INTRODUCTION

Dear Friends,
Thank you very much for purchasing our 3D Printer. For optimum performance, safety and convenience, please read the instruction before DIY.

I. User Information

1. Software:

a. **External Memory/Micro SD Capacity**
From 2G to 32GB;

b. **SYSTEM REQUIREMENTS**
Support XP\Win7\Win8\MAC\Linux

Note:
Because of 3D Printer are bare metal, you have to pay attention to electrical safety during use! And the power outlet must be three-hole grounded outlet!

2. Safety Precautions

Before installing and using the machine, be sure to read the following. Do not attempt any user manual does not describe to use the machine to avoid personal injury and property damage accidents may cause.

3. Select the proper placement

- The machine is suitable for placement in a ventilated, cool, dry and less dusty environments.
- Note that when using thermal printer surroundings, avoid placing on a thick carpet or against a wall.
- Do not place the machine near flammable materials or high heat.
- Do not place the unit in a larger vibration or other unstable environment.
- Do not pile heavy objects on the unit.

4. Follow the standard use of power

- Use the power cord supplied with this unit.
- Refer to the "Installation" section of the machine performance parameters to select the appropriate 220V power supply.
- Do not plug the power cord when your hands are wet.
- Please use a plug, be sure to fully inserted into the power outlet.
- Do not deliberately pull, twist the machine over the distribution line, to avoid causing an open or short circuit。

5. Note that in the printing process

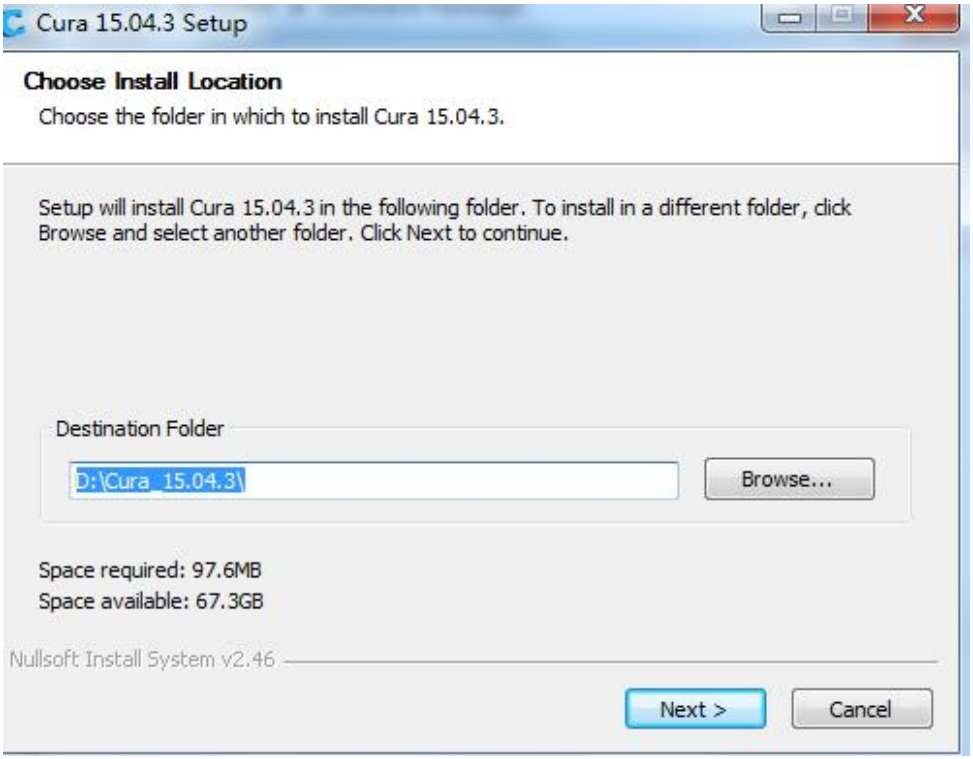
- Do not use the machine without the supervision of staff.
- Printing process and print just completed, avoid touching the printer's internal structure and prints, to prevent burns.
- If occur when printing printer smoke, abnormal noise, immediately turn off the power switch, the printer stops working, and contact your purchaser.
- Often do product maintenance
- Do not attempt to use the method described in the manual does not disassemble or modify the machine to prevent damage to the printer or other more serious accidents.
- Regularly in case of power failure, the printer clean with a dry cloth, wipe the dust and bonding of printed materials. If you must use a damp cloth to clean, do not use flammable solvents, flammable solvents to prevent contact with the printer's internal circuitry result in a fire or electric shock.

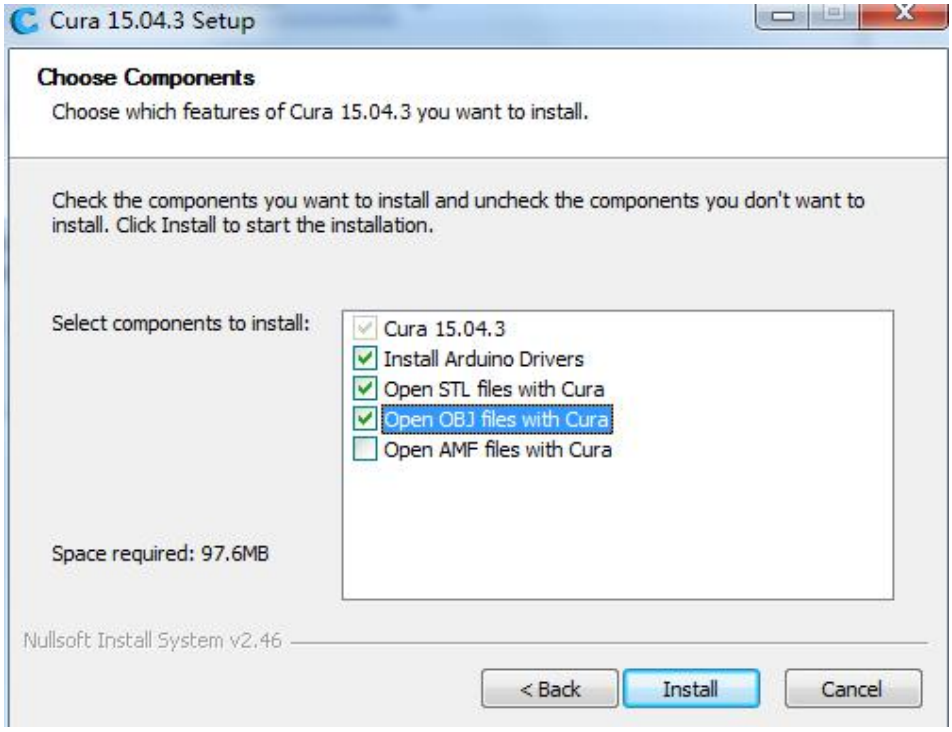
II. Cura Installation and Setup

1. Slice software installation

a. Install the software

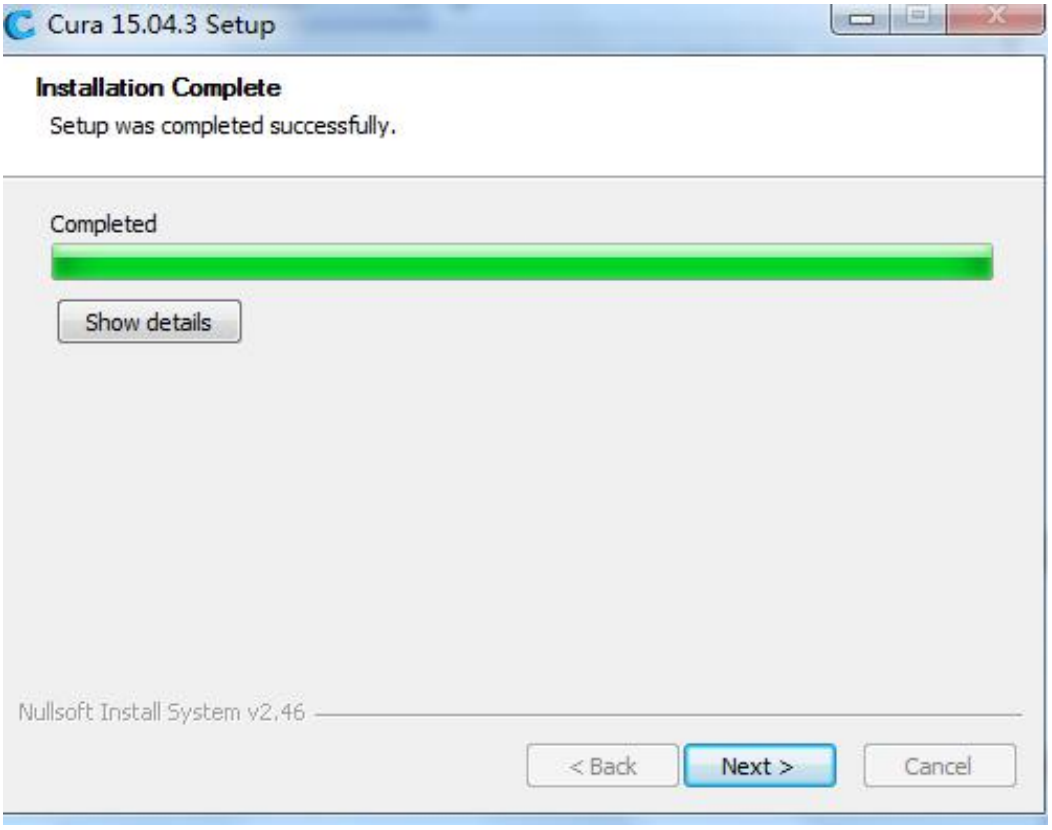
Click Setup Cura_15.04.3.exe, pop-up installation window, always click Next to complete the installation.

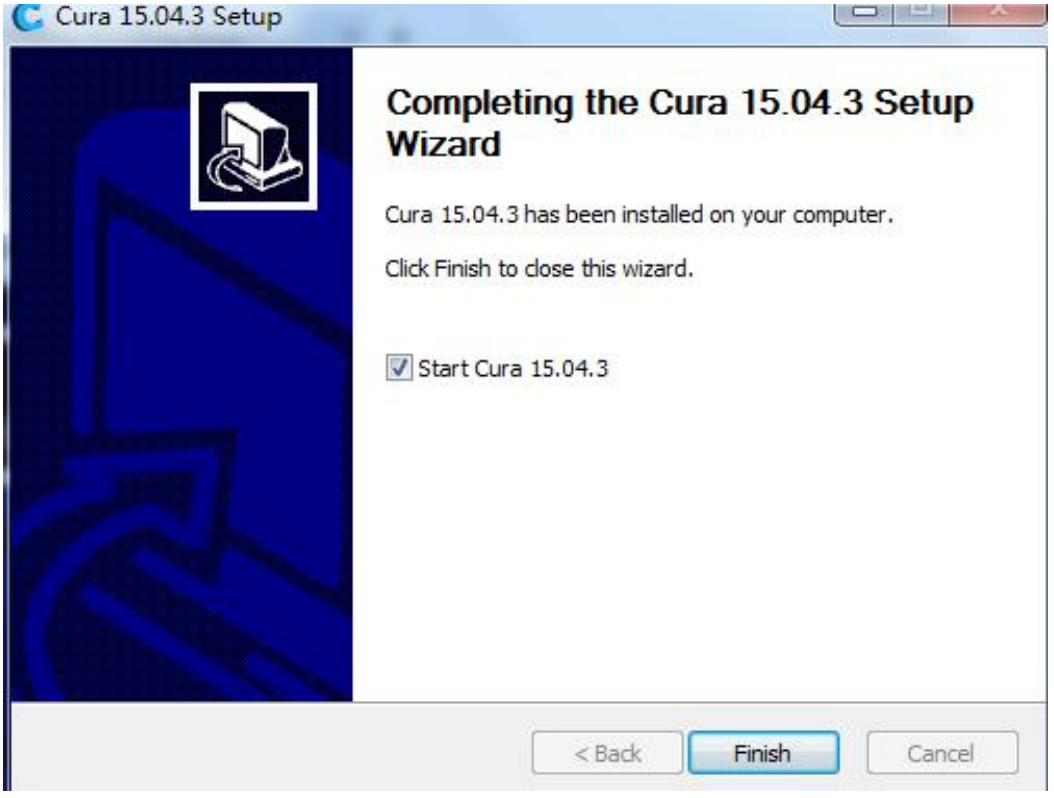


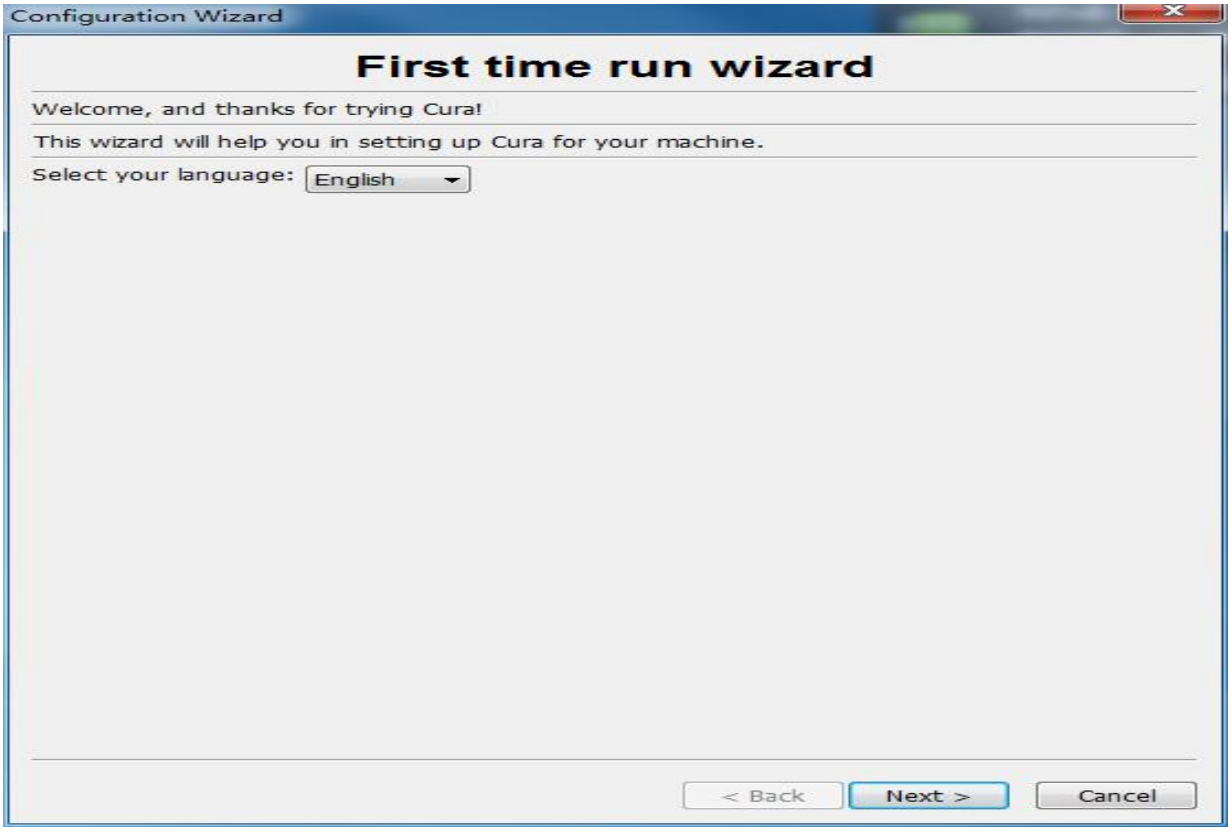


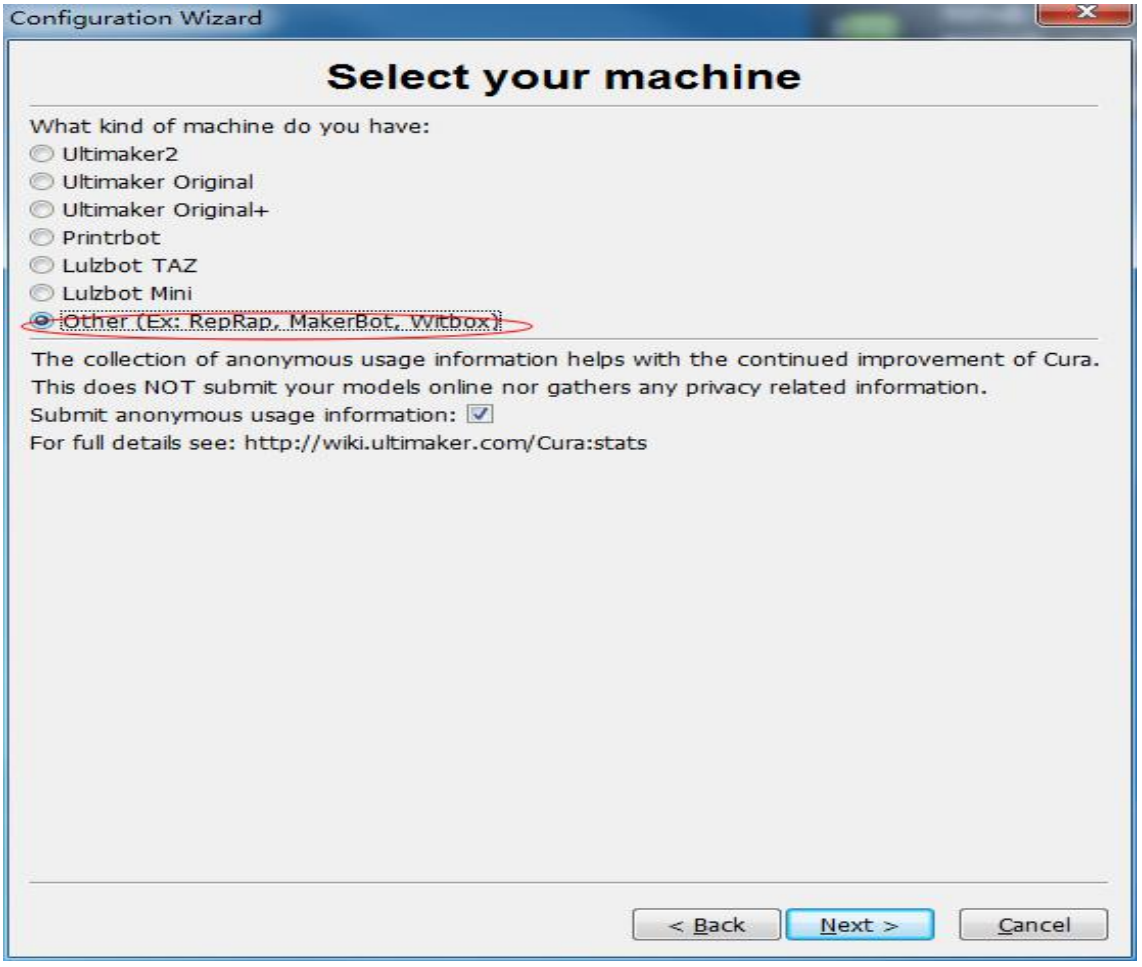

Device Driver Installation Wizard
Welcome to the Device Driver Installation Wizard!
This wizard helps you install the software drivers that some computers devices need in order to work.
To continue, click Next.
< 上一步(B)
下一步(N) >
取消

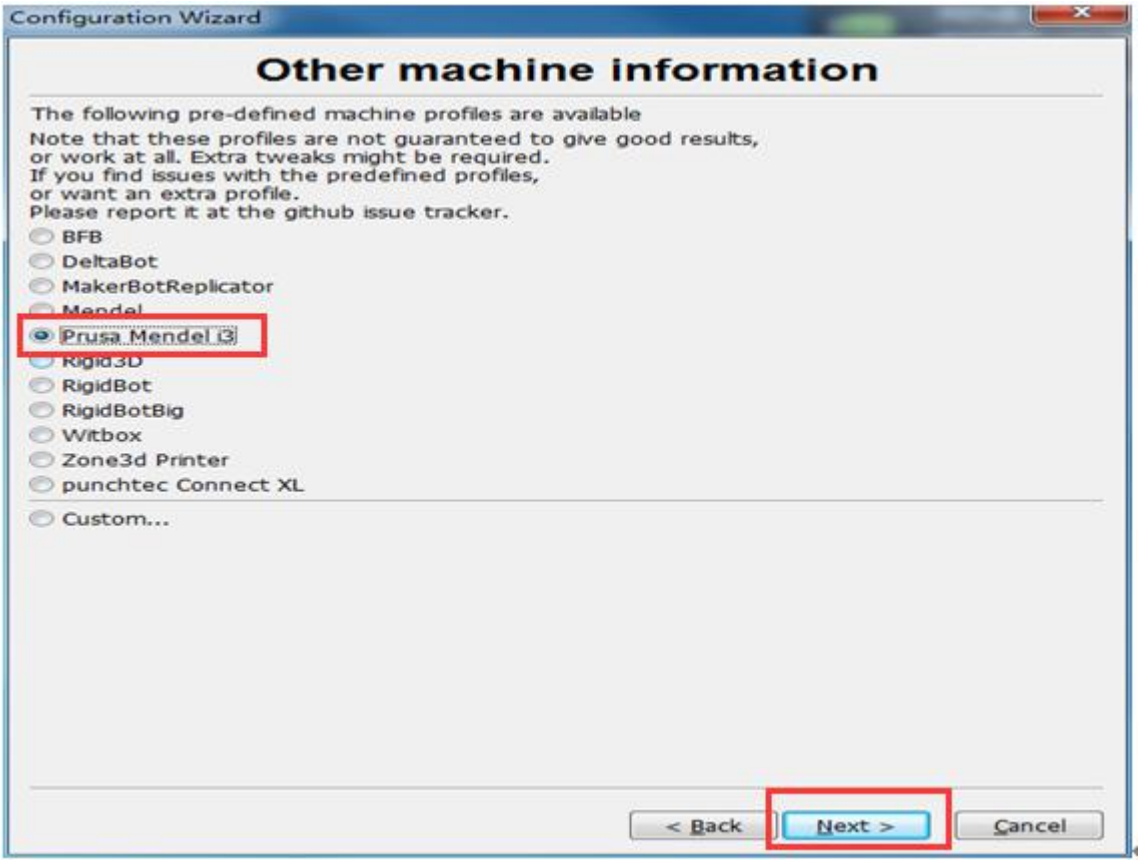

Device Driver Installation Wizard
Completing the Device Driver Installation Wizard
The drivers were successfully installed on this computer.
You can now connect your device to this computer. If your device came with instructions, please read them first.
Driver Name
Status
Arduino LLC (www.ardui...
Ready to use
< 上一步(B)
完成
取消


Cura 15.04.3 Setup
Installation Complete
Setup was completed successfully.
Completed
Show details
Nullsoft Install System v2.46
< Back
Next >
Cancel

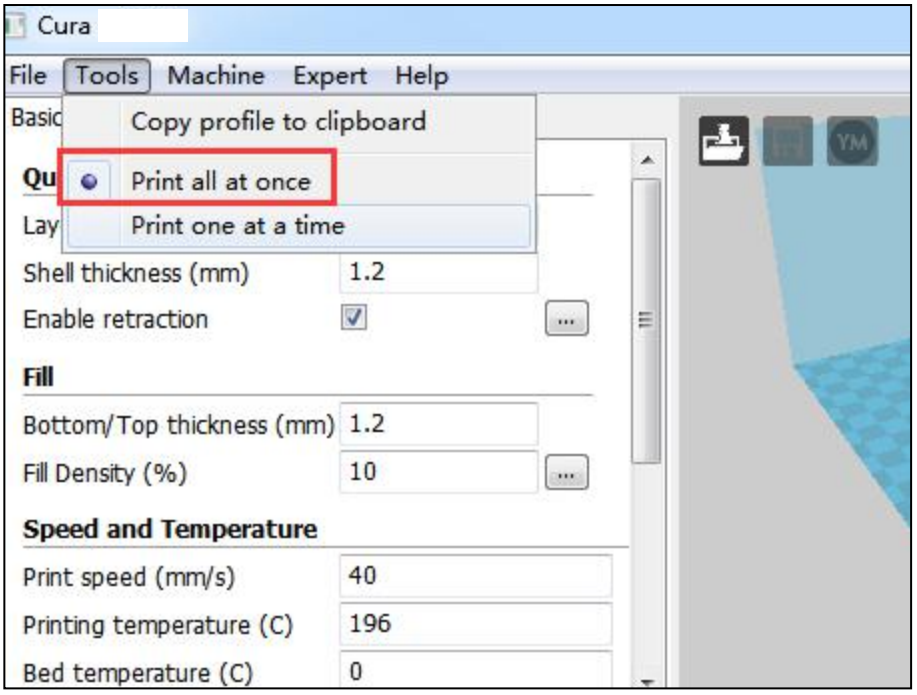

Cura 15.04.3 Setup
Completing the Cura 15.04.3 Setup Wizard
Cura 15.04.3 has been installed on your computer.
Click Finish to close this wizard.
Start Cura 15.04.3
< Back
Finish
Cancel

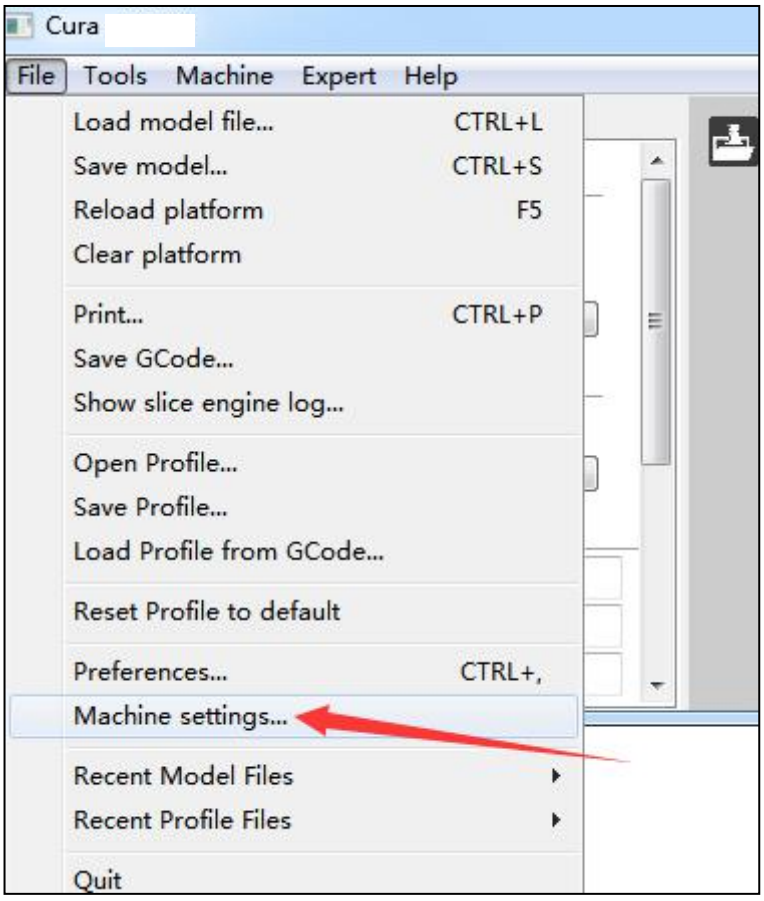

Configuration Wizard
First time run wizard
Welcome, and thanks for trying Cura!
This wizard will help you in setting up Cura for your machine.
Select your language:
English
< Back
Next >
Cancel

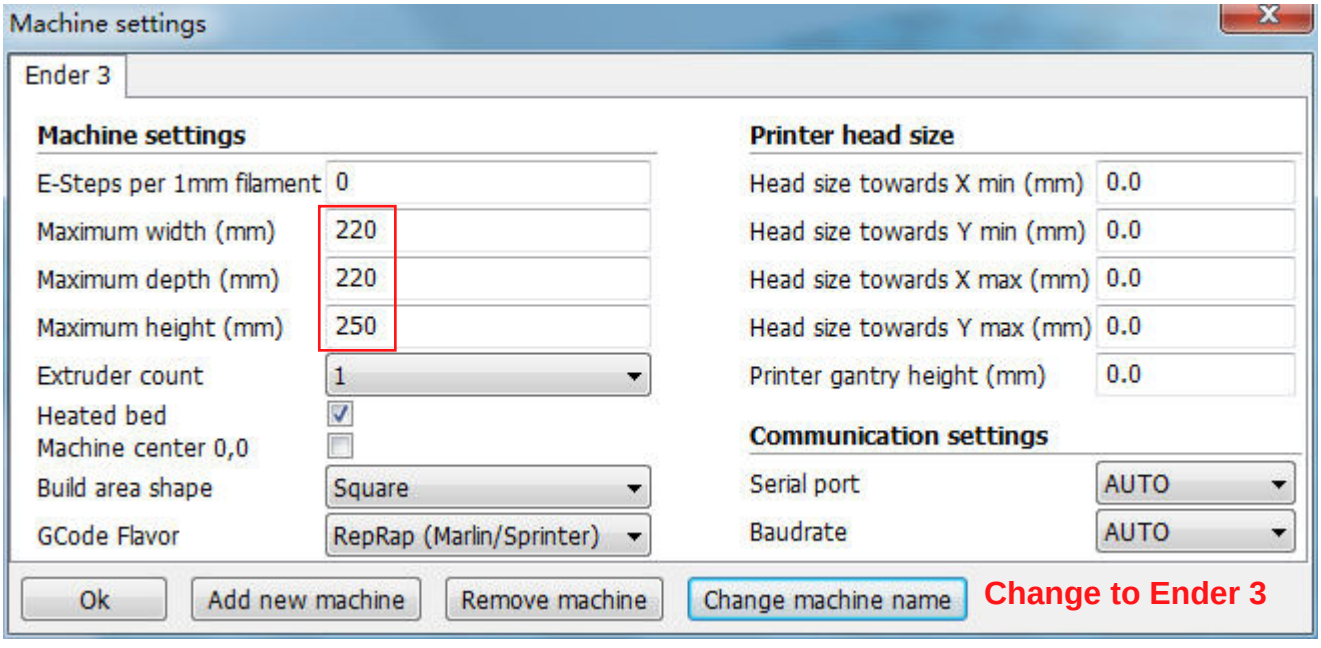

Configuration Wizard
Select your machine
What kind of machine do you have:
Ultimaker2
Ultimaker Original
Ultimaker Original+
Printrbot
Lulzbot TAZ
Lulzbot Mini
Other (Ex: RepRap, MakerBot, Witbox)
The collection of anonymous usage information helps with the continued improvement of Cura.
This does NOT 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



b. Select "print multiple model" from the "Tools"



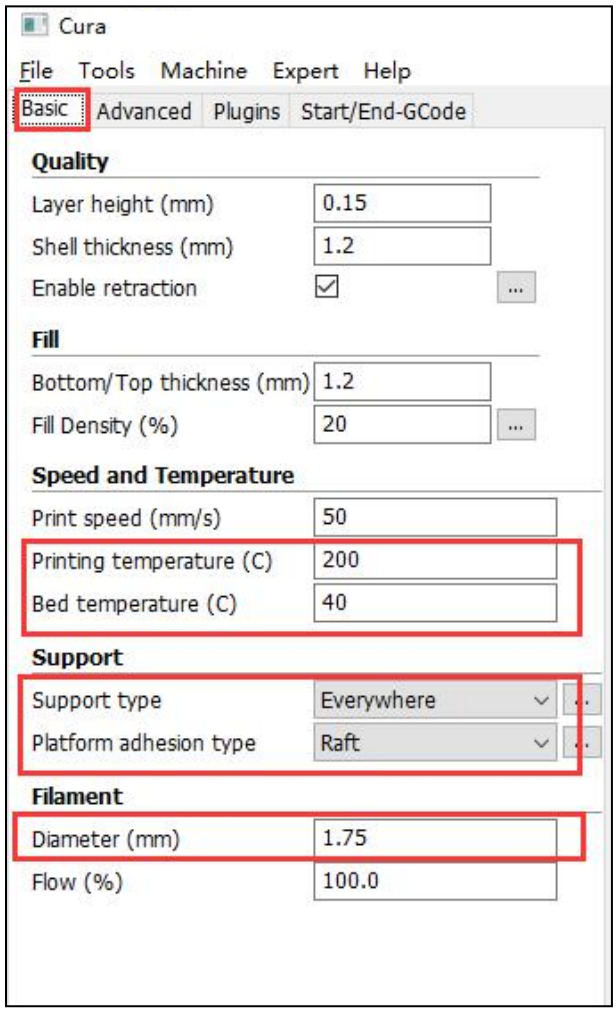

Cura
File
Tools
Machine
Expert
Help
Load model file... CTRL+L
Save model... CTRL+S
Reload platform F5
Clear platform
Print... CTRL+P
Save GCode...
Show slice engine log...
Open Profile...
Save Profile...
Load Profile from GCode...
Reset Profile to default
Preferences... CTRL+,
Machine settings...
Recent Model Files
Recent Profile Files
Quit

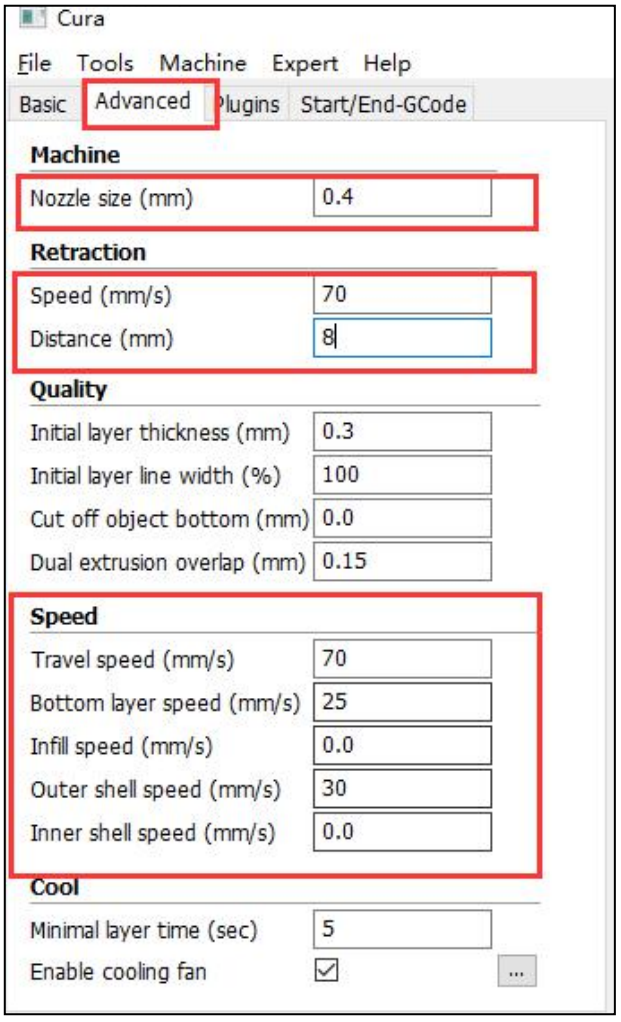

Machine settings
Ender 3
Machine settings
E-Steps per 1mm filament 0
Maximum width (mm) 220
Maximum depth (mm) 220
Maximum height (mm) 250
Extruder count 1
Heated bed
Machine center 0,0
Build area shape Square
GCode Flavor RepRap (Marlin/Sprinter)
Printer head size
Head size towards X min (mm) 0.0
Head size towards Y min (mm) 0.0
Head size towards X max (mm) 0.0
Head size towards Y max (mm) 0.0
Printer gantry height (mm) 0.0
Communication settings
Serial port AUTO
Baudrate AUTO
Ok
Add new machine
Remove machine
Change machine name
Change to Ender 3

2. Software parameter setting

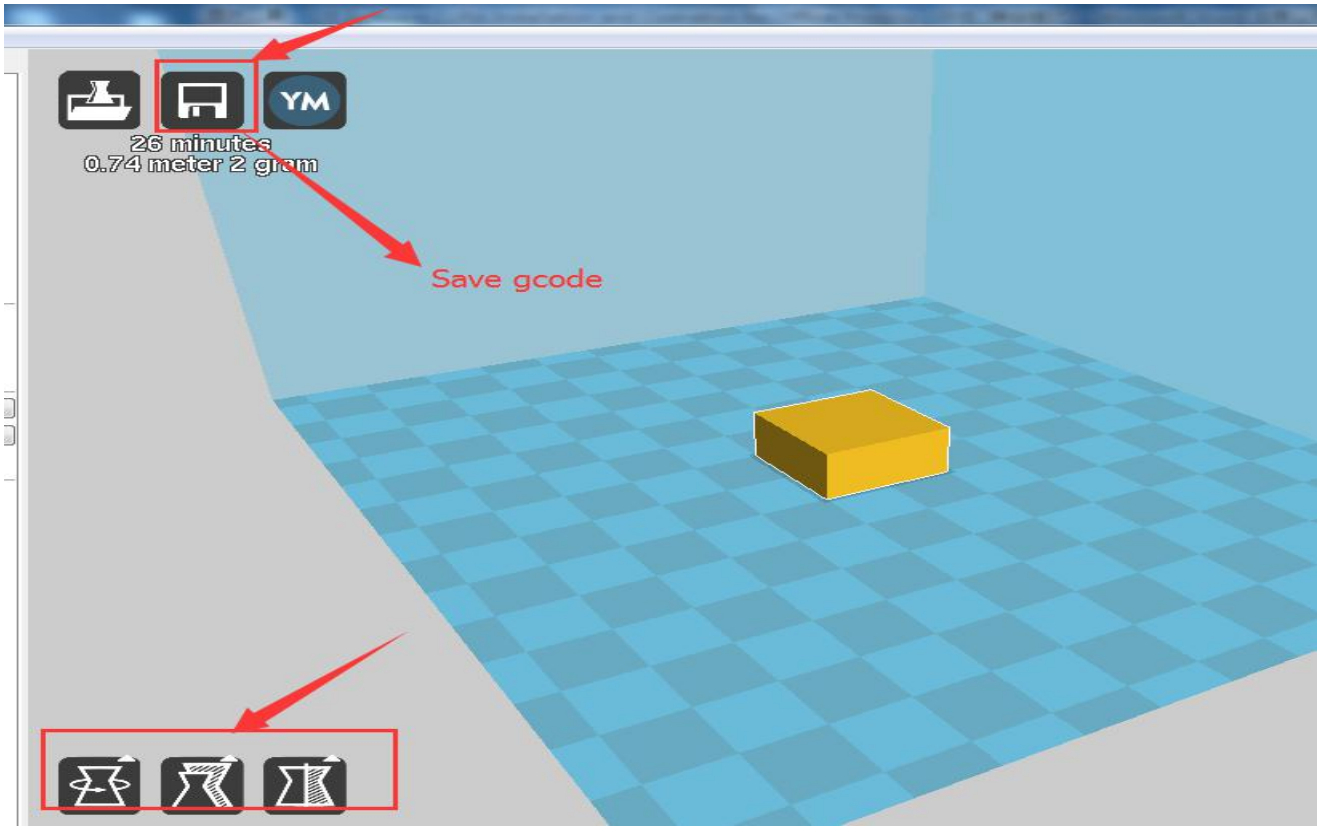
Open Cura_15.04.3.exe, you can modify all of the printing process control parameters display screen.

a. Recommended parameter settings:





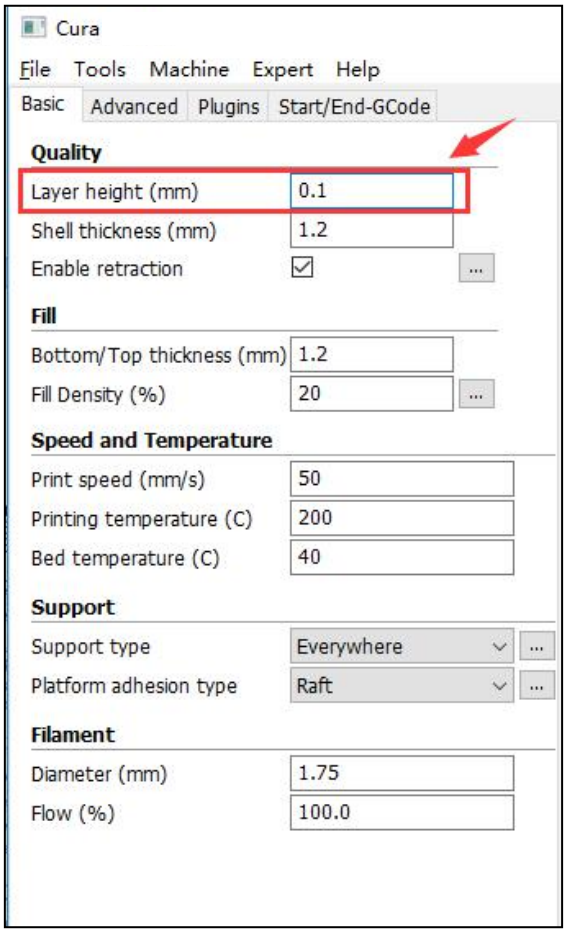
b. For in-depth study, refer to the following notes:

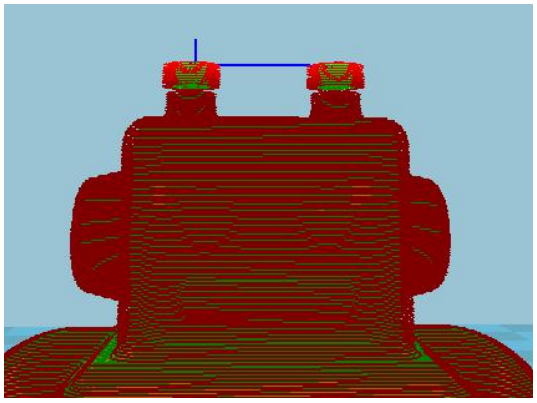


The saved "* .gcode" file name must be in English or number, can't named as Others,save the "* .gcode" file to your TF card and then insert it to the circuit board, and restart the power, Select "From to SD" in the main menu from LCD Control Screen,then select the saved "* .gcode" documents, press button to confirm to warms up and begin to print.

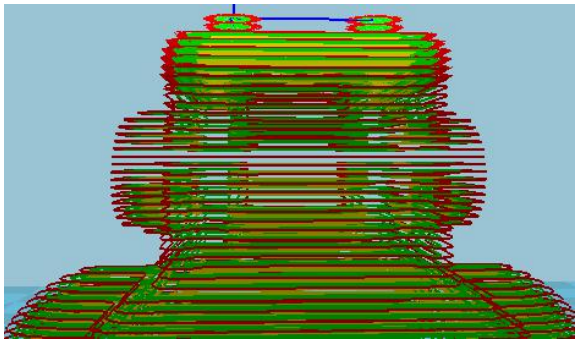
If the in-depth studies are needed, please read the following items:

1、Layer height





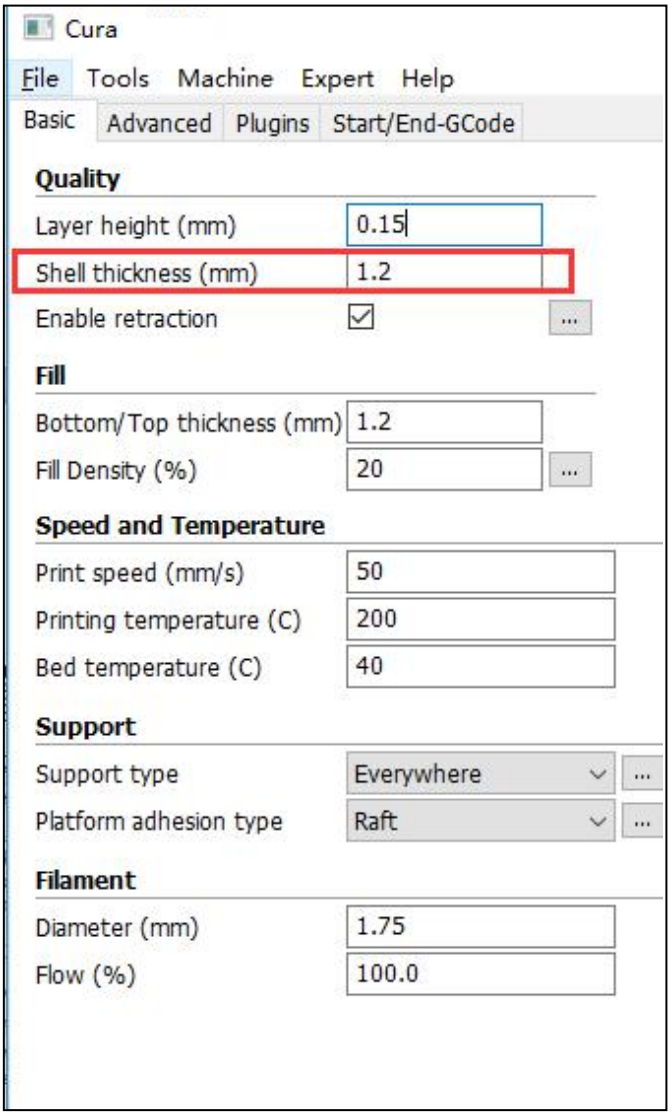
(0.1mm)

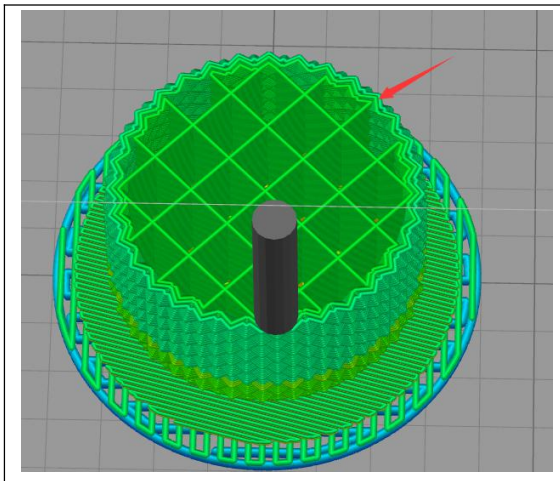


(0.3mm)

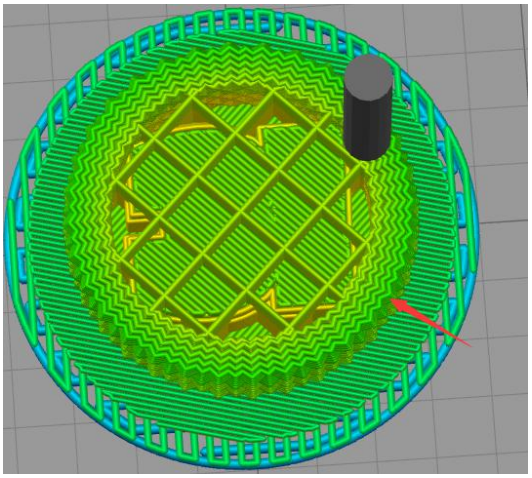
Explanation：Layer height in millimeters. This is most important setting to determine the quality of you print.Normal quality are 0.2mm, high quality is 0.1mm. You can go up to 0.25mm with a CR-10 for every for fast prints at low quality.

2、Shell thickness





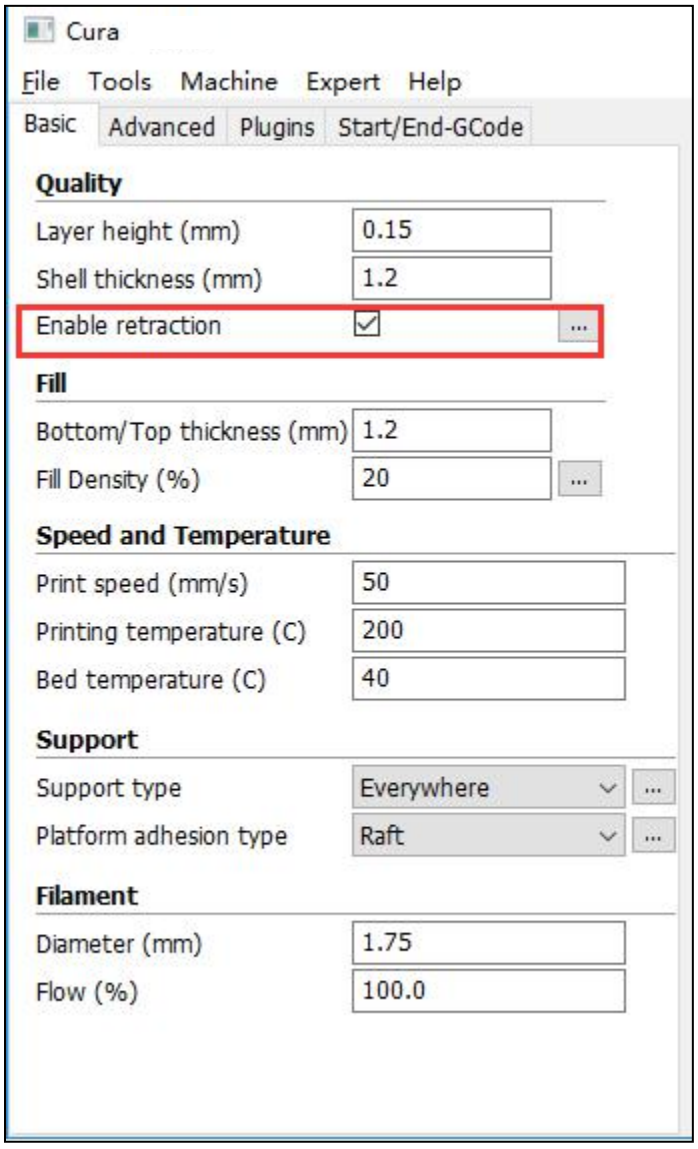
0.8mm(If the nozzle is 0.4mm)

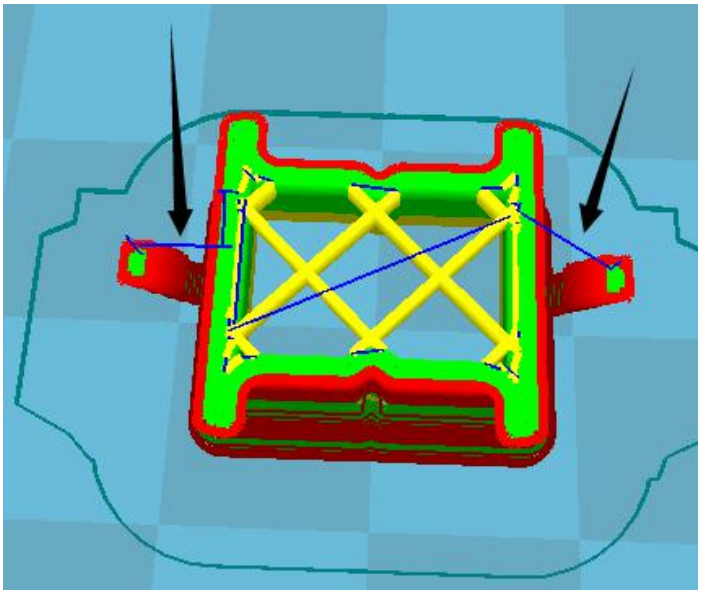


2.0mm(If the nozzle is 0.4mm)

Explanation：Thickness of the outside shell in the horizontal direction. This is used in combination with the nozzle size to define the number of perimeter lines and the thickness of those perimeter lines.

3、Enable retraction

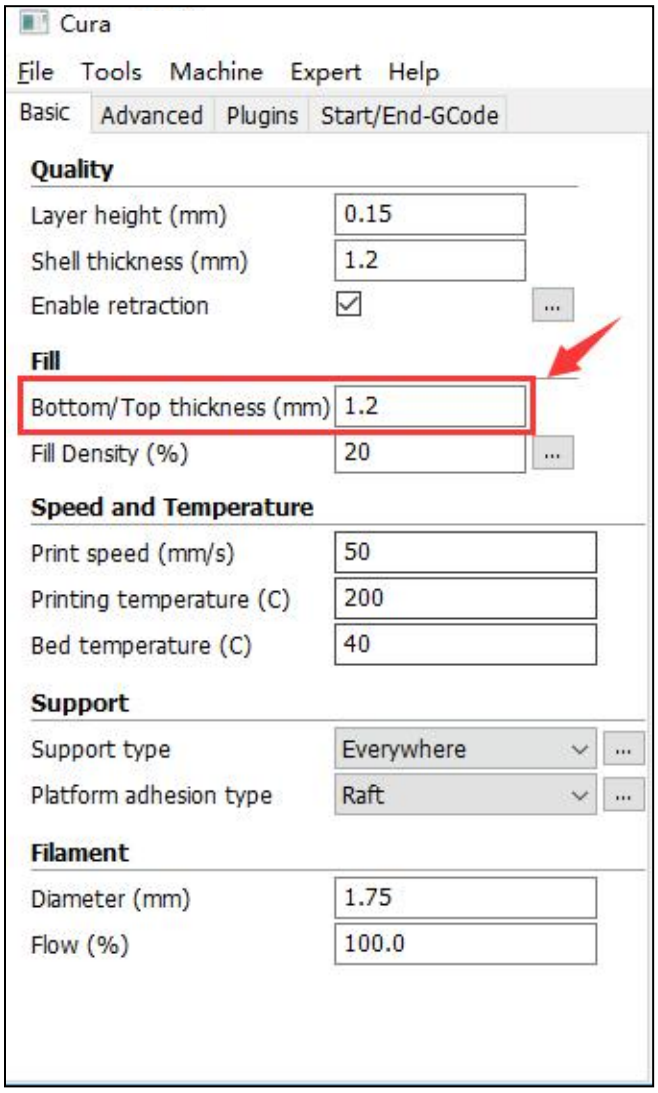


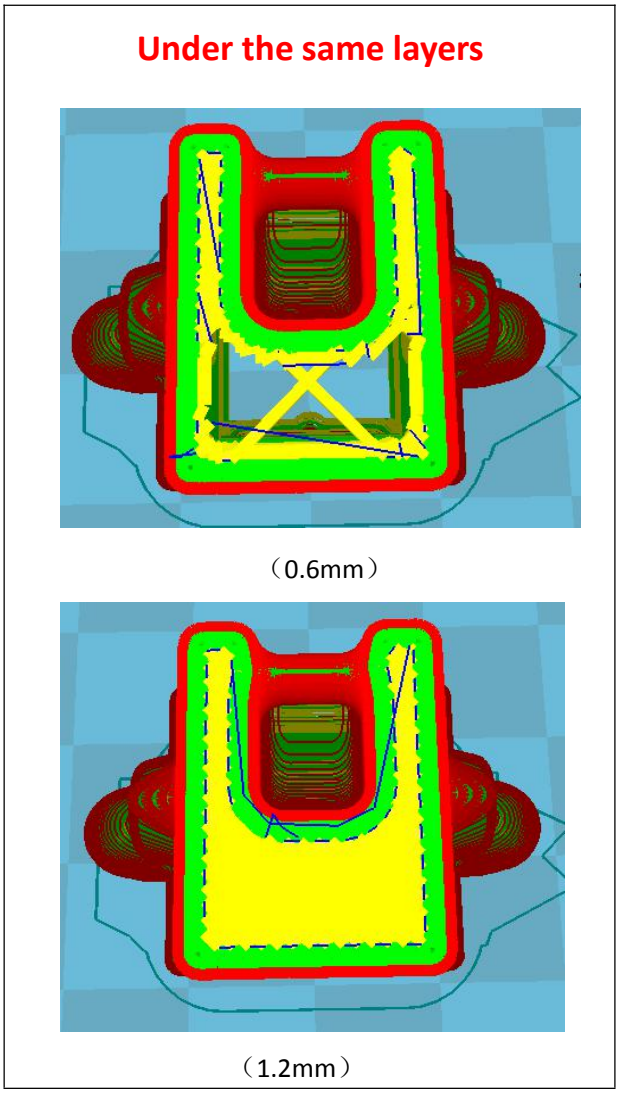


Explanation：

Retraction is for without letting the material flow out during printing.Otherwise it will affect printing surface.Details about the retraction can be configured in the advanced tap.

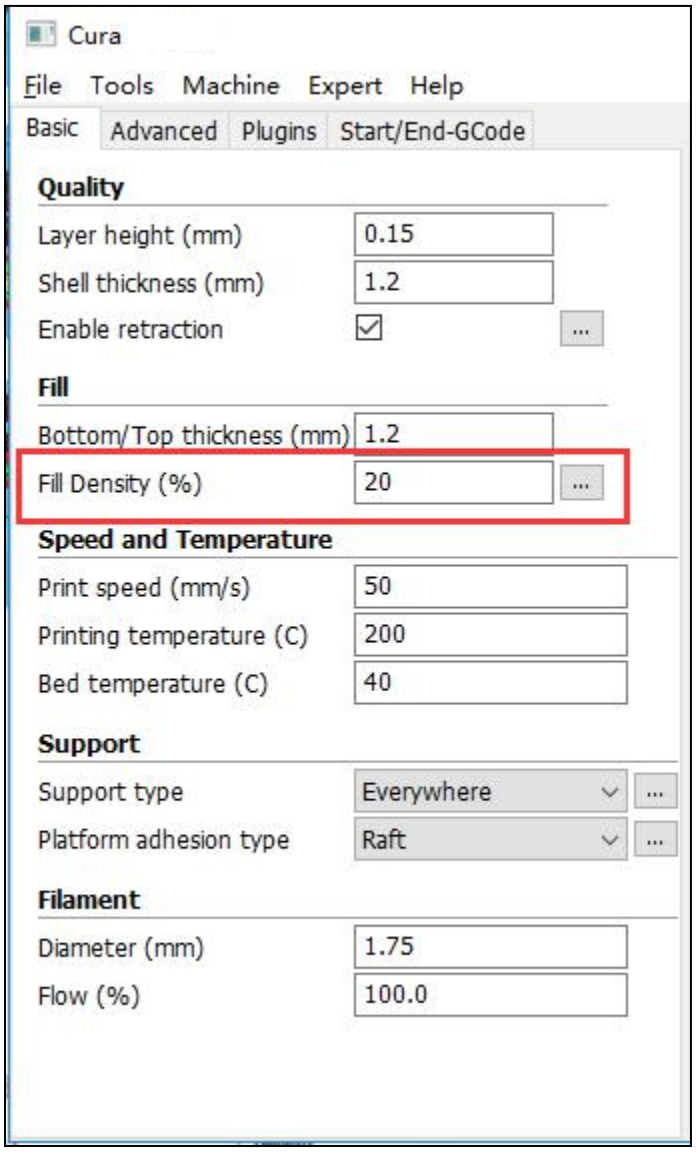
4、Bottom/Top thickness

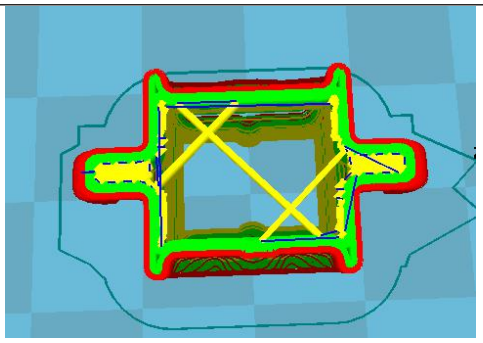




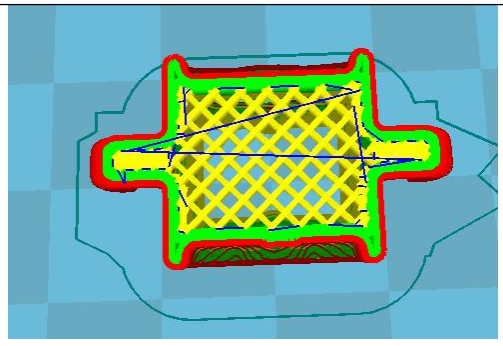
Explanation： This control the thickness of the bottom and top layers , the amount of solid layers put down is calculated by the layer thickness and this value .Having this value a multiple of the layer thickness makes sense .And keep it near your wall thickness to make an evenly strong part.

5、Fill Density(%)

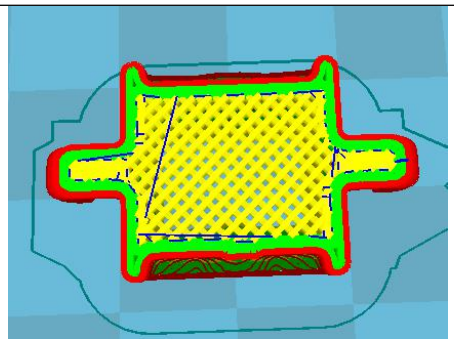




（10%）



（30%）

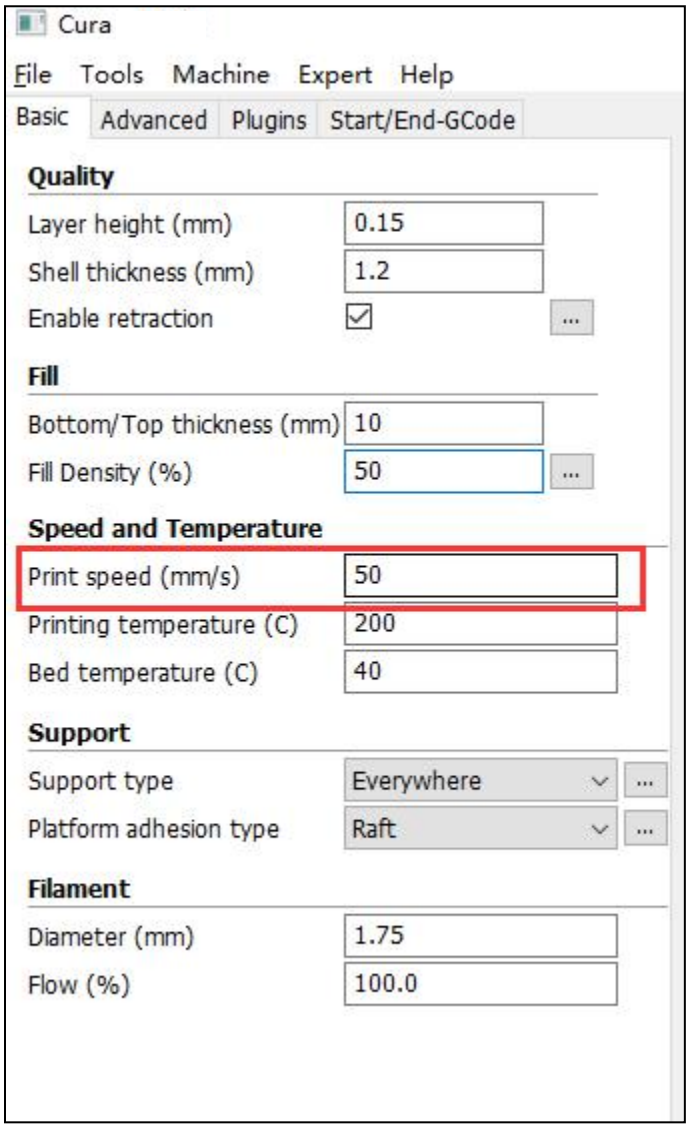


（50%）

Explanation：

For a solid part use 100%, for an empty part use 0%. A value around 20% is usually enough. It adjusts how strong the parts becomes.

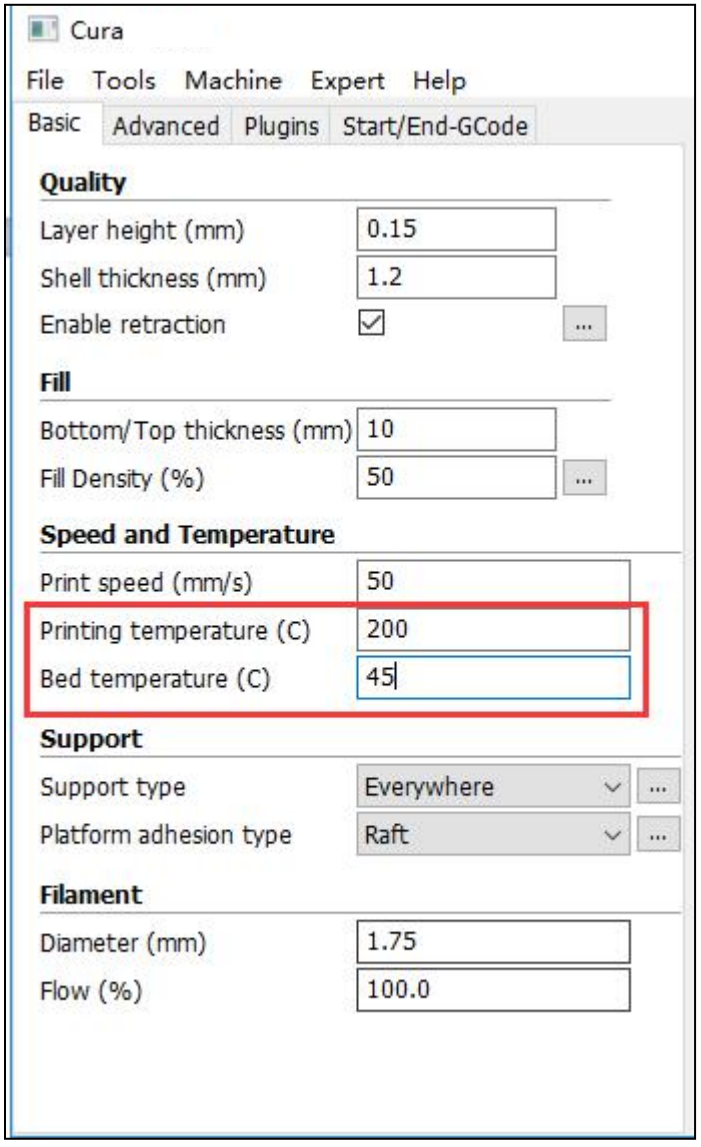
6、Print speed（mm/s）



Explanation： Speed at which printing happens .

Suggest 50-80, according to what you print. Faster speed, worse effect.

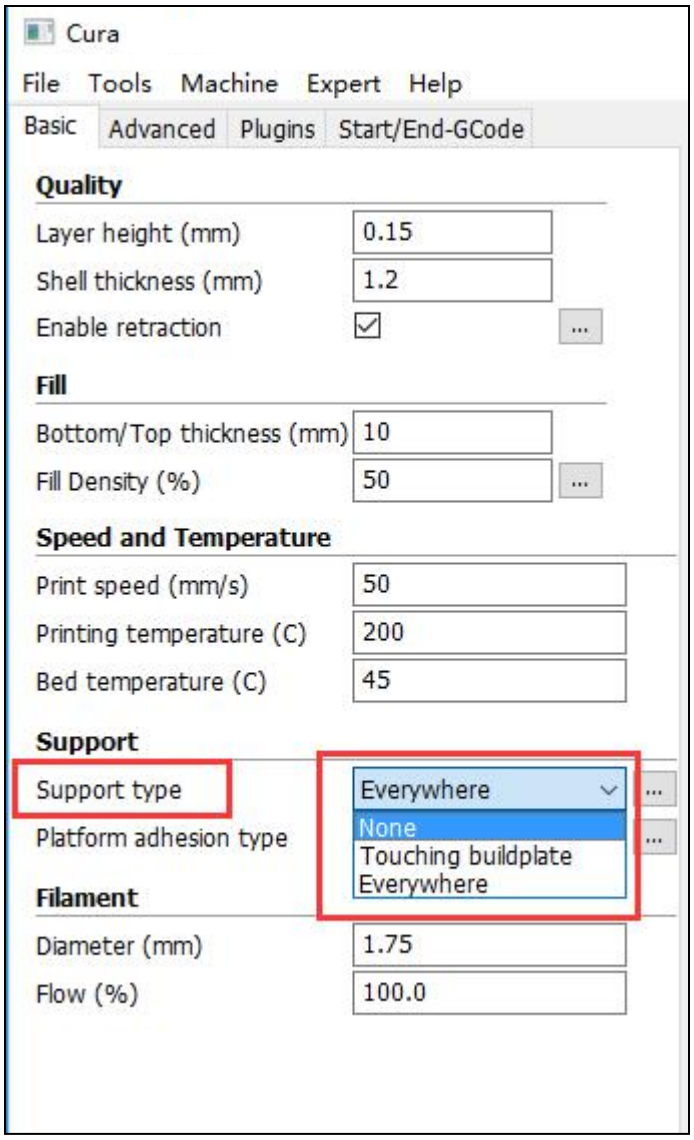
7、Print temperature(℃)

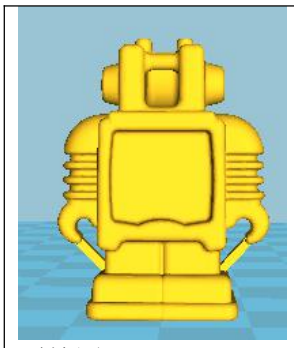


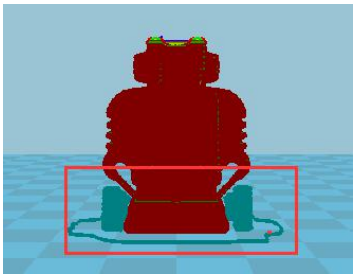
Explanation：

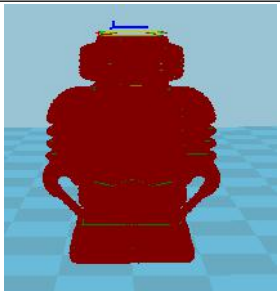
Printing temperature used for printing . PLA:nozzle is 190-220,generally 200;Bed is 45-50.

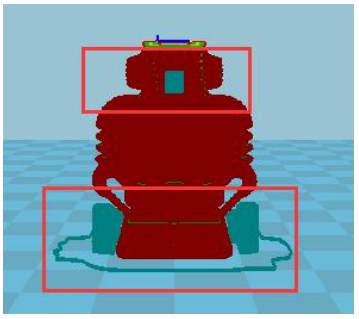
8、Support type



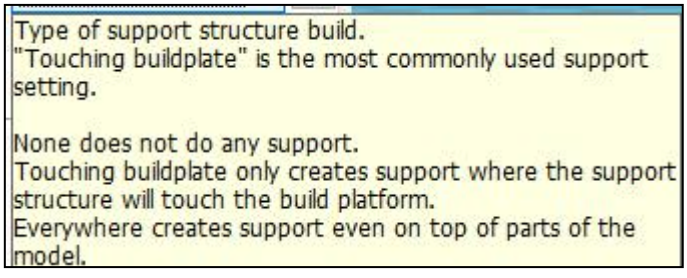

原始图


Touching buildplate

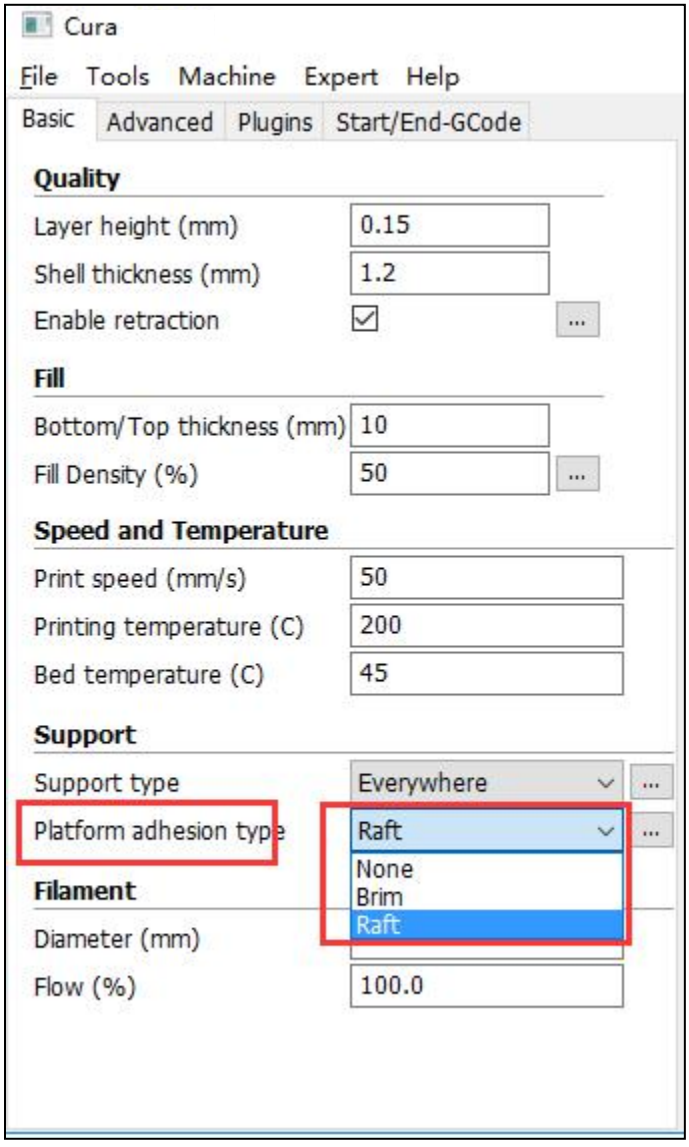

None

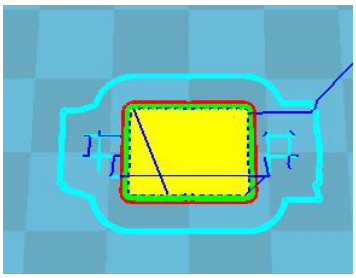

Everywhere

Explanation： The blue is support.

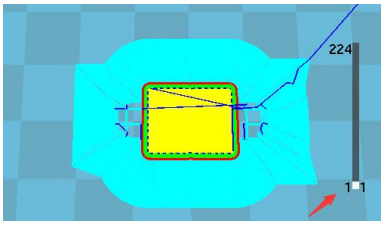


9、Platform adhesion type

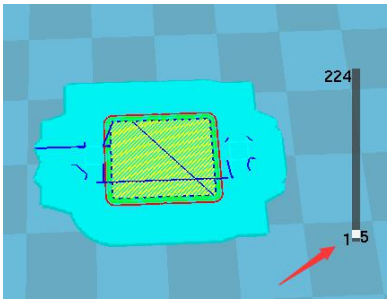




None

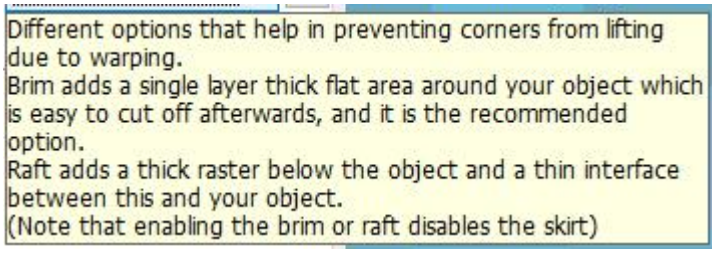


Brim

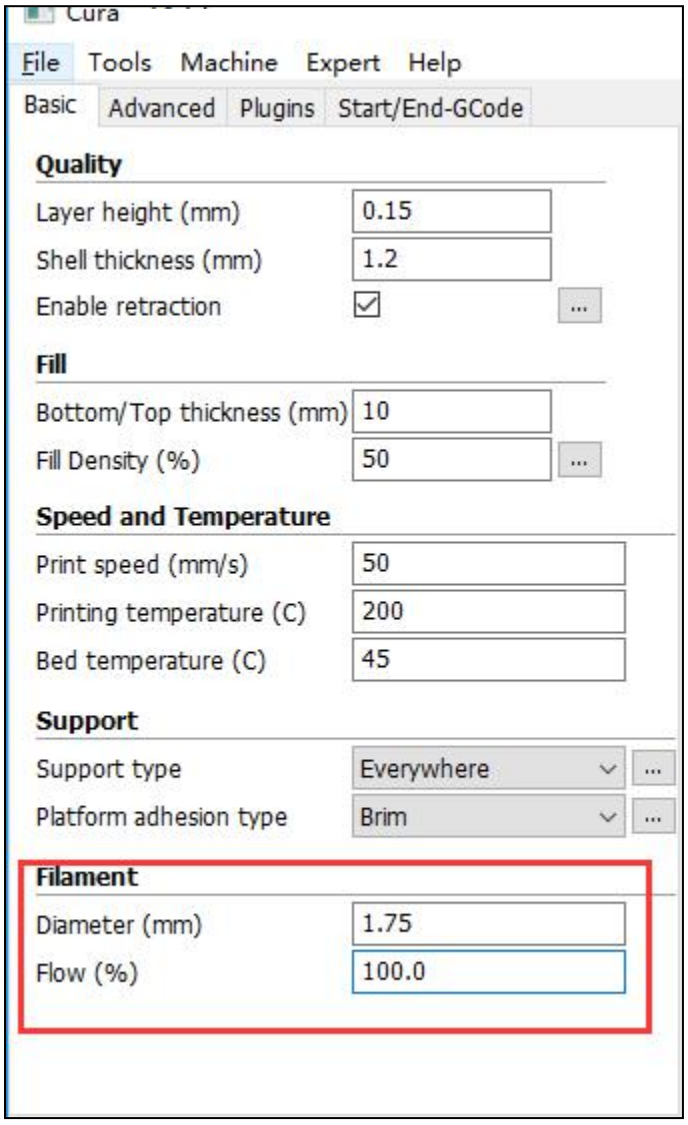


Raft

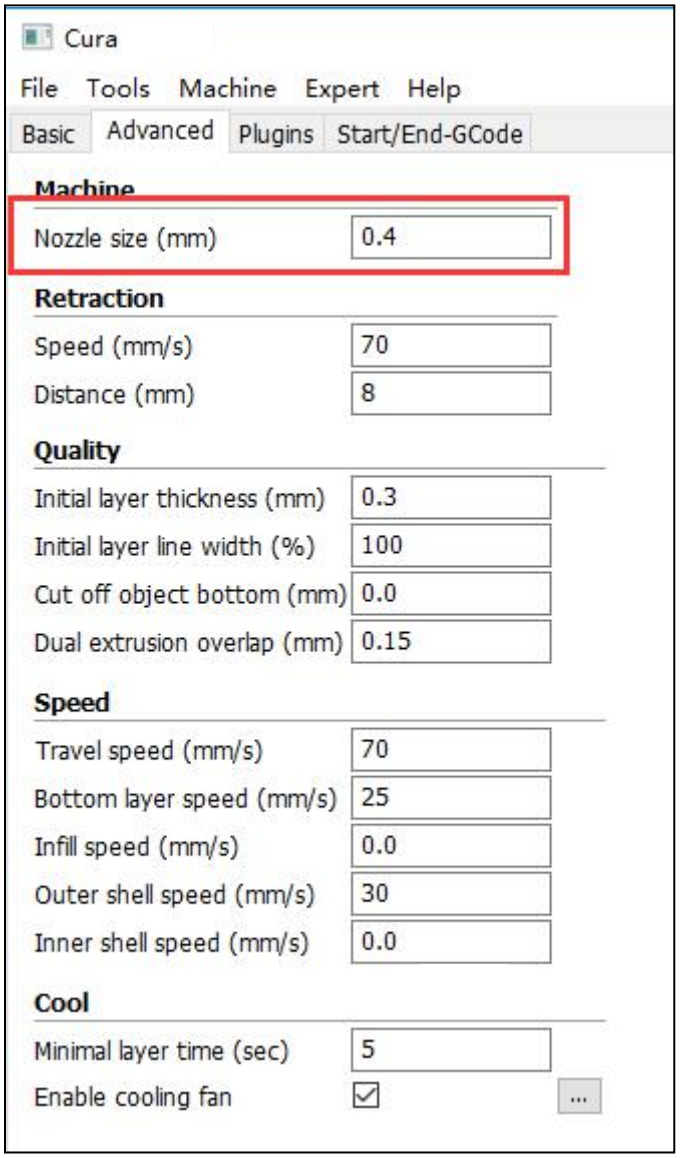
Explanation：



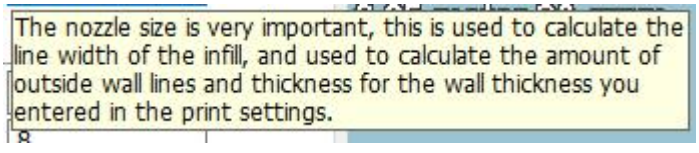
10、Filament


Cura
File Tools Machine Expert Help
Basic Advanced Plugins Start/End-GCode
Quality
Layer height (mm) 0.15
Shell thickness (mm) 1.2
Enable retraction
Fill
Bottom/Top thickness (mm) 10
Fill Density (%) 50
Speed and Temperature
Print speed (mm/s) 50
Printing temperature (C) 200
Bed temperature (C) 45
Support
Support type Everywhere
Platform adhesion type Brim
Filament
Diameter (mm) 1.75
Flow (%) 100.0

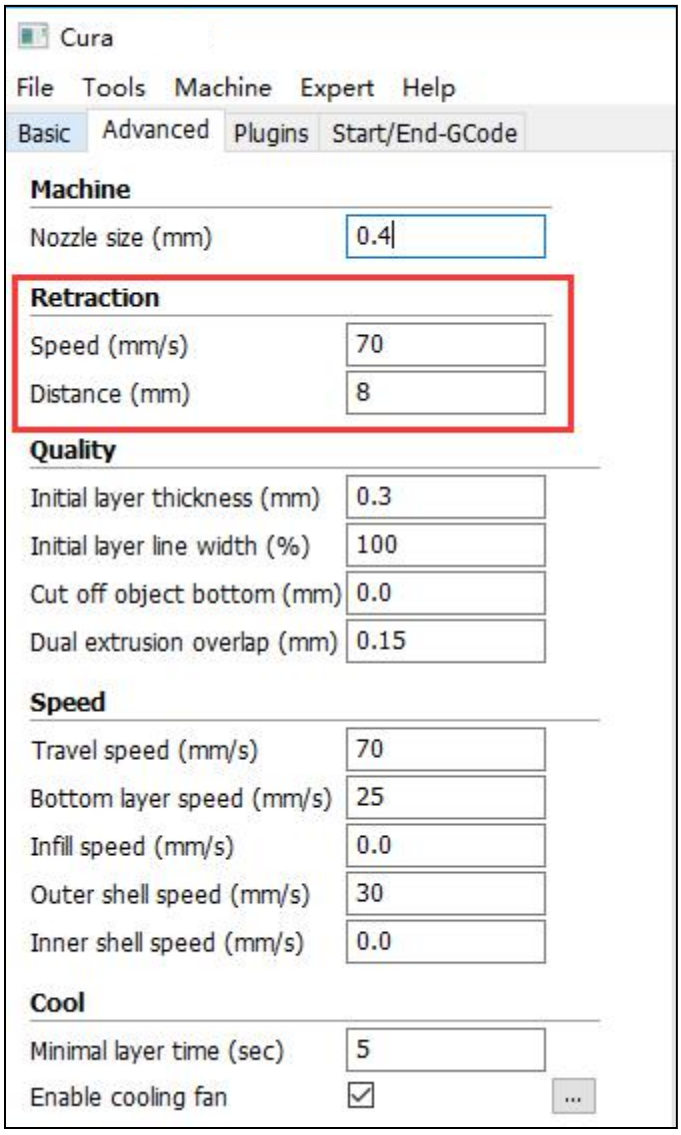
11、nozzle size（mm）



Explanation: The standard is 0.4mm. You can replace other size, 0.2, 0.3, 0.6, 0.8. The smaller size, the higher quality, the longer time. It may be clogged easily with small size.

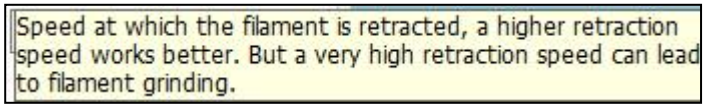


12、Retraction

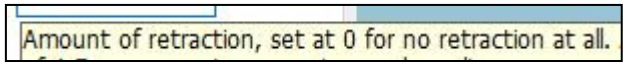


Explanation： Prevent stringing or oozing, speed sets 80, distance sets 8(10).

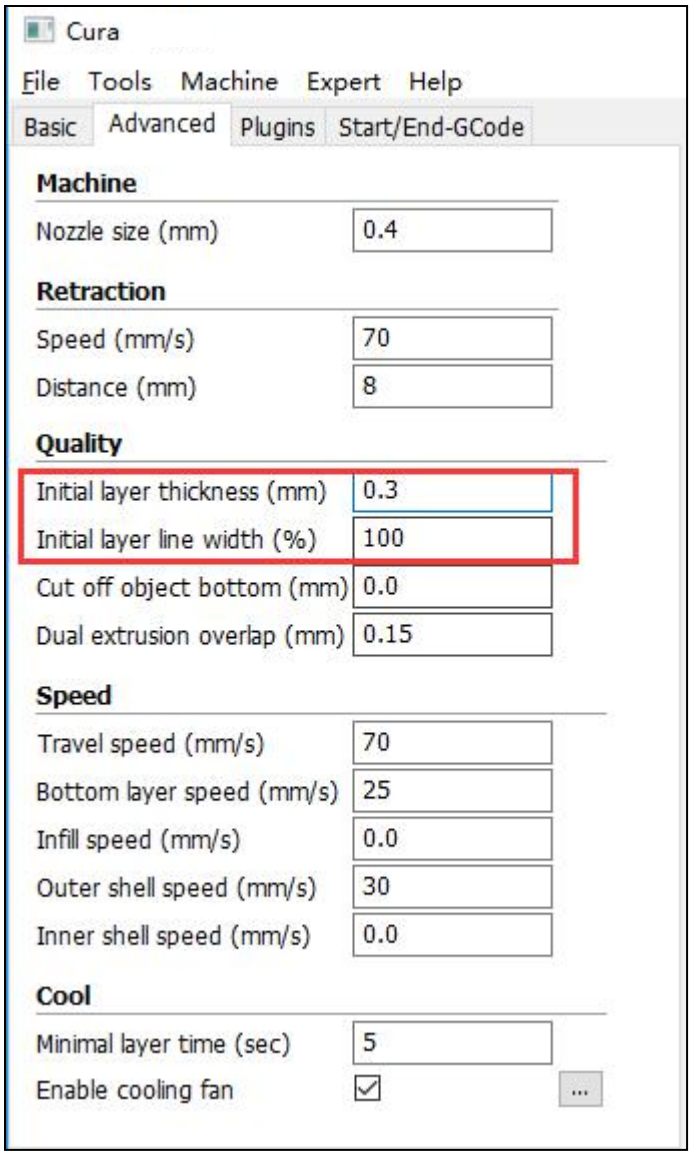
1、speed(mm/s)



2、distance（mm）

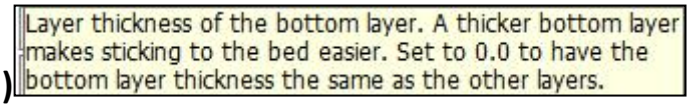


13、Quality

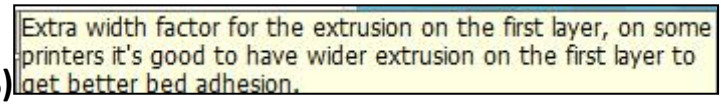


Explanation： Print the thickness of the first layer, initial line width setting 100% is more dense. Generally this as the default.

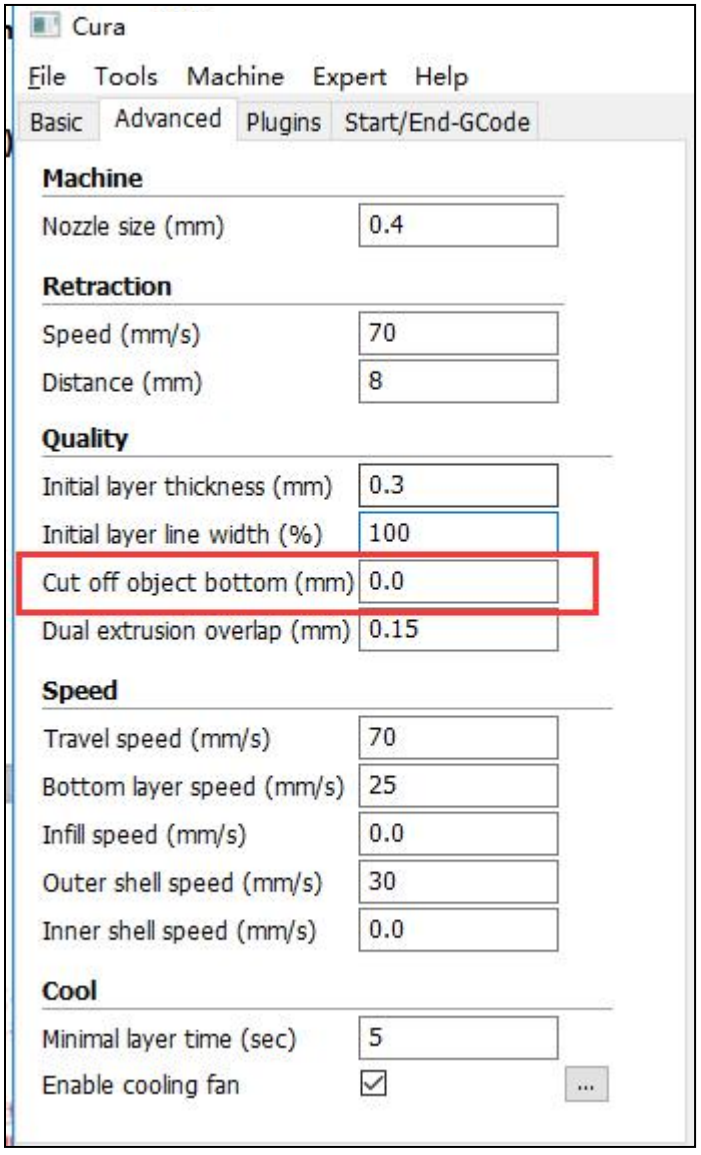
1、Initial layer thickness(mm)

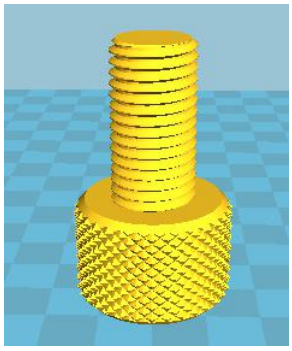


2、Initial layer line width(%)

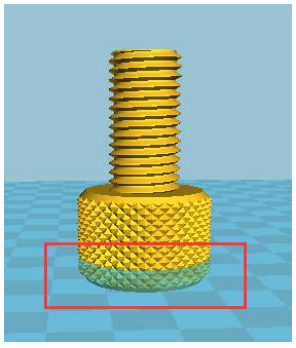


14、Cut off object bottom(mm)

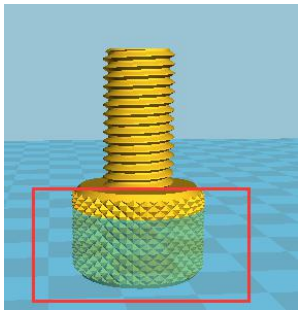




Cut off:0mm

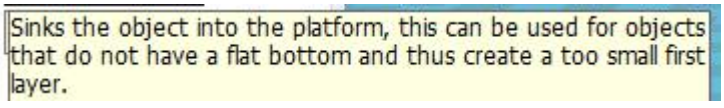


Cut off: 5mm

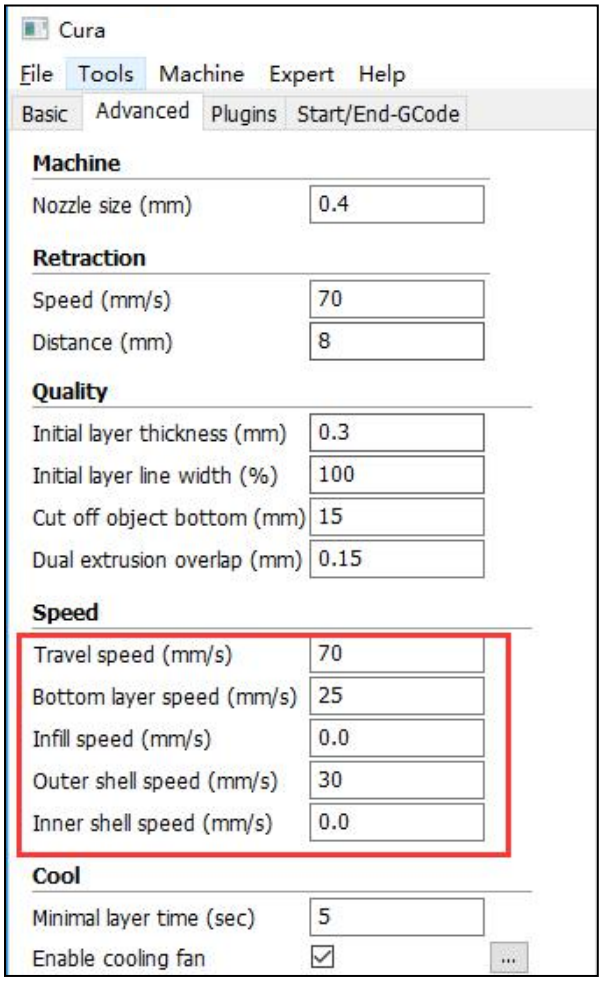


Cut off:15mm

Explanation：



15、Speed



Explanation：

1、travel speed(mm/s):Speed at which travel moves are done

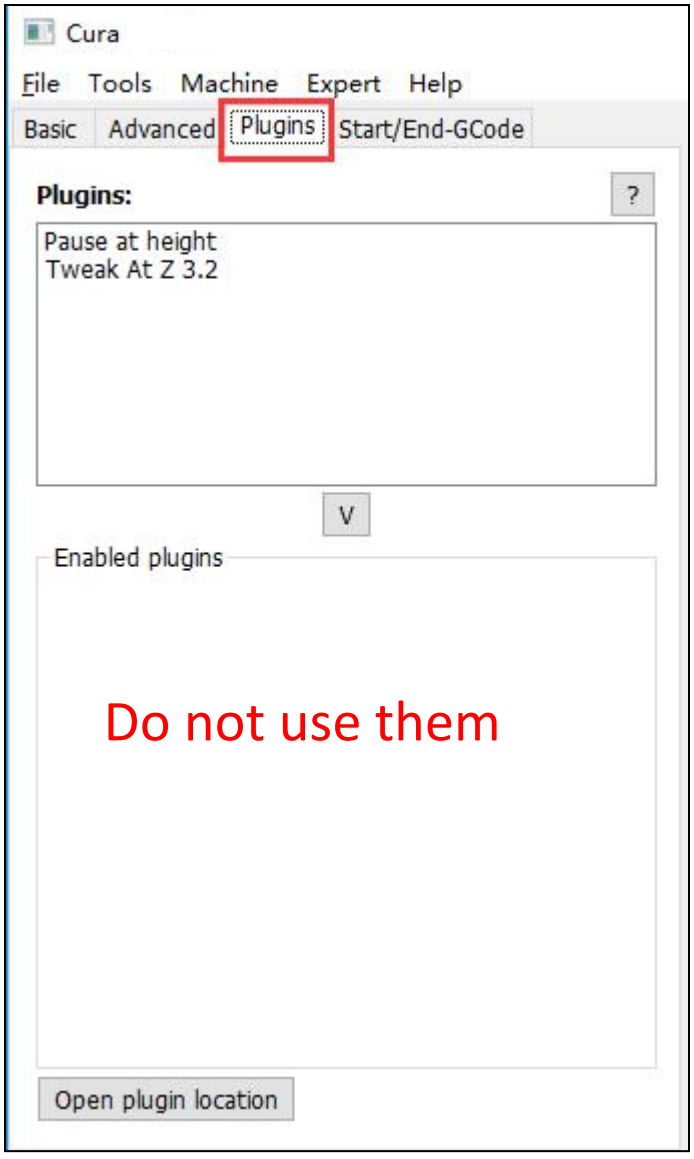
2、Bottom layer speed（mm/s）： Print speed for the bottom layer, you want to print the first layer slower so it sticks better to the printer bed.

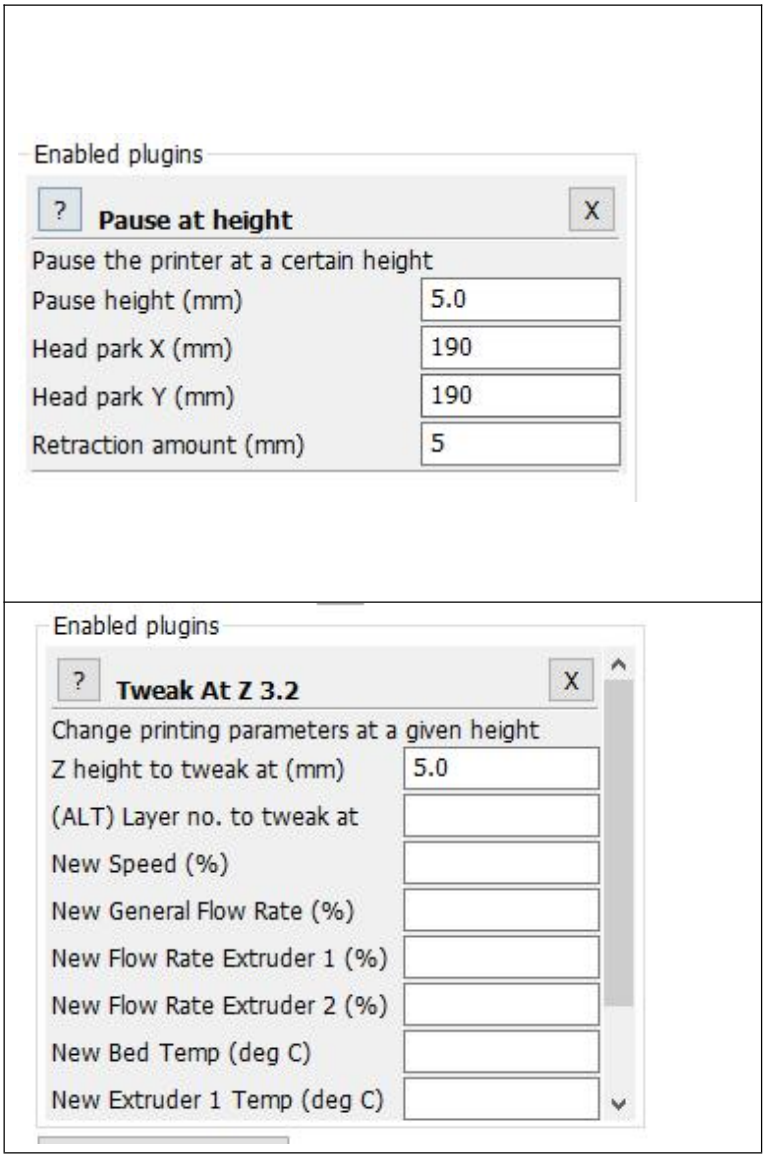
3、Infill speed（mm/s）: Speed at which infill parts are printed. If set to 0 then the print speed is used for the infill. Printing the infill faster can greatly reduce printing time, but this can negatively affect print quality.

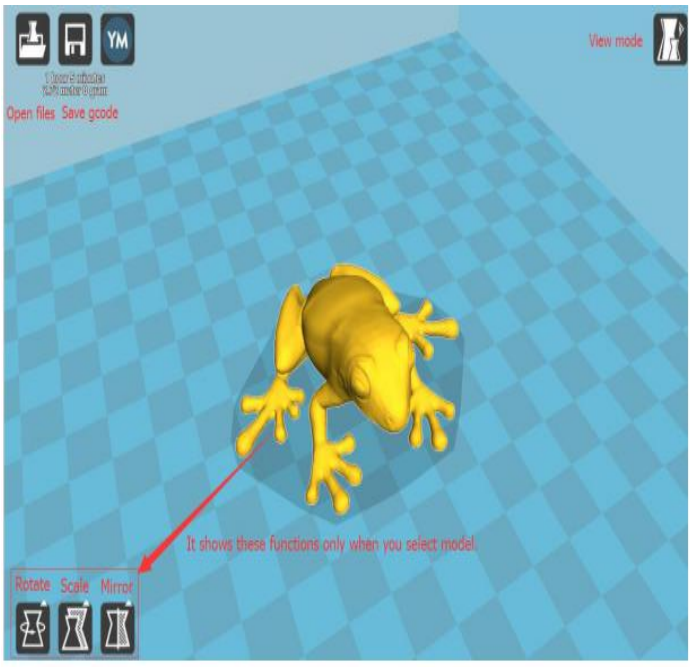
4、Out shell speed (mm/s): Speed at which outer shell is printed. If set to 0 then the print speed is used. Printing the outer shell at a lower speed improves the final skin quality. However, having a large difference between the inner shell speed and the outer shell speed will effect quality in a negative way.

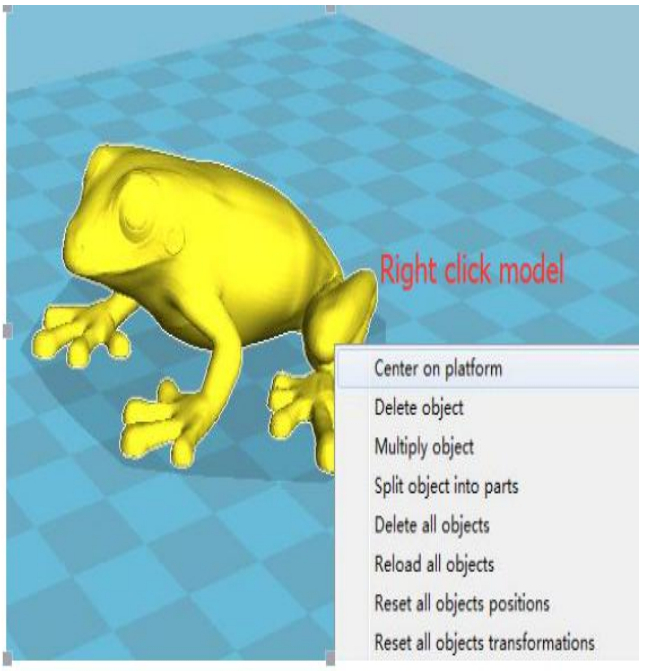
5、Inner shell speed (mm/s): Speed at which inner shells are printed. If set to 0 then the print speed is used. Printing the inner shell faster then the outer shell will reduce printing time. It is good to set this somewhere in between the outer shell speed and the infill/printing speed.

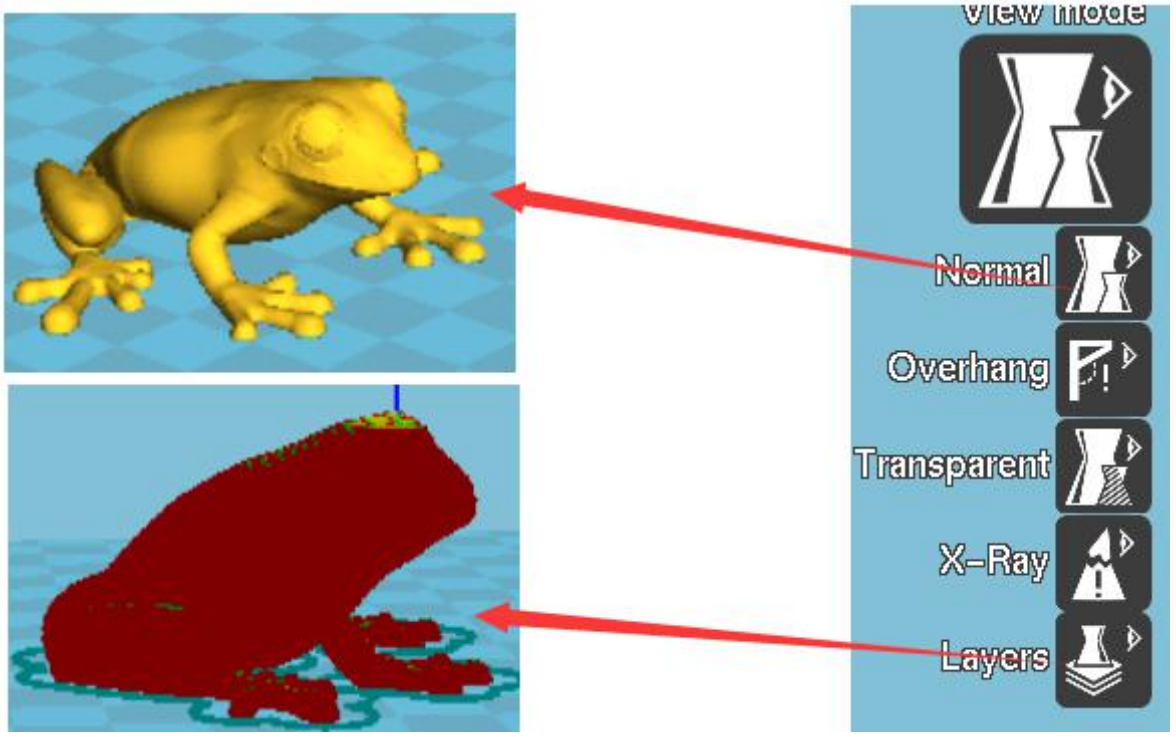
16、Plugins


Cura
File Tools Machine Expert Help
Basic Advanced Plugins Start/End-GCode
Plugins:
?
Pause at height
Tweak At Z 3.2
V
Enabled plugins
Do not use them
Open plugin location

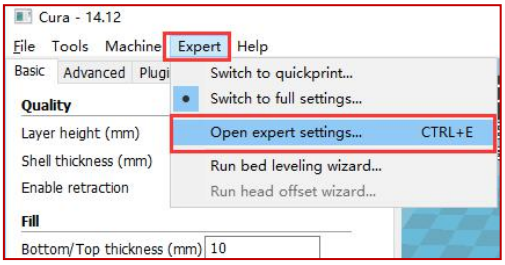

Enabled plugins
? Pause at height X
Pause the printer at a certain height
Pause height (mm) 5.0
Head park X (mm) 190
Head park Y (mm) 190
Retraction amount (mm) 5
Enabled plugins
? Tweak At Z 3.2 X
Change printing parameters at a given height
Z height to tweak at (mm) 5.0
(ALT) Layer no. to tweak at
New Speed (%)
New General Flow Rate (%)
New Flow Rate Extruder 1 (%)
New Flow Rate Extruder 2 (%)
New Bed Temp (deg C)
New Extruder 1 Temp (deg C)


View mode
Open files Save gcode
It shows these functions only when you select model.
Rotate Scale Mirror

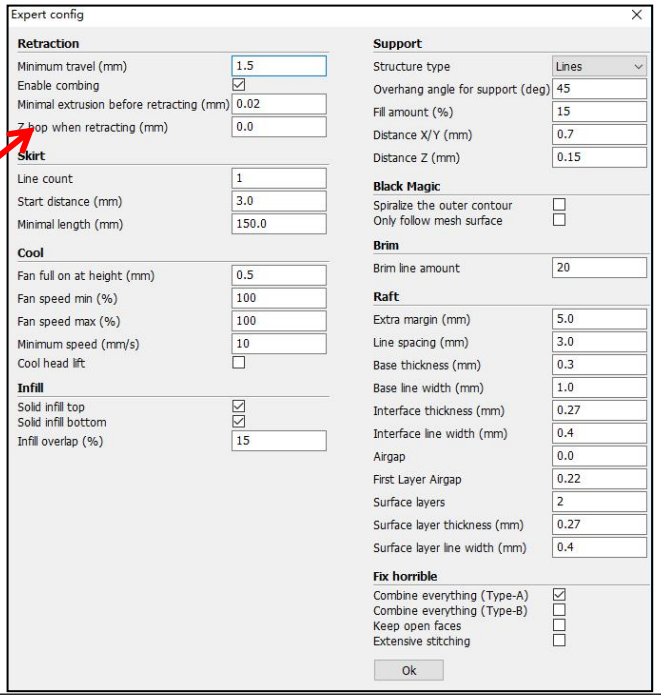

Right click model
Center on platform
Delete object
Multiply object
Split object into parts
Delete all objects
Reload all objects
Reset all objects positions
Reset all objects transformations

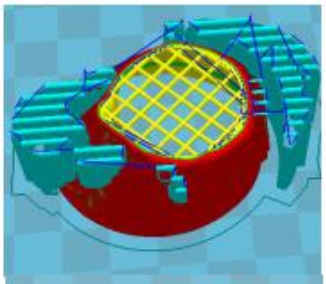

View mode
Normal
Overhang
Transparent
X-Ray
Layers

Expert settings



When a retraction is done, the head is lifted this amount to travel over the print. A value of 0.075mm works well. This feature has a lot of positive effect delta towers.



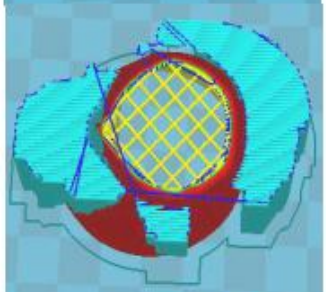


type:Line

Degree:60

Fill amount:15

Distance X/Y :0.7

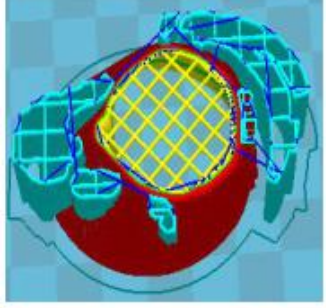


type:Line

Degree:60

Fill amount:50

Distance X/Y :3

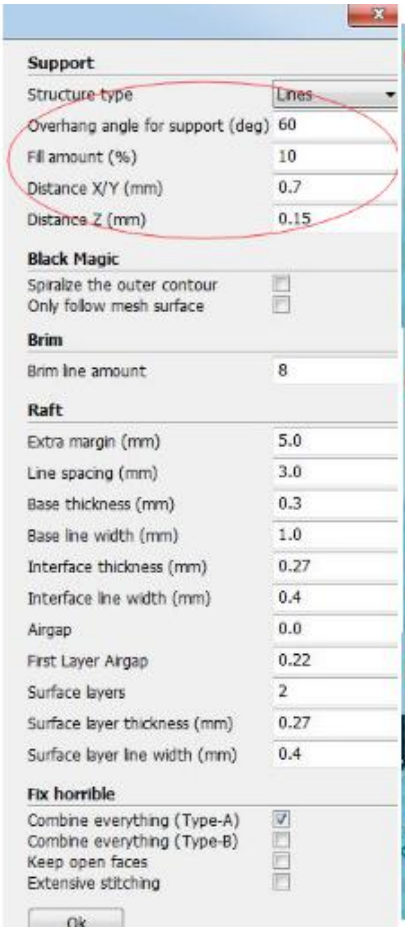


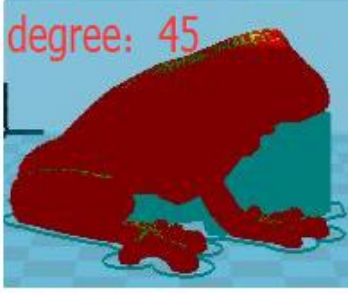
type:Gird

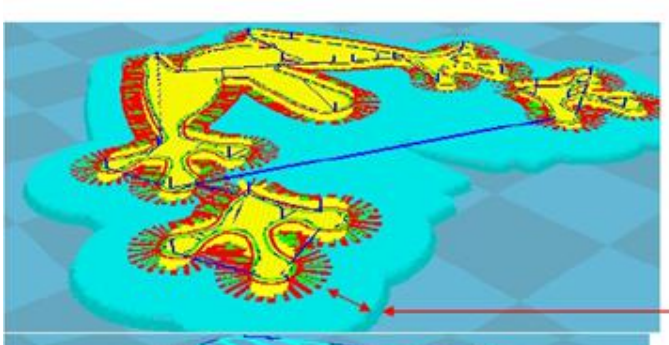
Degree:60

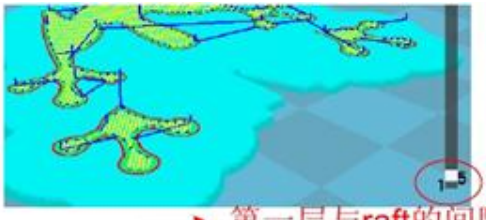
Fill amount:15

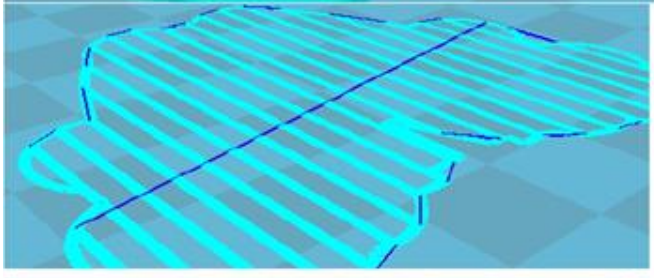
Distance X/Y :0.7



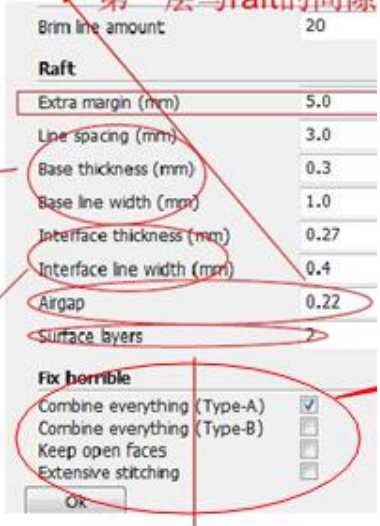


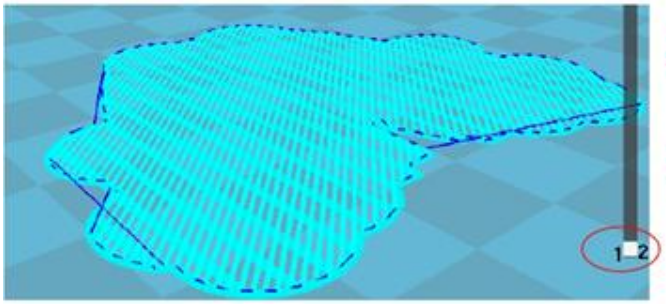






base line





Interface line

surface