

LiCai MICOLOR 3D Printer Product User Manual

STATEMENT

COPYRIGHT

This user manual contains proprietary information protected by copyright. Without our company prior written permission, any of the contents in this manual should not be copied and reproduced. This manual is copyrighted. Our company reserves the right of final interpretation to this manual.

DISCLAIMER

This manual is used for installation, operation and maintenance. It is not a product guarantee. Our company try our best to make sure that the content is accurate and complete in manual. But we do not assume any responsibility for the error or omission in this manual. We reserve the right of publishing this manual and modifying the error in it. There is no further notice for the manual information changing.

Our company is not responsible for the software reliability used for the device which is not provided by our company.

Table of Contents

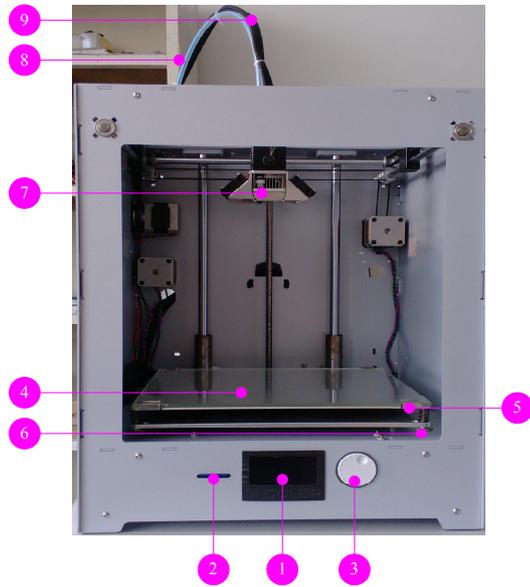
1. Product Introduction	1
1.1 Product at a glance.....	1
1.2 Specifications of the printer	2
1.3 Essential accessories.....	3
2. Printer Components Installation	4
2.1 Printer power installation	4
2.2 Filament components installation.....	4
3. First Use of the Printer	5
3.1 The first run wizard.....	5
3.1.1 Leveling the buildplate	5
3.1.2 Loading filament	8
3.1.3 Printing 3D model.....	10
3.1.4 Removing the print	11
4. Software Installation	12
4.1 Software Installation.....	12
4.2 Software Uninstallation	13
5. Software Use Introduction	14
5.1 Open and Exit	14
5.2 User Mode Instruction.....	15
5.2.1 [Load] Function	16
5.2.2 [Delete] Function	17
5.2.3 [Center] Function	18
5.2.4 [Rotate] Function	19
5.2.5 [Scale] Function	20
5.2.6 [Print Mode] Function.....	21
5.2.7 [Slice] Function.....	22
5.2.8 [View Mode] Function	23
5.2.9 [Save Mode] Function.....	24
5.2.10 [Print] Function.....	25
5.3 Setting Mode Instruction	26
5.3.1 Simple Setting Mode.....	26
5.3.2 Expert Setting Mode	26
5.4 Setting Parameters Intrusion.....	27
5.4.1 Basic Settings.....	27
5.4.2 Advanced Settings.....	31
5.4.3 Preference Settings.....	33
5.4.4 Machine Settings.....	34
5.4.5 Expert Settings	35
6. Maintenance and Frequently Asked Questions.....	39

6.1 Precautions	39
6.2 Maintenance	39
6.3 Frequently Asked Questions	40
7. Warranty and License.....	42
8. Contact Us.....	43

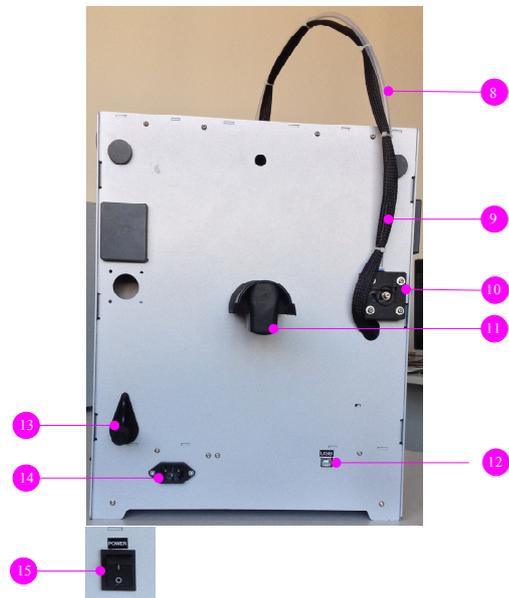
1. Product Introduction

1.1 Product at a glance

- 【1】 Display
- 【2】 SD-card slot
- 【3】 Push and rotate button
- 【4】 Build plate
- 【5】 Glass clamps
- 【6】 Buildplate screws
- 【7】 Print head
- 【8】 Filament guide tube
- 【9】 Print head cable



- 【10】 Filament feeder
- 【11】 Filament spool holder
- 【12】 USB socket
- 【13】 Filament guide holder
- 【14】 Power socket
- 【15】 Power switch



1.2 Specifications of the printer

Technical Description	Specifications
Product Dimensions	350x500x600mm
Product Weight	15kg
Build Volume	230x225x205mm
Filament	PLA/ABS
Layer Height	Minimum Layer Height = 0.02mm
Support	Auto
Positioning Precision	X Y Z: 12.5/12.5/5 micron
Print Speed	30mm/s – 300mm/s
Filament Diameter	3.0 mm
Nozzle Diameter	0.4 mm
Offline SD Print	Support
Software Bundle	MICOLOR
Operating Systems	Windows
Modeling Technology	Fused Deposition Modeling
Extruder Operating Temperature	180° - 260° C
Platform Operating Temperature	50° - 100° C
Operating Temperature	15° - 32° C
Storage Temperature	0° - 32° C
AC Input	90 ~ 264VAC 47 ~ 63Hz 3.5A/115VAC 1.8A/230VAC
Power Requirement	24VDC@12.5A

1.3 Essential accessories

There are a lot of essential accessories that comes with this 3D printer. All the following parts should be in the box, please check contents.

MICOLOR 3D Printer Parts List			
No.	Name	Quantity	Remarks
1	LiCai 3D printer material	1	At the bottom of the box.
2	LiCai 3D printer	1	
3	Filament spool holder	1	
4	Filament guide holder	1	
5	Nozzle special cleanup drill	1	
6	Test sample	1	
7	USB data cable	1	
8	Power line	1	
9	SD card	1	User manual, software and test model in it.
10	Warranty card	1	
11	Certificate	1	
12	Adjustment piece for nozzle height	1	

2. Printer Components Installation

2.1 Printer power installation

Power ON

1. Make sure the power switch 【15】 is in the OFF position;
2. Insert the power cable in the power socket 【14】 ;
3. Plug the external power supply into the external socket.

Power OFF

1. If the power supply has to be detached, the first step is to make sure the printer turned off;
2. Plug the power cable from the external socket;

CAUTION: Please make sure not to pull the cable. Excessive force may result in breaking this component.

2.2 Filament components installation

1. The spool holder attaches to the back of the printer, as shown in 【11】 .
2. Orientate the filament guider so that it will fit in the hole, put it in and rotate center-clockwise 90 degree's so that the little pin clicks into the tiny hole above.

The filament installation will be introduced during the printer operation.

3. First Use of the Printer

3.1 The first run wizard

When you switch on your printer, this display will light up and guide you how to use the printer first time.



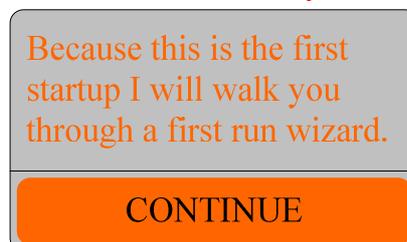
You can navigate through the display by the push and rotate button. By rotating you can select or control an action. By pushing you can confirm an action. When pushing the button you hear a “beep” sound to confirm your chosen action.



After the Welcome Screen on the display, click [CONTINUE] to go to the next step.

NOTE: If you don't see the Welcome Screen, navigate to [MAINTENANCE], [ADVANCED] and confirm for a [Factory reset]. You can use this function for the first run wizard at any time.

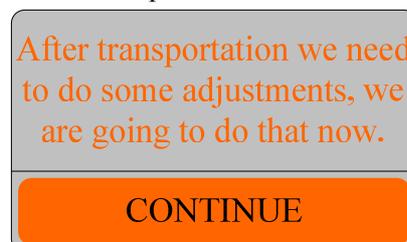
Follow the prompts to operate.



3.1.1 Leveling the buildplate

If the build plate is too far away from the nozzle your 3D-print might not stick to the build plate. Equally, if you adjust the build plate too close to the nozzle, the build plate can block the filament from extruding from the nozzle. It can also scratch the build plate.

NOTE: By leveling the build plate, the 3D-print will stick well to the build plate. Before leveling it is better to check out the state of build plate screws, neither too loose nor too tight.

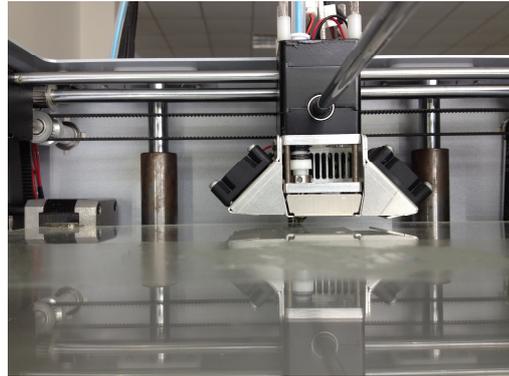


Level the build plate by the following steps.

First step: When the build stops lifting, you need rotate the button to control the height between build plate and nozzle to 1mm. See the specific operation in right figure. [CONTINUE] when finished.

Rotate the button until
the nozzle is a millimeter
away from the buildplate.

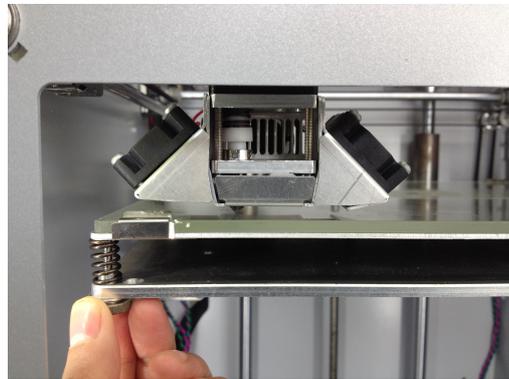
CONTINUE



Second step: Rotate this left build plate screw to control the height to 1mm. See the specific operation in right figure. [CONTINUE] when finished.

Turn left buildplate
screw till the nozzle is
a millimeter away
from the buildplate.

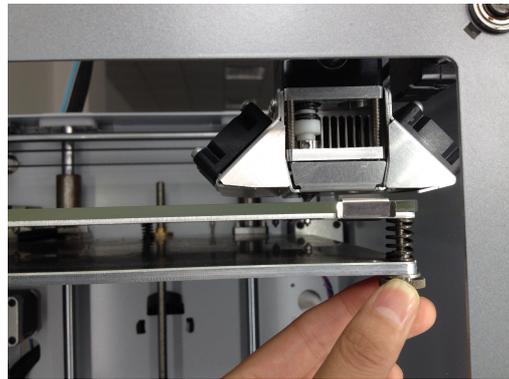
CONTINUE



Third step: Adjust a similar build plate screw on the right hand side. See the specific operation in right figure. [CONTINUE] when finished.

Turn right buildplate
screw till the nozzle is
a millimeter away
from the buildplate.

CONTINUE



NOTE: In the adjusting steps, the print head will automatically move to the adjustment point. Build plate needs manual adjustment to get its fit position.

Now we need to do this again to fine tune the build plate. Make sure you have a piece of paper within reach and press [CONTINUE] to go to the next step.

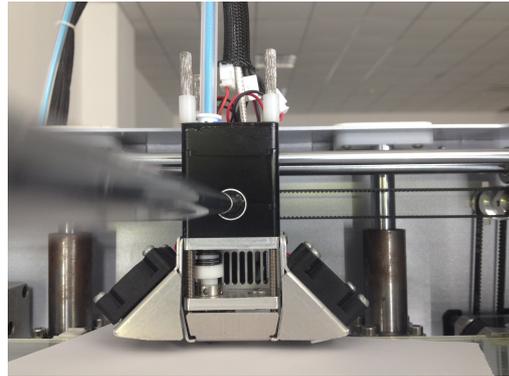
Repeat this step, but now use a sheet of paper to fine-tune the buildplate level.

CONTINUE

See the specific operation in right figure. [CONTINUE] when finished.

Slide a paper between buildplate and nozzle until you feel a bit resistance.

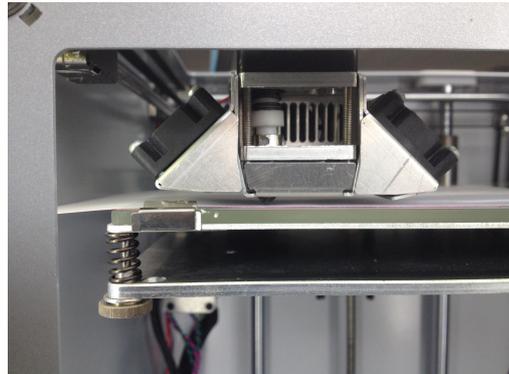
CONTINUE



See the specific operation in right figure. [CONTINUE] when finished.

Repeat this for the left corner.

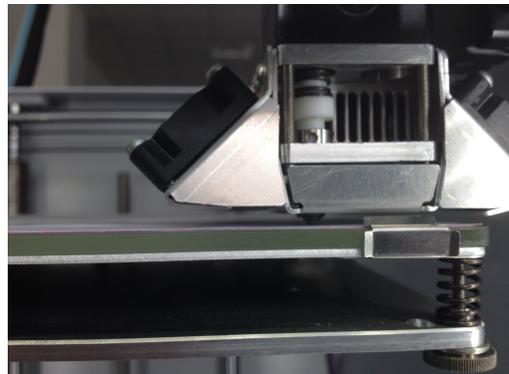
CONTINUE



See the specific operation in right figure. [CONTINUE] when finished.

Repeat this for the right corner.

CONTINUE



NOTE: If there is some trouble or it needs to level the build plate again, select [MAINTENANCE]-> [Build - plate] to level it again.

3.1.2 Loading filament

After completing the initial leveling build plate, you can press [CONTINUE] to load the filament for printer.

Now that we leveled the buildplate. The next step is to insert material.

CONTINUE

Please make sure the filament spool is positioned over the spool holder, guided along the filament guide in counter clockwise direction.



It takes a minute while the print head heats up. Please wait for the onscreen process bar to reach 100%.

NOTE: The print head will move itself to the front automatically.

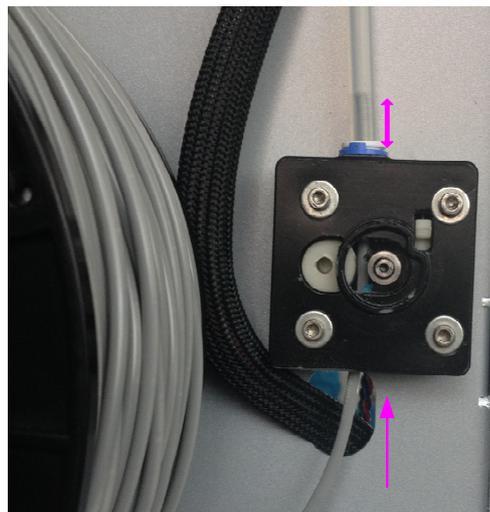
Please wait, printhead is heating for material loading.

Press [CONTINUE] to insert filament to the filament feeder.

Inset new material from the rear of printer, above the arrow.

CONTINUE

Take the filament and put it in the hole at the bottom of the filament feeder. Push the filament in the filament feeder until the filament is being grabbed by the knurled wheel. When the filament appears in the bodern tube about 1cm, you can press [CONTINUE] to make the filament loaded automatically by the printer.



Wait a few minute. When the filament is extruded from the nozzle, press the button to [CONATINUE].

NOTE: Don't be worried if the filament that initially comes out of the nozzle is not the color you expected. There's probably some filament left inside the nozzle. This is caused by testing the printer before packing. Wait until you see the color that you loaded comes out of the nozzle before you click the [CONTINUE].

Push button when material exits from nozzle...

CONTINUE

Next select the material you have inserted in the printer.

CONTINUE

Select the right filament type which is consistent to the label on the filament spool.

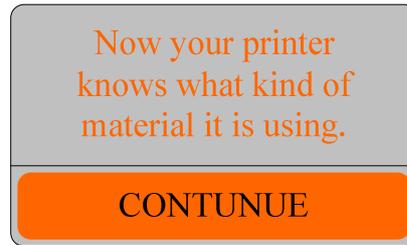
PLA	ABS
-----	-----

It is important to select the right filament type. The printer will load the print parameters to match with the filament type automatically. Press [YSE] to confirm the selection.

You have chosen PLA as material, Is this right?

YES	NO
-----	----

NOTE: Do not touch the extruder while it is heating, it is heatint up to 200° C.



3.1.3 Printing 3D model

Now the printer has leveled its build plate and has filament loaded, it can begin to print 3D model by clicking [CONTINUE].

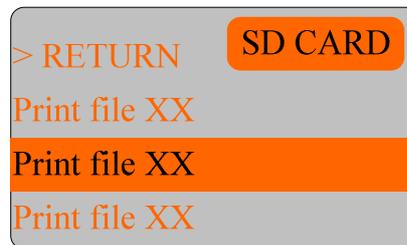


NOTE: Before printing, make sure that you add a thin layer of glue on the middle of the build plate. If the filament adhesion effect is not ideal on the glue layer, the layer can be made by using double-sided tape. When the build plate temperature is between 70° ~ 90° , the double-sided tape has a better adhesion effect. When the build plate temperature is above 90° , both glue and double-sided tape will make a bad effect to the print model.

Make sure the SD-card is inserted in the SD-card socket. If not, please find the SD-card that came with the printer and insert it. Press [CONTINUE] to begin print.



The display shows the files in SD-card. Rotate the button to select the file you would like to print. Then click the button to confirm the selcetion.



After you have choosen file to print, the printer will prepare itself and heat its print head nozzle and build plate. This will take up to 5 minutes. In the print process, the display will show the left print time, the process bar, tune and abort.

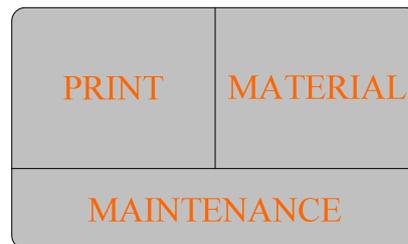
NOTE: Do not touch the extruder while it is heating and in use. Temperature can vary between 200° C~ 260° C

3.1.4 Removing the print

When your 3D print is finished, the display will indicate that the printer is cooling down. The progress bar indicates when it is safe to take out your print.

NOTE: Don't touch the print inside the printer while it is still cooling down. Wait until the display tells you "Print Finished. You can Delete the print." Some parts are cooling down from a temperature of max. 260° C and the print shaping progress will be affected.

Removing the print, you can select [PRINT] to continue printing, or [MATERIAL] to set the filament parameters, or [MAINTENANCE] to adjust other printer parameters.



When restart the printer, the display only indicates the [PRINT], [MATERIAL], [MAINTENANCE]. You can select the [MAINTENANCE]->[ADVANCED]->[Factory reset] to run the Welcome Screen.

4. Software Installation

4.1 Software Installation

Double click on the application icon to open the LiCai MICOLOR software installation wizard.



Install the software depending on the wizard.

If you select the **● Full installation**, you will get the installation interface directly.

If you select the **● Custom**, you will get the installation path selection interface and then obtain the installation interface.

When the progress bar reaches 100%, it pops up driver installation dialog.

Then some of Win7 system will pop up the corresponding installation instructions according to the system type. Press [OK] to continue the installation.

After the driver is completed, the software is installed.

When the software is installing, you can select [Cancel] to interrupt the installation. The system has not been modified. You can re-install the software.

After installation, shortcuts can be seen in the start menu and desktop.

4.2 Software Uninstallation

Select the Uninstall MICOLOR from the start menu. Then uninstall the software according to the uninstall wizard.

NOTE: All the generated files during the installation can be Deleted in uninstall progress. However, the files saved in the installation path during software use can not be Deleted. They need manually deleted.

5. Software Use Introduction

5.1 Open and Exit

Software open:

Double click the shortcut on the desktop or click the shortcut in the start menu to run the software.

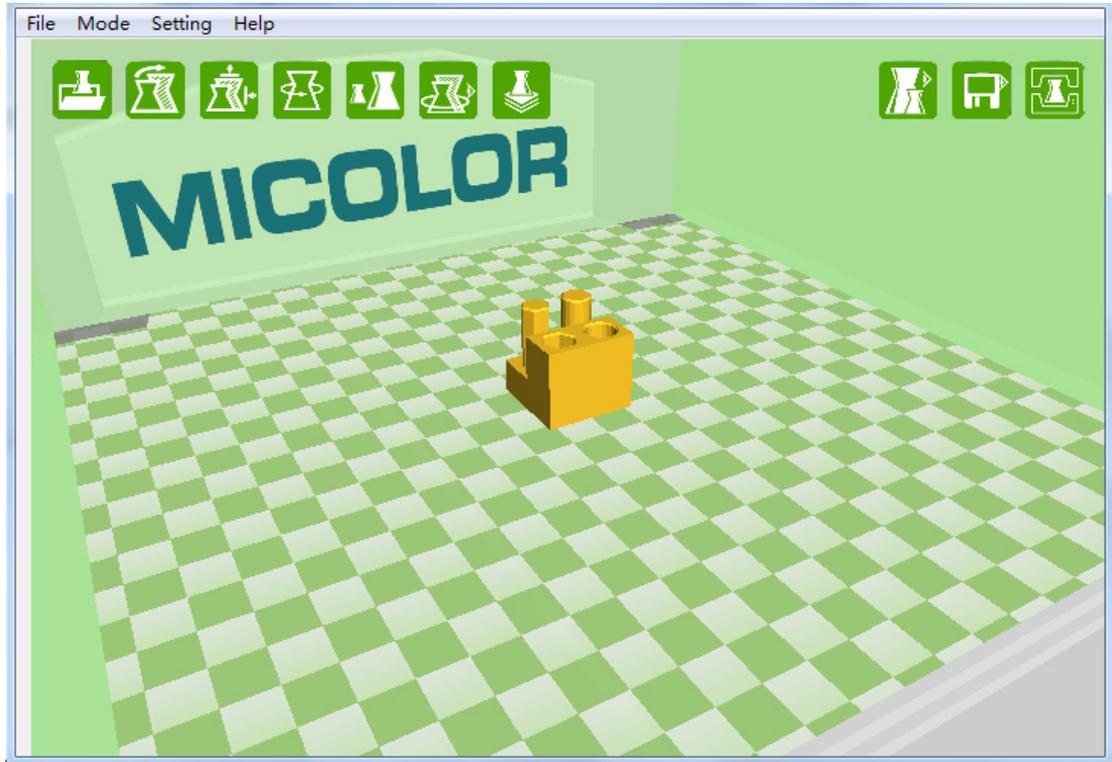
Software exit:

Method 1: Click the red X on the top right corner to close the software.

Method 2: Select the menu bar File -> Quit to close the software.

5.2 User Mode Instruction

When you open the software, the user mode is default. In this mode you can load 3D model for printing directly.



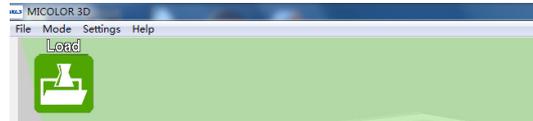
There are ten operation buttons in user mode. They are [Load], [Delete], [Center], [Rotate], [Scale], [Print Mode], [Slice], [View Mode], [Save Mode] and [Print]. The [Print Mode], [View Mode] and [Save Mode] are button groups. When the software is started, only [Load] button is visible. When you connect the printer and software with USB, you can operate the [Load], [Slice] and [Print] to achieve printing 3D model. The function of each button is explained as follow.

5.2.1[Load] Function

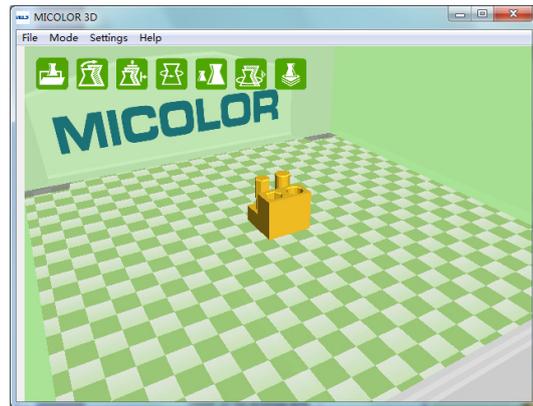
When you open the software, you can only see the [Load] button. Click the [Load] button to load 3D model file.

NOTE:

The size of selected button becomes a little bigger.



After the software finishing loading 3D model, the operational buttons are in visible state. A single 3D model is loaded in the center of the printing area.



Load multiple 3D models in the same method.

5.2.2 [Delete] Function

Before you operate [Delete] button, you had better some functions using mouse in printing area.

Select 3D model: You can select the 3D model by moving the mouse in the 3D model area. The selected 3D model will be brighten with the contour lines.



Drag 3D model: Hold and move the mouse left button, you can drag the selected 3D model.

If the color of 3D model is gray, it indicates that you have dragged the 3D model out of the printing area. In this condition, the 3D model cannot be printed. You need to drag it back into the printing area.

Rotate view: Hold and move the mouse right button to rotate the view. It is convenient for users to observe the 3D model from different perspectives.

Zoom view: Hold and move the mouse right and left buttons to zoom the view for user observation.

Select the 3D model and click [Delete] button, you can delete the 3D model you dislike.

When all 3D models are deleted, only the [Load] button is visible.

5.2.3 [Center] Function

Center 3D model: Select the 3D model and press [Center] button. This operation can set 3D model in the center of printing area.

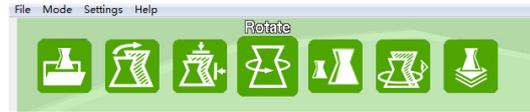


When there are multiple 3D models in printing area, one centered model will lead to other model out of printing area. Please make sure all models are in printing area.

5.2.4 [Rotate] Function

Rotate 3D model: Select 3D model and click [Rotate] button.

Choose different color coil to rotate the model at different direction.



Choosing red coil can make the model rotated in Z-axis direction.

Choosing green coil can make the model rotated in X-axis direction.

Choosing yellow coil can make the model rotated in Y-axis direction.

When model rotation is completed, click the [Rotate] button again to make the rotating coils invisible.

5.2.5 [Scale] Function

Scale 3D model: Select the 3D model and click [Scale] button. Then the model size can be seen. Hold the selected size cubic and move the left mouse button to resize the model.

After model is finished scaled, press [Scale] button again to cancel the size display. You can find the menu bar File -> Save model to save the modified model for future print use.



5.2.6 [Print Mode] Function

[Print Mode] button is a group button. When there are multiple 3D models in the printing area, you can choose [All] or [One] to change the generated Gcode. Slicing is needed after changing this group button. Slicing result will be shown in [Mode view] button group.

When [All] button is available, the 3D models in printing area will be sliced together. The generated Gcode can make them printed simultaneously.



When [One] button is available, the 3D models in printing area will be sliced one by one. The generated Gcode can make them printed separately.

NOTE: When [One] button is selected, the height of 3D model can not be more than 55mm, and the distance between models is not easy too small. Otherwise the generated Gcode is still together.

5.2.7 [Slice] Function

When no changing operations are done to 3D models in printing area, you can press [Slice] to generate slicing progress and obtain the Gcode for printing.

NOTE: Don't Delete, drag, rotate or zoom models when models are being sliced.



When the progress bar under [Slice] button is 100%, it indicates the required printing time, the length and the weight of used filament. At the same time, [View Mode], [Save Mode] and [Print] are visible.

The required printing time is related to the size and complexity of 3D models. It is necessary to slice again when 3D models are moved and changed.

5.2.8 [View Mode] Function

[View Mode] is button group. This button group can show models view or Gcode view when it is changed.

After models have finished slicing, selecting [Model] shows the models.

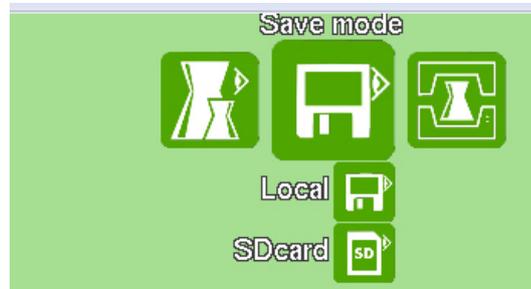


After models have finished slicing, selecting [GCode] shows the Gcode view.
Moving printing layers bar can view the printing path corresponding layer.

5.2.9 [Save Mode] Function

[Save Mode] is button group. This button group can save Gcode to different location.

When models slicing progress is finished, choosing [Local] can save the Gcode to the local device, such as your computer.



When models slicing progress is finished, choosing [SD card] can save the Gcode to the SD card. Before saving, make sure the following:

- a) Is SD card is inserted in SD-card socket;
- b) In menu bar, Setting -> Machine Setting -> Gcode flavor is **UltiGCode**.

The saving path will be shown when progress bar under [SD card] button is 100%.

When there is no available SD card or other removable storage device, the prompt will appear.

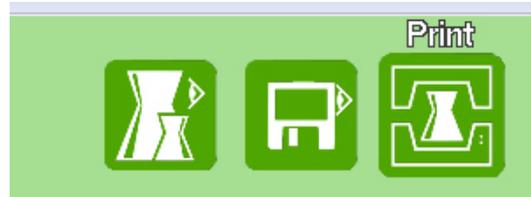
When there are some removable storage devices, the selection dialog will pop up to help you make a choice which device is used to save Gcode.

NOTE: If Gcode is saved to SD card, the Gcode flavor must be UltiGCode before slicing. Otherwise the printer will not recognize Gcode in SD card.

5.2.10 [Print] Function

When 3D models have finished loading and slicing, [Print] button can be used. Before [Print] button clicking, make sure the following:

1. Software has been connected to printer by USB ;
2. Before slicing, in menu bar Setting -> Machine Setting -> Gcode flavor is **RepRap**.



After clicking [Print] button, the printing dialog will pop up if there is no error message. There are two type printing dialog: Basic and Expert.

Basic printing dialog only shows the printing progress bar and temperature about print head and print bed

You can do some real-time operation to printer when using expert printing dialog. In menu bar, Setting -> Preference Setting -> Print window type can be changed if necessary.

5.3 Setting Mode Instruction

In setting mode, users can set printer parameters, slicing parameters and other parameters which will effect print result. There are simple setting mode and expert setting mode in software. It needs to slice again to make the changing parameters work.

5.3.1 Simple Setting Mode

In menu bar, selecting Mode -> Simple Setting Mode can make a simple setting to print .

NOTE: Filament setting in software must be consistent to printer used. If not, changing filament type in simple setting mode. If Gcode flavor is UltiGCode, there is no filament type setting in this mode.

5.3.2 Expert Setting Mode

In menu bar, selecting Mode -> Expert Setting Mode can make a expert setting to print, such as Basic setting, Advanced setting, Plugins and Start\End Gcode. Move the mouse to parameter setting box, you can see the corresponding parameter description. Click the [...] button, you can make the relative settings. Only a few parameters have [...] button.

In menu bar, Setting -> Expert Setting is available. This setting also will affect the printing effect.

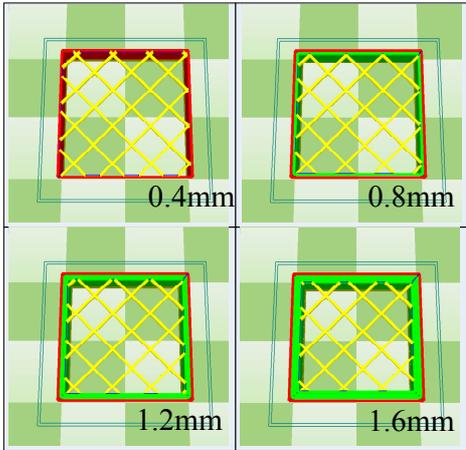
NOTE: Expert Setting is only available in Expert Setting Mode.

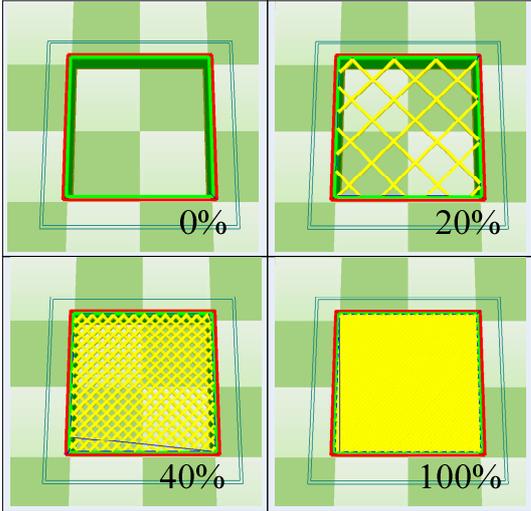
When printer is printing by USB or SD card, it needs different Gcode flavor. Choosing Setting -> Machine Setting changes Gcode flavor.

5.4 Setting Parameters Intrusion

Setting parameters are composed by basic settings, advanced settings, preference settings, machine settings and expert settings. The following lists will be described separately.

5.4.1 Basic Settings

Basic settings		
Preference		
Layer height	0.1mm	Normal quality are 0.1mm. High quality is 0.05mm. If you want to reduce print time, the layer height can be 0.2mm.
Shell thickness	0.8mm	Thickness of the outside shell in the horizontal direction. This is used in combination with the nozzle size to define the number of perimeter lines and the thickness of those perimeter lines. (If the shell thickness is 0.4mm and the nozzle is 0.4mm, there will be one perimeter line, and some necessary fill.) 
Enable retraction	True	Retract the filament when the nozzle is moving over a none-printed area. The print speed is travel speed. More retraction settings are in [Expert]->[Retraction].
Fill		
Bottom/Top thickness	1.0 mm	Having this value a multiple of the layer thickness makes sense. It controls the print layers of bottom and top. If layer thickness is 0.1mm, there are 10 layers in bottom/top to print. This thickness should not be too small, or it is not well to cover the bottom and top surface.

<p>Fill Density</p>	<p>20 %</p>	<p>This value controls the fill density when filling the insides of your print. For a solid part use 100%, for an empty part use 0%. A value around 20% is usually enough.</p> <p>This will not affect the outside of the print and only adjusts how strong the part becomes.</p> <p>The fill rate can increased to get strong print and extend the printing time for the price.</p> 
<p>Speed and Temperature</p>		
<p>Print speed</p>	<p>30 mm/s</p>	<p>This value is closely associated with the print quality and print time.</p> <p>You can improve print speed to reduce printing time, or reduce speed to improve print quality.</p>
<p>Printing temperature</p>	<p>200° C</p>	<p>This value is defined in terms of material properties.</p> <p>The printhead temperature is generally 200° C for PLA.</p>
<p>Bed temperature</p>	<p>50° C</p>	<p>This value is defined in terms of material properties.</p> <p>The buildplate temperature is generally 30 ~ 50° C for PLA.</p>
<p>Support</p>		
<p>Support Type</p>	<p>None Touching buildplate Everywhere</p>	<p>None: There is no support for print model.</p> <p>Touchingbuildplate: It only creates support where the support structure will touch the build platform.</p> <p>Everywhere: It creates support even on top of parts of the model.</p> <p>More support type settings are in [Expert Settings]->[Model Support].</p>

		
<p>Platform adhesion type</p>	<p>None Brim Raft</p>	<p>None: The print model does not need platform adhesion when the model has a large bottom area. More this type settings are in [Expert Settings]->[None Adhesion Type]. Adding the none type adhesion makes sure that the extruder extrudes filament smoothly when printing.</p> <p>Brim: It adds a single layer thick flat area around print model to prevent edge lifting. It is the recommended option. More this type settings are in [Expert Settings]->[Brim Adhesion Type].</p> <p>Raft: It generates a thick raster below the model and a thin interface between this and model. More this type settings are in [Expert Settings]->[Raft Adhesion Type].</p>

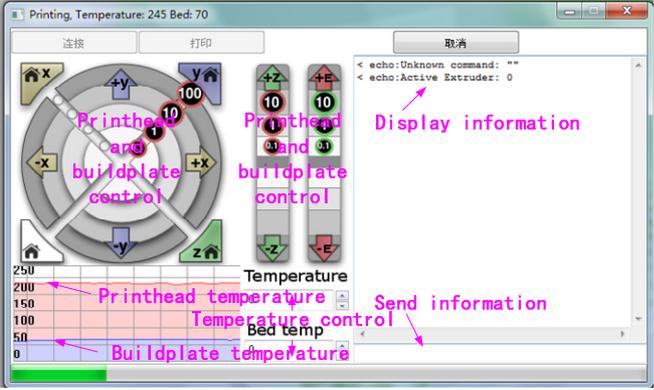
Filament		
Diameter	3.0mm	It is the material diameter.
Flow	110%	Flow compensation. The amount of material extruded is multiplied by this value.

5.4.2 Advanced Settings

Advanced Settings		
Machine		
Nozzle size	0.4mm	This value is important. It is used to calculate the line width of the infill, the amount of outside wall lines and thickness for the wall thickness.
Retraction		
Speed	24mm/s	Speed at which the filament is retracted. A higher retraction speed works better. But a very high retraction speed can lead to filament grinding.
Distance	4.5mm	Distance at which the filament is retracted. Set at 0 for no retraction at all. The actual retraction distance needs to be set according to the properties of the material and the model. The reference value is 4.5mm.
Quality		
Initial layer thickness	0.3 mm	A thicker bottom layer makes model sticking to the platform easier. Set as 0 for having the bottom layer thickness the same as the other layers.
Initial layer line width	100%	This value affects the adhesion effect of the bottom layer on the platform.
Cut off object bottom	0.0 mm	This value is the height that the model is sunk into the platform. When the model has a small bottom area, this value can make begin printing at a larger area. Or you only want to print the model at a certain height.
Dual extrusion overlap	0.15 mm	This value bonds the different colors together.
Speed		
Travel speed	150 mm/s	Speed at which travel moves are done.
Bottom layer speed	20 mm/s	In order to get a good adhesion result, it is recommend to print bottom layer at a lower speed than the normal print speed.
Infill speed	0 mm/s	Set as 0 for making the same speed as the print speed

		in [Basic Settings]. This value can be increased to shorten printing time.
Outer shell speed	0 mm/s	Set as 0 for making the same speed as the print speed in [Basic Settings]. Printing the outer shell at a lower speed improves the final skin quality. However, having a large difference between the inner shell speed and the outer shell speed will effect quality in a negative way.
Inner shell speed	0 mm/s	Set as 0 for making the same speed as the print speed in [Basic Settings]. This value is better in the range of outer shell speed and infill speed.
Cool		
Minimal layer time	5 sec	This value is the shortest time spent in a layer. It makes print layer to cool down before the next layer is put on top. If the layer will be placed too fast the printer will slow down to make sure it has spent at least this amount of seconds printing this layer. And turn on the fan for cooling if needed.
Enable cooling fan	True	Turning on the cooling fan is in favor of model shaping.

5.4.3 Preference Settings

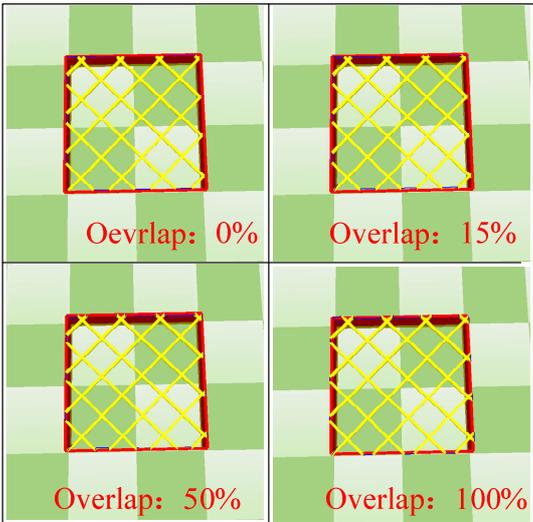
Preference Settings		
Print Window		
Print window type	Basic Pronterface UI	<p>Basic:</p>  <p>Pronterface UI:</p> 
Color		
Model color		<p>Change the color that model displays.</p> 
Filament settings		
Density	1240 kg/m3	These values can be used to calculate the filament weight and cost when some model is printed.
Cost	0 price/kg	
Cost	0 price/m	
SD-card setting		
Auto detect SD card driver	True	Set true as default.

5.4.4 Machine Settings

Machine Settings		
Machine		
E-Steps per 1mm filament	0	For device debugging, users do not need modify it.
Maximum width	230 mm	They are the inherent properties of the machine.
Maximum depth	225mm	
Maximum height	205mm	
Extruder count	1	The printer only support single printhead now.
Heated bed	True	
Machine center 0,0	False	The default setting set the left front corner as 0,0. When it is true, set bed center as 0,0.
Build area shape	Square	The default setting.
Gcode flavor	RepRap UltiGCode	RepRap: This flavor has a start and end Gcode, printed through USB or SD card. UltiGcode: This flavor uses the machine parameters for starting and ending a printing.
Printhead size		
Head size towards X min	40.0 mm	
Head size towards Y min	10.0 mm	
Head size towards X max	60.0 mm	
Head size towards Y max	30.0 mm	
Printhead gantry height	55.0 mm	
Communication settings		
Serial port	AUTO	The default settings are AUTO.
Baudrate	AUTO	

5.4.5 Expert Settings

Expert settings		
Retraction		
Minimum travel	1.5 mm	Minimum travel distance needed for a retraction to happen. When travel distance is less than this value, there is no retraction. It is better to prevent a lot of retractions in a small area, leading to material grinding.
Enable combing	True	
Minimal extrusion before retracting	0.02 mm	Minimum extrusion needed for a retraction to happen. When actual extrusion is less than this value, there is no retraction. It is better to prevent a lot of retractions in a small area, leading to material grinding.
Z hop when retracting	0.0 mm	When a retraction is done, the head is lifted by this amount to travel over the print model.
None adhesion type	Settings for platform adhesion: None.	
Line count	1	Set 0 for disabling the none adhesion type. This type is used for pre-extrusion and checking whether the platform is suitable for printing. Small model can add some lines if needed.
Start distance	3.0 mm	The minimal distance between the line and the model first layer. Multiple lines will be put outwards from this distance.
Minimal length	150.0mm	The minimal length of the line. If this minimal length is not reached it will add more lines to reach this minimal length. If the line count is set to 0 this is ignored.
Cool		
Fan full on at height	5.0mm	The height at which the fan is turned on completely. For the layer below this height, the fan speed is scaled linearly with the fan off at layer 0.
Fan speed min	40%	When the fan is turned on, it is enabled at this speed. If cool slows down the layer, the fan is adjusted

		between the min and max speed. Minimal fan speed is used if the layer is not slowed down due to cooling.
Fan speed max	100%	When fan is turned on, the fan speed is adjusted between the min and max speed. If print speed slows down, the fan speed is max. The max can be set as 200%.
Minimum speed	10mm/s	The minimal layer time can cause the print to slow down so much that the nozzle starts to ooze. The minimal feedrate protects against this. Even if print speed gets slowed down it will never be slower than this minimal speed.
Cool head lift	False	Lift the head if the minimal speed is hit because of cool slow down, and wait the extra time so the minimal layer time is always hit.
Infill		
Solid infill top	True	Create a solid top surface, if set to false the top is filled with the fill density in [Basic settings]
Solid infill bottom	True	Create a solid bottom surface, if set to false the bottom is filled with the fill density in [Basic settings]
Infill overlap	15%	<p>Amount of overlap between the infill and the walls. This ensures that the walls connect firmly to the infill.</p> 
Support		
Structure	Grid	The type of support structure. Grid is very strong and

	Lines	can come off in 1 piece, however, sometimes it is too strong. Lines are single walled lines that break off one at a time. Which is more work to Delete, but as it is less strong it does work better on tricky prints.
Overhang angle for support	60 deg	The minimal angle that overhangs need to have to get support. With 90 degree being horizontal and 0 degree being vertical.
Fill amount	15 %	Amount of infill structure in the support material. Less material gives weaker support which is easier to Delete.
Distance X/Y	0.7 mm	Distance of the support material from the print in the X/Y directions. According to the model property, set the distance to make the support does not stick to the model.
Distance Z	0.15 mm	Distance from the top/bottom of the support to the model. A small gap here makes it easier to Delete the support but makes the model a bit uglier.
Model outer wall process		
Spiralize the outer contour	False	Spiralize is smoothing out the Z moves of the outer edge. This will create a steady Z increase over the whole model. This feature turns a solid object into a single walled model with a solid bottom.
Only follow mesh surface	False	Only follow the mesh surfaces of the 3D model, do not do anything else. No infill, no top/bottom.
Brim adhesion type	Settings for platform adhesion: Brim.	
Brim line amount	20	The amount of lines used for a brim. More lines means a larger brim which sticks better, but this also makes model print area smaller.
Raft adhesion type	Settings for platform adhesion: Raft.	
Extra margin	5.0 mm	If the raft is enabled, this is the extra raft area around the model which is also rafted. Increasing this margin will create a stronger raft while using more material and leaving less area for print.

Line spacing	3.0 mm	The distance between the centerlines of the raft line.
Base thickness	0.3 mm	The thickness of base layer lines which are put down.
Base line width	1.0 mm	The width of base layer lines which are put down.
Interface thickness	0.27 mm	Thickness of each surface layer.
Interface line width	0.4 mm	Width of the lines for each surface layer.
Airgap	0.22 mm	Gap between the last layer of the raft and the first printing layer.
Surface layers	2	Amount of surface layers put on top of the raft, these are fully filled layers on which the model is printed.
Fix horrible	As with all [Fix horrible] options, results may vary and use at your own risk.	
Combine everything (Type-A)	True	This expert option adds all parts of the model together. The result is usually that internal cavities disappear. Depending on the model this can be intended or not. Enabling this option is at your own risk. Type-A is dependent on the model normals and tries to keep some internal holes intact.
Combine everything (Type-B)	False	This expert option adds all parts of the model together. The result is usually that internal cavities disappear. Depending on the model this can be intended or not. Enabling this option is at your own risk. Type-B ignores all internal holes and only keeps the outside shape per layer.
Keep open faces	False	This expert option keeps all the open bits of the model intact. Normally the software tries to stitch up small holes and Delete everything with big holes, but this option keeps bits that are not properly part of anything and just goes with whatever is left. This option is usually not what you want, but it might enable you to slice models otherwise failing to produce proper paths.
Extensive stitching	False	Extensive stitching tries to fix up open holes in the model by closing the hole with touching polygons. This algorithm is quite expensive and could introduce a lot of processing time.

6. Maintenance and Frequently Asked Questions

6.1 Precautions

6.1.1. Printer can not continue printing when power off is on. It is recommended to place printer where people walk less and external power can not easily be plugged. Make sure printer is in a stable state on power.

6.1.2. When printing is finished, Delete print objects and clean print platform. Make sure there is no residual filament on it.

6.1.3. When printing is finished, clean the print nozzle. Make sure there is no filament to block the nozzle.

6.2 Maintenance

6.2.1. Change Filament

Go to [MATERIAL] tab on the main menu of the display. Press the button, next go to [CHANGE]. The printer will heat up, so the filament can be Deleted automatically. Coil the Deleted filament on filament spool. Using tape or other material fix the filament head to prevent filament scattered.

6.2.2. Print Platform Clean

After a lot of printing, there sometimes can be a little excess glue stuck to the glass build plate. This can cause an uneven print surface and it is recommended to clean it once in a while.

Make sure the print is powered off. Carefully slide the front glass clamps and click them off carefully. Then take the print platform out.

NOTE: Before operation, make sure the build platform is at the bottom of printer. This is prevent you not damage print head or Z-axis.

To dissolve and clean off the excess glue by clean water. Then put it back into its original position. To be sure that your next print will be successful you might want to level your build platform. You can do this under [MAINTENANCE].

NOTE: The build platform is position on an electrical heating area so make sure it really is dry.

6.2.3. Filament Feeder Clean

After a lot of printing, the wheel in the filament feeder can accumulate small plastic particles. You can clean this by blowing air on the wheel or using a simple brush.

6.2.4. XYZ-Axis Lubricating

After approximately half year the threaded rod on the z-stage has to be lubricated. A green grease is include with printer. This grease is the lubrication which has to be used on the threaded z-stage rod.

Make sure you spread 10 drops of grease over the entire exposed threaded rod. With you next print on printer the thread will grease itself by moving up and down.

If you notice small wavers in the outside surfaces of your 3D printed objects. It is recommended to add a single drop of sewing machine oil onto the X and Y steel axes bars. Delete and excess oil. This will improve the smooth movement of your printer.

6.3 Frequently Asked Questions

6.3.1. In printing process, the filament will not be well adhered to the print platform.

Cause: ① The build platform is not properly leveled.
 ② The glue layer is not thick enough.

Solution: To ①, follow the instructions under [MAINTENANCE] and then [BUILD-PLATE] to level it again.

 To ②, clean the glue layer and make the layer again.

2. It is difficult to take off the print model from the build plate.

You can heating the buildplate to 30-40° C for helping to take off the print model from buildplate.

3. The print head extrudes the filament brokenly.

The print head may be blocked by dust or filament. PLA-type material must not be heated to high temperature. The material becomes carbonization easily when temperature exceeds 230 °, and further clogs the nozzle. You can use the slightly smaller than the diameter of 0.4mm metal devices to dredge nozzle and observe spinning. If the situation does not improve, the further processing can be done. Delete the nozzle guidewire tube, take a short material, top whittled pencil, manually squeeze into the nozzle, extrusion blockage, repeatedly until smooth extrusion nozzle materials.

Please contact us when you can not solve the problem of nozzle. The printer needs for repair or replacement.

Contact: Log www.micolor.cn.

6.3.4. Filament flow stops and filament is ground down by the feeder.

Cause: ① Nozzle could be blocked or partially blocked.
 ② Too much pressure is put on the filament by the feeder.
 ③ Too little pressure is put on the filament by the feeder, causing the filament to slip.

Solutions: To ①, send the Question 3 solutions.

 To ②, release the pressure on the feeder a bit by adjusting the build plate screw clockwise.

 To ③, put more pressure on the feeder a bit by adjusting the pressure screw counter clockwise.

6.3.5. Printer display only shows “ERROR - STOPPED”, “Temp Sensor”.

Cause: A problem with the temperature measurement has been detected, the printer has been switched off for safety reasons.

Solution: Please contact us for electronics inspection and maintenance.

Contact: www.micolor.cn

6.3.6. If the object has not yet finished printing and the filament has been used up, can filament be added to continue printing?

The printer can not continue to print the model. The left filament can not be held by printer feeder, and it can not reach the print head. Then the filament is not able to be extruded from the print head. It is better to estimate the printing amount of object before printing. Make sure there is enough filament for printing.

6.3.7. In printing process, the object appears print layer offset.

Cause: X or Y axis fastening screws become loose.

Solution: Adjust the X or Y axis tightness of fastening screws. Print to see the effect again.

If the above operation can not solve the problem, please contact us to repair or replace the X or Y axis.

Contact: www.micolor.cn

7. Warranty and License

The 3D printer produced by our company is guaranteed for user in its one-year warranty period: the device hardware and material (complimentary tools and accessories not included) from the date of purchase of are the absence of quality defects. If the company receives notice of such defects during the warranty period, the company will choose to repair or replace defective equipment indeed. Replacement device may be a new product or performance is not less than the original equipment products.

In the case of proper installation and use, during the warranty period, free replacement can not run the software. If the Company is unable to repair as warranted, customer will be entitled to demand a replacement during the warranty period, or even return. Warranty does not apply to equipment defects due to the following reasons:

- maintenance or calibration, improper or inadequate.
- Use non-software, interfaces, components, or materials provided by the Company.
- disassemble the equipment.
- environmental norms specified by the device operation.
- improper site preparation or maintenance.
- equipment damage and other non-company product quality problems caused.

Within the local extent permitted by law, the above is the only warranty in addition no other express or implied, written or oral, or the terms of the warranty, and the company is not for any particular purpose of merchantability, satisfactory quality and fitness for hint warranty terms. Some countries / regions, states, autonomous regions or provinces do not allow limitations on implied warranty, within the local extent permitted by law, this warranty statement is the only proprietary compensation. Apart from the above, the Company or its suppliers be liable for any loss of data under any circumstances or direct, special, incidental, consequential (including lost profits or loss of data) or be responsible for other losses, whether it is based on contract, tort or other reasons. Some countries / regions, the law states, autonomous regions or provinces do not allow the exclusion or limitation of incidental or consequential damages, so the above limitation or exclusion may not apply to users.

In addition to the extent permitted by law, this warranty statement does not exclude nor limit or modify the mandatory statutory rights of this equipment sales to such customers when applicable.

8. Contact Us

Welcome to visit our company website www.micolor.cn to leave your views and expectation of technical support.