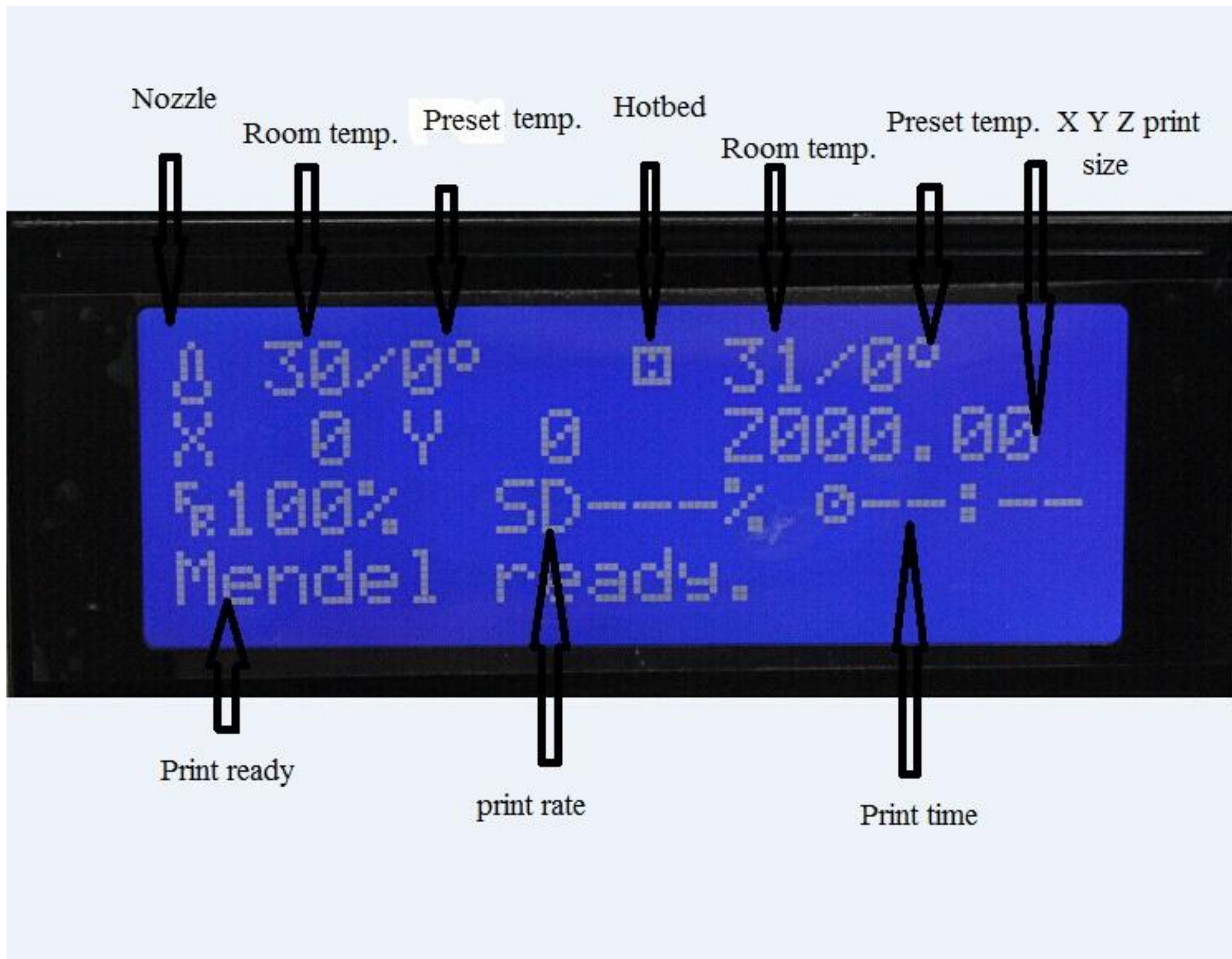


A-3 Content

1.Introduction of LCD display Interface.....	P2-3
2.Software installation and instructions.....	P4-37
3.Filament loading.....	P38-44
4.Hot bed adjustment.....	P45-49
5.Initial printing.....	P50-58
6.FAQ.....	P59-63





Cura14.07 User Manual

Where to find Cura 14.07

1. From our elaborate DVD packed.

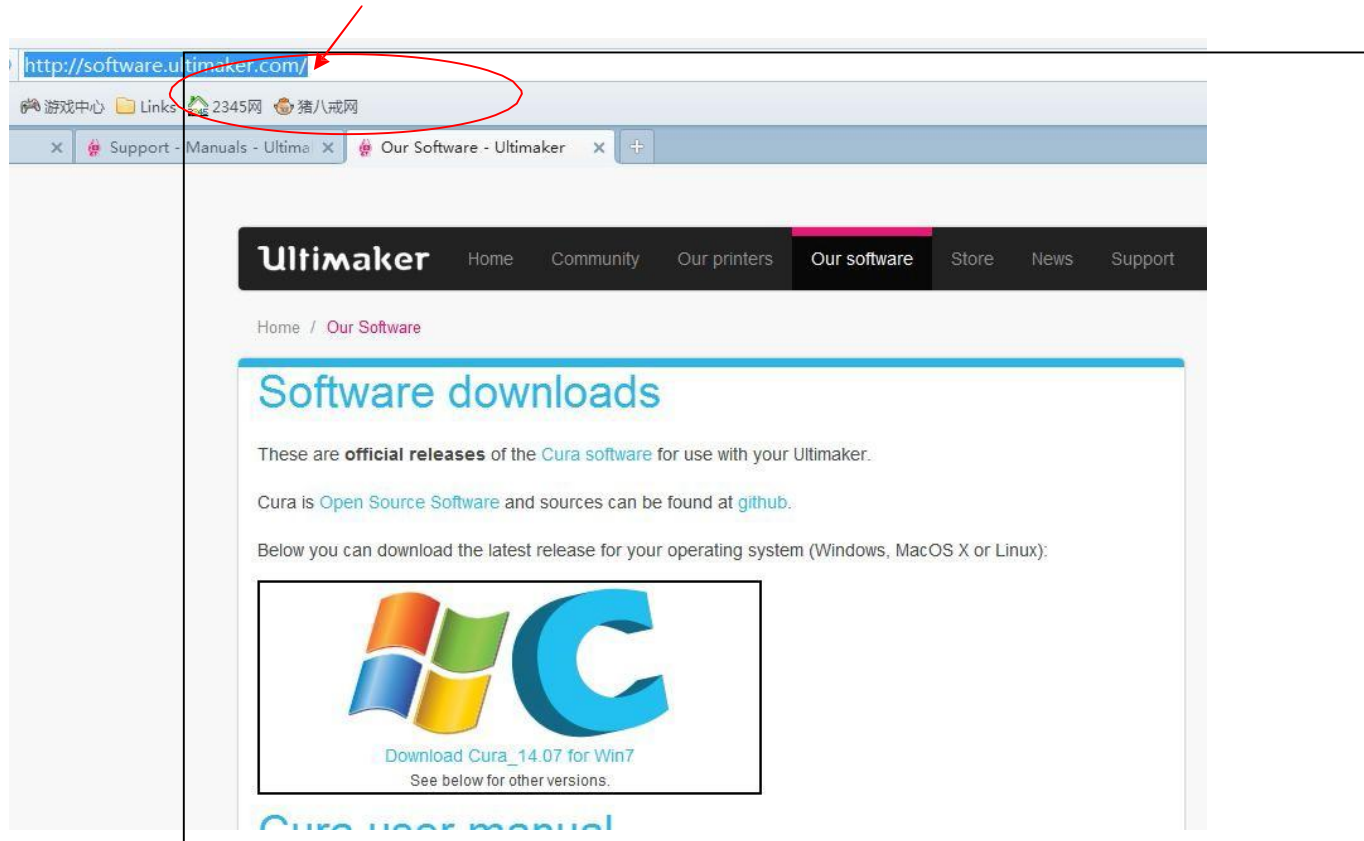
Assembly Instruction	2015/8/14 15:28	文件夹	
Software	2015/8/14 15:31	文件夹	
User Manual	2015/8/14 15:30	文件夹	
A3-Packing List 2015-7-16	2015/8/13 18:41	Microsoft Office...	17 KB

Cura_14.07	2014/9/4 10:49	应用程序	18,377 KB
xinpeizhi	2015/7/2 13:37	配置设置	11 KB

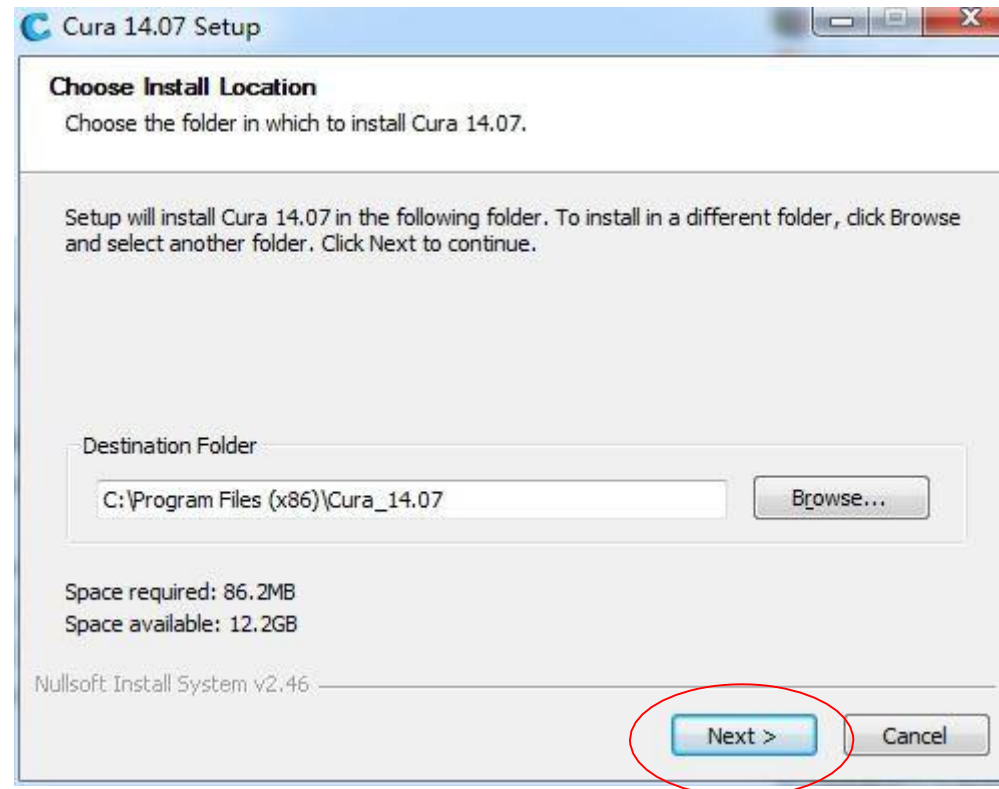
configuration files

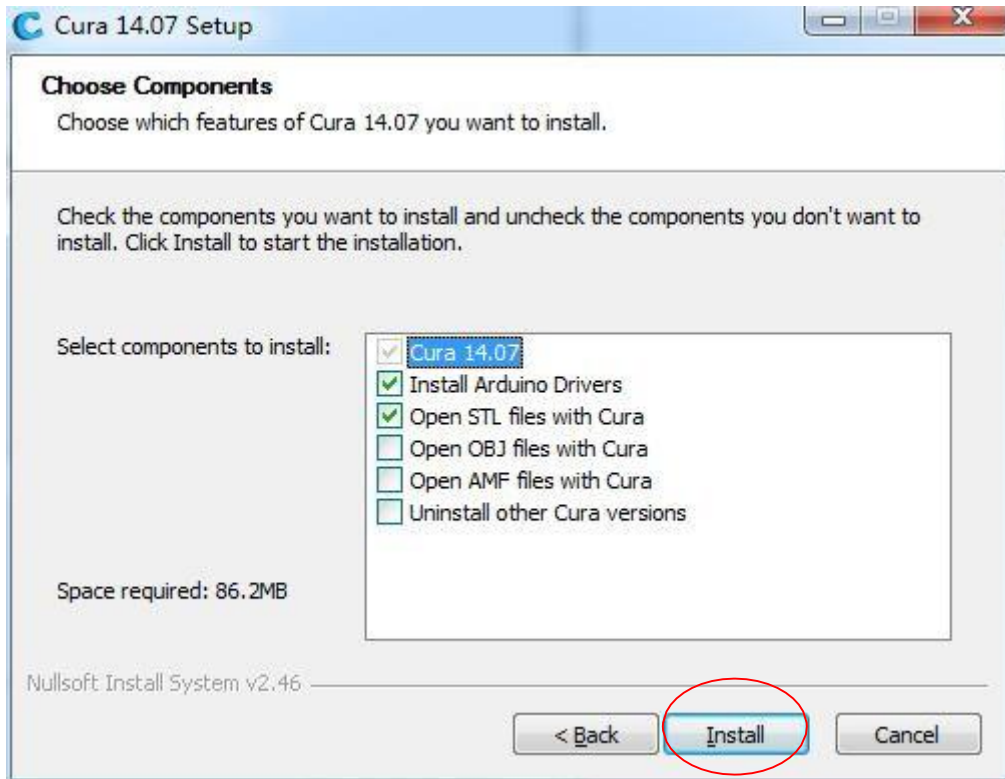
2.From the website of Ultimake.

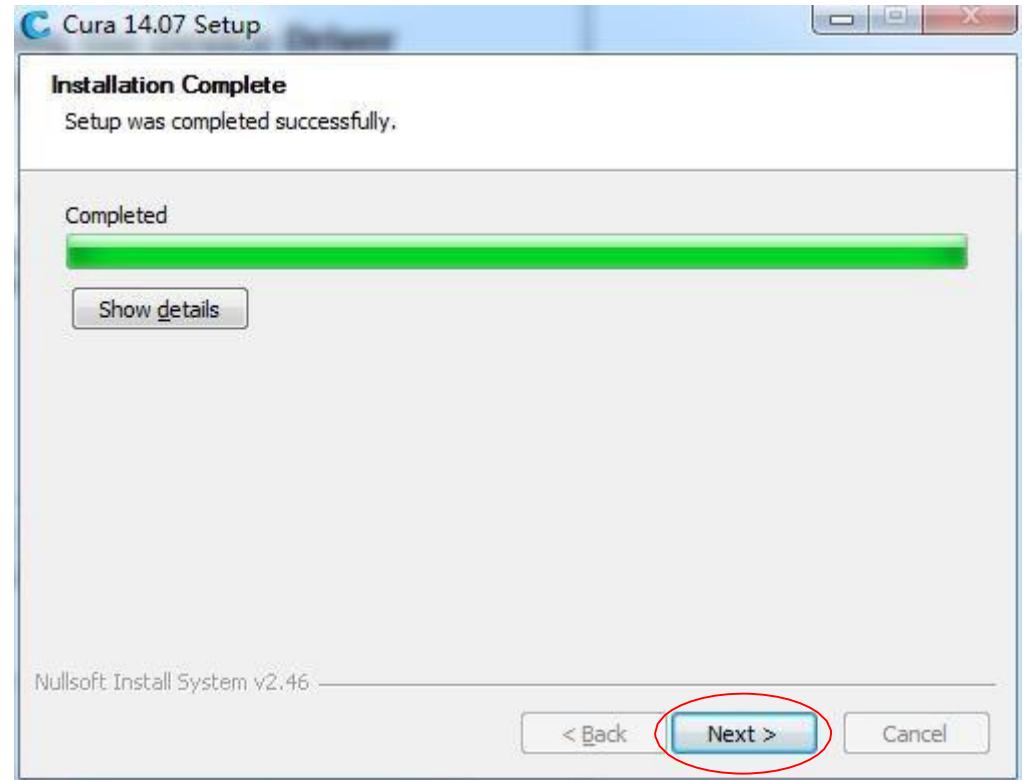
Here is the link: <http://software.ultimaker.com>



Steps of Software installation









Select
Other(Ex:ReRap,MakerBot)

Configuration Wizard

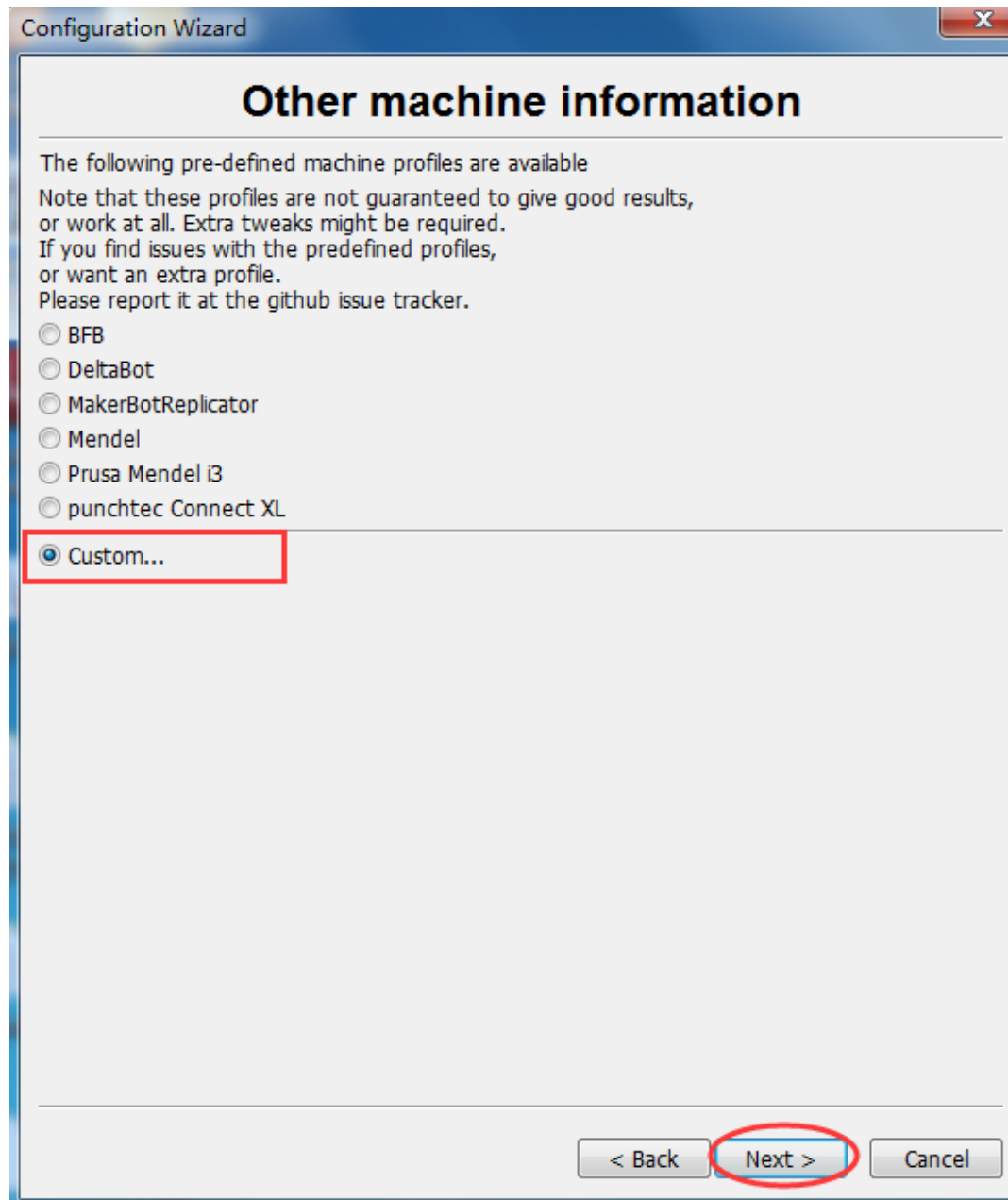
Select your machine

What kind of machine do you have:

- Ultimaker2
- Ultimaker Original
- Printrbot
- Other (Ex: RepRap, MakerBot)

The collection of anonymous usage information helps with the continued improvement of Cura.
This does NOT submit your models online nor gathers any privacy related information.
Submit anonymous usage information:
For full details see: <http://wiki.ultimaker.com/Cura:stats>

< Back **Next >** Cancel



Configuration Wizard

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

Machine width X (mm)

Machine depth Y (mm)

Machine height Z (mm)

Nozzle size (mm)

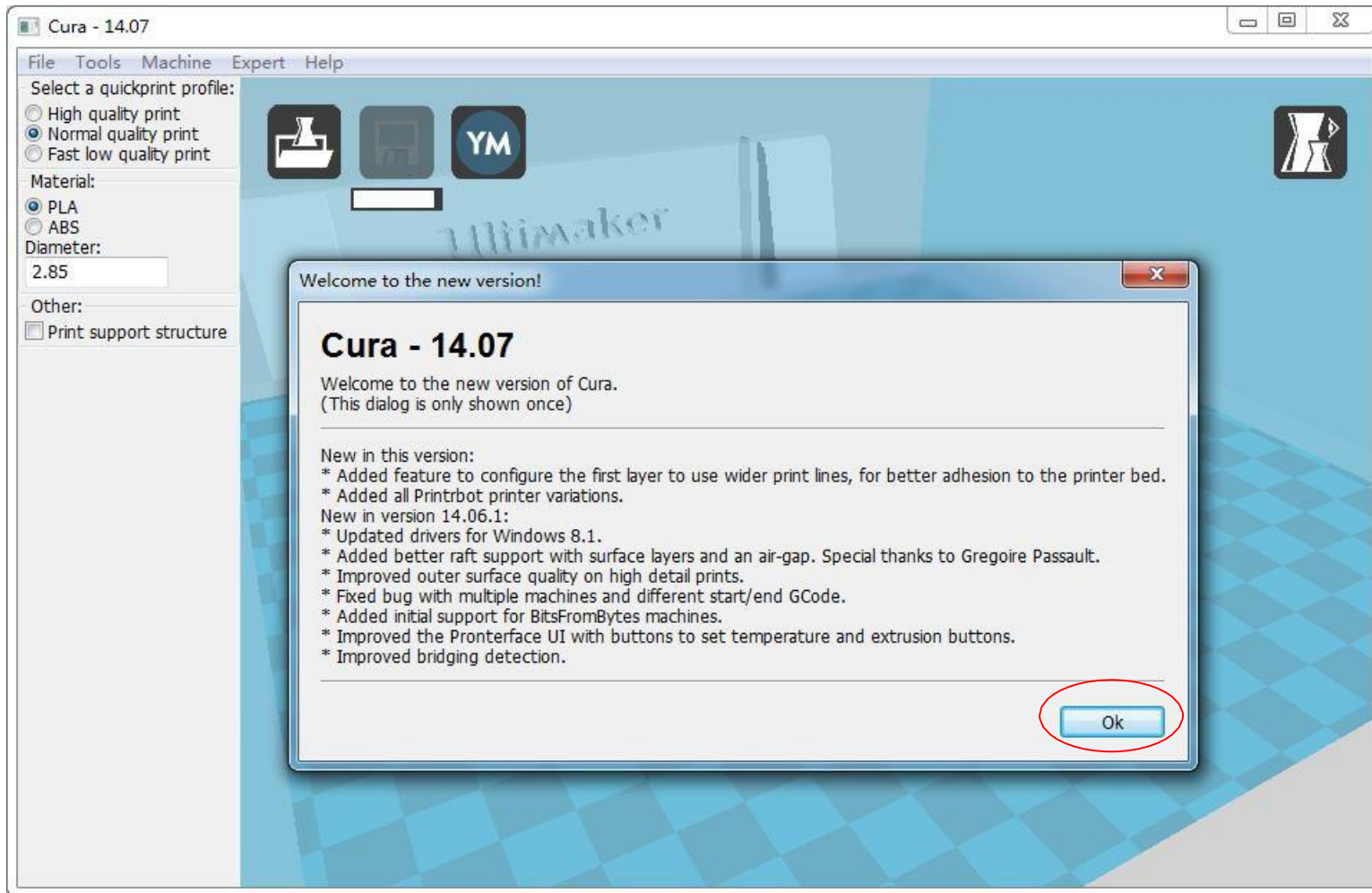
Heated bed

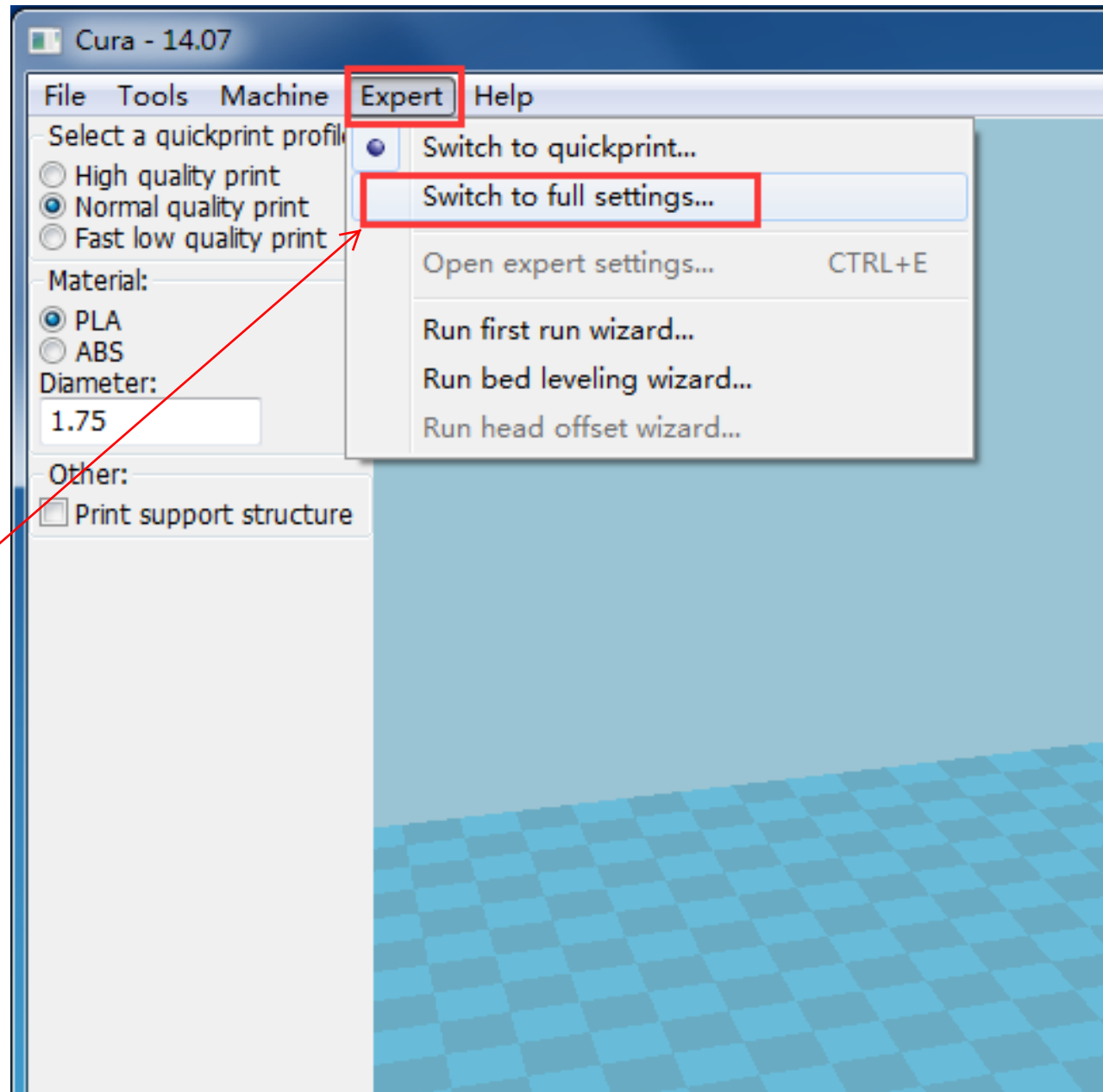
Bed center is 0,0,0 (RoStock)

< Back **Finish** Cancel

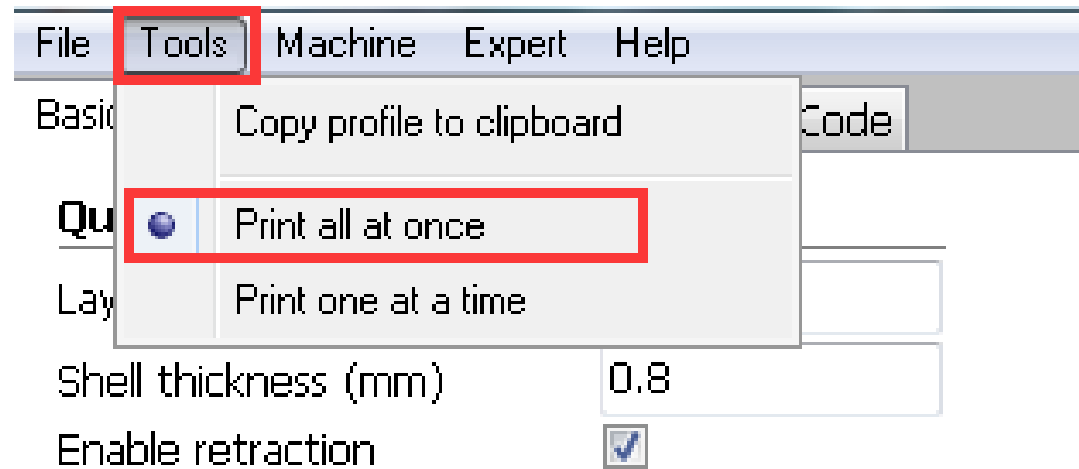
Type the size as pic show.

Click “ok”.



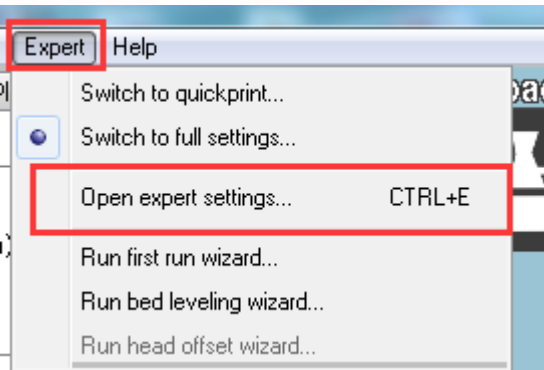


Switch to full settings for parameter setting.



Pls choosing “Print all at once” in case of failure while over one models printing.

Support type setting



Expert config

Retraction

- Minimum travel (mm) 1.5
- Enable combing
- 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) 5.0
- Fan speed min (%) 100
- Fan speed max (%) 100
- Minimum speed (mm/s) 10
- Cool head lift

Infill

- Solid infill top
- Solid infill bottom
- Infill overlap (%) 15

Support

- Structure type **Lines**
- Overhang angle for support (deg) 60
- Fill amount (%) 15
- Distance X/Y (mm) 0.7
- Distance Z (mm) 0.15

Black Magic

- Spiralize the outer contour
- Only follow mesh surface

Brim

- Brim line amount 20

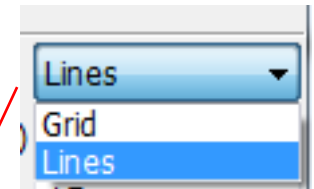
Raft

- Extra margin (mm) 5
- Line spacing (mm) 1.0
- Base thickness (mm) 0.3
- Base line width (mm) 0.7
- Interface thickness (mm) 0.2
- Interface line width (mm) 0.2
- Airgap 0.22
- Surface layers 2

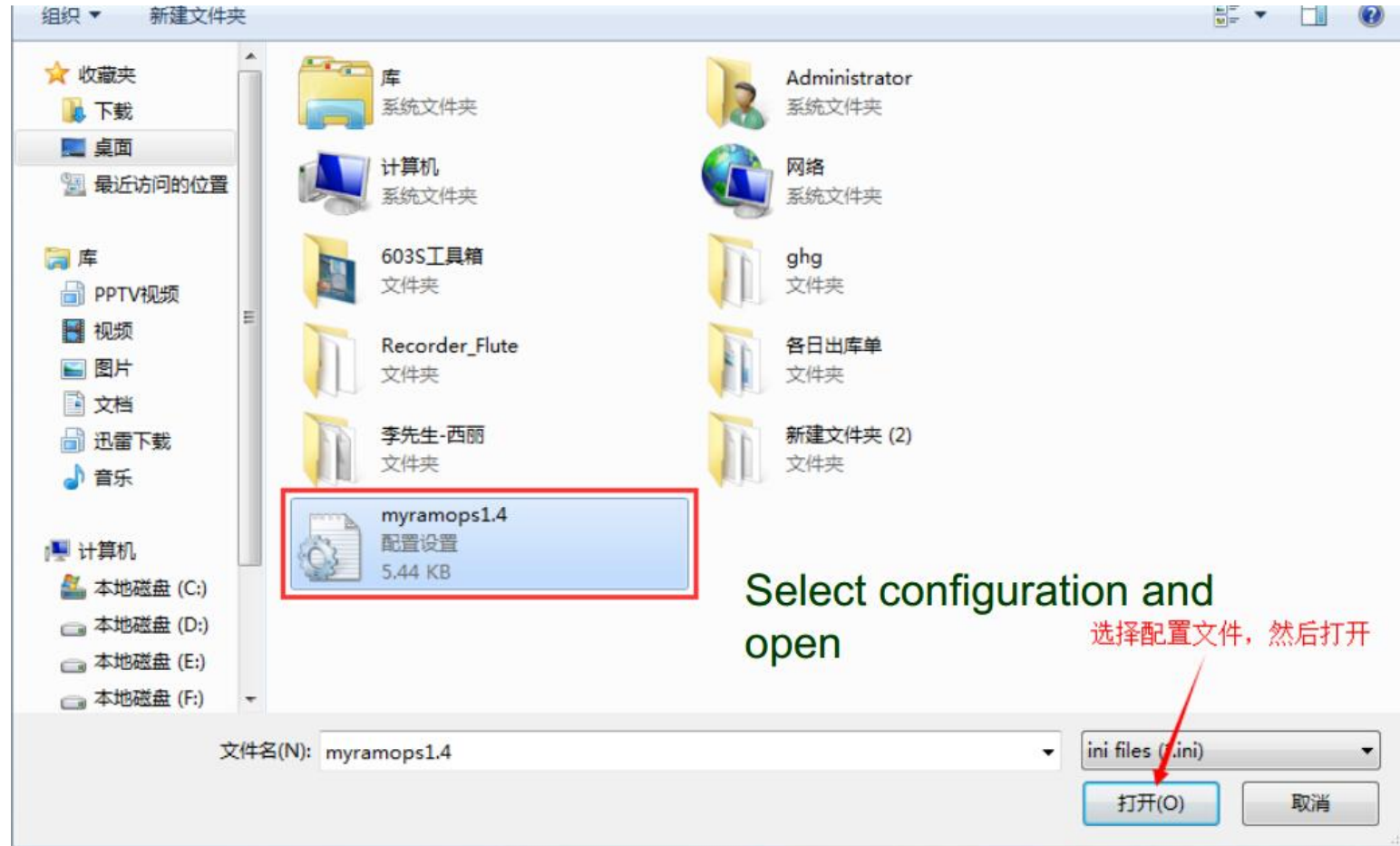
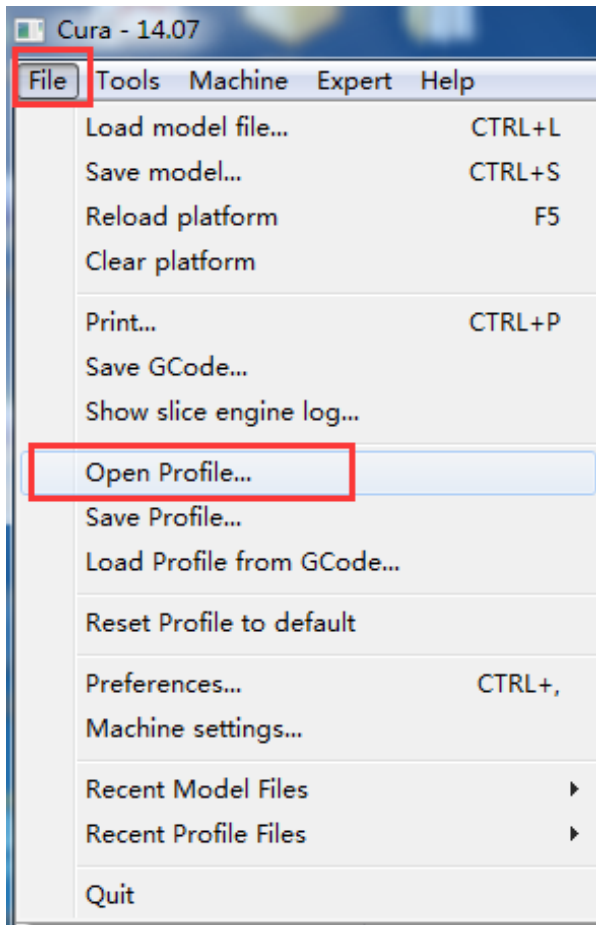
Fix horrible

- Combine everything (Type-A)
- Combine everything (Type-B)
- Keep open faces
- Extensive stitching

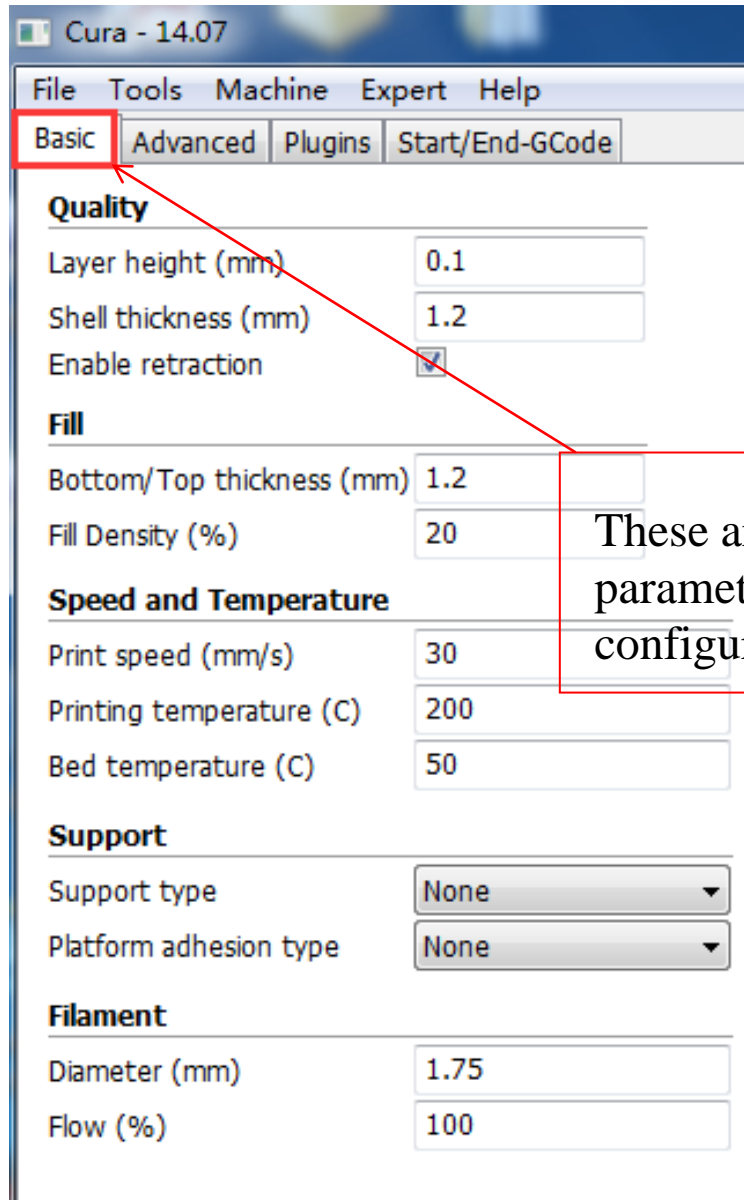
Ok



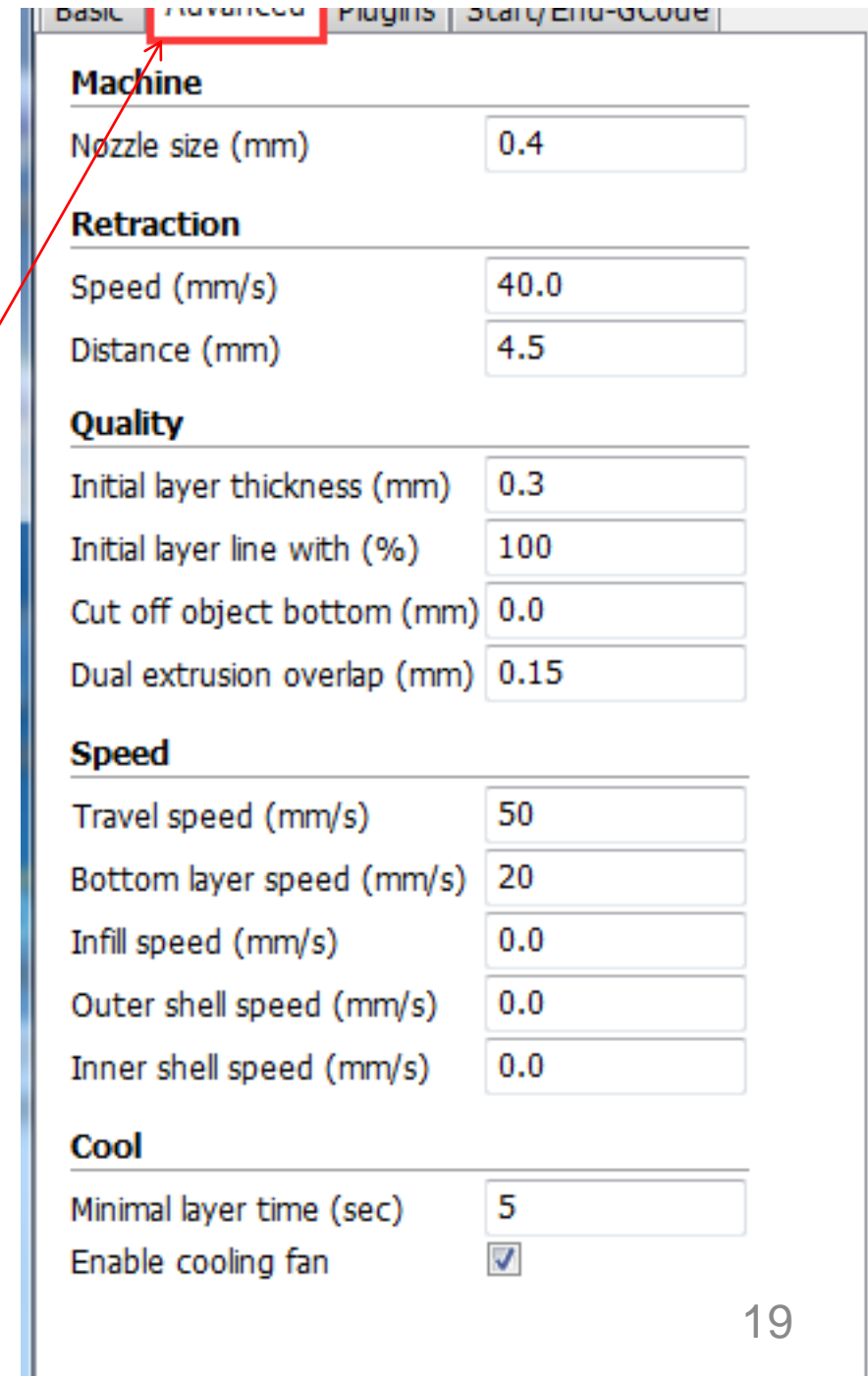
Load Configuration file.



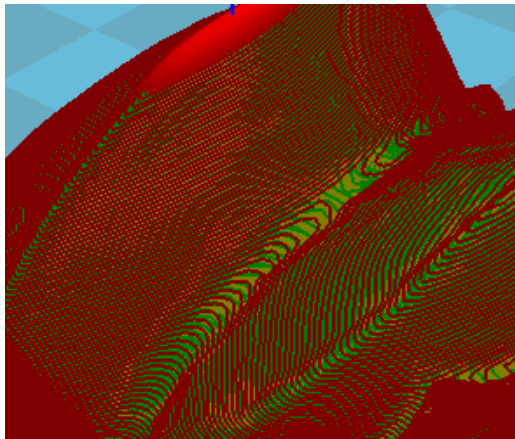
After configuration file loading



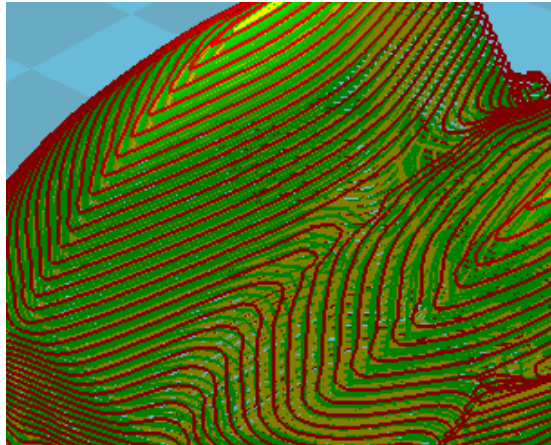
These are the standard print parameters after loading configuration file.



When set layer height at 0.1mm, print time is longer and higher print precision. Print time will be shorten half when setted at 0.2mm comparing 0.1mm lay height, and prcesion is also good. The shortest time but not good precision is at 0.3mm.



left:0.1mm
layer height

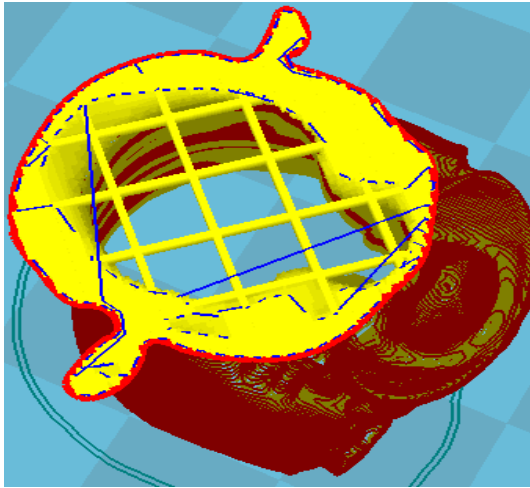


Right:0.3mm
layer height

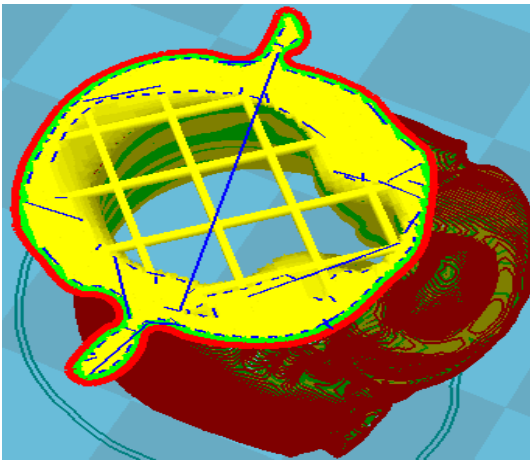
File Tools Machine Expert Help	
Basic Advanced Plugins Start/End-GCode	
Quality	
Layer height (mm)	0.1
Shell thickness (mm)	0.8
Enable retraction	<input checked="" type="checkbox"/>
Fill	
Bottom/Top thickness (mm)	0.6
Fill Density (%)	20
Speed and Temperature	
Print speed (mm/s)	50
Printing temperature (C)	220
Support	
Support type	None
Platform adhesion type	None
Filament	
Diameter (mm)	2.85
Flow (%)	100.0

Shell thickness

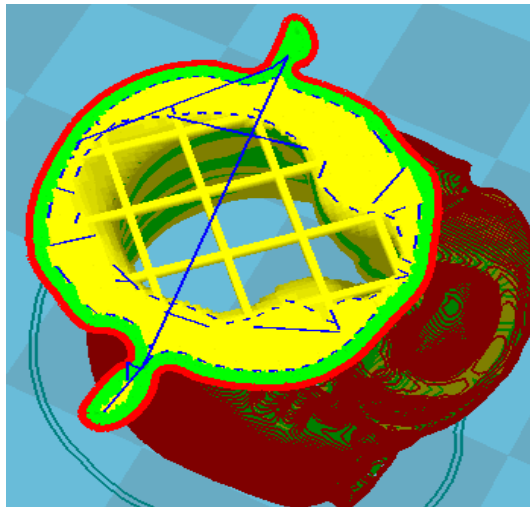
0.4



0.8



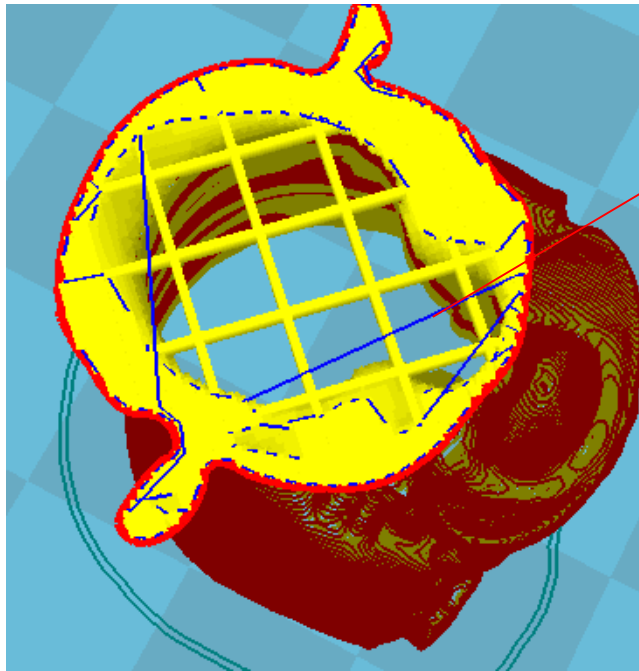
1.2



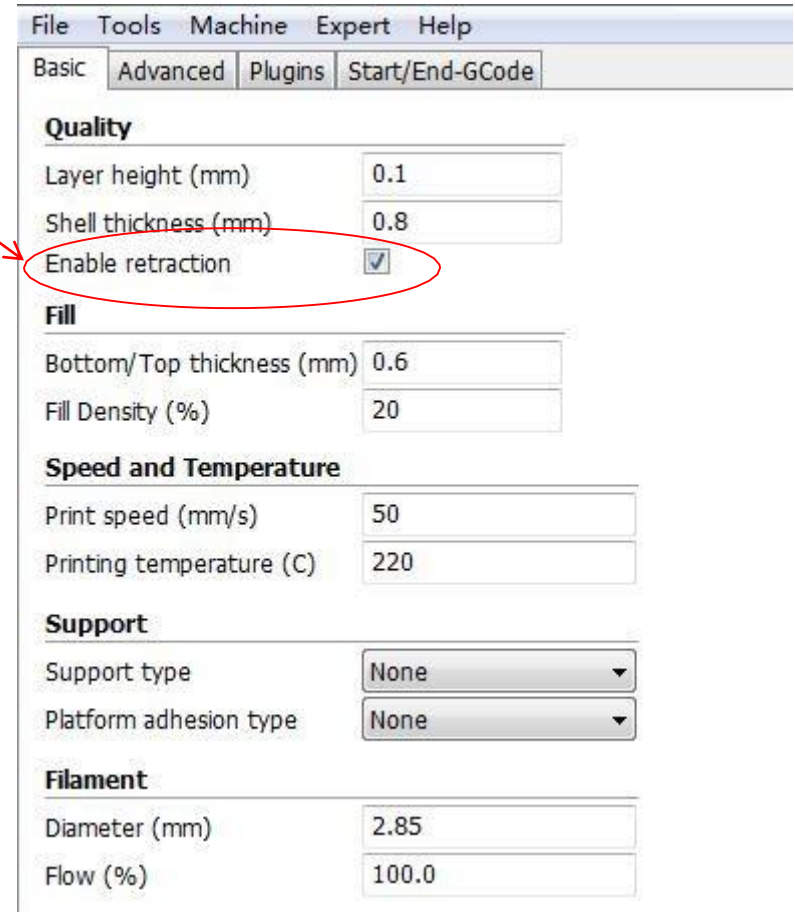
File Tools Machine Expert Help	
Basic Advanced Plugins Start/End-GCode	
Quality	
Layer height (mm)	0.1
Shell thickness (mm)	0.8
Enable retraction	<input checked="" type="checkbox"/>
Fill	
Bottom/Top thickness (mm)	0.6
Fill Density (%)	20
Speed and Temperature	
Print speed (mm/s)	50
Printing temperature (C)	220
Support	
Support type	None
Platform adhesion type	None
Filament	
Diameter (mm)	2.85
Flow (%)	100.0

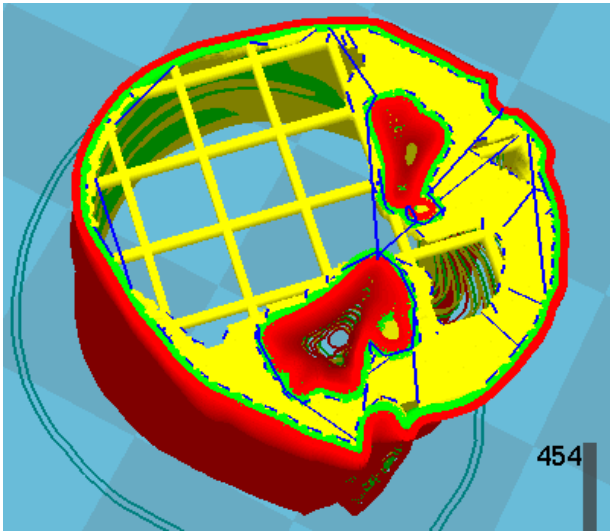
1. Shell will be very thin when set shell thickness at 0.4mm.
2. Print time will be longer when shell thickness at 1.2mm.
3. Normally, it is best at 0.8mm.
4. Shell thickness should be the integral multiple of the nozzle diameter.

Enable retraction function: not to let the fused filament leak off under force of gravity during printing moving. Otherwise, the appearance of printed sample will be effected.

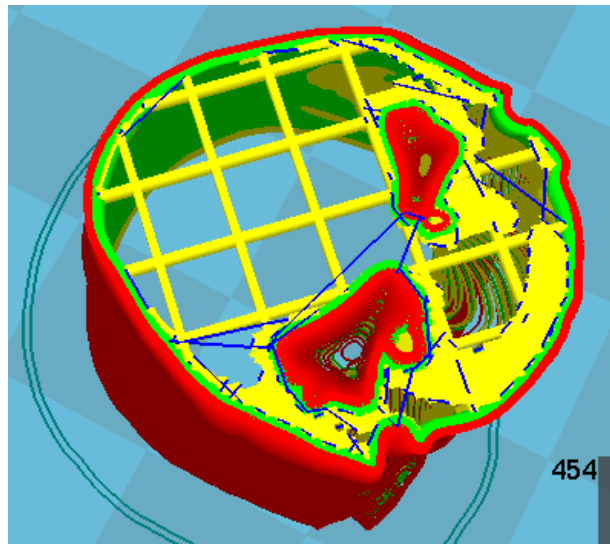


Blue stand
for the
thread
leaked out.





Bottom/Top thickness: 1.2mm



Bottom/Top thickness: 0.6mm

When the fill density less 20% , there will be a hole at the top when thickness at 0.6mm. Normally, 1.2mm is better.

Same filling

Quality

Layer height (mm) 0.1

Shell thickness (mm) 0.8

Enable retraction

Fill

Bottom/Top thickness (mm) 0.6

Fill Density (%) 10

Speed and Temperature

Print speed (mm/s) 50

Printing temperature (C) 220

Support

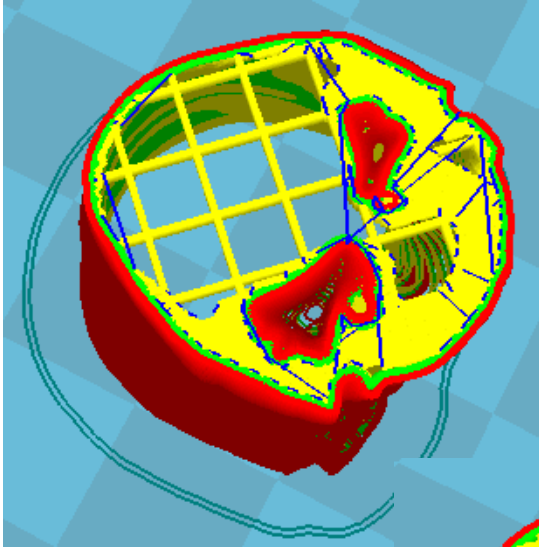
Support type None

Platform adhesion type None

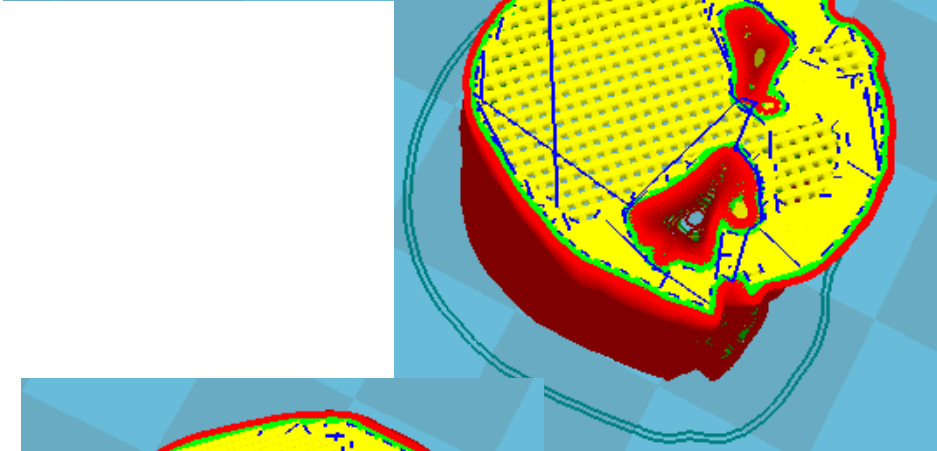
Filament

Diameter (mm) 2.85

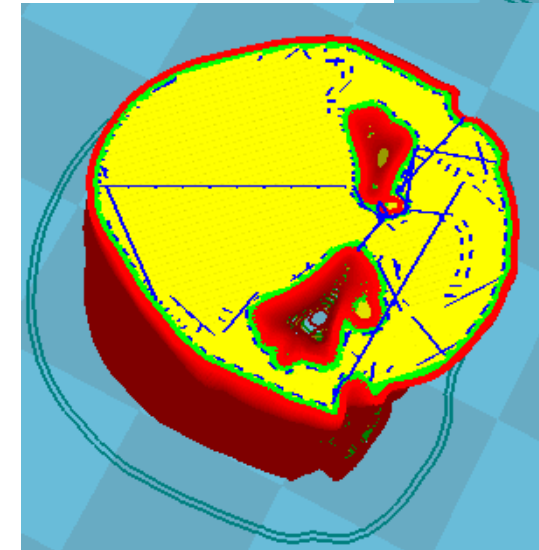
Flow (%) 100.0



20%



50%



90%

Fill density: 10% is OK if no strength requirement. Increase the fill density if high strength requirement, but printing time increased accordingly, and will print accumulation in slop.

Quality	
Layer height (mm)	0.1
Shell thickness (mm)	0.8
Enable retraction	<input checked="" type="checkbox"/>
Fill	
Bottom/Top thickness (mm)	0.6
Fill Density (%)	10
Speed and Temperature	
Print speed (mm/s)	50
Printing temperature (C)	220
Support	
Support type	None
Platform adhesion type	None
Filament	
Diameter (mm)	2.85
Flow (%)	100.0

This speed is initial configuration file loaded. You could alter the speed you want to print. Speed is inversely proportional to accuracy. Normally, we suggest at 40-60mm/s.

PLA:

Nozzle: 190-220

Hot bed: 50-80

ABS:

Nozzle: 245-255

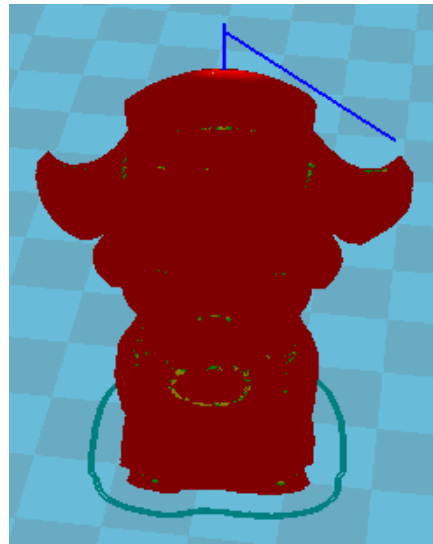
Hot bed: 90-100

Speed faster, layer height more, temperature should be set higher. On the contrary, temperature could be set lower.

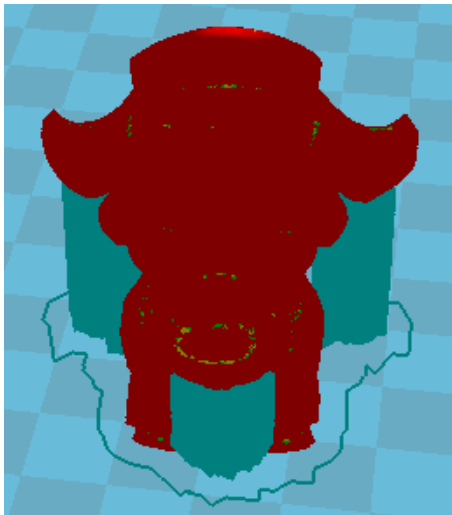
Quality	
Layer height (mm)	0.1
Shell thickness (mm)	0.8
Enable retraction	<input checked="" type="checkbox"/>
Fill	
Bottom/Top thickness (mm)	1.2
Fill Density (%)	50
Speed and Temperature	
Print speed (mm/s)	30
Printing temperature (C)	200
Bed temperature (C)	50
Support	
Support type	None
Platform adhesion type	None
Filament	
Diameter (mm)	1.75
Flow (%)	100



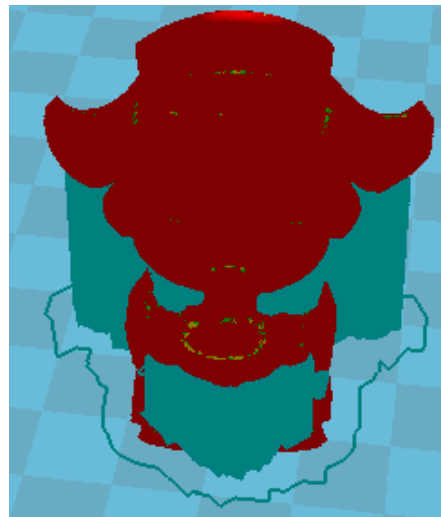
Original



None

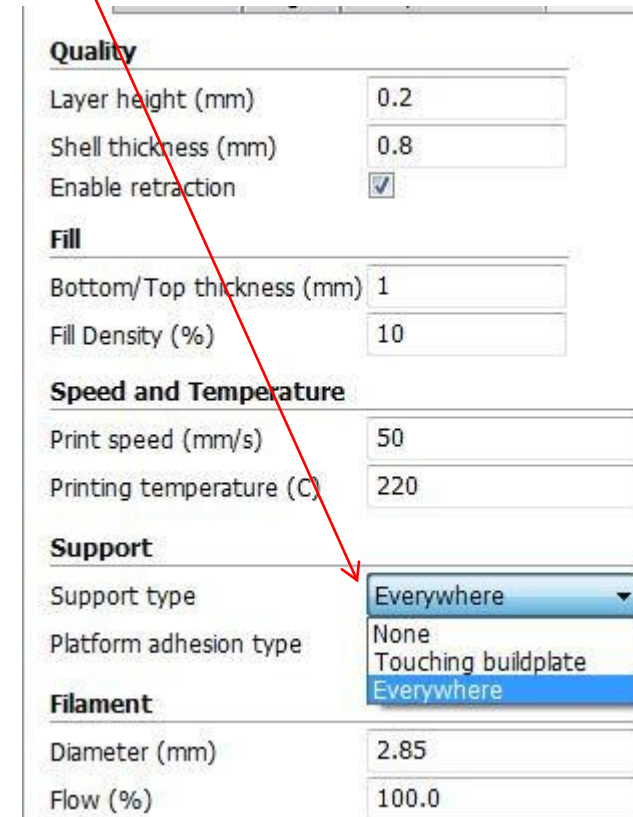


Touching buildplate



Everywhere

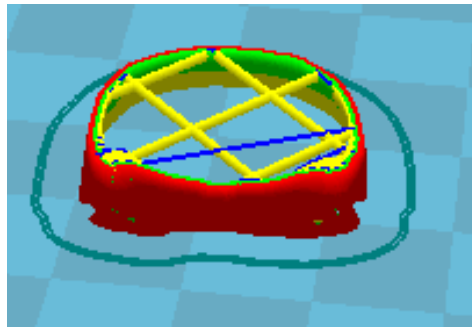
Support type: None, Touching buildplate and Everywhere.



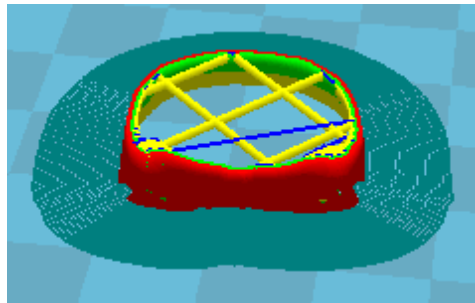
Normally support is needed for complicated structure and hanging model, Everywhere type may touch the model and not good looking appearance, To avoid support, model need to be rotated to a right position.

Support setting

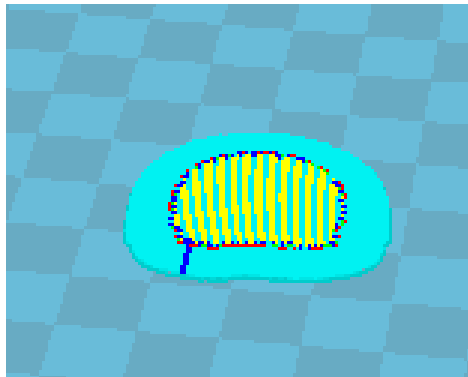
Platform adhesion setting



None

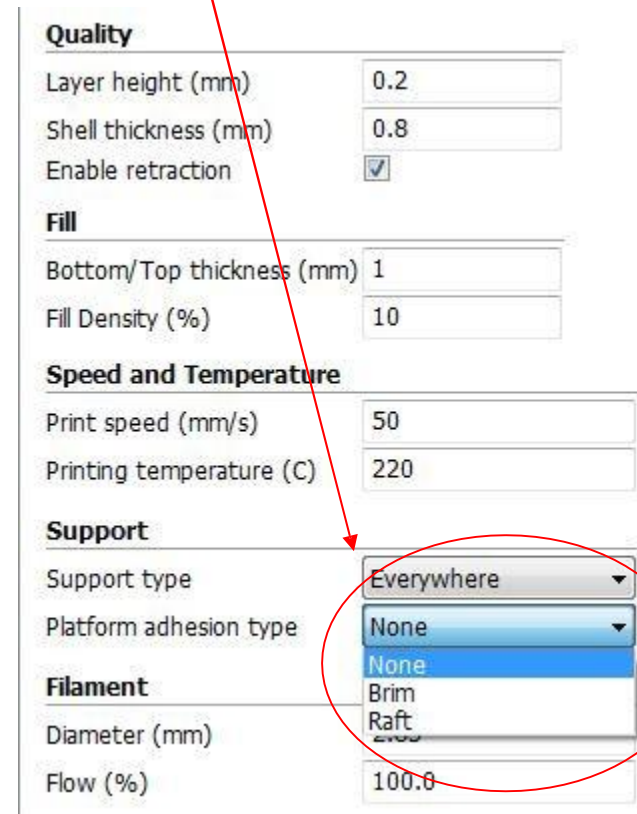


Brim



Raft

Platform adhesion type: None, Brim and Raft



For normal models, select “None” is OK if hot bed adjustment done and good blue masking tap. Or, select Raft, but hard to separate raft from model.

Filament setting

2.85mm is auto for Cura system, 1.75mm is our machine standard.

Flow rate, increasing the flow rate is as same as decreasing the filament diameter. Increasing too much, there will be salient point in model appearance, decreasing too much, model will be sparse cause of few filament flow.

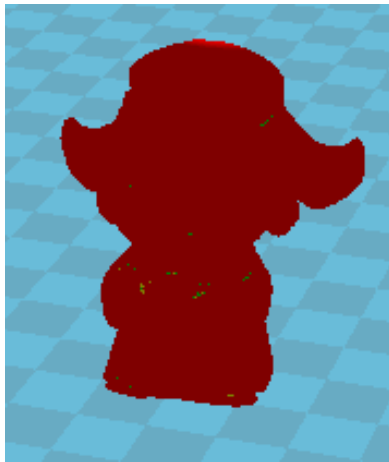
Quality	
Layer height (mm)	0.2
Shell thickness (mm)	0.8
Enable retraction	<input checked="" type="checkbox"/>
Fill	
Bottom/Top thickness (mm)	1
Fill Density (%)	10
Speed and Temperature	
Print speed (mm/s)	50
Printing temperature (C)	220
Support	
Support type	Everywhere
Platform adhesion type	Raft
Filament	
Diameter (mm)	2.85
Flow (%)	100.0

Other specifications setting

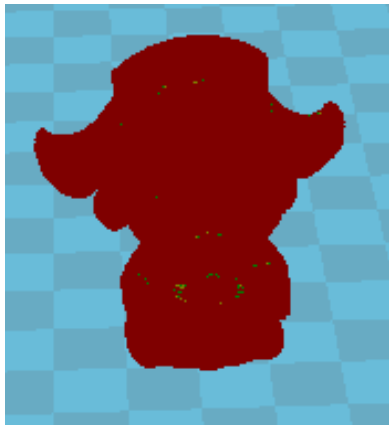
Fixed value don't need to change after configuration
File loading.

Basic	Advanced	Plugins	Start/End-GCode
Machine			
Nozzle size (mm)		0.4	
Retraction			
Speed (mm/s)		40.0	
Distance (mm)		4	
Quality			
Initial layer thickness (mm)		0.3	
Initial layer line with (%)		100	
Cut off object bottom (mm)		0.0	
Dual extrusion overlap (mm)		0.15	
Speed			
Travel speed (mm/s)		150.0	
Bottom layer speed (mm/s)		20	
Infill speed (mm/s)		0.0	
Outer shell speed (mm/s)		0.0	
Inner shell speed (mm/s)		0.0	
Cool			
Minimal layer time (sec)		5	
Enable cooling fan		<input checked="" type="checkbox"/>	

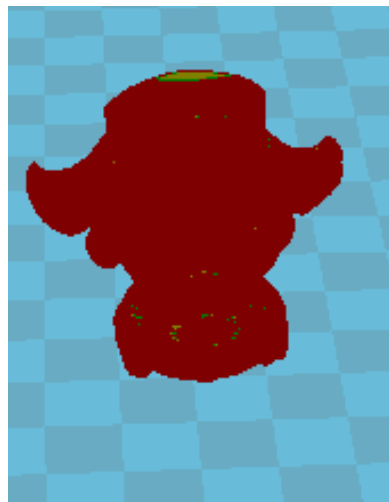
Cooling fan



Cut off: 0mm



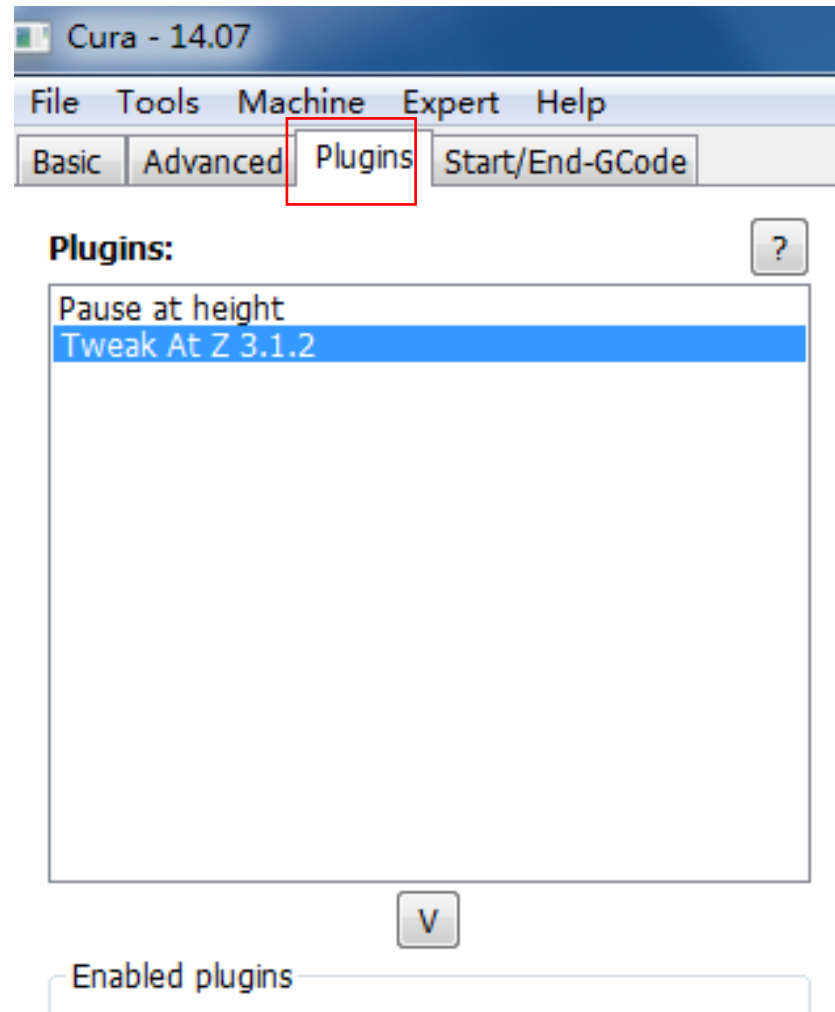
Cut off: 3mm



Cut off: 8mm

Cut off object bottom setting.

Machine	
Nozzle size (mm)	0.4
Retraction	
Speed (mm/s)	40.0
Distance (mm)	4
Quality	
Initial layer thickness (mm)	0.3
Initial layer line width (%)	100
Cut off object bottom (mm)	0
Dual extrusion overlap (mm)	0.15
Speed	
Travel speed (mm/s)	150.0
Bottom layer speed (mm/s)	20
Infill speed (mm/s)	0.0
Outer shell speed (mm/s)	0.0
Inner shell speed (mm/s)	0.0
Cool	
Minimal layer time (sec)	5
Enable cooling fan	<input checked="" type="checkbox"/>



Different firmware have its own customized order, normally, don't change for these two items.

Loading model

Save gcode to
SD card

Connect to "youmagine" web



Model display
type

Left key: model moving

Middle key: scale view

Right key: model rotary

Shift+Right key: translation

view

Select model by left key,
as image show.

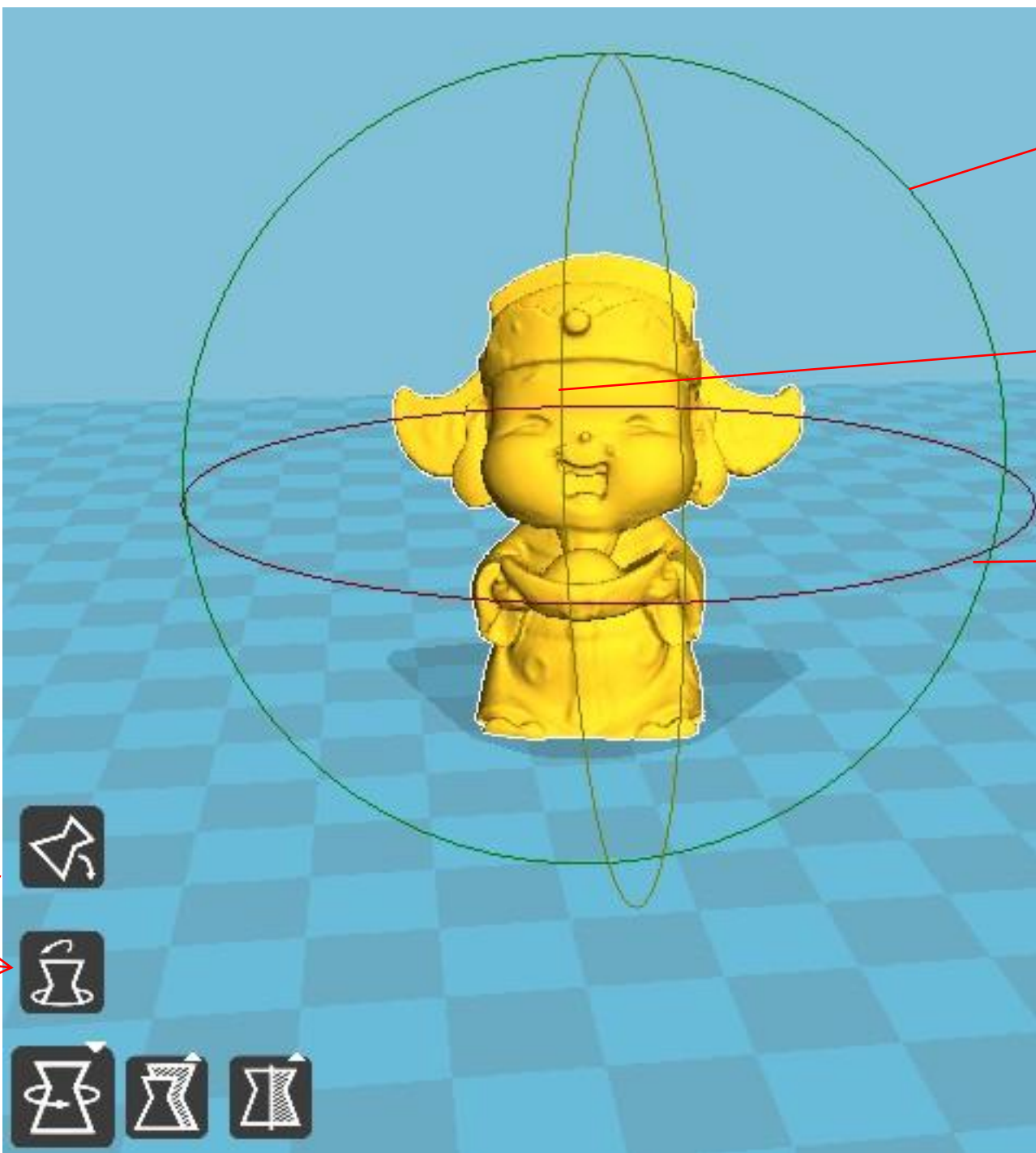
Rotate

scale

Mirror

Lay flat

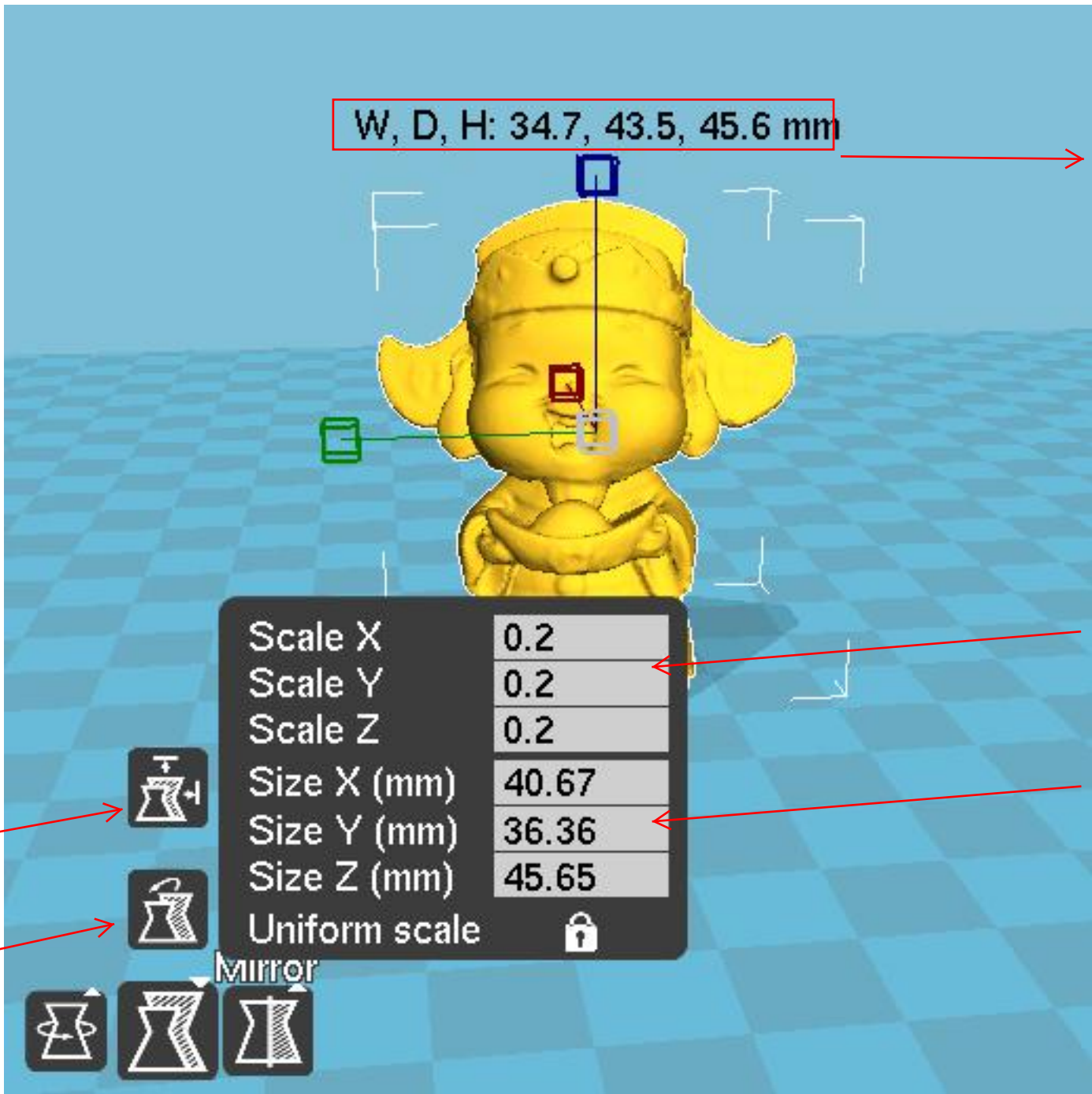
Reset



Green: X axis

Yellow: Y axis

Red: Z axis



W, D, H: 34.7, 43.5, 45.6 mm

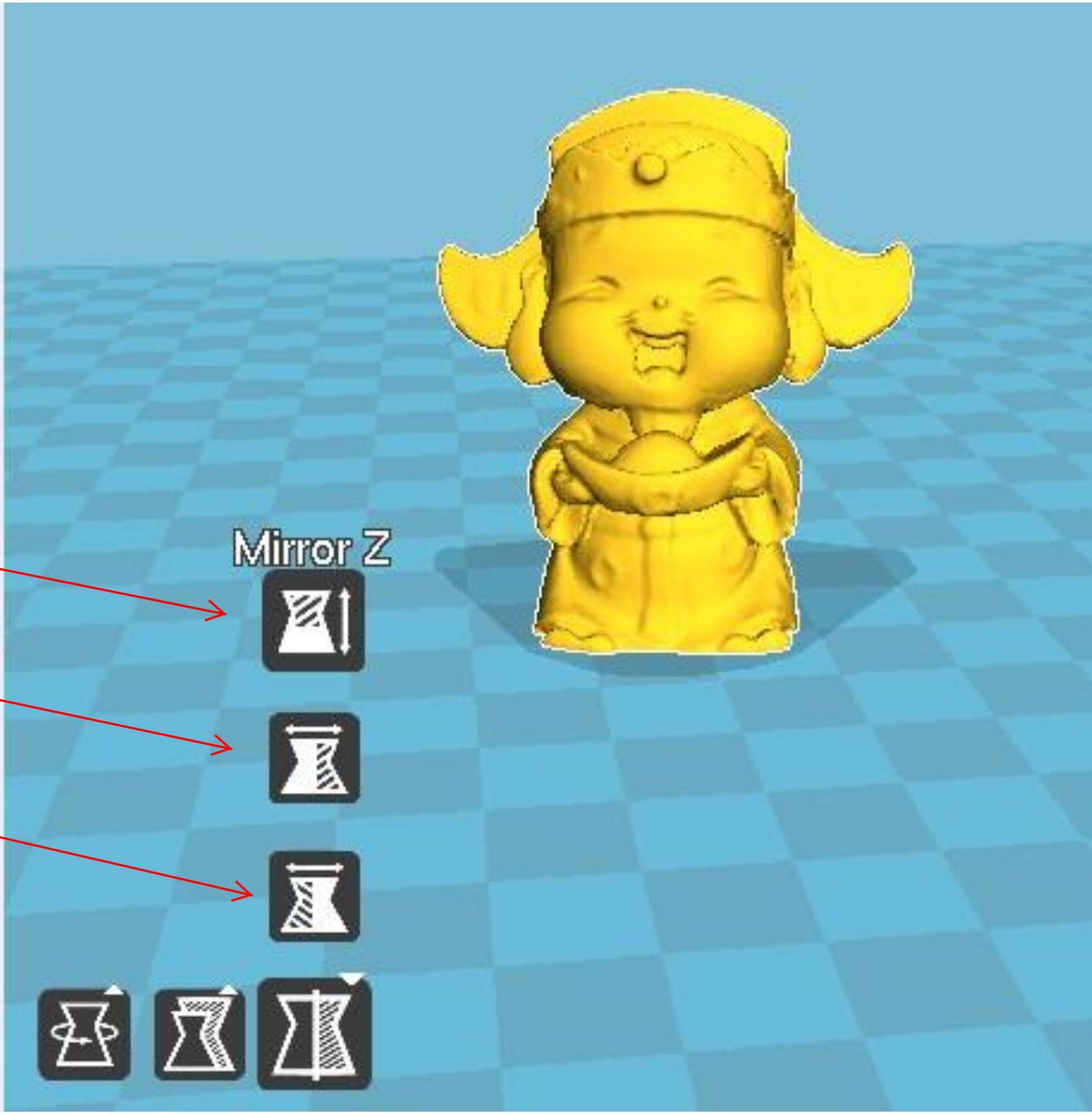
W,D,H size

Scale rate

Modify the data if you want.

Max size

Reset



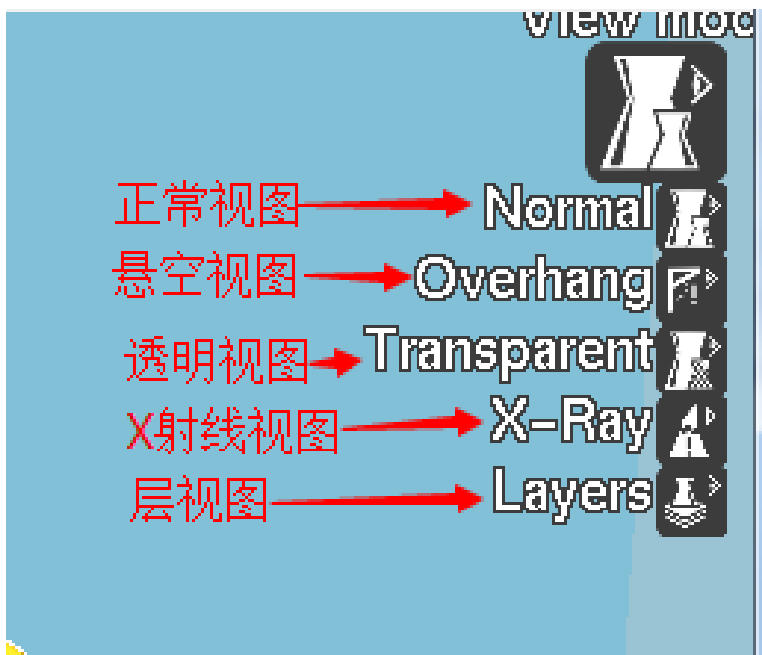
Mirror Z

Mirror Z

Mirror Y

Mirror X

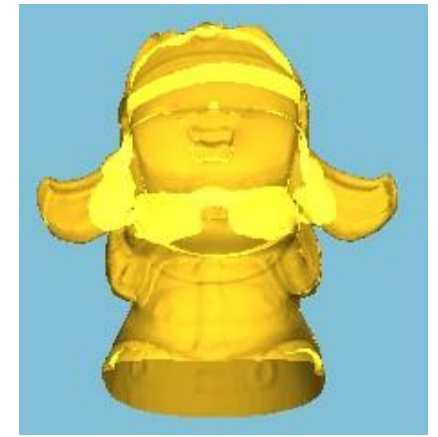
Mirror image is opposite of actual printing model.



A



B



C

A. Normal

B. Overhand: to view overhand position (Red)

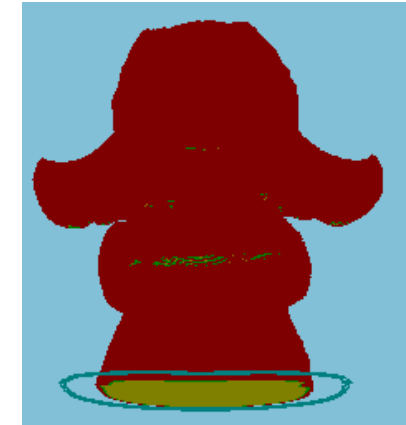
C. Transparent

D. X-Ray

E. Layers:to see the printing process

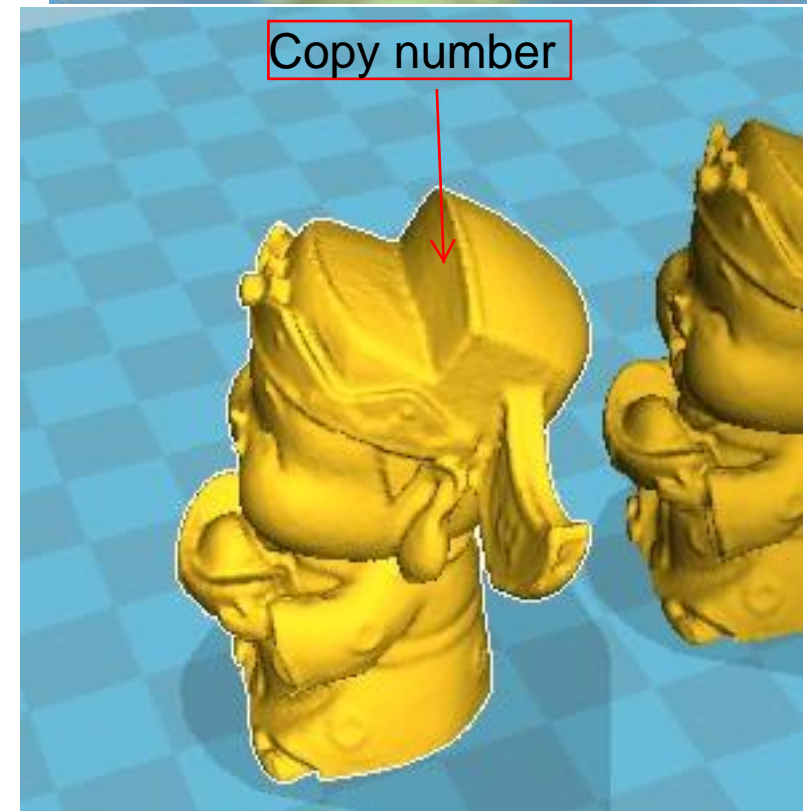


D



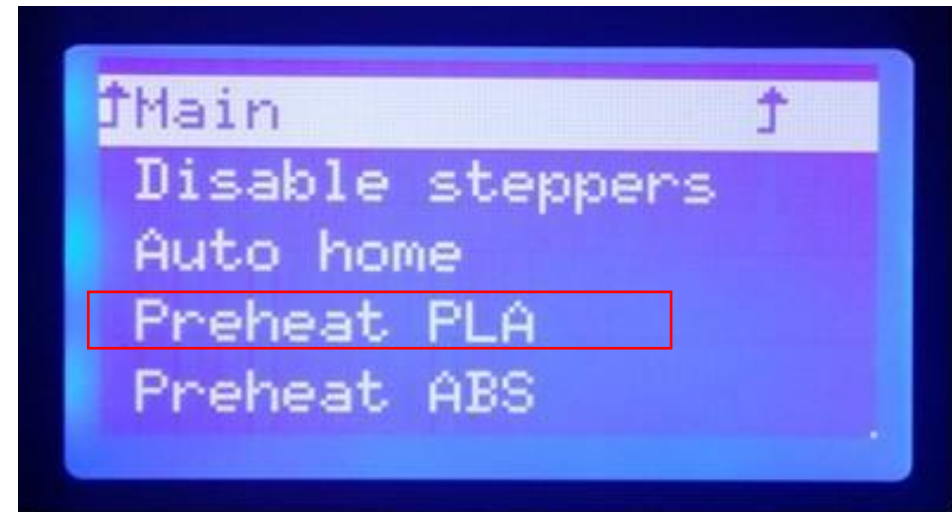
E

Select model and right click



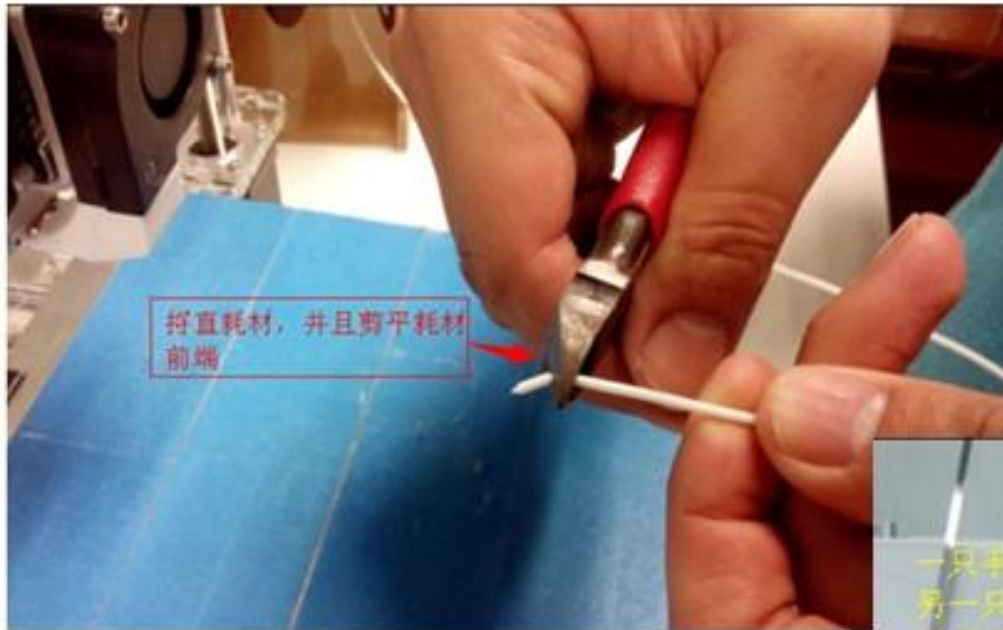
Filament loading guide

Press button--Prepare--Preheat PLA



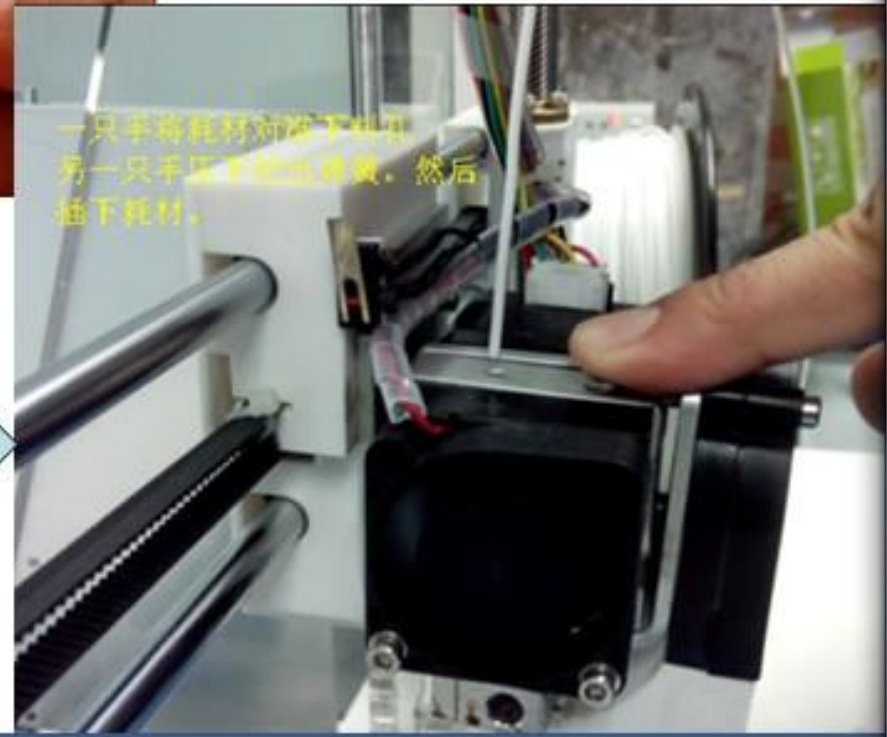
After the extruder temp. reach 180 °C. Operate as below.

Cutting a small bevel angle at the beginning of filament.



捋直耗材，并且剪平耗材前端

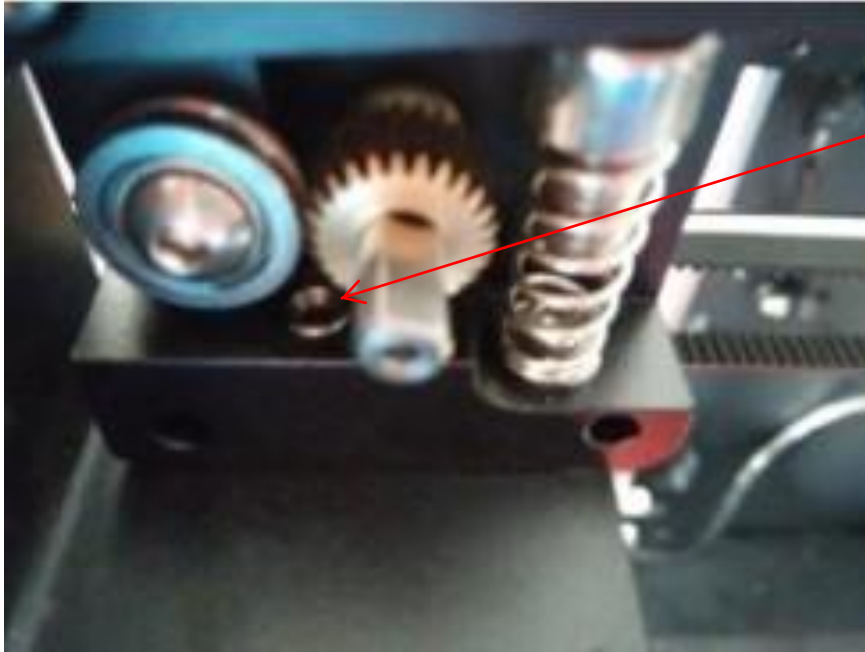
← 捋直耗材，并且剪平耗材前端。如图所示：



一只手将耗材对准下料孔，另一只手压下挤出弹簧，然后插下耗材。

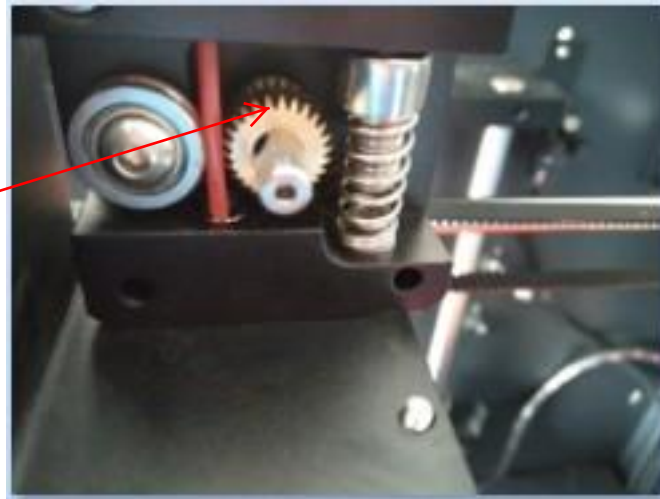
一只手将耗材对准下料孔，另一只手压下挤出弹簧，然后插下耗材。如图所示：

Insert the filament by a hand, press the lock(nut) down by another hand meanwhile.

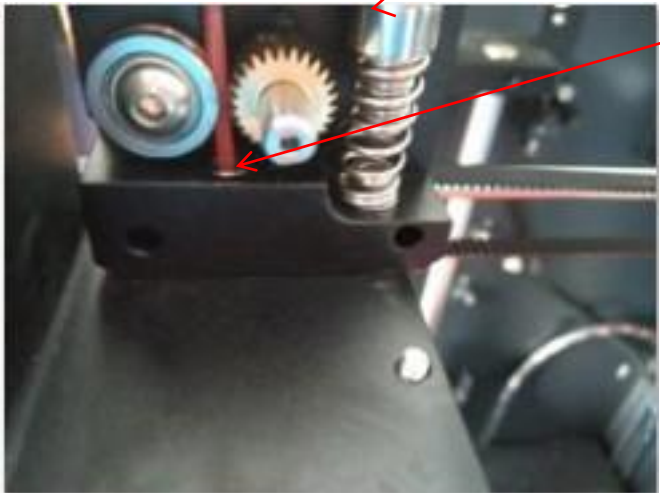


Inner feeding mouth

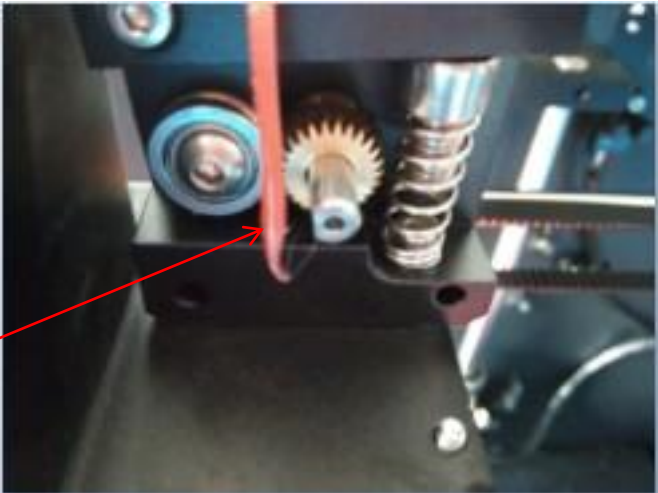
Filament not feed into the right feeding mouth.

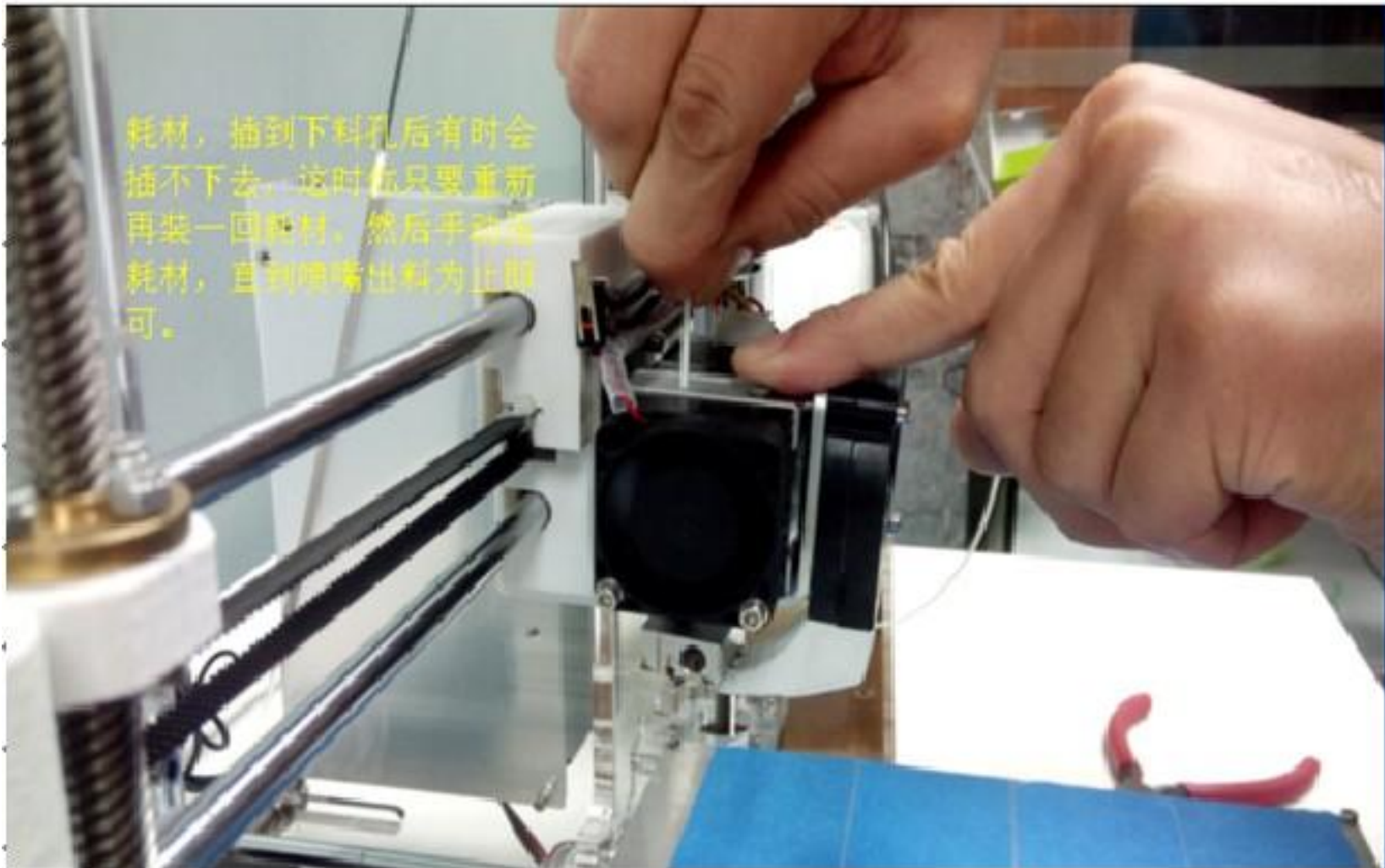


Filament at the right feed mouth



Filament after dissolution





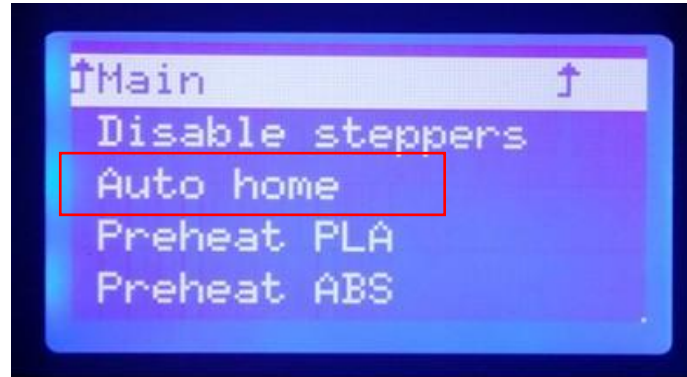
After filament loaded, push filament by hand till homogeneous thread flowing out from nozzle.

Tips

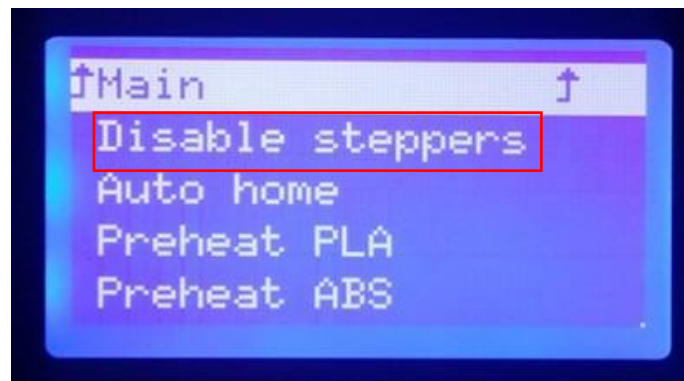
- 1.To avoid filament replacement problem, please dont feed the all filament to the feeding mouth when one roll of filament is going to finished .New filament should be changed in time.
- 2.When new filament replacement, pls preheat the printer first, and then press down tight spring by hand, and feeding the filament for a moment and then pull out quickly .Remember that not to pull out hard or cold to avoid nozzle irreparable damage.
- 3.Maintenance: After the machine works for a period of time, please add drops of lubricant to the polished rod of some moving parts, to reduce the abrasion of machine.

Hot bed adjustment

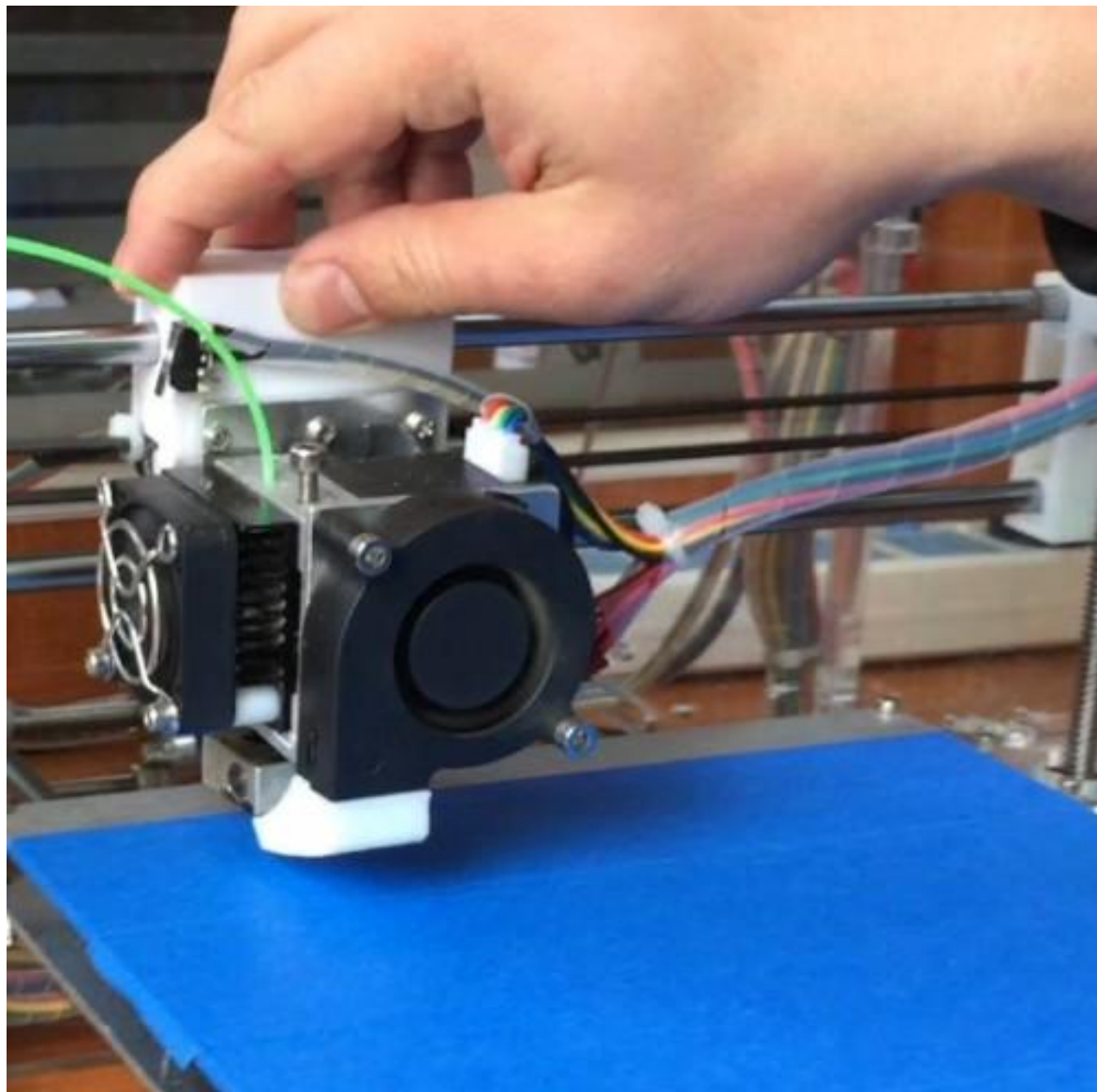
1. Press the button - “Prepare” - “Auto home”



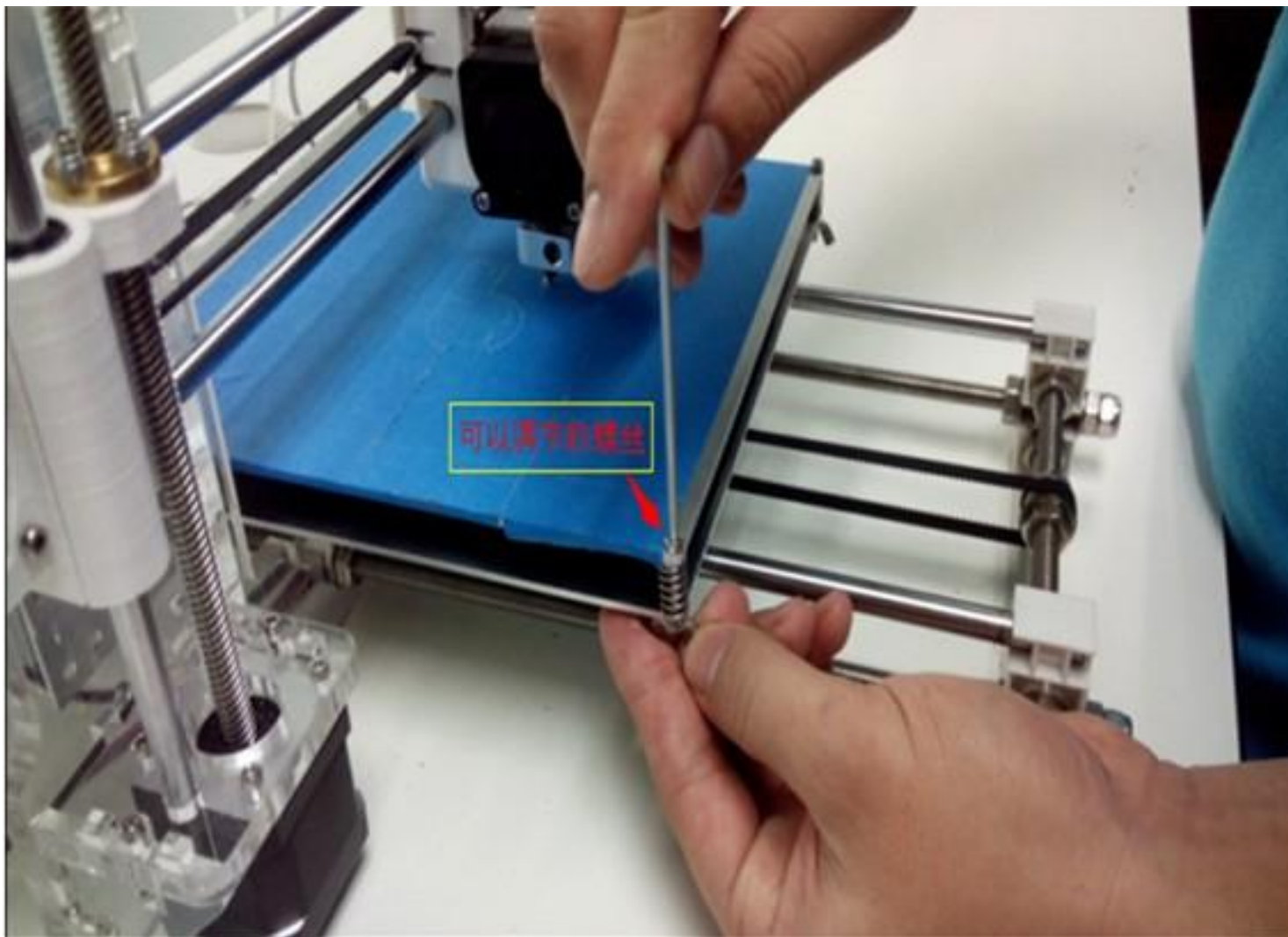
2. Press button - “Prepare” - “Disable steppers”



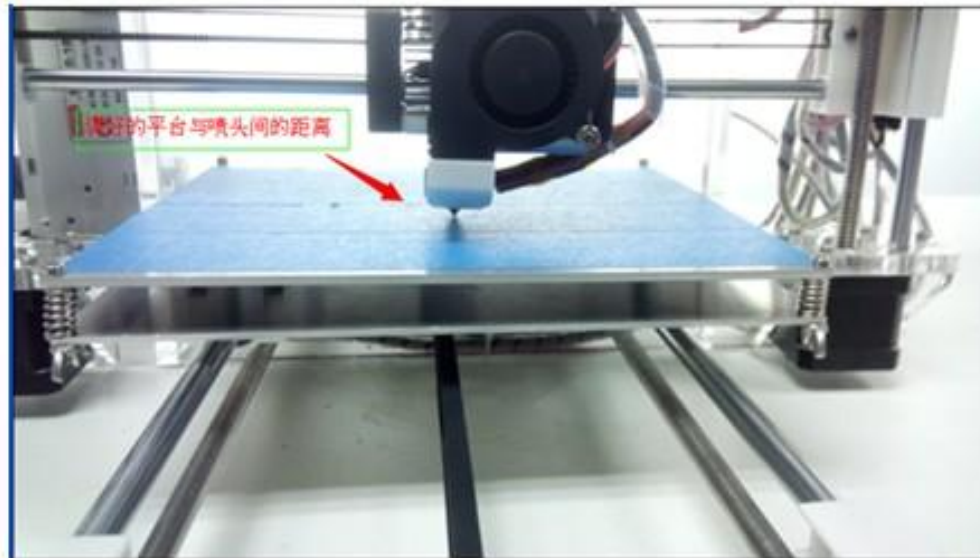
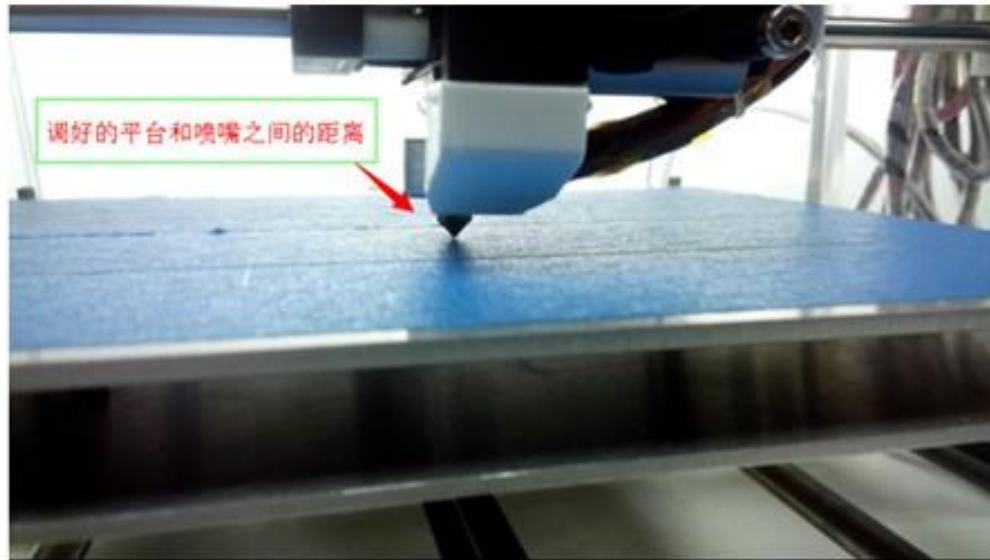
3. Move the extruder part all around the corner over the hot bed and check the distance between nozzle and platform.



4. To loose the screw when distance too far, on the contrary, tighten it when too close. The correct distance is not to scratch the blue making tape but just close to it. Shown as picture:

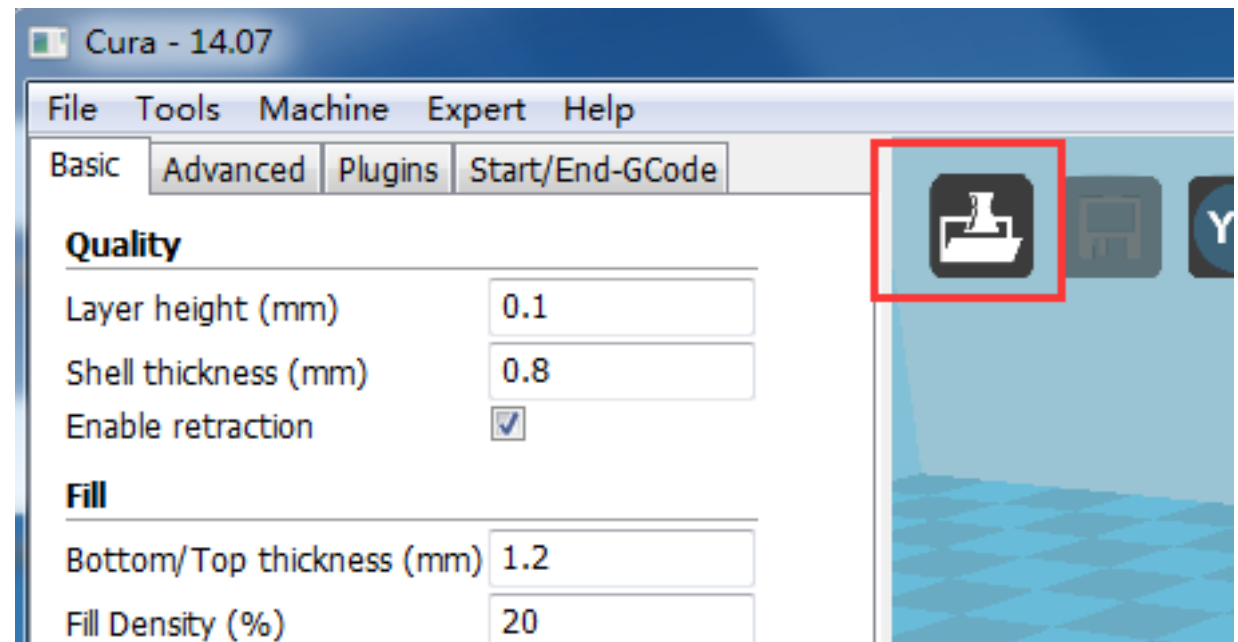
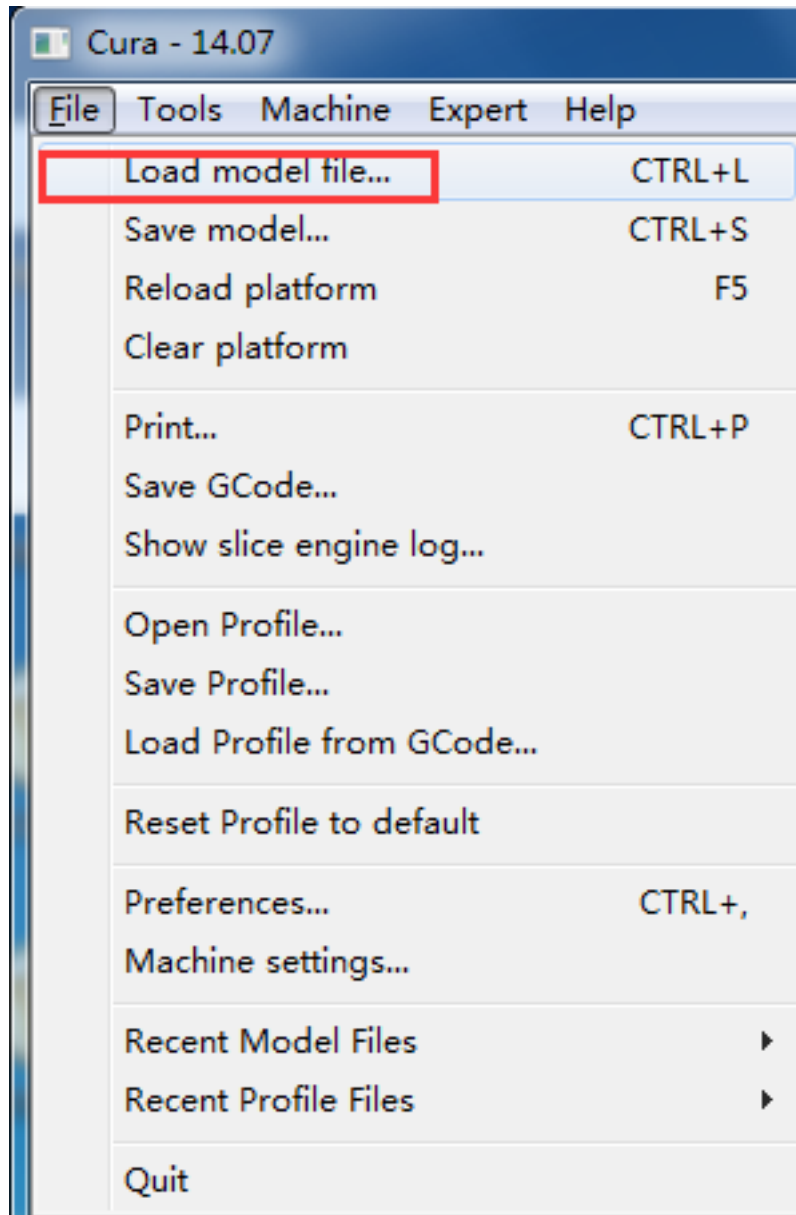


5. After adjustment, The space between nozzle and platform is about 0.15-0.25mm, the same as thickness as A4 paper. Shown as picture:

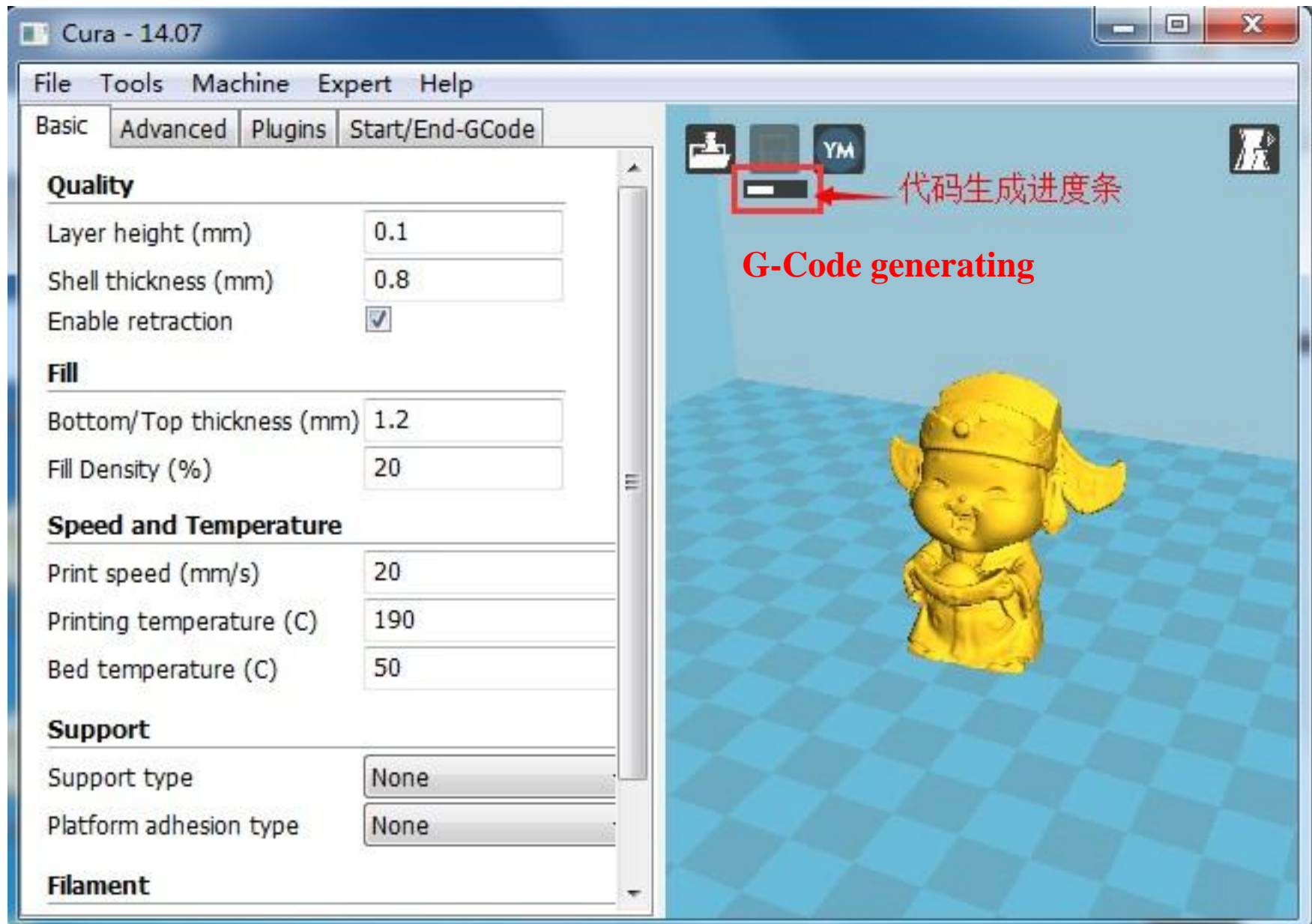


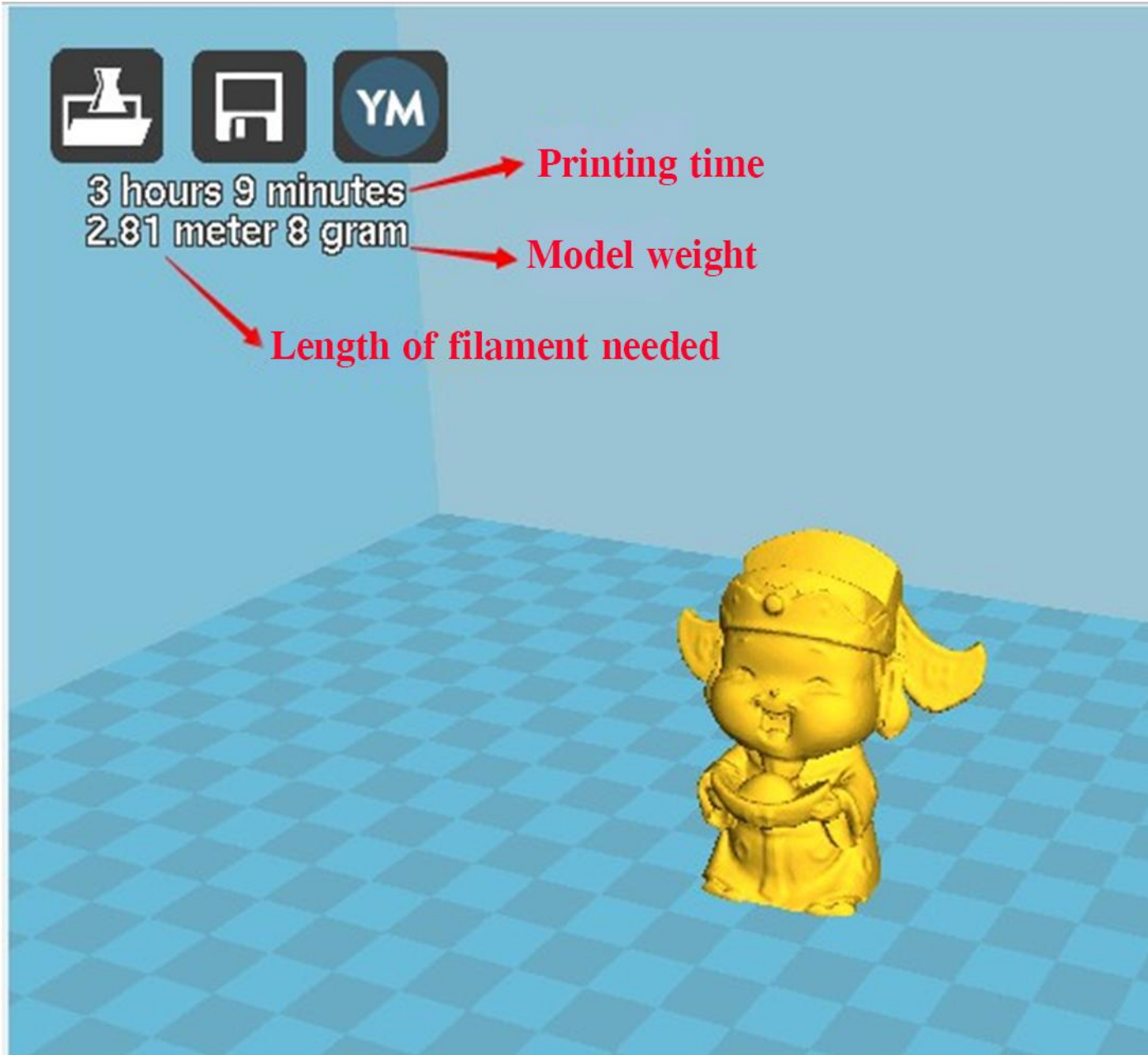
Initial printing

SD card offline printing



1. File loaded, both STL and OBJ file is available for this slicing software.





2. G-Code save to computer or SD card.



3. File--save GCode

1. Save to your PC →



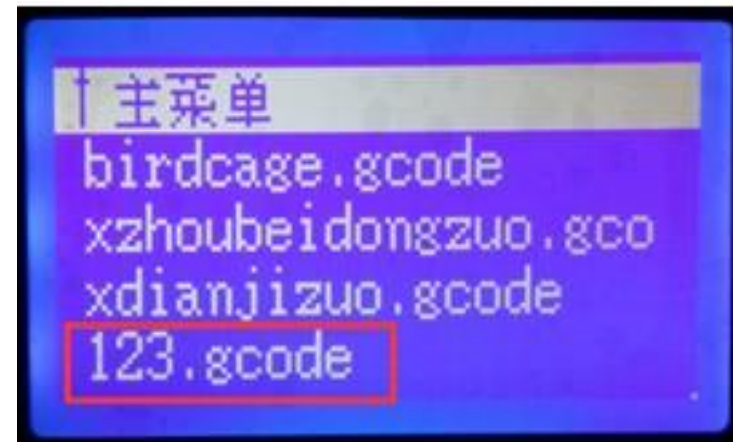
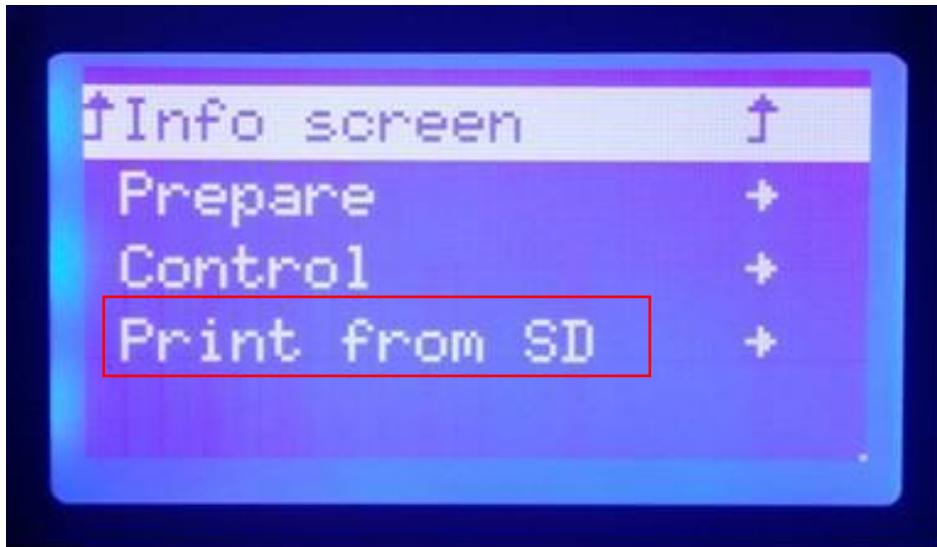
2. Save to SD card →



Insert SD card
to the card slot →



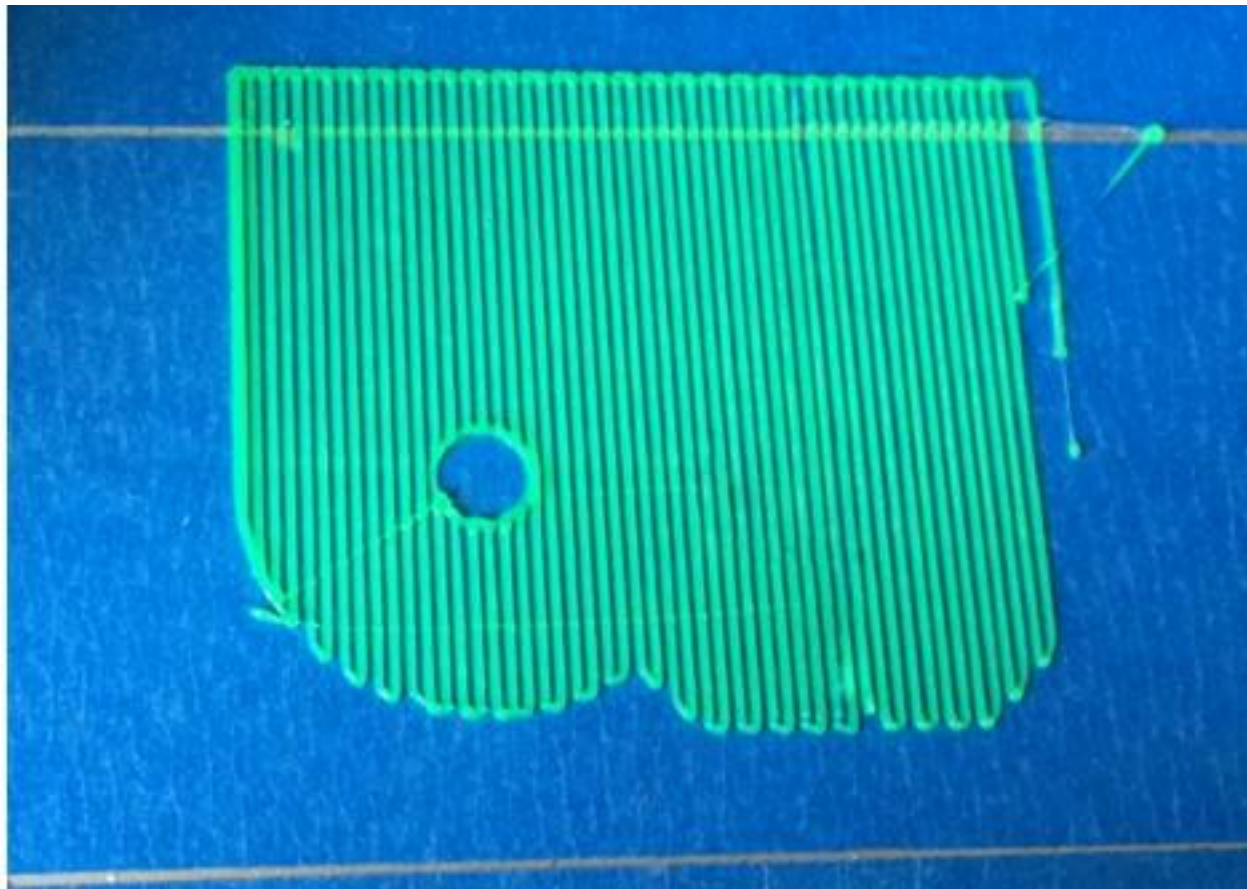
3. Start to print.



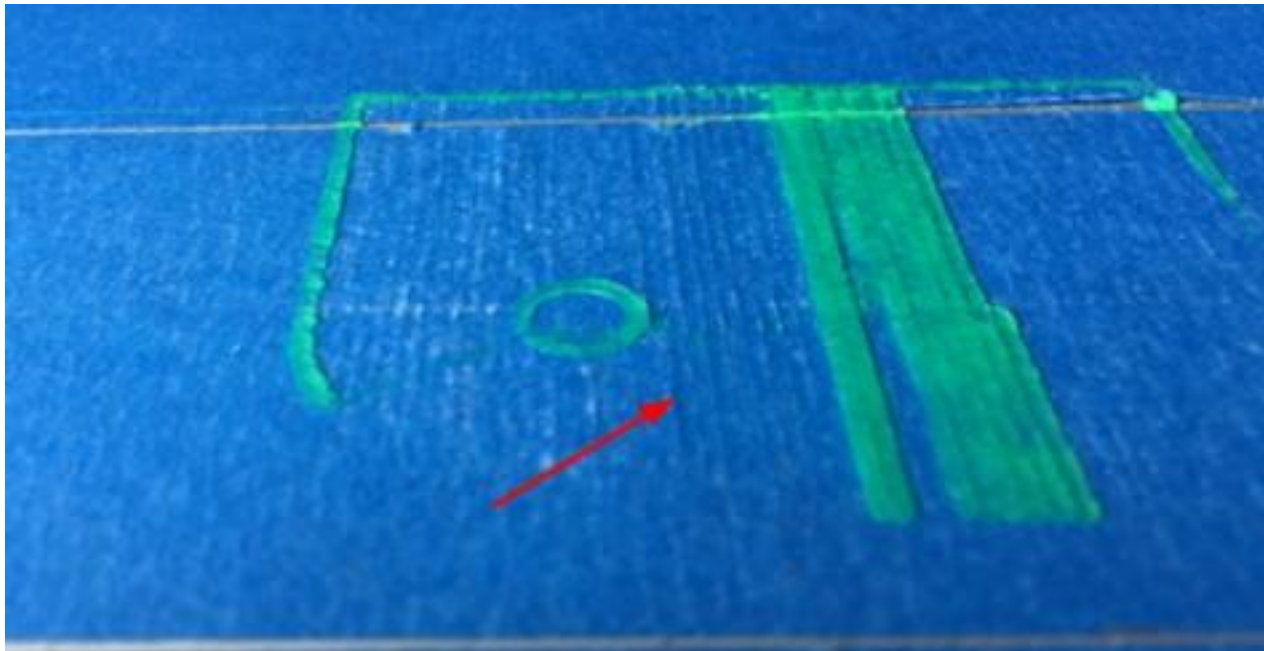
Select “Print from SD” from LCD display---Selecting the gcode you are going to print (such as : 123.gcode). The printer will be automatically heating up to temp.setted after pressing the rotary knob, then printer will automaticlly work up.

4. The distance between nozzle and platform is one of the most important factor that effect the printing result.

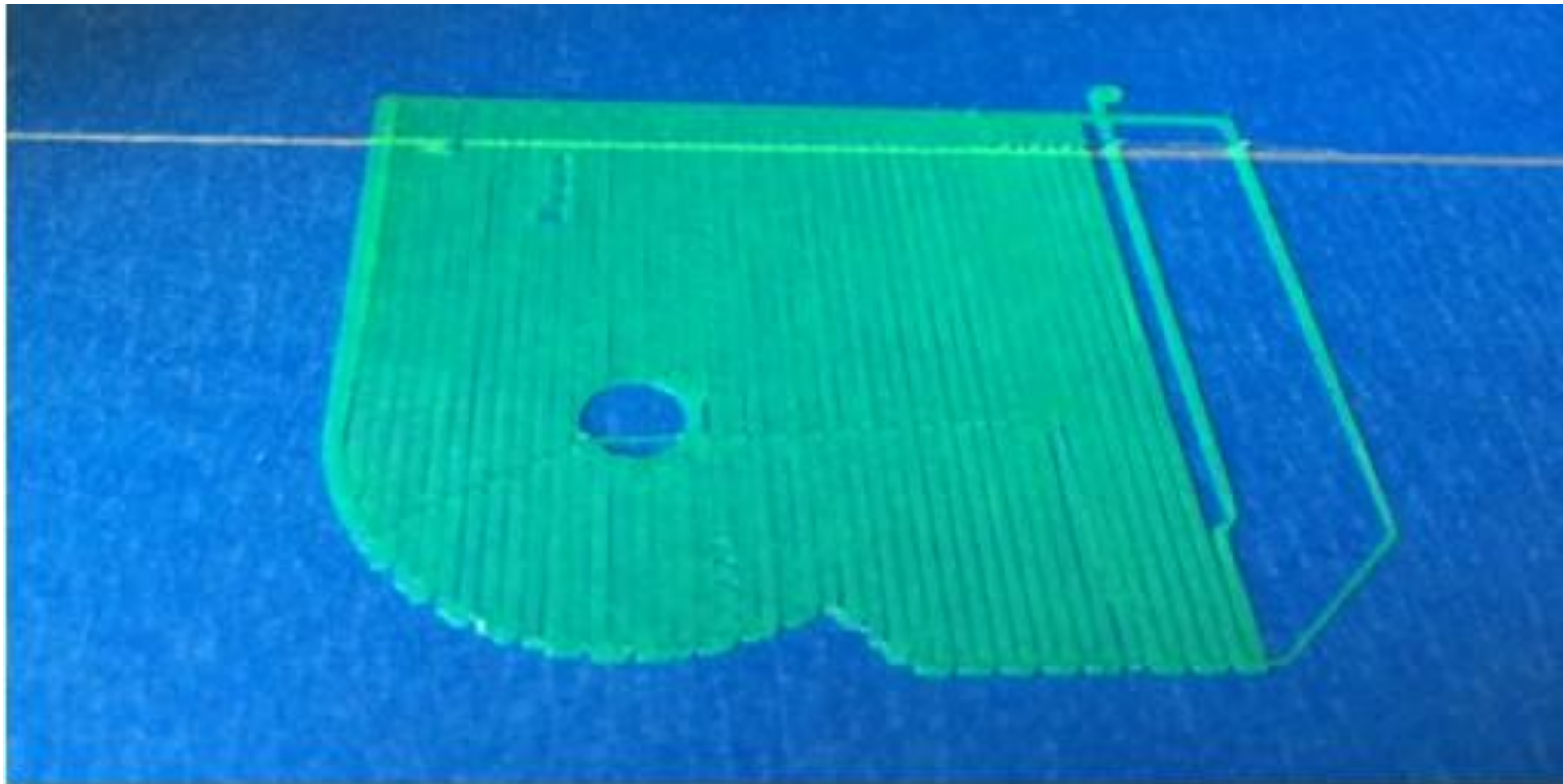
Far distance: Fine round ,uneven,gap and tilt .In this case, printing not good and easy to move.shown as picture:



Close distance: stop spinning, not good print, what's more, it will damage the nozzle, shown as picture:



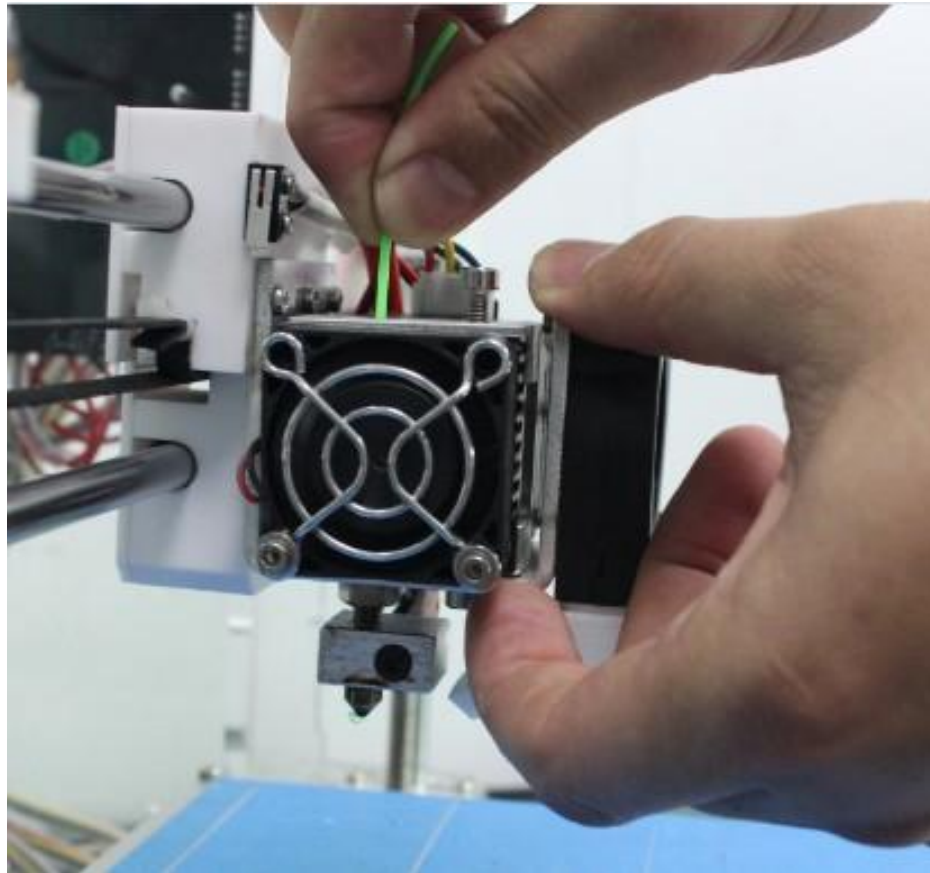
Right distance: Flat skinning, gapless, shown as picture:

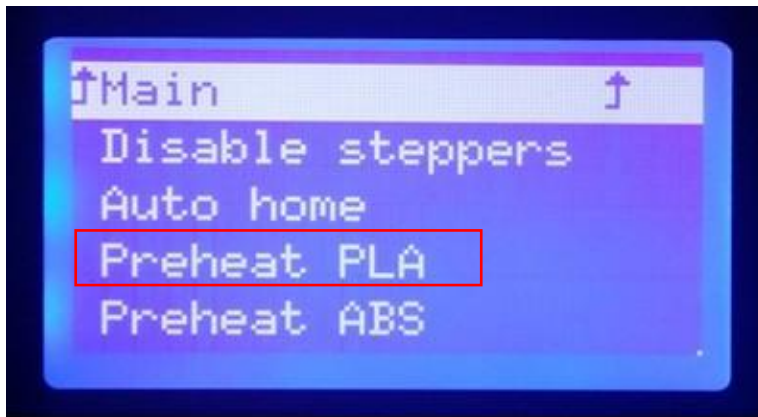


FAQ

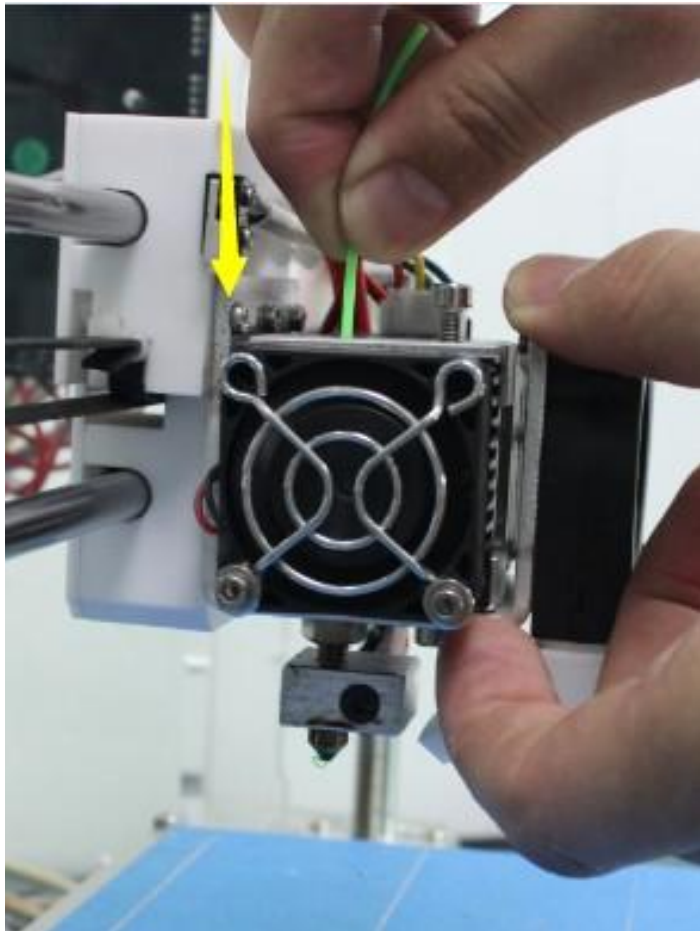
Extruder blocking

Situation A: when there is a little filament remanent and cannot pull out.



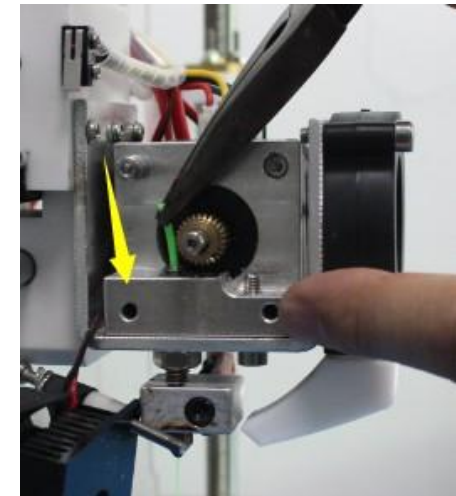
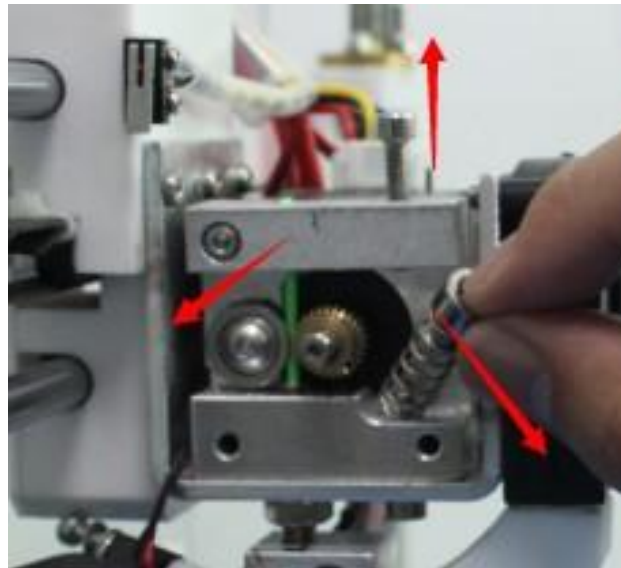
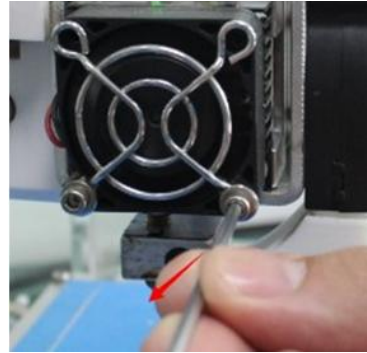
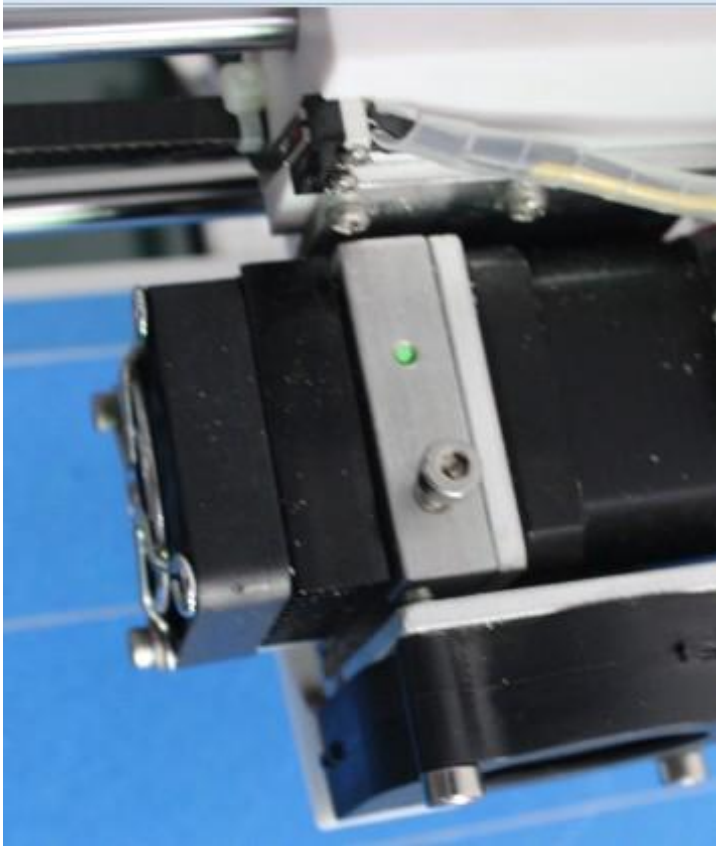


Select preheat PLA (it depends on ABS/PLA) Till temp. up to set temp. (below steps are proceed during heating)



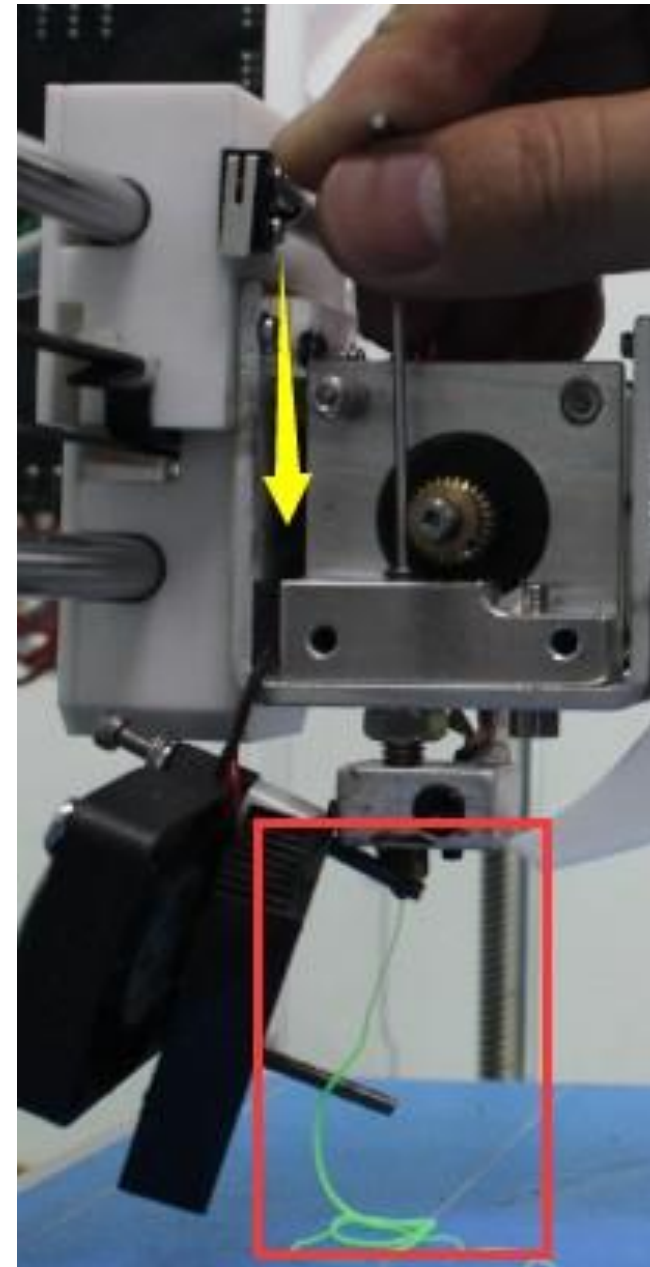
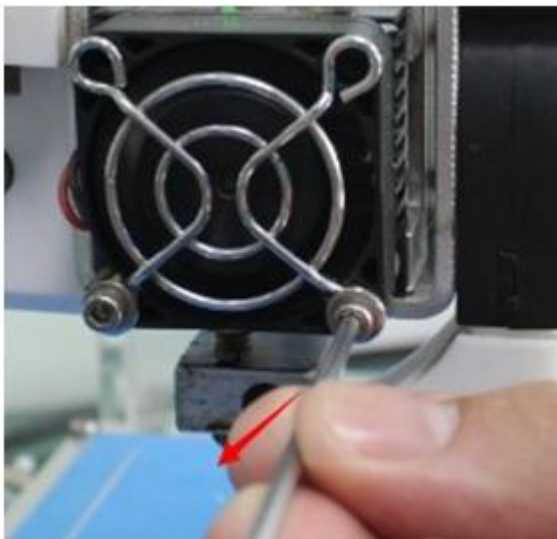
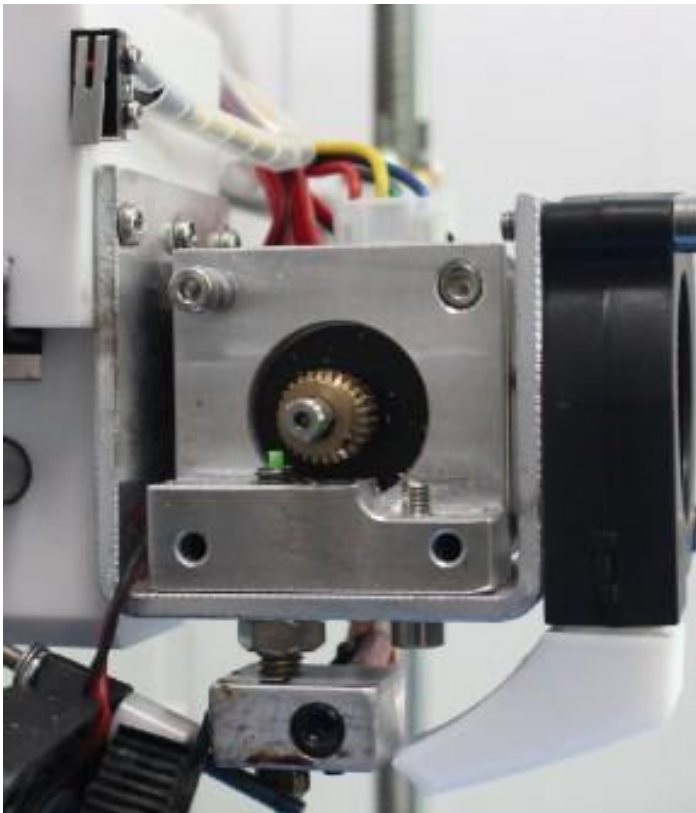
Push the filament by a hand for a little section, press the lock(nut) down by another hand at meanwhile, and then pull it out rapidly.

Situation B: When all filament feed into feeding mouth.



Keep preheating, use a allen wrench to dismantle the fan and heat sank part, and use the smallest hexagon wrech in the tool box (show as picture) to press the remain filament till it pull out from nozzle completely.

Situation C: When whole filament feed into the feeding mouth.



1. Loose the M4 screw
2. Take out the spring
3. Pull down Clamping piece

Use the smallest hexagon wrench and press the remain filament continuely till filament flow out completely. then, assemble all accessories again.

Thank you !

Any questions, please feel free to contact with us.