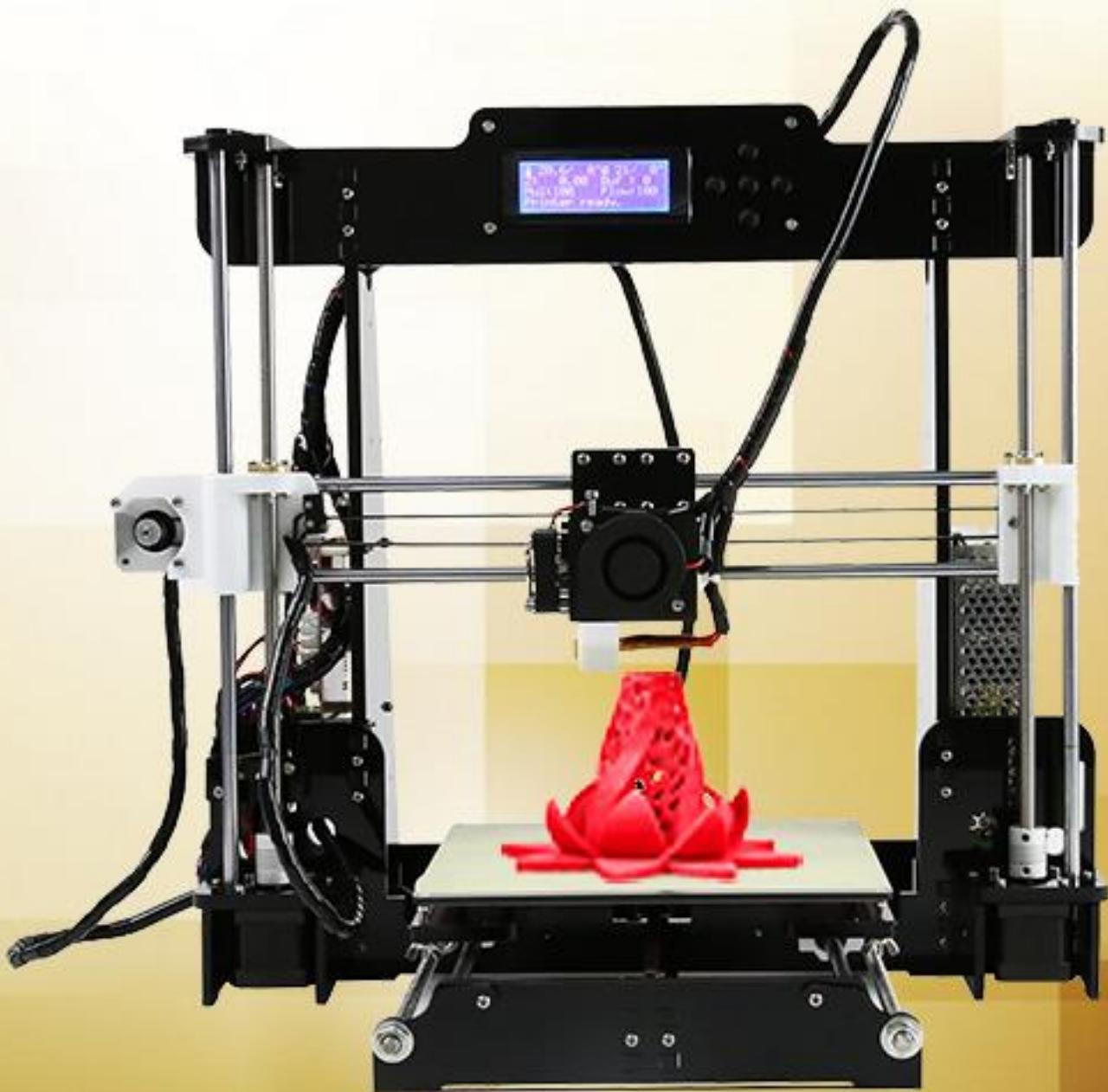


Operation Instruction

Model:A8



Contents

INTRODUCTION	2
A. Security Considerations.....	3
B. Product Details	4
1.Specifications.....	4
2.Machine parts	4
3.Extruder drawing.....	5
4.Tool List	6
C. Cura Software	9
1.Installation of Cura14.07.....	9
1.1 File location in the TF card	9
2. Cura Setting.....	18
2.1 Clear platform.....	18
2.3 Layer height settings	23
D. Printing Operation	46
1. Introduction of Display	46
2. Install Filament.....	56
2.1 Set Preheat Mode	56
2.2 Filament Installation.....	57
2.3 Pull out filament	59
3. Platform Adjustment	60
4. Printing	62
E. FAQ and Solution	65
1. Z Axis Ajustment.....	655
2. Nozzle blocking.....	67
3. FAQ	69
F. Maintenance.....	70
G. Maintenance policy	71

INTRODUCTION

A8 FDM 3D printer can print CAD 3D printer model to real . A8 uses Acrylic to build its frame while it uses linear bearings , belts and threaded rods to build X , Y , Z axis . It enables A8 to print steadily with no vibration .

Note:

1. All statement included in this Instructions have been checked carefully , if any typographical errors or misunderstanding , we have the final interpretation .
2. No notification if any update .

A. Security Considerations

To avoid danger when using 3D printer , please pay attention to precautions below .

◎ Danger

During Operation , the maximum temeprature of nozzle can be 260 °C while hotbed can be 100 °C . For your safety , during printing or cooling down , do not touch the nozzle , hotbed and models under printing . Power works at 110V/220V 50HZ AC and supply ground needed . Do not use other power supply , or it may cause components damage , fire or electric shock . And we take no responsibility for this .

△ Warning

We suggest wering protective goggles when removing auxiliary support materials . Some filaments will emit slight irritant gases , so we suggest to use 3D printer in a ventilated environment .

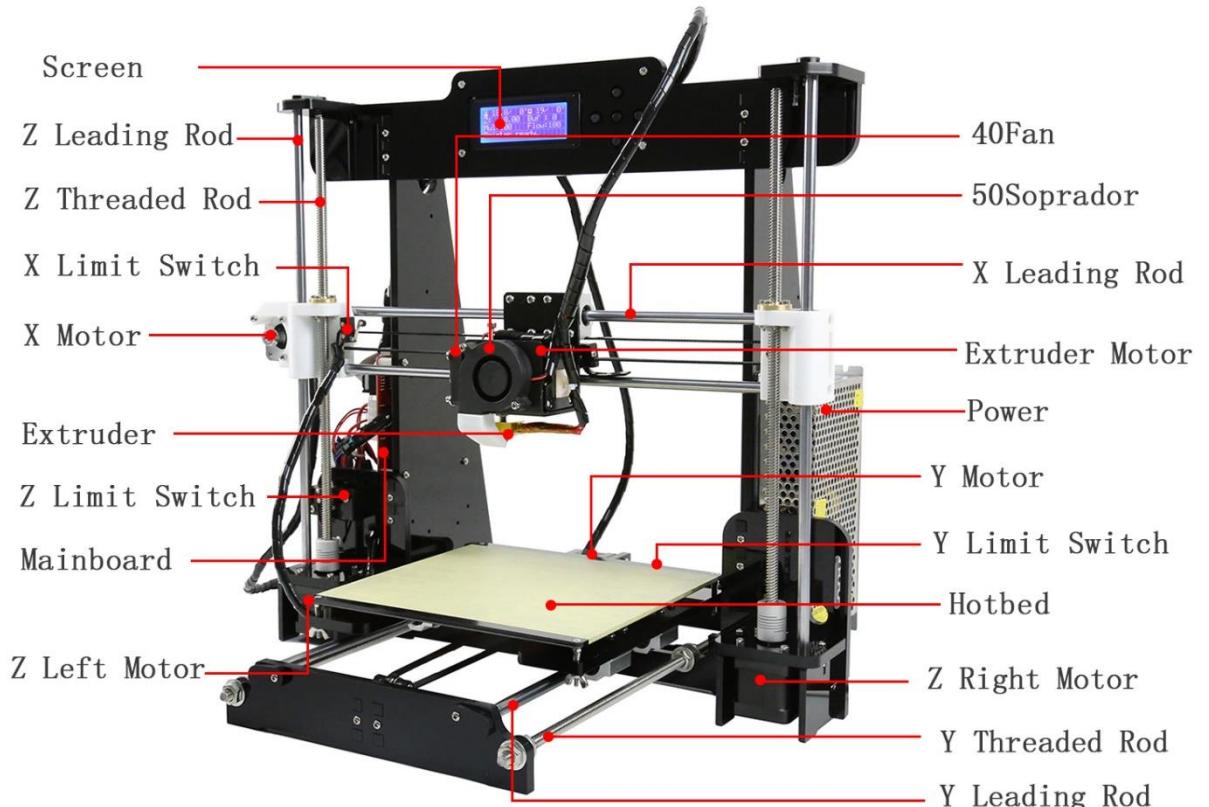
Note: ABS filament will emit a bit toxic gases when it melts .

B. Product Details

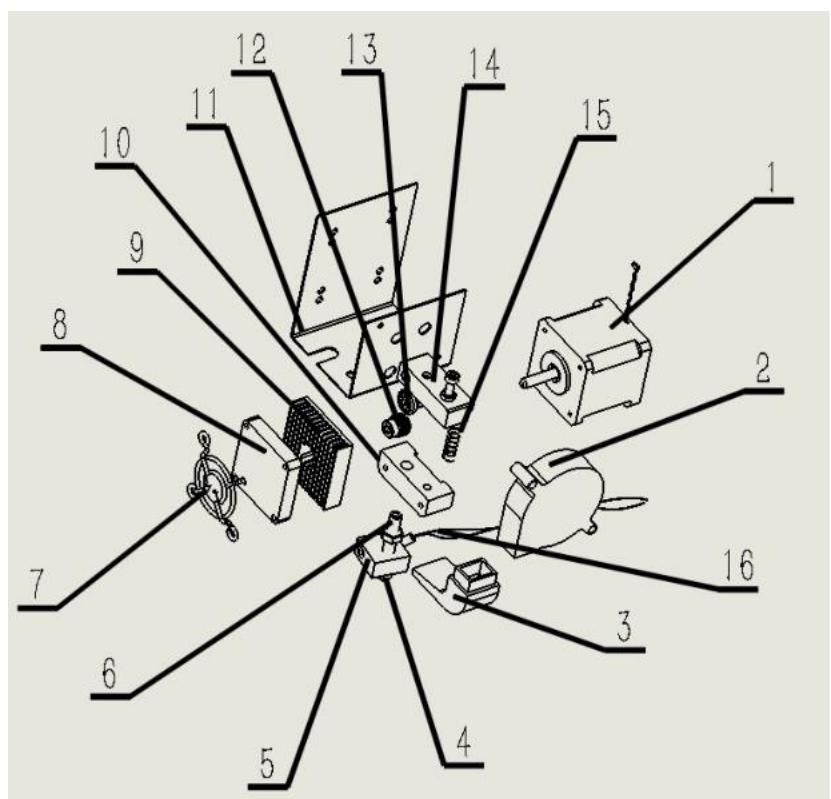
1.Specifications

Model: A8	Nozzle diameter: 0.4mm
Layer thickness: 0.1-0.3mm	Machine size: 500*400*450mm
Printing speed: 10-120mm/s	Machine weight: 7.5KG
X Y axis position accuracy: 0.05mm	Packing size: 510*345*215mm
Z axis position accuracy: 0.015m	Gross weight: 9.2KG
Printing material: ABS,PLA	Build size: 220*220*240mm
Material tendency: PLA	LCD screen: Yes
Filament diameter : 1.75mm	Offline printing: SD CARD
Software language: Multi-Language	File format: STL、G-Code、OBJ
Function of support: automatically	OS: windows(linux、mac)
Software: Cura	Working condition: 10-30°C, Humidity 20-50%

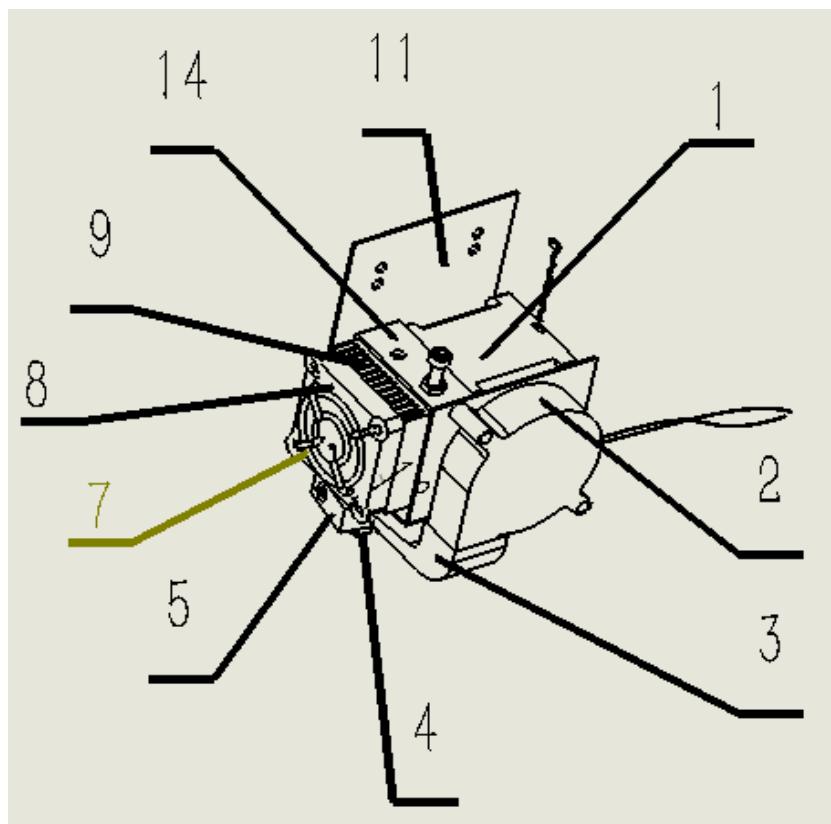
2.Machine parts



3.Extruder drawing

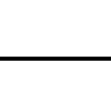


No.	Part Name	Quantity
1	Extruder Motor	1
2	Blower	1
3	Wind Mouth	1
4	Nozzle(0.4mm)	1
5	Heating Block	1
6	Throat	1
7	Fan Cover	1
8	Fan	1
9	Heat Sink	1
10	Extruder Seat	1
11	Bend Parts	1
12	Brass Wheel	1
13	U-Bearing	1
14	Briquetting	1
15	Spring	1
16	Heating Pipe	1



4. Tool List

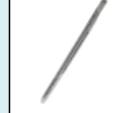
Item	Picture	Name	QTY	Check	Item	Picture	Name	QTY	Check
1-1		Hot bed fixed aluminum plate	1		2-1		Mainboard	1	
1-2		220mm*220mm*3mm Hot bed	1		2-2		Left Z axis nut support	1	
1-3		Plastic nippers	1		2-3		Right Z axis nut support	1	
1-4		1.5M Power line	1		2-4		Wind mouth	1	
1-5		5mm*160mm Screwdriver	1		2-5		1.6M Belt	1	
1-6		Four parts below in this bag	1		2-6		1.5M USB wire	1	

1-6-1		40*10 Fan	1		2-7		5015 Air blower	1	
1-6-2		40*11 Cooling fin	1		2-8		Y axis belt bearing support	1	
1-6-3		Fan cover	1		2-9		Seven parts below in this bag	1	
1-6-4		M3*45 Screw 2pcs M3 Spacer 8pcs	1		2-9-1		Z axis Limit switch fixed plate	2	
1-7		Screw bag include below screws	1		2-9-2		Y axis motor support	1	
1-7-1		M3*18 screw 52pcs	1		2-9-3		Y axis Limit switch fixed plate	1	
1-7-2		M3 Nut 64pcs	1		2-9-4		Y axis belt fixation clamp	2	

1-7-3		M8 Nut 16pcs M8 Spacer 12pcs	1		2-9-5		Guide rod back up plate	6	
1-7-4		M4*8 screw 28pcs M4*14 screw 4pcs	1		2-10		Side support plate	2	
1-7-5		M3*30 screw 14pcs	1		2-11		Filament support plate	2	
1-7-6		M3*12 screw 15pcs	1		2-12		Filament support plate connecting plate	2	
1-7-7		M2*12 screw 4pcs wing nut 4pcs Spring 4pcs	1		2-13		Screen baffle plate	1	
1-7-8		M3*25 screw 3pcs	1		2-14		Z axis motor support Plate	4	
1-8		Three parts below in this bag	1		2-15		8GB TF card and card reader	1	

1-8-1		Wire 70CM	1		3-1		Bottom support plate	1	
1-8-2		Limit switch A 20CM Limit switch B 55CM Limit switch C 70CM	3		3-2		Top support plate	1	
1-8-3		Pillar washer M3*7 4pcs Pillar washer M3*15 4pcs	8		3-3		Back plate	1	
1-9		Three parts below in this bag	1		3-4		Front plate	1	
1-9-1		3mm*130mm Screwdriver	1		3-5		Support plate lock plate	2	
1-9-2		Hex wrench M1.5 Hex wrench M2 Hex wrench M2.5 Hex wrench M3	4		3-6		Z axis motor fixed plate	2	
1-9-3		Open spanner	1		3-7		Y axis motor fixed plate	1	

1-10		Four parts below in this bag	1		3-8		Extruder	1	
1-10-1		4.5M Winding pipe	1		3-9		X axis motor	1	
1-10-2		Belting	10		3-10		Y axis motor	1	
1-10-3		R clip	3		3-11		Z axis motor	2	
1-10-4		Locating piece	2		3-12		Linear bearing	7	

1-11		LCD 2004 screen	1		3-13		Guide rod 436mm 2pcs Guide rod 380mm 4pcs	6	
1-12		X Motor line 40CM Y Motor line 40CM Left Z Motor line 40CM Right Z Motor line 90CM Extruder Motor line 90CM	5		3-14		T type lead screw M8*345mm 2pcs Threaded rod M8*400mm 2pcs Threaded rod M8*150mm 1pcs	5	
1-13		Heat bed line 90CM	1		3-15		Power Supply	1	

C. Cura Software

1. Installation of Cura14.07

a: Where can I find the software?

- 1) SD card with shipment; 2) download from Internet;

b: Installation process

- 1) From SD card with shipment

Insert SD card and open the file

1.1 File location in the TF card

1) Insert SD card , open the file

名称	修改日期	类型	大小
Installation Instruction	2016/7/7 星期四 ...	文件夹	
Print Model STL	2016/6/22 星期三 ...	文件夹	
Software	2016/7/7 星期四 ...	文件夹	
Test file GCODE	2016/6/22 星期三 ...	文件夹	
Tool List&other pictures	2016/7/7 星期四 ...	文件夹	

名称	修改日期	类型	大小
CH340G Drive	2016/7/7 星期四 ...	文件夹	
Cura 14.07	2016/7/7 星期四 ...	文件夹	
RepetierHost_1_0_5	2016/7/7 星期四 ...	文件夹	

名称	修改日期	类型	大小
Cura download link.txt	2016/7/1 星期五 ...	文本文档	1 KB
Cura_14.07.exe	2015/8/11 星期二 ...	应用程序	18,377 KB

2) Download from Internet

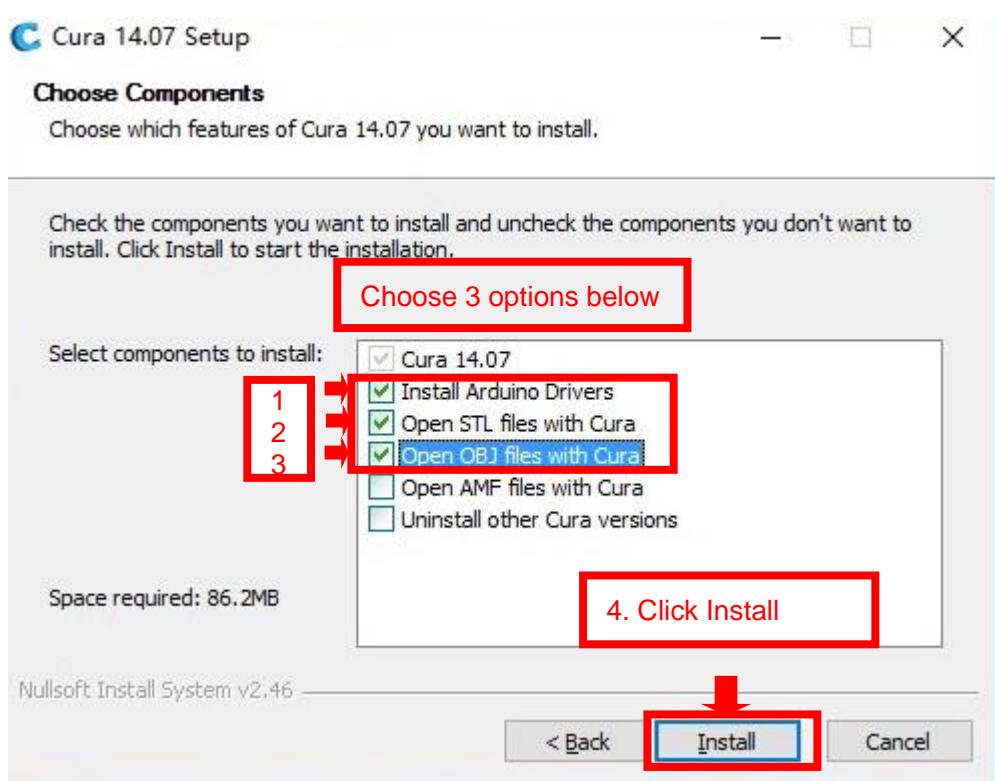
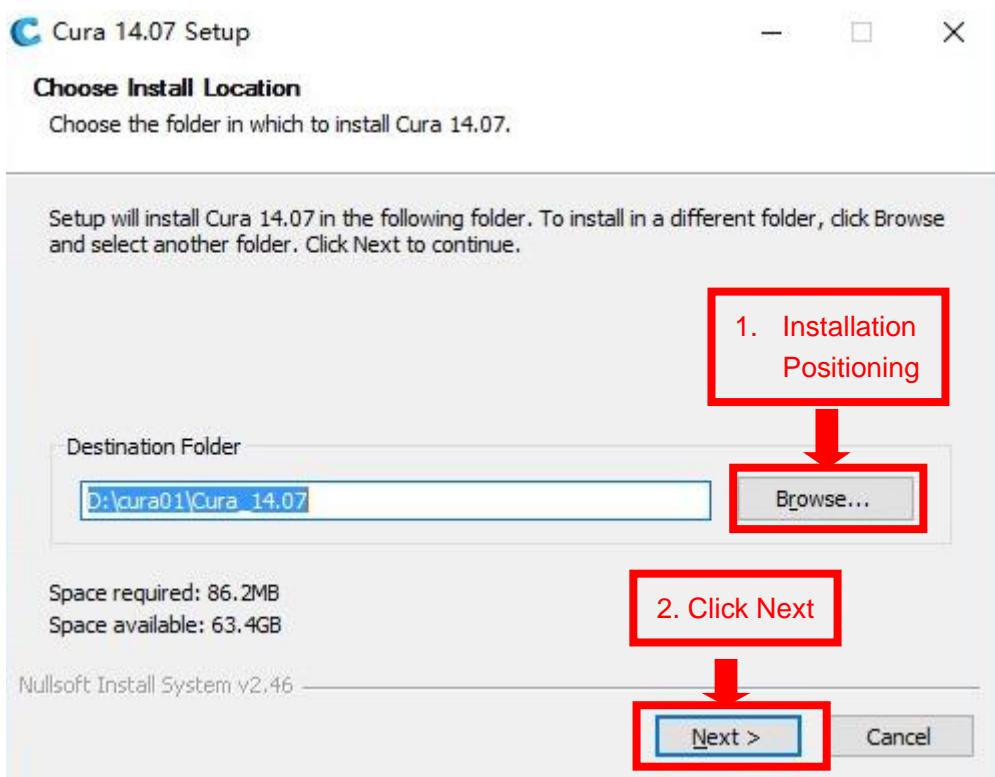
Official Website: <https://ultimaker.com/en/cura-software/list>

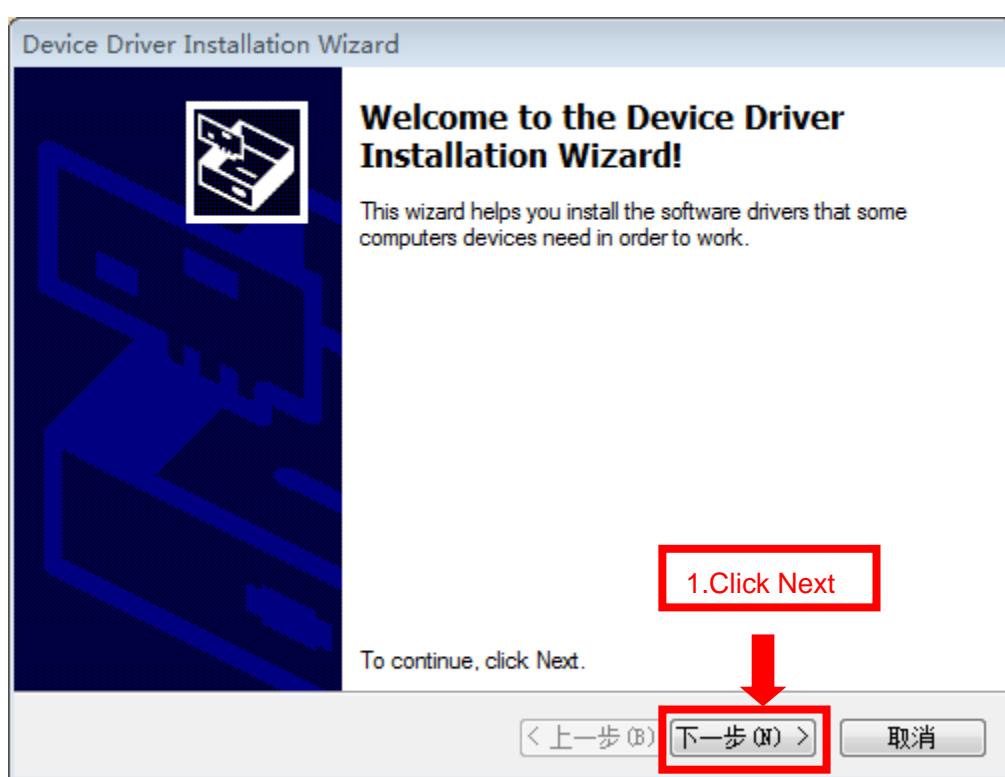
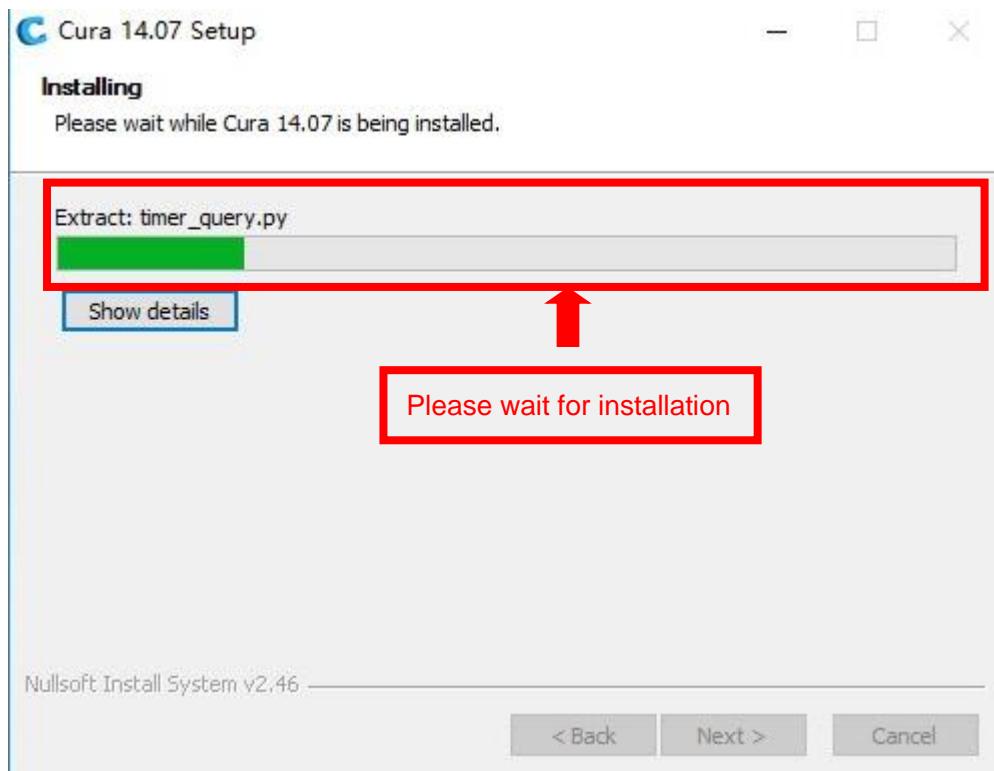
Choose corresponding software to download

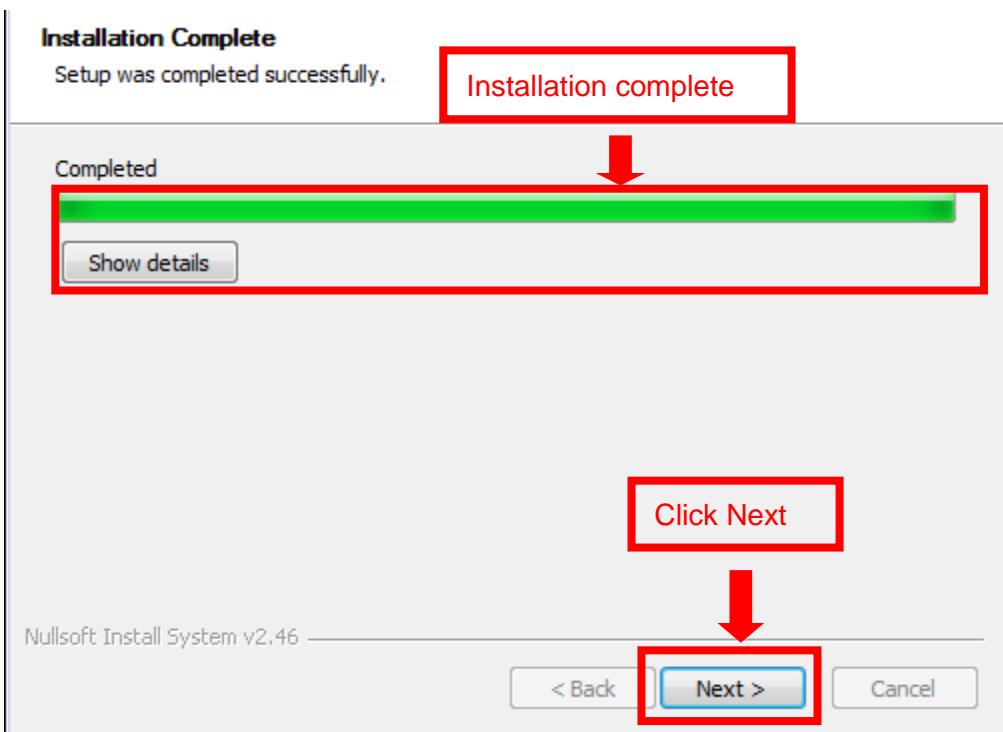
WINDOWS

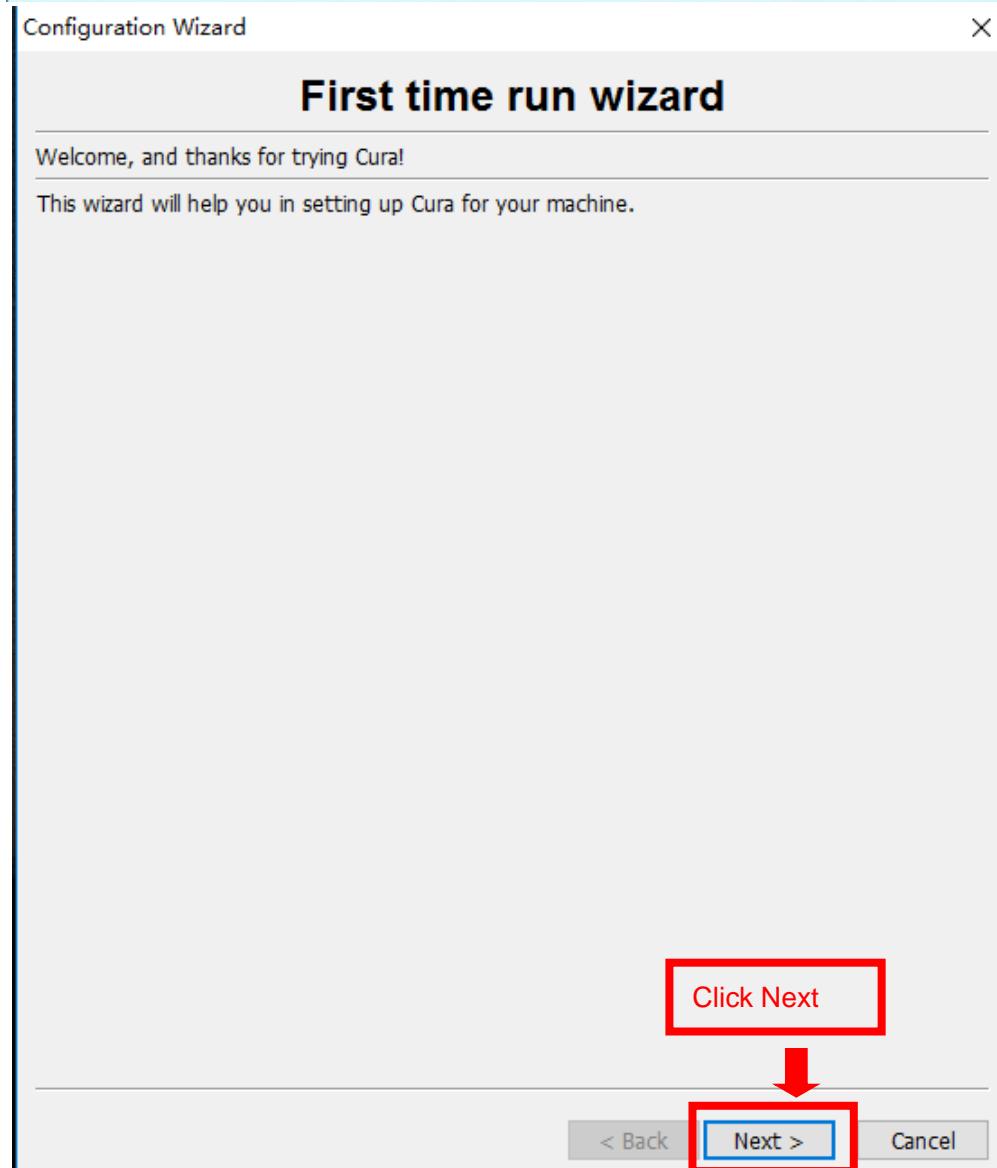
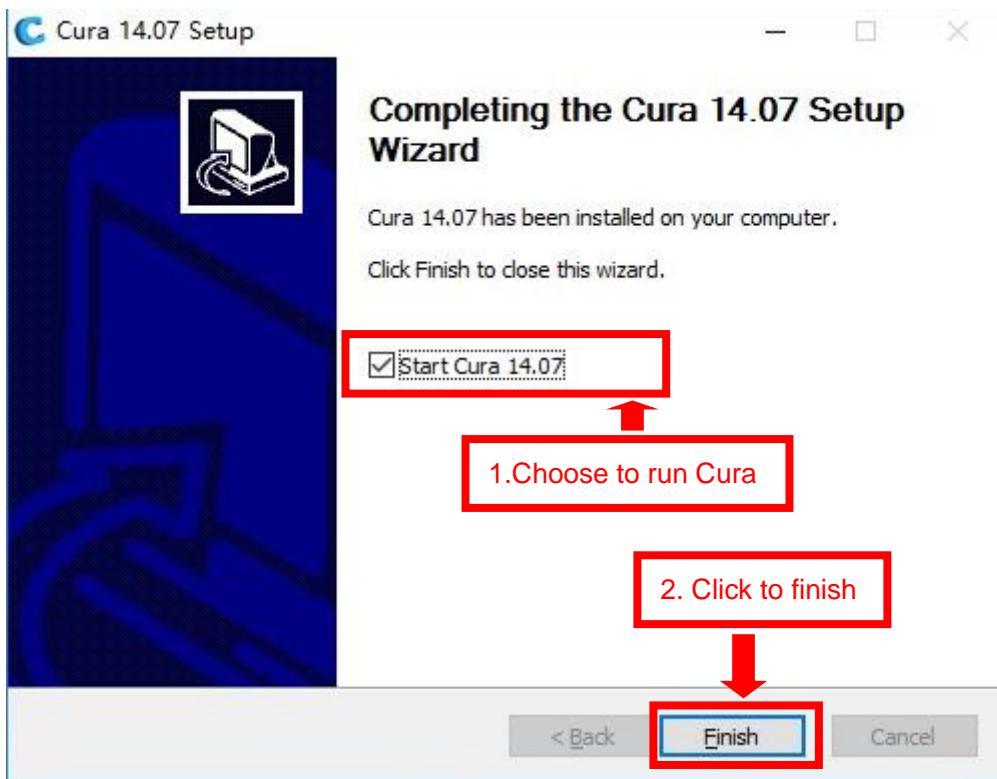
Version: 2.1.2 32 bit	Release date: 6/7/16
Version: 2.1.2 64 bit	Release date: 6/7/16
Version: 15.04.6	Release date: 6/7/16
Version: 15.04.5	Release date: 3/17/16
Version: 15.04.4	Release date: 1/5/16
Version: 15.04.03	Release date: 11/4/15
Version: 15.04.2	Release date: 7/28/15
Version: 15.04	Release date: 4/15/15
Version: 15.02.1	Release date: 2/19/15
Version: 15.01	Release date: 1/30/15
Version: 14.12	Release date: 12/15/14
Version: 14.09	Release date: 9/19/14
Version: 14.07	Release date: 7/3/14
Version: 14.06	Release date: 6/16/14
Version: 14.03	Release date: 3/17/14
Version: 14.01	Release date: 1/10/14
Version: 13.12	Release date: 12/23/13
Version: 13.11	Release date: 11/22/13
Version: 13.10	Release date: 10/18/13
Version: 13.06.4	Release date: 6/26/13
Version: 13.04	Release date: 4/26/13
Version: 13.03	Release date: 3/8/13
Version: 12.12	Release date: 12/24/12
Version: 12.11	Release date: 11/12/12
Version: 12.10	Release date: 11/8/12

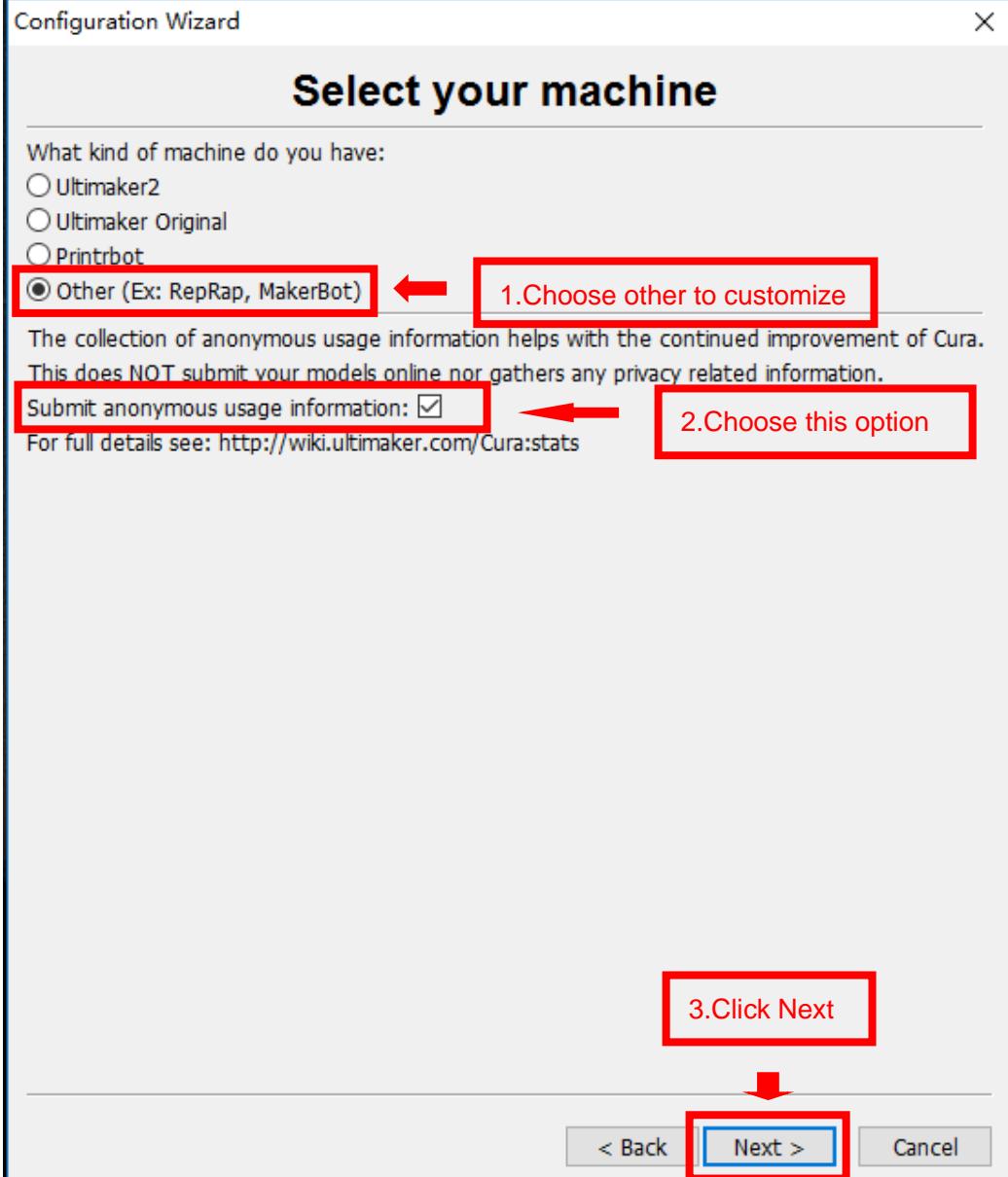
b. Software Installation Process











Other machine information

The following pre-defined machine profiles are available

Note that these profiles are not guaranteed to give good results,
or work at all. Extra tweaks might be required.

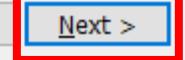
If you find issues with the predefined profiles,
or want an extra profile.

Please report it at the github issue tracker.

- BFB
- DeltaBot
- MakerBotReplicator
- Mendel
- Prusa Mendel i3
- punchtec Connect XL
- Custom... 

1. Choose to customize

2. Click Next

< Back  Next > Cancel

Custom RepRap information

RepRap machines can be vastly different, so here you can set your own settings.

Be sure to review the default profile before running it on your machine.

If you like a default profile for your machine added,
then make an issue on github.

You will have to manually install Marlin or Sprinter firmware

Machine name

A8

This is A8 parameter

You can customize name here.

Machine width (mm)

220

We default these data .

Machine depth (mm)

220

You can fill in according to
actual condition

Machine height (mm)

240

Nozzle size (mm)

0.4

Heated bed

Choose this option

Bed center is 0,0,0 (RoStock)

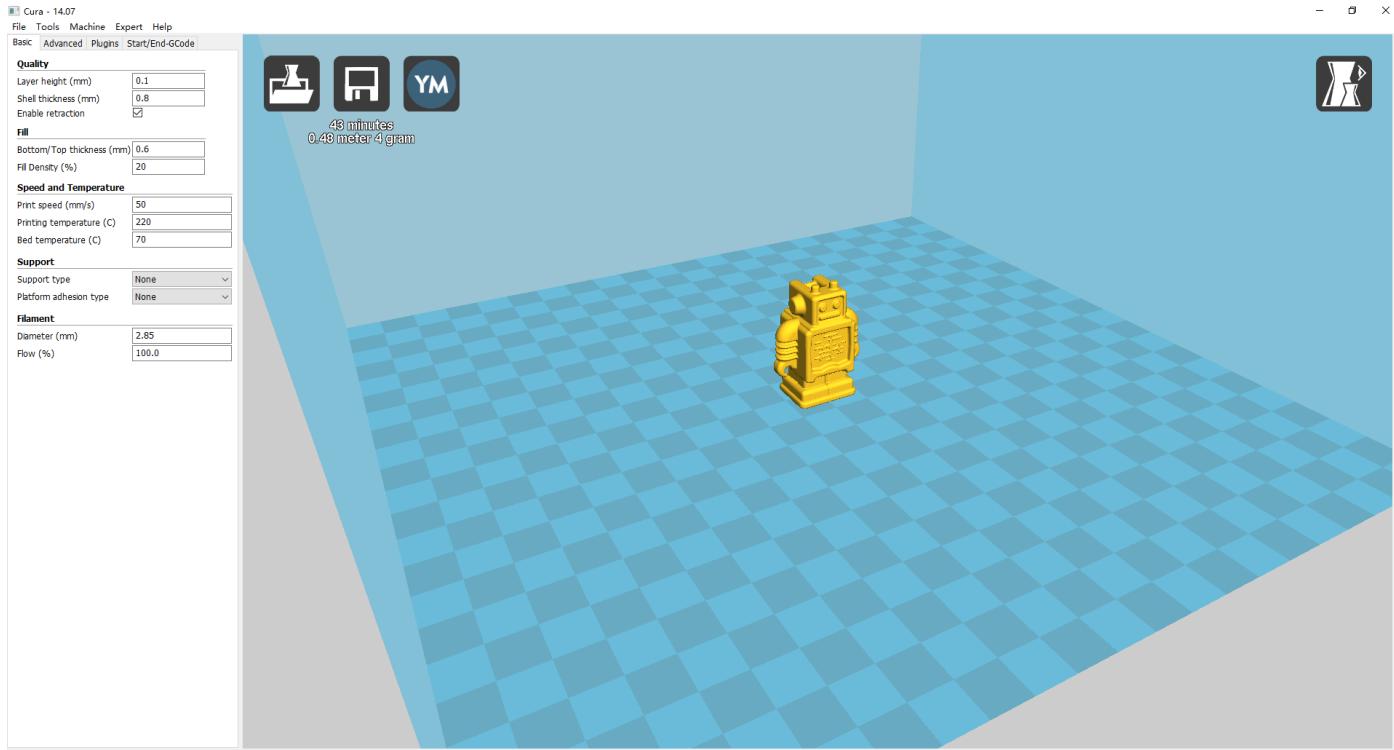
DO NOT choose this option !!!

1.Click to finish

< Back

Finish

Cancel



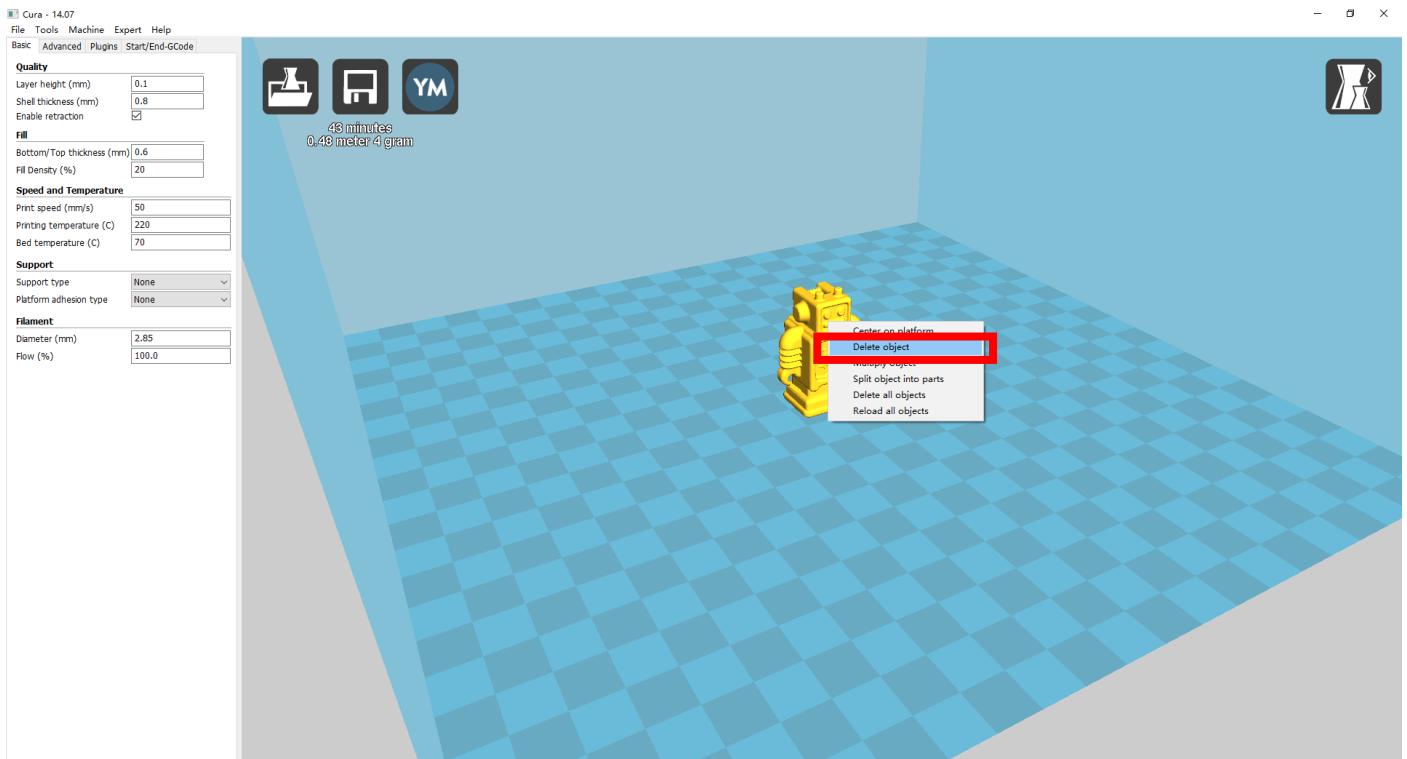
Now you have finished the installation. Next , enter Cura .

2. Cura Setting

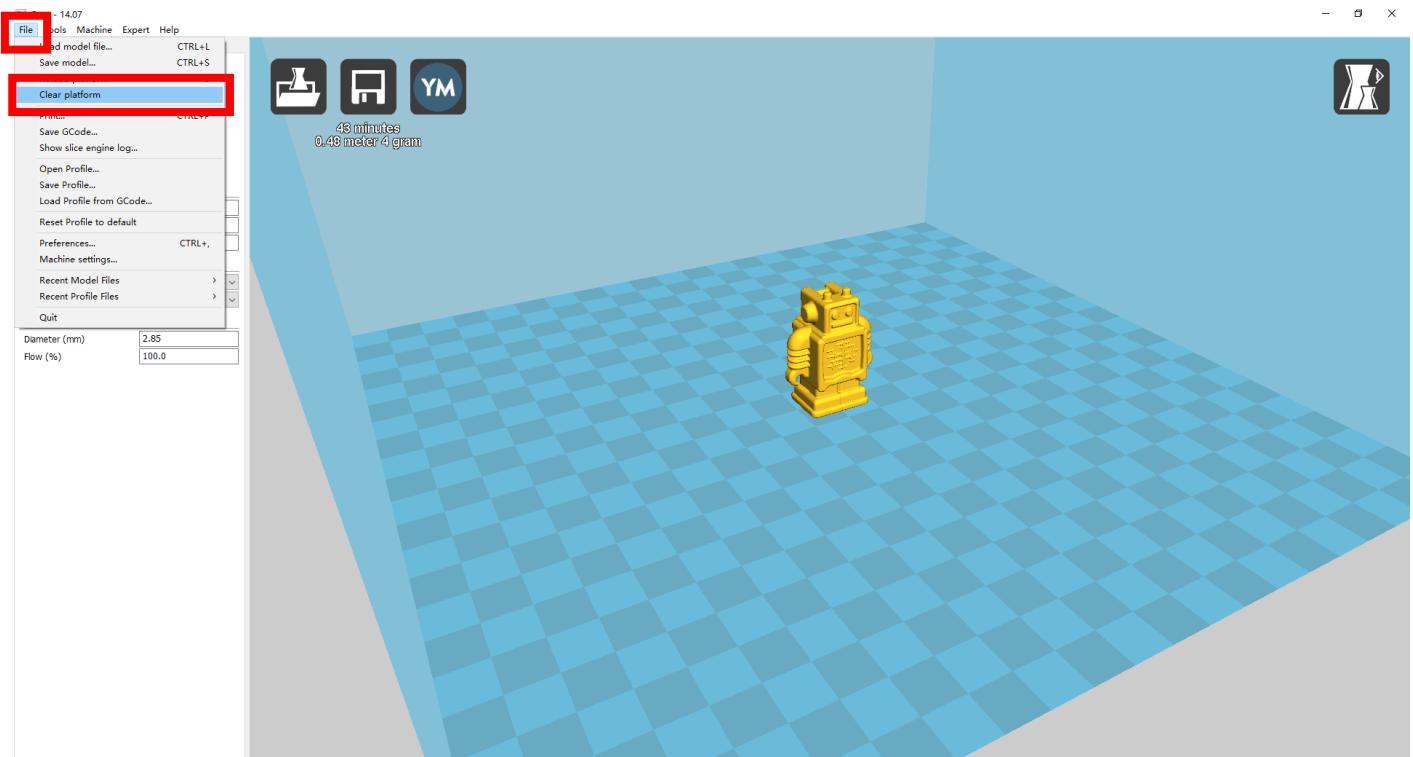
2.1 Clear platform

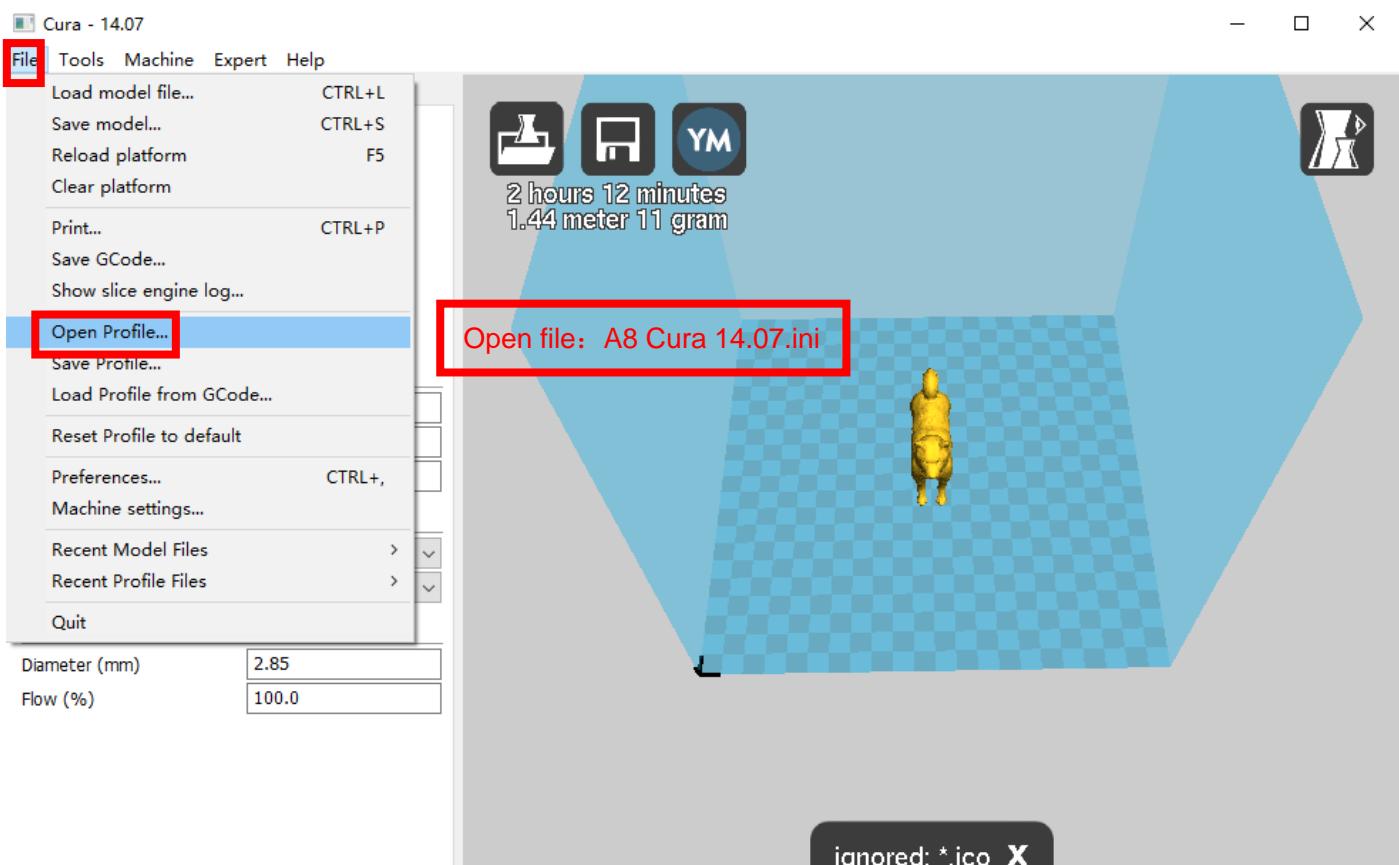
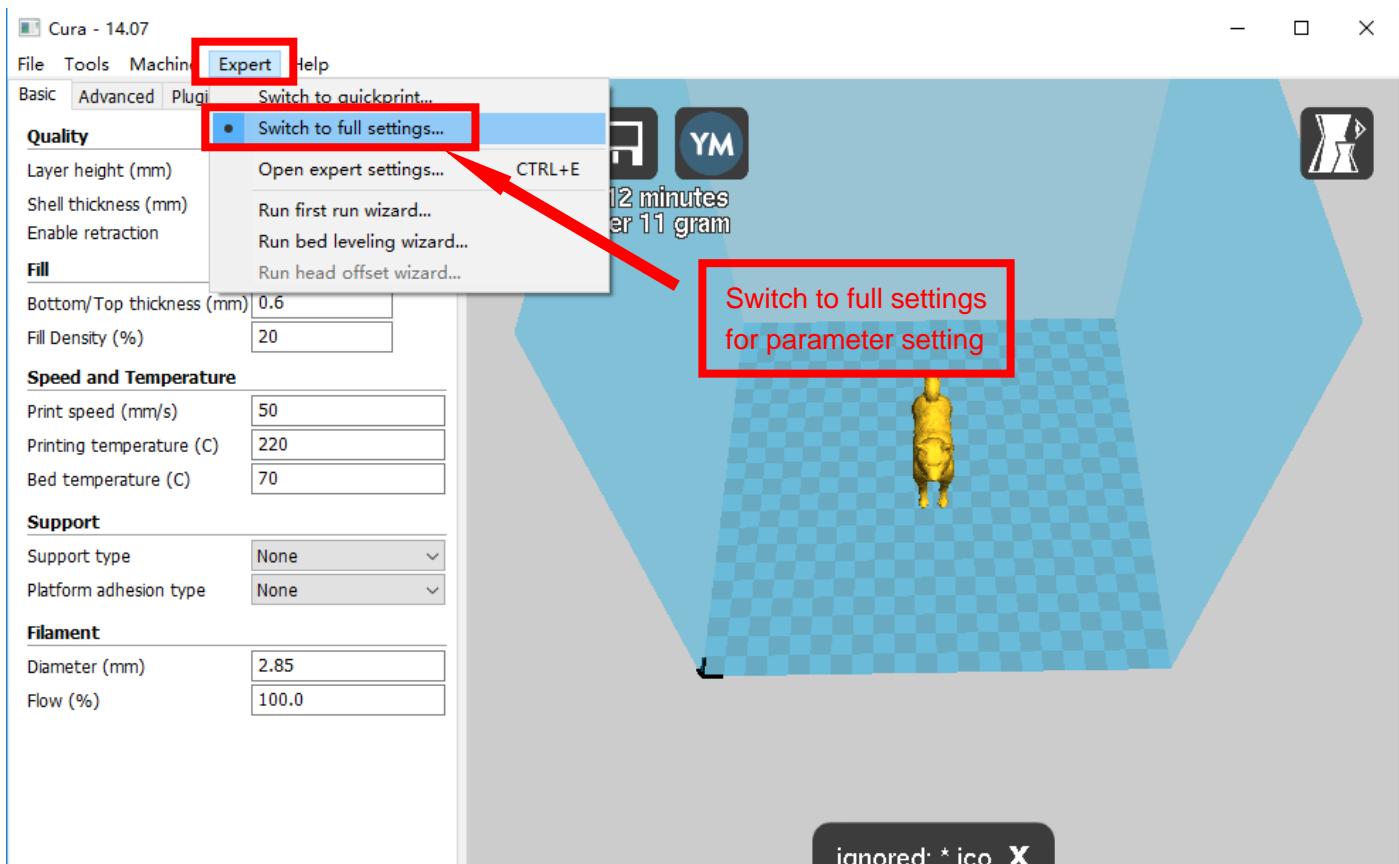
Delete the dog. Two ways for you :

- 1.Move mouse to dog ,right click, click “delete object”.



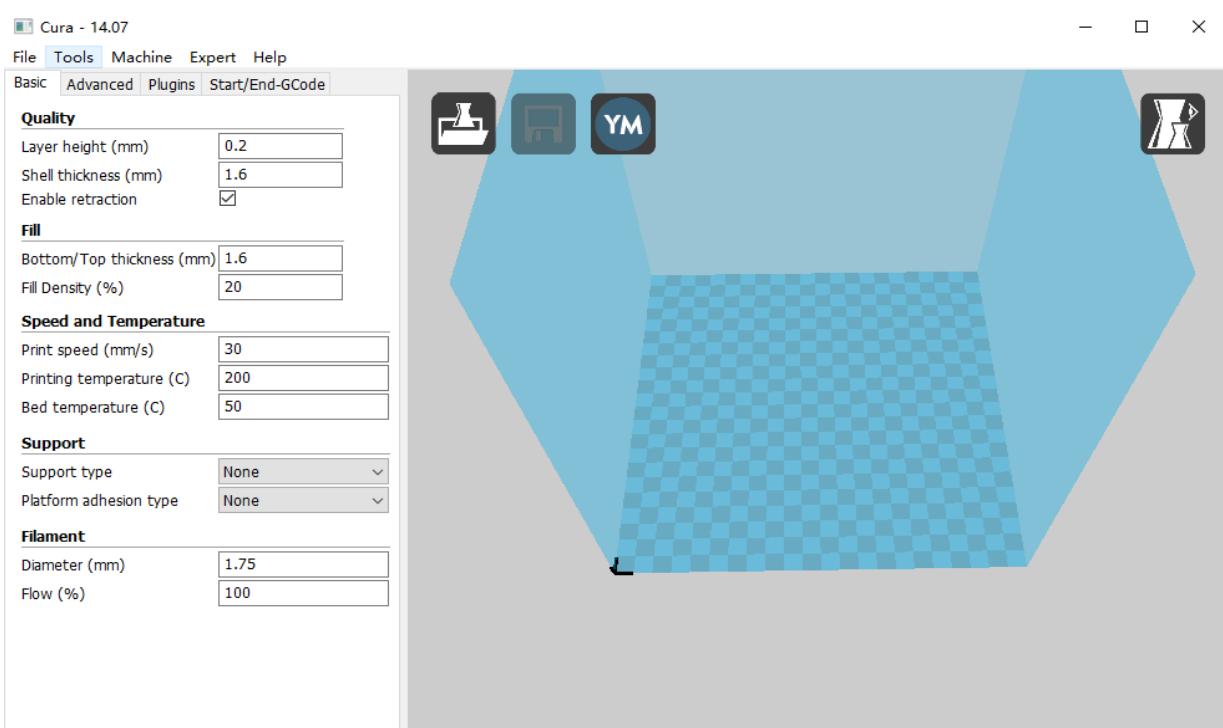
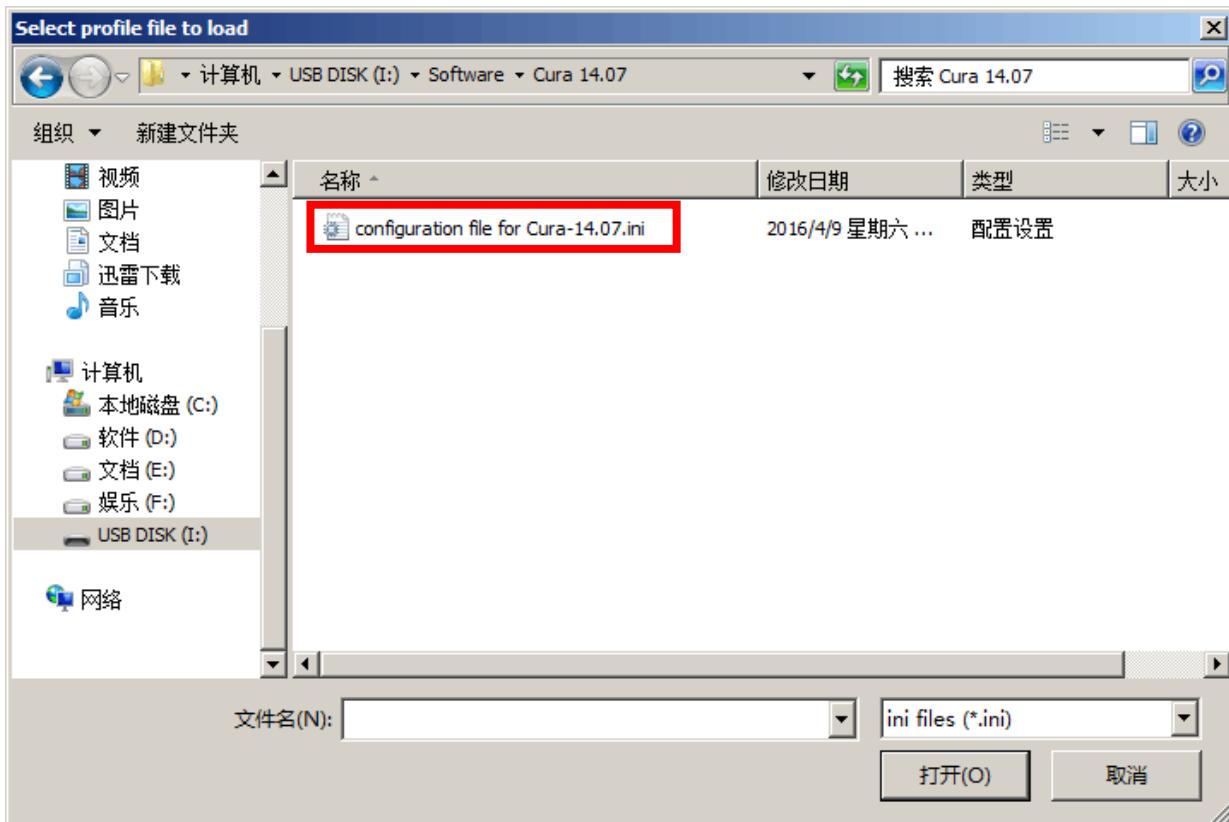
2. Left click “File” , choose “Clear platform”.

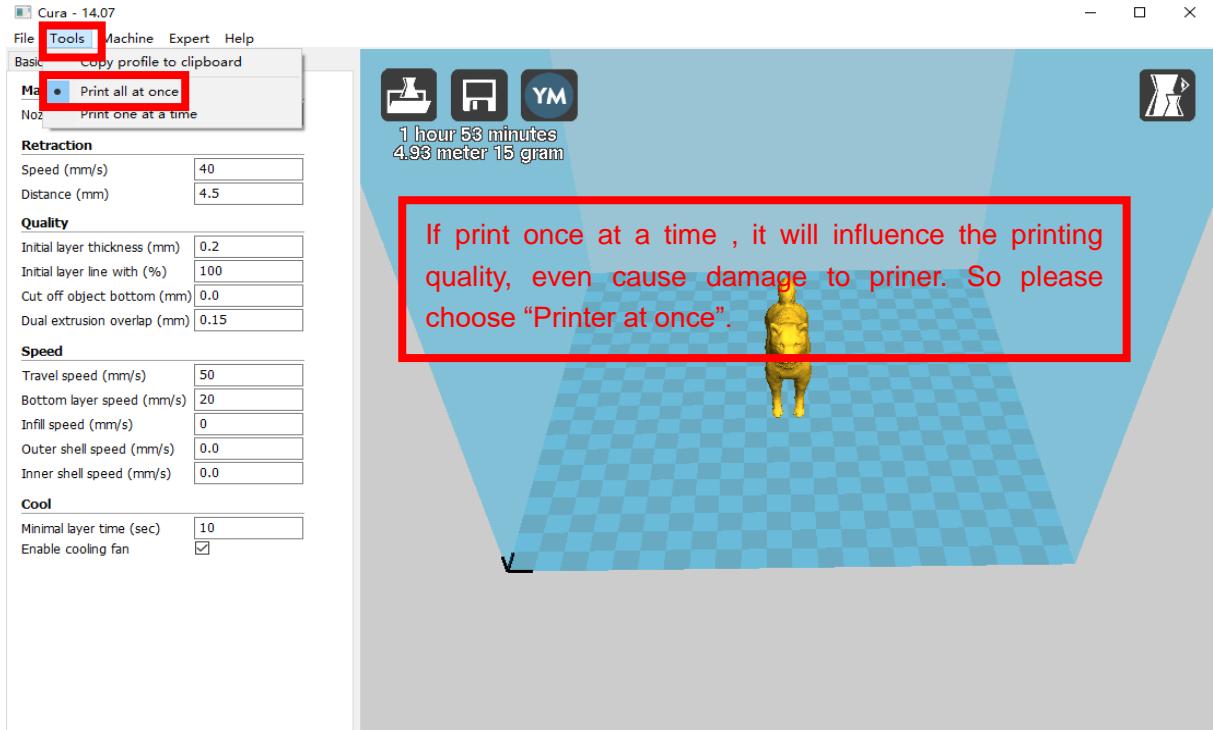




Position of configuration file: Computer/SD card)/ configuration file for cura-14.07

(suggestion :keep this file copy to your computer)





2.3 Layer height settings

Cura - 14.07

File Tools Machine Expert Help

Basic Advanced Plugins Start/End-GCode

Quality

- Layer height (mm)
- Shell thickness (mm)
- Enable retraction

Fill

- Bottom/Top thickness (mm)
- Fill Density (%)

Speed and Temperature

- Print speed (mm/s)
- Printing temperature (C)
- Bed temperature (C)

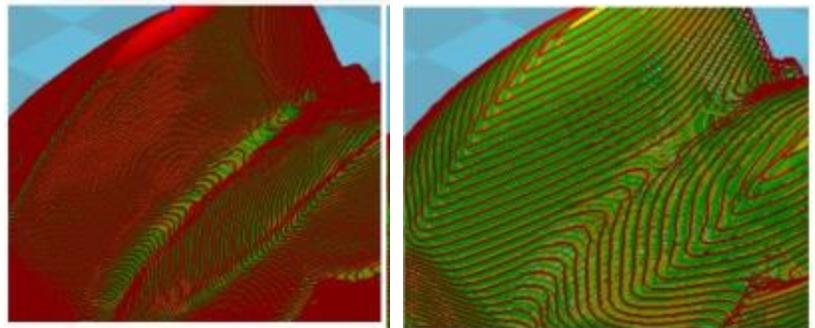
Support

- Support type
- Platform adhesion type

Filament

- Diameter (mm)
- Flow (%)

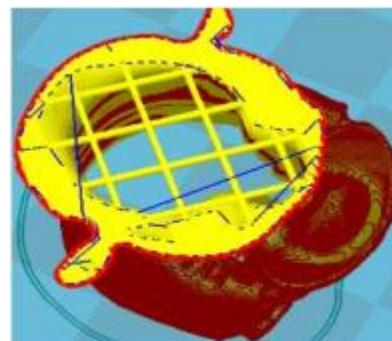
Layer height: 0.1mm cost long time but have the best printing precision. 0.2mm cost half time compared to 0.1mm , but have general printing precision. 0.3 cost less time with not good precision. It defaults 0.2mm.



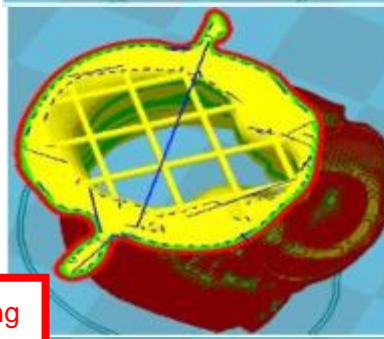
Left: 0.1mm
Layer height

Right: 0.2mm
Layer height

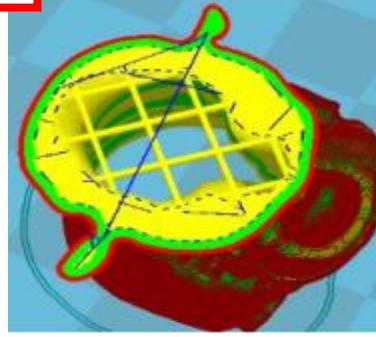
Layer height setting



0.8mm

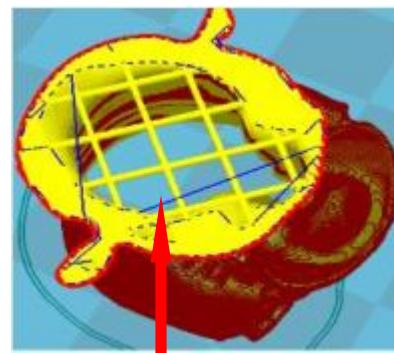
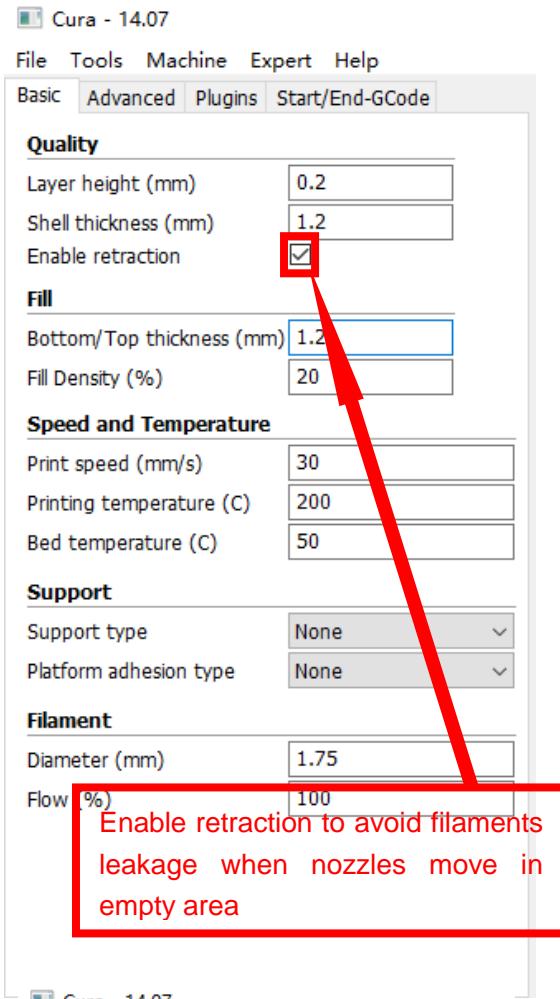


1.2mm



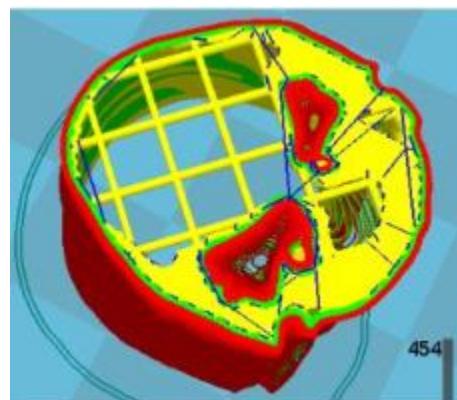
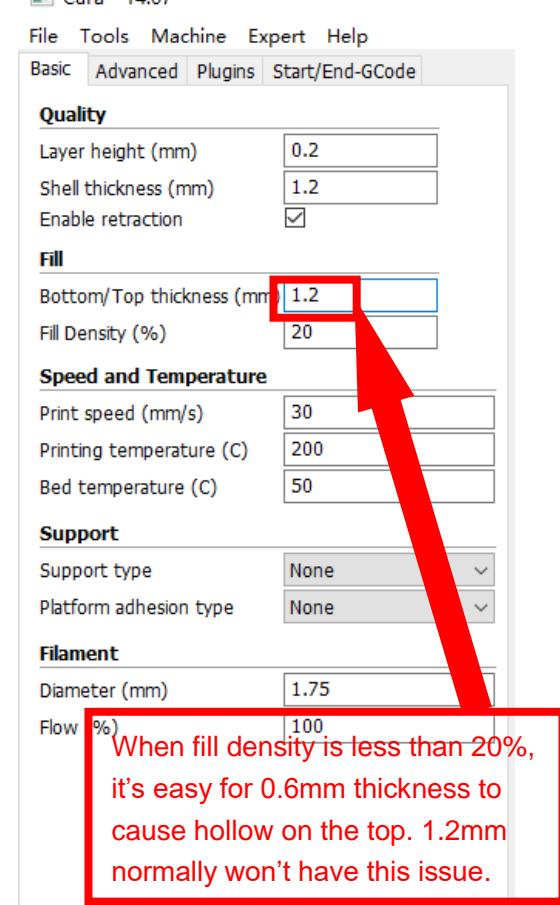
2mm

0.8mm is too thin for shell, 2mm shell costs more time, 1.2mm is relatively better. Please use the integer multiple of nozzle diameter.



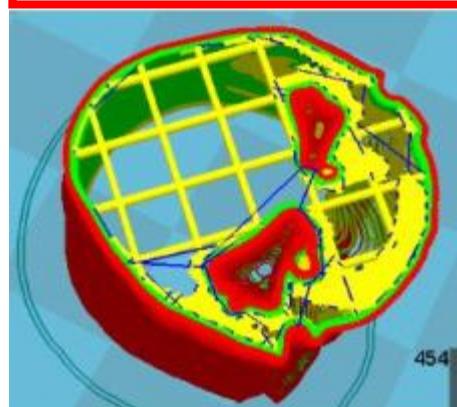
The Blue line is the leak filament

Enable retraction



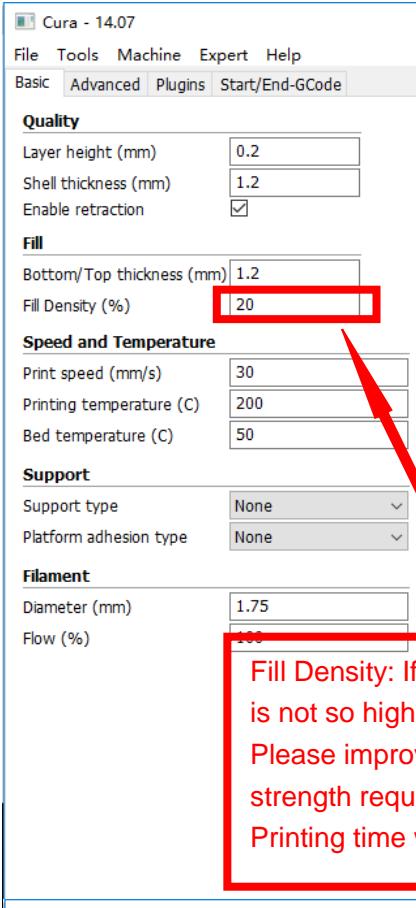
Bottom/Top thickness: 1.2mm

Under the same fill density

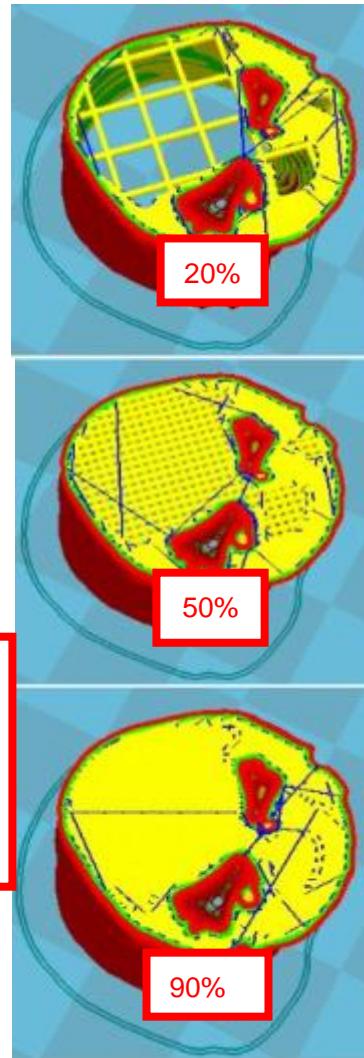


Bottom/Top thickness: 0.6mm

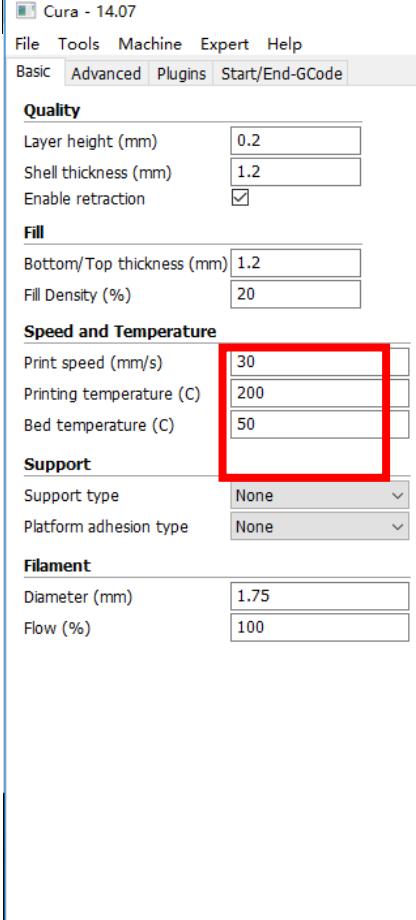
Bottom/Top thickness setting



Fill Density Setting



Fill Density: If strength requirement is not so high , set 10% is ok.
Please improve fill density when strength requirement gets higher.
Printing time will go up as well.



Printing Speed setting

This is default speed. If other settings aren't changed, it prints more accurately while the printing process takes more time.
High printing speed takes less time while it cannot print accurately , making the model have bad quality.
Normally 40-60 print speed is suitable for printing.

Pringting Temperature

PLA filament temperature setting: nozzle: 190-210 °C hotbed: 40-60°C

ABS filament temperature setting: nozzle: 230-250°C hotbed: 60-90°C

Quality

Layer height (mm) 0.2
 Shell thickness (mm) 1.2
 Enable retraction

Fill

Bottom/Top thickness (mm) 1.2
 Fill Density (%) 20

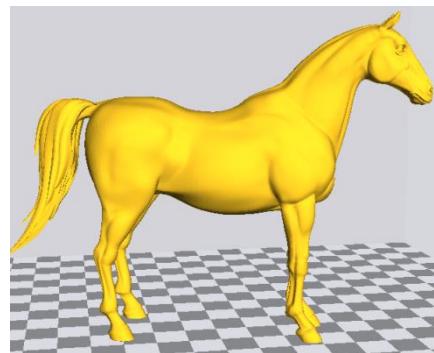
Speed and Temperature

Print speed (mm/s) 30
 Printing temperature (C) 200
 Bed temperature (C) 50

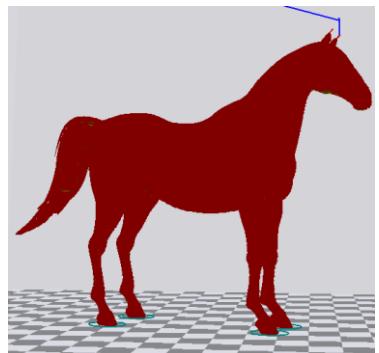
Support

Support type None
 Platform adhesion type None
 Touching buildplate
 Everywhere

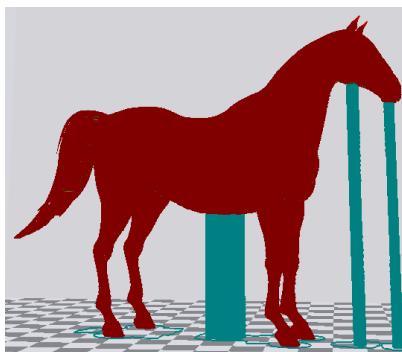
Filament
 Diameter (mm) 1.75
 Flow (%) 100



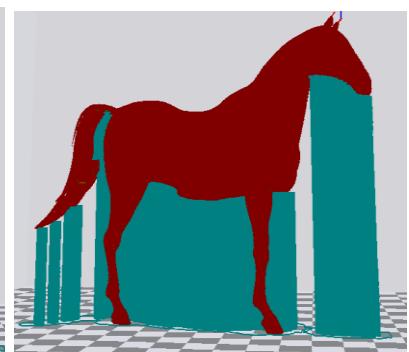
Original Model



Support type: None



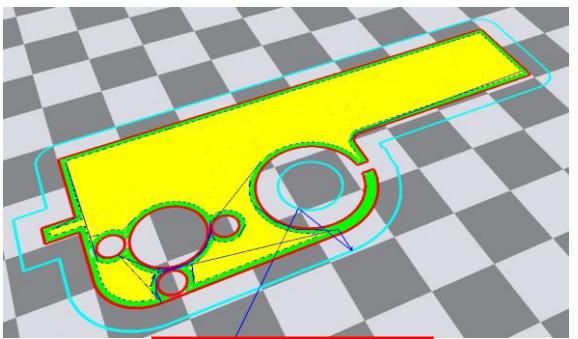
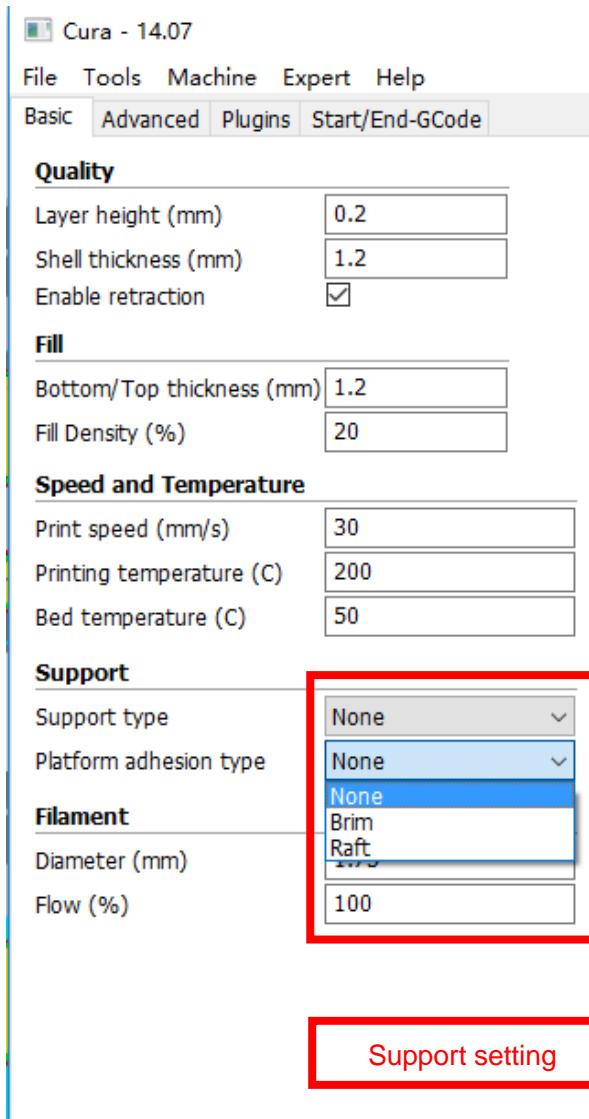
Support type: Touching



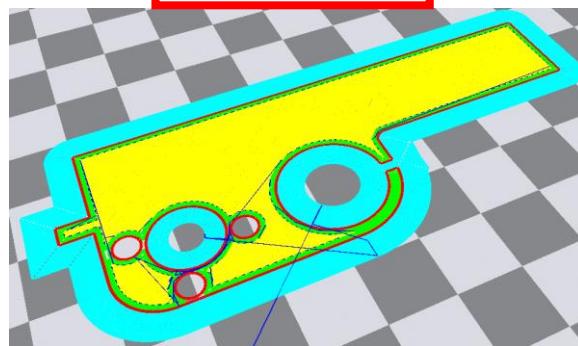
Support type: Everywhere

Support type setting

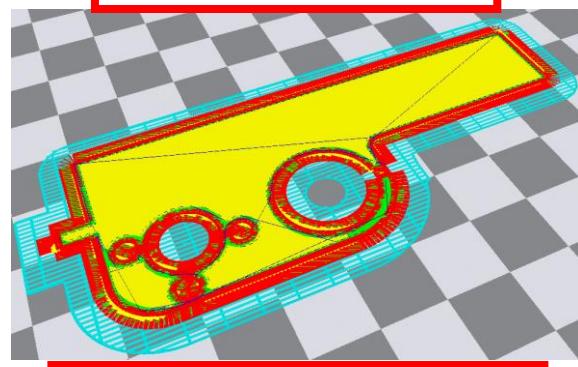
Attention: Normally we add support to complex model or model with vacant parts. It may have influence on the surface if you choose everywhere. You'd better circle around the model and try to avoid unnecessary support.



None: no support

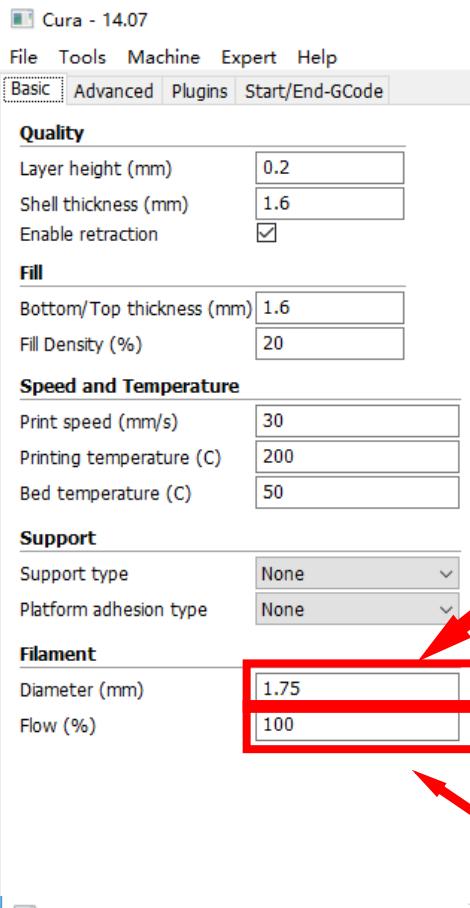


Brim: Touch with the edge

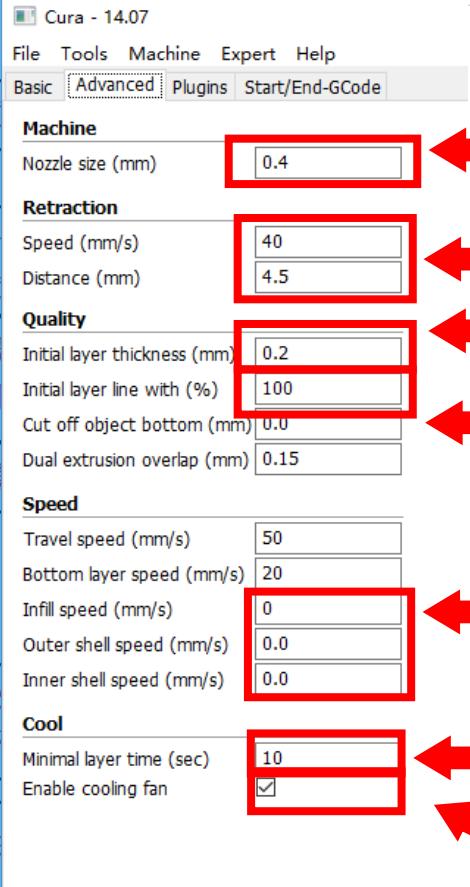


Raft: Totally touch with the bottom

Attention: please choose None if the printing platform is ready and the high temperature adhesive tape is good. Please choose Brim when the model is small . Choosing Raft makes it difficult to separate model from the platform



Attention: A8 use 1.75mm filament



Attention:

Flow is proportion of filament , we suggest to use 100
Increasing flow & decrease diameter has the similar effect.

Model surface gets many bumps when flow is too big;model frame gets flimsy if flow is too small.

We suggest not to change it , A8 default 0.4mm

We suggest not to change it , or use the date in the picture

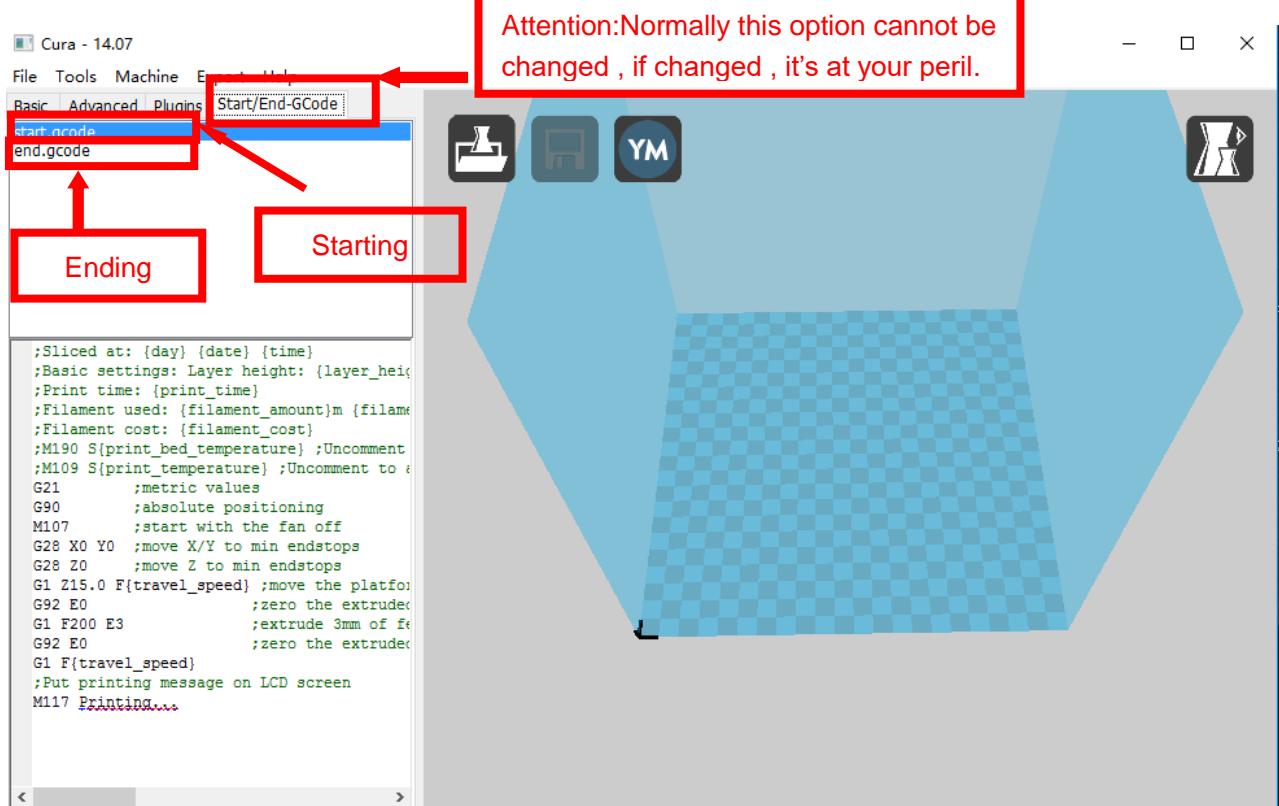
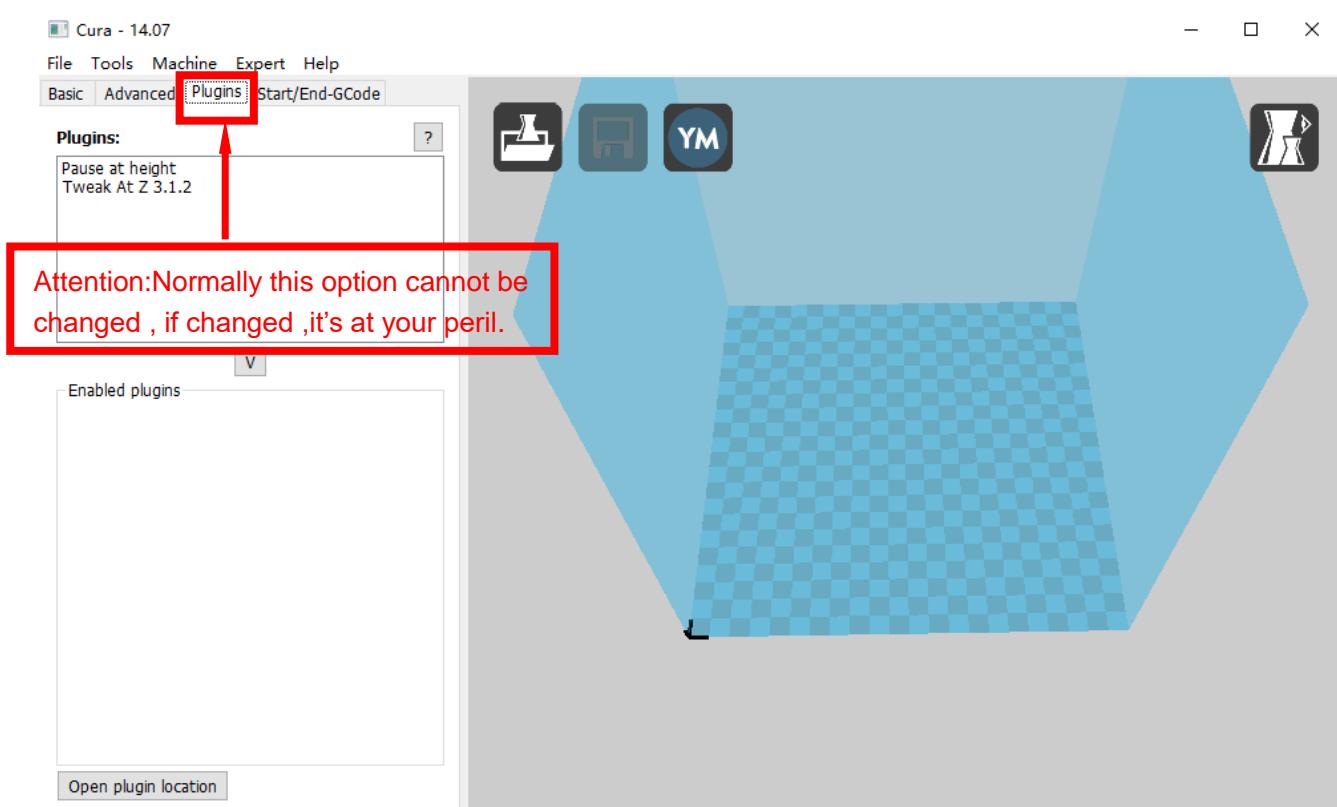
We suggest 0.2mm to avoid initial layer tilt,0.3mm is more easy to seperate from the platform.

Initial layer line proportion

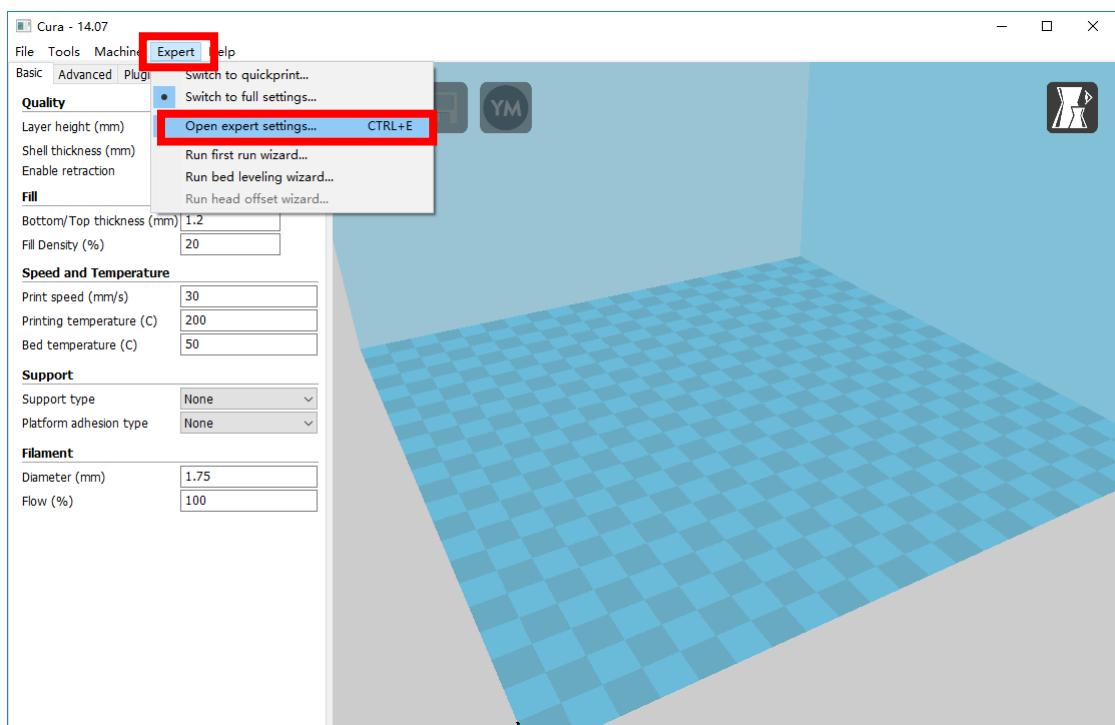
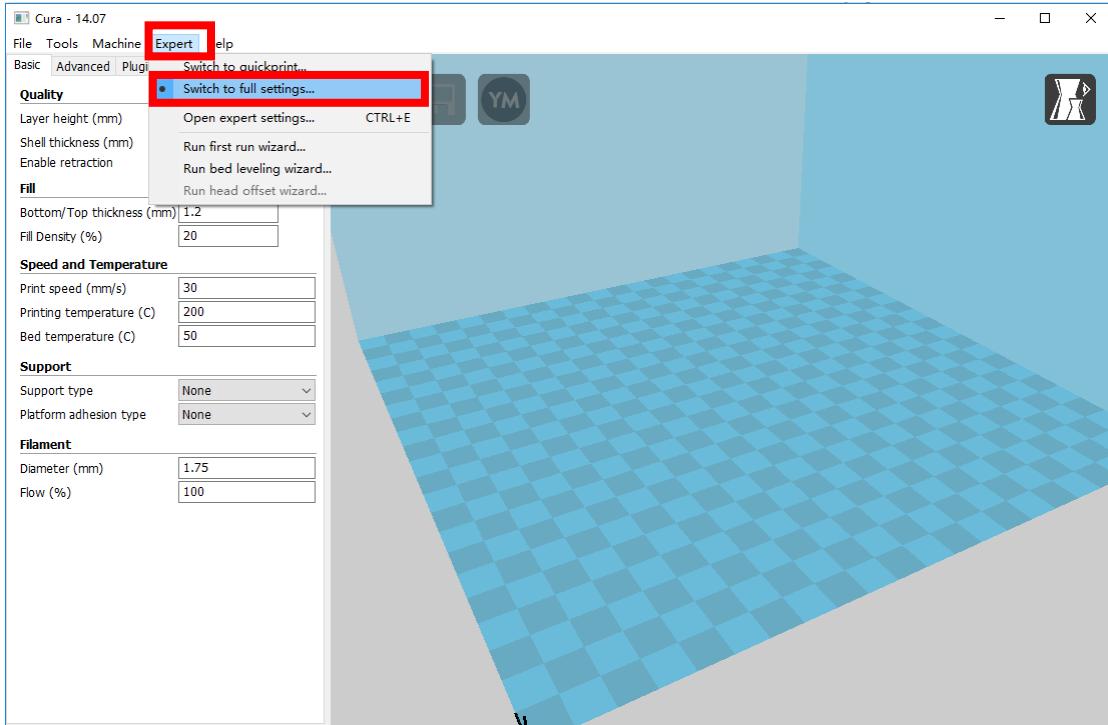
"0" means using default speed

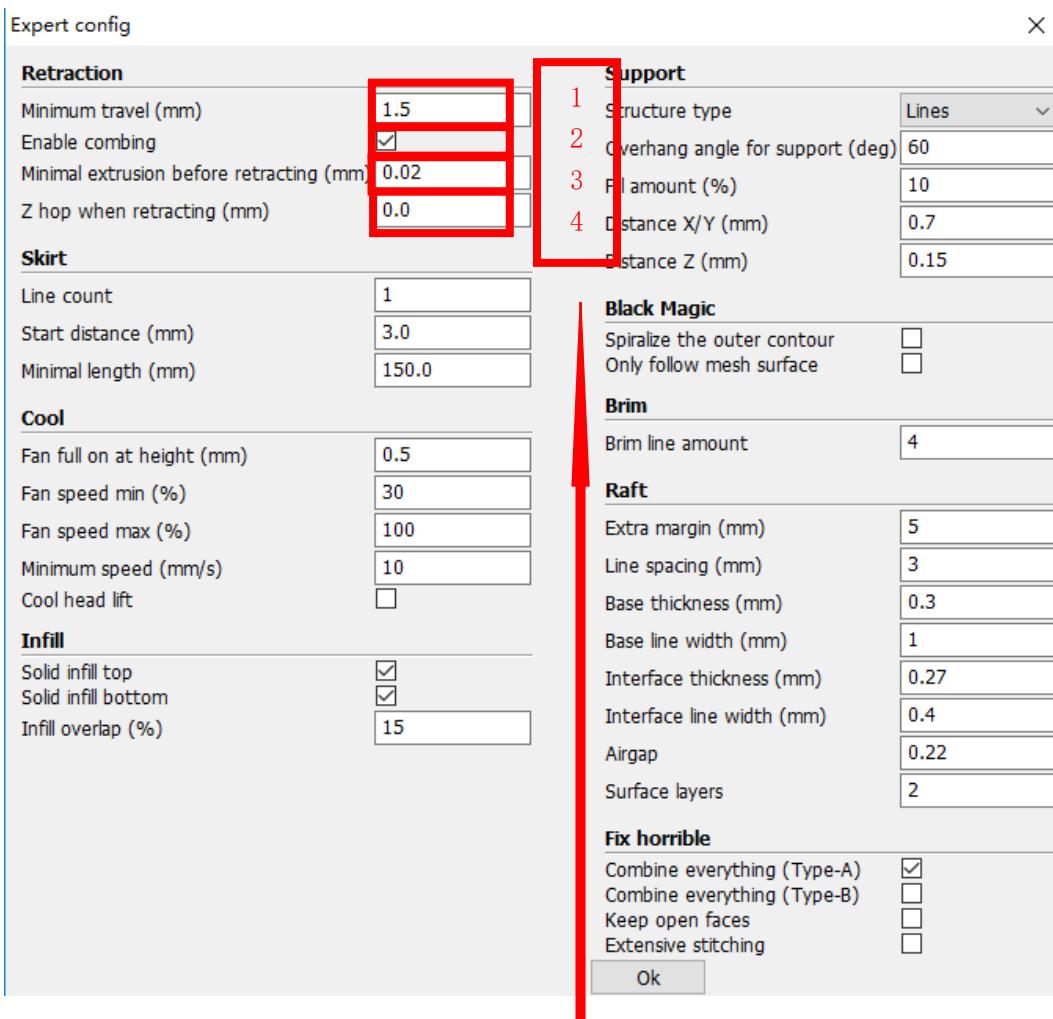
Min printing time for each layer. When the time is less than 10 , it prints slower. It's better to decrease time when printing thin and long models.

We suggest not choose this when printing ABS.

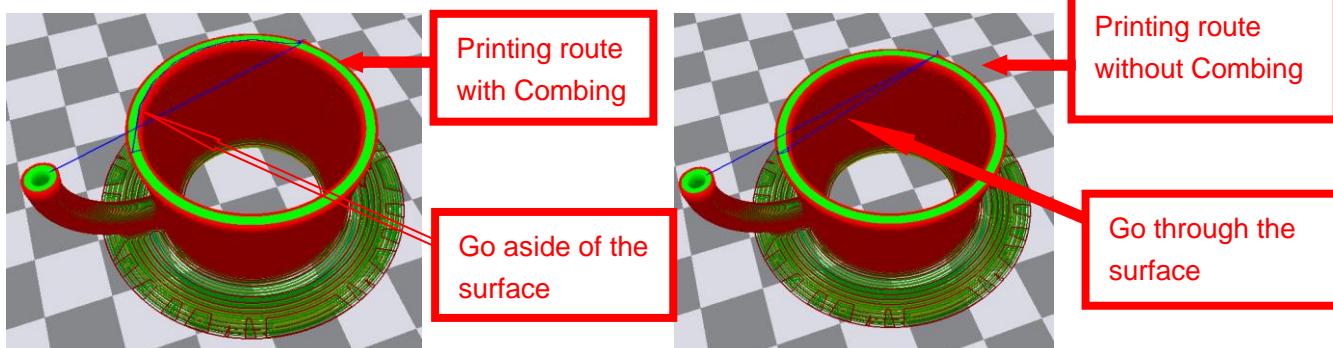


3.Expert Setting





1. The minimum length before retraction . Used to avoid frequent retraction. No need to change.
2. Enable Combing: Digital for surface quality , the nozzle will try not to go through surface , that's why Cura is better than Slic3r.
3. Minimum extrusion length , to avoid frequent extrusion.



Expert config

Retraction

Minimum travel (mm)	1.5
Enable combing	<input checked="" type="checkbox"/>
Minimal extrusion before retracting (mm)	0.02
Z hop when retracting (mm)	0.0

Skirt

Line count	1
Start distance (mm)	3.0
Minimal length (mm)	150.0

Cool

Fan full on at height (mm)	0.5
Fan speed min (%)	30
Fan speed max (%)	100
Minimum speed (mm/s)	10
Cool head lift	<input type="checkbox"/>

Infill

Solid infill top	<input checked="" type="checkbox"/>
Solid infill bottom	<input checked="" type="checkbox"/>
Infill overlap (%)	15

Support

Structure type	Lines
Overhang angle for support (deg)	60
Fill amount (%)	10
Distance X/Y (mm)	0.7
Distance Z (mm)	0.15

Black Magic

Spiralize the outer contour	<input type="checkbox"/>
Only follow mesh surface	<input type="checkbox"/>

Brim

Brim line amount	4
------------------	---

Raft

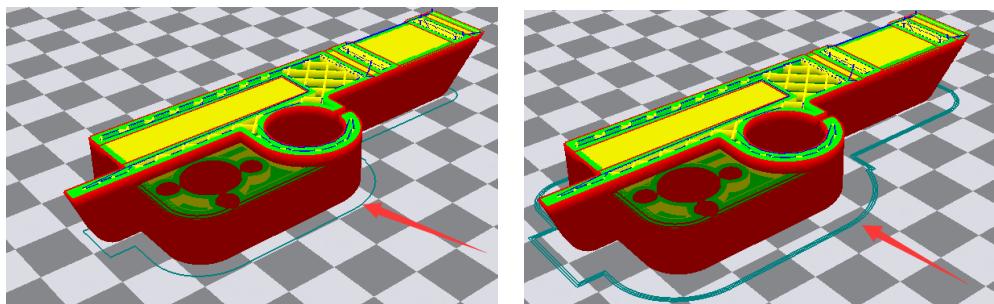
Extra margin (mm)	5
Line spacing (mm)	3
Base thickness (mm)	0.3
Base line width (mm)	1
Interface thickness (mm)	0.27
Interface line width (mm)	0.4
Airgap	0.22
Surface layers	2

Fix horrible

Combine everything (Type-A)	<input checked="" type="checkbox"/>
Combine everything (Type-B)	<input type="checkbox"/>
Keep open faces	<input type="checkbox"/>
Extensive stitching	<input type="checkbox"/>

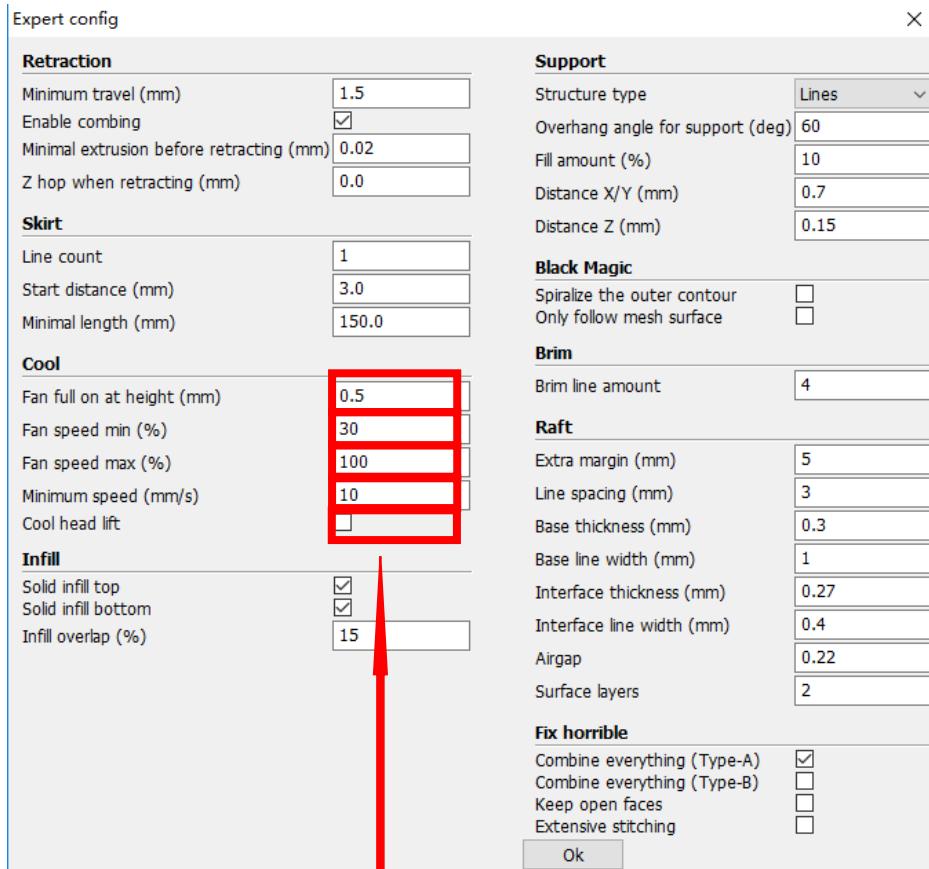
Ok

Skirt is to avoid extruder unfilled before printing , and it appears only when platform attachment type is None. Normally “1” is ok . Change it to “0” when your model reaches the maximum size , or the printing size will be too big.



Peripheral line quantity: 1
Start distance: 3

Peripheral line quantity: 1
Start distance: 3

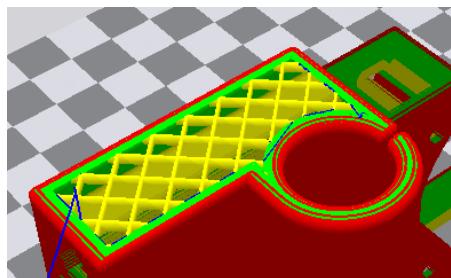


1. To ensure the attachment of model to platform , fan won't start at the beginning.
- 2-4. Fan speed min & max : If they are not equal , the soft ware will choose a suitable speed during them.
5. Condition to choose cool head lift : When it's printing with the minimum speed but still cannot reach the minimum time , you need to choose cool head lift . But it may cause filament leak.

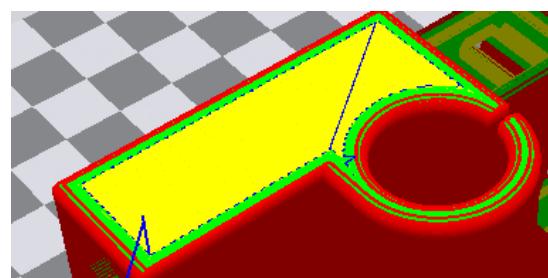
Expert config X

Retraction		Support	
Minimum travel (mm)	1.5	Structure type	Lines ▼
Enable combing	<input checked="" type="checkbox"/>	Overhang angle for support (deg)	60
Minimal extrusion before retracting (mm)	0.02	Fill amount (%)	10
Z hop when retracting (mm)	0.0	Distance X/Y (mm)	0.7
Skirt		Distance Z (mm)	
Line count	1	0.15	
Start distance (mm)	3.0	Black Magic	
Minimal length (mm)	150.0	Spiralize the outer contour	<input type="checkbox"/>
Cool		Only follow mesh surface	<input type="checkbox"/>
Fan full on at height (mm)	0.5	Brim	
Fan speed min (%)	30	Brim line amount	4
Fan speed max (%)	100	Raft	
Minimum speed (mm/s)	10	Extra margin (mm)	5
Cool head lift	<input type="checkbox"/>	Line spacing (mm)	3
Infill		Base thickness (mm)	0.3
Solid infill top	<input checked="" type="checkbox"/>	Base line width (mm)	1
Solid infill bottom	<input checked="" type="checkbox"/>	Interface thickness (mm)	0.27
Infill overlap (%)	15	Interface line width (mm)	0.4
		Airgap	0.22
		Surface layers	2
Fix horrible			
Combine everything (Type-A) <input checked="" type="checkbox"/>			
Combine everything (Type-B) <input type="checkbox"/>			
Keep open faces <input type="checkbox"/>			
Extensive stitching <input type="checkbox"/>			
Ok			

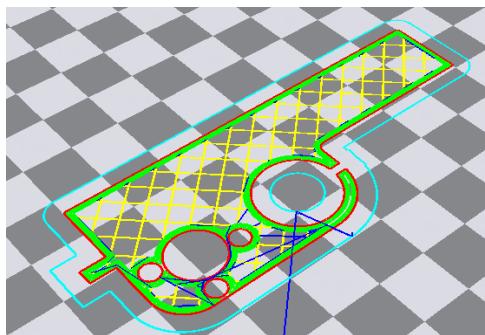
If no solid infill top , the only qualification is the surface thickness .
Please check the detail below.



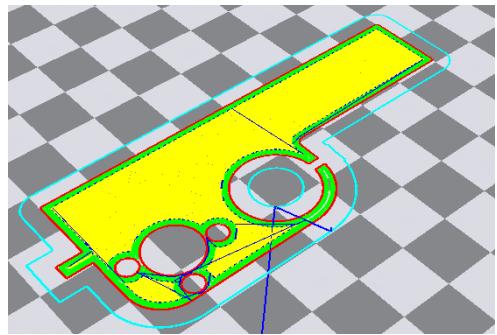
No solid infill top



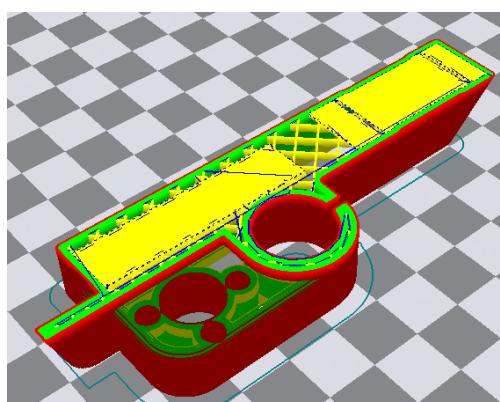
With solid infill top



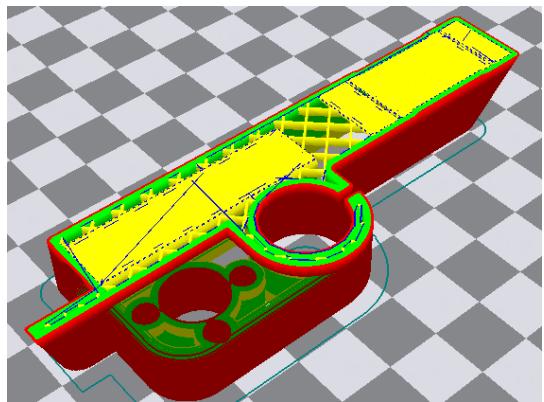
No solid infill bottom



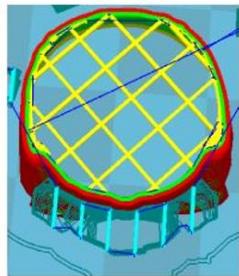
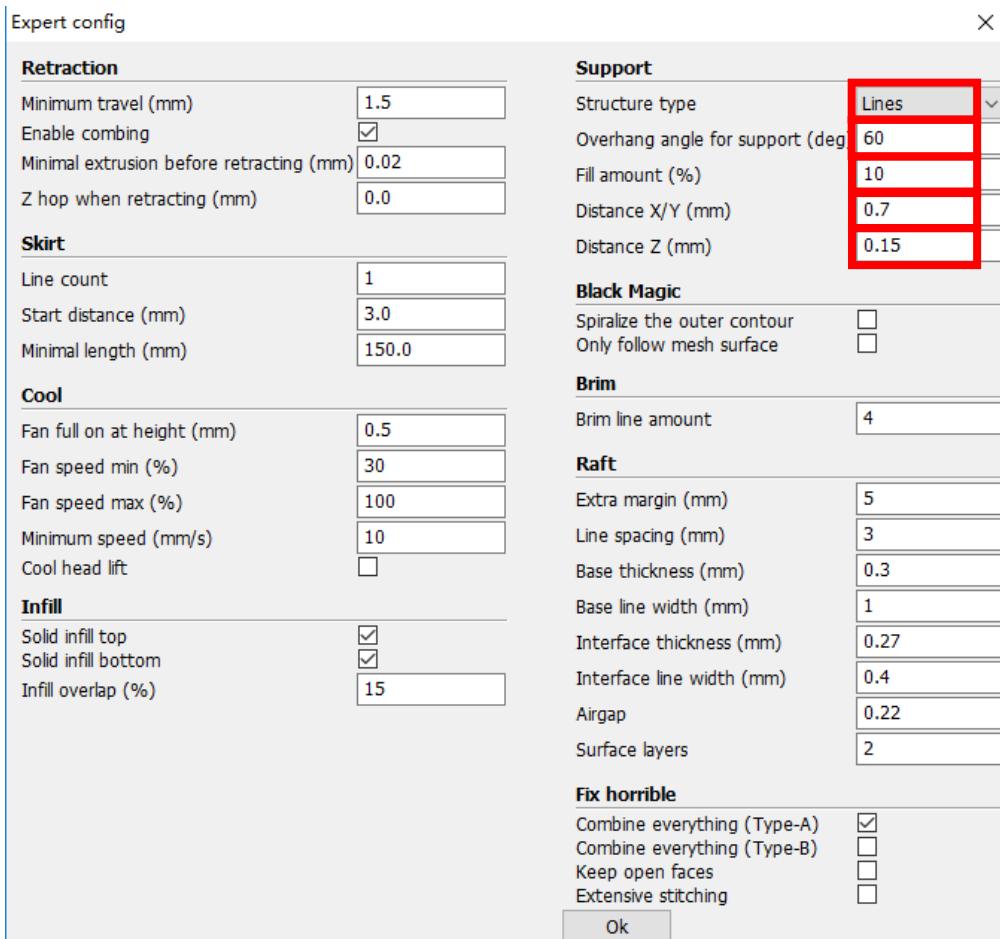
With solid infill bottom



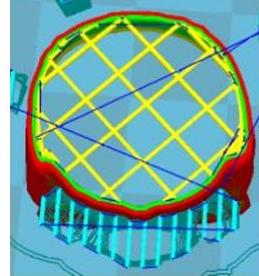
Infill coverlap: 20



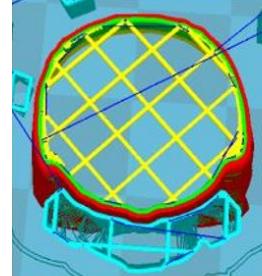
Infill coverlap: 40



Structure types:lines
Infill covertap: 15
Distance X/Y: 0.7



Structure types:lines
Infill covertap: 30
Distance X/Y: 0.7

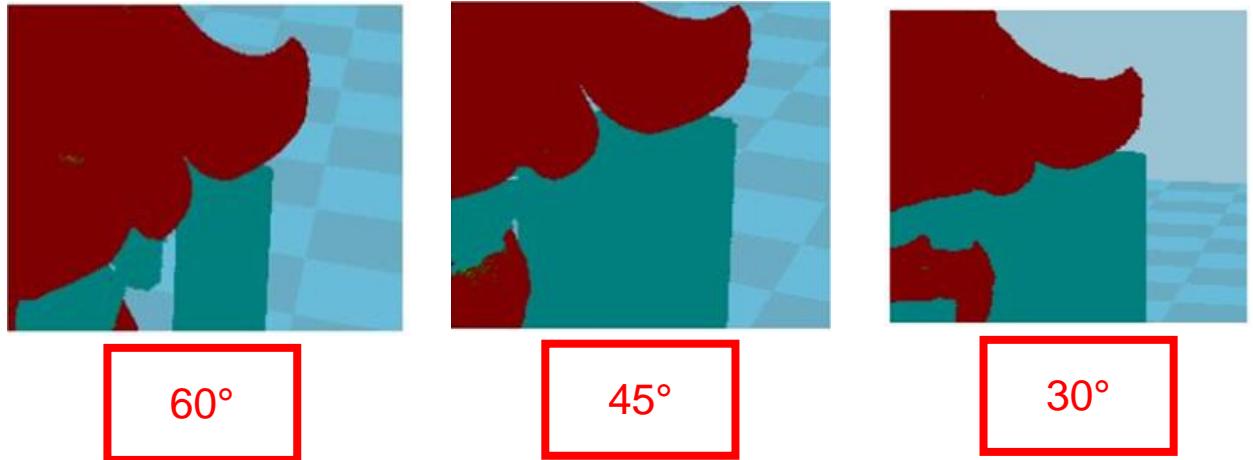


Structure types:grids
Infill covertap: 15
Distance X/Y: 0.7

These above are examples , you can set these options according to actual requirements. The biggest progress Cura has made is the kinds of support structure types , making it easier to separate from the model.

Expert config

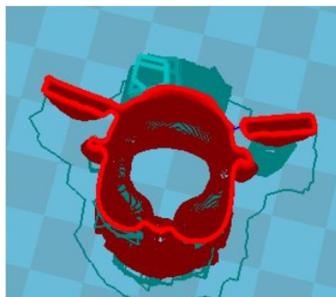
Retraction	Support		
Minimum travel (mm)	1.5	Structure type	Lines
Enable combing	<input checked="" type="checkbox"/>	Overhang angle for support (deg)	60
Minimal extrusion before retracting (mm)	0.02	Fill amount (%)	10
Z hop when retracting (mm)	0.0	Distance X/Y (mm)	0.7
Skirt	Black Magic	Distance Z (mm)	0.15
Line count	1	Spiralize the outer contour	<input type="checkbox"/>
Start distance (mm)	3.0	Only follow mesh surface	<input type="checkbox"/>
Minimal length (mm)	150.0	Brim	
Cool	Raft		
Fan full on at height (mm)	0.5	Extra margin (mm)	5
Fan speed min (%)	30	Line spacing (mm)	3
Fan speed max (%)	100	Base thickness (mm)	0.3
Minimum speed (mm/s)	10	Base line width (mm)	1
Cool head lift	<input type="checkbox"/>	Interface thickness (mm)	0.27
Infill	Interface line width (mm)	0.4	
Solid infill top	<input checked="" type="checkbox"/>	Airgap	0.22
Solid infill bottom	<input checked="" type="checkbox"/>	Surface layers	2
Infill overlap (%)	15	Fix horrible	
<input type="button" value="Ok"/>			



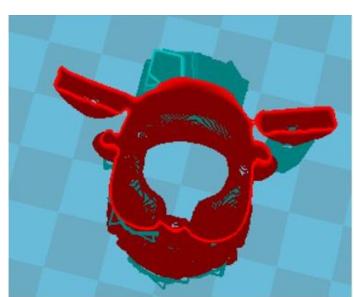
It's difficult to separate if the distance between support and the supported place is too close; The surface will be influenced if the distance is too far.
 Different angle will generate different support , you can try the examples we provide above which will have different effect.

Expert config X

Retraction	Support
Minimum travel (mm) 1.5	Structure type Lines
Enable combing <input checked="" type="checkbox"/>	Overhang angle for support (deg) 60
Minimal extrusion before retracting (mm) 0.02	Fill amount (%) 10
Z hop when retracting (mm) 0.0	Distance X/Y (mm) 0.7
Skirt	Distance Z (mm) 0.15
Line count 1	Black Magic
Start distance (mm) 3.0	Spiralize the outer contour <input type="checkbox"/>
Minimal length (mm) 150.0	Only follow mesh surface <input type="checkbox"/>
Cool	Brim
Fan full on at height (mm) 0.5	Brim line amount 4
Fan speed min (%) 30	Raft
Fan speed max (%) 100	Extra margin (mm) 5
Minimum speed (mm/s) 10	Line spacing (mm) 3
Cool head lift <input type="checkbox"/>	Base thickness (mm) 0.3
Infill	Base line width (mm) 1
Solid infill top <input checked="" type="checkbox"/>	Interface thickness (mm) 0.27
Solid infill bottom <input checked="" type="checkbox"/>	Interface line width (mm) 0.4
Infill overlap (%) 15	Airgap 0.22
	Surface layers 2
	Fix horrible
	Combine everything (Type-A) <input checked="" type="checkbox"/>
	Combine everything (Type-B) <input type="checkbox"/>
	Keep open faces <input type="checkbox"/>
	Extensive stitching <input type="checkbox"/>
	Ok



When choosing “Spiralize the outer contour” : Z axis rises while X,Y axis moves , and only a hollow bottom and a single layer of surface.



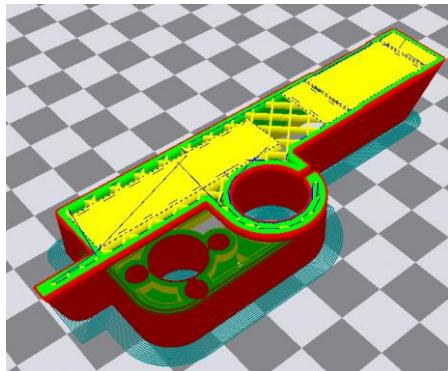
When choosing “Only follow mesh surface” : The nozzle prints along the surface.

Attention: The software defaults not open the option above , you'd better not turn it on .

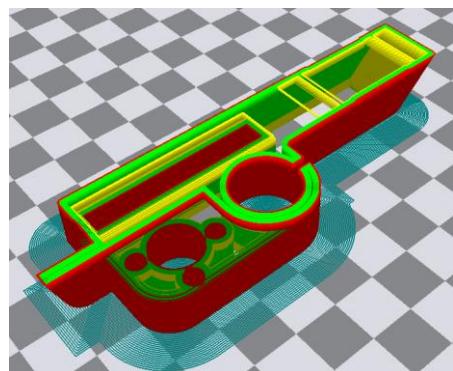
Expert config

Retraction	Support
Minimum travel (mm) <input type="text" value="1.5"/>	Structure type <input type="text" value="Lines"/>
Enable combing <input checked="" type="checkbox"/>	Overhang angle for support (deg) <input type="text" value="60"/>
Minimal extrusion before retracting (mm) <input type="text" value="0.02"/>	Fill amount (%) <input type="text" value="10"/>
Z hop when retracting (mm) <input type="text" value="0.0"/>	Distance X/Y (mm) <input type="text" value="0.7"/>
Skirt	Distance Z (mm) <input type="text" value="0.15"/>
Line count <input type="text" value="1"/>	Black Magic
Start distance (mm) <input type="text" value="3.0"/>	Spiralize the outer contour <input type="checkbox"/>
Minimal length (mm) <input type="text" value="150.0"/>	Only follow mesh surface <input type="checkbox"/>
Cool	Brim
Fan full on at height (mm) <input type="text" value="0.5"/>	Brim line amount <input type="text" value="4"/>
Fan speed min (%) <input type="text" value="30"/>	Raft
Fan speed max (%) <input type="text" value="100"/>	Extra margin (mm) <input type="text" value="5"/>
Minimum speed (mm/s) <input type="text" value="10"/>	Line spacing (mm) <input type="text" value="3"/>
Cool head lift <input type="checkbox"/>	Base thickness (mm) <input type="text" value="0.3"/>
Infill	Base line width (mm) <input type="text" value="1"/>
Solid infill top <input checked="" type="checkbox"/>	Interface thickness (mm) <input type="text" value="0.27"/>
Solid infill bottom <input checked="" type="checkbox"/>	Interface line width (mm) <input type="text" value="0.4"/>
Infill overlap (%) <input type="text" value="15"/>	Airgap <input type="text" value="0.22"/>
	Surface layers <input type="text" value="2"/>
	Fix horrible
	Combine everything (Type-A) <input checked="" type="checkbox"/>
	Combine everything (Type-B) <input type="checkbox"/>
	Keep open faces <input type="checkbox"/>
	Extensive stitching <input type="checkbox"/>

Ok



Brim line amount: 10



Brim line amount: 20

Guidance to use Brim if needed: Expert → Expert Settings → Support → Support Types → Birm.

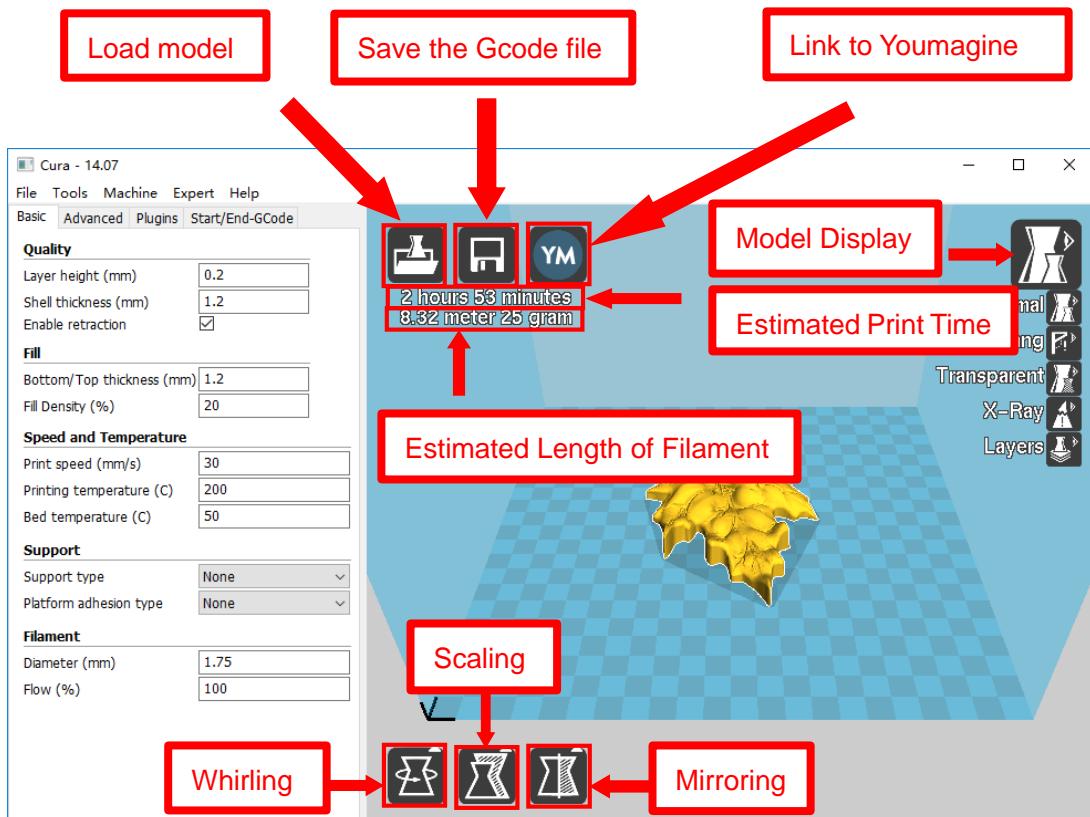
The images above are only for reference , please set the parameter according to actual requirement.

Expert config X

Retraction	Support
Minimum travel (mm) 1.5	Structure type Lines
Enable combing <input checked="" type="checkbox"/>	Overhang angle for support (deg) 60
Minimal extrusion before retracting (mm) 0.02	Fill amount (%) 10
Z hop when retracting (mm) 0.0	Distance X/Y (mm) 0.7
Skirt	Distance Z (mm) 0.15
Line count 1	Black Magic
Start distance (mm) 3.0	Spiralize the outer contour <input type="checkbox"/>
Minimal length (mm) 150.0	Only follow mesh surface <input type="checkbox"/>
Cool	Brim
Fan full on at height (mm) 0.5	Brim line amount 4
Fan speed min (%) 30	Raft
Fan speed max (%) 100	Extra margin (mm) 5
Minimum speed (mm/s) 10	Line spacing (mm) 3
Cool head lift <input type="checkbox"/>	Base thickness (mm) 0.3
Infill	Base line width (mm) 1
Solid infill top <input checked="" type="checkbox"/>	Interface thickness (mm) 0.27
Solid infill bottom <input checked="" type="checkbox"/>	Interface line width (mm) 0.4
Infill overlap (%) 15	Airgap 0.22
	Surface layers 2
	Fix horrible
	Combine everything (Type-A) <input checked="" type="checkbox"/>
	Combine everything (Type-B) <input type="checkbox"/>
	Keep open faces <input type="checkbox"/>
	Extensive stitching <input type="checkbox"/>
	Ok

Guidance to use Raft if needed: Expert → Expert Settings → Support → Support Types → Raft.

The images above are only for reference , please set the parameter according to actual requirement.



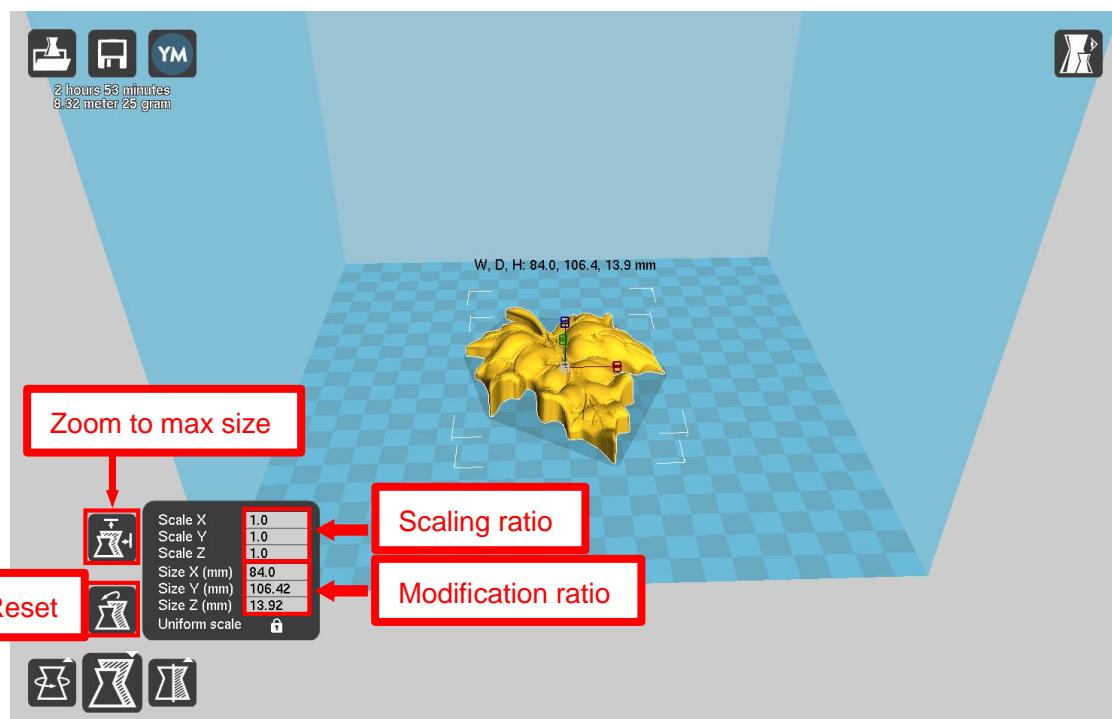
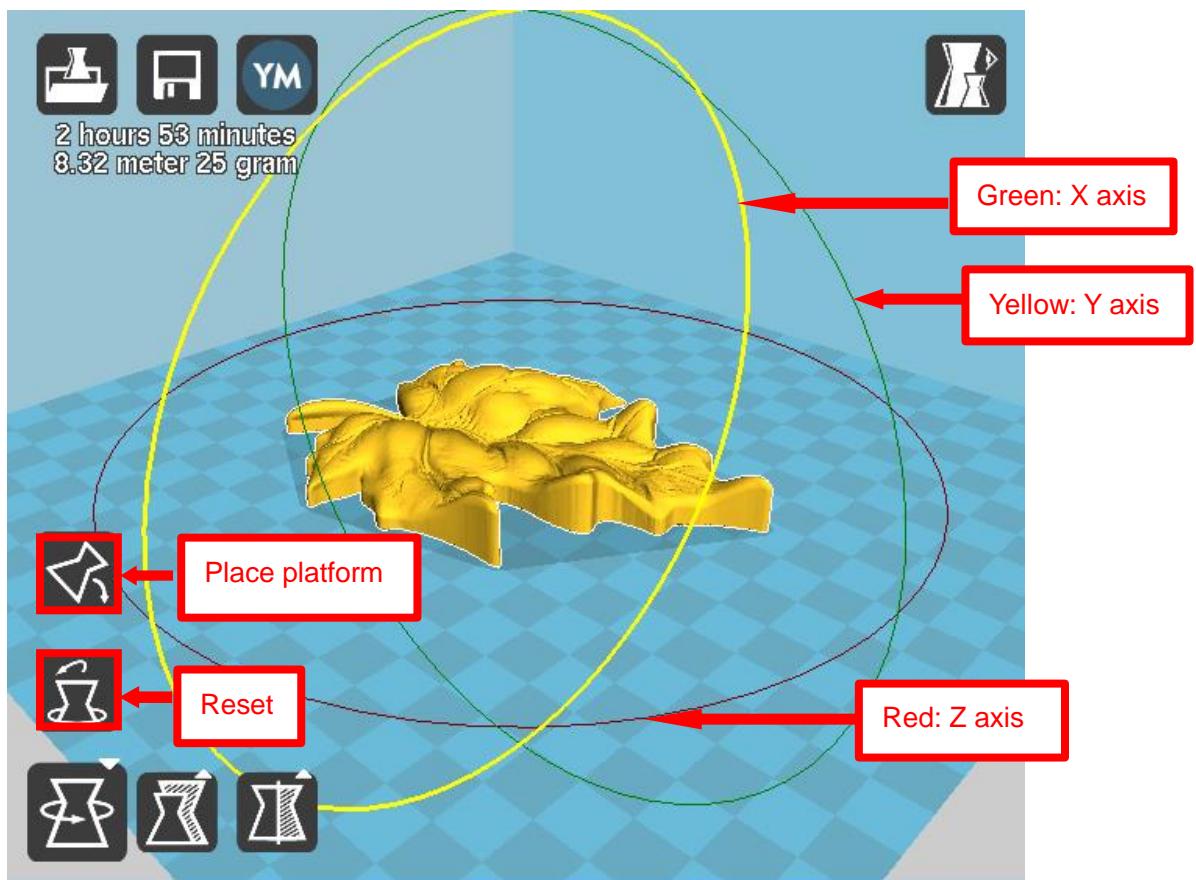
Left click the model and you will see the icon of "whirling, scaling, mirroring."

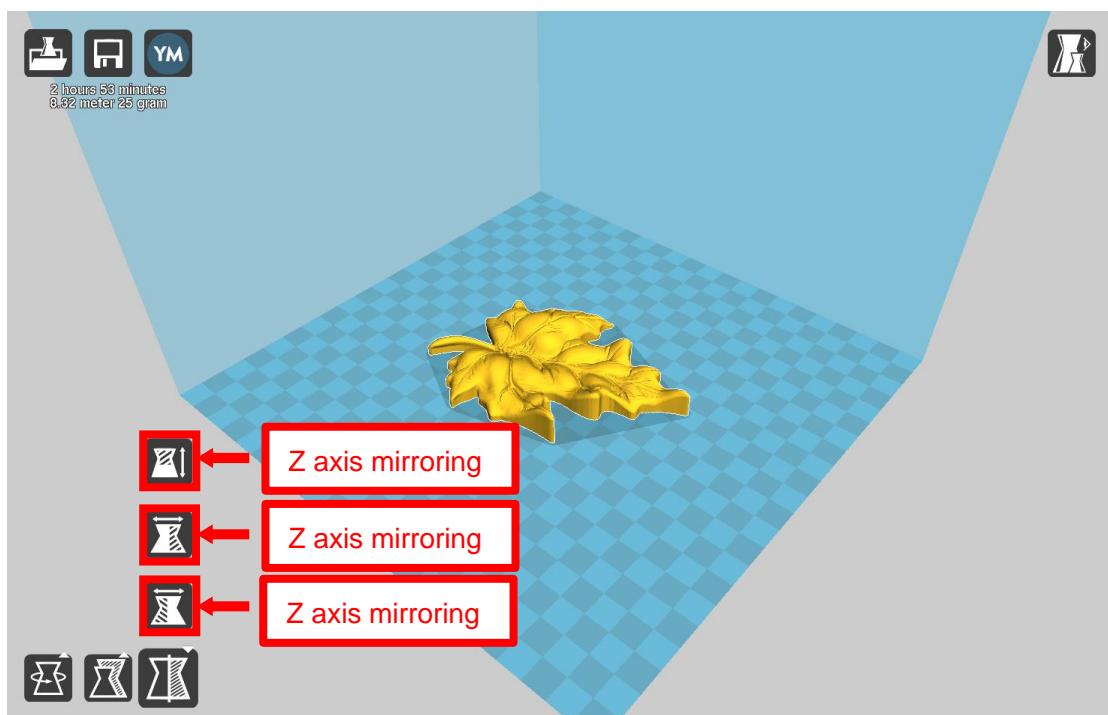
Left click to select model and move → move model.

Slide mouse wheel → scaling.

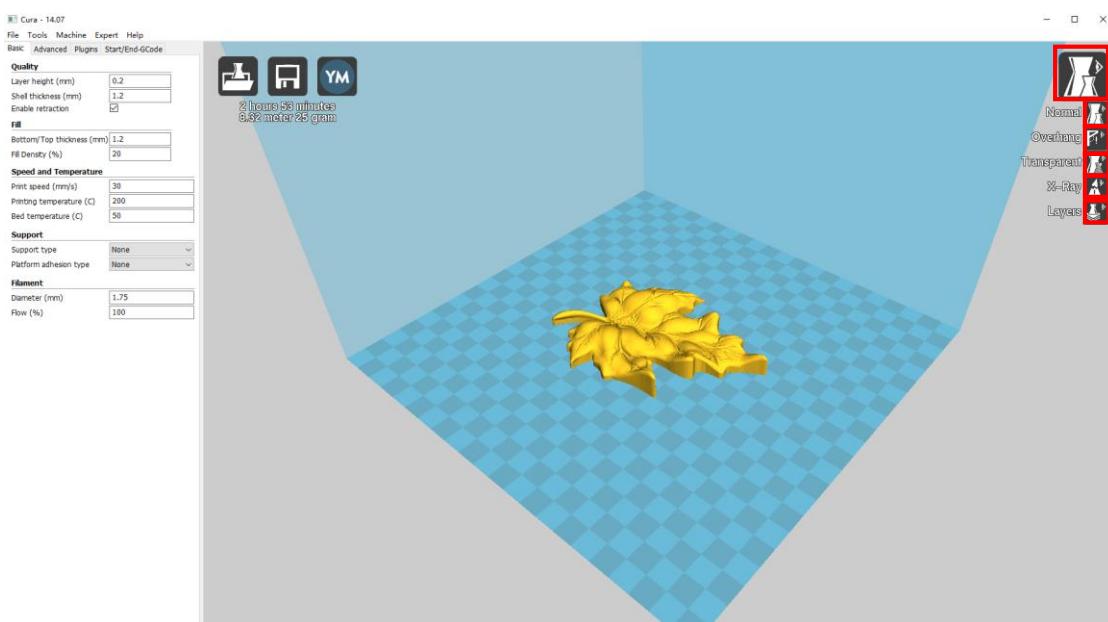
Right click to select model and move → whirling.

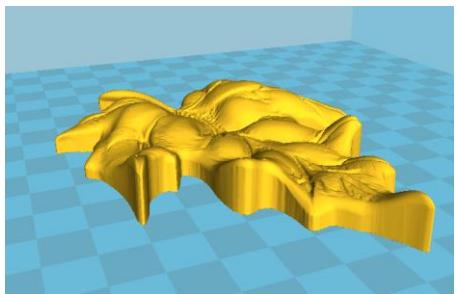
Shift + right click platform and move → move platform



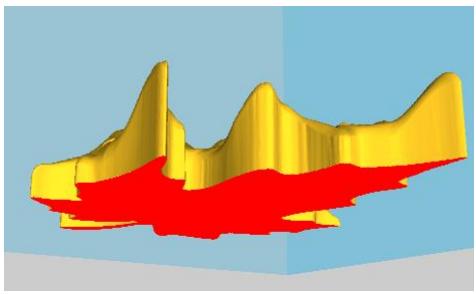


Attention: The model will be converse after mirroring.

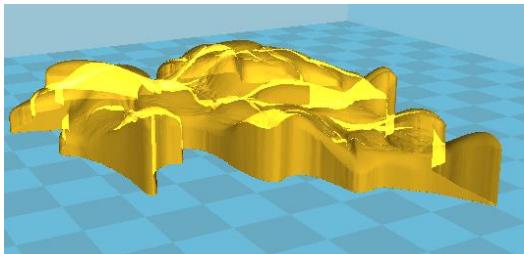




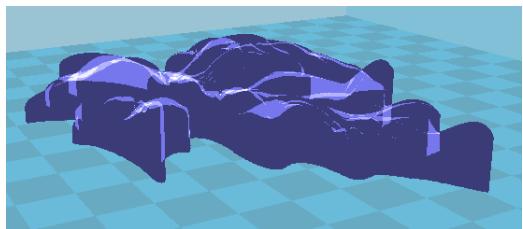
Normal: Most used.



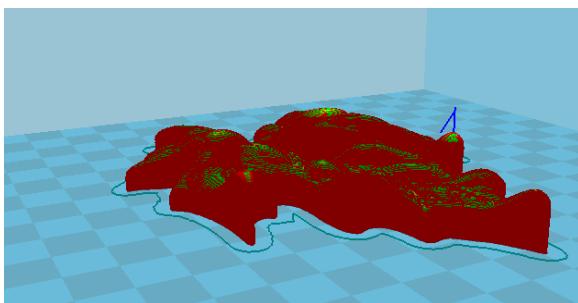
Overhang: Used to see the vacant part.



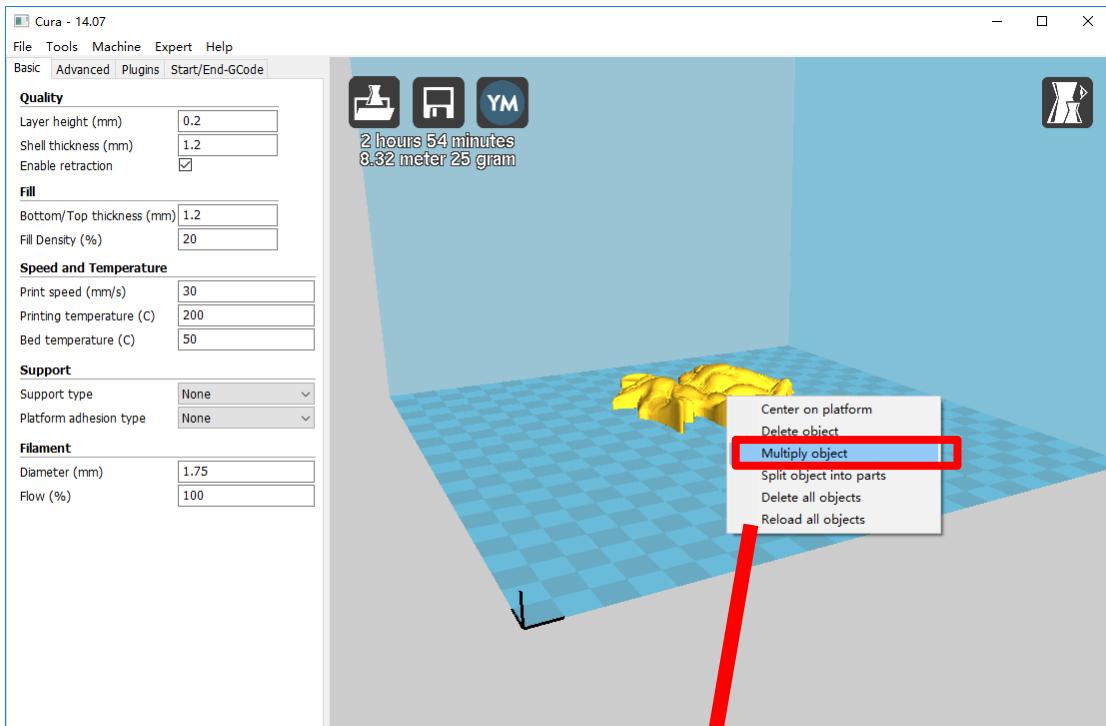
Transparent



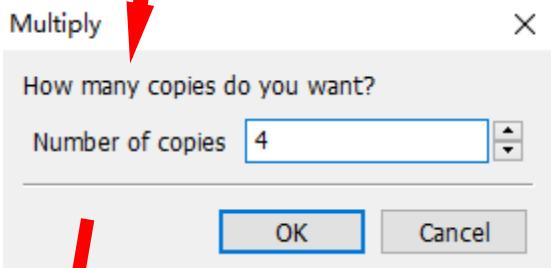
X-Ray



Layers:Used to simulate the effect of each layer and the path.



Right click model → Left click “Multiply object” → Example: copy 4 model



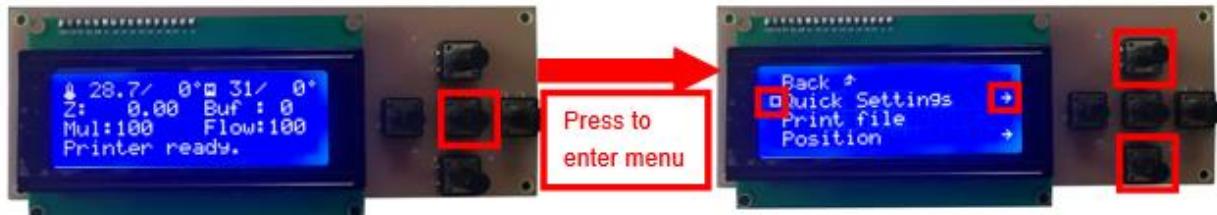
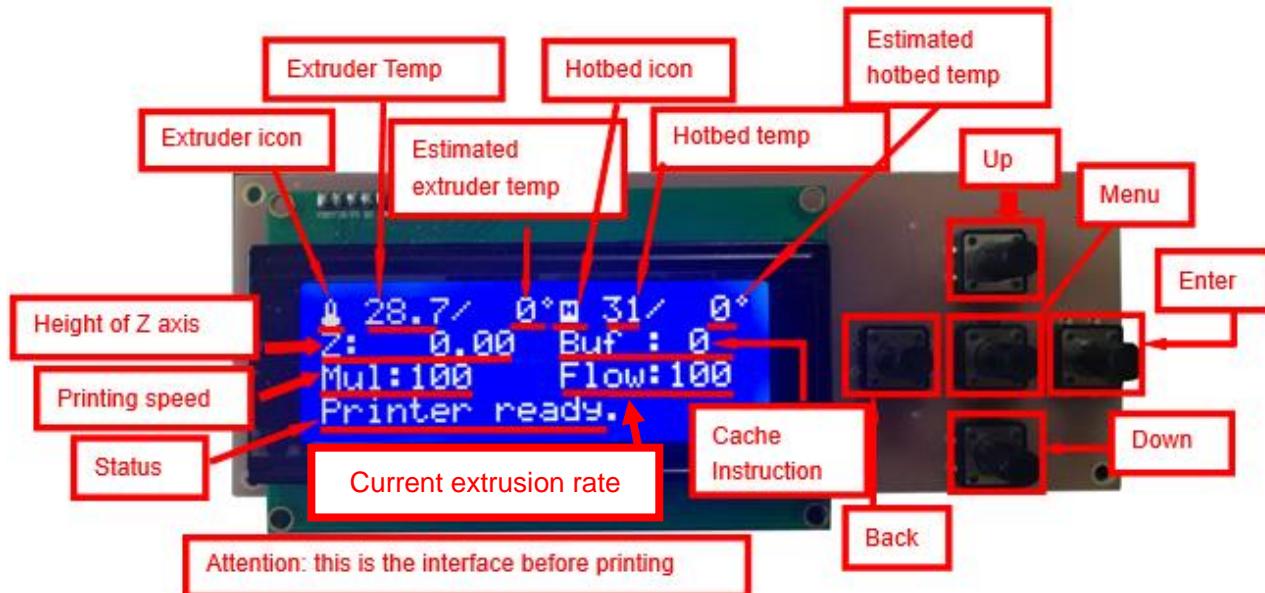
5 models in total



Attention: You can try other functions by yourself.

D. Printing Operation

1. Introduction of Display



Whole interface of menu

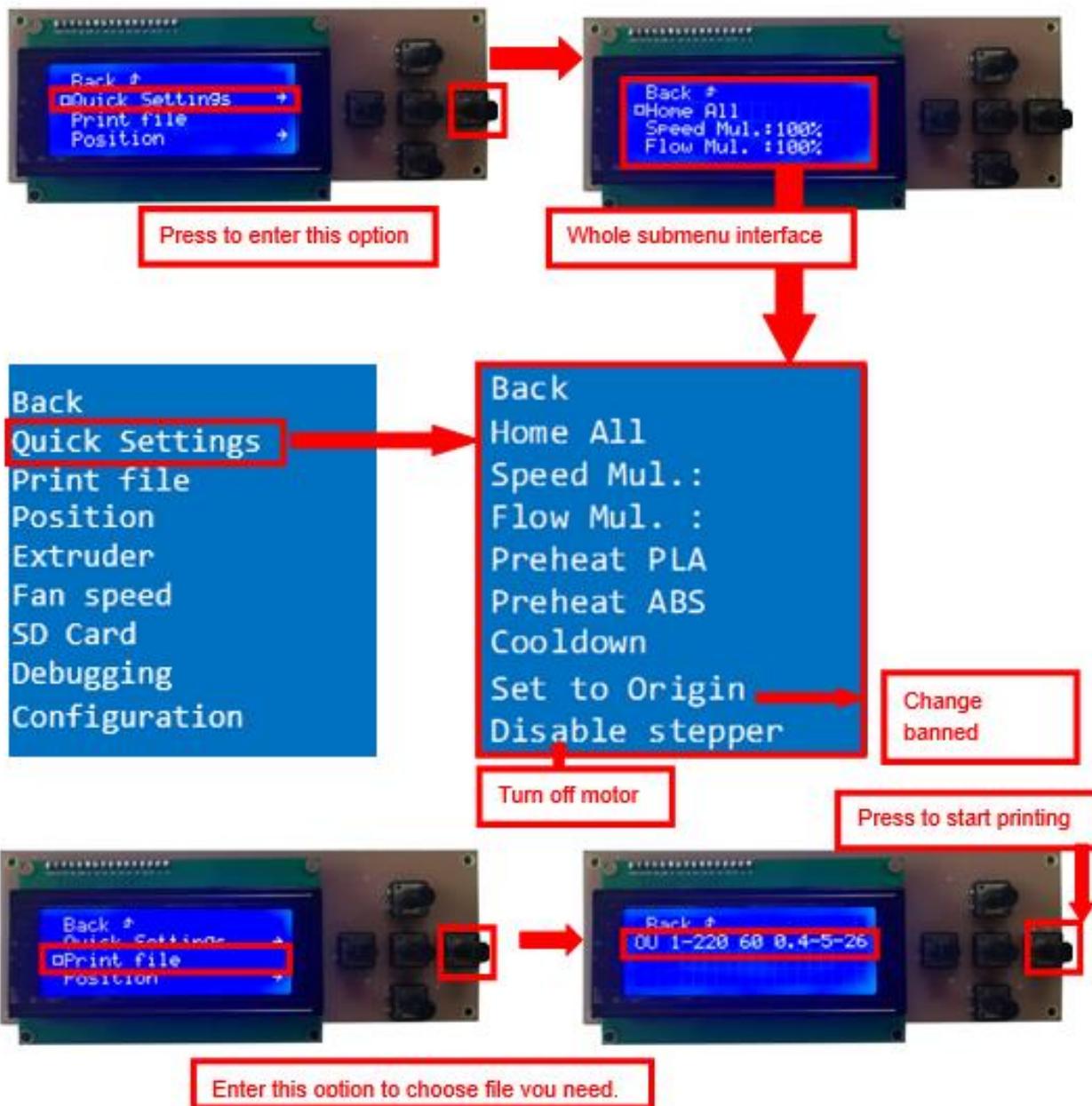
Back
Quick Settings
Print file
Position
Extruder
Fan speed
SD Card
Debugging
Configuration

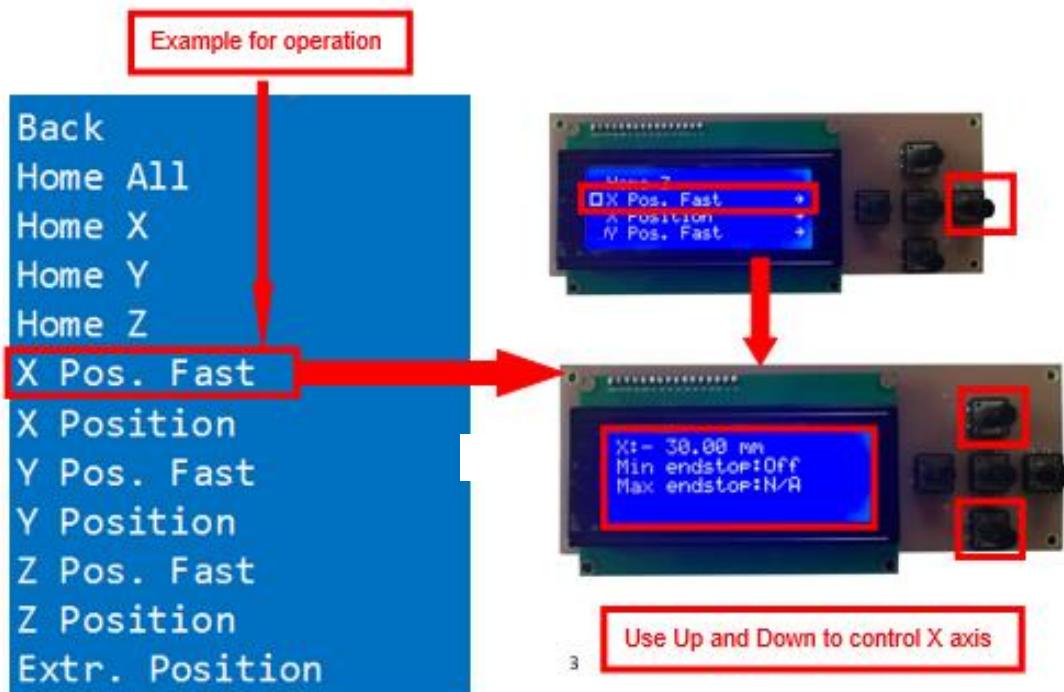
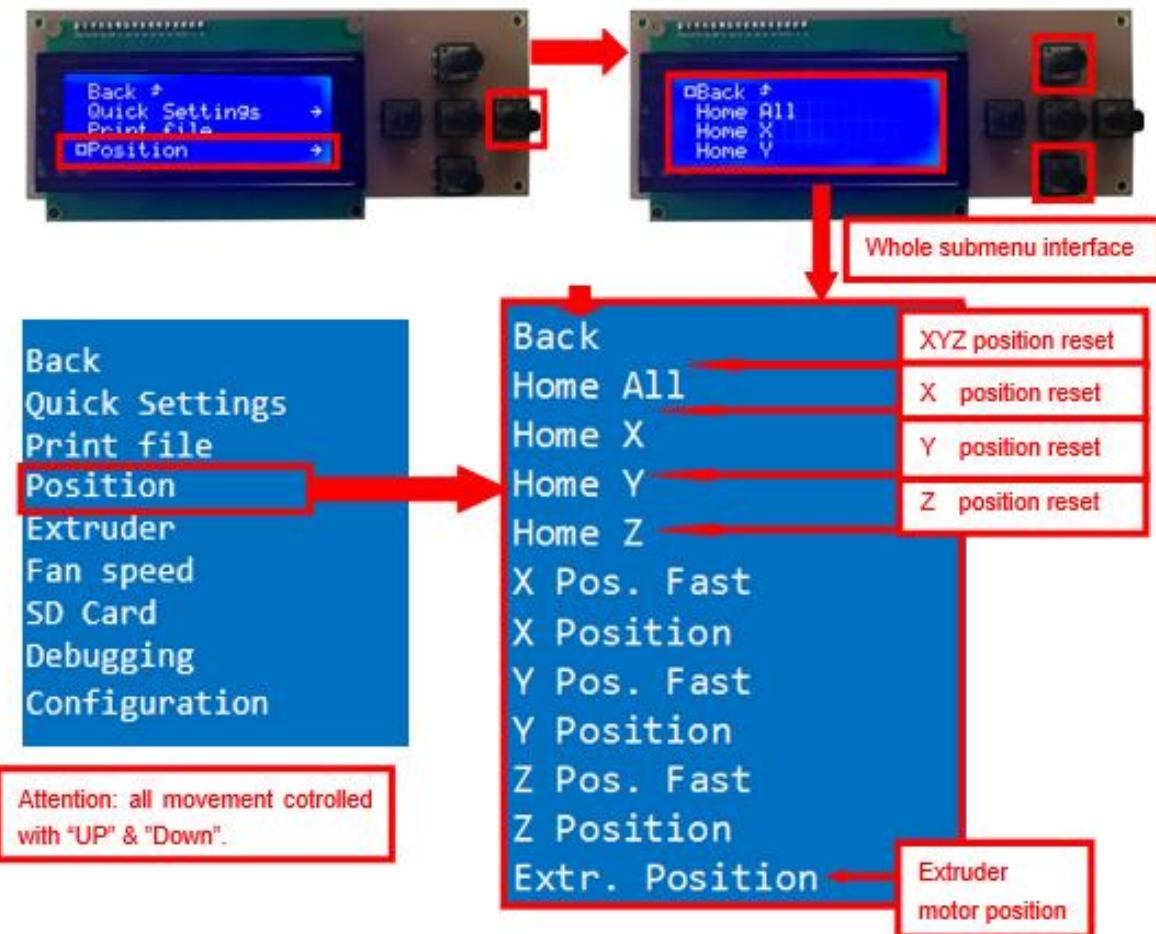
Change banned

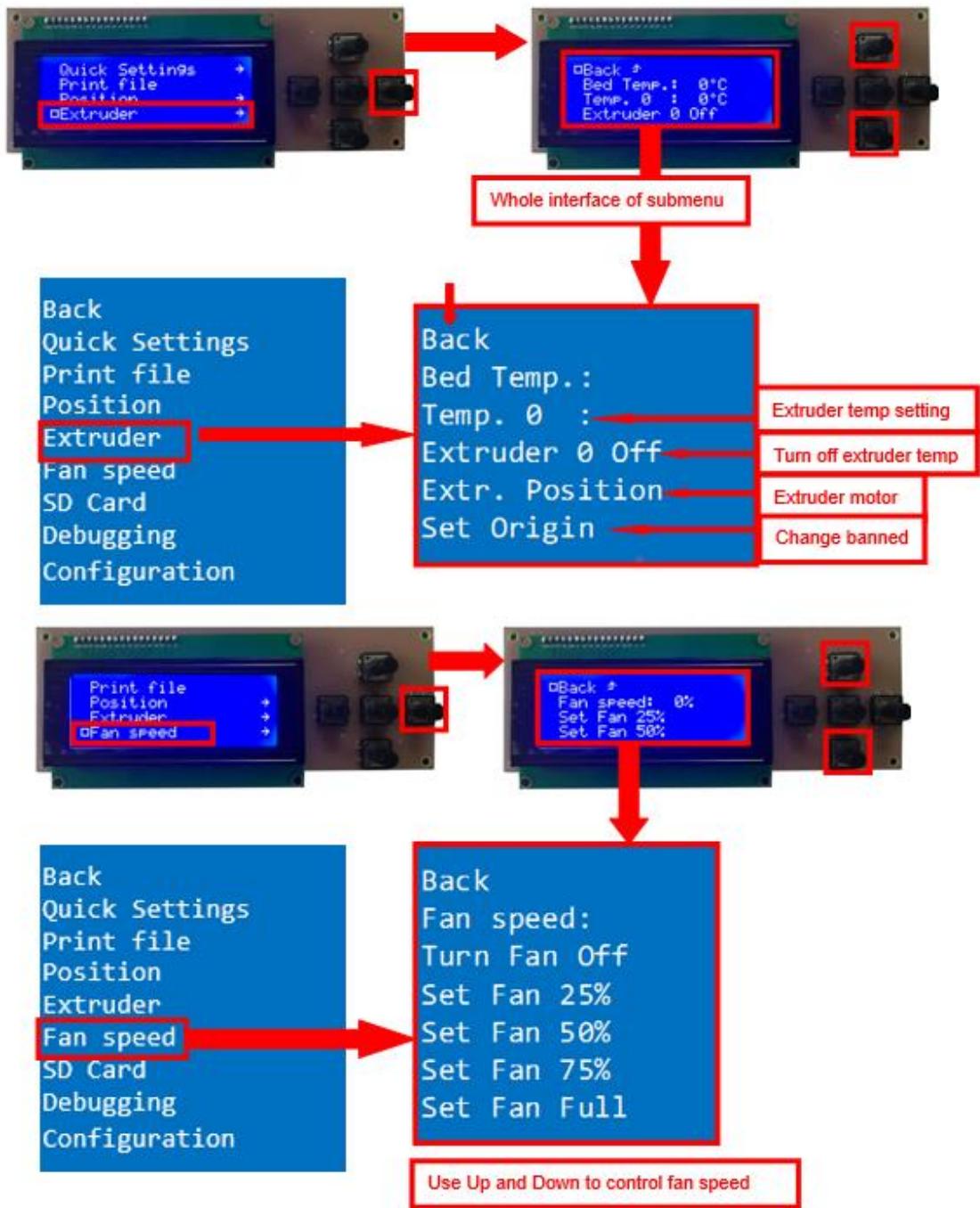
Change banned

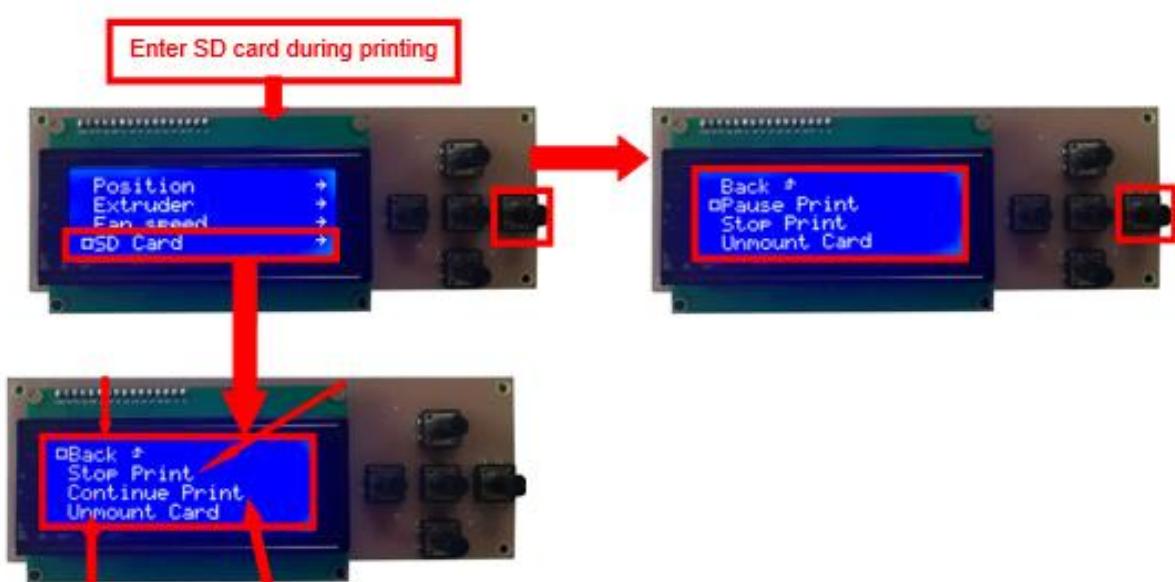
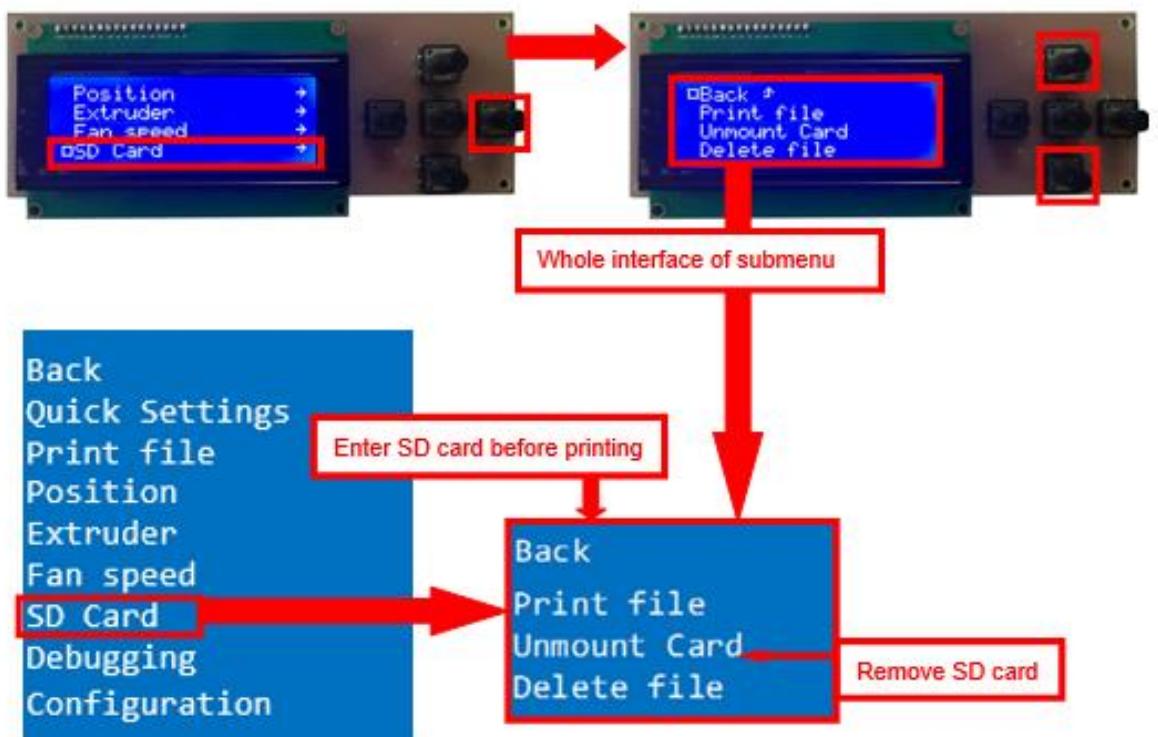
: it means you can enter this option to other settings

→ : it represents you can enter this option.

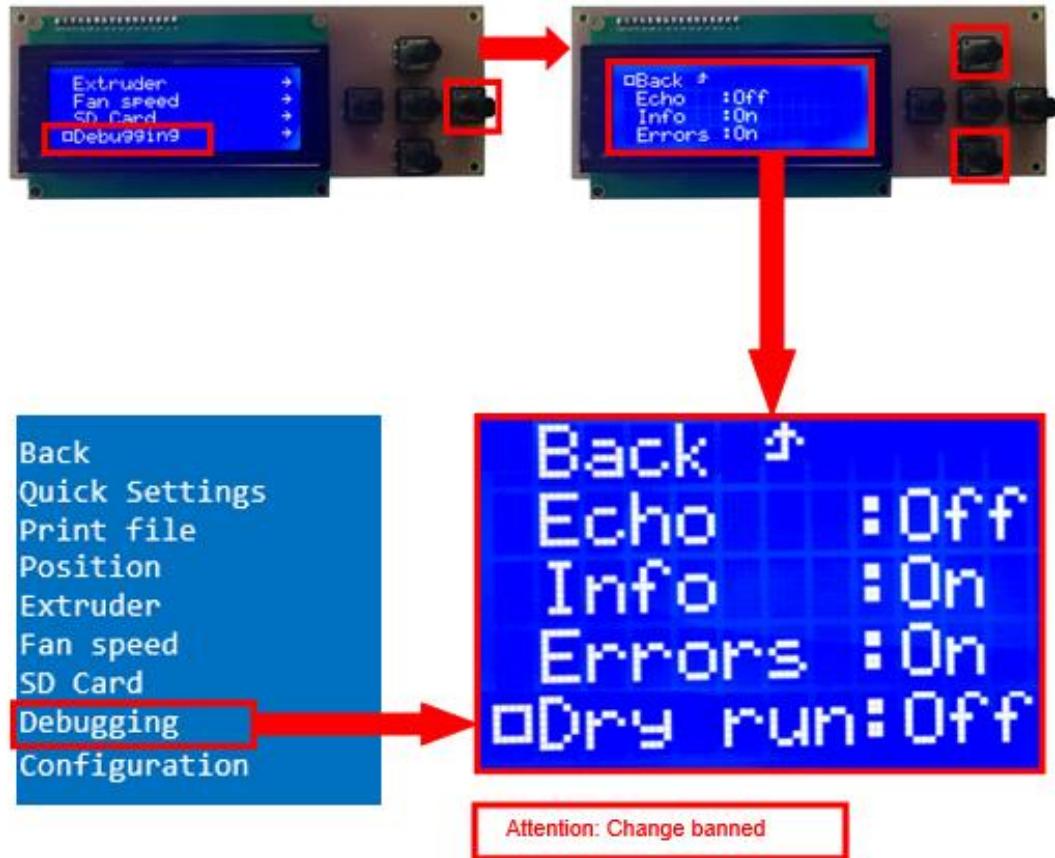




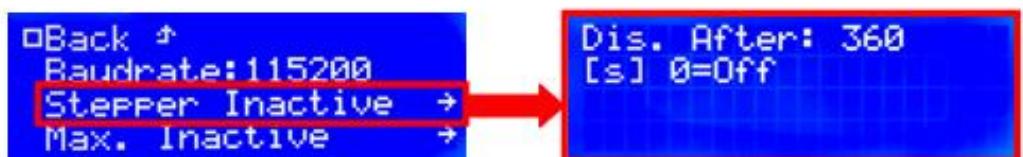
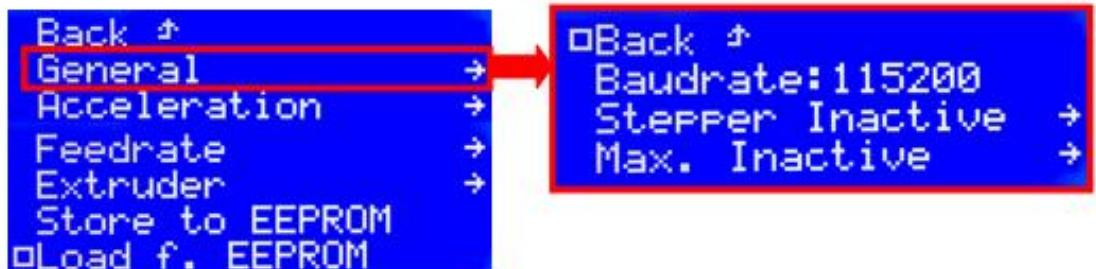
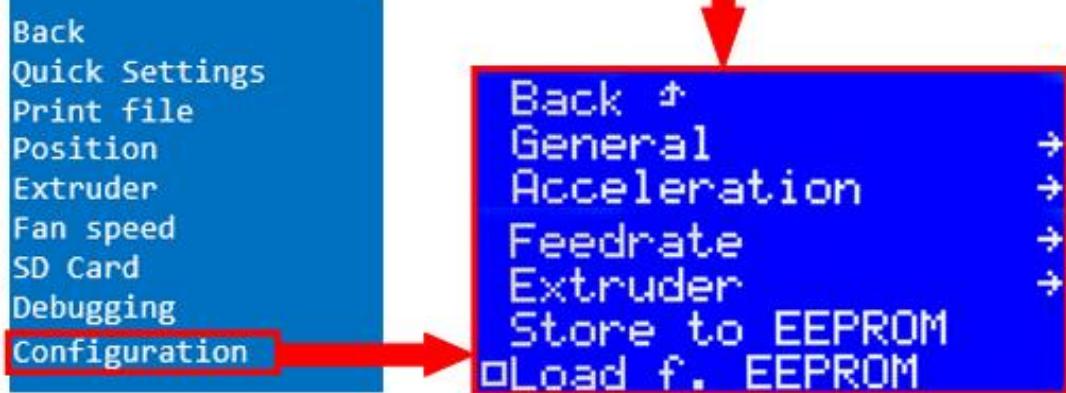
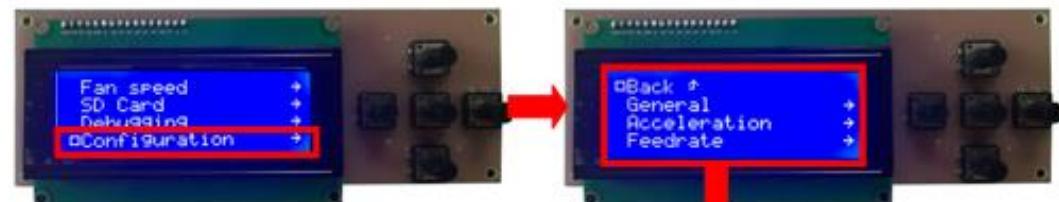




A8 can pause print during printing , during printing , enter "SD card" → "Pause print".
 It will stop printing in a few seconds. If you want to continue printing , enter "SD card"
 → "Continue print"



Attention: Change banned of these settings



Back ↺
Baudrate: 115200
Stepper Inactive →
Max. Inactive →

Dis. After: 0
[s] 0=Off

Back ↺
General →
Acceleration →
Feedrate →
Extruder →
Store to EEPROM
Load f. EEPROM

Back ↺
Print X: 400
Print Y: 400
Print Z: 400

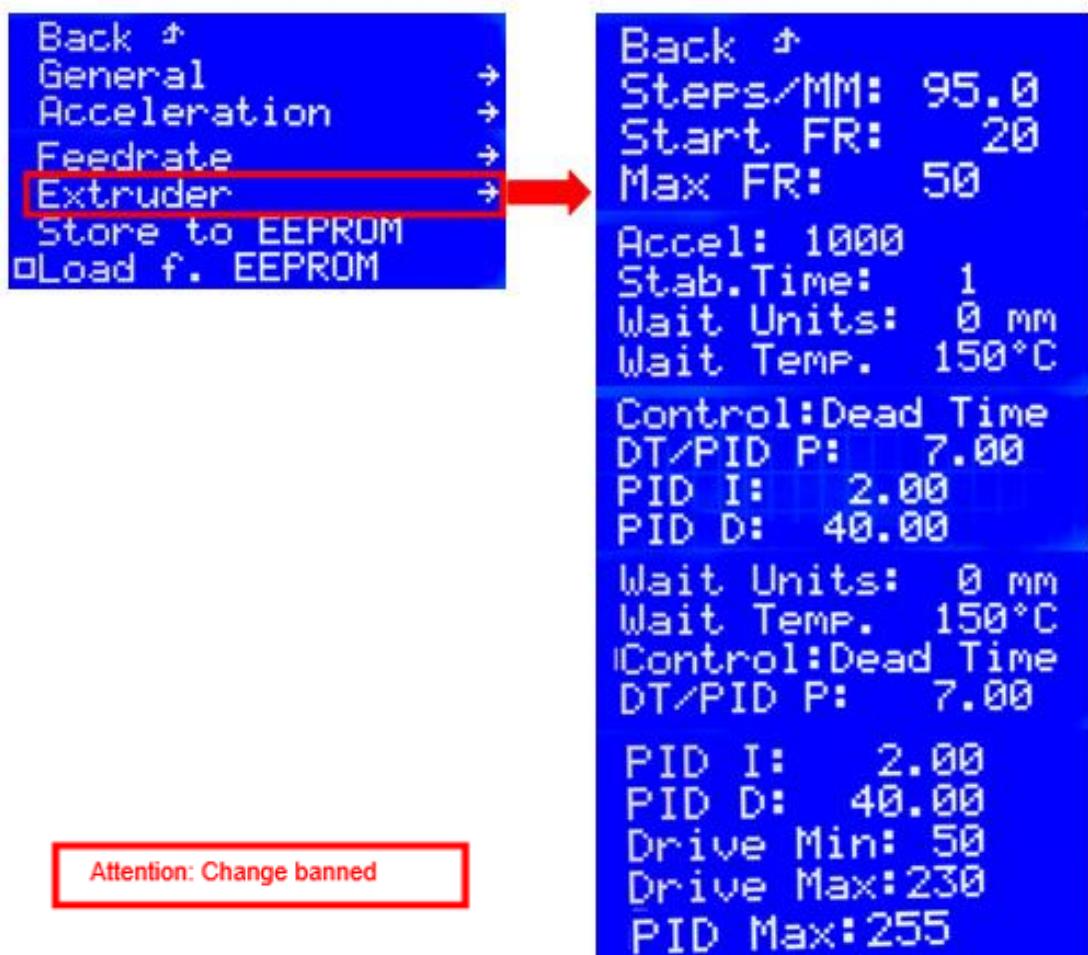
Move X : 400
Move Y : 400
Move Z : 400
Jerk : 20.0

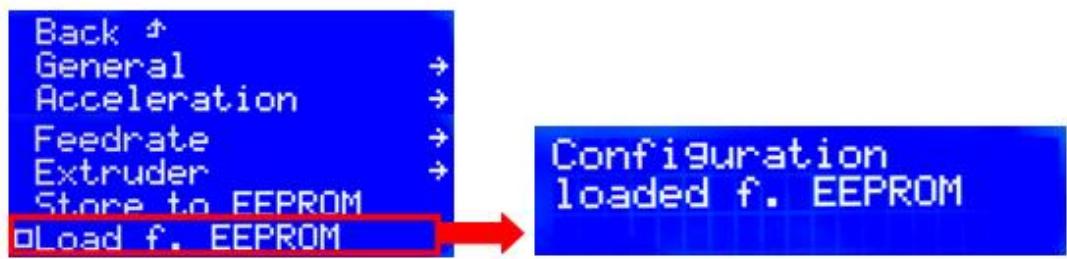
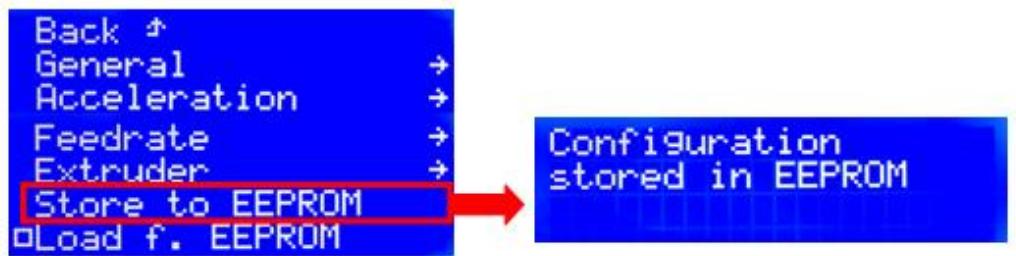
Z-Jerk : 0.3

Back ↺
General →
Acceleration →
Feedrate →
Extruder →
Store to EEPROM
Load f. EEPROM

Back ↺
Max X: 400
Max Y: 400
Max Z: 8
Home X: 100
Home Y: 100
Home Z: 4

Attention: Change banned





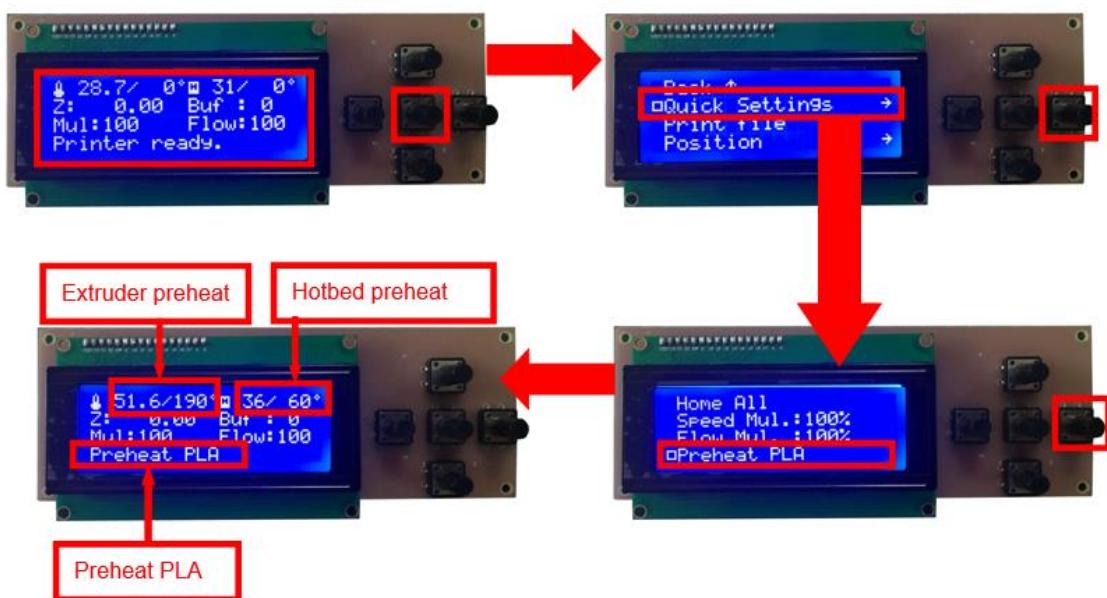
Attention: Change banned

2. Install Filament

2.1 Set Preheat Mode

Attention: Before filament installation, you need to preheat the printer. Use PLA as reference as below:

Press Menu → Quick Settings → Preheat PLA
The printer begins to preheat (You can back to main interface to check)

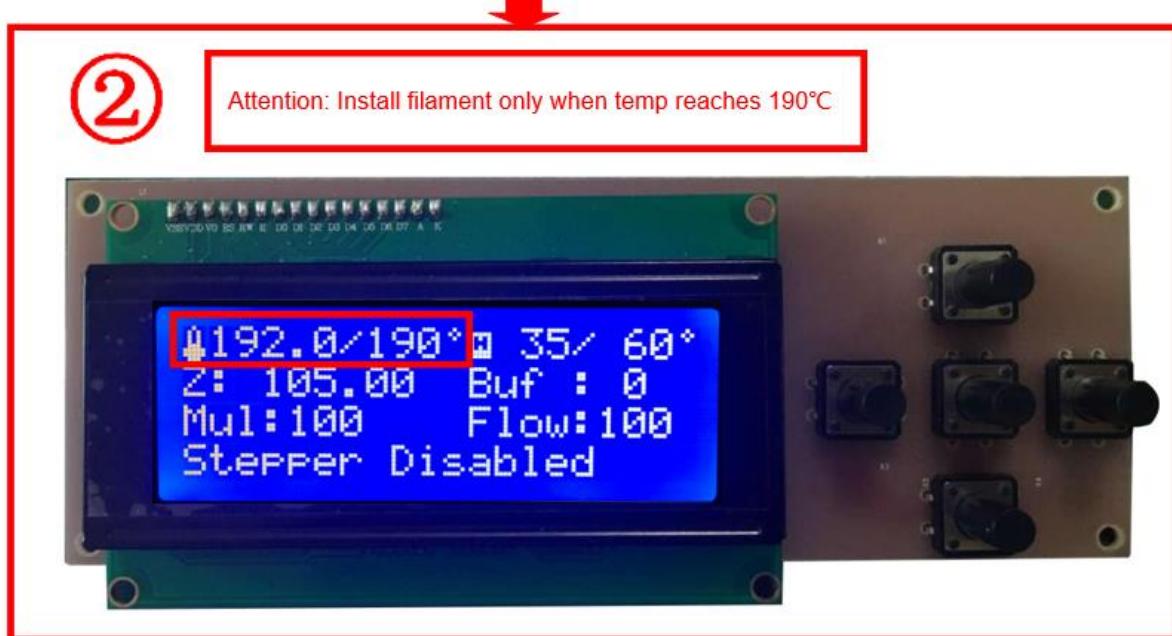
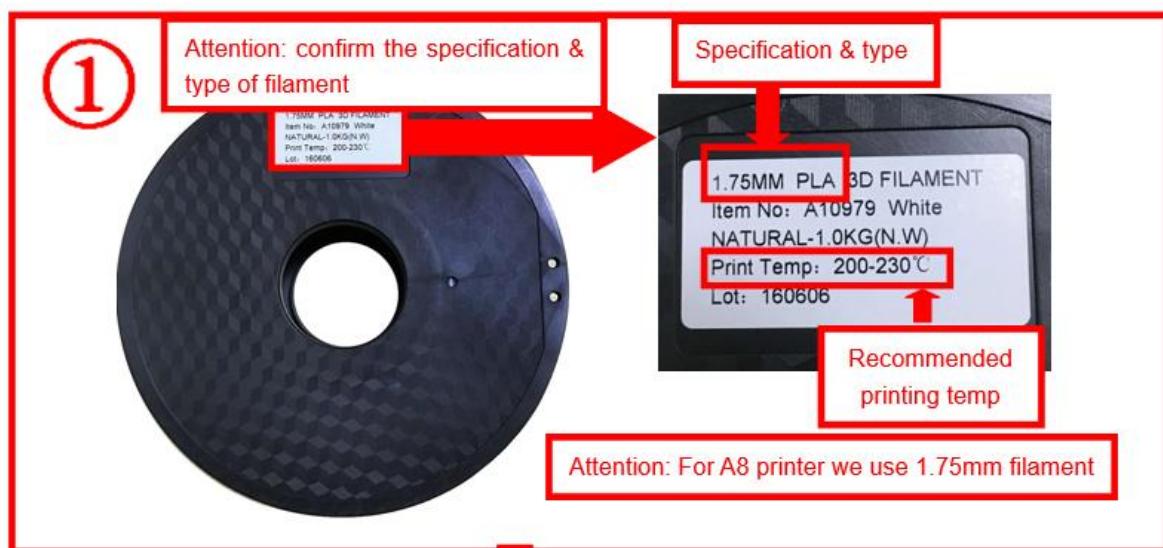


Attention: Please choose "Preheat ABS" if you want to print with ABS.

2.2 Filament Installation

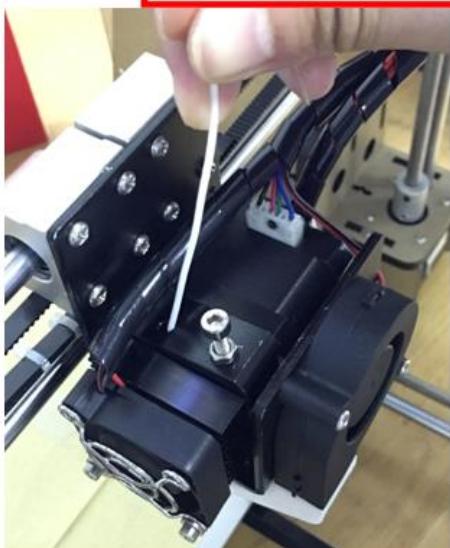
Attention: Put filament into printer only when extruder temp reaches 190°C. (Use PLA as example)

Confirm Extruder Temp has reached 190°C → 1 roll PLA → Stroke the filament head straight → Press extruder screw. Meanwhile ,hold the white wind mouth → Meanwhile, stick filament into the extruder quickly until filament goes out from the nozzle → Filament installation succeed

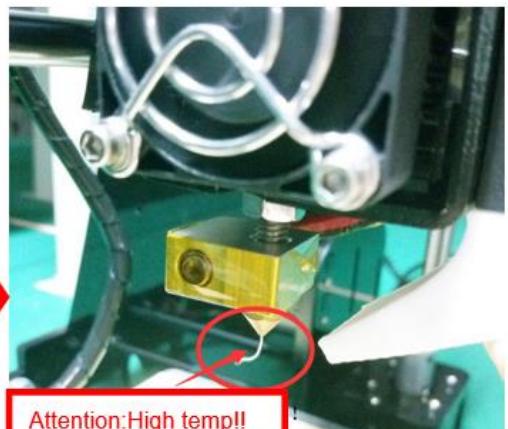


③

Stroke the filament head straight → Press extruder screw. Meanwhile ,hold the white wind mouth → Meanwhile, install filament into the extruder quickly until filament liquid goes out from the nozzle



④



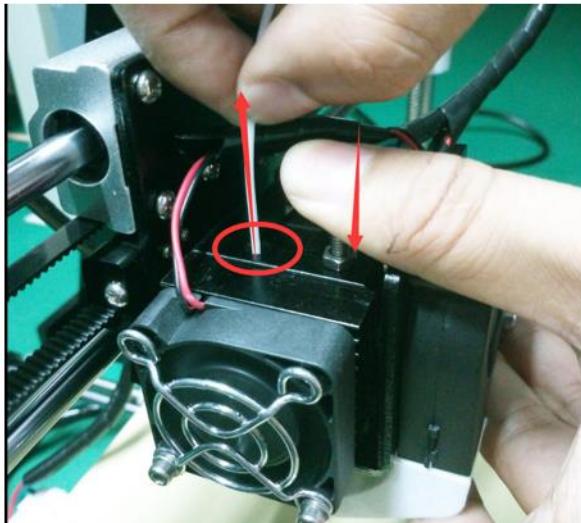
If the shape of output filament is average , means installation is good.

Press extruder screw, hold on wind mouth. Stick filament in quickly until filament does out from extruder

Filament installation succeed

2.3 Pull out filament

When Change filament/Long-term not in use of printer , you need to pull out filament.



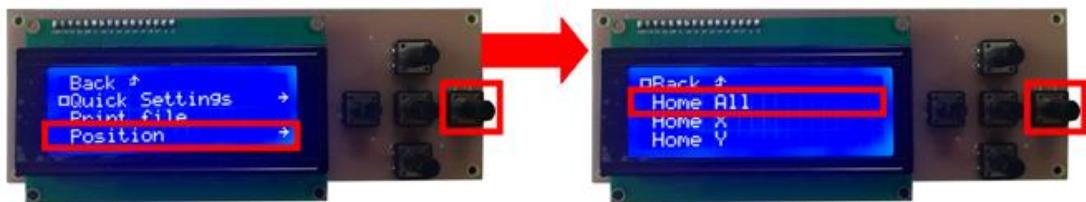
Use PLA as example

1. Preheat extruder to 190°C
2. Press extruder screw, hold wind mouth.
Meanwhile, stick filament down for a few length, then pull out with average speed.

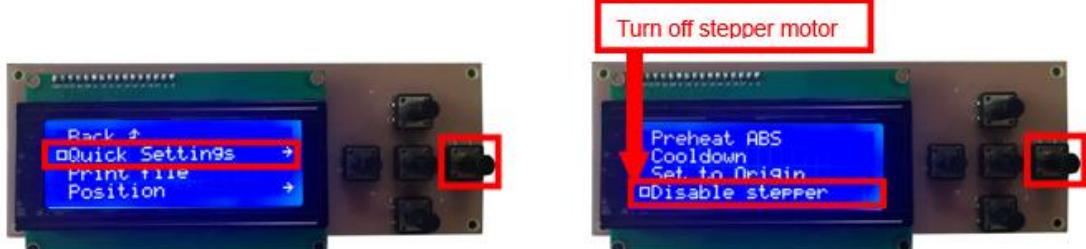
Precautions:

1. Do not stick down for long length in order to avoid failure of pulling out. Replace filament timely.
2. Please confirm you have preheated the extruder to 190°C. Do not pull out before 190°C , or it will cause irreparable damage.

3. Platform Adjustment

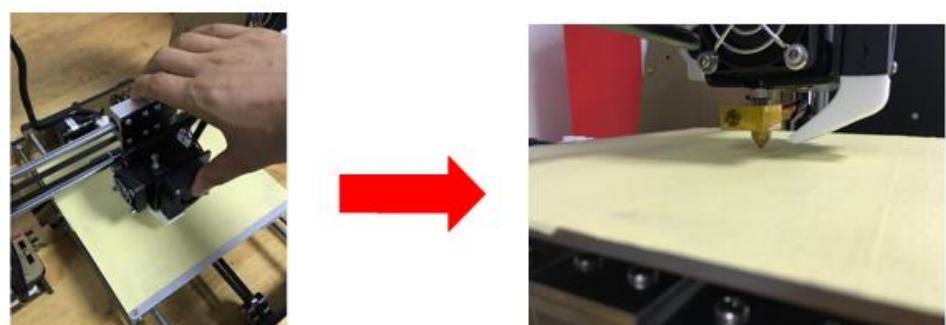


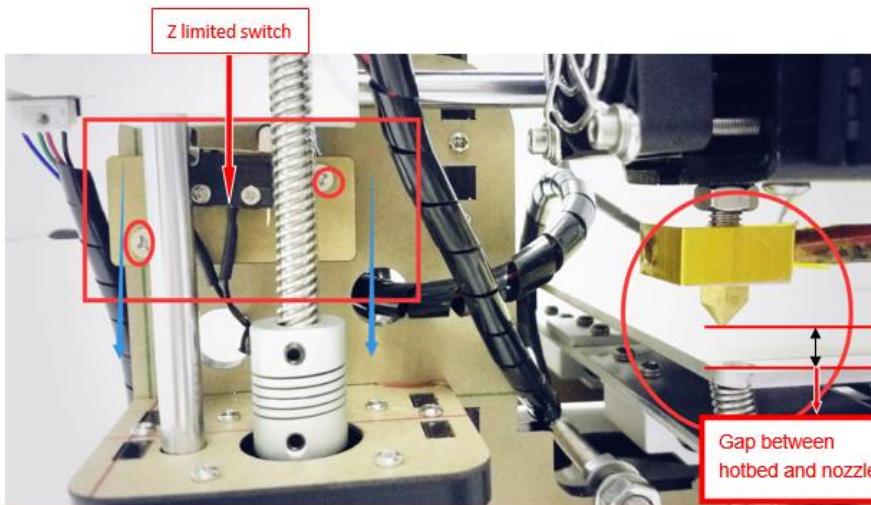
1. Choose "Position" → "Home all" , printer will move to limited switch until it stops.



2. Turn off stepper motor: based on step 1 , enter "Quick settings" → "Disable stepper"

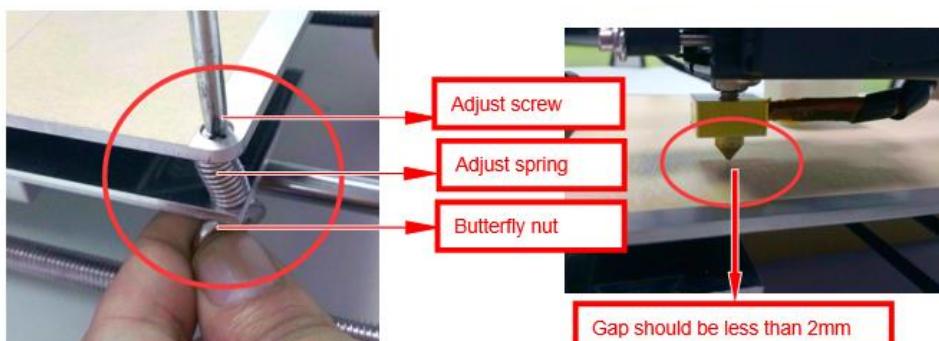
3. Please manually move nozzle to platform and check the gap between nozzle and platform.





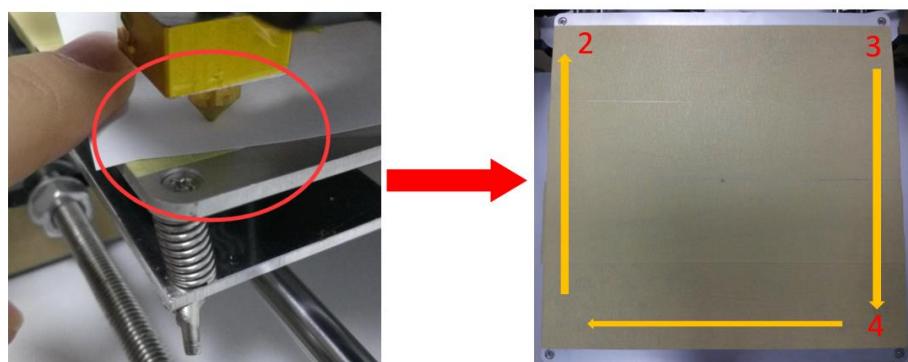
4. When the gap is more than 2mm, you need to adjust the height of Z limited switch.

Example: When the gap is 12mm , you need to adjust limited switch down by 10mm.The rest 2mm can adjust by spring on the hotbed.



After adjustment of Z limited switch, you need to reset printer and turn off stepper motor, move extruder to the center. You can check the gap better with these conditions.

5. Adjust the gap to about 0.2mm to satisfy printing needs. Move extruder to a corner of platform, adjust the springs one by one . Use 1 or 2 A4 paper to test if they can go through the gap with some resistance.



After adjustment of springs, reset printer and close stepper motor to test. Use A4 paper to test the gap.

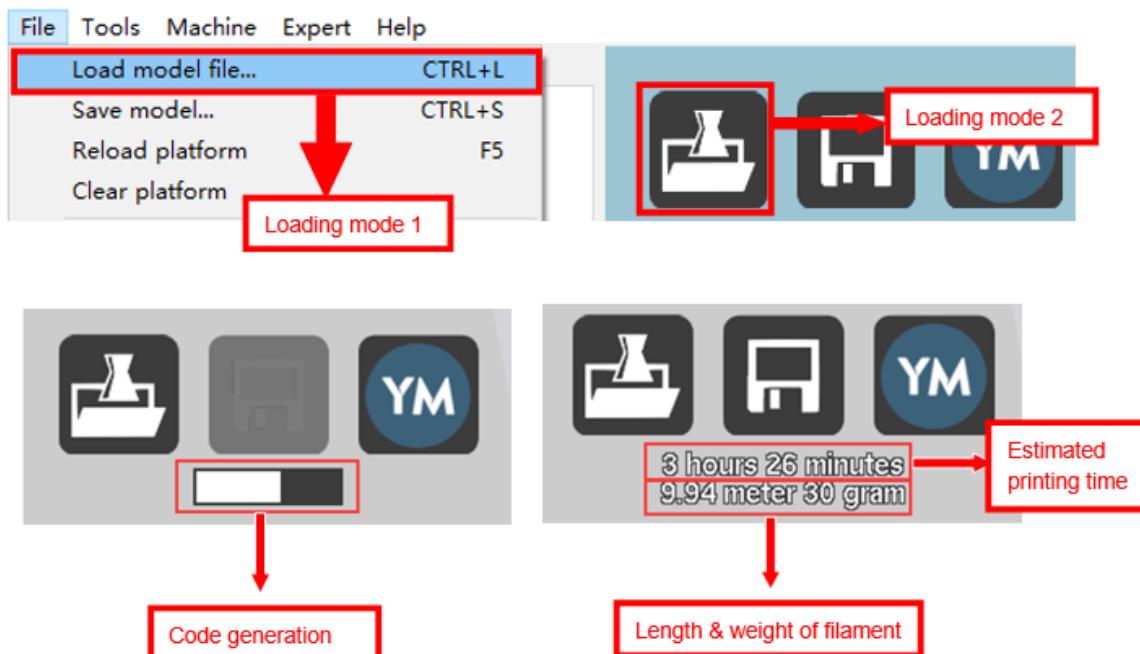
Tips: When you are familiar with the printer with time going by, we can adjust while it's printing. It's because the printing speed is slow at the beginning so that there's enough time for adjustment. Meanwhile, the printing effect will be better.

4.Printing

1) SD Card Offline Printing

a. Loading mode

Cura supports STL file & G-code file.

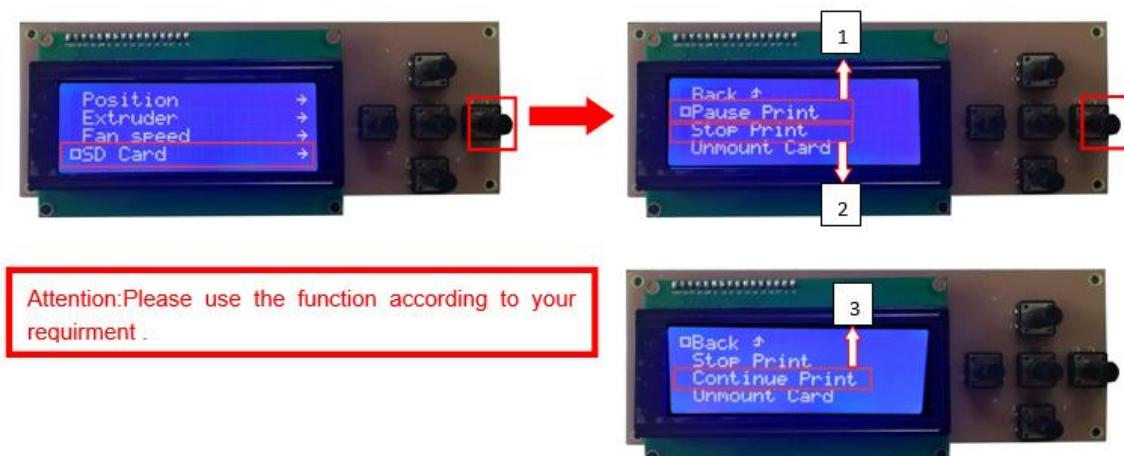


b. Code Saving

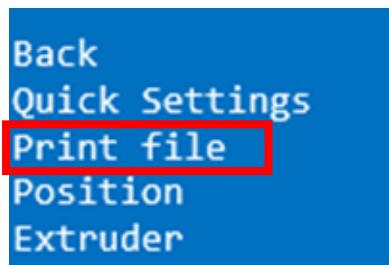
Copy file to SD card . Then connect SD card to printer, click reset. Picture below shows the location of print file , there are 2 methods to find print file.

c. Introduction of Stop print , Pause print , Continue Print:

Only when the printer is printing can we use Stop print, Pause print, Continue Print.



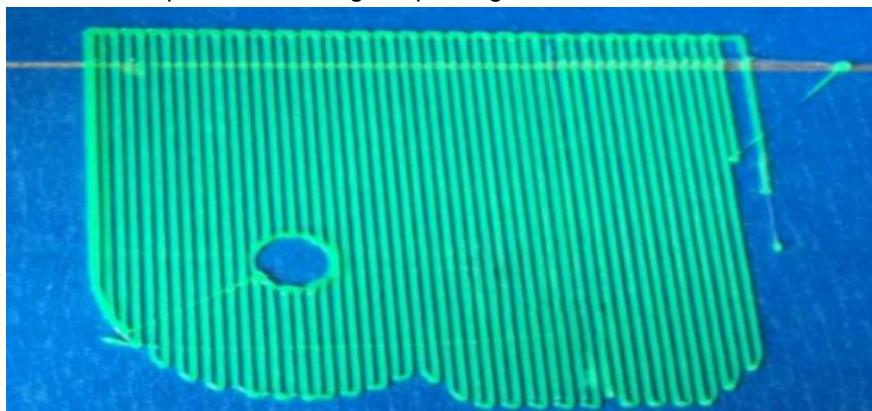
d. Print model



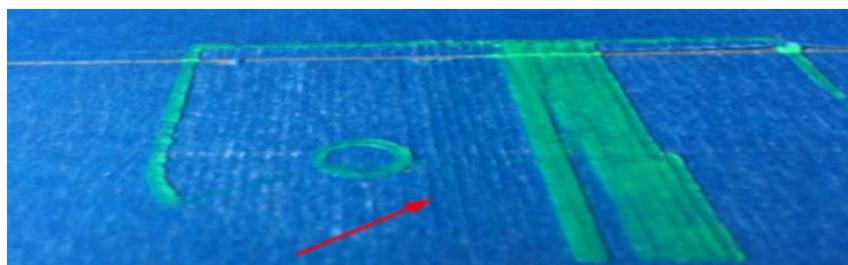
The printer will start printing automatically when the extruder & hotbed reaches estimated temp after you choose print model .
Attention: If the print bottom fails to stick to platform tight , you can adjust the platform to make it tight during printing.

e. Judgment of the gap between nozzle and platform.

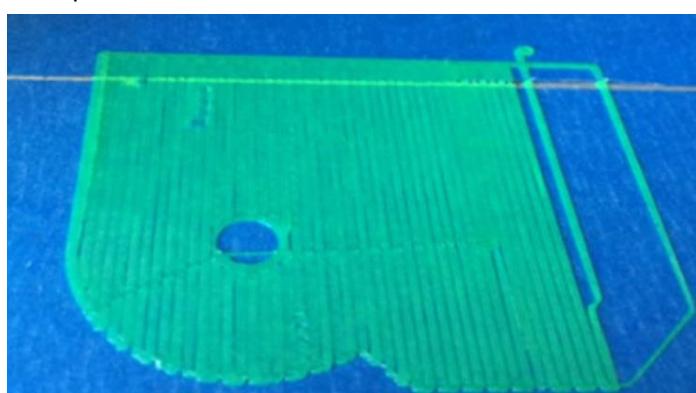
1. Too big gap: The printed model is uneven, curled with gap. It means the gap is too big for filament to reach the platform, making the printing effect so bad.



2. Too close gap: The printed model edge has irregular projections. It means the gap is too close to print normally. Sometimes it even cannot output filament



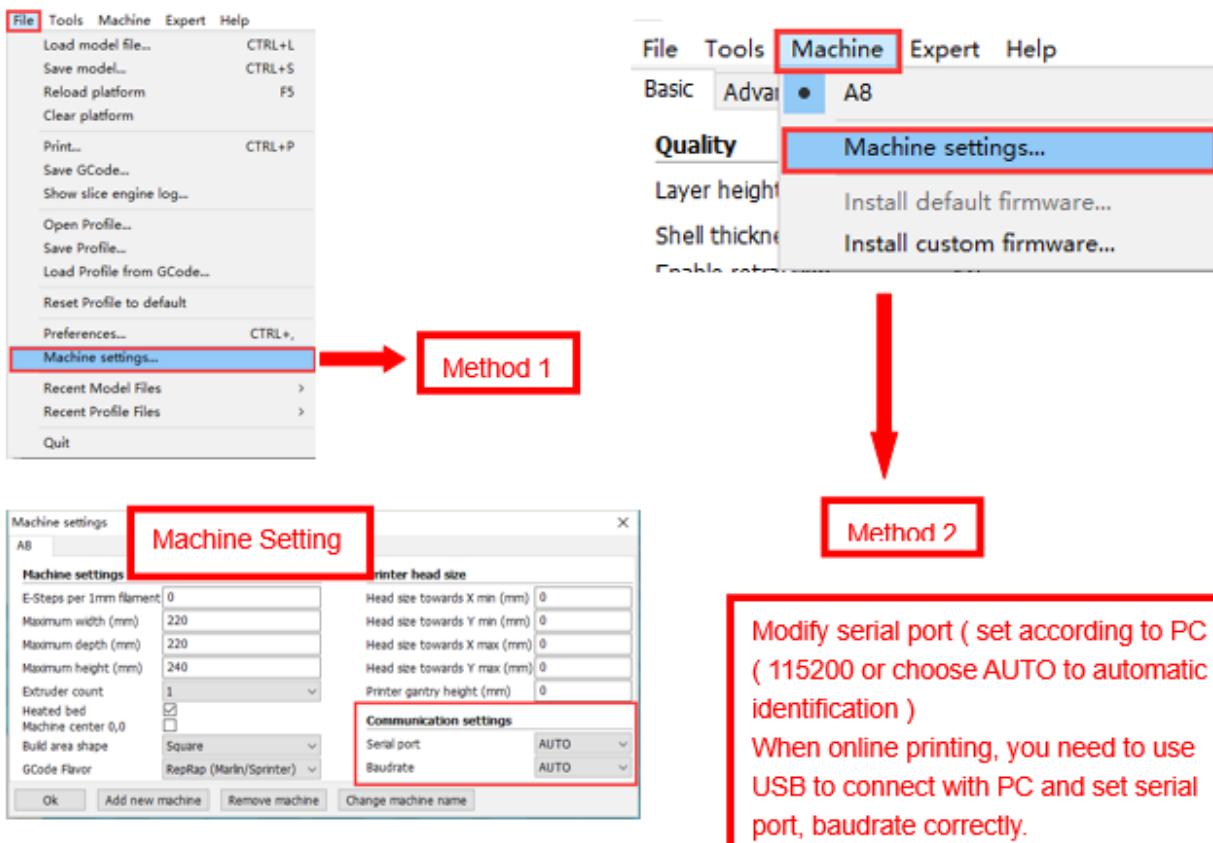
3. Appropriate distance: Printed model flat with no gap, no glitches. It means the distance is appropriate to print.



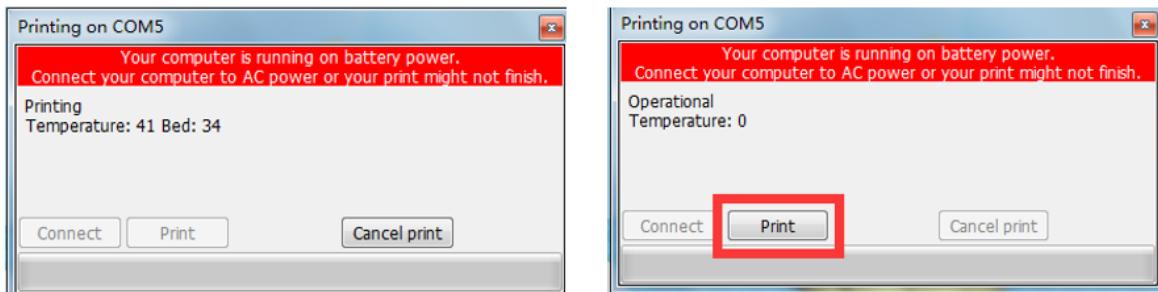
Wait to print complete after gap adjustment.

2) USB Online Printing

a. Machine settings (Use USB to connect to PC)



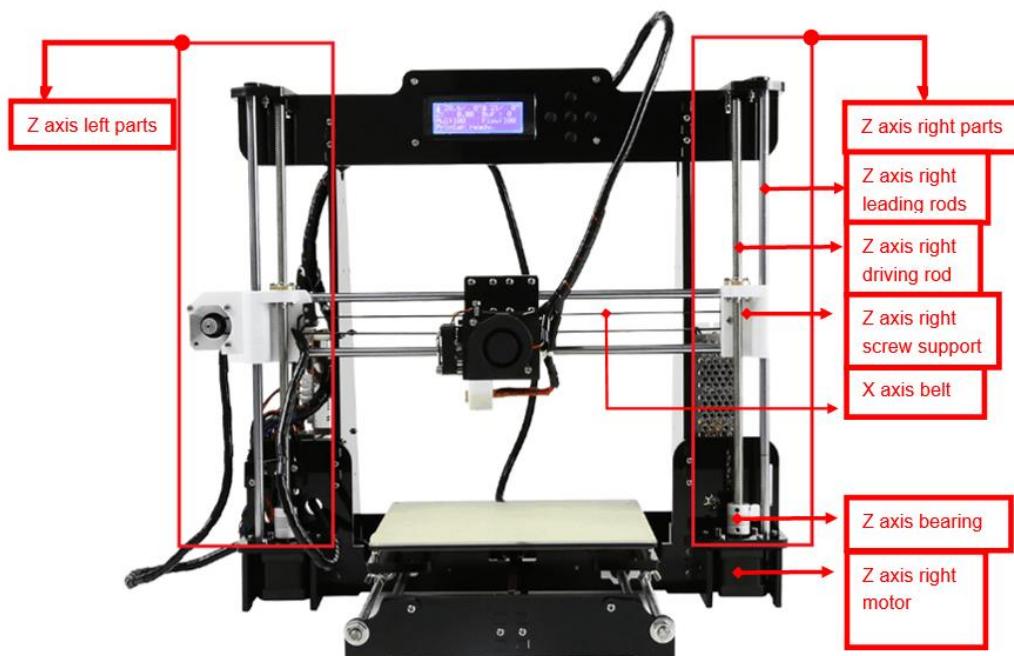
b. Online Pringting



The printer will start printing automatically when the extruder & hotbed reaches estimated temp after you choose print model. You can also modify temp in this interface.

E. FAQ and Solution

1. Z Axis Adjustment



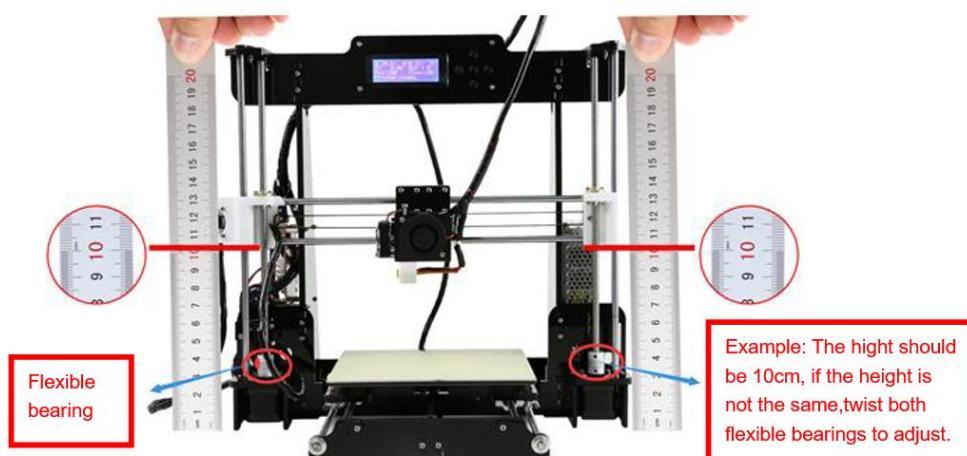
Use right Z axis for reference as shown above.

Reasons for Z axis's not smooth movement:

1. The height of both Z axis screw support is apparently different.
2. Large deviation of leading rod and motor rod's concentricity.
3. X belt is too tight.

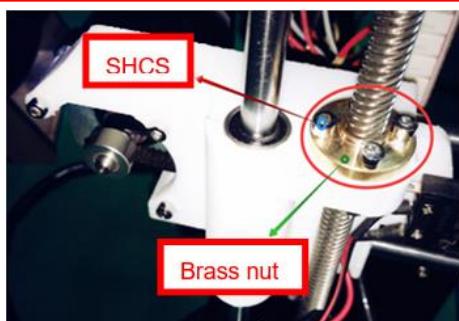
During installation, we need to test moving parts:

1. Preparation: Before Z axis moving adjustment, please confirm the height of both Z axis screw support is the same. (Keep the same height of two white parts)

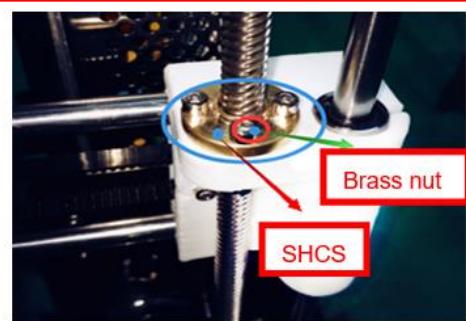


2. Adjust concentricity

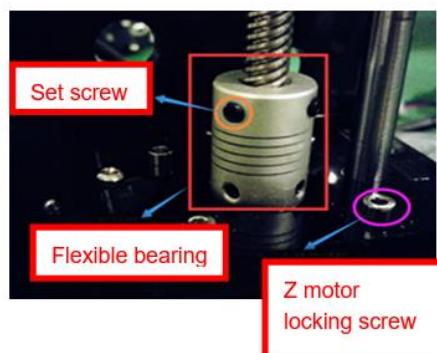
1. Click to adjust Z axis to move (Position → Z pos.Fast → +/-). If it cannot move smoothly, you need to adjust the unsmooth side's screw support. Try to keep them at the same height.
2. We can also tight/loose the Z motor screw according to requirements. This is to correct the deviation in the first time installation. Please take steps as follows to lock screws,



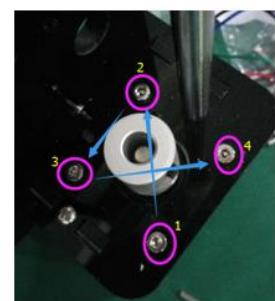
Z axis left screw support



Z axis right screw support



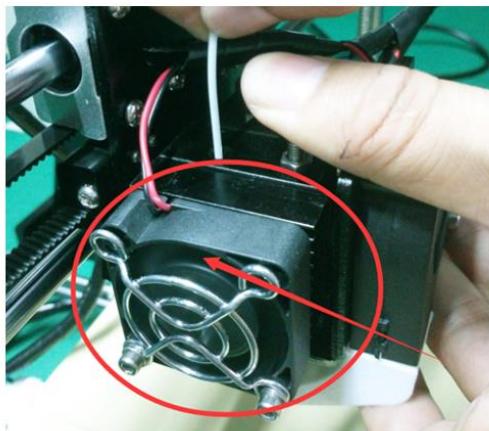
Z motor & Flexible bearing



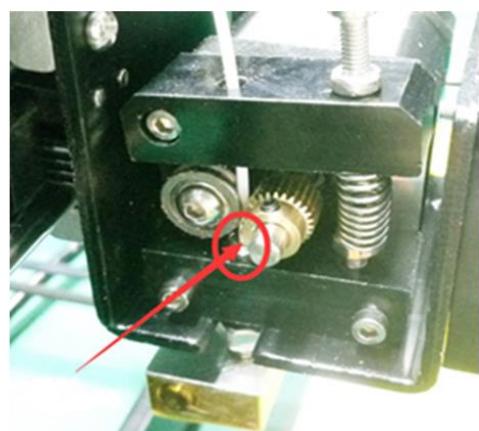
Z motor screw locking sequence

2. Nozzle blocking

Tips: We have removed fan to show obviously. Please consider movement according to actual requirement.

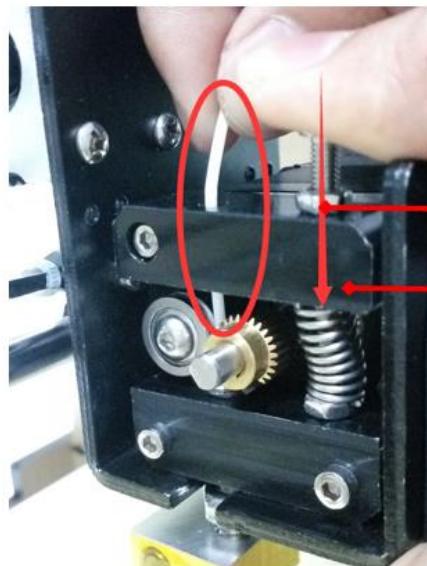
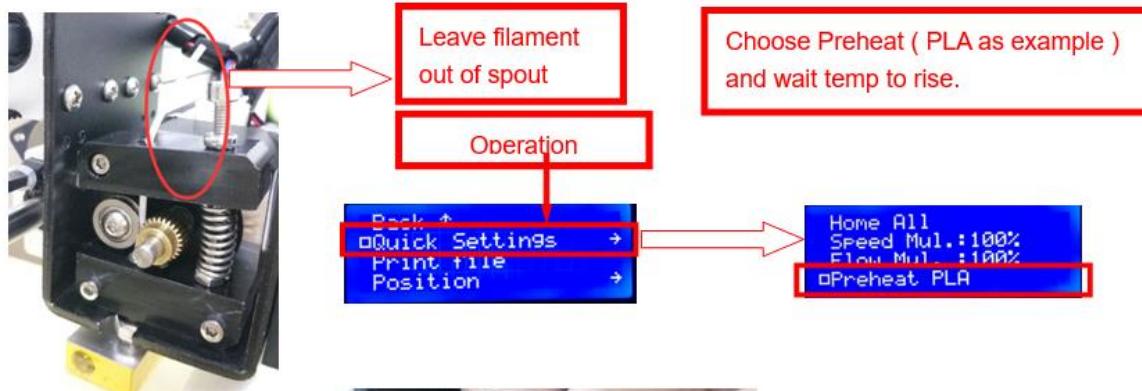


Fan

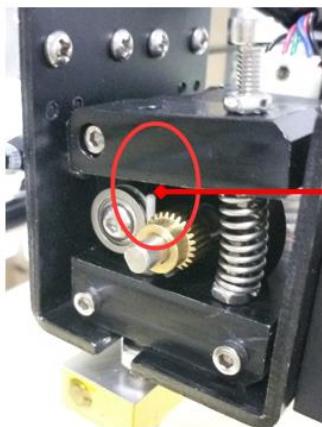


Fan removed

A. Only a little filament left in the nozzle and difficult to take out:



B. Filament full filled in spout



Leave filament in the spout



Allen wrench

Use the minimum allen wrench or some filament (1.75mm), press aligned with the remnant filament until filament comes out through nozzle.

3. FAQ

No.	Symptom	Reason	Method
1	Print model dislocation	Synchronous wheel/belt loose.	Tighten set screws or fasten belt
2	Glitch with the print model	Too high temp or slicing problem.	Extruder temp is too high and retracting speed & distance is too small
3	Foamy print model	Low temp or not smooth filament entering.	Raise extruder temp or check if brass nut and bearing is good. Replace a nozzle if methods above can't solve the problem.
4	Printer model is warped	Hotbed level isn't well adjusted.	Adjust hotbed
5	Unavailable G-code transformation	Wrong setting/wrong save path	Choose right machine type and change the right path
6	Software installation failed	Different OS	Reset OS
7	Unusual temp	Broken temp sensor	Change a new one

F. Maintenance

Important maintenance tips:

1. maintenance of X,Y,Z axis: Add some lubricants on the rods to reduce friction when the machine works noisy and a little bit shake.
2. Please refer to the USER MANUAL before printing, do preparation of hot bed adjustment first.
3. When finished printing, the filament should keep sealing, avoid moisture.
4. Preheat the extruder at the beginning of 2 nd time printing, let extruder auto-push filament for a while.
5. Machine should do some regular maintenance, drop some lubricating oil on thread rod, polished rod and bearings to avoid fatigue wear.
6. Do not let the fan and air-condition blow to the hot bed when printing.
7. Keep the working condition at “Temp:10-30°C, Humidity:20-70%”.

G. Maintenance policy

1. This product executes regulations of “Product Warranty Card”.
2. Please contact supplier or customer service if the product have any problems . Do not repair it by yourself, otherwise you need to bear all the consequences.