

Dear Geeetech users,

If your printer does not work properly, please follow the following steps to repair it:

1. If the printer is unable to work, please refer to the corresponding section of this manual to deal with according to the fault phenomenon.
2. If this maintenance manual fails to solve your problem, please contact the after-sales staff and describe the problem in short and precise terms.

Thank you for choosing Geeetech products!



[Important] Please read the manual carefully before using the printer.



Please visit website: <https://www.geeetech.com/> for more product information.



Technical support email: <https://www.geeetech.com/contactus.html>.



Scan the following QR code to join the Facebook Group:



Mizar M Printing Guide

I. Slicing Software Guide	4
1. Cura	4
2. EasyPrint	8
3. Prusa Slicer	12
4. Simplify 3D	18
II. Printing Problem Guide	23
1. Blocked	23
1.1 Unblock with a needle.....	23
1.2 Push old filament through.....	24
1.3 Dismantle and rebuild the hot end.....	24
2. Grinding Filament	26
2.1 Increase the nozzle temperature by 5-10C°.....	27
2.2 Clean the hot end.....	27
2.3 Decrease the print speed.....	27
3. Not Sticking to the Bed	27
3.1 Level the print bed.....	27
3.2 Adjust the Z-offset.....	27
3.3 Clean the print platform.....	28
3.4 Apply the build plate adhesion.....	28
4. Warping	29
4.1 Use a heated print platform.....	29
4.2 Apply an adhesive to the print bed.....	29
4.3 Add build plate adhesion.....	29
5. Layer Shift	29
5.1 Check that the printer is on a stable base.....	29
5.2 Check the print bed is secure.....	30
5.3 Decrease print speed.....	30
5.4 Check the belts.....	30
5.5 Check the rods.....	32
6. Vibrations and Ringing	32
6.1 Reduce vibration.....	32
6.2 Make sure everything is tight.....	32
6.3 Add oil.....	32
6.4 Reduce the print speed.....	33
7. Under-Extrusion	33
7.1 Check the hot end for clogging.....	33
7.2 Decrease the print speed.....	33
7.3 Extruder Problem.....	33
8. Print Looks Melt and Deformed	34
8.1 Poor Cooling.....	34
8.2 Print temperature is too high.....	34
9. String	34

9.1	Enable retraction	34
9.2	Decrease print temperature	35
9.3	The inner and outer wall speed difference.....	35
10.	Zits and Blobs.....	35
10.1	Lower temperature & speed	36
10.2	Use coasting.....	36
10.3	Fix incorrect extrusion.....	36
III.	Common Hardware Issues	37
1.	Hot End.....	37
1.1	Hot end is shaking	37
1.2	Heating failed	37
2.	Extruder.....	38
2.1	Clicking or slipping	38
2.2	Grinding filament.....	38
3.	Hotbed.....	38
3.1	Heating failed	38
3.2	Shaking	39
3.3	Print is stuck to Hotbed.....	39
3.4	Automatic heating	39
4.	Belt.....	40
4.1	Install the belt tensioner	40
5.	Data Reading	40
5.1	Unable to read TF card.....	40
5.2	Unable to connect to the computer	41
6.	Other maintenance.....	41
6.1	Z-axis leadscrews.....	41
6.2	Nozzle	41
6.3	Stepper Motor.....	42

I. Slicing Software Guide

1. Cura



Mizar M Cura Configuration.rar

1.1 The Mizar M configuration applies to Cura 4.13.1 and Cura 5.0/5.1.

1.2 File Path

Cura 5.X

Copy "geeetech_Mizar_M.def.json" to Ultimaker Cura
5.X\share\cura\resources\definitions

Copy "geeetech_Mizar_M_extruder0.def.json" and
"geeetech_Mizar_M_extruder1.def.json" to Ultimaker Cura
5.X\share\cura\resources\extruders

Cura 4.X

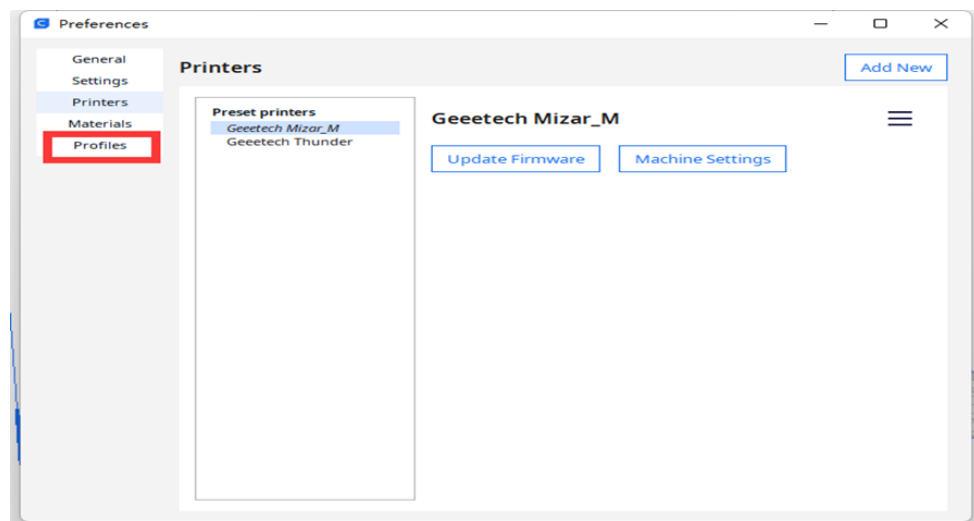
Copy "geeetech_Mizar_M.def.json" to Ultimaker Cura
4.X\resources\definitions

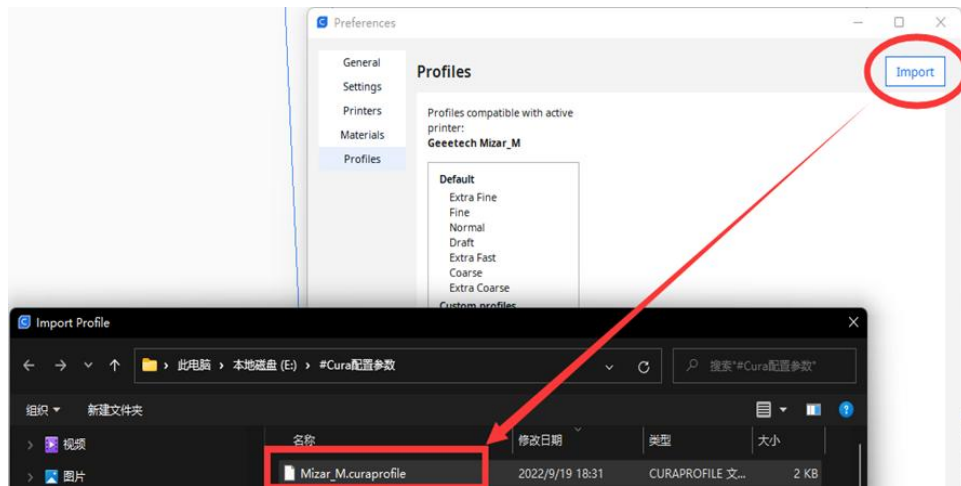
Copy "geeetech_Mizar_M_extruder0.def.json" and
"geeetech_Mizar_M_extruder1.def.json" to Ultimaker Cura
4.X\resources\extruders

1.3 Restart the Cura software, add the printer Mizar M and import the profile
Mizar_M.curaprofile



Mizar_M.curaprofile.rar

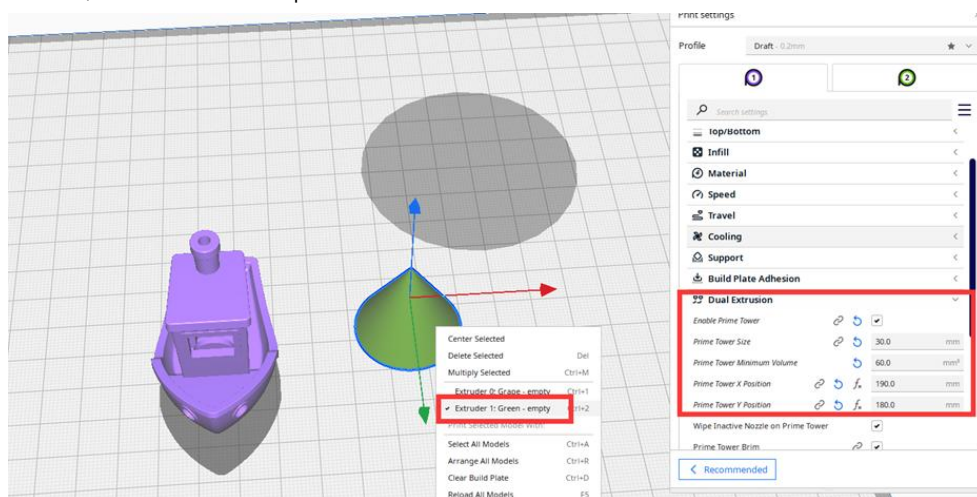




1.4 Separate-Color Printing

Applied to both Mix-Color module and Separate-Color module

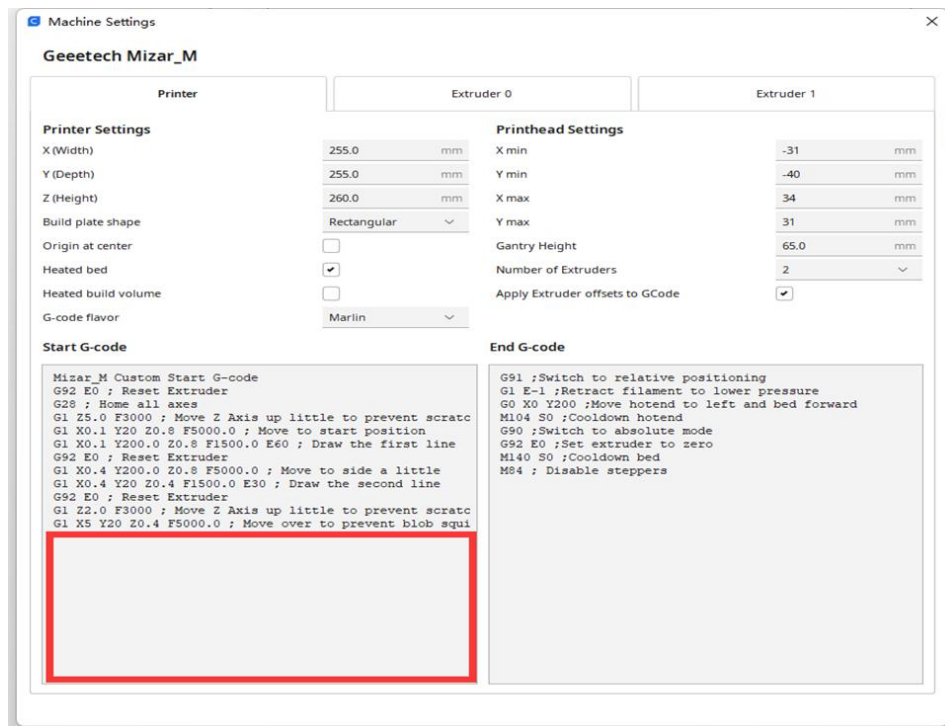
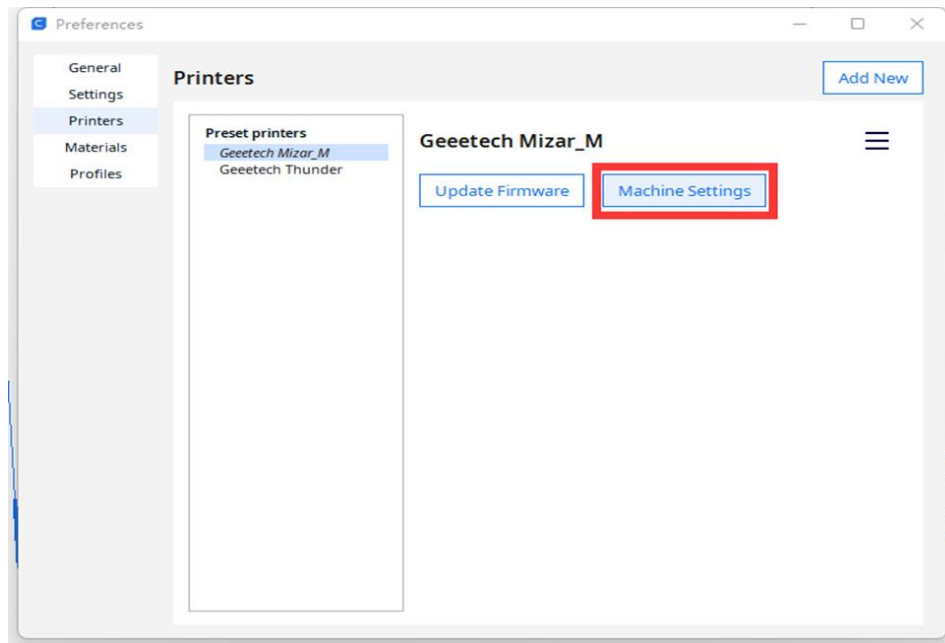
Import 2 or more models, respectively set the extrusion head used by each model, set the size and position of the Prime Tower.



1.5 Mix-Color Printing

Make sure to install the Mix-Color module in advance

According to the different effect of the model you want, enter Mizar M printer management settings and edit different contents under the content of Start G-code. The details are as follows. By the way, Mix-Color printing can disable the Prime Tower setting option.



a) Mix-color printing in a fixed proportion (Mix-Color Module)

M163 ;Set a mix factor

M164 ;Save mix factor

For example, E0/E1 extruded 100%, 0 or 0, 100%, respectively

M163 S0 P1

M163 S1 P0

M164 S0 ;T0=100/0

;Then we create the first Tools T0, which will be red.

M163 S0 P1; 100% Red

M163 S1 P0; 0% Green

;Now save this color scheme in our Tools T0 we use the command:

;M164 S (index), where index is our Tools, and then click:

M164 S0; Store color mixing in tools T0=100/0/0

Now we create the Tools Green, And keep it on the Tools T1:

M163 S0 P0; 0% Red

M163 S1 P1; 100% Green

M164 S1; Store color Mixing in Tools T1

b) Mix-color printing in gradient proportions (Mix-Color Module)

Gradient for a tall vase : In this example no tool index is assigned to the Gradient. So the Gradient is enabled using M166 S1 and disabled with M166 S0.

```
M166 A0 Z250 I0 J1 S1
```

Gradient as a Virtual Tool : In this example the Gradient has been given a tool alias of 3. So the Gradient is selected with T3, and all other Tn commands will select a static mix.

```
M166 A0 Z250 I0 J1 T3
T3
```

To use the static mix stored in T0...

```
M166 T
T0
```

M163 S0 P1

M163 S1 P0

M164 S0 ;T0=100/0

M163 S0 P0

M163 S1 P1

M164 S1 ;T1=0/100

M163 S0 P0.5

M163 S1 P0.5

M164 S2 ;T2=50/50

M166 A0 Z100 I0 J1 S1 ;Gradient Usage Instructions 1

;In this example no tool index is assigned to the Gradient. So the Gradient is enabled using M166 S1 and disabled with M166 S0

M163 S0 P1

M163 S1 P0

M164 S0 ;T0=100/0

M163 S0 P0

M163 S1 P1

```
M164 S1 ;T1=0/100
M163 S0 P0.5
M163 S1 P0.5
M164 S2 ;T2=50/50
M166 A0 Z100 I0 J2 T3 ;Gradient Usage Instructions 2
;In this example the Gradient has been given a tool alias of 3. So the Gradient
is selected with , and all other commands will select a static mix.
```

- c) G-code Instructions
- ;M163 Set a mix factor
 - ;M164 Save mix factor
 - ;M166 Gradient for Marlin 2.0
 - ;A= Start z height
 - ;Z= End z height
 - ;I= Start Virtual Tool
 - ;J= End Virtual Tool
 - ;S= Enable / disable the gradient in manual mode
 - ;T=A tool index to reassign to the gradient. If no index is given, cancel the tool assignment.

Detailed G-code instructions, please refer to the Marlin's official website <https://marlinfw.org/meta/gcode/>

Note: Do not use separate-color module to print mix-color gcode, clogging and damaging Teflon tube.

2. EasyPrint

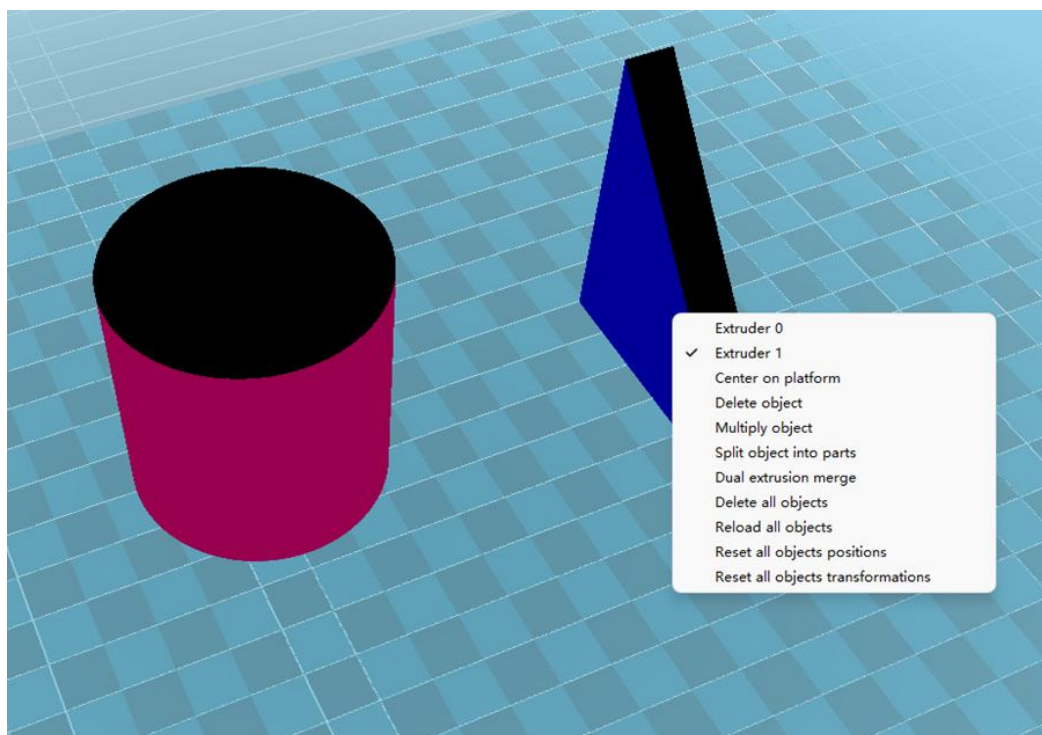
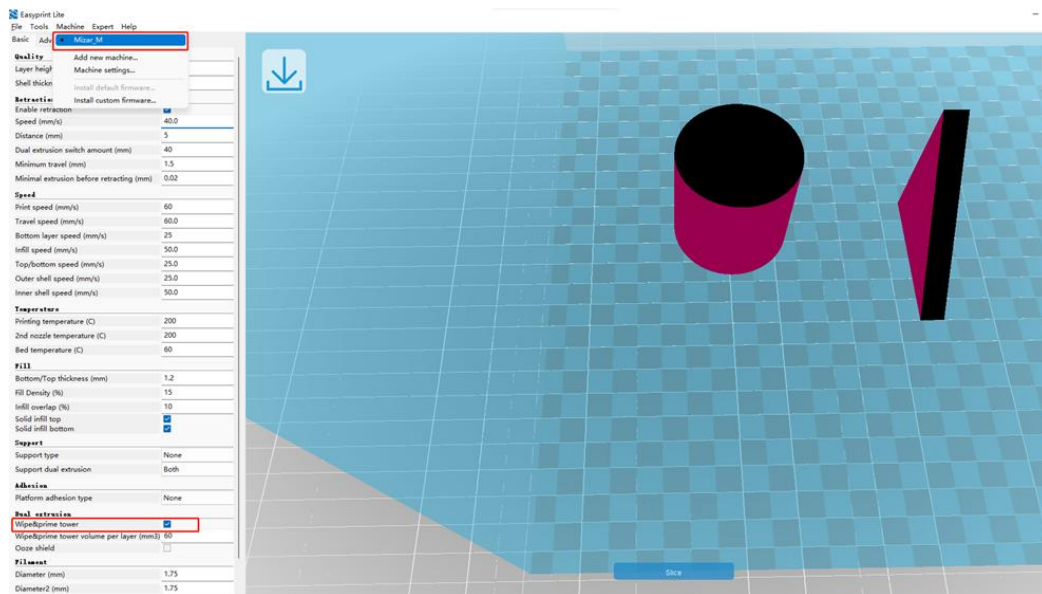


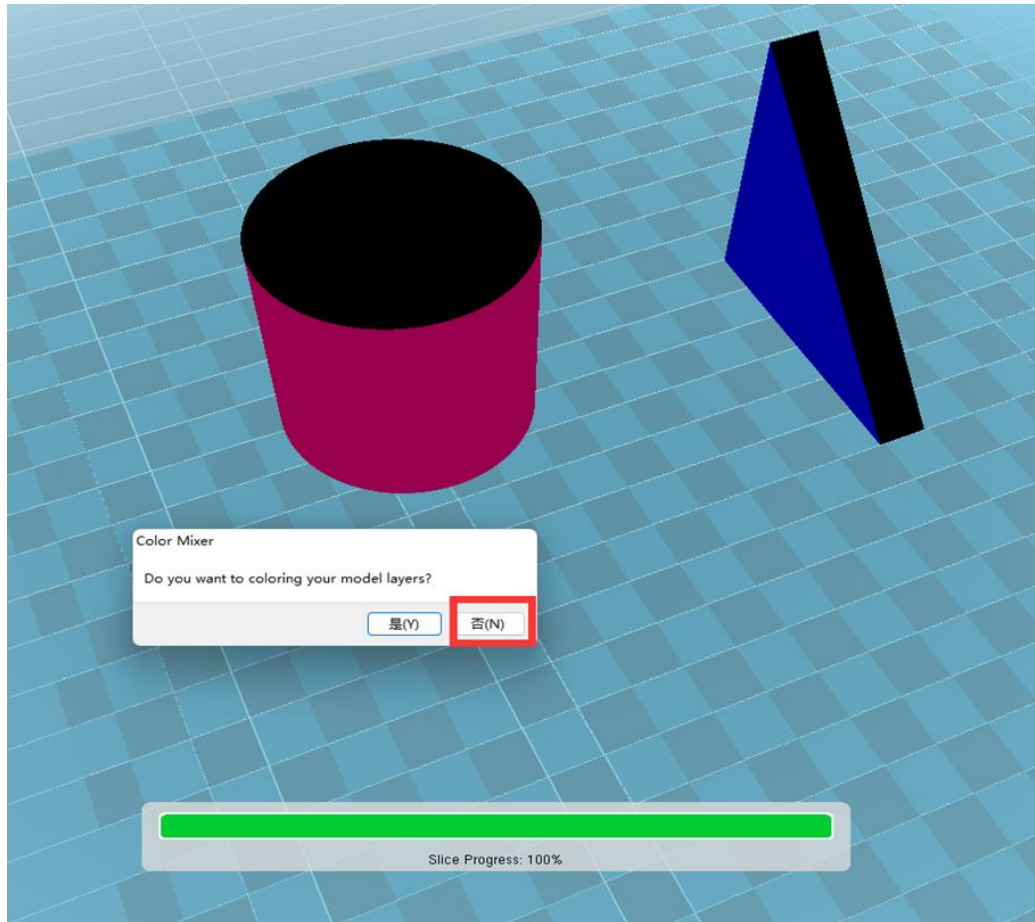
Mizar_M-EasyPrint-config.rar

2.1 Separate-Color Printing

Applied to both Mix-Color module and Separate-Color module

Import 2 or more models, the size of a single model is no more than 200M, add a prime tower, and set the extrusion head for each model. After slicing, select "No" in the prompt box for whether to mix colors.

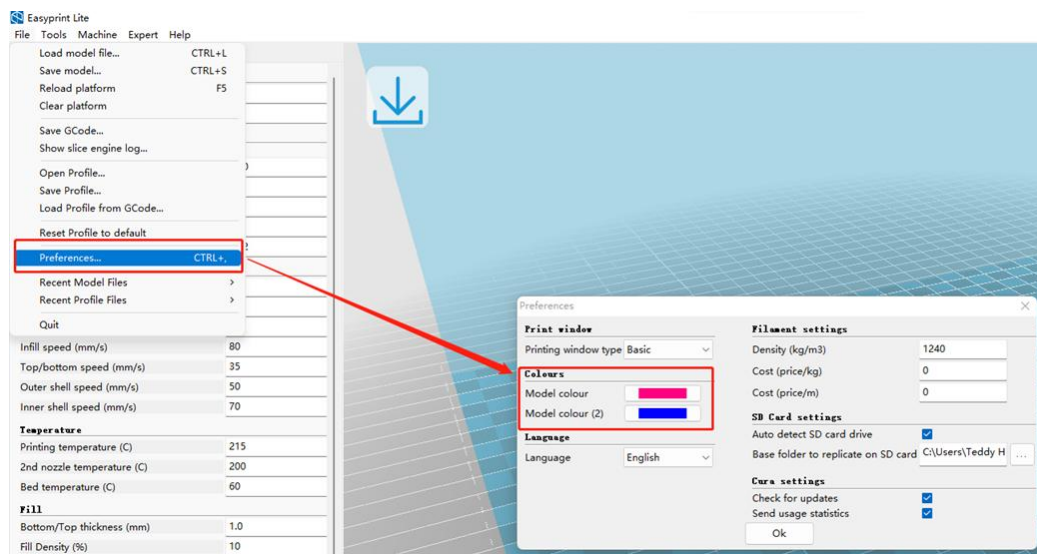


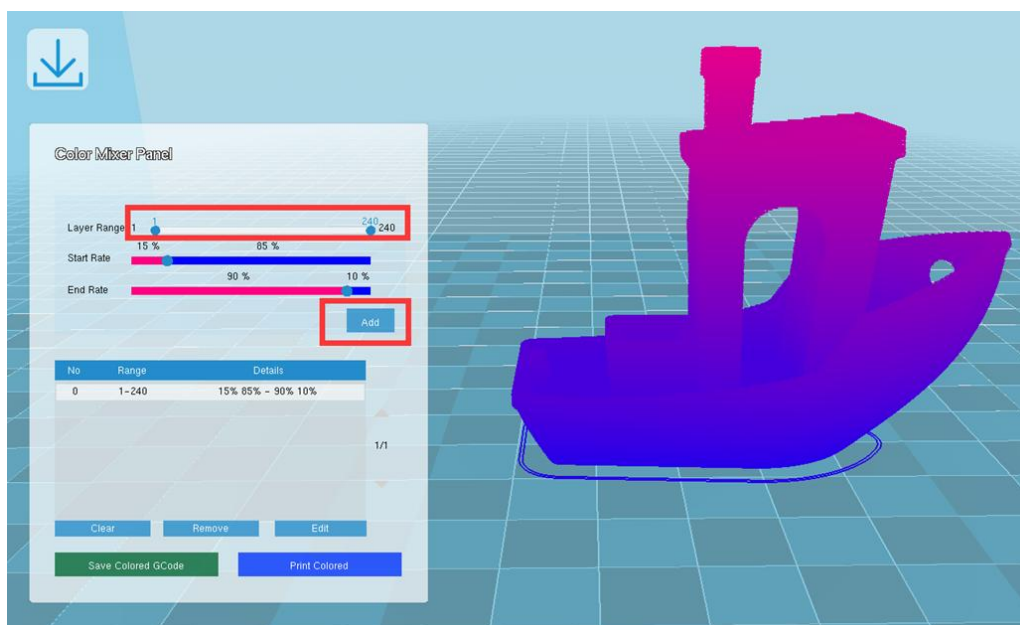
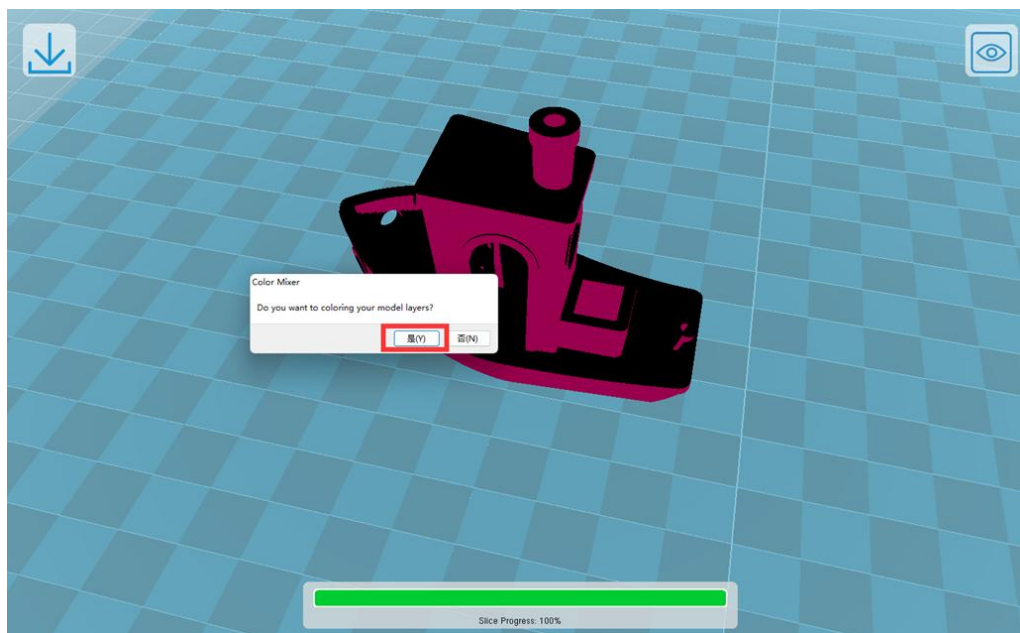


2.2 Mix-Color Printing

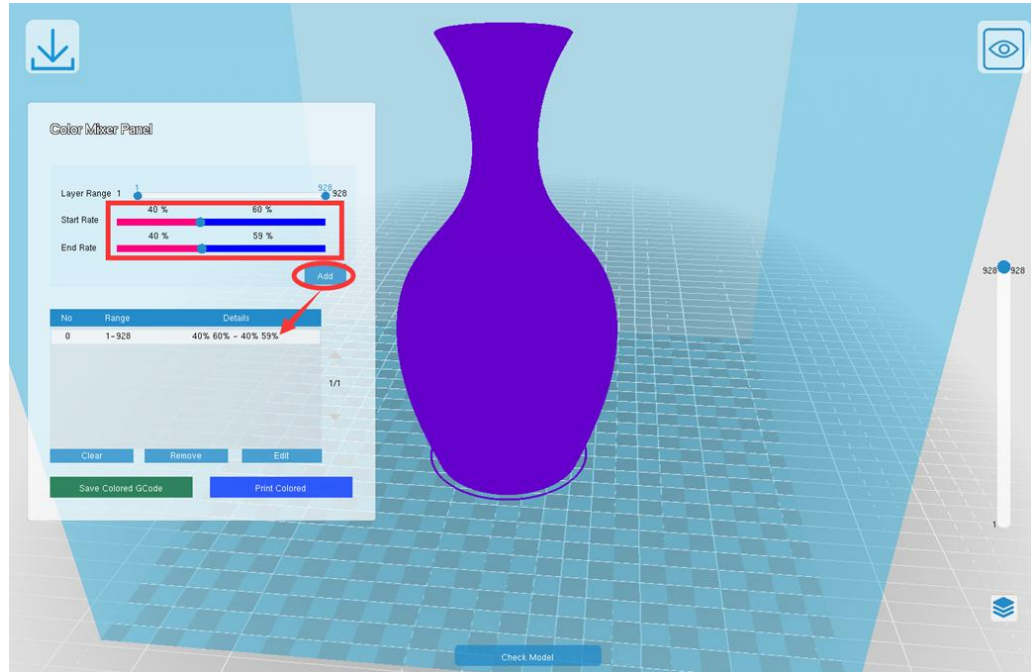
Make sure to install the Mix-Color module in advance

- a) Mix-color printing in gradient proportions (Mix-Color Module)
 - Easy Print adopts visual color mixing effect slice, please set the E0/E1 extruder color in advance, import the model, select the color mixing effect after slicing.





- b) Mix-color printing in a fixed proportion (Mix-Color Module)
Keep the Start Rate and End Rate consistent, and click "Add" to complete the section.



3. Prusa Slicer

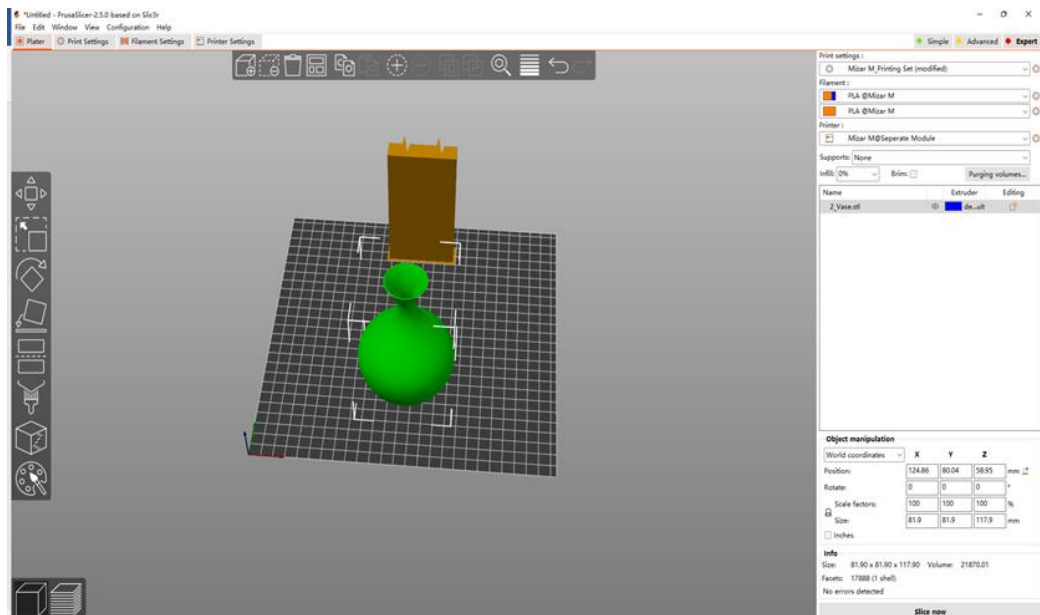
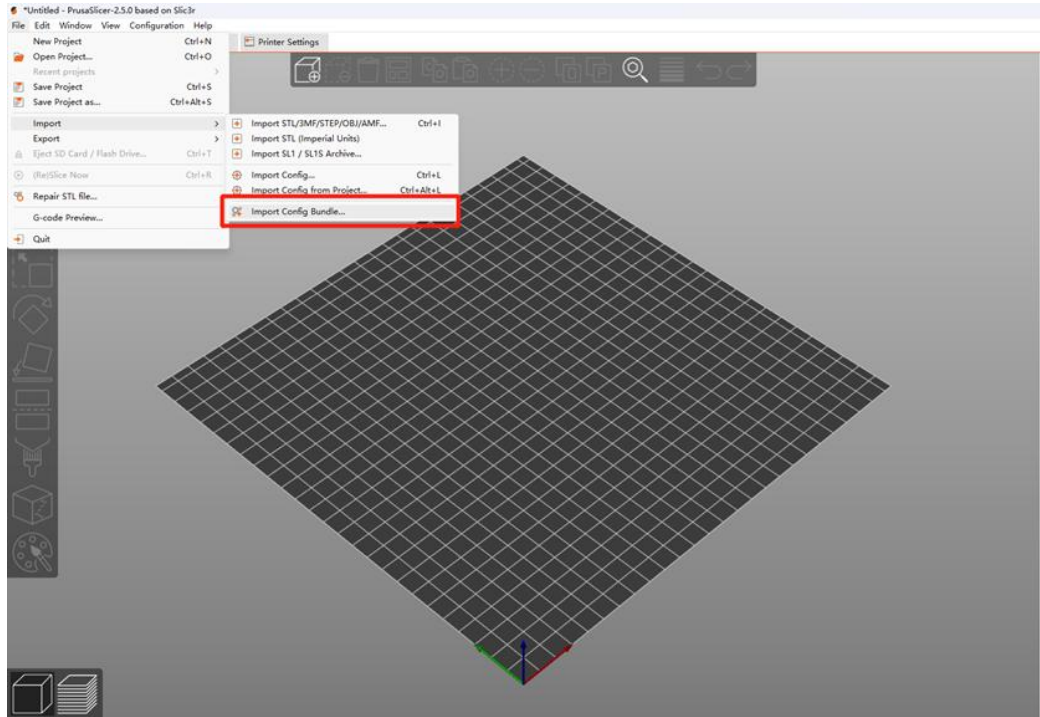


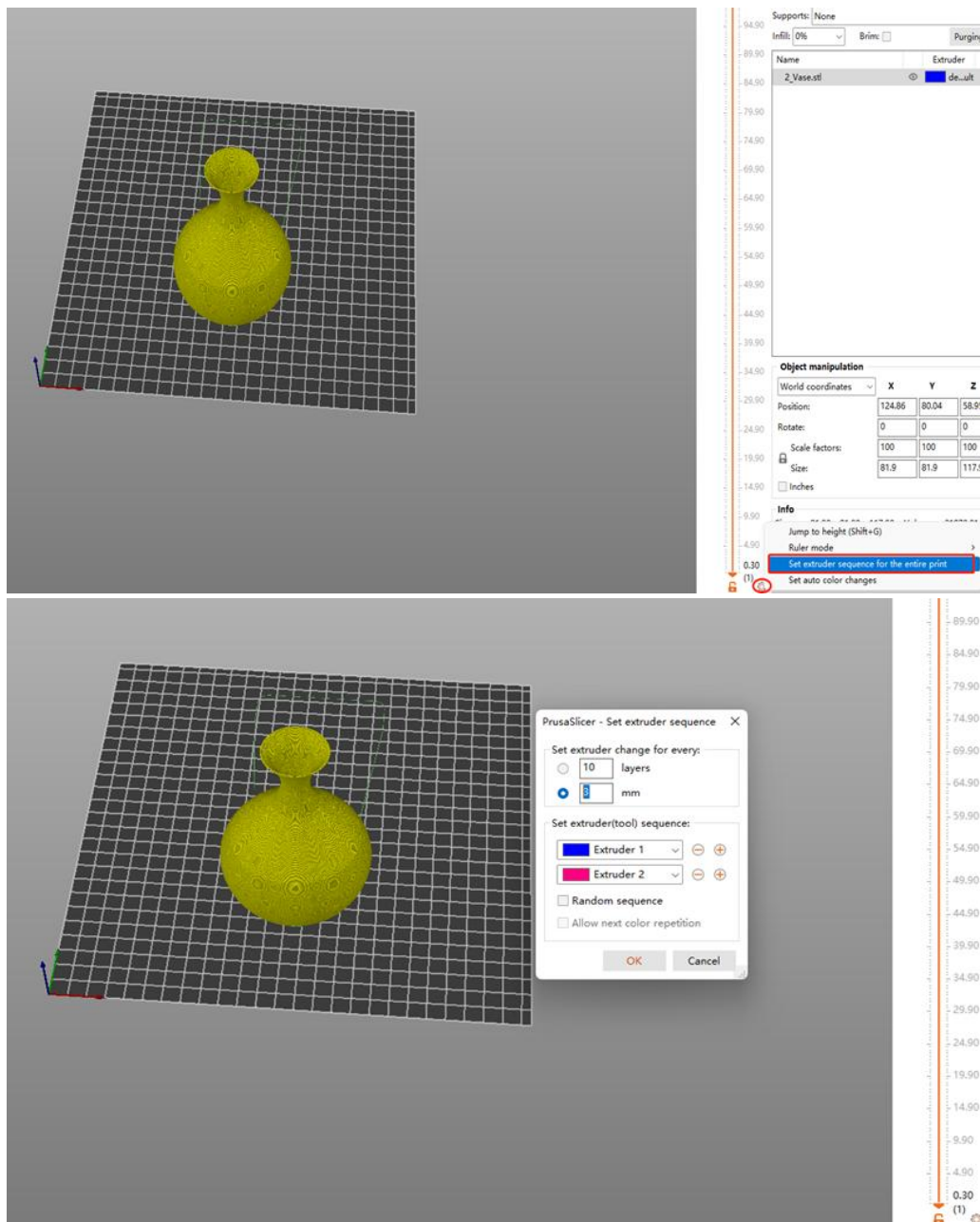
PrusaSlicer_config_bundle for Mizu

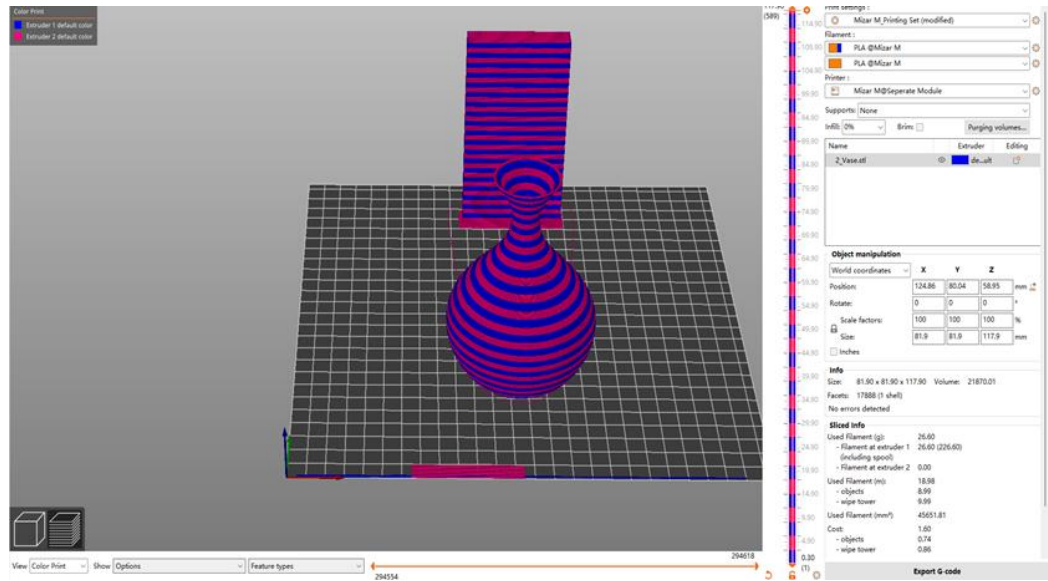
3.1 Separate-Color Printing

Applied to both Mix-Color module and Separate-Color module

- a) PrusaSlicer can print a single model with different height separations. Import the configuration file and the stl model file, set the printing order and height after slicing, and re-slice to preview as shown in the figures.







- b) PrusaSlicer also has the capability to color separate printing after coloring parts of a single model.
- Through the tool Multimaterial painting multi-layer painting palette, the left and right keys respectively select the area to be allocated to different extruder printing, in order to prevent the printing of tainted colors or 3D Touch impact on the prime tower, it is necessary to change the shape and volume of the prime tower appropriately. What is more worth mentioning is that the prime tower can be made smaller and save filament. The function of "No sparse layers(EXPERIMENTAL)" in the printing setting should be enabled, but the size of the printing head should be paid attention to in this function to prevent collision with the model.

Mizar M_Printing Set (modified)

- Layers and perimeters
 - Infill
 - Skirt and brim
 - Support material
 - Speed
 - Multiple Extruders**
 - Advanced
 - Output options
 - Notes
 - Dependencies

Extruders

- Perimeter extruder: 2
- Infill extruder: 2
- Solid infill extruder: 2
- Support material/raft/skirt extruder: 1
- Support material/raft interface extruder: 1

Ooze prevention

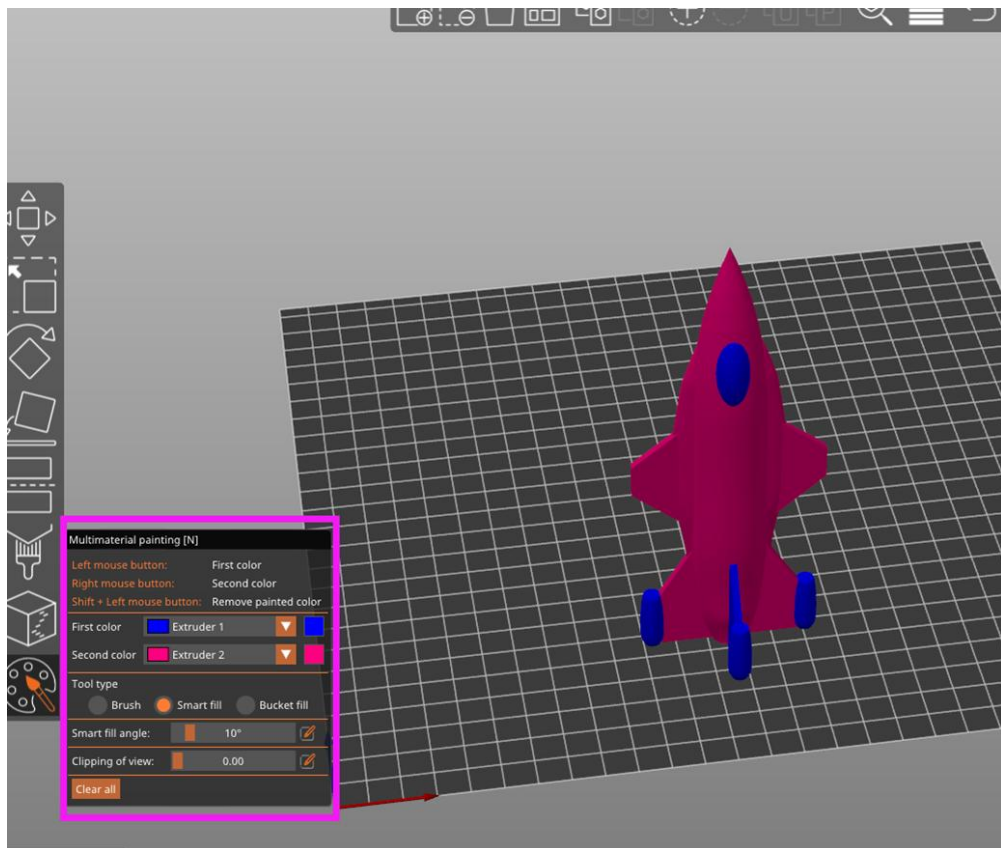
- Enable:
- Temperature variation: -5 Δ°C

Wipe tower

- Enable:
- Position X: 96.3805 mm
- Position Y: 180.3253 mm
- Width: 60 mm
- Wipe tower rotation angle: 0 °
- Wipe tower brim width: 2 mm
- Maximal bridging distance: 10 mm
- No sparse layers (EXPERIMENTAL):**
- Prime all printing extruders:

Advanced

- Interface shells:
- Maximum width of a segmented region: 0 mm (zero to disable)



The amount of material to be purged during a filament change can be set by clicking on the **Purging volumes...** button in the right toolbar.



The sheep on the right has the white color contaminated by the blue pigment from the previous filament

Simple mode

By default, you can simply set how much material will be purged when the filament is being unloaded or loaded. This lets you optimize the amount of wasted filament by, for example, decreasing the amount to purge when loading black filament. On the other hand, you can increase the value when transitioning to very light colors, to ensure they are not contaminated by previous colors.

Example:

	unload	load
Black	80	30
White	20	70

Transition **Black to White** = Black unload + White load = $80 + 70 = 150 \text{ mm}^3$

Transition **White to Black** = White unload + Black load = $20 + 30 = 50 \text{ mm}^3$

i If you switch to the Advanced mode, you can see the resulting purge volume calculated for each color change combination, with the current settings from the Simple mode.

4. Simplify 3D

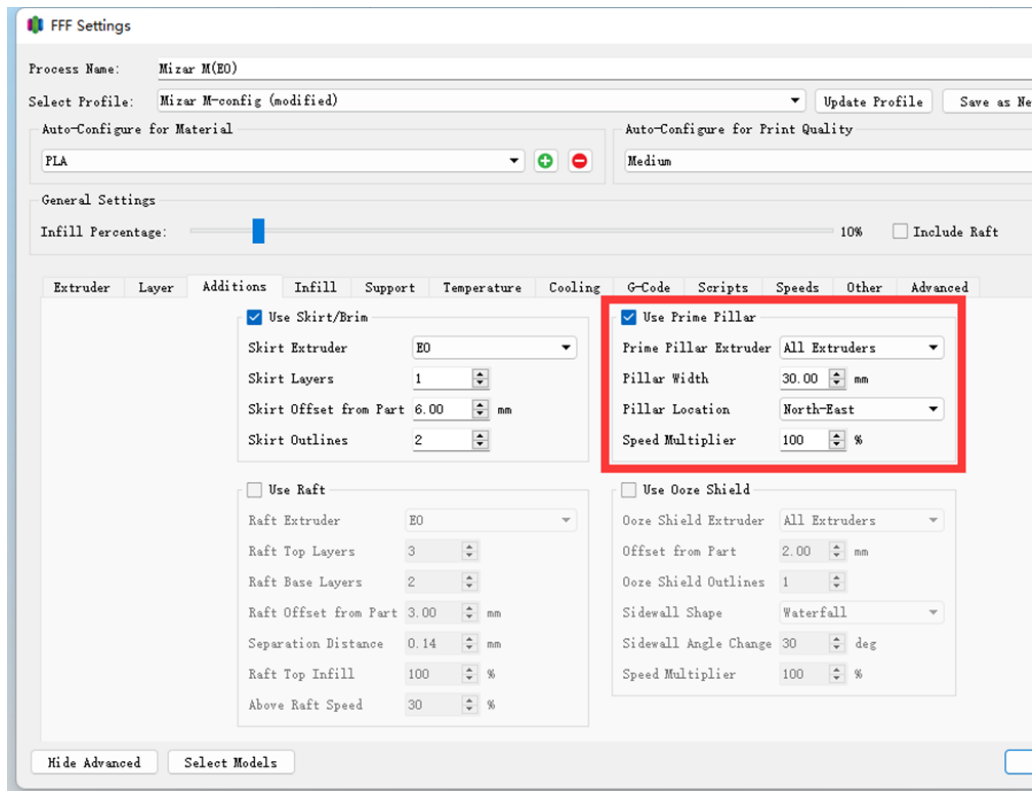
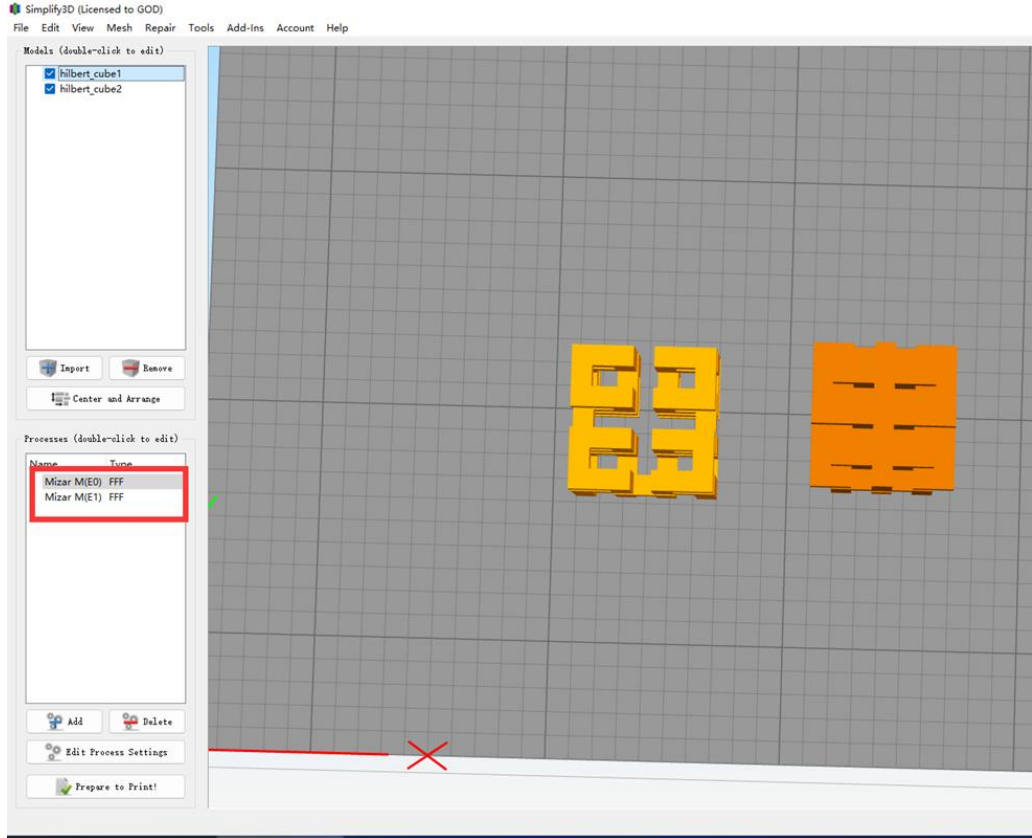


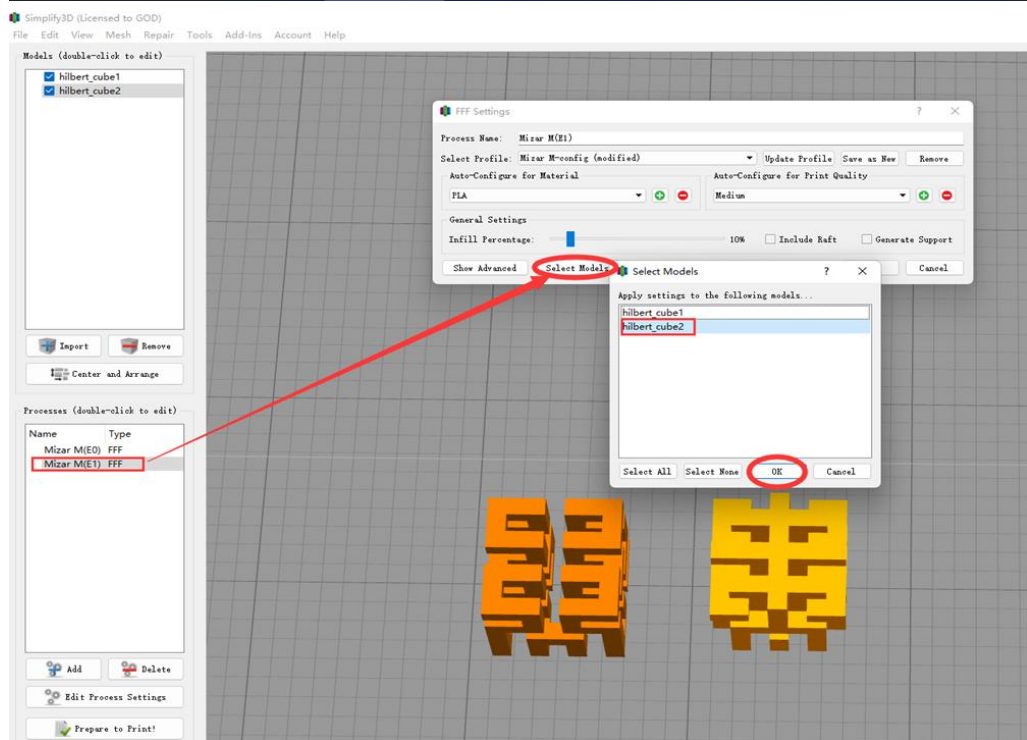
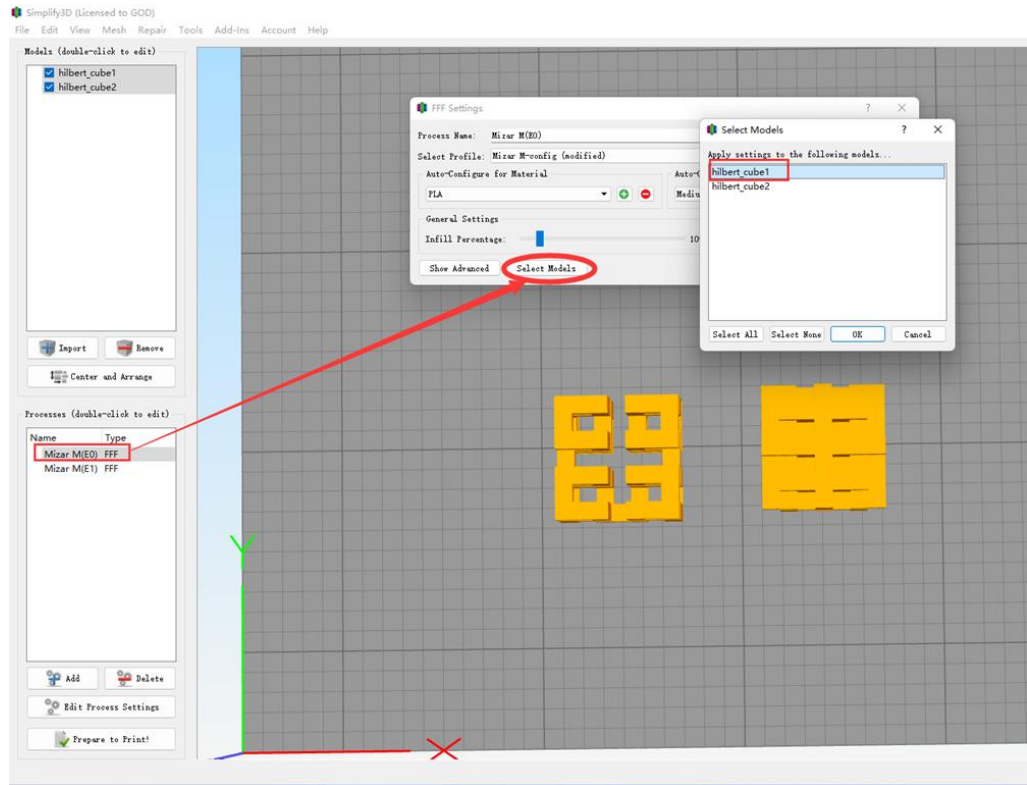
Simplify3D config
for Mizar_M.rar

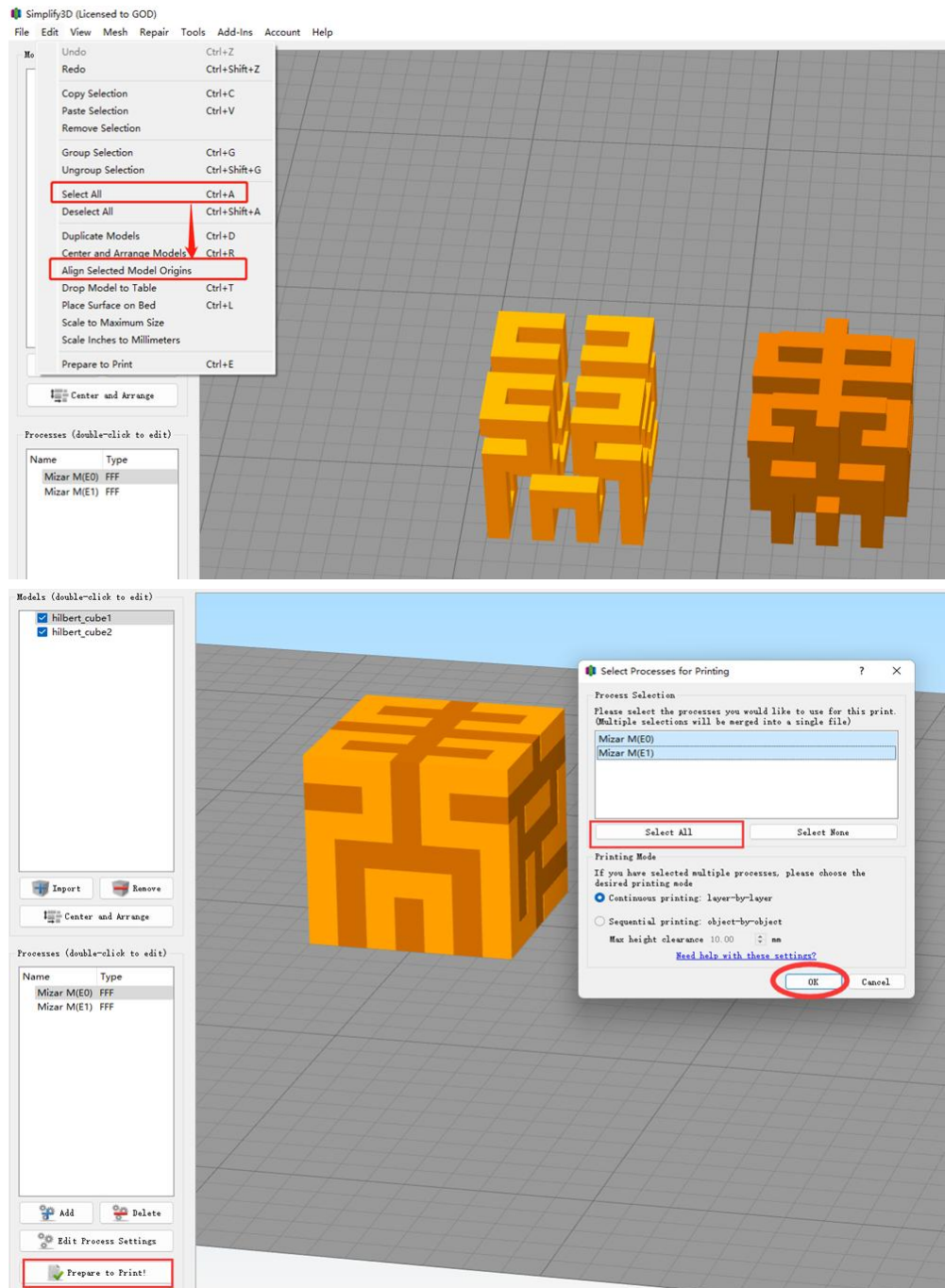
4.1 Separate-Color Printing

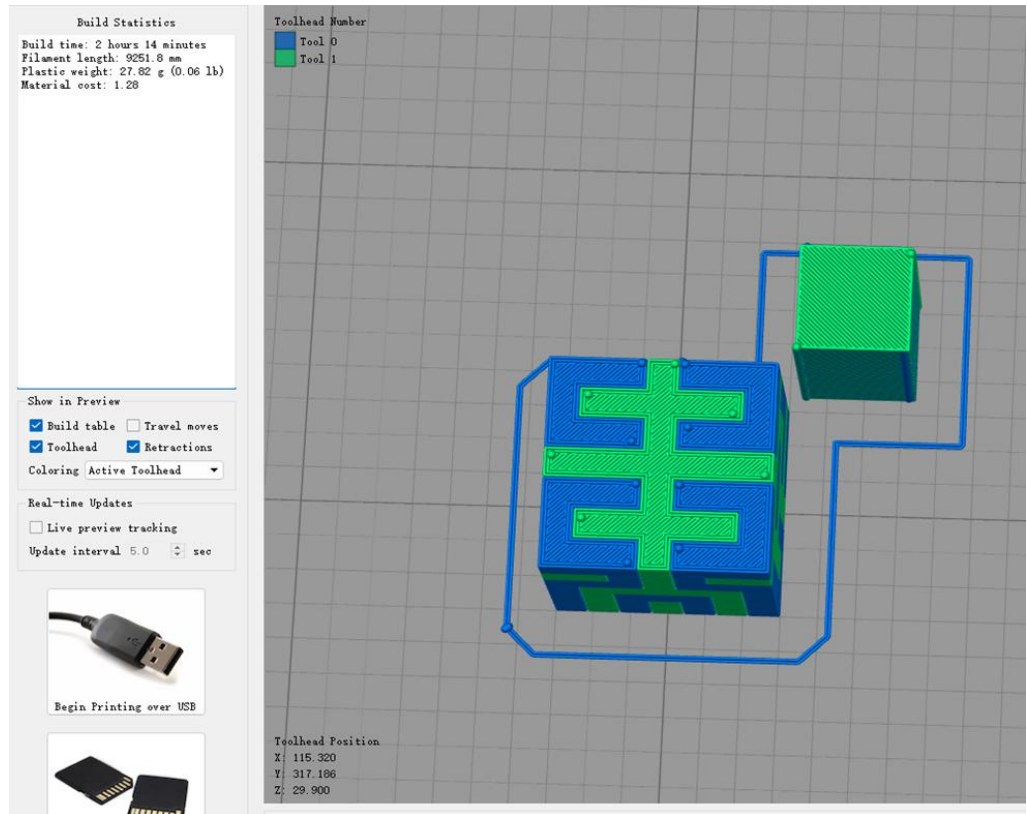
Applied to both Mix-Color module and Separate-Color module

Simplify3D needs to set up separate FFF processes for extruder E0/E1. Set Mizar M(E0) and Mizar M(E1) as shown below. Import 2 or more models, set the extruder head for each model, and set the size and location of the prime tower (for each process). Select all models after Ctrl A, enter the menu and select "Align Selected Model Origins". Select two processes to complete slicing.





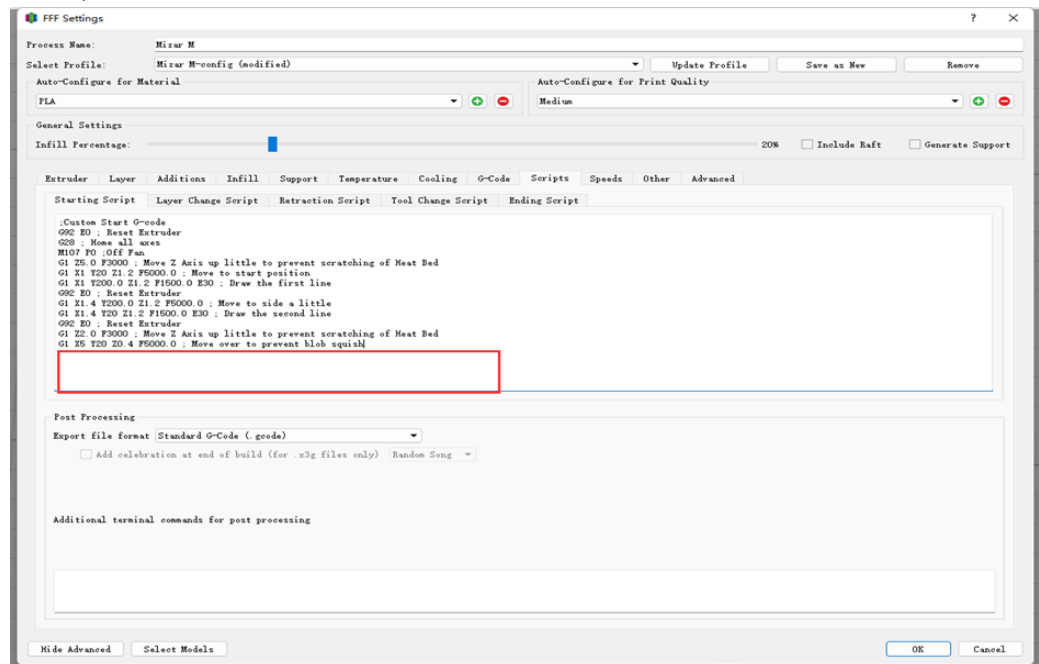




4.2 Mix-Color Printing

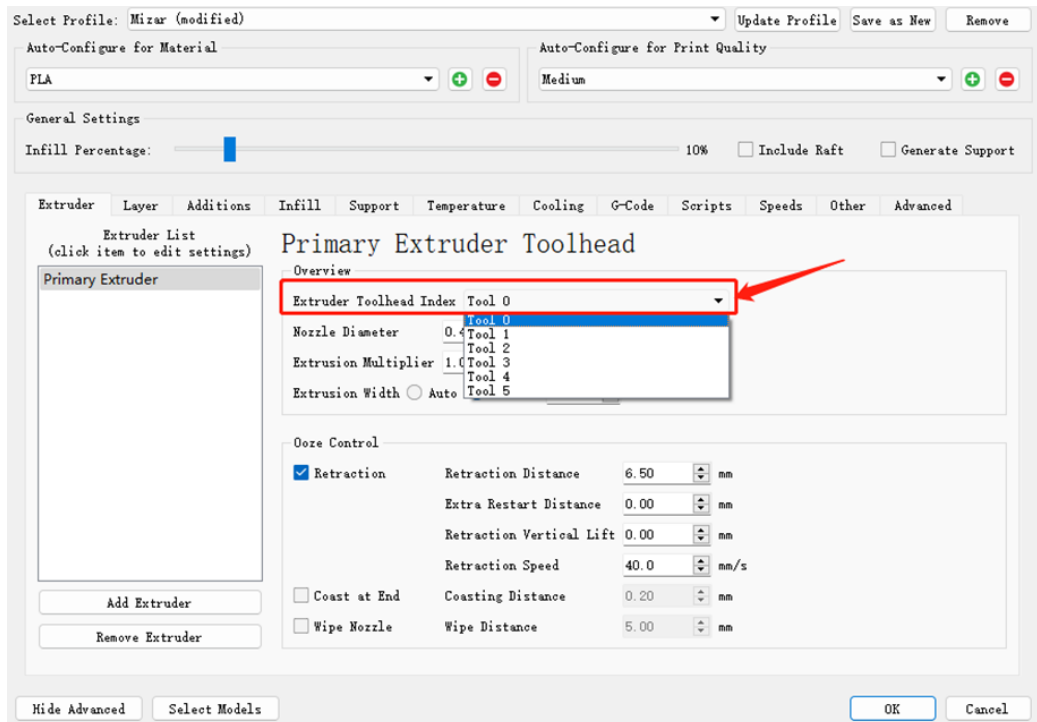
Make sure to install the Mix-Color module in advance

Start G-code is same with Cura. For the script, see Cura mix-color printing description in 1.5.



4.3 When you use a single extruder, Simplify3D deposit the default E0 extruder. If

you need to switch E1, select the extruder number "Tool 1" from the advanced options, as shown in the below figure.

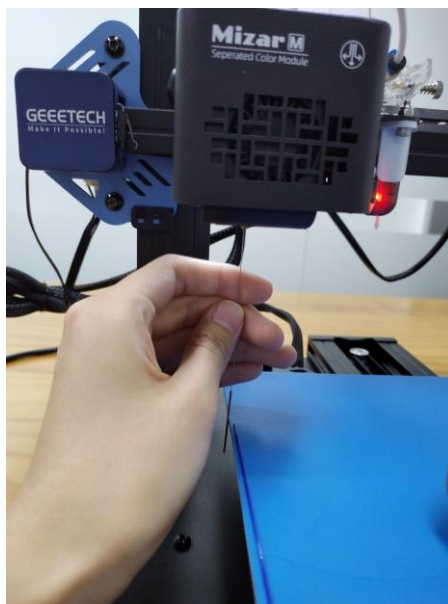


II. Printing Problem Guide

1. Blocked

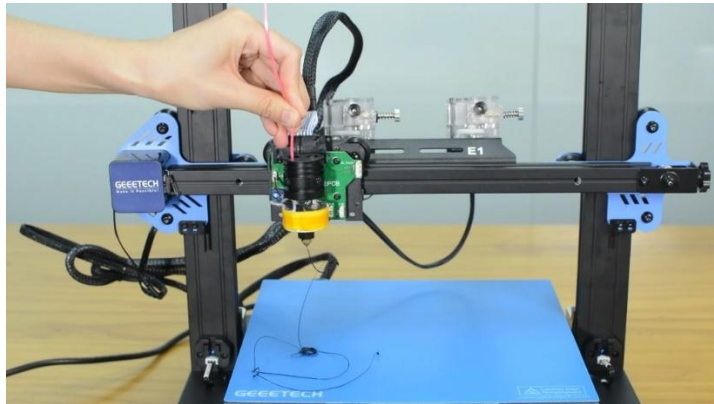
1.1 Unblock with a needle

For PLA, heat up the nozzle temperature to 220 and use a needle to poke the nozzle.



1.2 Push old filament through

Heat up the hot end to 220 °C for PLA and then use another piece of filament to push this through from the top to try to force the stuck filament in the nozzle out.



1.3 Dismantle and rebuild the hot end

In extreme cases when the nozzle remains blocked, you'll need to do a little surgery and dismantle the hot end.

a) Gradient Color Module

Be careful not to poke the hot end with the purge rod, otherwise the tube inside the hot end will be damaged.

Filament1 Filament2



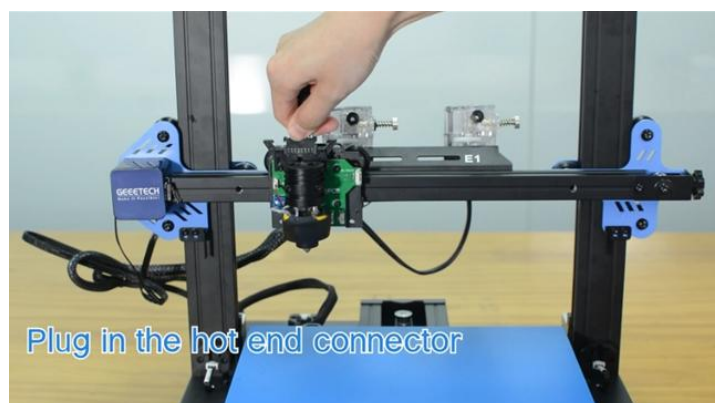
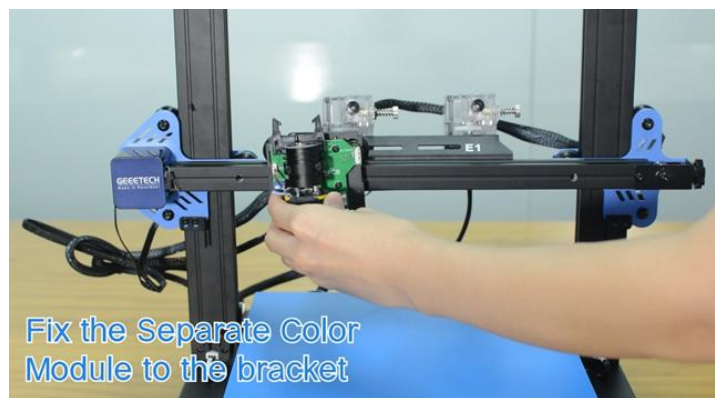
Internal structure of the hotend of the Separated Color Module.

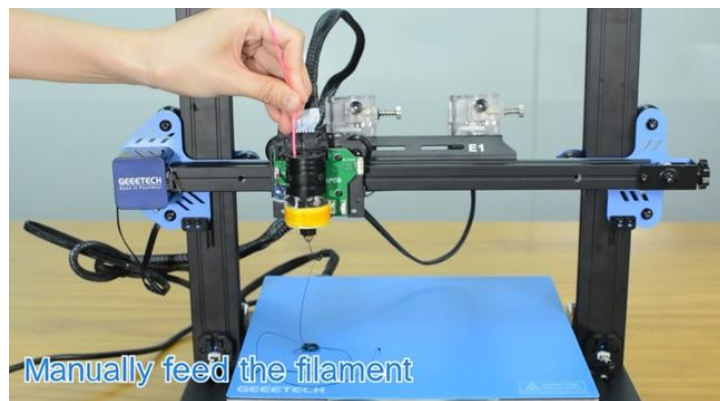
Only one filament is allowed into the hotend.





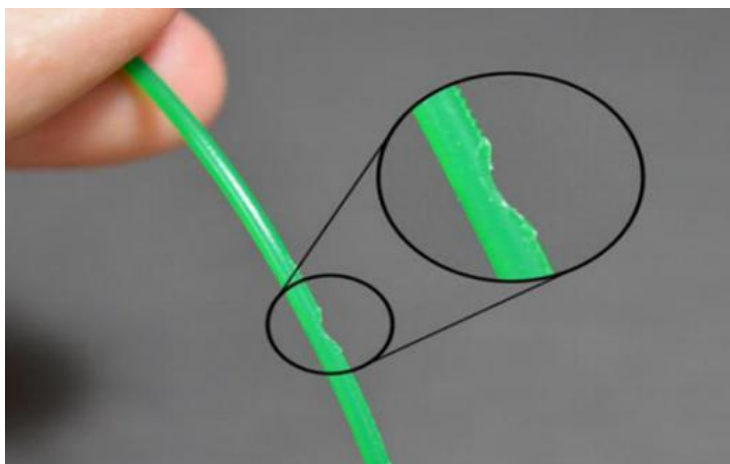
b) Separate Color Module





2. Grinding Filament

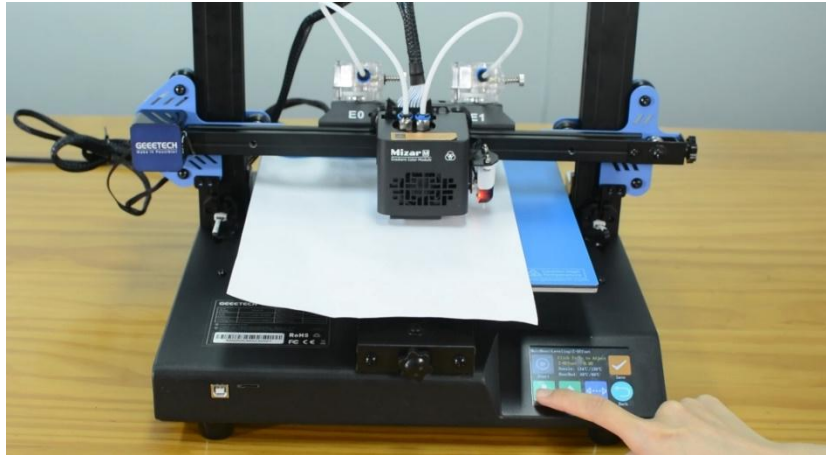
If the filament cannot move, but the gear continues to turn, the gear may plane some of the plastic away from the filament, leaving no place for the gear to hold on to the filament.



- 2.1 Increase the nozzle temperature by 5-10C°
If the print speed is too fast, the extruder will push the filament too quickly. Filament may not have enough time to melt and even clog in the nozzle. A clogged nozzle will then exert pressure back on the extruder, resulting in clicking.
- 2.2 Clean the hot end
A blocked hot end will restrict the flow of the filament, causing the extruder keep clicking.
- 2.3 Decrease the print speed
If the print speed is too fast, the extruder will push the filament too quickly. Filament may not have enough time to melt and even clog in the nozzle. A clogged nozzle will then exert pressure back on the extruder, resulting in clicking.

3. Not Sticking to the Bed

- 3.1 Level the print bed
Please refer to the auto-leveling or manual leveling video tutorial.
- 3.2 Adjust the Z-offset
Find the Z-axis offset option in your printer' s settings and make small adjustments — into the positive to raise the nozzle away from the bed, and negative to lower it closer.



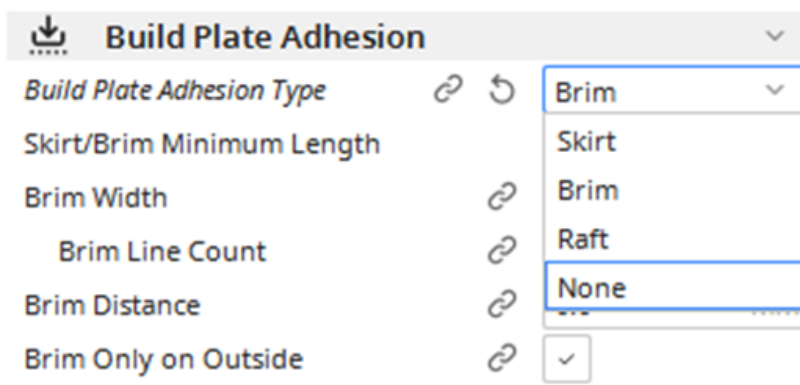
3.3 Clean the print platform

If the filament remaining on the Mylar sheet is difficult to remove, you can add a little water to the residual filament, or wipe it with a damp rag, then use a scraper to remove it. **Be careful to remove the mylar sheet and leave it away from the machine before operating it, and do not let the printer get wet. After use, wipe the Mylar sheet clean.**



3.4 Apply the build plate adhesion

It's convenient to apply some glue on the print bed to get a more adhesive first layer. And also, smaller items and those with only a small footprint in contact with the platform will require some type of Build Plate Adhesion. These can be added in your slicer software - — look for “Brim” and “Raft” .



4. Warping

Warping is common as it's caused by the natural characteristics of plastics. As the ABS or PLA filament cools it starts to contract very slightly; the problem of warping arises if the plastic is cooled too quickly.



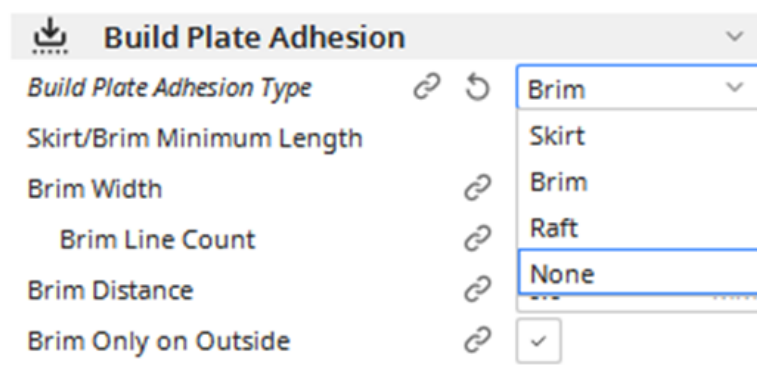
4.1 Use a heated print platform

If you get that temperature right then the first layer will stay flat on the print platform. For PLA, 60C° is enough, but for ABS and PETG, they're usually need higher hot bed temperature.

4.2 Apply an adhesive to the print bed

If you still find your print lifting at the edges then apply a tiny amount of stick glue evenly on the bed to increase adhesion.

4.3 Add build plate adhesion



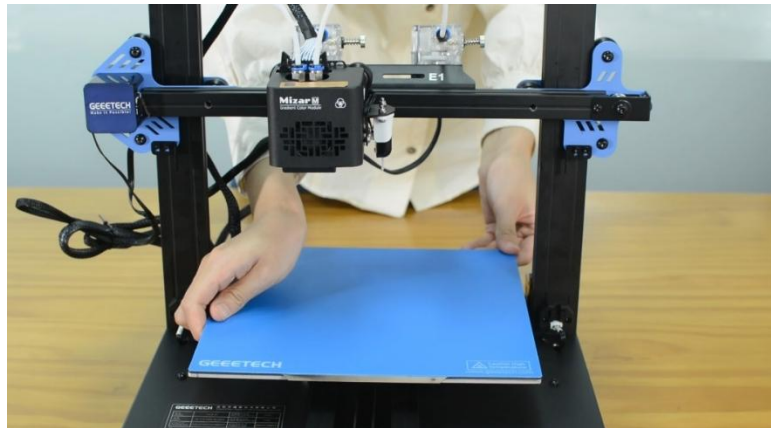
5. Layer Shift

5.1 Check that the printer is on a stable base

Place the printer on a stable base and in a location where it will avoid being knocked, poked and generally fiddled with.

5.2 Check the print bed is secure

If the hot bed wobbles, secure the eccentric isolation column under the hot bed with a wrench.

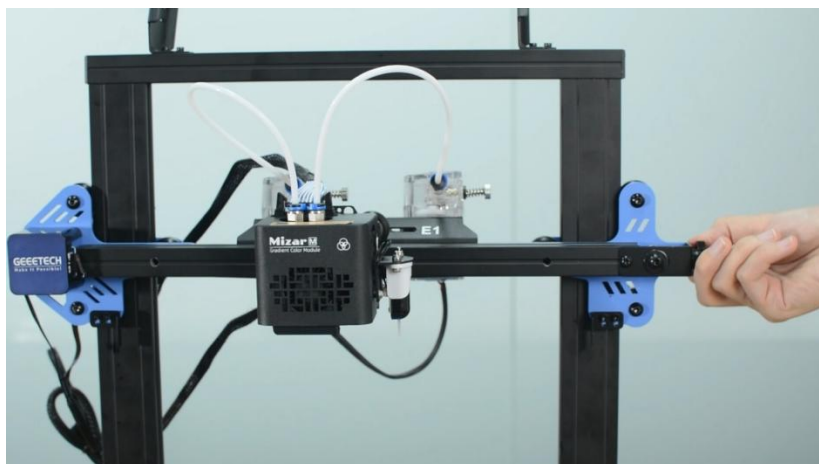


5.3 Decrease print speed

Too fast speed may result in the filament flowing in the correct quality the rest of the machine may struggle to keep up. If you hear a clicking during printing this could be a sign that the printer is going too fast.

5.4 Check the belts

The tension in each belt should be the same, if not then you' ll need to adjust the belt position to even out the belt tension. Over time the rubber belts will stretch, if there is quite a bit of play in the belts then it' s time to replace them with new ones.





5.5 Check the rods

Over time debris can build up on the rods which means that at some points along their length they encounter an increase in friction. This can affect the free movement of the head and cause layer shifting. A quick wipe and re-oil of the rods usually solves the issue.

6. Vibrations and Ringing

The issue of waves is usually down to one of two things, and more commonly a combination of the two. Vibrations and speed.



6.1 Reduce vibration

Make sure that the surface you use is solid and there are no visual vibrations.

6.2 Make sure everything is tight

As part of your maintenance routine make sure everything is bolted and tight.

6.3 Add oil

Check all the rods are clear of dust and dirt and then add a drop of oil to ensure everything is well lubricated.

6.4 Reduce the print speed

Printing too fast may cause the machine to shake and produce ringing, in which case you need to reduce the printing speed.

7. Under-Extrusion

There are several possible causes. First, the diameter of the filament used does not match the diameter set in the slicing software. Secondly, the amount of material that is extruded is too low because of faulty slicer software settings. Alternatively, the flow of the material through the extruder is restricted by dirt in the nozzle.



7.1 Check the hot end for clogging

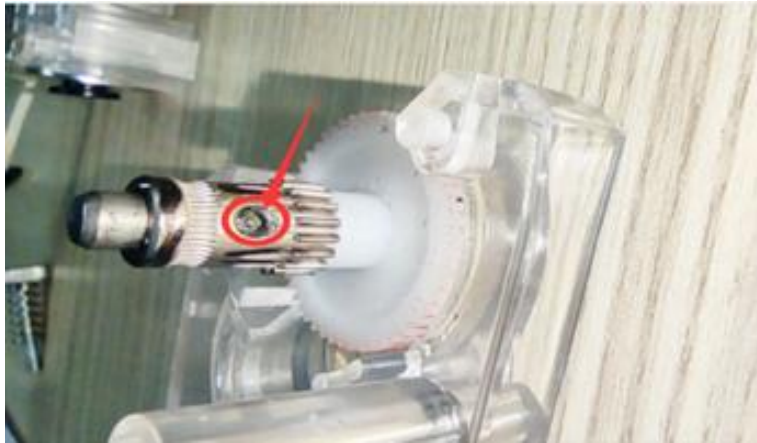
The most common cause of under-extrusion is a clogged nozzle. Please clean the nozzle and hot end first.

7.2 Decrease the print speed

For Mizar M, recommended print speed is 60-80mm/s. When the printing speed is too fast, the hot end does not have enough time to melt the filament, resulting in under extrusion.

7.3 Extruder Problem

Due to long-term printing, the grub screw in the gear is loose or detached. Please tighten it as shown in the figure.



Calibrate the extruder referring to:

<https://teachingtecht.github.io/calibration.html#esteps>

8. Print Looks Melt and Deformed

8.1 Poor Cooling

The most common cause of overheating is that the filament is not cooled in time, so the shape of print is easily deformed. Turn on the part cooling fan at full speed after the first layer printed.

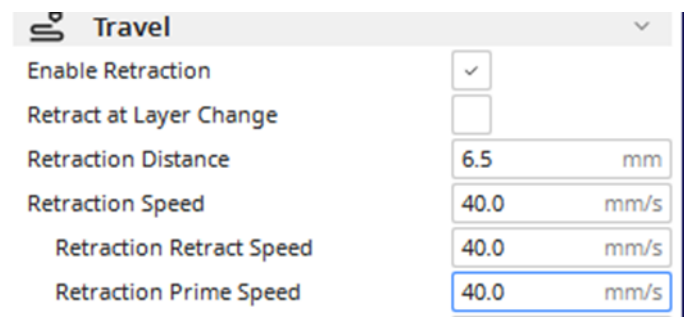
8.2 Print temperature is too high

If you've used the part cooling fan but still not working, you may need to lower the print temperature by 5-10C° (5C° as the minimum unit).

9. String

9.1 Enable retraction

Retraction distance determines how much of the filament will be pulled back from the nozzle. Generally speaking, the more filament pulled back from the nozzle, the less likely the nozzle is to string when moving. Normally, the retraction distance of the Bowden extruder is set to 5-10mm.



Another related setting the retraction speed, which determines how fast the filament can be pulled out of the nozzle. If the speed is too slow, the

filament will hang out of the nozzle and start leaking before they can move to a new position. If too fast, the filament may be break from the nozzle. The retraction speed between 40-100mm/s is good. The ideal value depends on the actual filament you are using.

Test retraction referring to: <http://retractioncalibration.com>

Calibration Generator 1.3.4
Instructions
Facebook Group
Donate

Warning!!! If you stop the gcode generated midprint restart your printer. It uses a mixture of relative and absolute movements.

X Dimension 2.20	Y Dimension 2.20	Nozzle Diameter 0.4
Start Retraction Distance 0.5	Increment Retraction 0.5	Filament Diameter 1.75
Start Retraction Speed 10	Retraction Speed Inc 10	Layer Height 0.2
Start Temp 200	Increment Temp 0	Extrusion Multiplier 1
Fan Speed 40	Fan Speed Increment 0	Bed Temp 50
Layers per Text 25	Number of Tests 15	Travel Speed 100
Print Speed 40	Custom Gcode G29	Generate GCode

Notes:
 For delta set bed size to 0 for x and y
 For direct drive start with .5 starting retraction and .25 increment
 For bowden start with .5 starting retraction and .5 increment

9.2 Decrease print temperature

If the print temperature is too high, filament in the nozzle will become very sticky and will flow out of the nozzle more easily. Decreasing the nozzle temperature by 5 to 10C° will have a noticeable impact on the final print quality.

9.3 The inner and outer wall speed difference

Reduce the difference between the inner wall and the outer wall print speed. It is recommended that the difference be set within 15%.

According to experience and verification, small inner and outer wall speed difference ensures a key factor of print quality, please refer to the example diagram below:

Speed (mm/s) (⚠The inner and outer wall speed difference is 15%, too large speed difference will reduce the print quality)	Default	250
	Inner Shell	225
	Outer Shell	188
	Top/Bottom	250

10. Zits and Blobs

The extruder must constantly stop and start extruding as it moves to different portions of the build platform. Most extruders are very good at producing a uniform extrusion while they are running, however, each time the extruder is turned off and on again, it can create extra variation.



10.1 Lower temperature & speed

Too high a nozzle temperature can cause over-extrusion, and too high a printing speed can reduce print quality because there's less time for the material to settle before the next layer is added.

10.2 Use coasting

When you've enabled coasting in Cura, the nozzle stops extruding material towards the end of its extrusion path. Instead, the material that's oozing from the nozzle is used to complete the layer.

Experimental

Enable Coasting

Coasting Volume	0.064	mm ³
Minimum Volume Before Coasting	0.8	mm ³
Coasting Speed	90.0	%

Add wall line count also helps.

Quality

Walls

Wall Thickness	0.8	mm
Wall Line Count	5	
Wall Transition Length	0.4	mm

10.3 Fix incorrect extrusion

If your printer isn't extruding material consistently and in the right amounts, printing issues will occur.

Calibrate the extruder referring to:

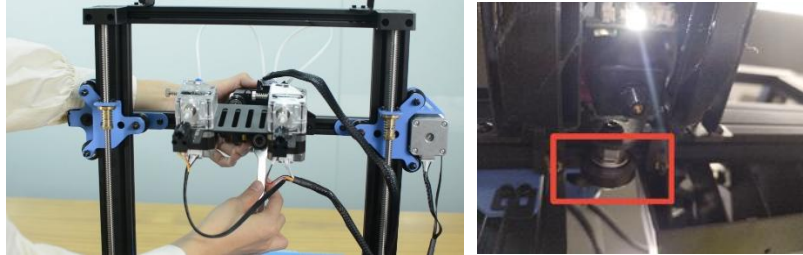
<https://teachingtecht.github.io/calibration.html#esteps>

III.Common Hardware Issues

1. Hot End

1.1 Hot end is shaking

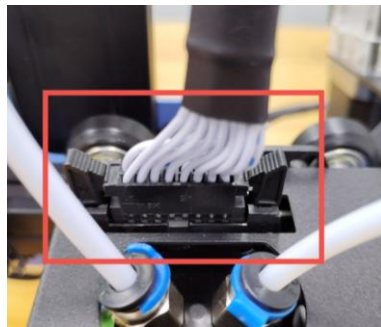
You can adjust the eccentric nut with the wrench until the pulley and the profile just touch. If it is tight, lag will happen and if it is loose, it will shake.



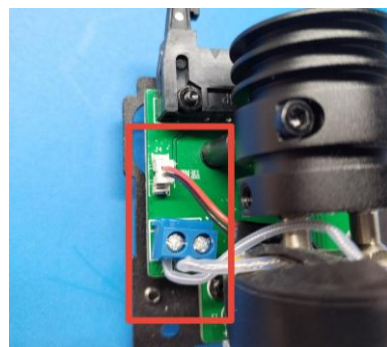
1.2 Heating failed



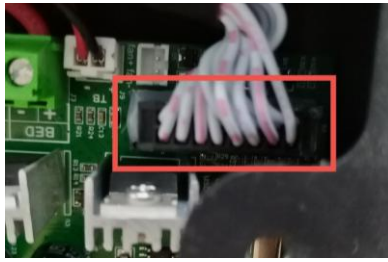
a) Check the hot end connector for loose or broken.



b) Check the connection of the heater cartridge and thermistor.



- c) Check the connection of the hot end on the motherboard.



2. Extruder

2.1 Clicking or slipping

Lower the print speed: If you print at too high a speed, the extruder will push the material too quickly, and it may not have enough time to absorb heat and melt properly. This partially melted filament can't leave the nozzle and may even clog it. A clogged nozzle will then exert pressure back on the extruder, resulting in clicking.

Increase the print temperature: To ensure a smooth flow of filament, it's crucial that the temperature of the hot end is set accurately. If you set the temperature too low, the filament may not melt properly.

2.2 Grinding filament

Adjust extruder tension: If it's too tight, slightly loosen your extruder's feeder (gear part) grip on the filament.

Replace the Teflon tube: If your Bowden tube or hot end lining are compromising filament movement, you may need to replacing it.

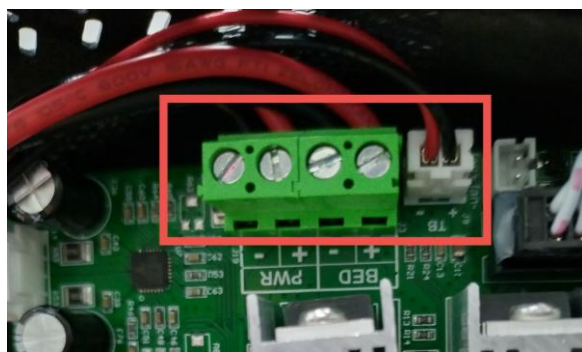
Clean hot end: Ensure that your hot end is clean and doesn't have any residue on it. Check that it doesn't have any other issues that could cause a jam.

3. Hotbed

3.1 Heating failed



Check the connection of the hotbed and thermistor.



3.2 Shaking

If the hot bed wobbles, secure the eccentric isolation column under the hot bed with a wrench.



3.3 Print is stuck to Hotbed

Please preheat the hot bed to above 80 degrees and gently scrape it with a spatula.

If the filament remaining on the Mylar sheet is difficult to remove, you can add a little water to the residual filament, or wipe it with a damp rag, then use a scraper to remove it. **Be careful to remove the mylar sheet and leave it away from the machine before operating it, and do not let the printer get wet. After use, wipe the Mylar sheet clean.**



3.4 Automatic heating

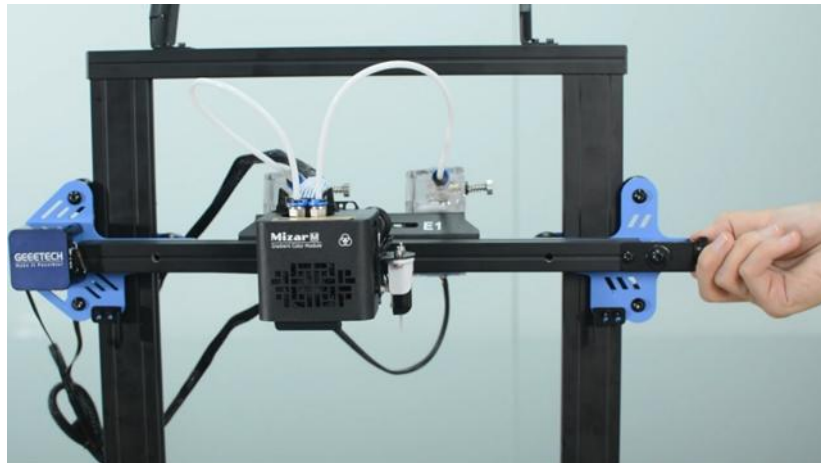
If the hot bed temperature automatically rise, is generally caused by the motherboard problem, it is recommended that you replace the motherboard or directly contact the after-sales personnel to deal with.

4. Belt

If it's too loose, the belt can slack and even skip teeth. Further, they won't respond well to sudden changes in speed and direction, with surface quality degradation particularly noticeable at higher printing speeds or in heavier setups, such as direct-drive extruders. Too tight, and the motors will be stressed and overheat.

4.1 Install the belt tensioner

Rotate the adjustable knob on the belt tensioner to tighten the belt. You want the belt tight enough so that when you move the carriage across its axis, the motion is smooth but not wobbly. If the carriage wobbles, you should tighten the belt; if the motion is too hard or rough where you feel every bump along the axis, you should slightly loosen the belts.



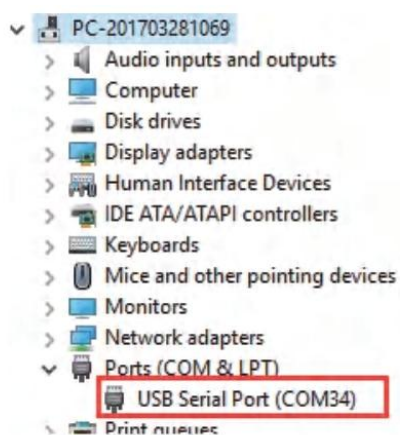
5. Data Reading

5.1 Unable to read TF card

Try another TF card. If still can not read after many attempts, maybe the card slot is broken. Please contact after sales service for help.

5.2 Unable to connect to the computer

Please check if the printer driver CH340 is installed. The baud rate of the serial port is 250000.

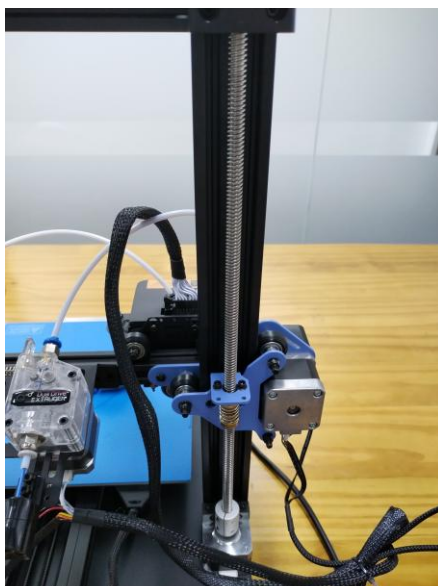


6. Other maintenance

6.1 Z-axis leadscrews

Mechanical friction and movement impairments will require more torque from the stepper motors, often leading to overheating.

Both the linear rods with ball bearings and the linear rails with carriage slides require some sort of lubrication to reduce the wear and tear generated by friction.



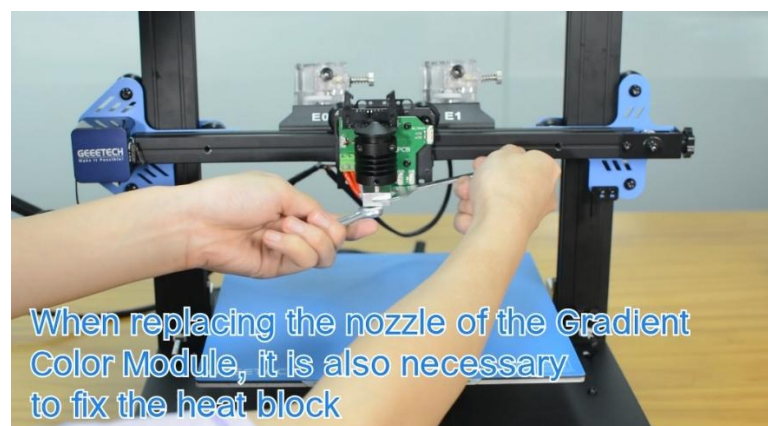
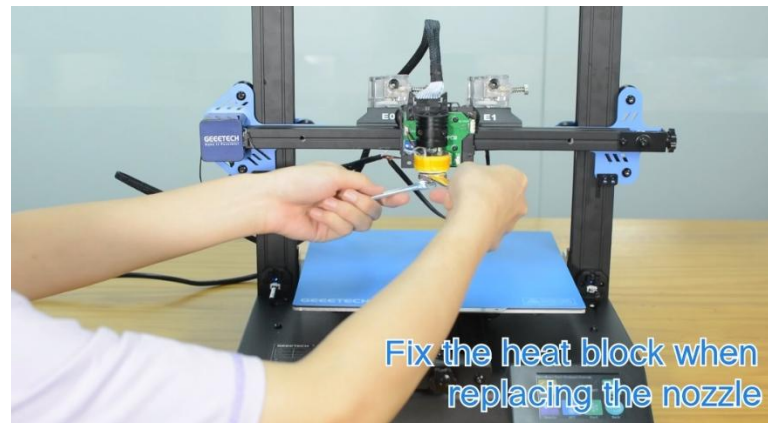
6.2 Nozzle

Try and keep the nozzle as clean as possible. Heat it up and use a wire brush to remove all melted and un-melted plastic that's been accumulating over successive prints.

As the nozzle degrades over a long period of time, it's quite difficult to spot

this damage.

Preheat the nozzle to 200C° before replacing it, details are shown below.



6.3 Stepper Motor

First, never install a motor (or anything, for that matter) on a printer while it's plugged in and on. This is hazardous because if done incorrectly, you risk shocking yourself, frying delicate components, and starting an electrical fire. Always power off the printer and unplug it before commencing any kind of work.

Second, never unplug a stepper motor that's being powered. If you paid attention to the first step, this isn't a problem, but if you didn't, you'll likely end up frying your stepper motor driver and possibly your control board.