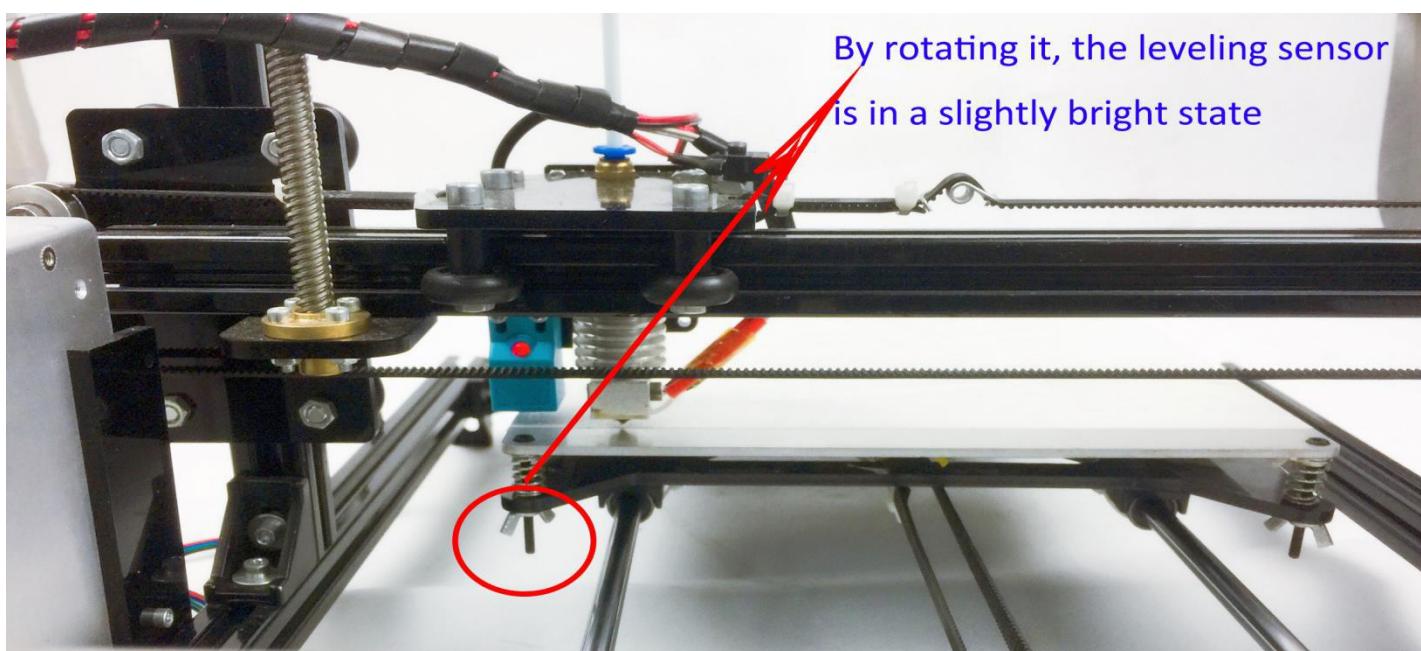
Move the Z axis to 10

(The purpose of doing so is to prevent the scratches when the nozzle goes down to the next coordinate)

Then enter: g1 x180 y180 ,The nozzle will go to **A** coordinate



Move the Z axis to zero

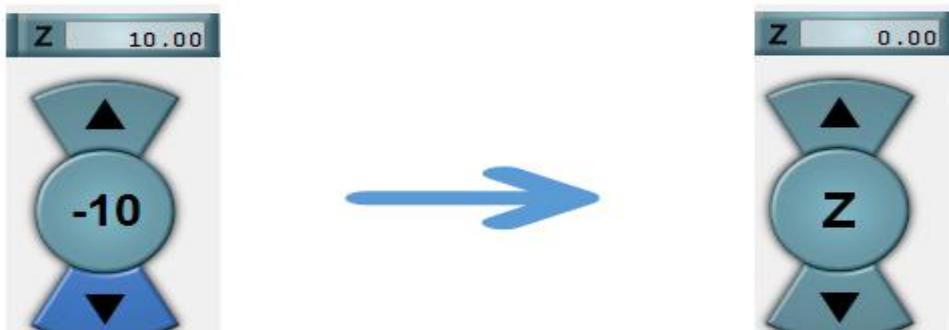




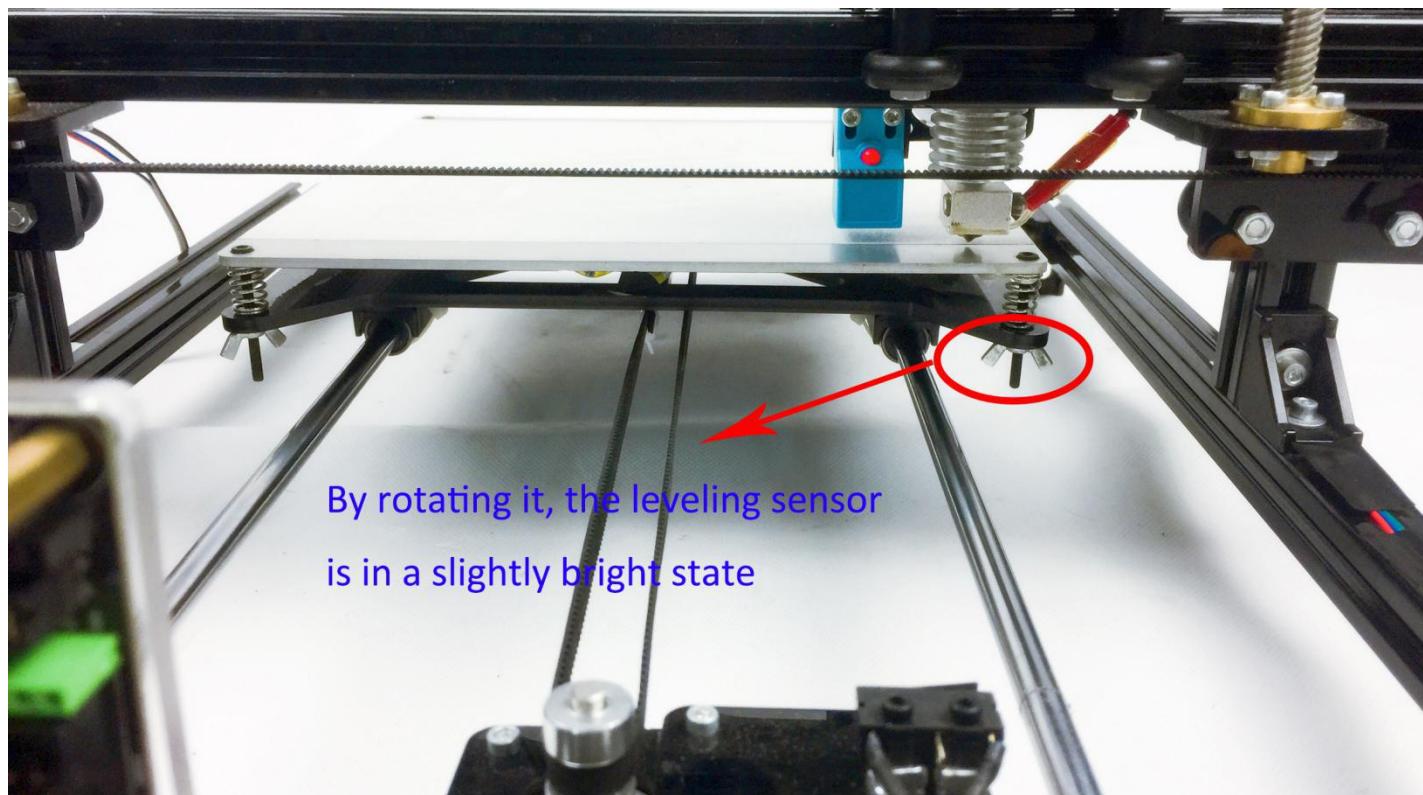
Move the Z axis to 10

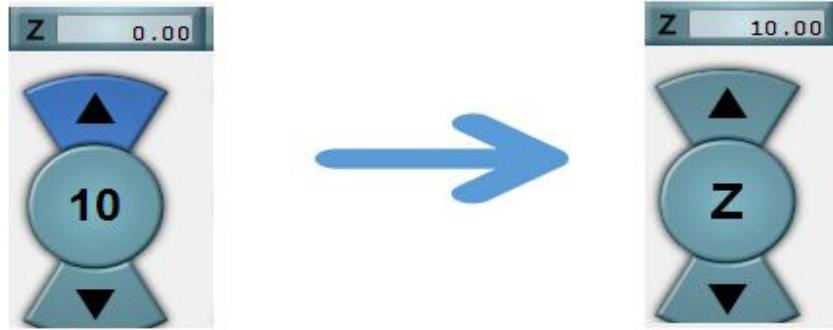
(The purpose of doing so is to prevent the scratches when the nozzle goes down to the next coordinate)

Then enter: g1 x10 y180 ,The nozzle will go to **B** coordinate



Move the Z axis to zero





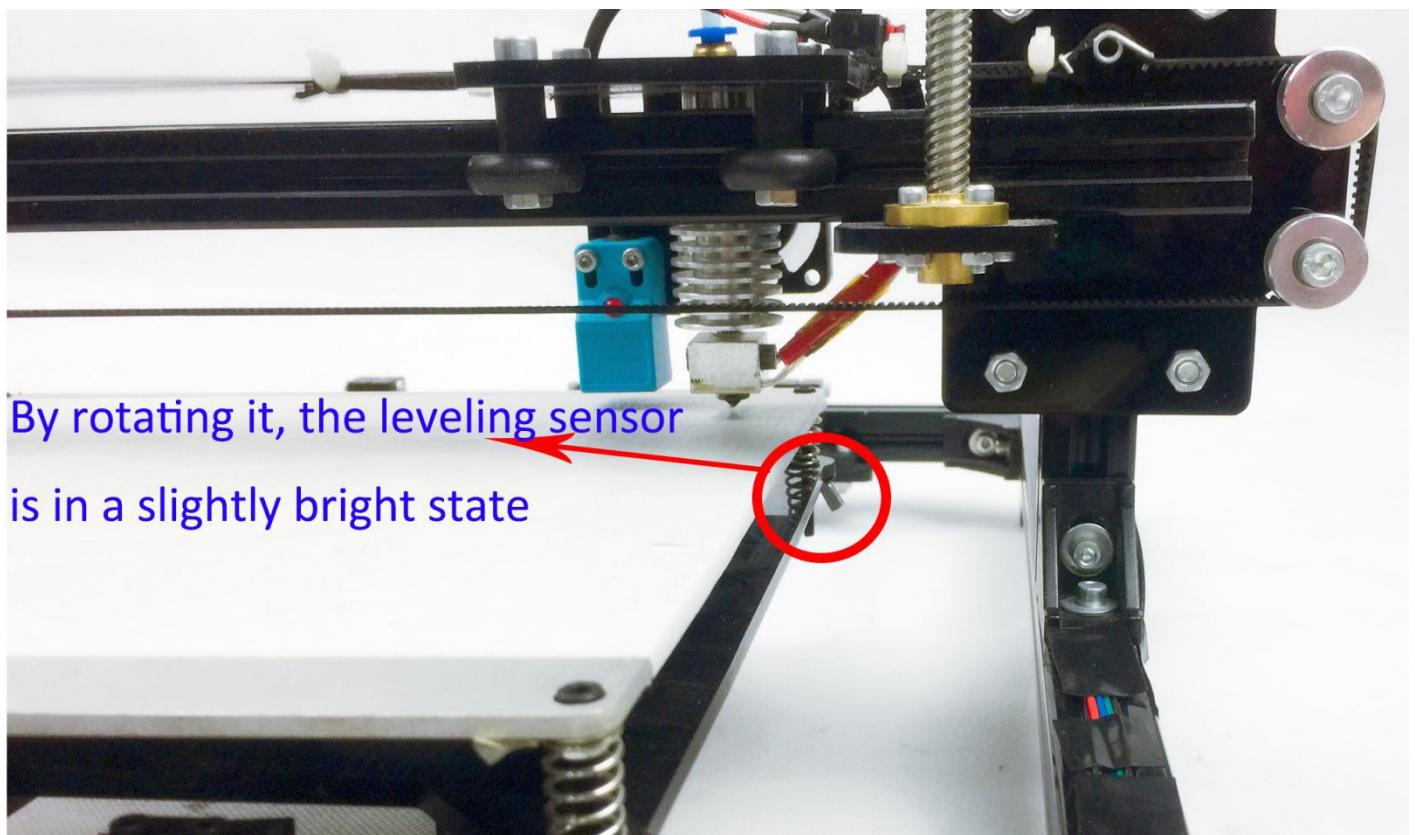
Move the Z axis to 10

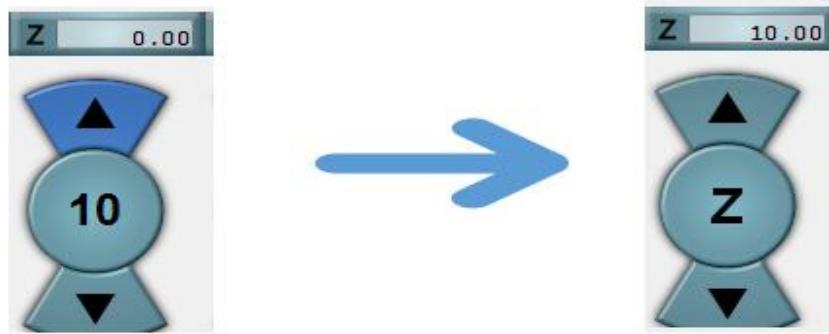
(The purpose of doing so is to prevent the scratches when the nozzle goes down to the next coordinate)

Then enter: g1 x10 y10 ,The nozzle will go to C coordinate



Move the Z axis to zero

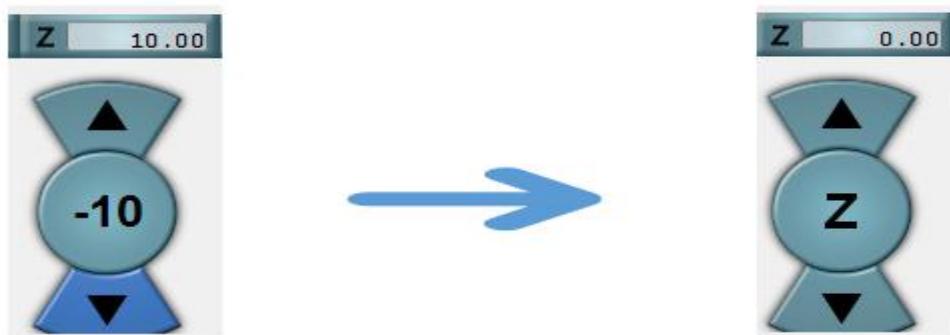




Move the Z axis to 10

(The purpose of doing so is to prevent the scratches when the nozzle goes down to the next coordinate)

Then enter: g1 x180 y10 ,The nozzle will go to **D** coordinate



Move the Z axis to zero

Repeat two to three times to ensure that the results are more accurate.

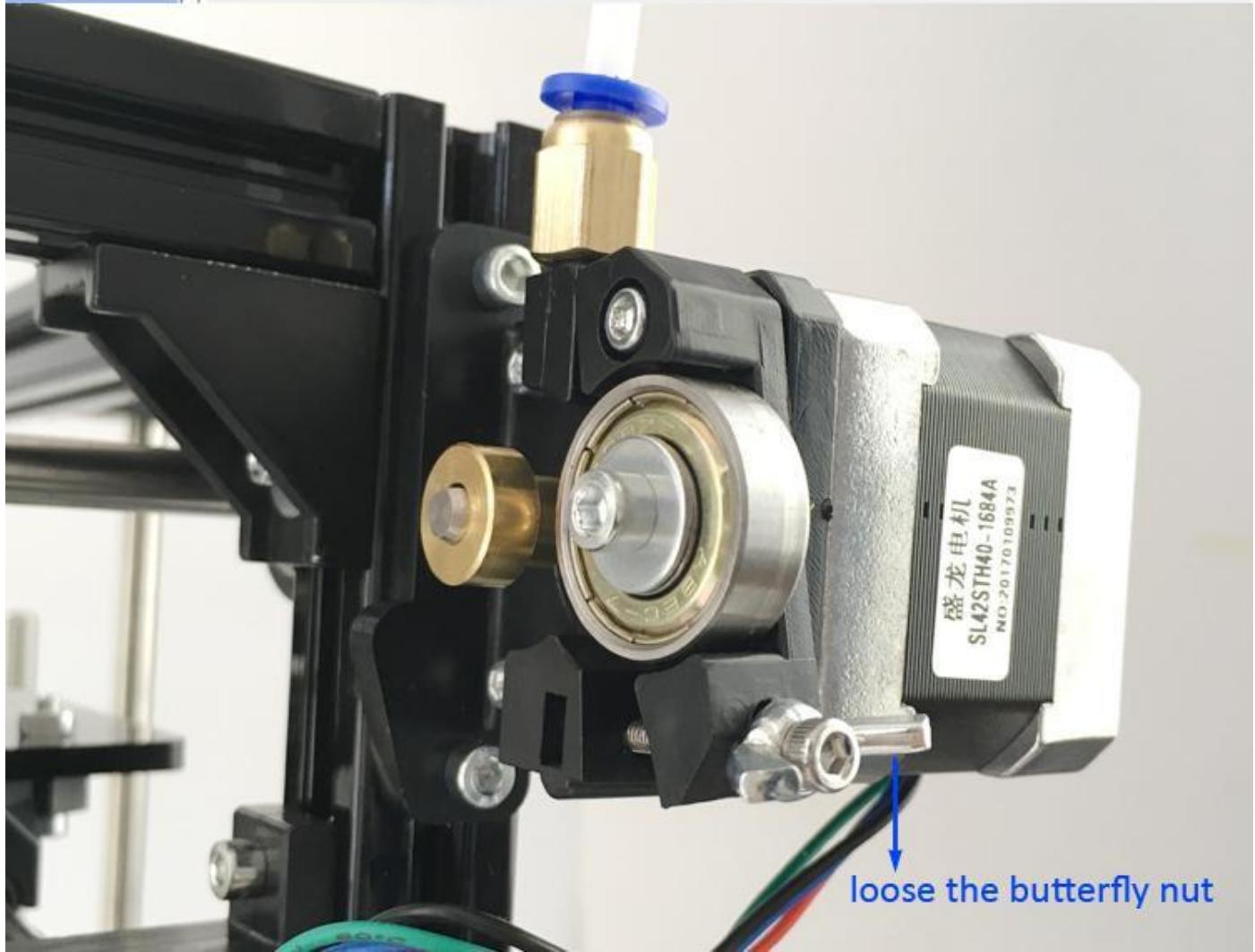
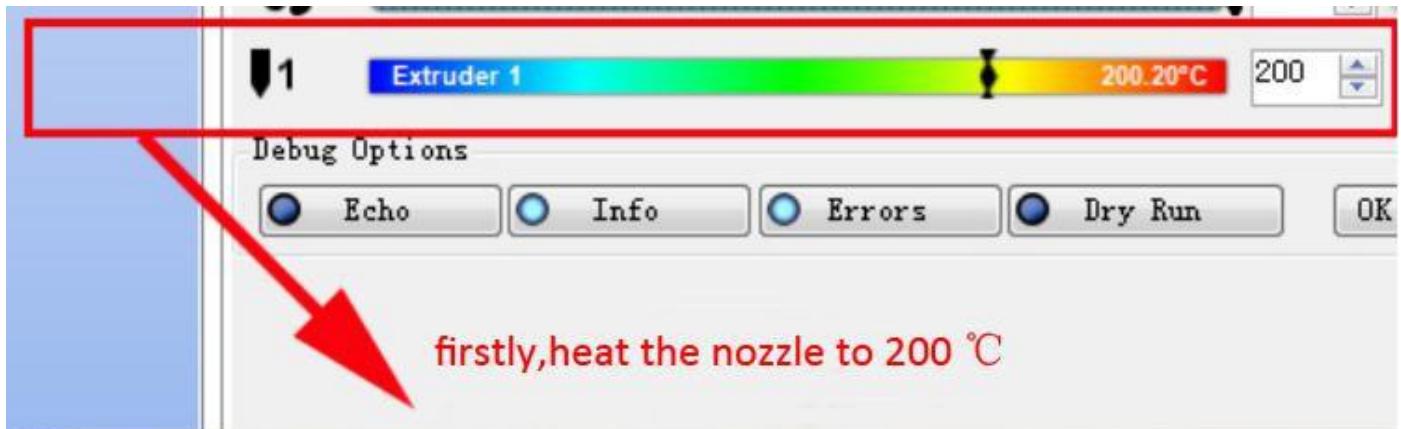
Now the heated bed is leveling ,preparation before leveling is complete.

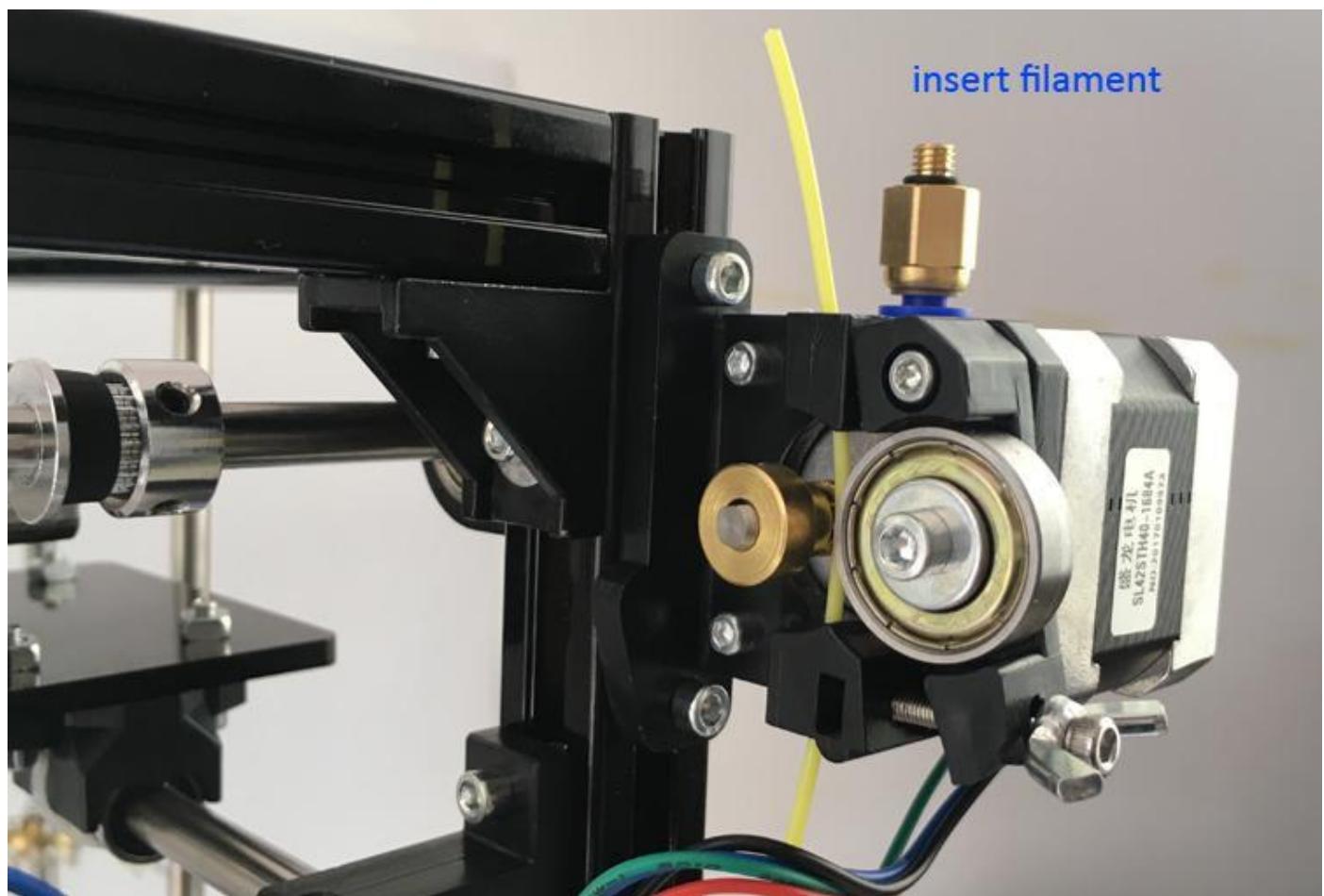
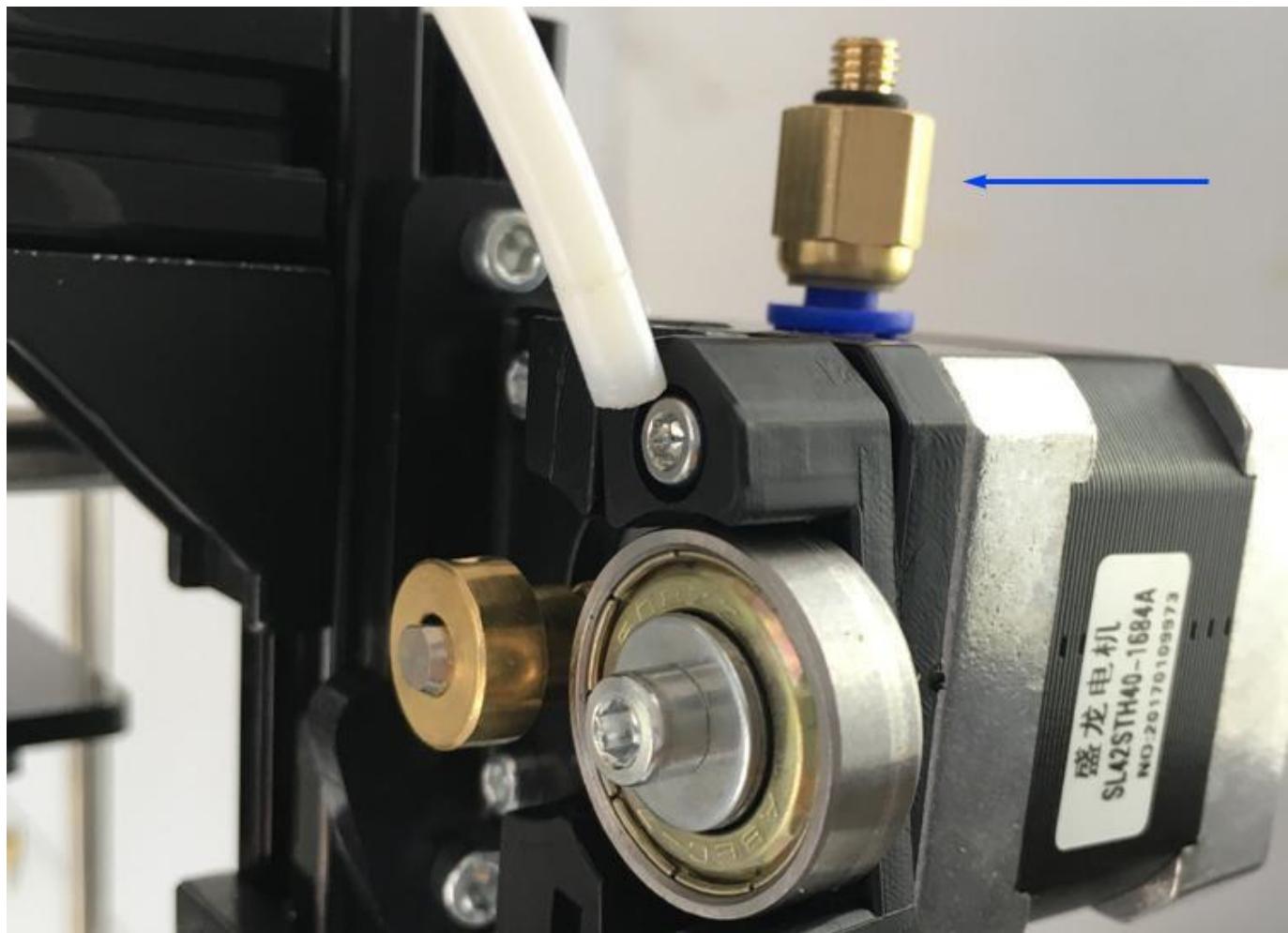
G29 is autolevel command,after you load model and slice it ,the regular succession is g28 go home,g29 autolevel ,then you can click start print

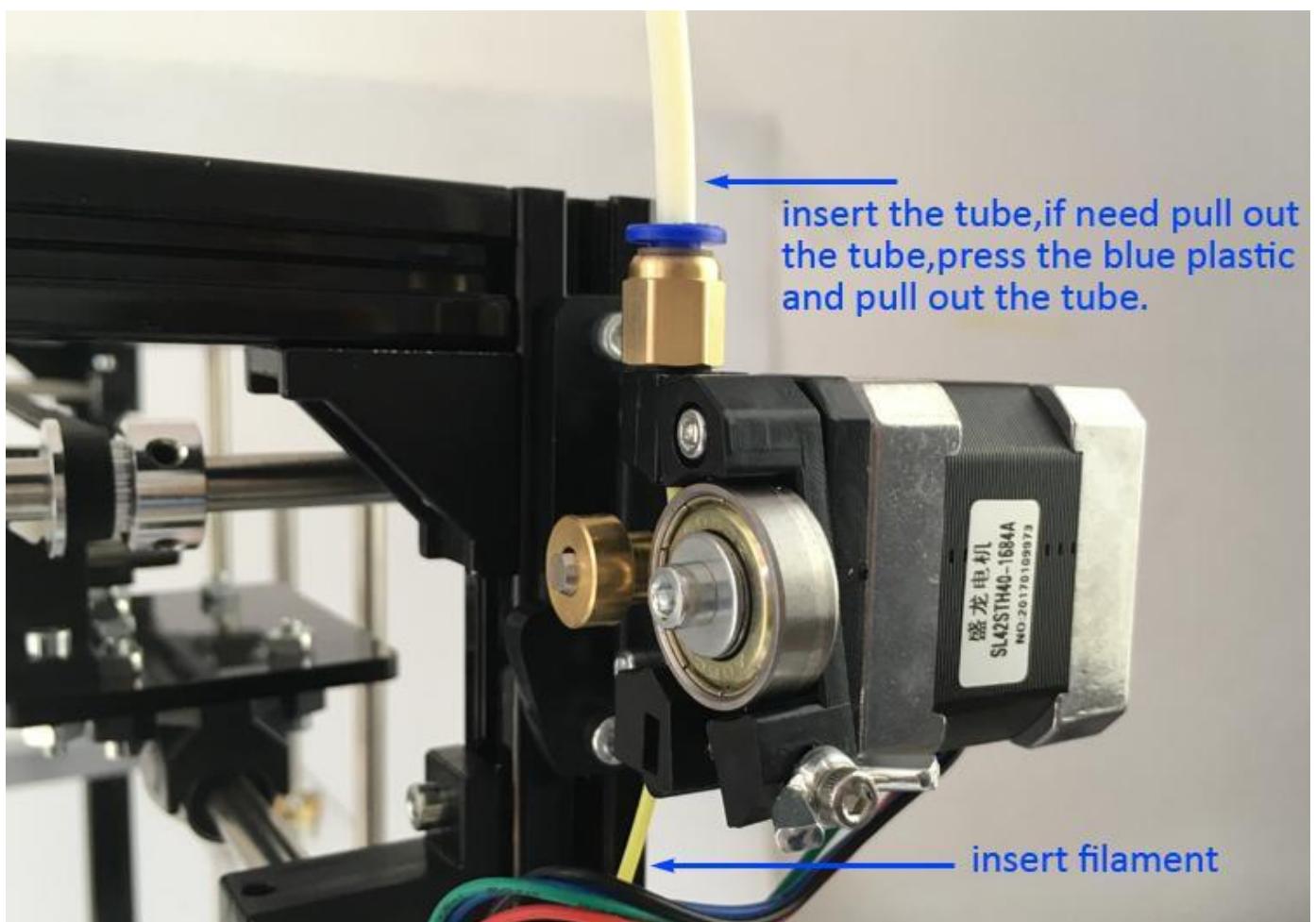
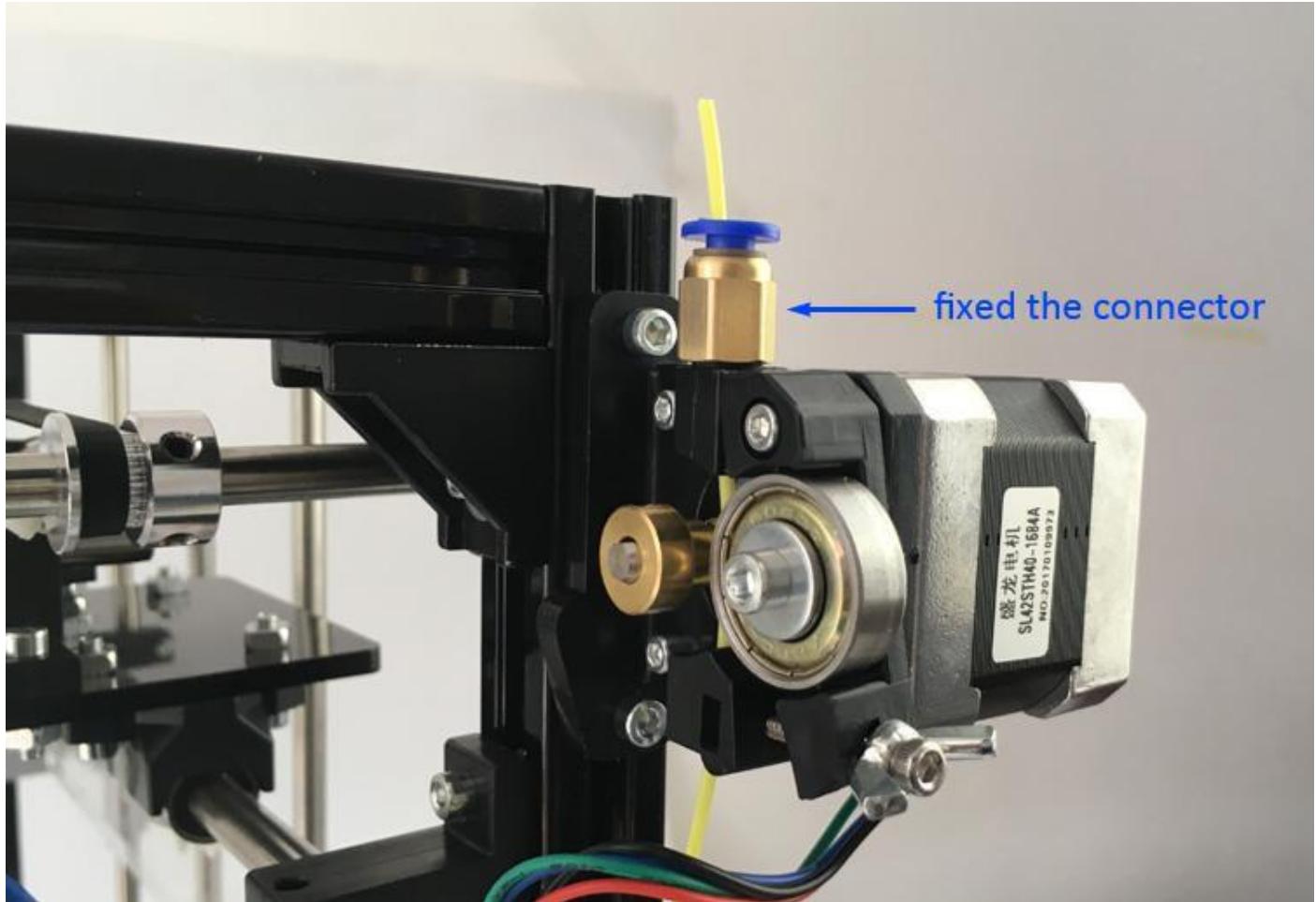
2,Start printing

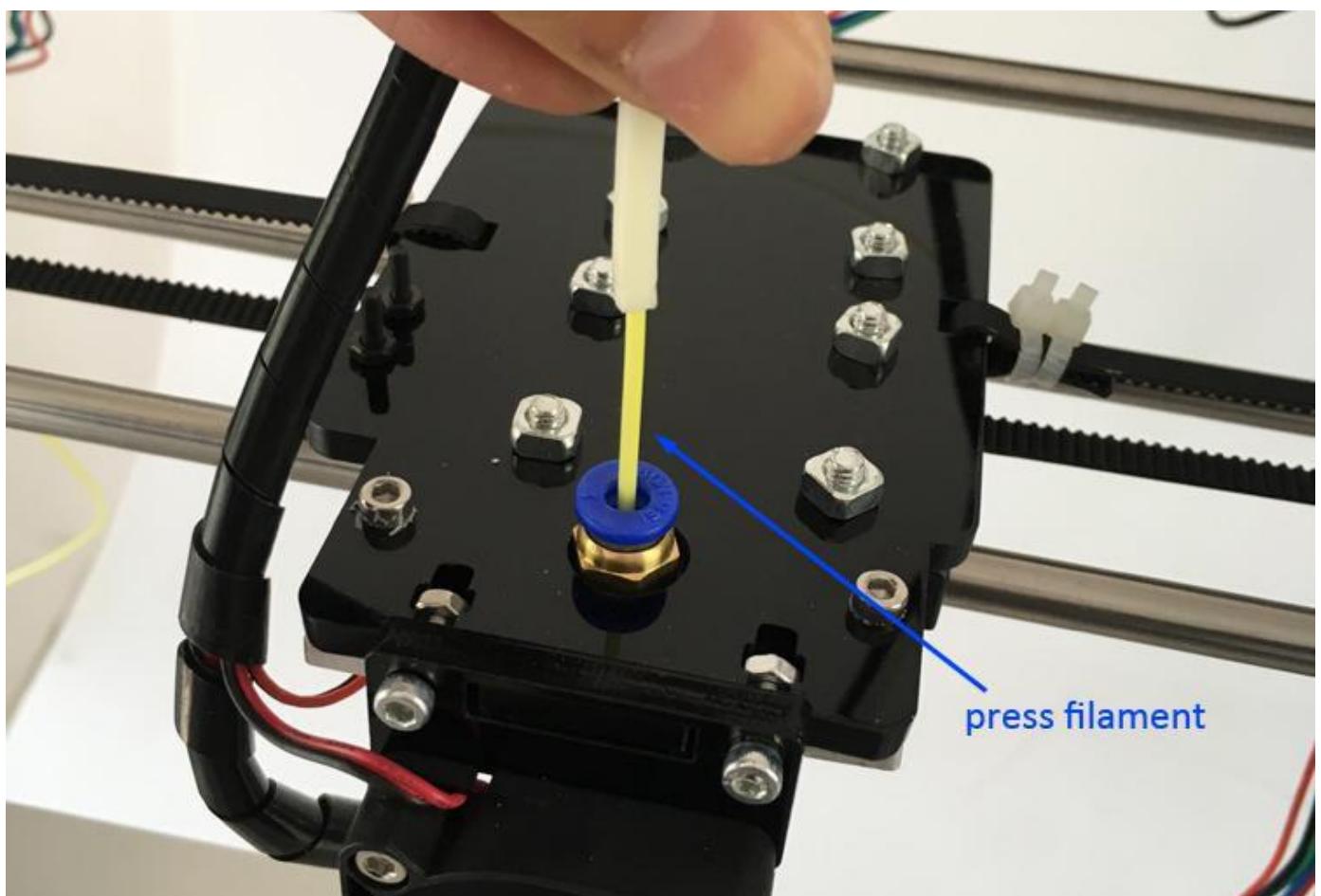
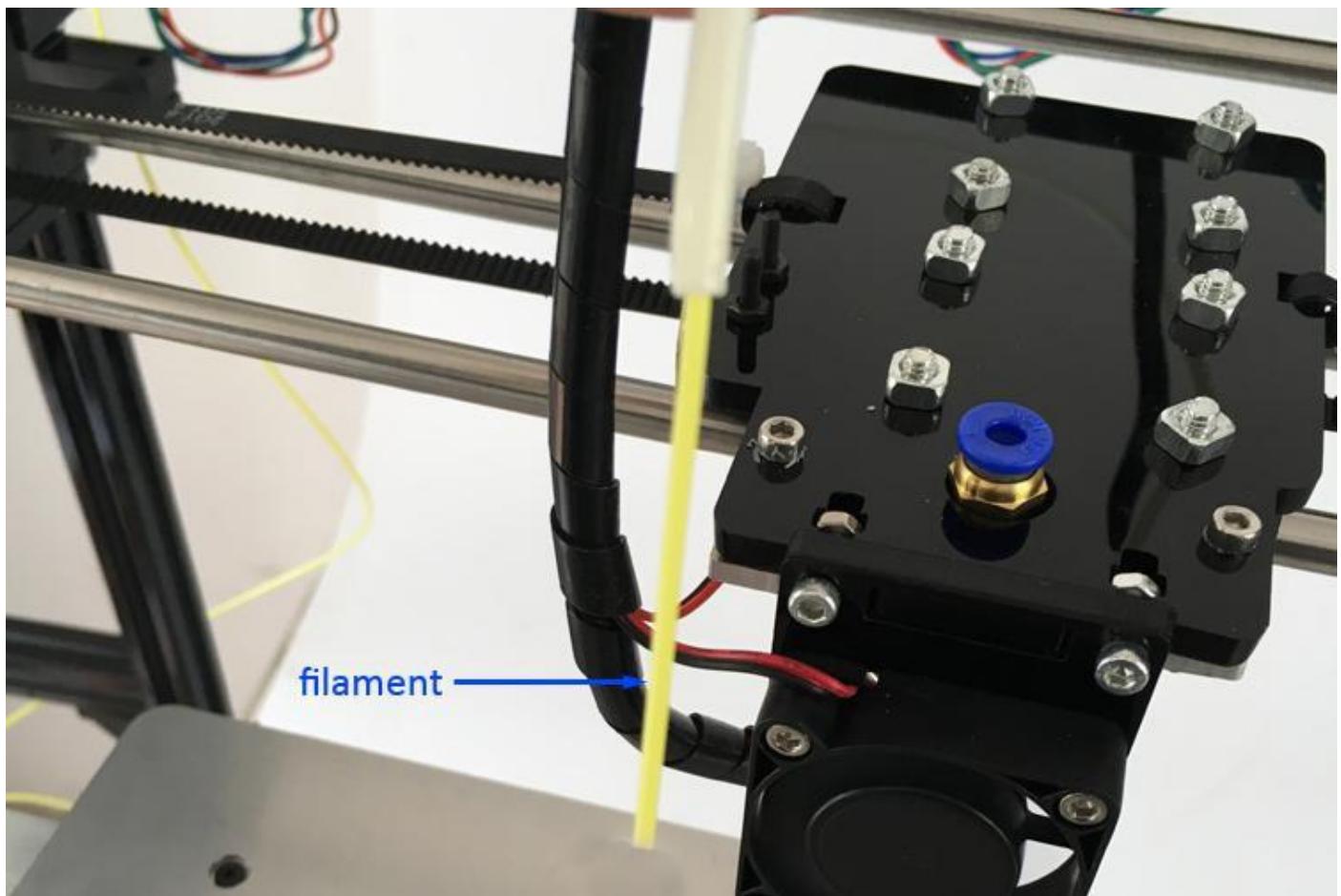
1 Feeder filament

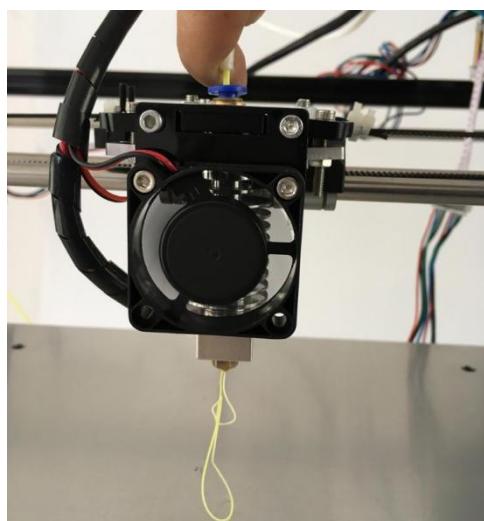
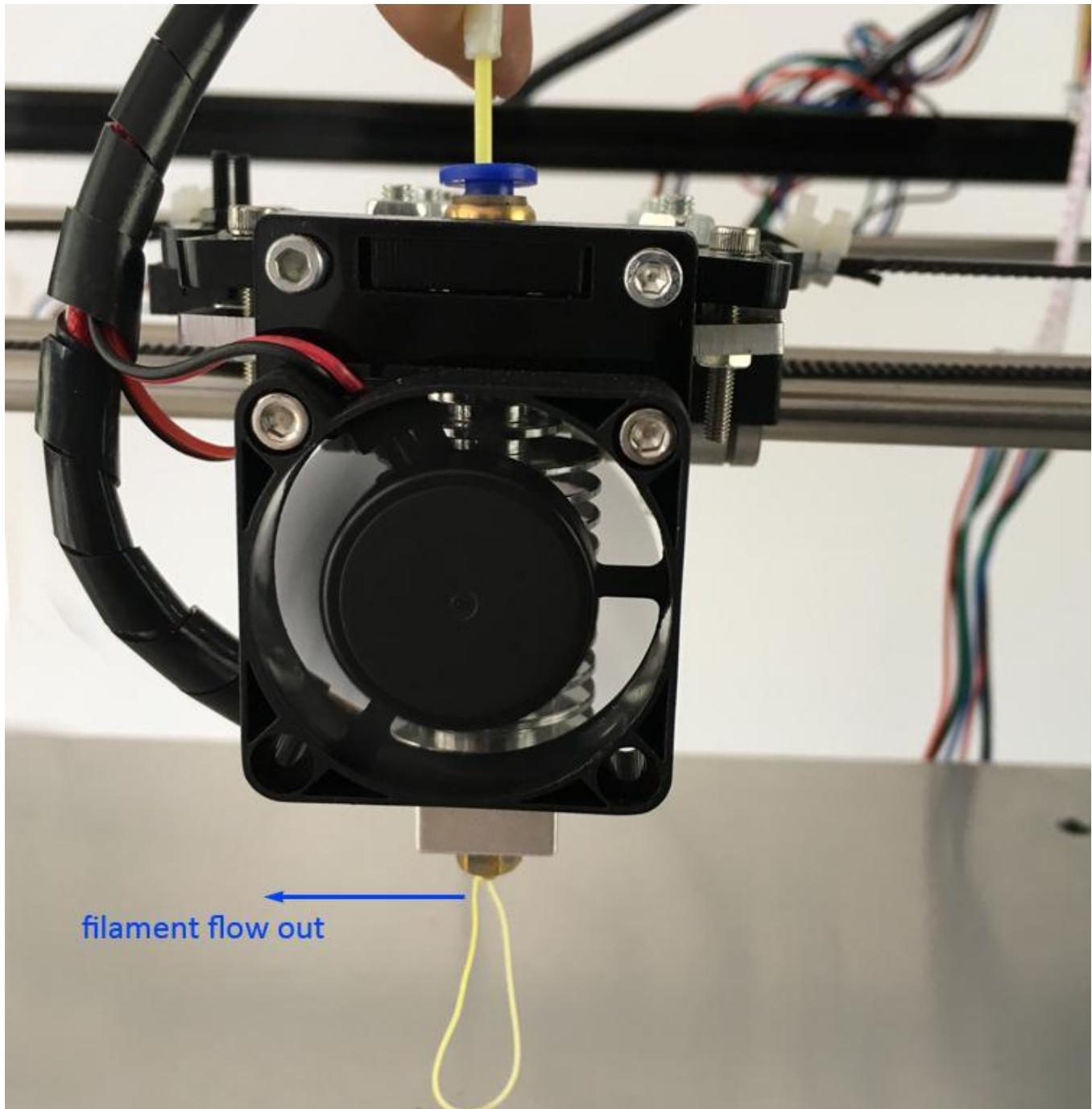
Before printing,connect USB and power supply,then feeder filament.

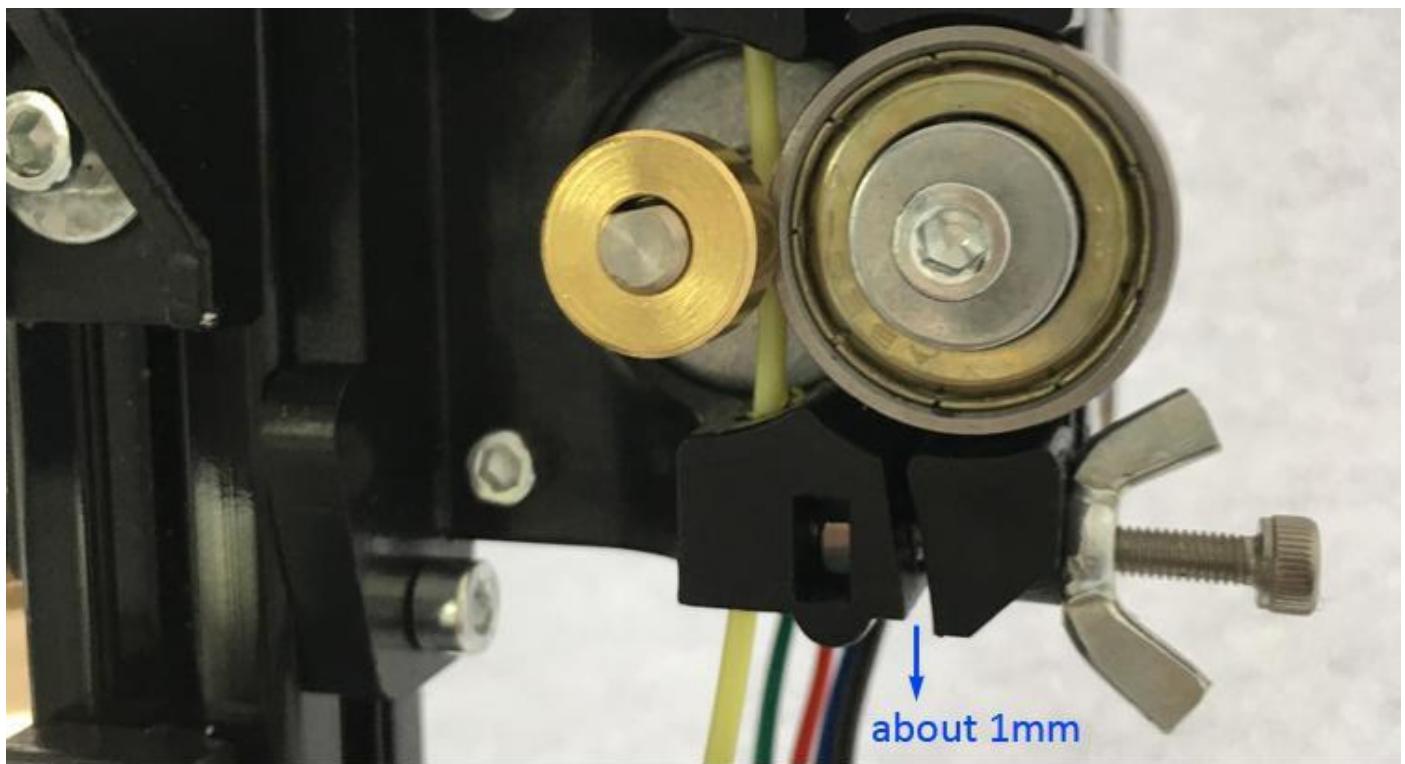
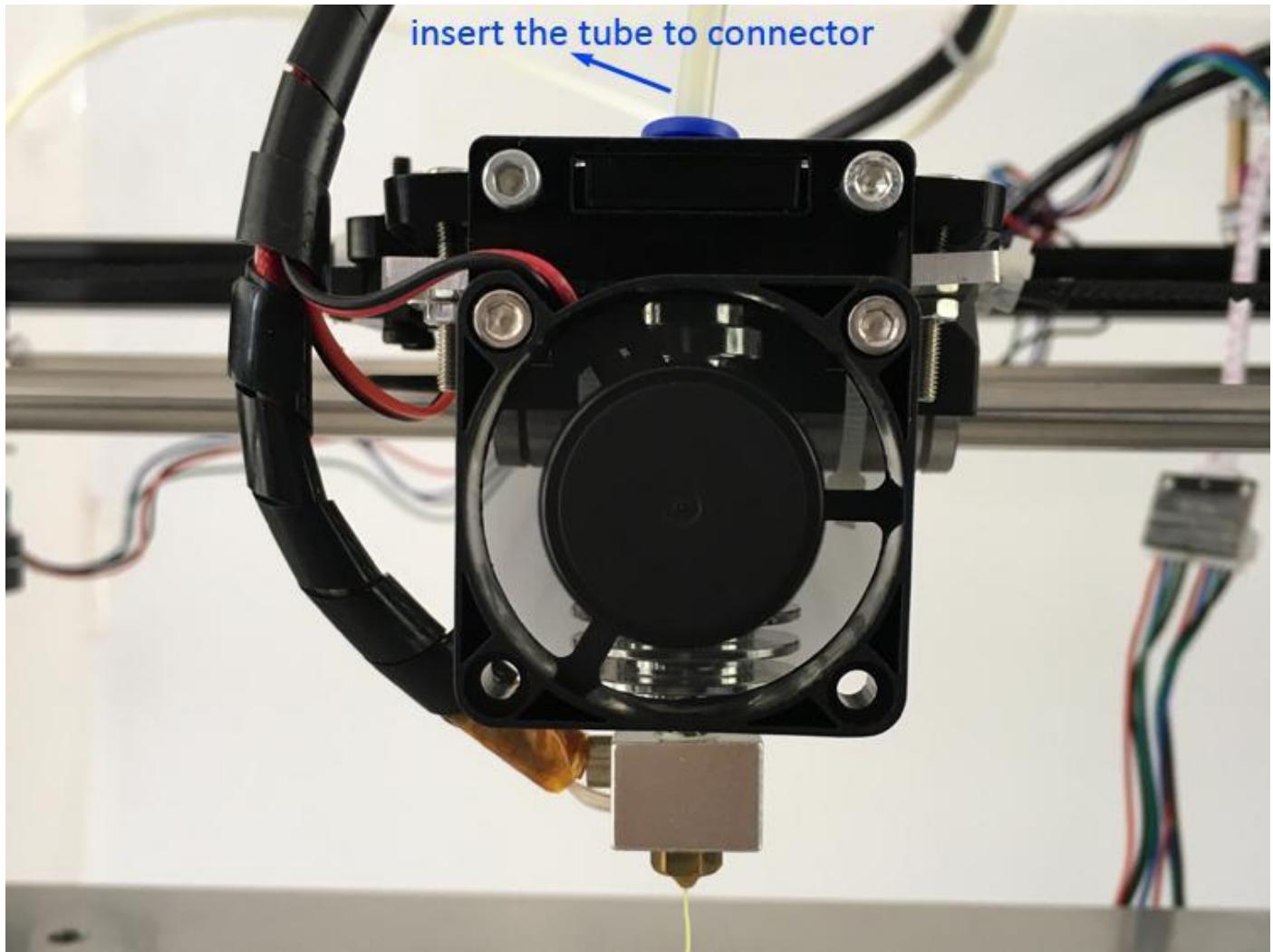








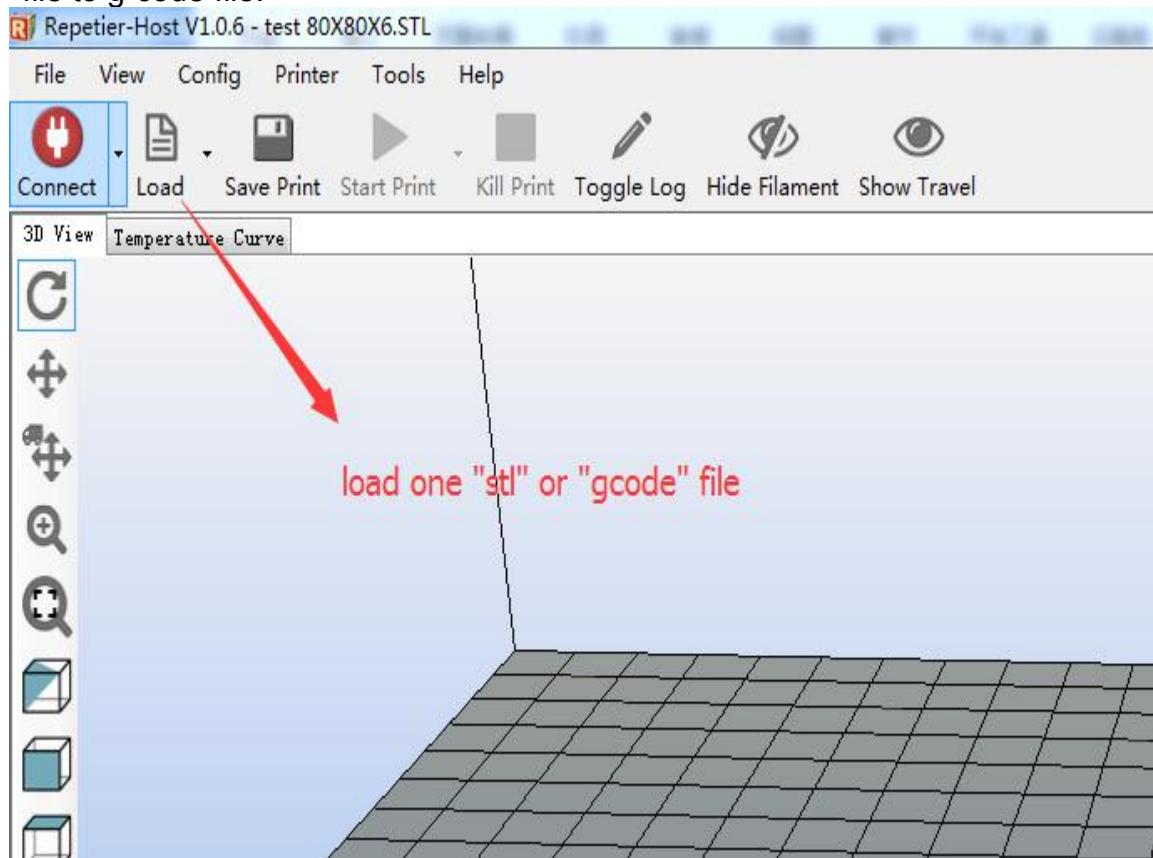




tight the butterfly nut, the butterfly can not tight too much, the gap is about 1 mm. if tight too much, will stop the filament.

2 Slice

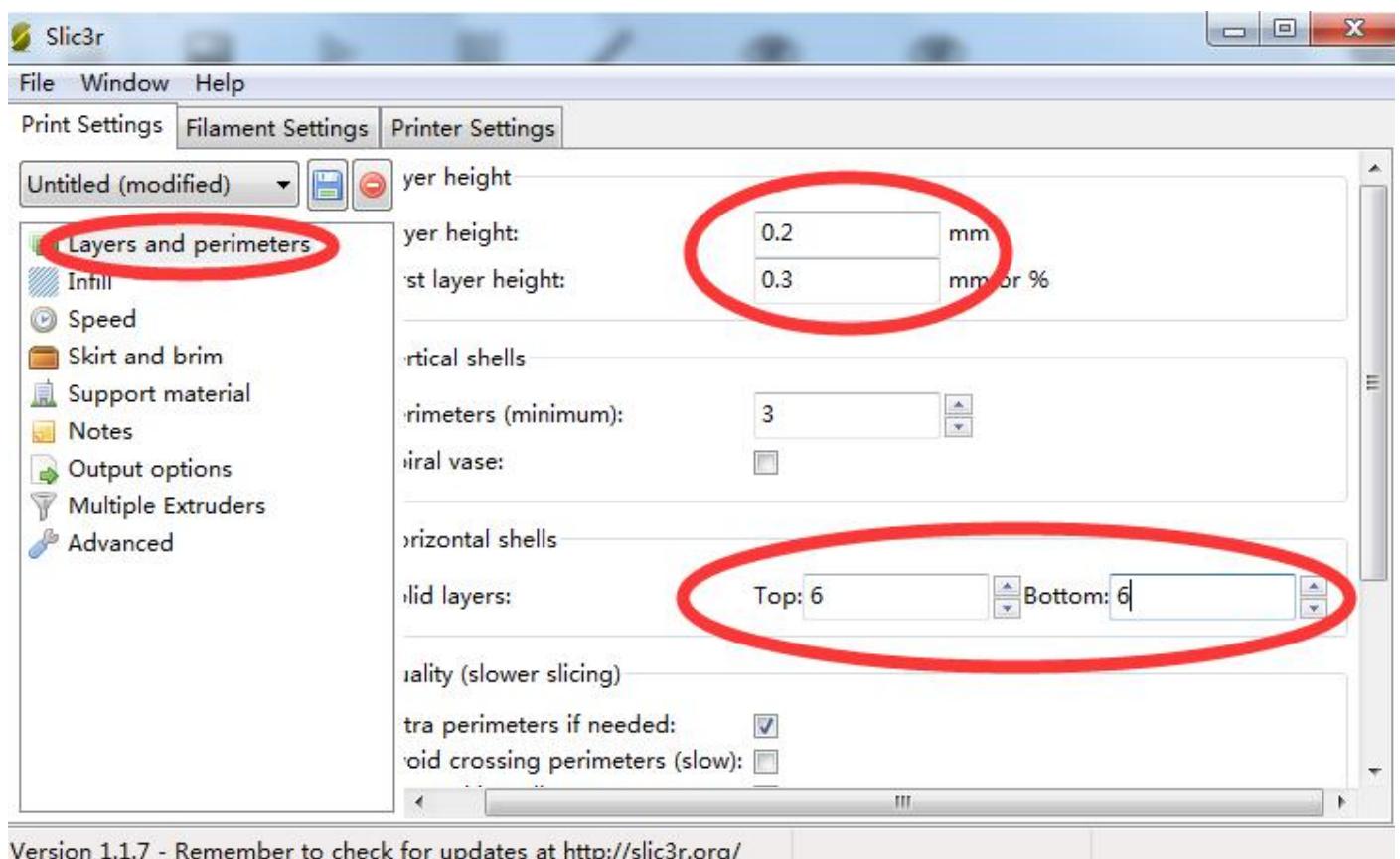
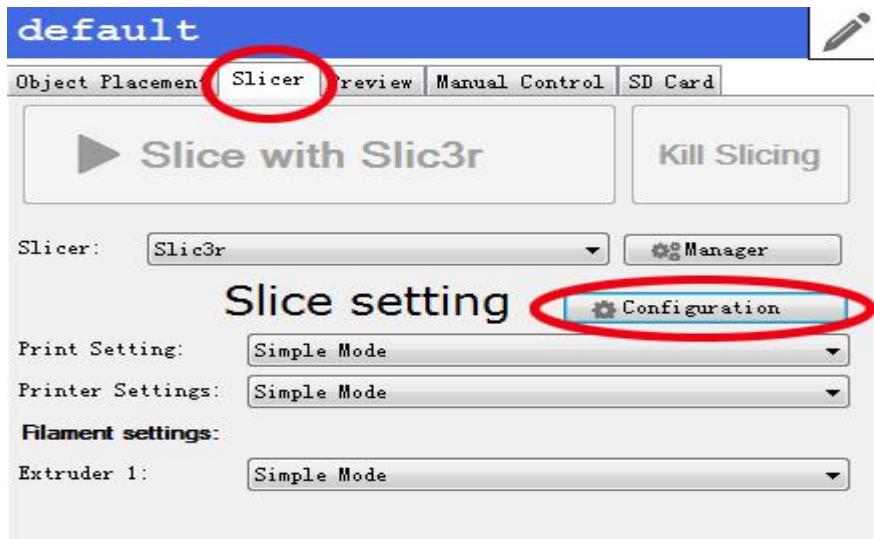
The 3d printer only can Identify g-code files, so need slice first, transfer the stl file to g-code file.



bot.gcode	2017/3/24 11:24	GCODE 文件	1,324 KB
bot.stl	2017/3/24 11:24	证书信任列表	764 KB
cov.gcode	2017/3/24 11:24	GCODE 文件	527 KB
cov.stl	2017/3/24 11:24	证书信任列表	317 KB
D20.gcode	2017/3/24 11:24	GCODE 文件	182 KB
D20.stl	2017/3/24 11:24	证书信任列表	529 KB
m10-8c.gcode	2017/3/24 11:24	GCODE 文件	1,045 KB
m10-8c.stl	2017/3/24 11:24	证书信任列表	529 KB
m10-20.gcode	2017/3/24 11:24	GCODE 文件	2,405 KB
m10-20.stl	2017/3/24 11:24	证书信任列表	978 KB

choose the file you will print

3 slice setting



File Window Help

Print Settings Filament Settings Printer Settings

My Settings (modified ▾)



Layers and perimeters

Infill

Speed

Skirt and brim

Support material

Notes

Output options

Multiple Extruders

Advanced

Infill

Fill density:

20 %

Fill pattern:

rectilinear

Top/bottom fill pattern:

rectilinear

Reducing printing time

Combine infill every:

1 layers

Only infill where needed:



Advanced

Solid infill every:

0 layers

Fill angle:

45 °

Solid infill threshold area:

70 mm²

Only retract when crossing
perimeters:



Infill before perimeters:



File Window Help

Print Settings Filament Settings Printer Settings

My Settings (modified ▾)



Filament

Cooling

PLA filament

Filament

Diameter:

1.75

mm

Extrusion multiplier:

1

Temperature (°C)

Extruder:

First layer: 210

Other layers: 205

Bed:

First layer: 0

Other layers: 0

Print Settings Filament Settings Printer Settings

My Settings (modified)

Enable

Keep fan always on:

Enable auto cooling:

If estimated layer time is below ~30s, fan will run at 100% and print speed will be reduced so that no less than 30s are spent on that layer (however, speed will never be reduced below 10mm/s).
If estimated layer time is greater, but still below ~60s, fan will run at a proportionally decreasing speed between 100% and 80%.
During the other layers, fan will always run at 80% except for the first layer.

Fan settings

Fan speed: Min: 80 Max: 100
Bridges fan speed: 100 %
Disable fan for the first: 1 layers

Cooling thresholds

Enable fan if layer print time is below: 60 approximate seconds
Slow down if layer print time is below: 30 approximate seconds
Min print speed: 10 mm/s

this fan is turbo fan, can be controled by repetier

File Window Help

Print Settings Filament Settings Printer Settings

My Settings

General
Custom G-code
Extruder 1

Size

Nozzle diameter: 0.4 mm

Position (for multi-extruder printers)

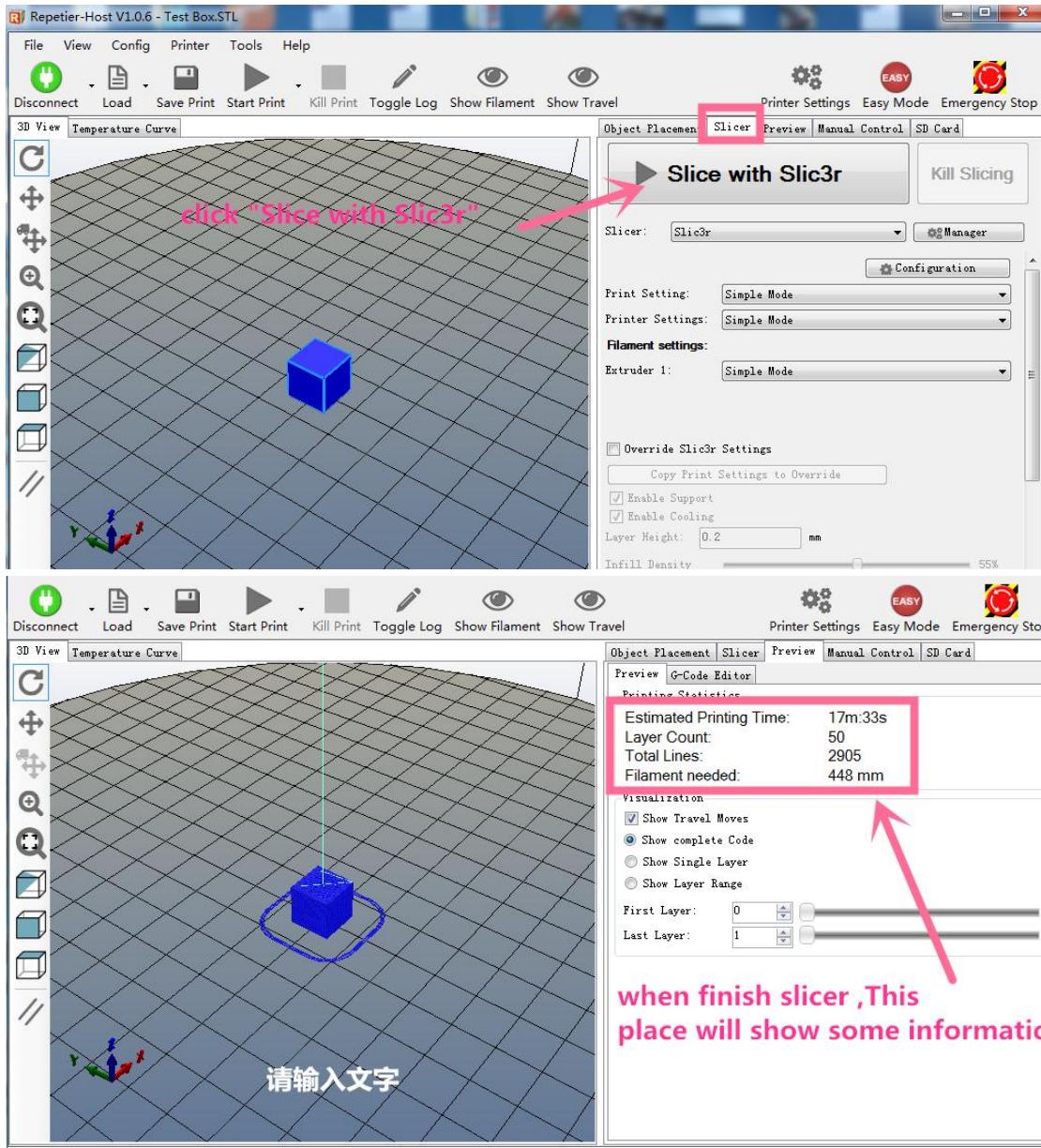
Extruder offset: x: 0 y: 0 mm

Retraction

Length: 5 mm (zero to disable)
Lift Z: 0 mm
Speed: 30 mm/s
Extra length on restart: 0 mm
Minimum travel after retraction: 2 mm
Retract on layer change:
Wipe while retracting:

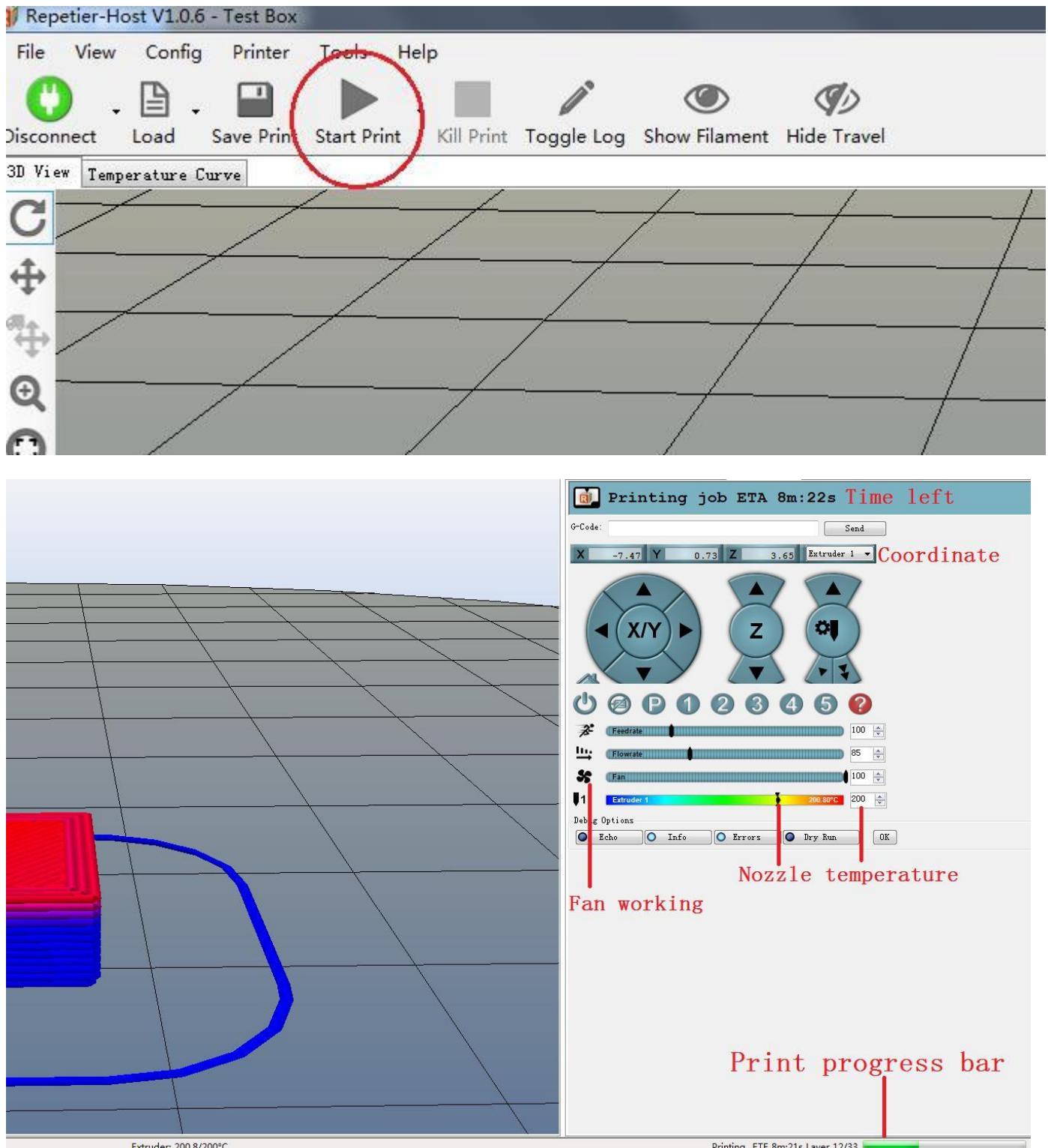
Retraction when tool is disabled (advanced settings for multi-extruder setups)

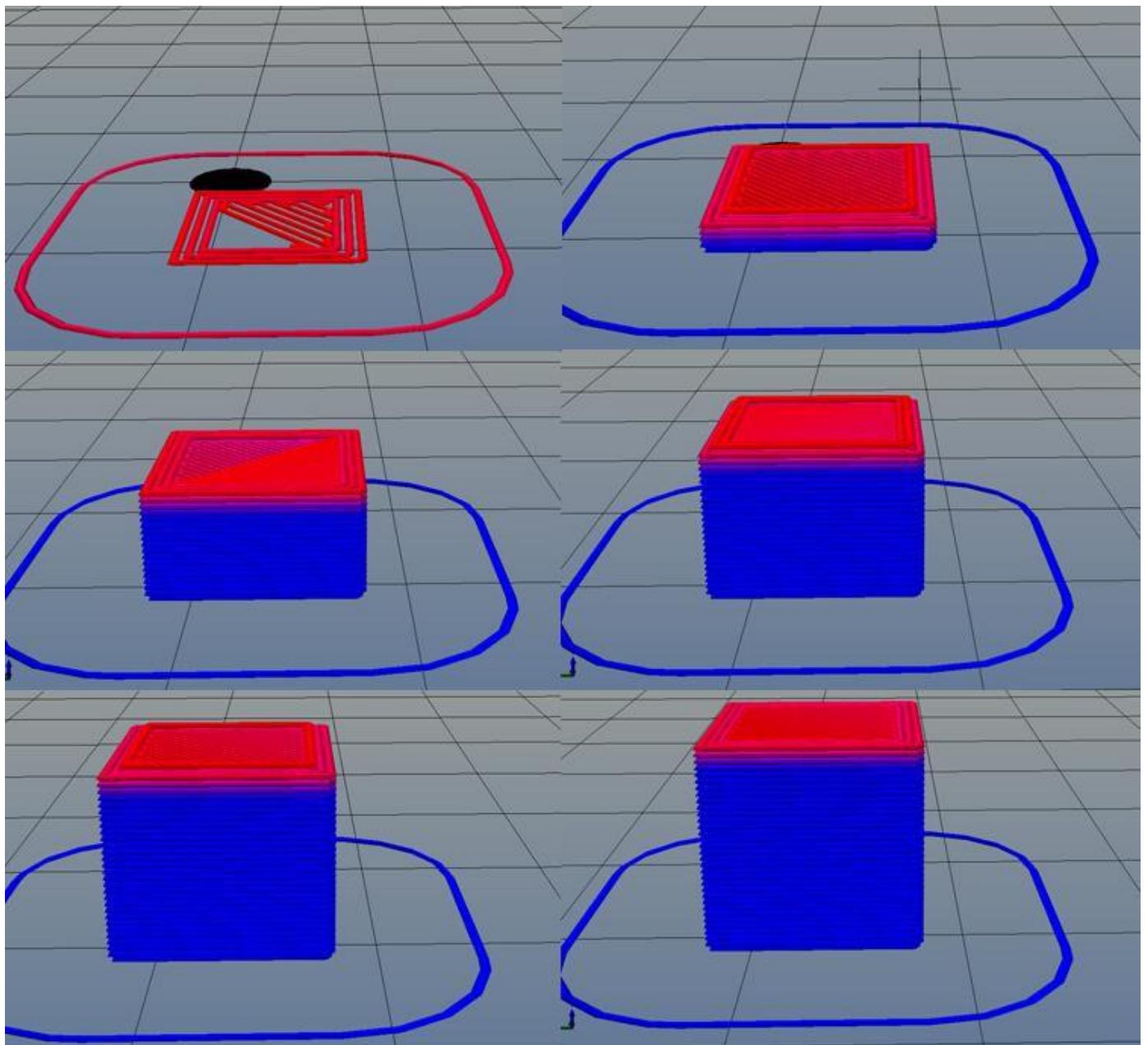
Length: 10 mm (zero to disable)
Extra length on restart: 0 mm



4 printing

Complete sliced, could start printing, click the “Start print”, will print the model, the panel display printing progress.





If when print ,the distance between nozzle and bed is big (filament can't stick at bed),you can reduce the offset ,E.g:change -1.4 to -1.5,-1.8,-2.0.

If when print ,the distance between nozzle and bed is too small(nozzle hit bed),you can add the offset in marlin ,eg:change -1.4 to -1.2,-1.0,-0.8.

```

/*
 *  Z Probe to nozzle (X,Y) offset, relative to (0, 0).
 *  X and Y offsets must be integers.
 *
 *  In the following example the X and Y offsets are both positive:
 *  #define X_PROBE_OFFSET_FROM_EXTRUDER 10
 *  #define Y_PROBE_OFFSET_FROM_EXTRUDER 10
 *
 *  +--- BACK ---+
 *  |           |
 *  L | (+) P | R <- probe (20,20)
 *  E |           | I
 *  F | (-) N (+) | G <- nozzle (10,10)
 *  T |           | H
 *  | (-)     | T
 *  |           |
 *  O--- FRONT ---+
 *  (0,0)
*/
#define X_PROBE_OFFSET_FROM_EXTRUDER 15      // X offset: -left +right [of the nozzle]
#define Y_PROBE_OFFSET_FROM_EXTRUDER -11       // Y offset: -front +behind [the nozzle]
#define Z_PROBE_OFFSET_FROM_EXTRUDER -2        // Z offset: -below +above [the nozzle]

```

The value is at configuration.h ,481line.

3 Marlin

Sometimes we will encounter some problems that need to be changed and uploaded marlin

1 Arduino

The marlin is the printer program which run the printer,it's also named firmware.

Arduino is an working environment for firmware, only installing the arduino on PC,then can open marlin and flash program to board, if you are not familiar with the software,pls don't change any parameters,thank you.

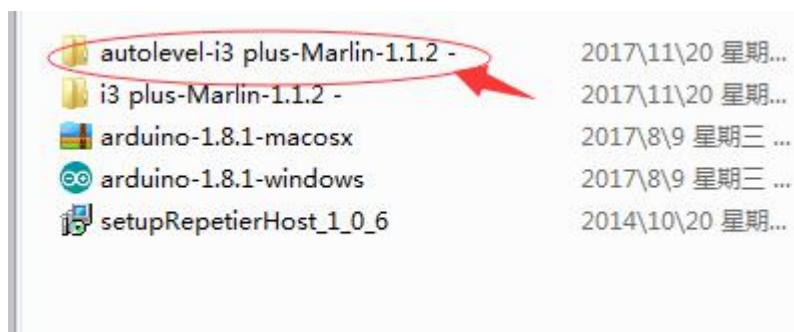
	autolevel-i3-Marlin-1.1.2 -	2017/8/2
	i3-Marlin-1.1.2 -	2017/8/2
	arduino-1.8.1-macosx	2017/8/9
	arduino-1.8.1-windows	2017/8/9
	autolevel-i3-Marlin-1.1.2 -	2017/8/2
	i3-Marlin-1.1.2 -	2017/8/2
	setupRepetierHost_1_0_6	2014/10/

2 Marlin

1 Open marlin

open marlin, if it's compressed file,must unzip it.

find the icon green color,this format is “ino”, click and open it in arduino.



least_squares_fit.cpp	2017/5/31 星期...	CPP 文件	3 KB
least_squares_fit.h	2017/5/31 星期...	H 文件	3 KB
M100_Free_Mem_Chk.cpp	2017/5/31 星期...	CPP 文件	12 KB
macros.h	2017/5/31 星期...	H 文件	7 KB
Makefile	2017/5/31 星期...	文件	16 KB
Marlin.h	2017/5/31 星期...	H 文件	16 KB
Marlin	2017/5/31 星期...	Arduino file	2 KB
Marlin_main.cpp	2017/5/31 星期...	CPP 文件	402 KB
MarlinConfig.h	2017/5/31 星期...	H 文件	2 KB
MarlinSerial.cpp	2017/5/31 星期...	CPP 文件	14 KB
MarlinSerial.h	2017/5/31 星期...	H 文件	7 KB

2.settings

Firmware could only be upload succeed when the settings like the above Go to tools > board >arduino mega 2560

Go to tools > port, select the COM port Go to tools

> programmers > USBtinyISP



2 Upload Firmware

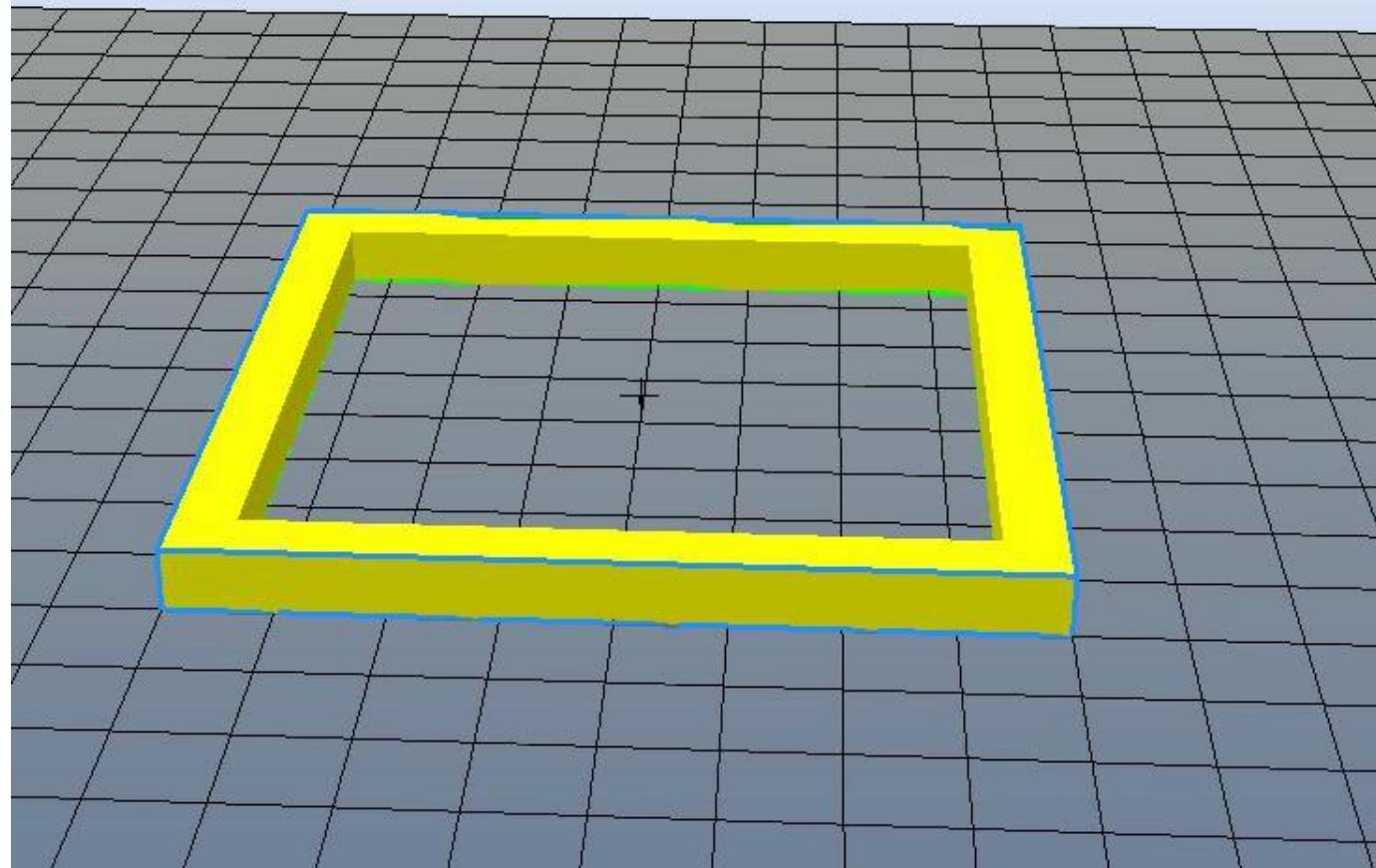
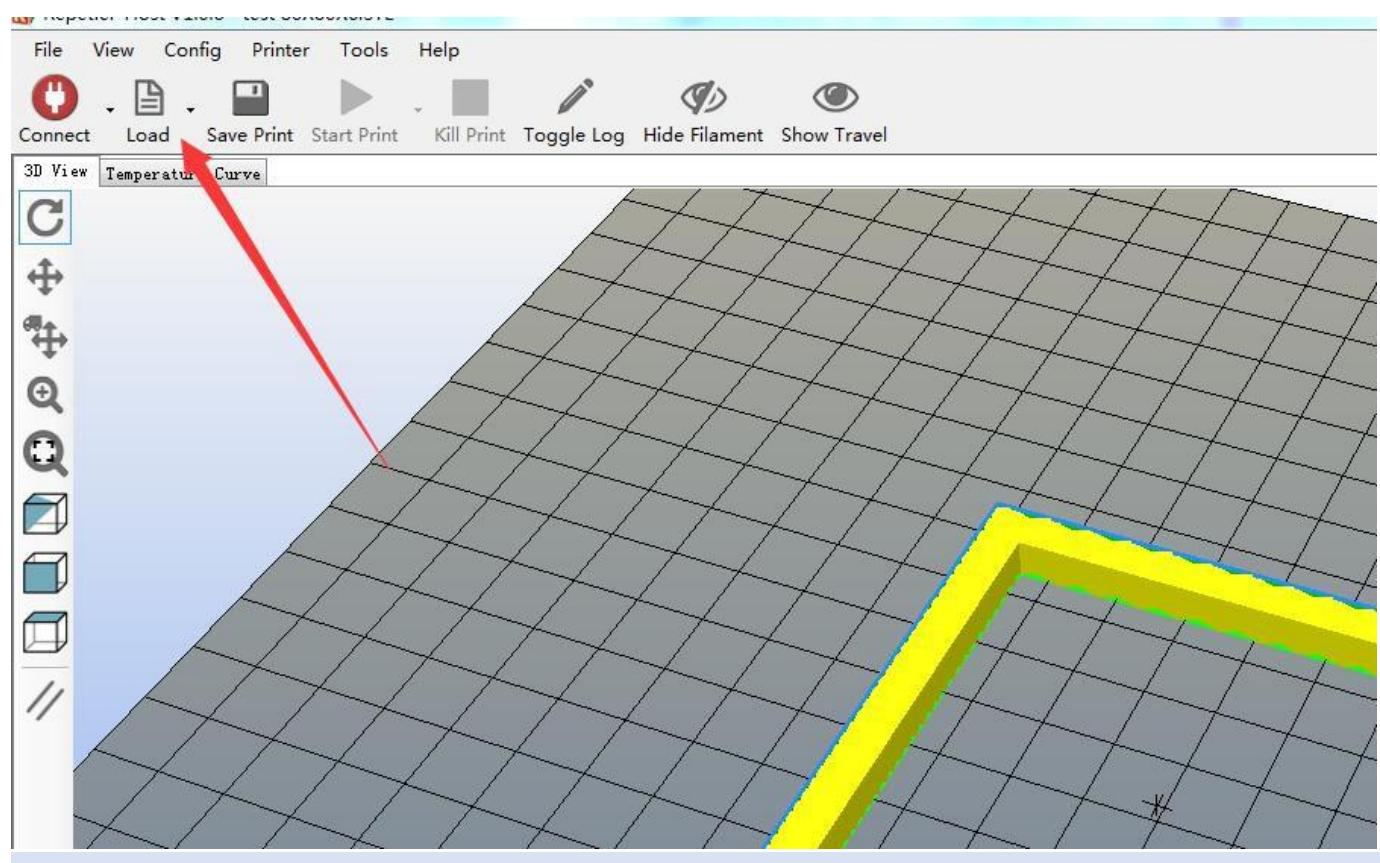
Click the 上传 (Upload) button to upload the Marlin Firmware Sketch to MKS Controller Board. Once upload successful, message “Done”.

pls disconnect repetier when flash firmware !!!

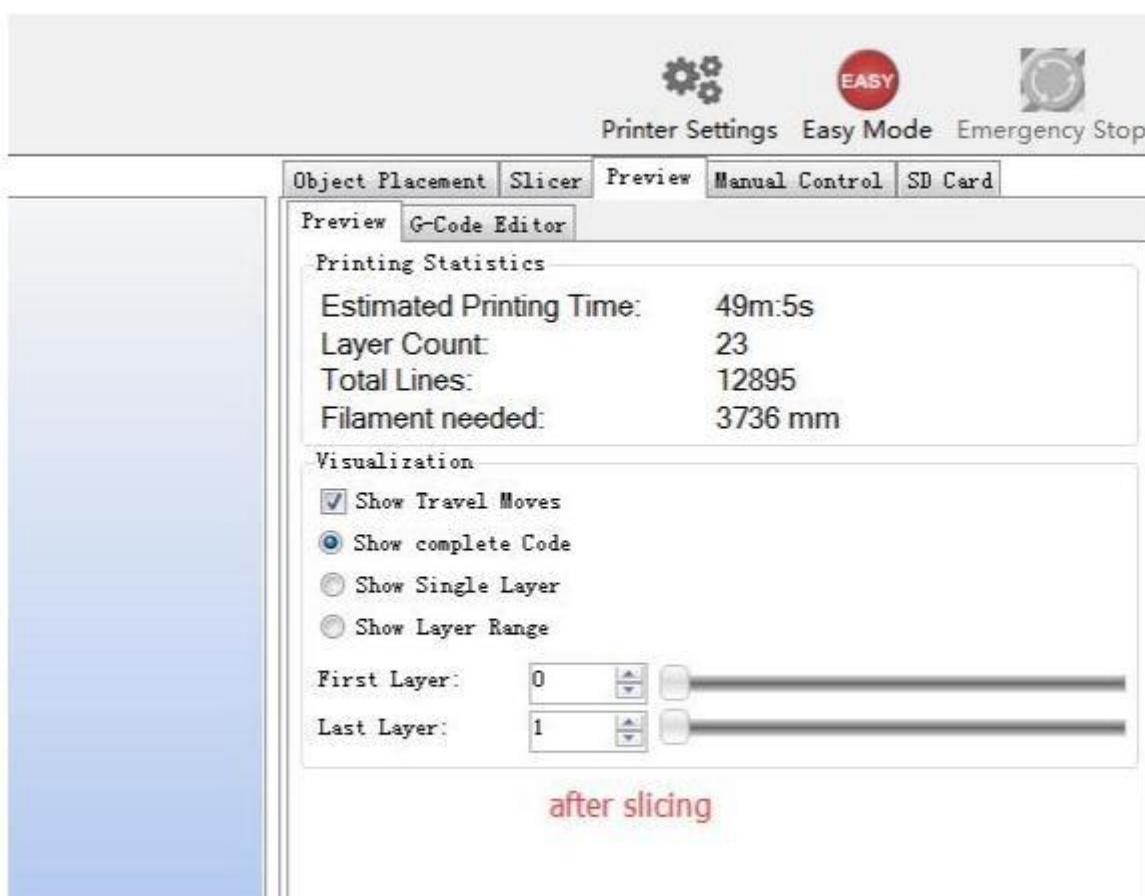
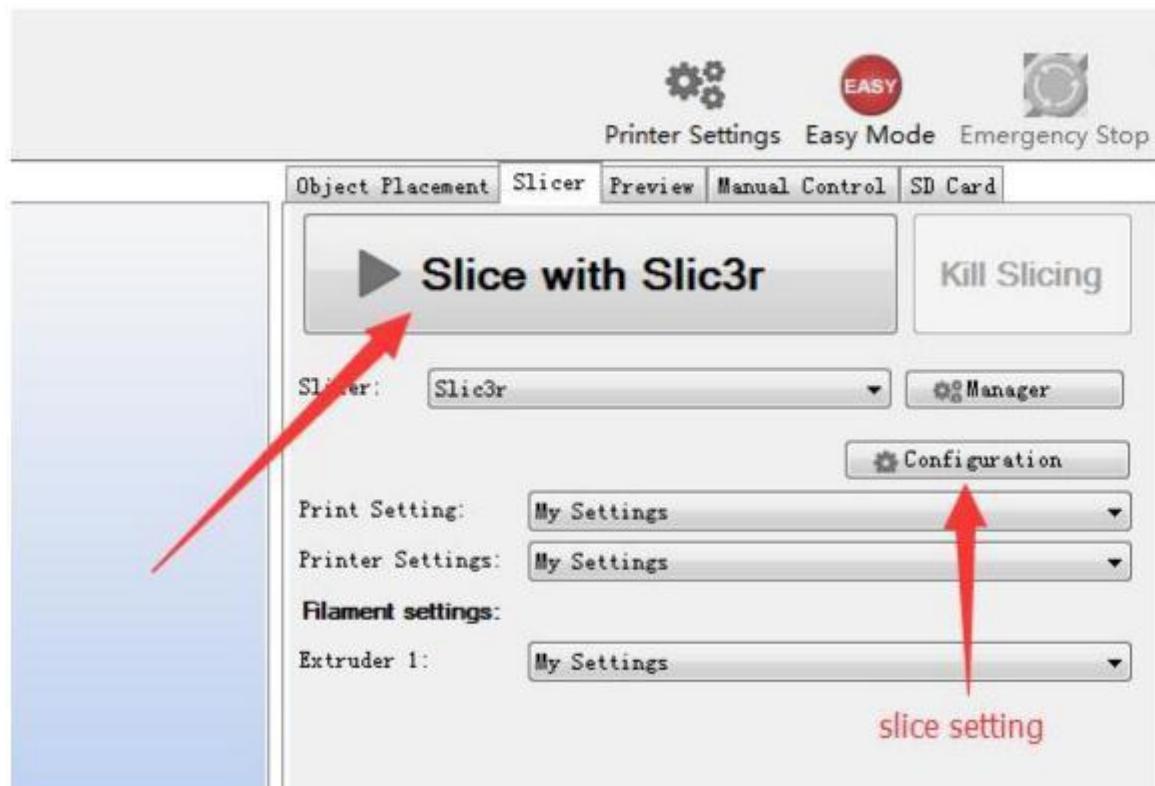


4 Offline printing

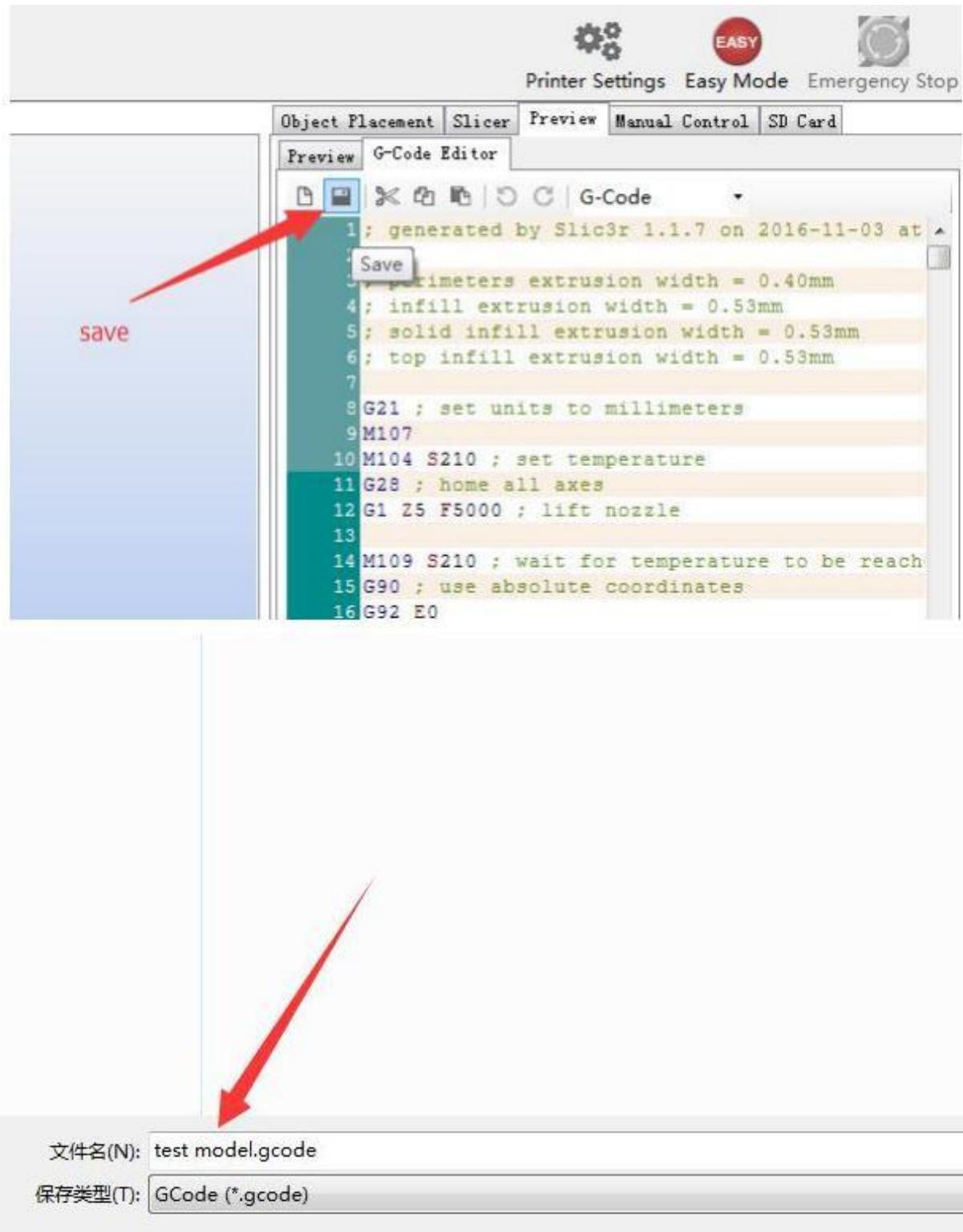
1 load the model



2 slice



3 Export Gcode file



new folder				
	名称	修改日期	类型	大小
	test model.gcode	2016/11/3 0:10	GCODE 文件	406 KB

↑ Gcode file

4 Copy

Copy the Gcode file to SD card and insert the SD card into LCD

5 Printing

Click on Print



Then choose the Gcode which you have copied

ReadyPrint>ChooseFile



bak_pic



filament
base.g



CH340G_U
SB to TT



Marlin



PLA Mode
ls



Page up



Page down



Back

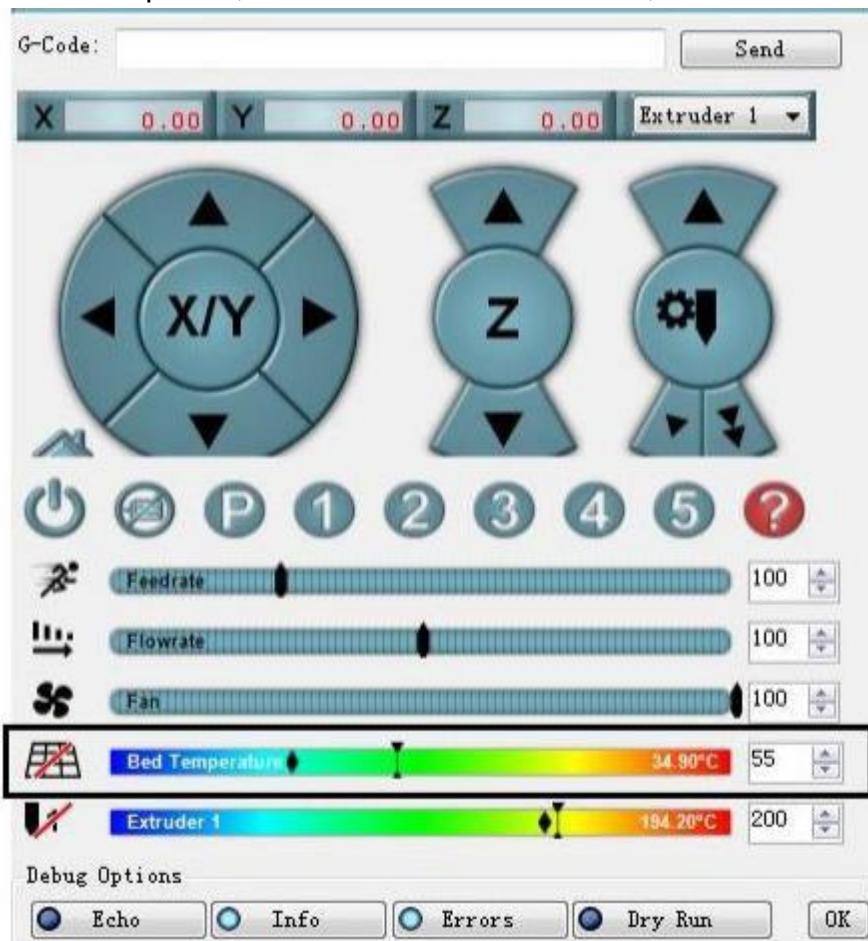
MKS TFT32_L V3.0 3 3

5 Heated bed

if use PLA filament,no need open heated bed.

If use ABS filament,need open heated bed, set the heated bed temp 85-100°C Must connect heat bed line to board heated bed sensor to board.

If use repetier to control the printer, can see heated bed choose,can set heated bed temp.



6 Problem and advice

1. The computer can not install arduino and other software

this situation often occur in the windows10, you can download Arduino1.8.1 online, pay attention to the correct version.

2. The computer cann't installed the drive

first enter the device manager to find the port item, double-click the unknown device port (unplug the motherboard, the port disappears, connect the motherboard, the port appears, the port is the printer port), click the 'update driver' button, then click find the driver software the computer ', and then in the' Browse 'to find your driver address, the next step, that's ok.

If you connect the motherboard and computer, in the device manager port, nothing did appear, there may be a problem with the motherboard, please contact us

3.marlin upload failed

Marlin toolbar configuration may be wrong, the board (2560), processor (2560), port (2560),

programmer (USBtinyISP) need to select the right

Repetier connection is not disconnected

Arduino don't be installed on right location ,don't be installed in C drive

arduino version wrong, the correct version is 1.6.1

4.repetier interface nozzle temperature is 0, the display shows "MINTEMP"

Nozzle temperature sensor / white line in the motherboard slot inside the loose, another direction to tighten

may be the nozzle temperature line / white line is broken, measured with a multimeter, the resistance is 100K, you can also insert the hot bed temperature Sensor into the nozzle temperature sensor slot to check.

5 if the motor steering is wrong(For example ,control it rises, it drops)

```
// @section extruder

#define DISABLE_E false // For all extruders
#define DISABLE_INACTIVE_EXTRUDER true // Keep only the active extruder enabled.

// @section machine

// Invert the stepper direction. Change (or reverse the motor connector) if an axis goes the wrong way.
#define INVERT_X_DIR false
#define INVERT_Y_DIR true
#define INVERT_Z_DIR true
// Enable this option for Toshiba stepper drivers
// #define CONFIG_STEPPERS_TOSHIBA

// @section extruder

// For direct drive extruder v9 set to true, for geared extruder set to false.
#define INVERT_E0_DIR false
#define INVERT_E1_DIR false
#define INVERT_E2_DIR false
#define INVERT_E3_DIR false
#define INVERT_E4_DIR false
```

This is the standard motor steering setting, here you can change the motor steering according to your need

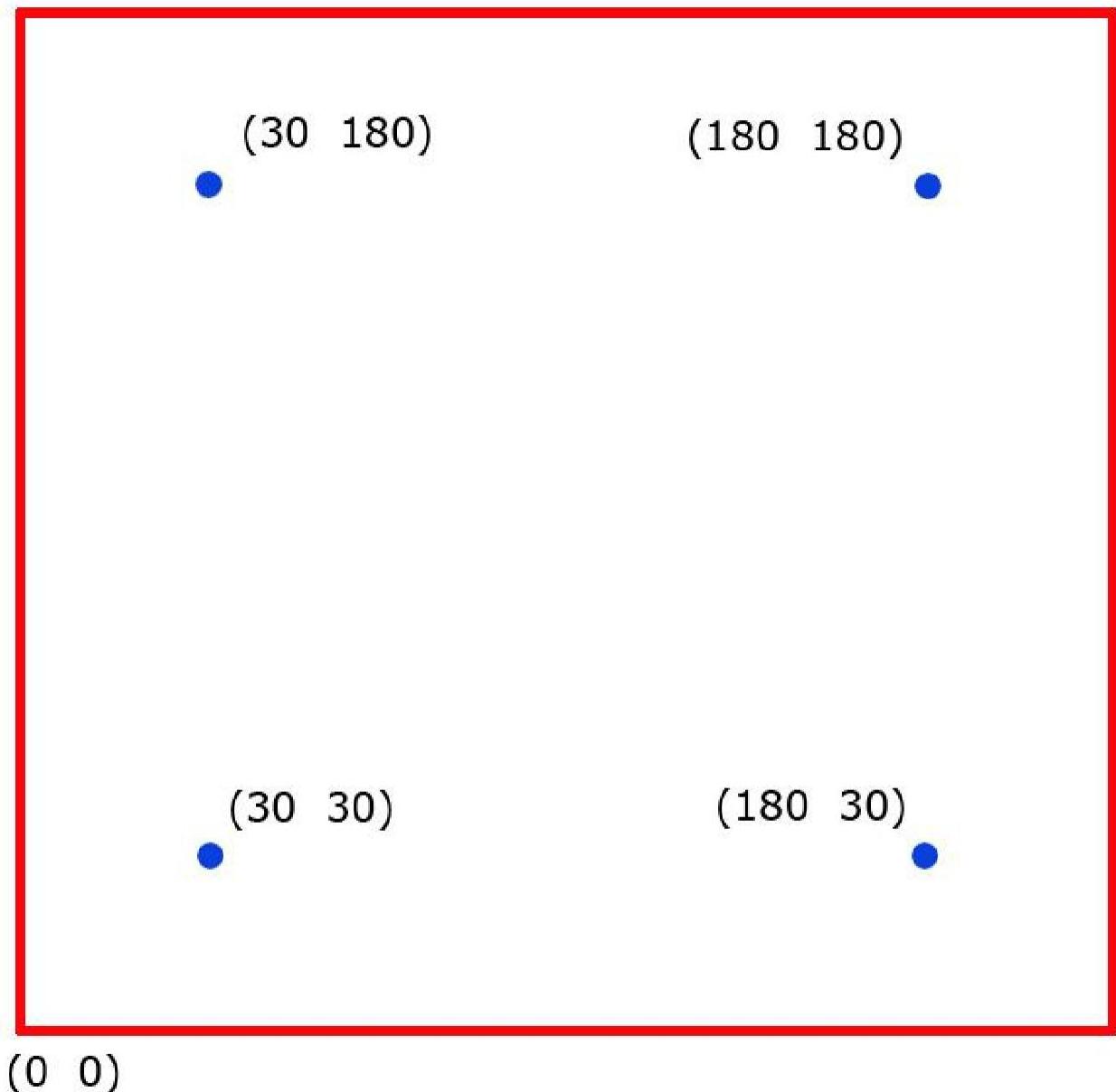
7 if your auto-level sensor is bad or not

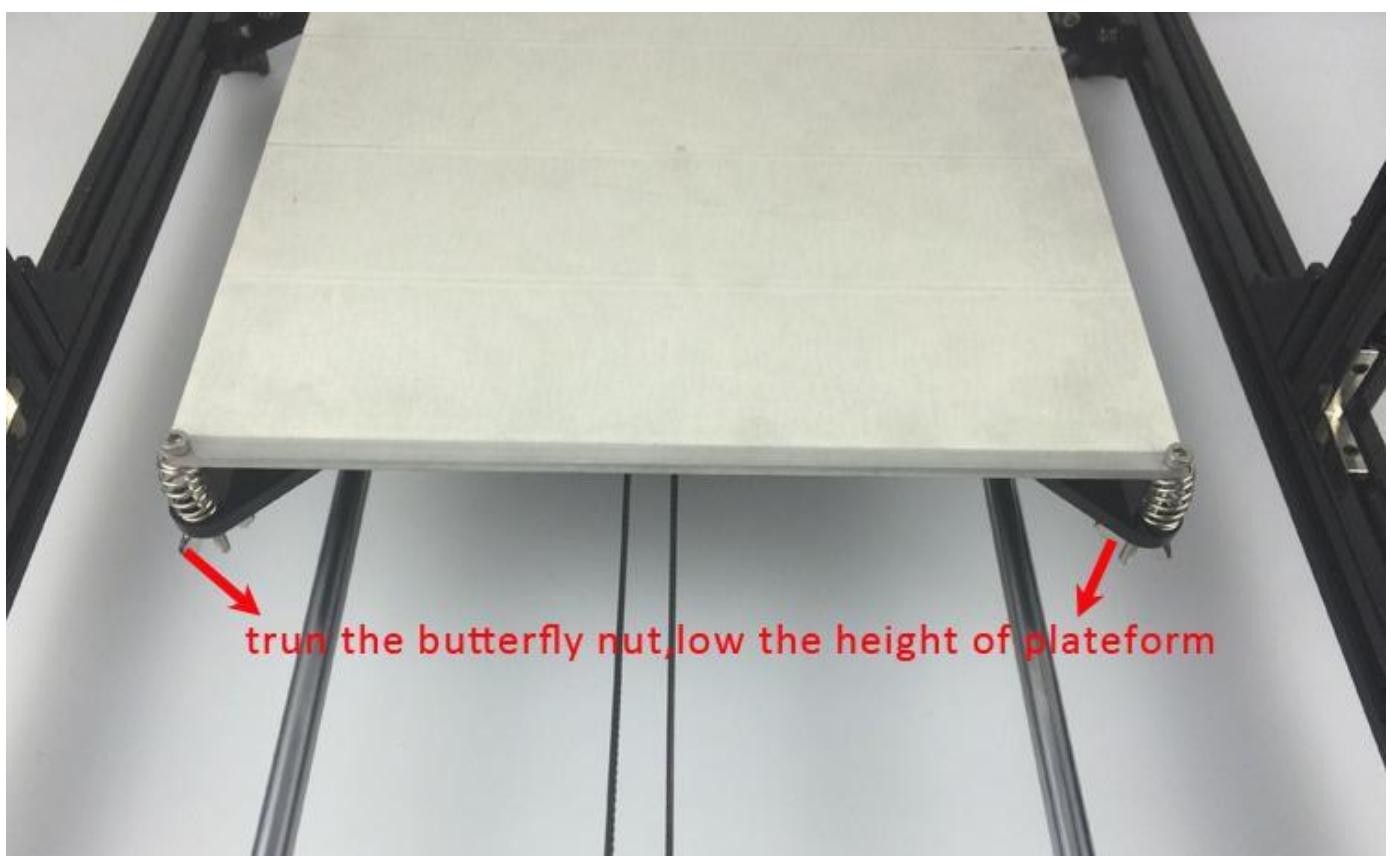
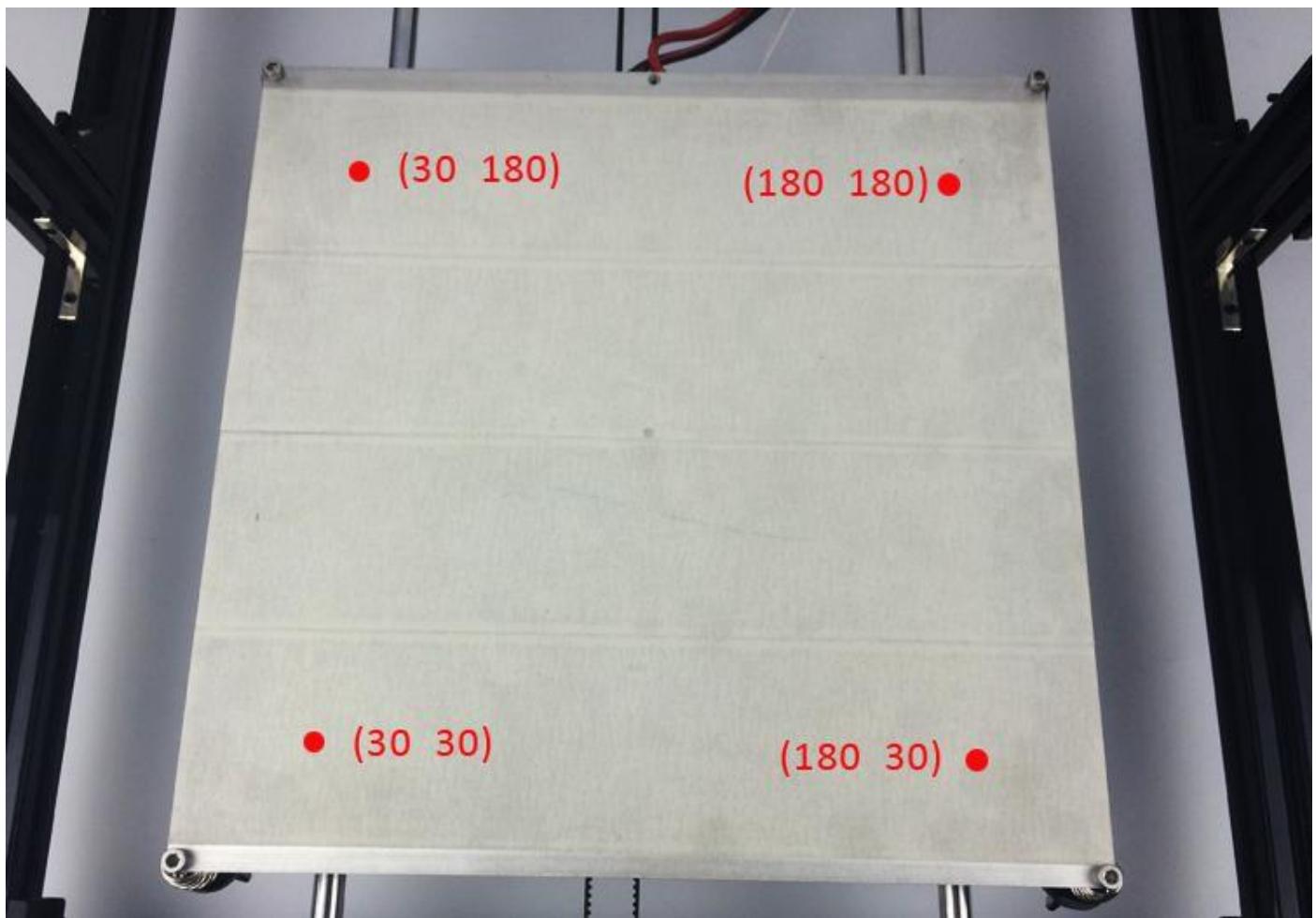
working ,you can use the manual leveling

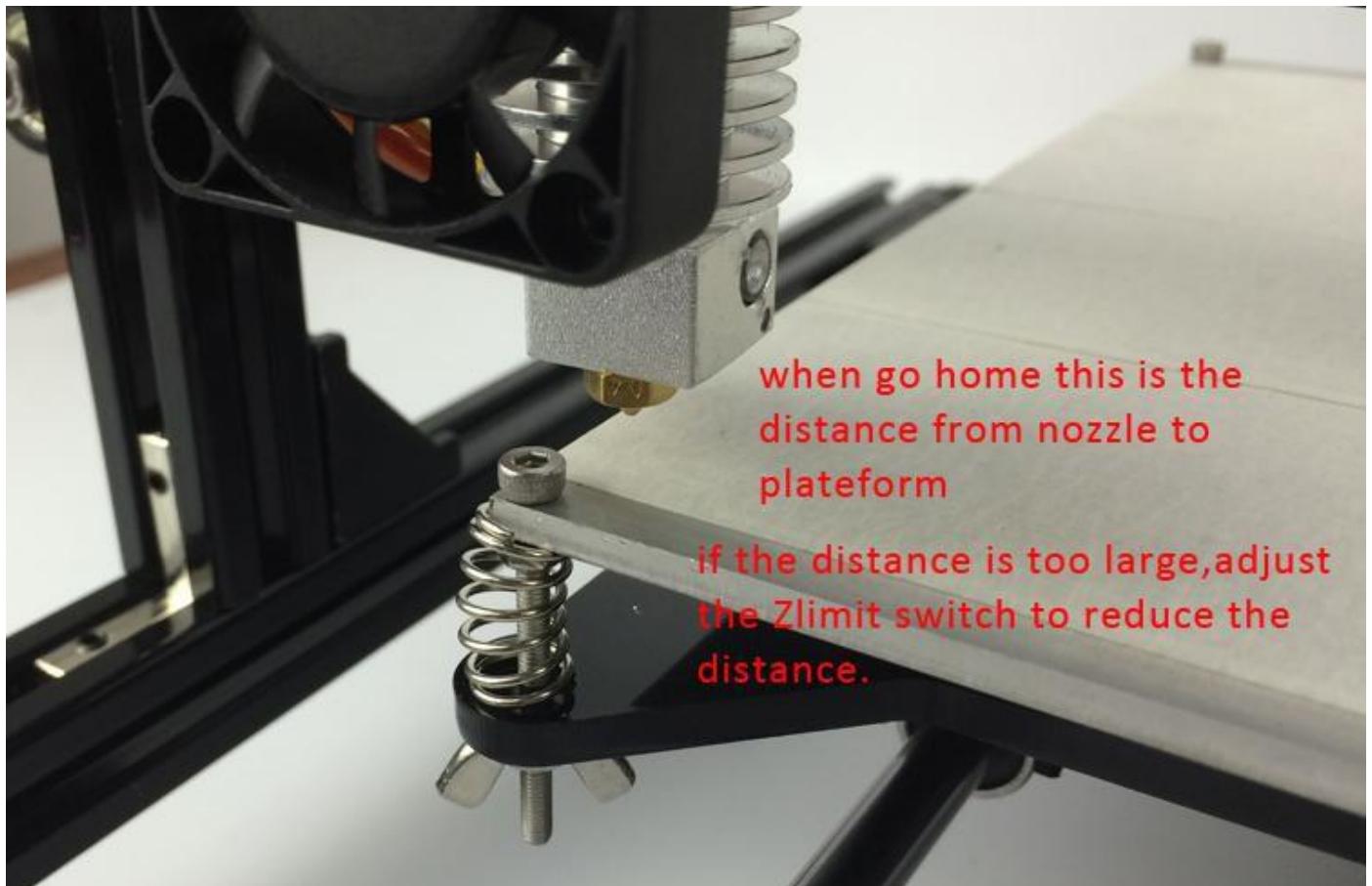
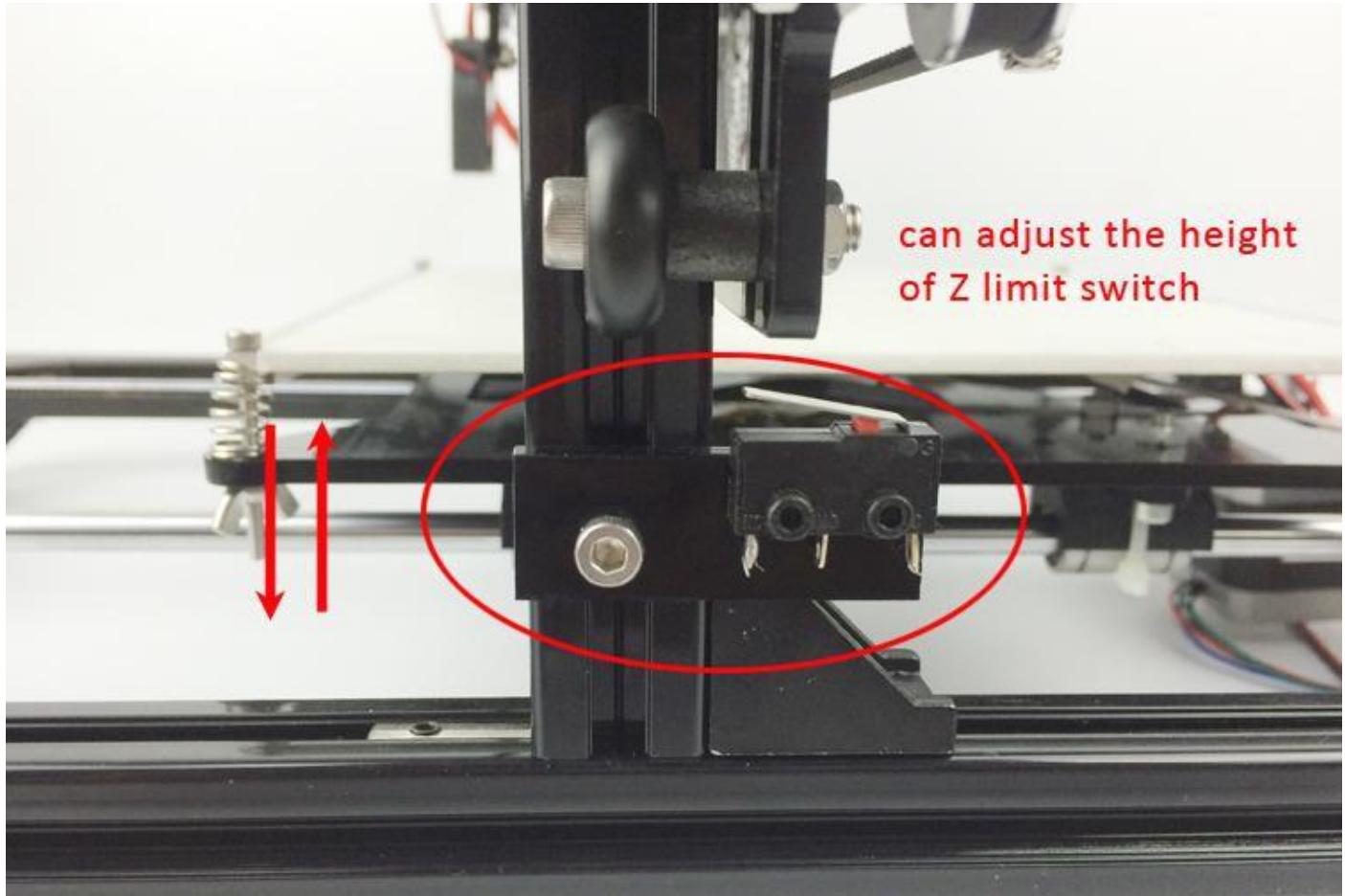
First you should installed the Z limit switch instead of the Auto-level sensor(refer to the the Assembly instructions from page 89 to page91)

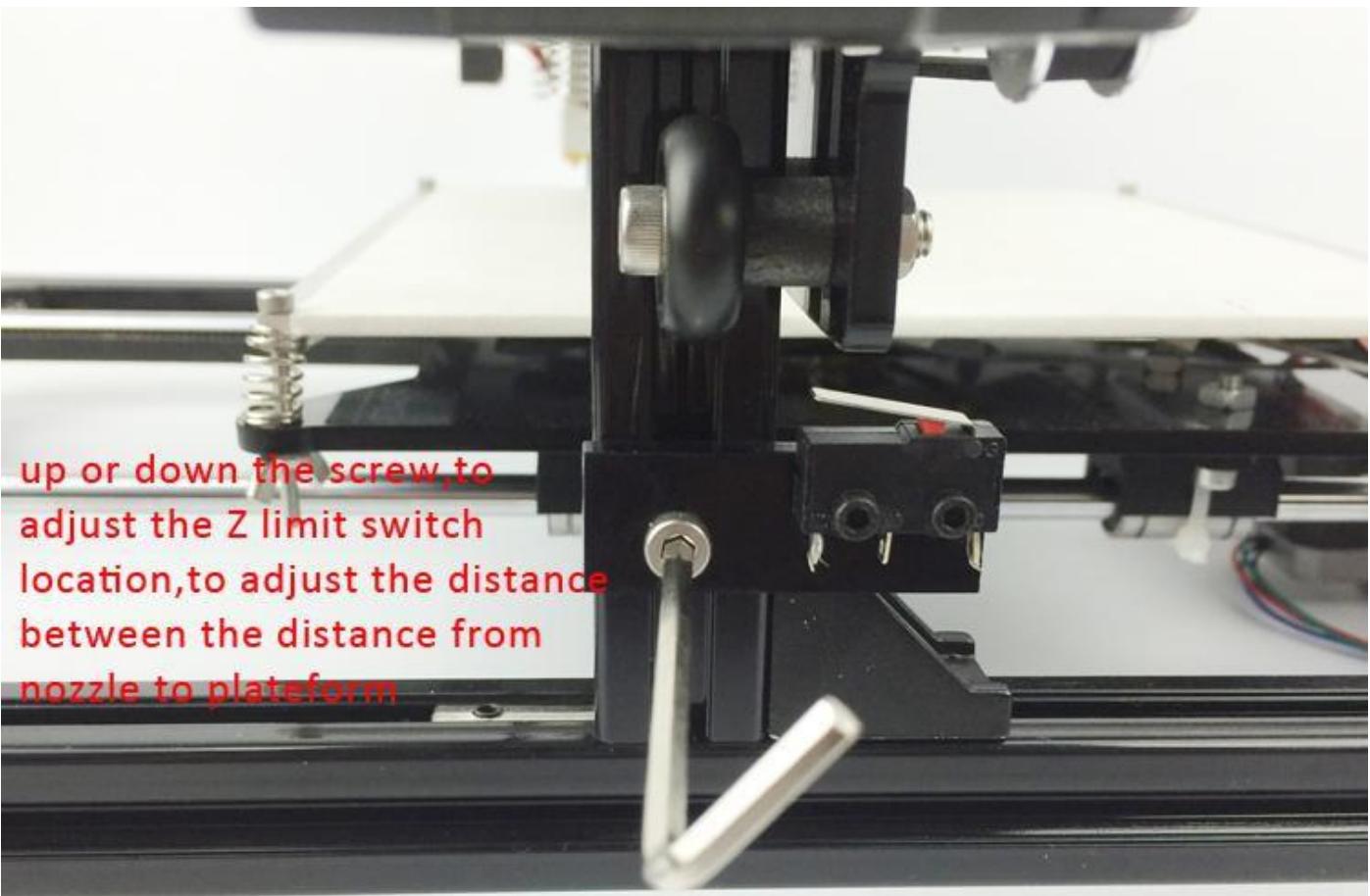
1,Calibration

Select 4 points, the 4 points coordinate is (30 30) , (180 30),(180 180),(30 180). Ensure the nozzle to bed distance,this distance could go through one paper.as the pic show.

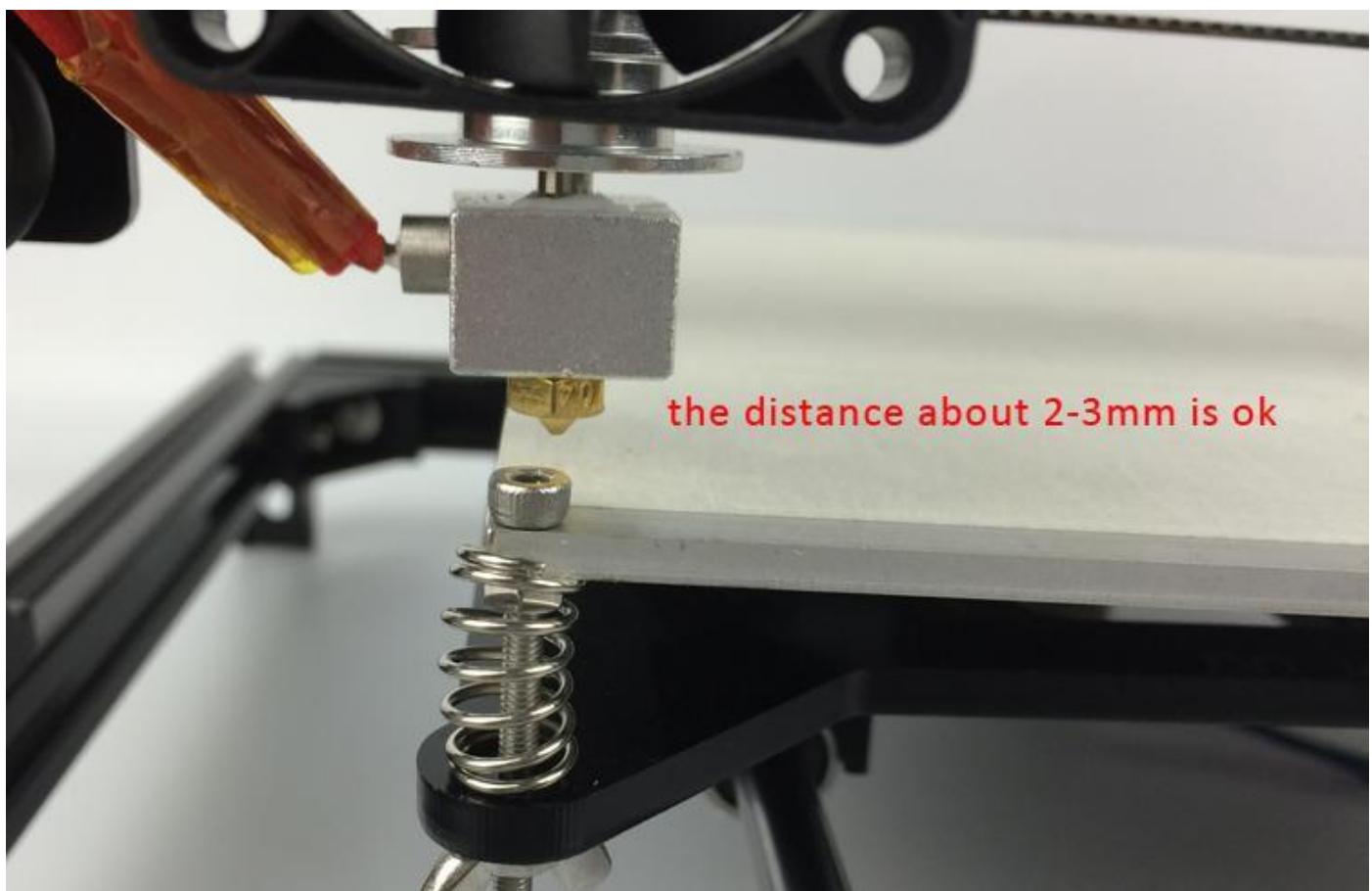




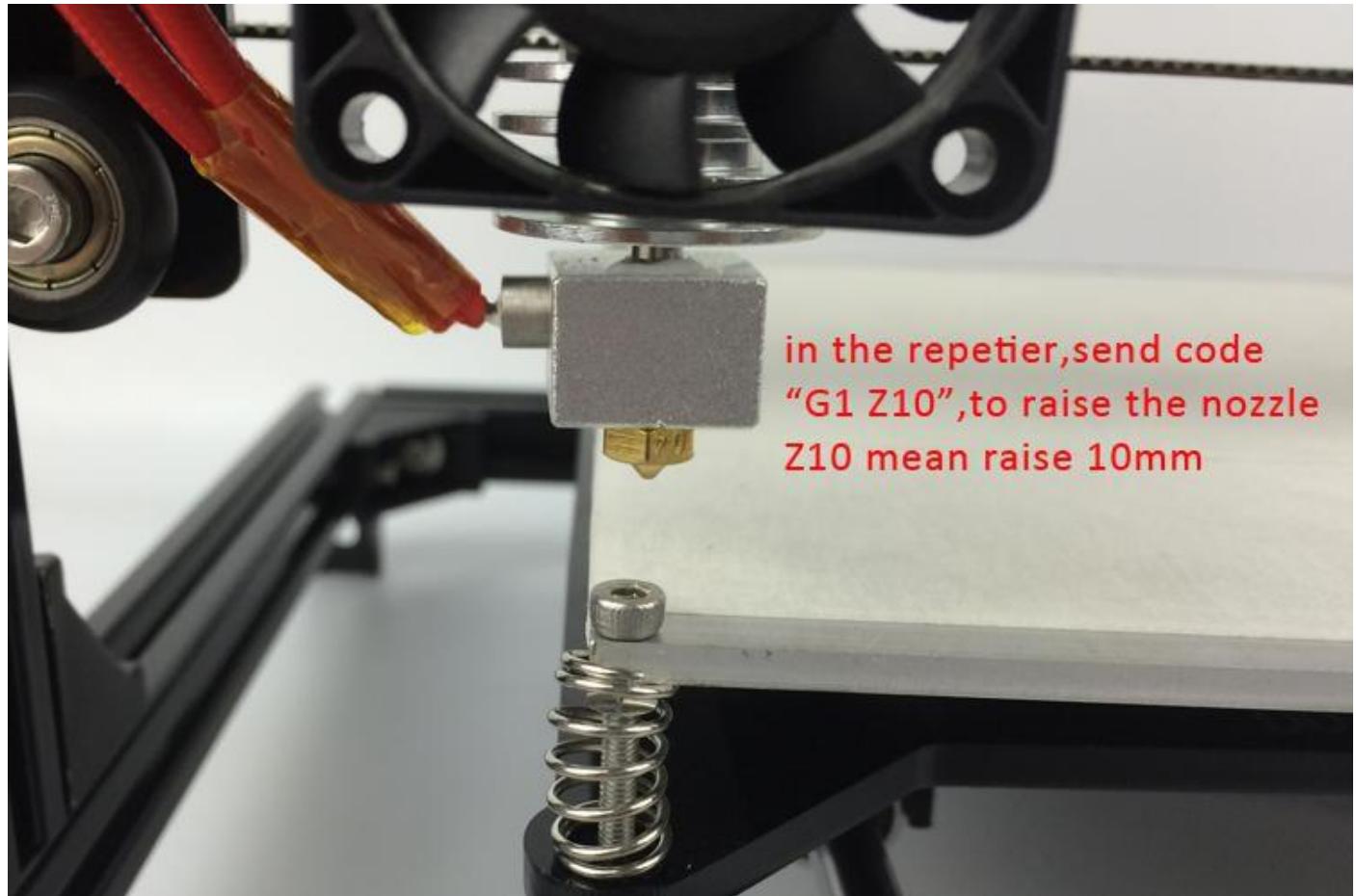




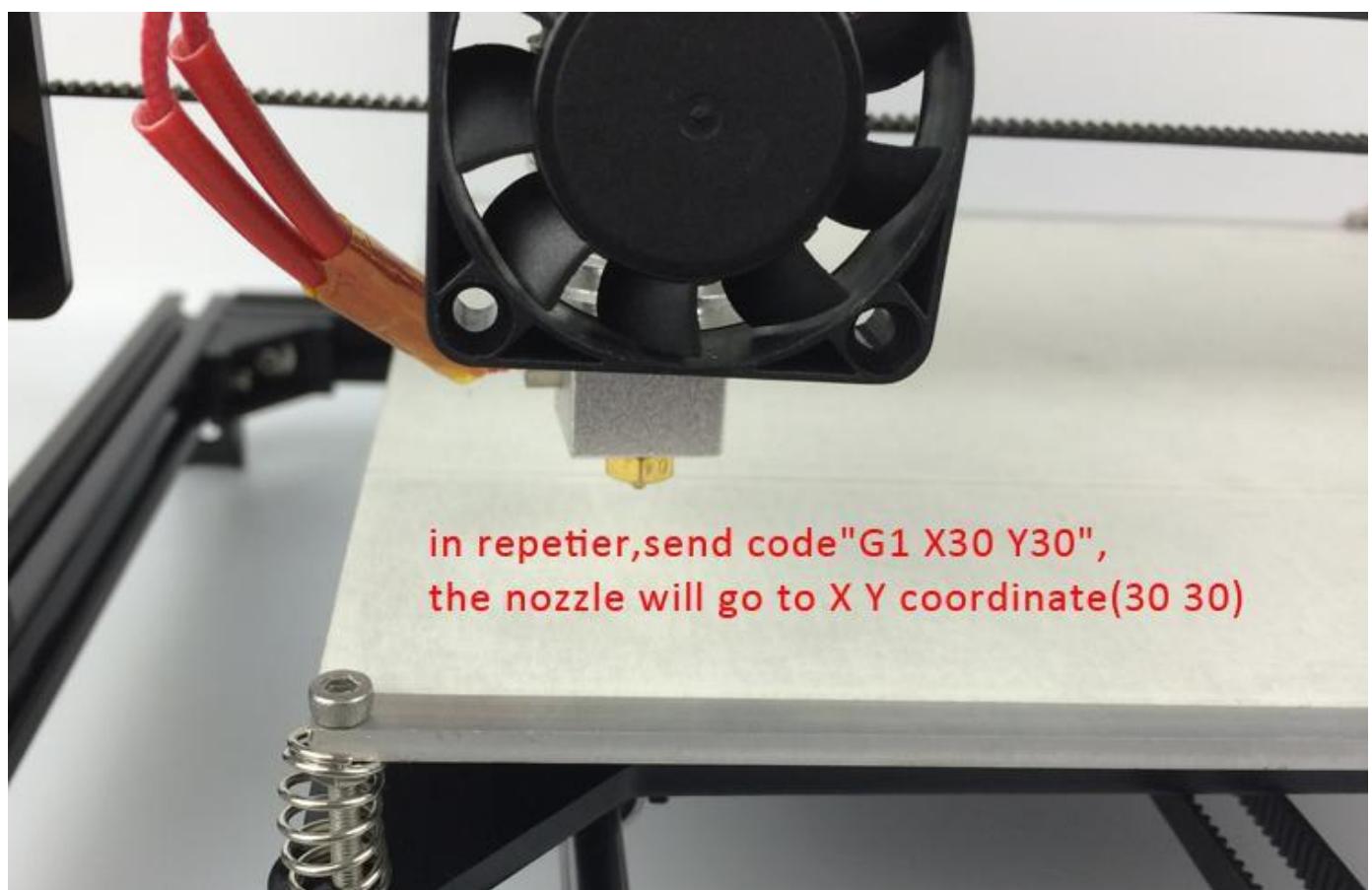
up or down the screw,to
adjust the Z limit switch
location,to adjust the distance
between the distance from
nozzle to platform



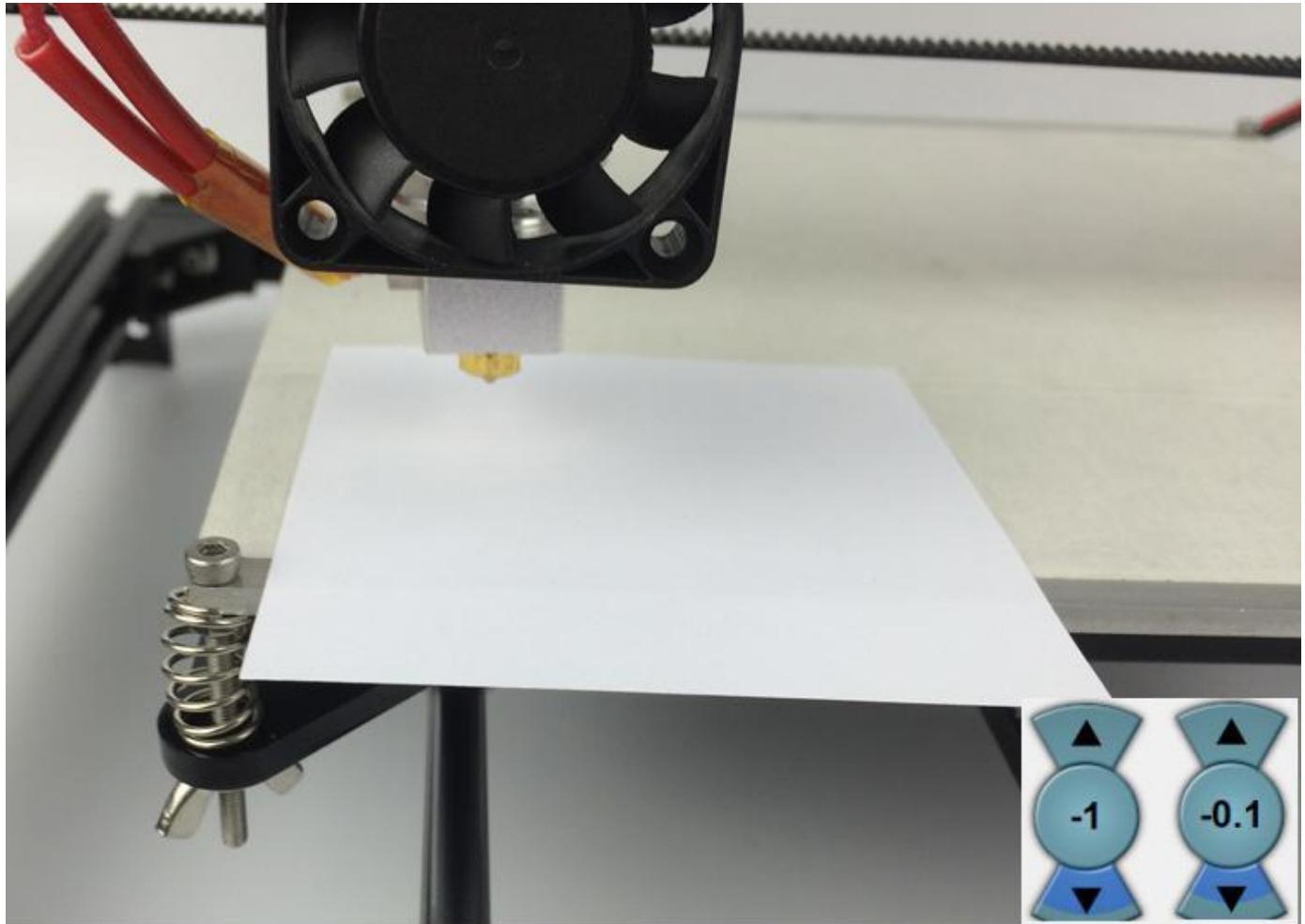
the distance about 2-3mm is ok



in the repetier,send code
“G1 Z10”,to raise the nozzle
Z10 mean raise 10mm



in repetier,send code "G1 X30 Y30",
the nozzle will go to X Y coordinate(30 30)

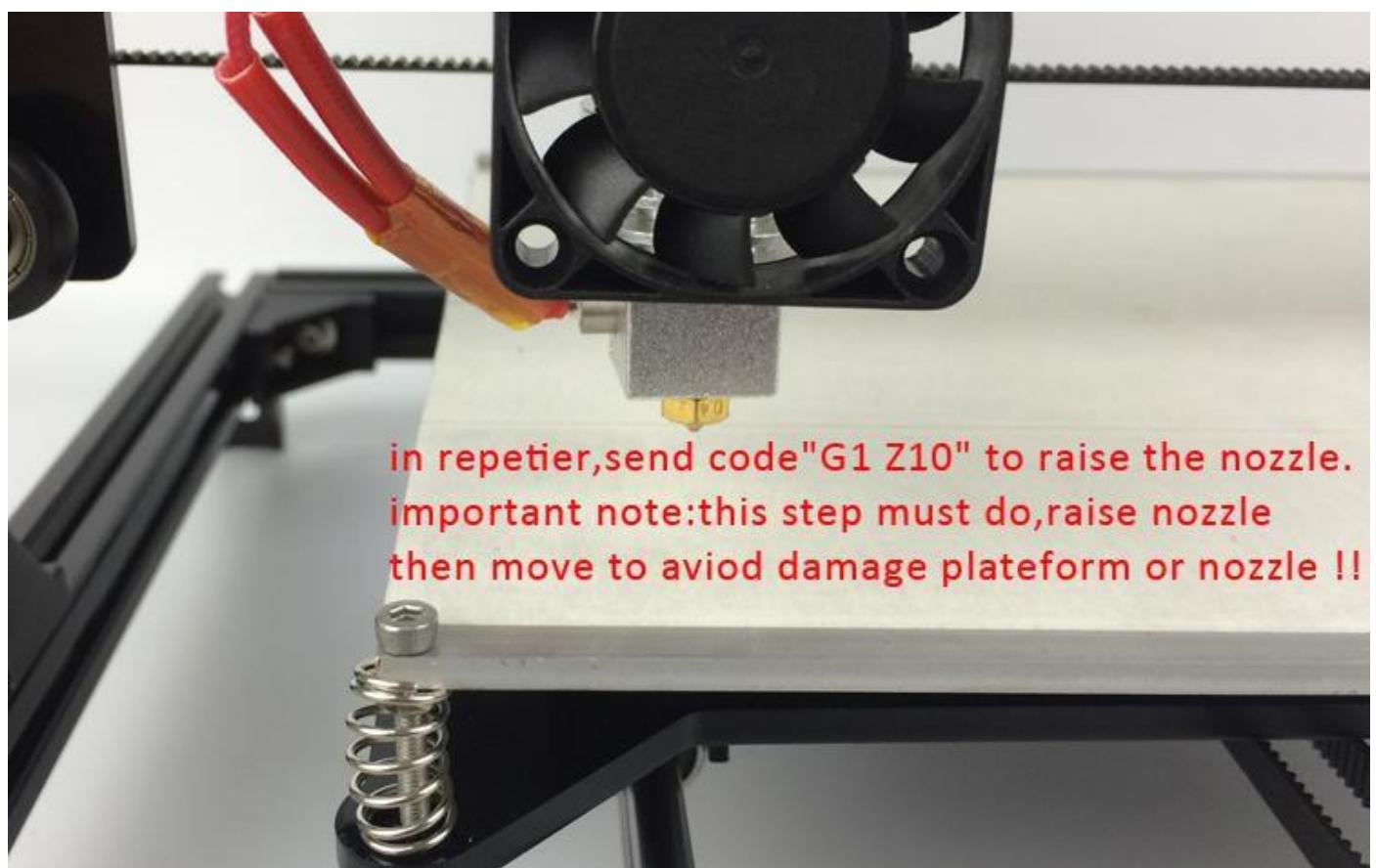
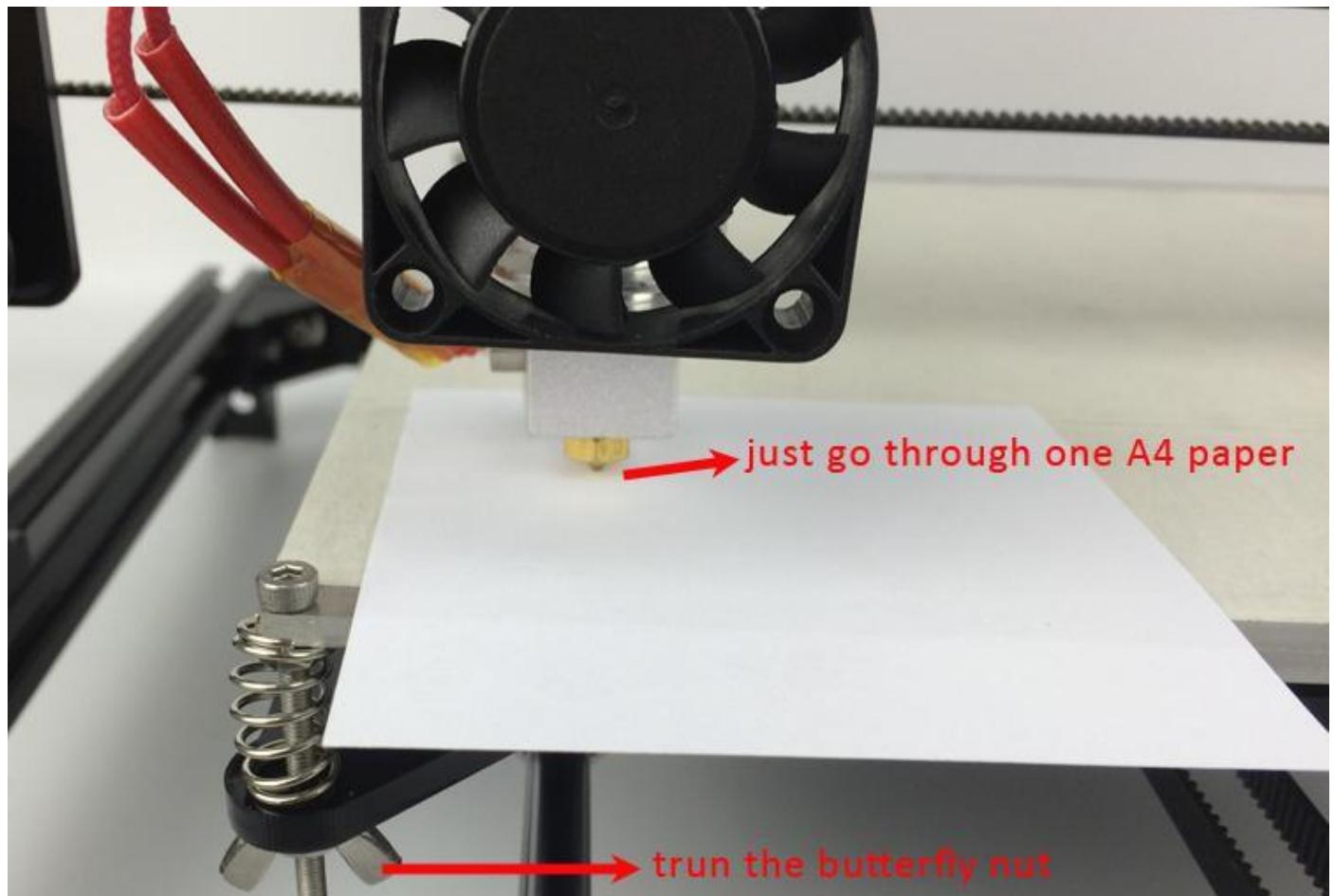


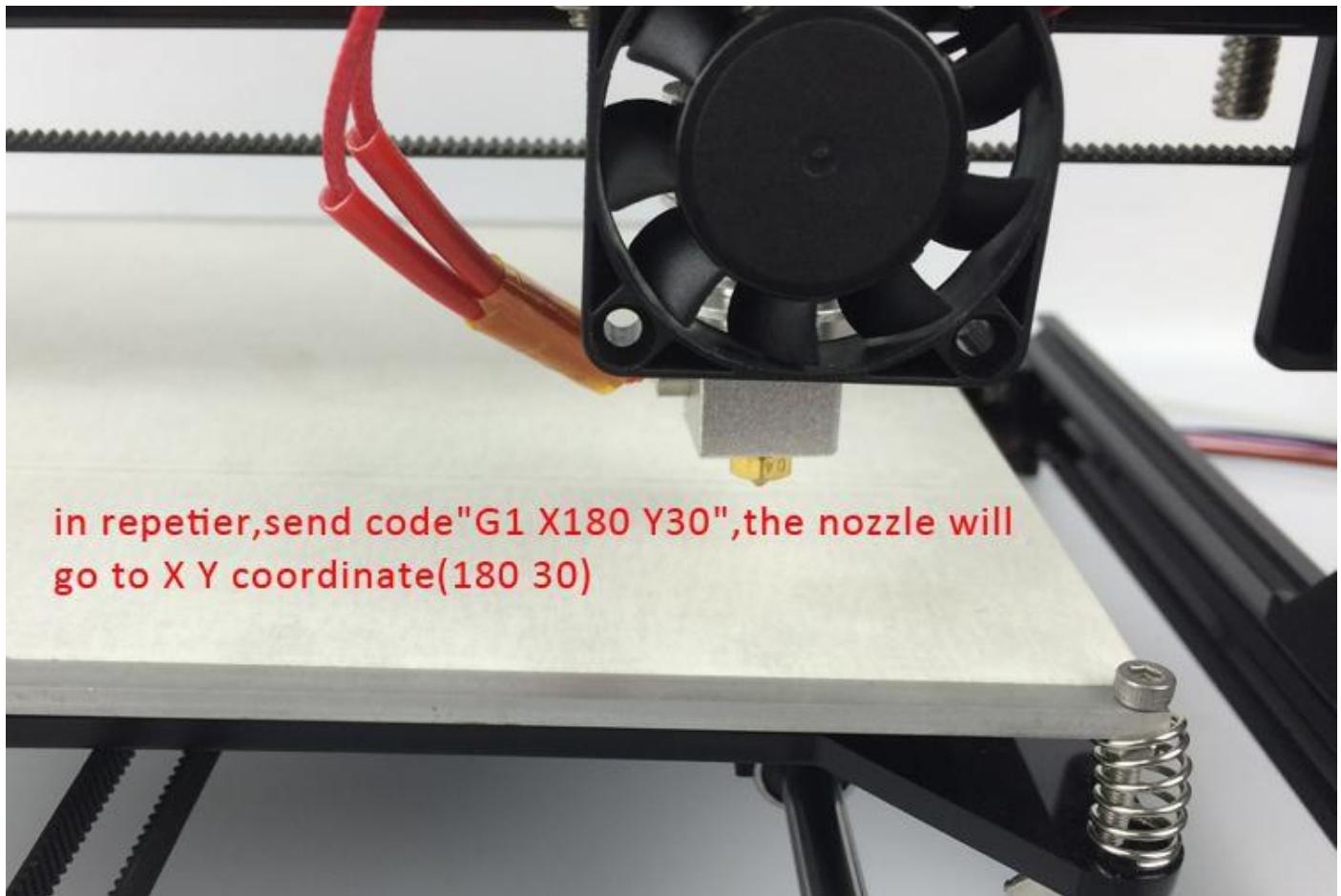
1,put one paper one the platform

2,lower nozzle height via repetier,1mm/step, or 0.1mm/step

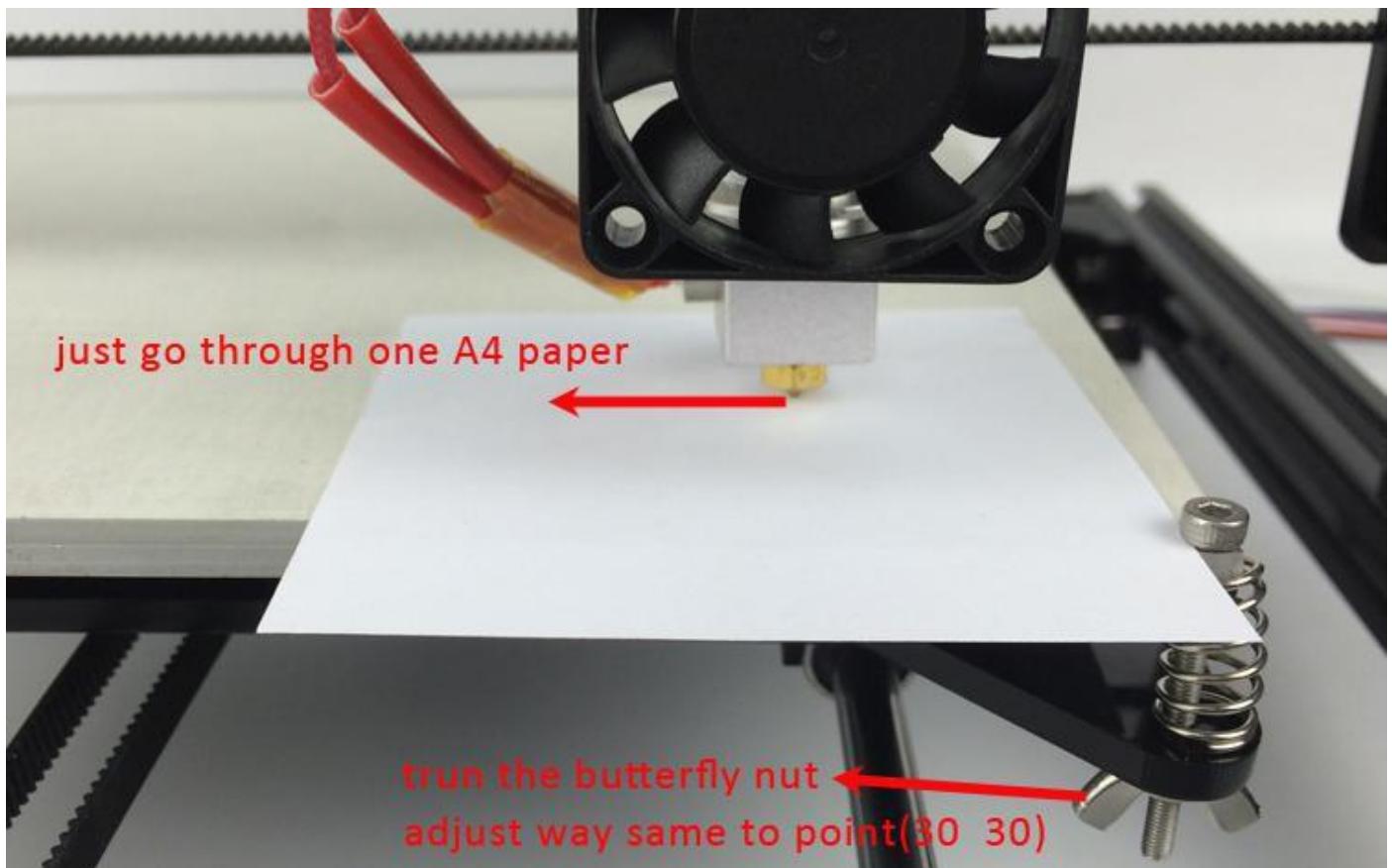
3,when nozzle reach lowest(Z coordinate is 0) still not reach the platform,adjust the butterfly nut to raise the platform, ensure one paper just can go through the distance between the nozzle and platform.this point is ok.

5,if the nozzle has no reach lowest(Z coordinate is 0),the nozzle has hited the platform.adjust the butterfly nut to lower the platform,then lower nozzle height to lowest ,ensure one paper just can go through the distance between the nozzle and platform.this point is ok.

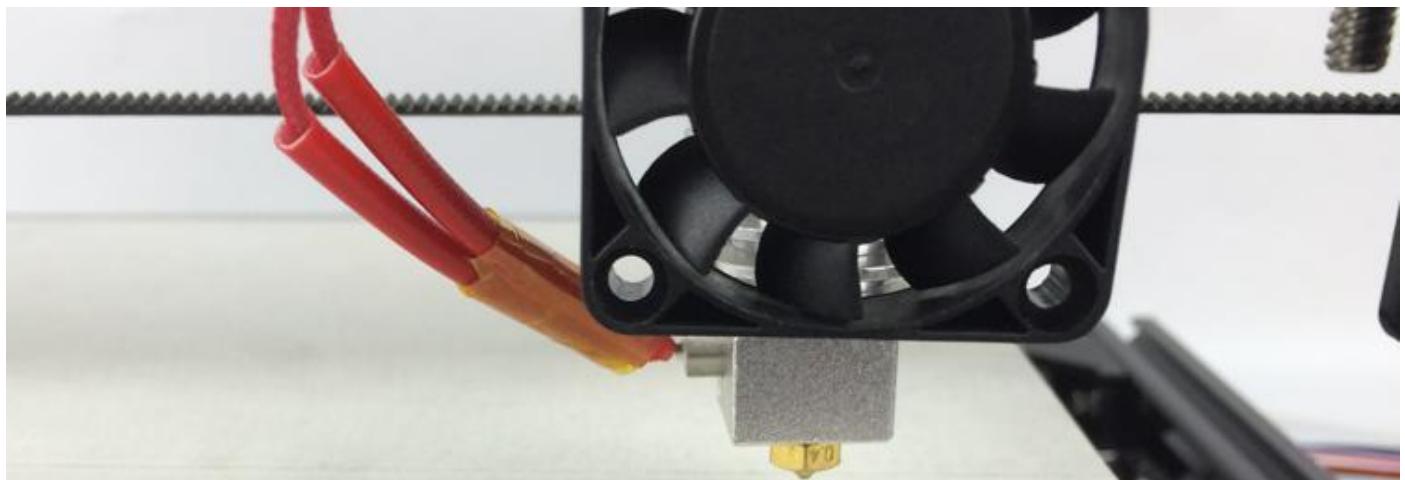




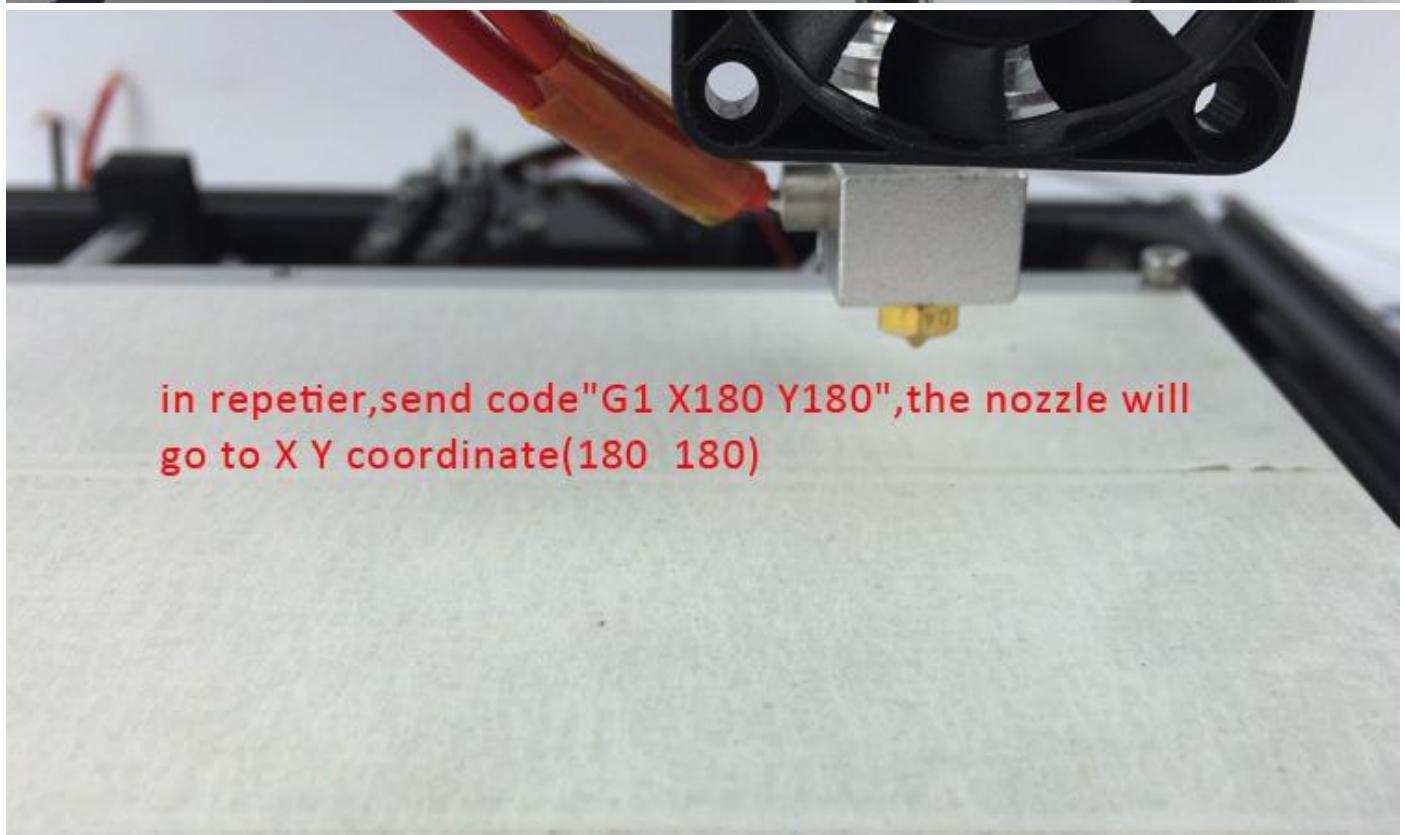
in repetier,send code"G1 X180 Y30",the nozzle will go to X Y coordinate(180 30)



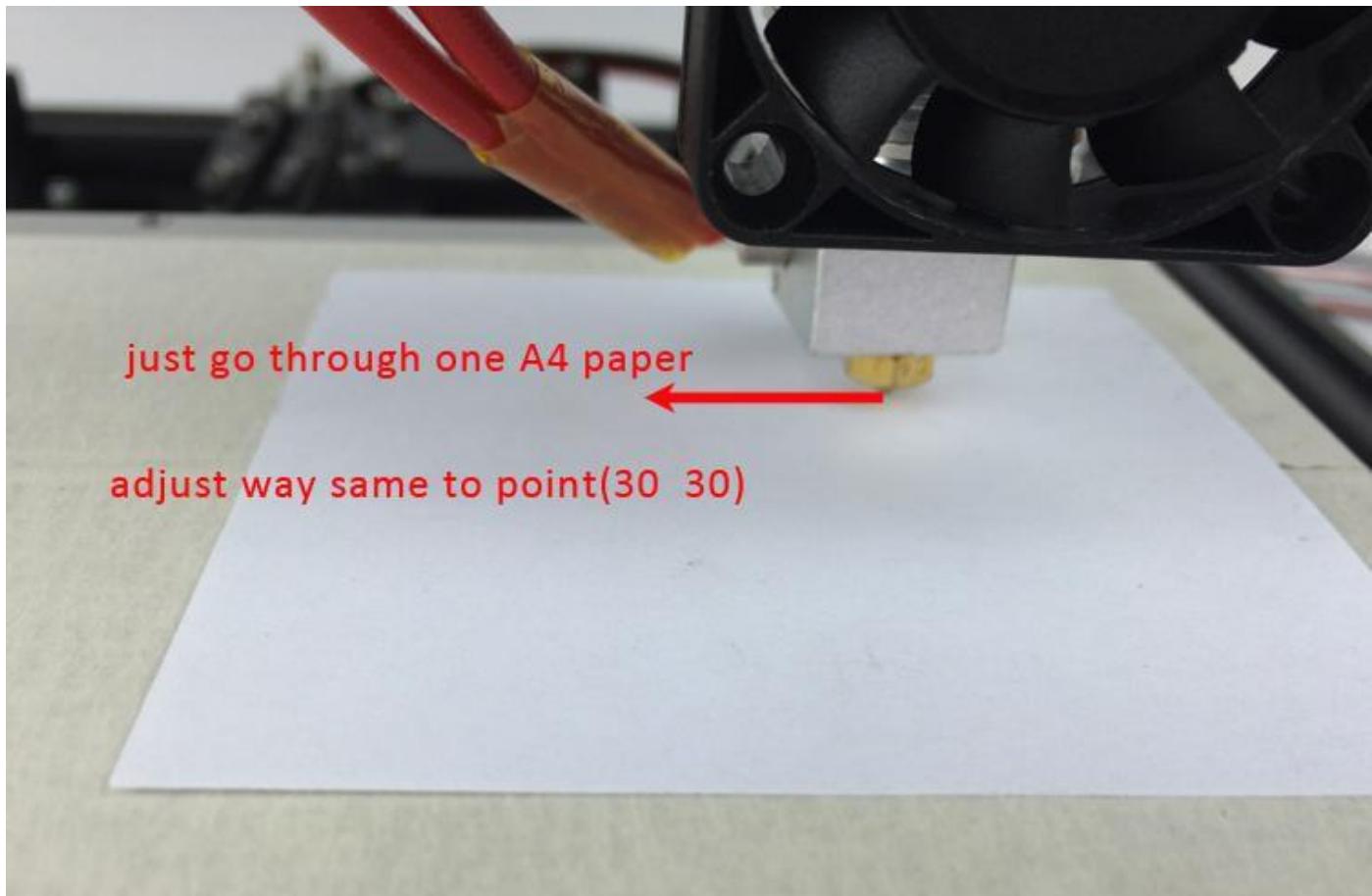
turn the butterfly nut
adjust way same to point(30 30)



in repetier,send code"G1 Z10" to raise the nozzle 10mm

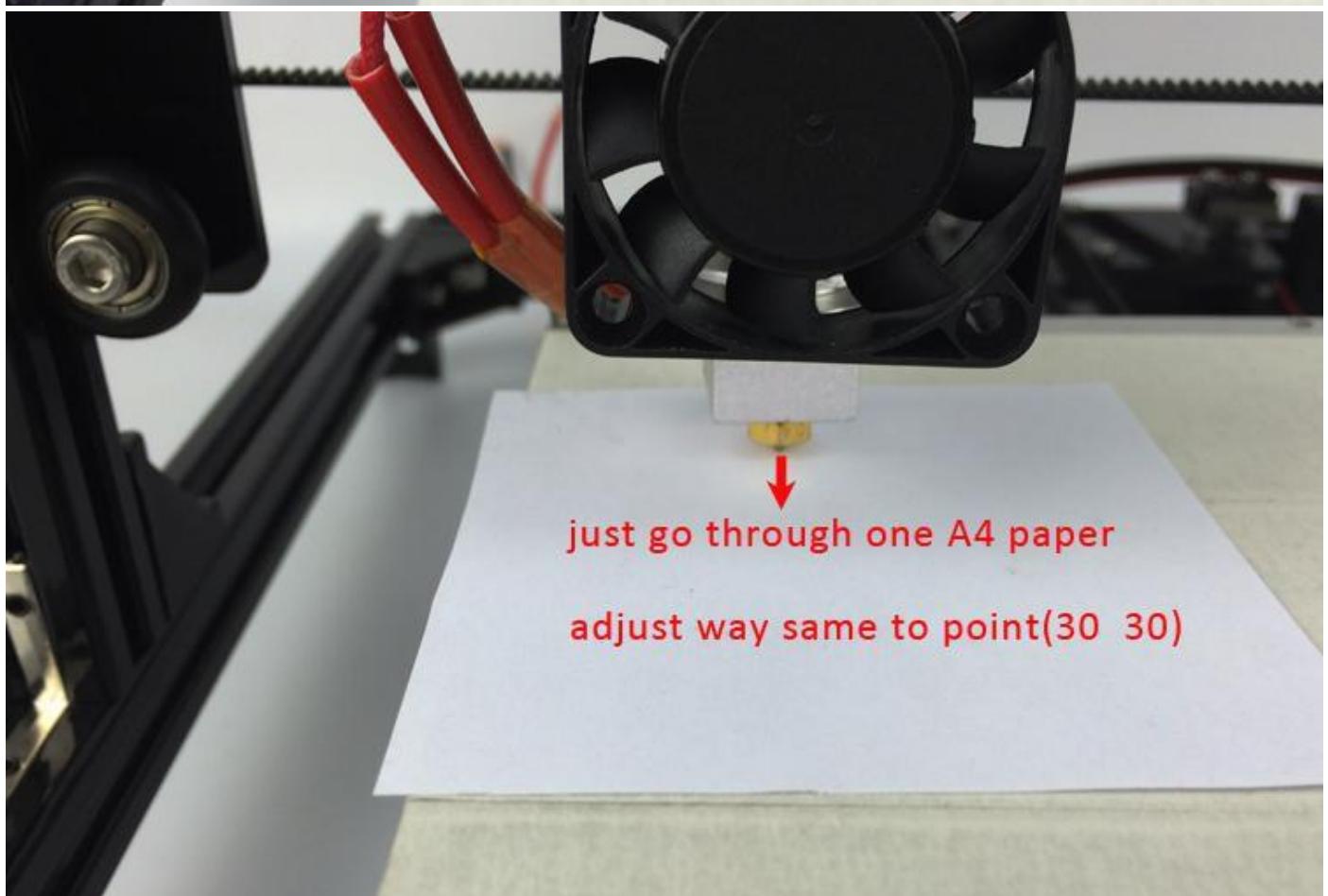


in repetier,send code"G1 X180 Y180",the nozzle will go to X Y coordinate(180 180)



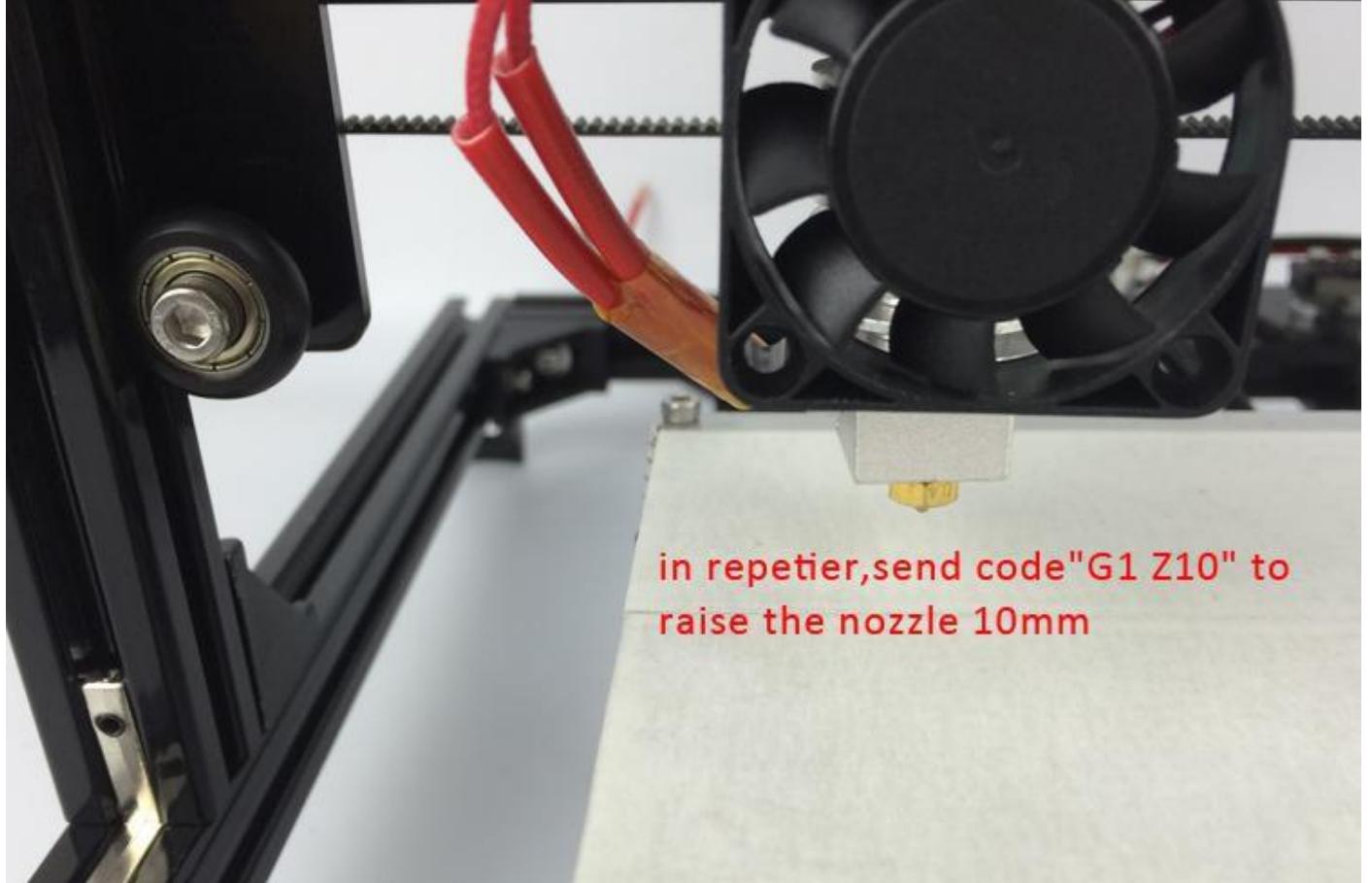


in repetier, send code "G1 X30 Y180", the nozzle will go to X Y coordinate(30 180)

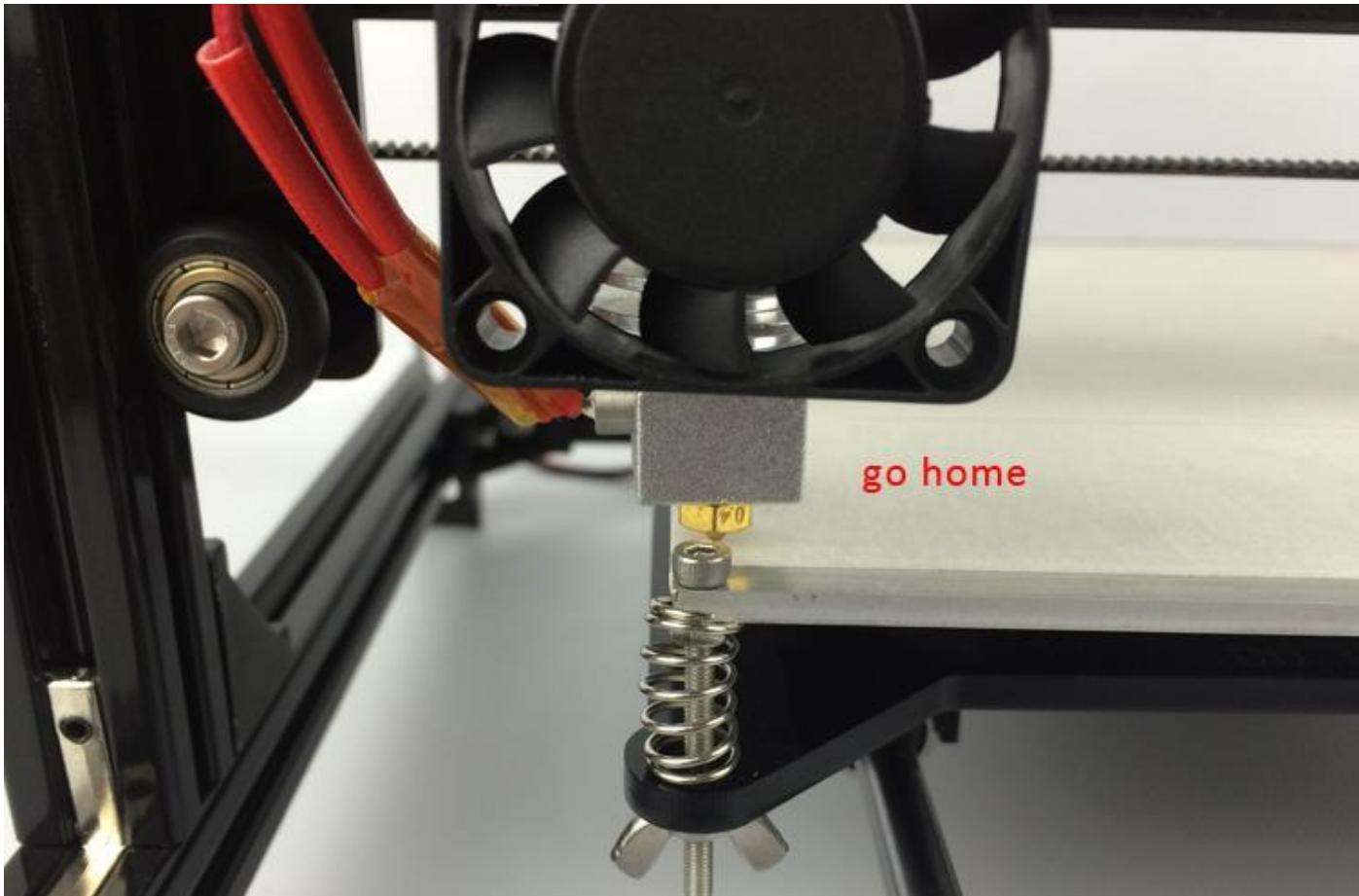


just go through one A4 paper

adjust way same to point(30 30)



in repetier,send code "G1 Z10" to raise the nozzle 10mm



go home