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.
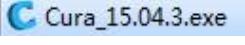- Do not deliberately pull, twist the machine over the distribution line, to avoid causing an open or short circuit。
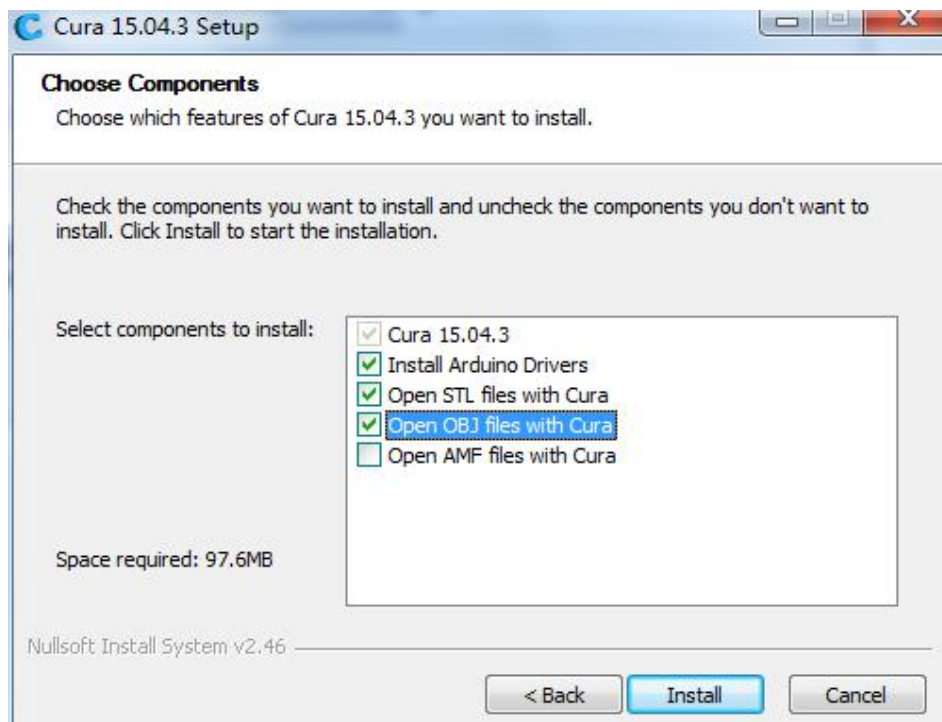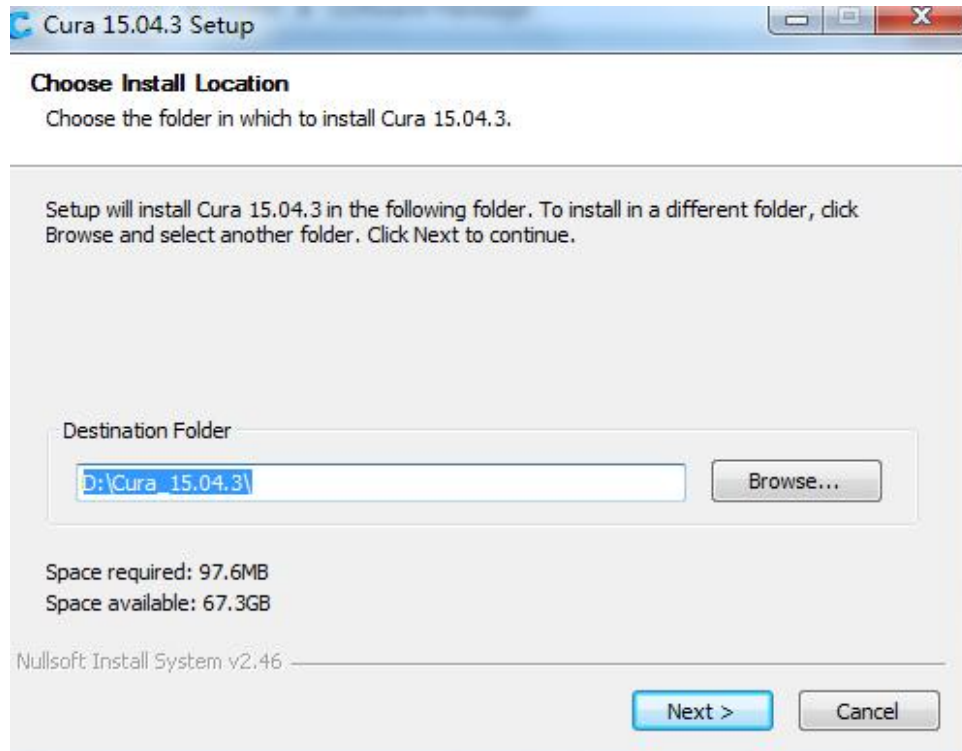
5. Note that in the printing process

- 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.
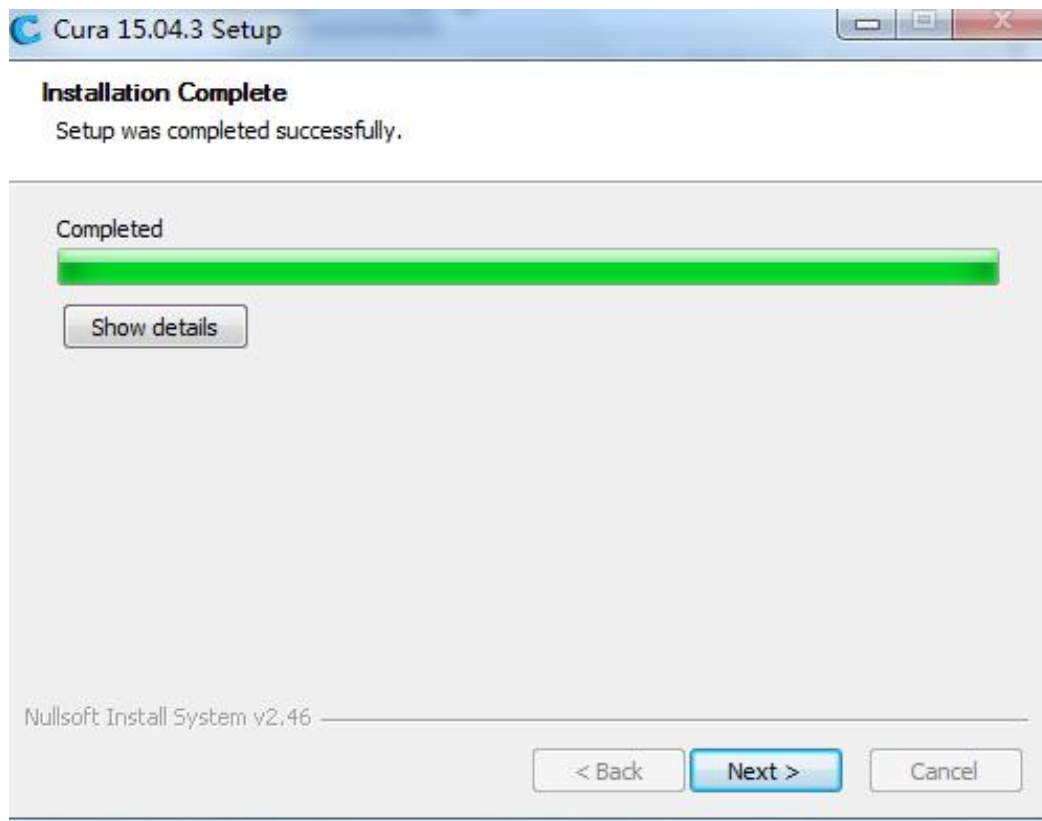
II. Cura Installation and Setup
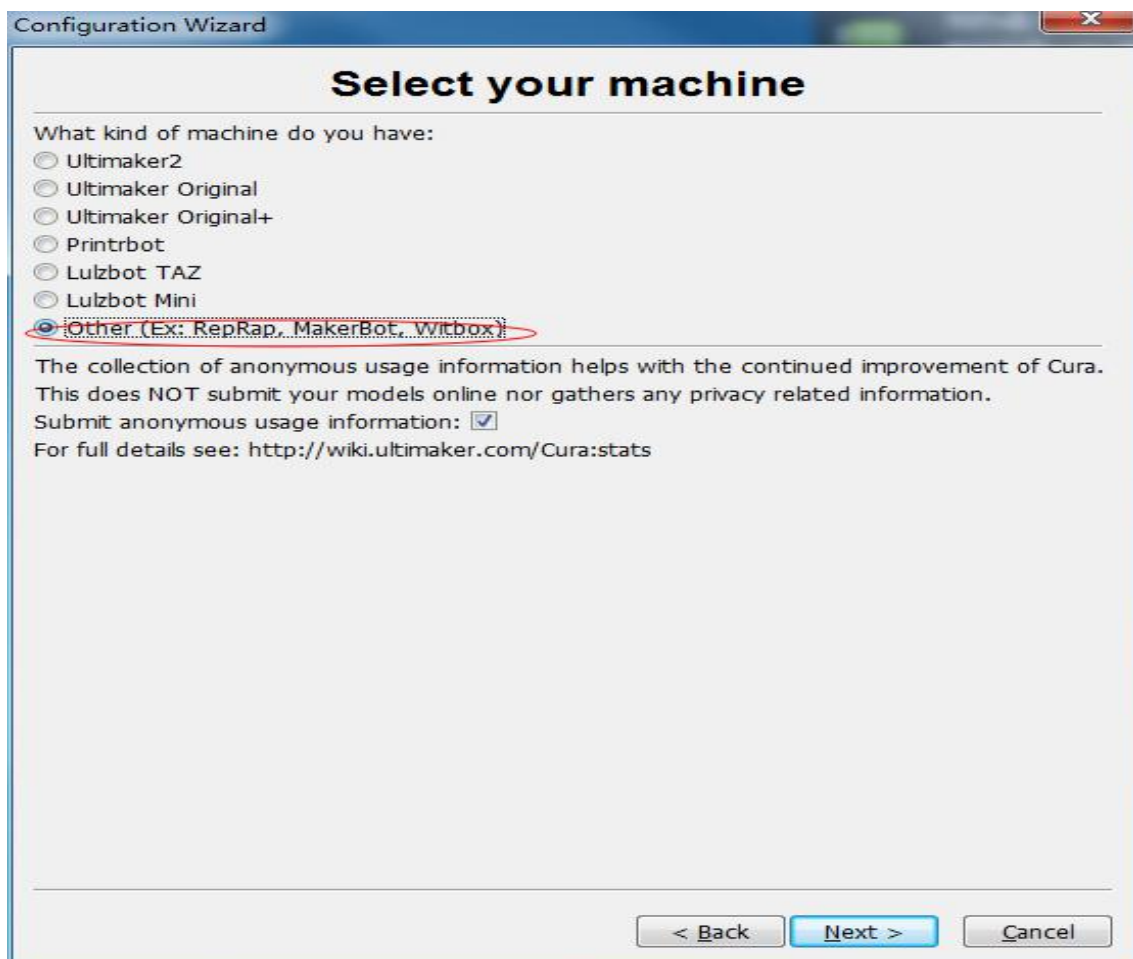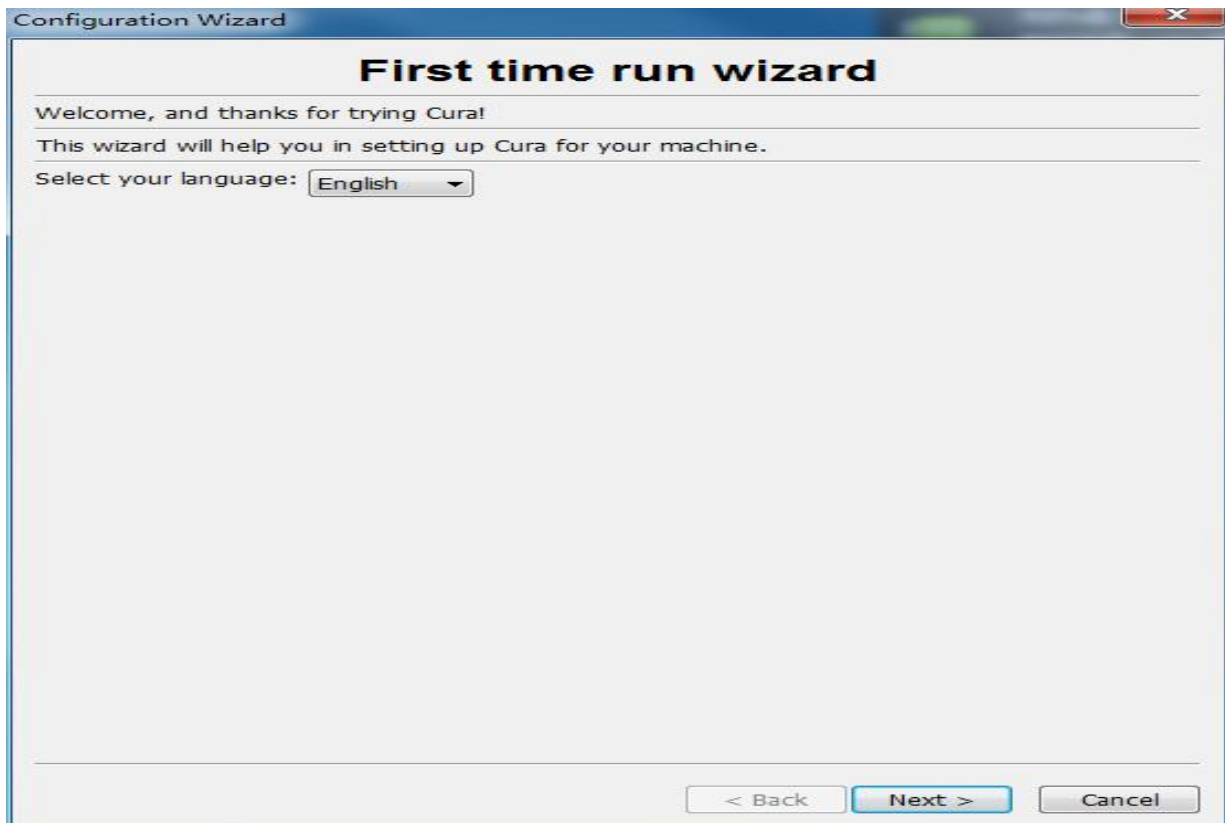
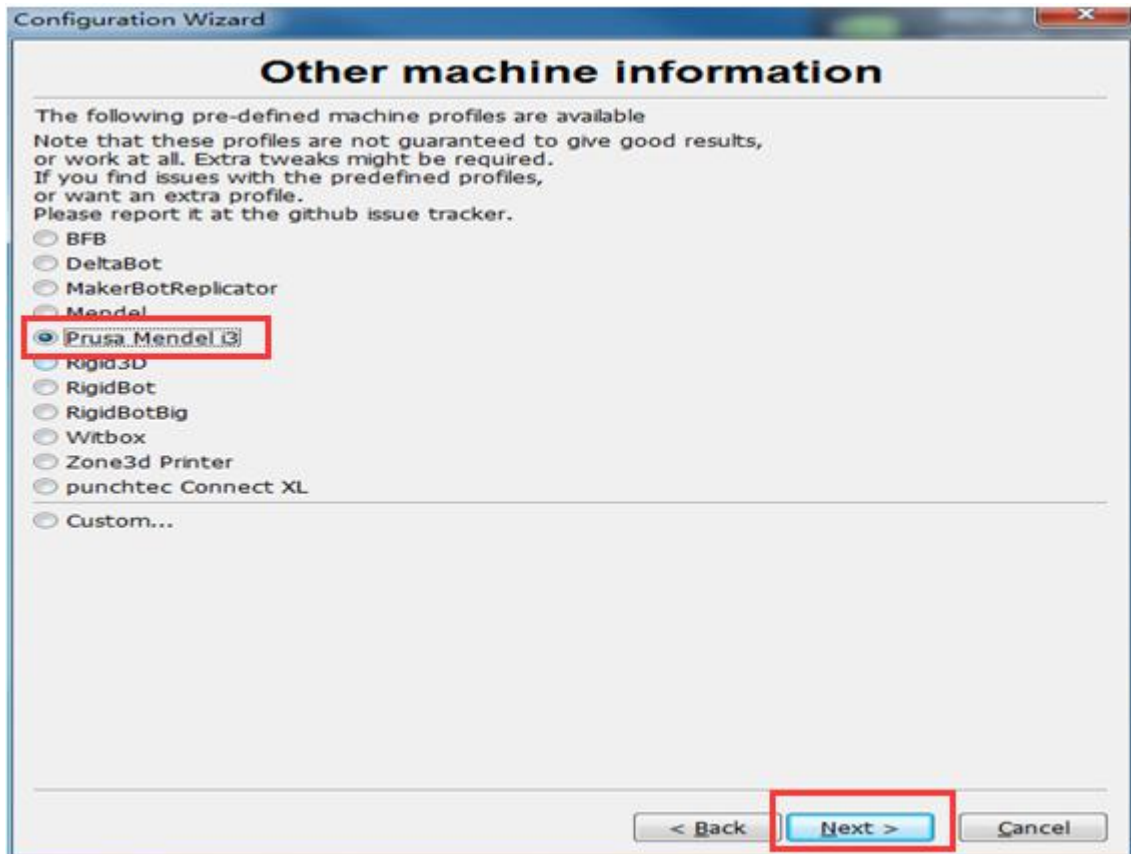## 1. Slice software installation

a. Install the software

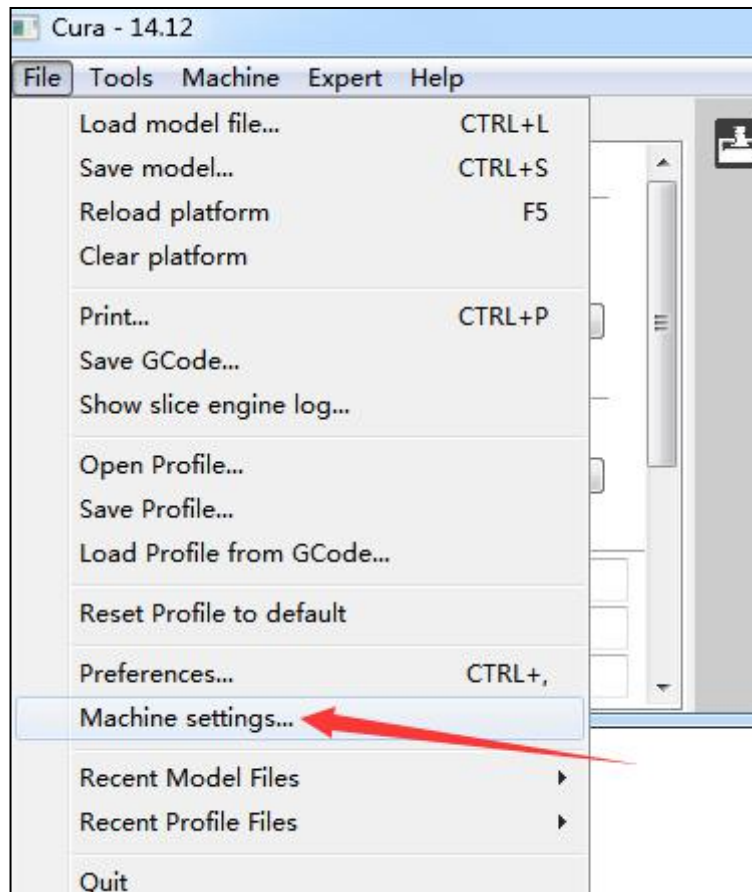Click Setup ![C Cura_15.04.3.exe], pop-up installation window, always click Next to complete the installation.

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]

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

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 - 14.12

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

Cr-7	Cr-8	Cr-2020	Cr-5	Cr-10	Ender-1	Ender-2	Ender-4

Machine settings

E-Steps per 1mm filament	0
Maximum width (mm)	220
Maximum depth (mm)	220
Maximum height (mm)	300
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-4

2. Software parameter setting

Open **C Cura_15.04.3.exe** , you can modify all of the printing process control parameters display screen.
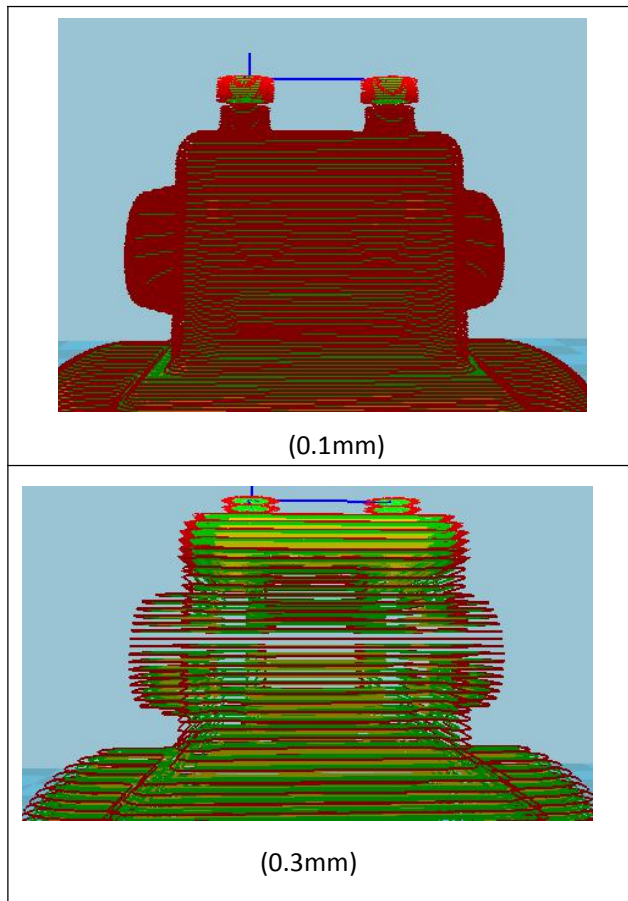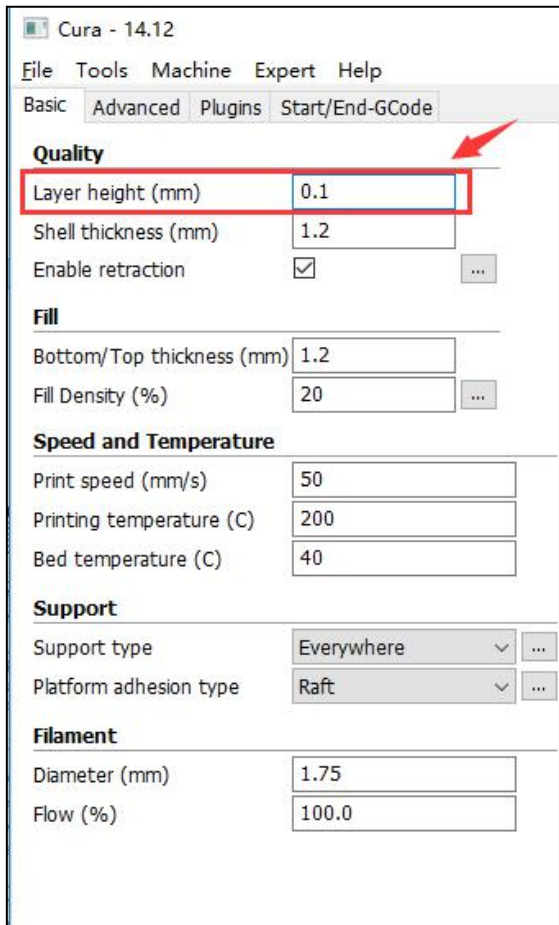
a. Recommended parameter settings:

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

Cura - 14.12

File Tools Machine Expert Help

Basic Advanced Plugins Start/End-GCode

Quality

Layer height (mm)	0.1
Shell thickness (mm)	1.2
Enable retraction	☑

Fill

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

Speed and Temperature

Print speed (mm/s)	50
Printing temperature (C)	200
Bed temperature (C)	40

Support

Support type	Everywhere
Platform adhesion type	Raft

Filament

Diameter (mm)	1.75
Flow (%)	100.0

(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

Cura - 14.12

Basic | Advanced | Plugins | Start/End-GCode

Quality

Layer height (mm)	0.15
Shell thickness (mm)	1.2
Enable retraction	☑

Fill

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

Speed and Temperature

Print speed (mm/s)	50
Printing temperature (C)	200
Bed temperature (C)	40

Support

Support type	Everywhere
Platform adhesion type	Raft

Filament

Diameter (mm)	1.75
Flow (%)	100.0

0.8mm(If the nozzle is 0.4mm)
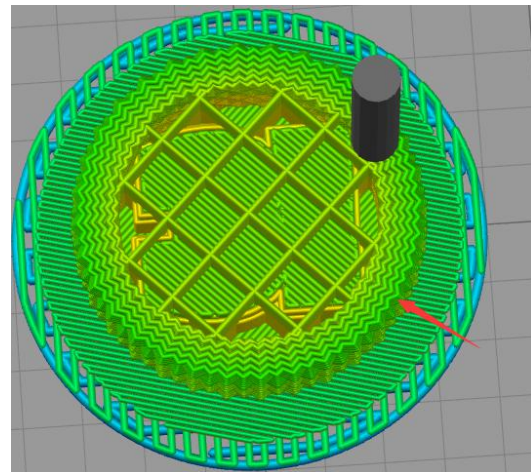
2.0mm(If the nozzle is 0.4mm)

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
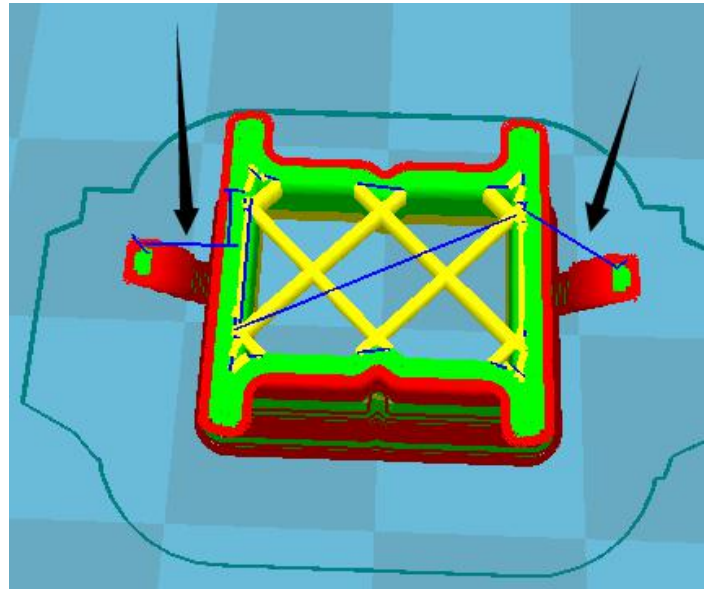
Explanation：

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

Cura - 14.12

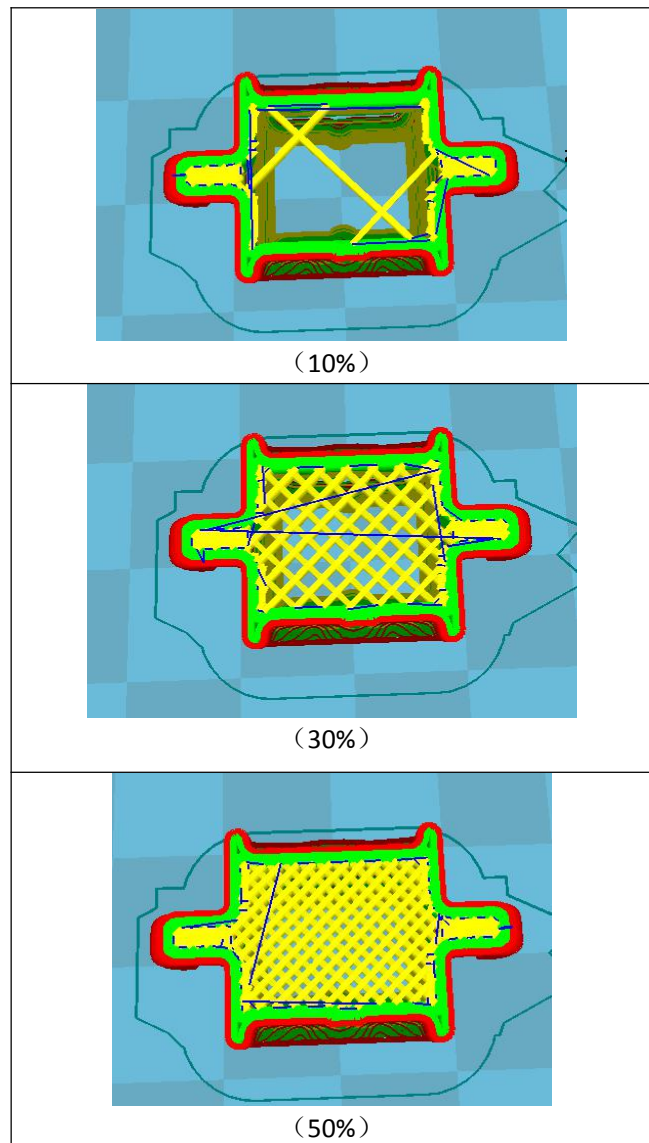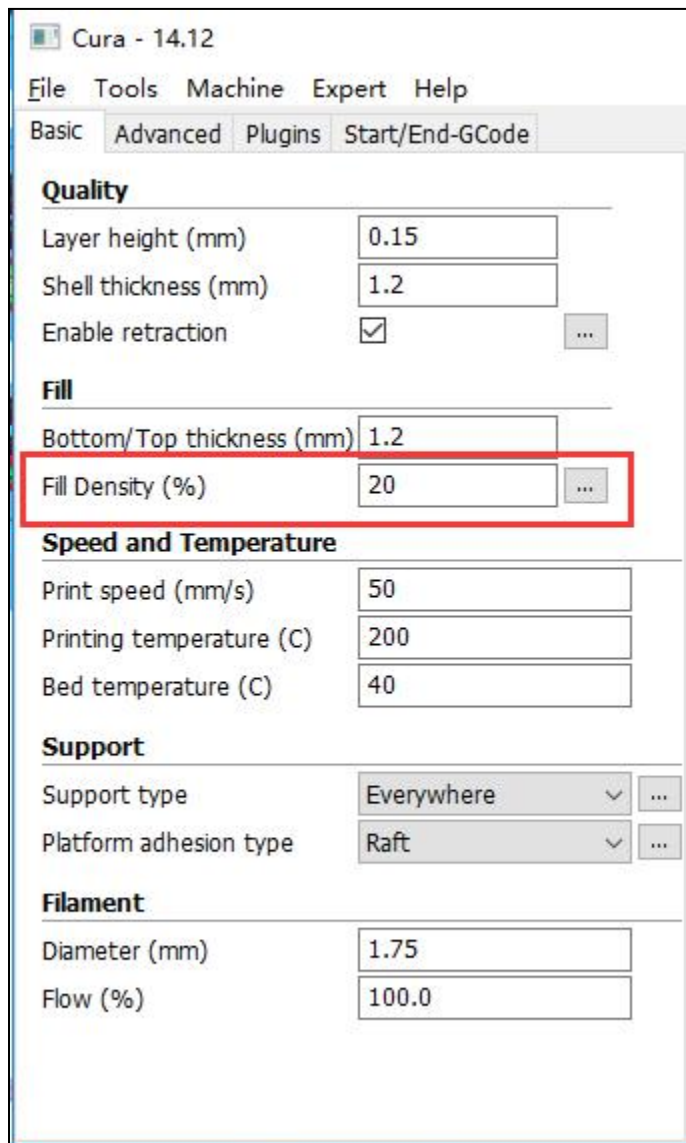File Tools Machine Expert Help

Basic Advanced Plugins Start/End-GCode

Quality

Layer height (mm)	0.15
Shell thickness (mm)	1.2
Enable retraction	☑ ...

Fill

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

Speed and Temperature

Print speed (mm/s)	50
Printing temperature (C)	200
Bed temperature (C)	40

Support

Support type	Everywhere ...
Platform adhesion type	Raft ...

Filament

Diameter (mm)	1.75
Flow (%)	100.0

Under the same layers
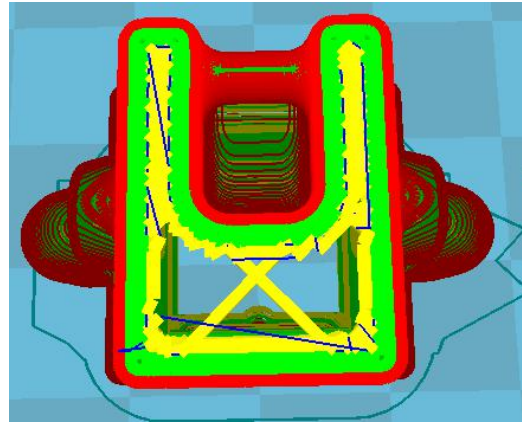
（0.6mm）
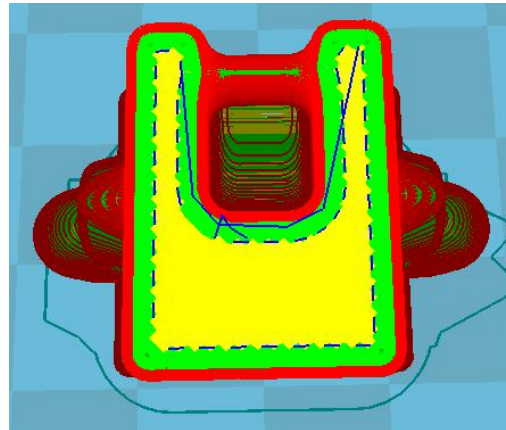
（1.2mm）

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%）
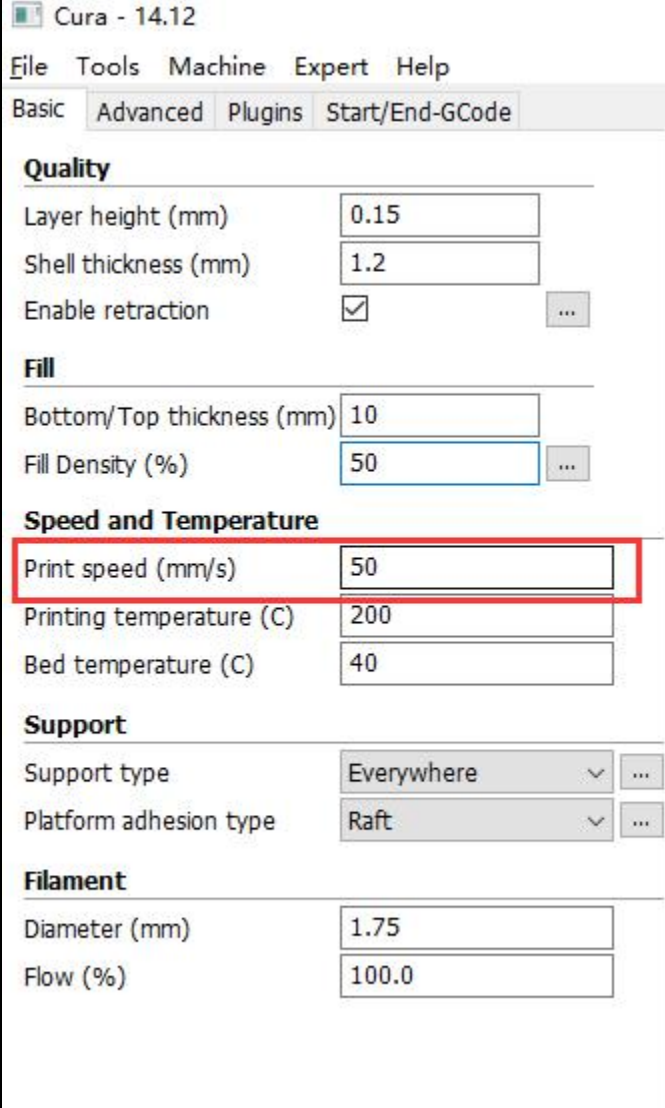
Explanation：

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）
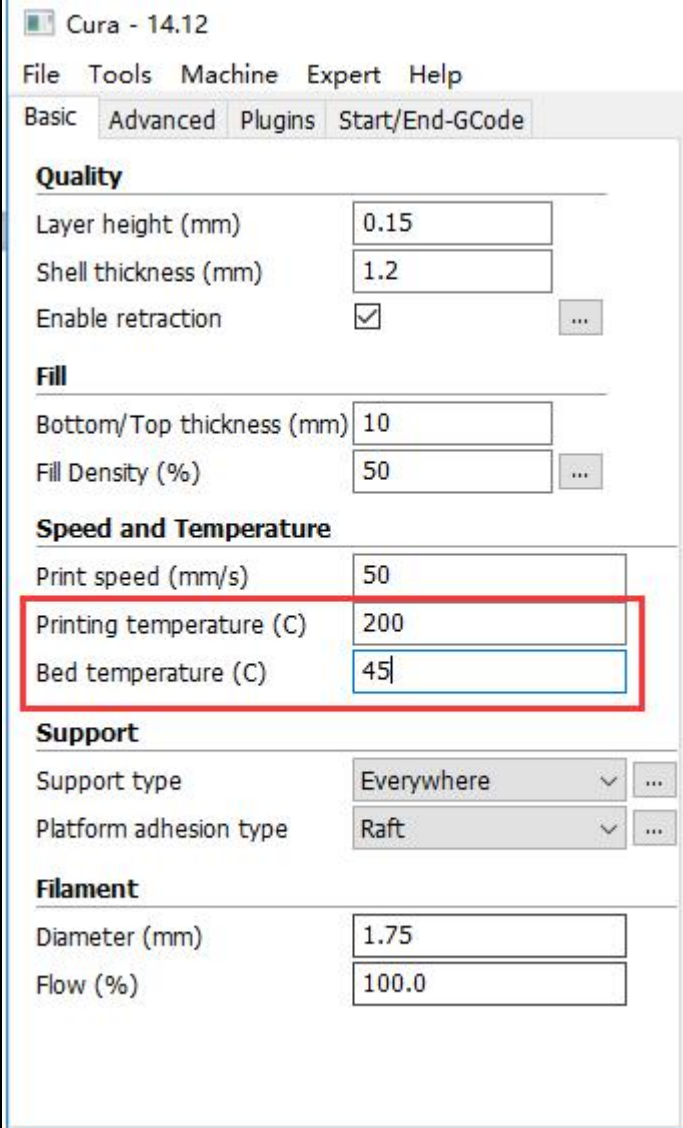
```
Cura - 14.12
File   Tools   Machine   Expert   Help
Basic   Advanced   Plugins   Start/End-GCode

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)            40

Support
Support type              Everywhere    ⌄   ...
Platform adhesion type    Raft          ⌄   ...

Filament
Diameter (mm)                  1.75
Flow (%)                       100.0
```

Explanation：Speed at which printing happens .

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

7、 Print temperature(℃)

Cura - 14.12

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

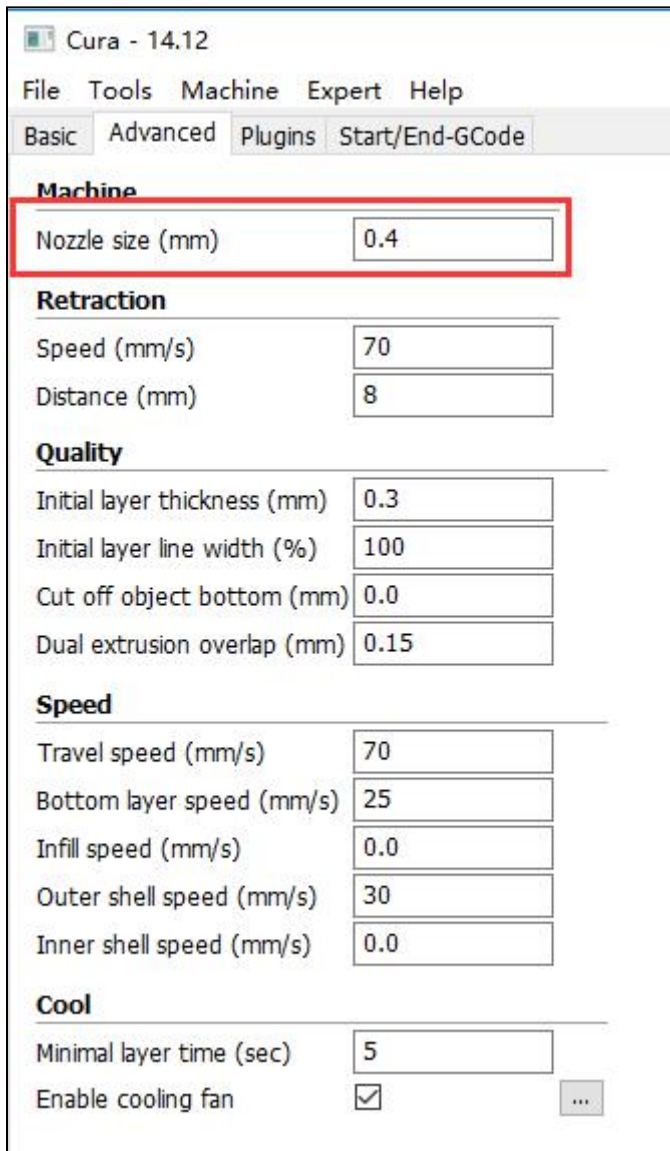Support type   Everywhere   ∨   ...

Platform adhesion type   Raft   ∨   ...

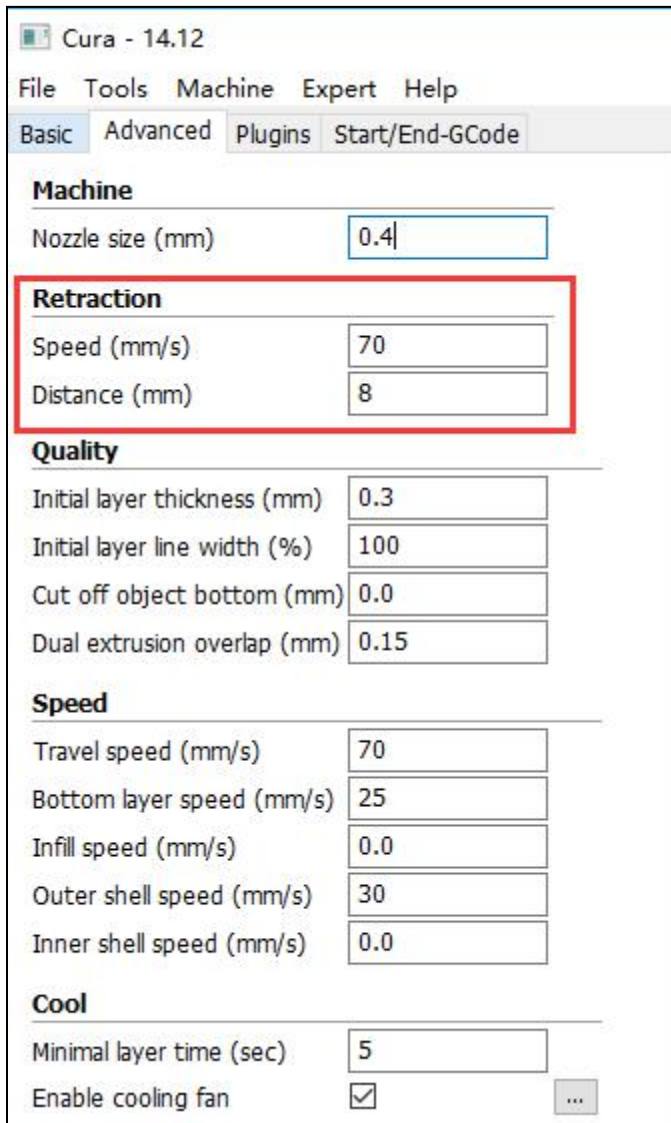**Filament**

Diameter (mm)   1.75

Flow (%)   100.0

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
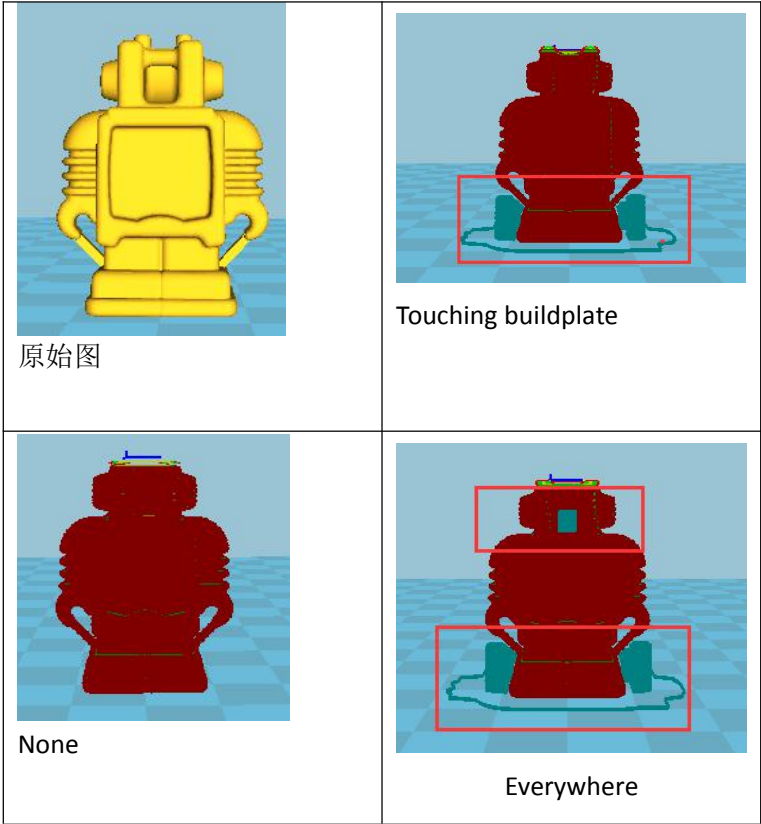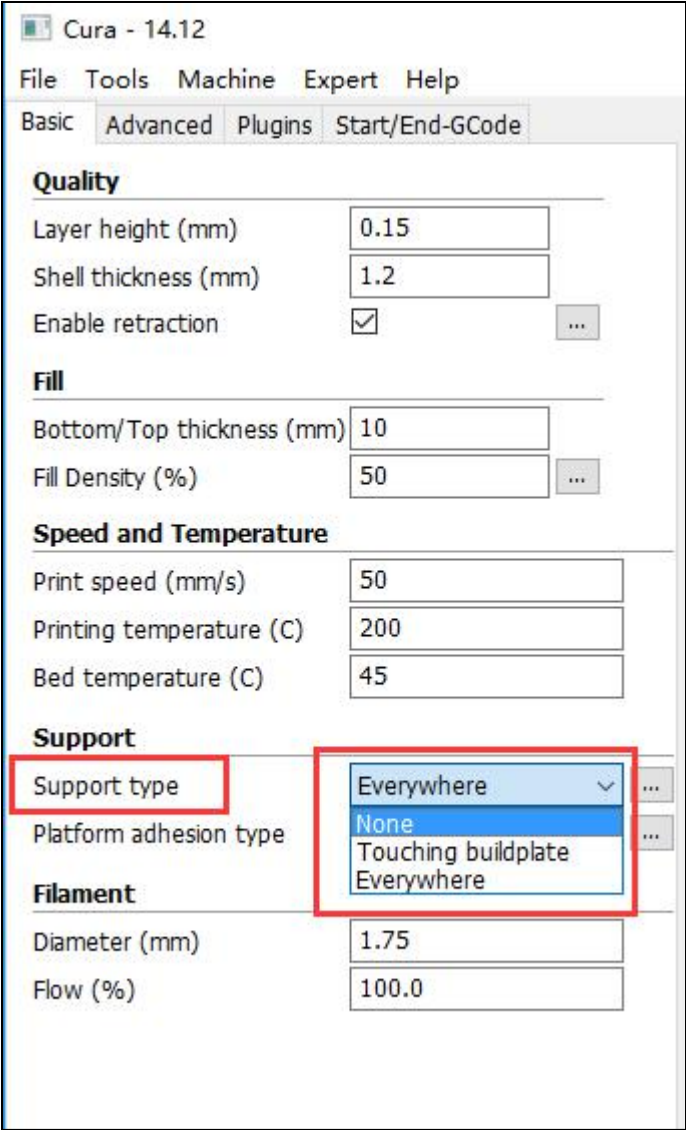
Explanation：The blue is support.

Type of support structure build.
"Touching buildplate" is the most commonly used support setting.

None does not do any support.
Touching buildplate only creates support where the support structure will touch the build platform.
Everywhere creates support even on top of parts of the model.

9、Platform adhesion type

Cura - 14.12

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	Raft ∨ ...
	None
	Brim
	Raft

Filament

Diameter (mm)	
Flow (%)	100.0
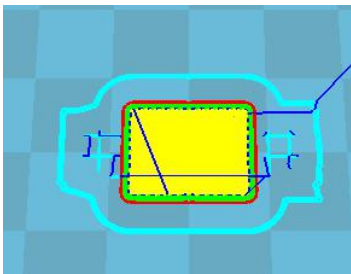
None
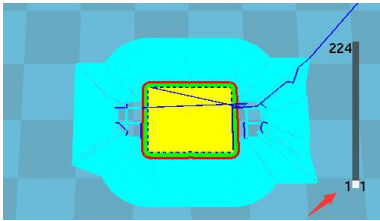
Brim
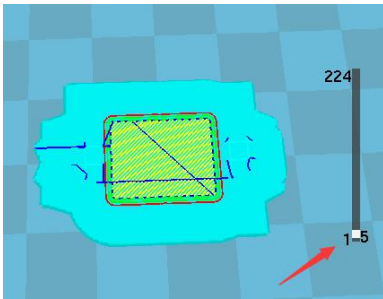
Raft

Explanation：

Different options that help in preventing corners from lifting due to warping.
Brim adds a single layer thick flat area around your object which is easy to cut off afterwards, and it is the recommended option.
Raft adds a thick raster below the object and a thin interface between this and your object.
(Note that enabling the brim or raft disables the skirt)

10、Filament

Cura - 14.12

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.

The nozzle size is very important, this is used to calculate the line width of the infill, and used to calculate the amount of outside wall lines and thickness for the wall thickness you entered in the print settings.

12、Retraction

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

1、speed(mm/s)

Speed at which the filament is retracted, a higher retraction speed works better. But a very high retraction speed can lead to filament grinding.

2、distance（mm）

Amount of retraction, set at 0 for no retraction at all.

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)

Layer thickness of the bottom layer. A thicker bottom layer makes sticking to the bed easier. Set to 0.0 to have the bottom layer thickness the same as the other layers.

2、Initial layer line width(%)

Extra width factor for the extrusion on the first layer, on some printers it's good to have wider extrusion on the first layer to get better bed adhesion.

14、Cut off object bottom(mm)

Cura - 14.12

File Tools Machine Expert Help

Basic Advanced Plugins Start/End-GCode

Machine

Nozzle size (mm) 0.4

Retraction

Speed (mm/s) 70
Distance (mm) 8

Quality

Initial layer thickness (mm) 0.3
Initial layer line width (%) 100
Cut off object bottom (mm) 0.0
Dual extrusion overlap (mm) 0.15

Speed

Travel speed (mm/s) 70
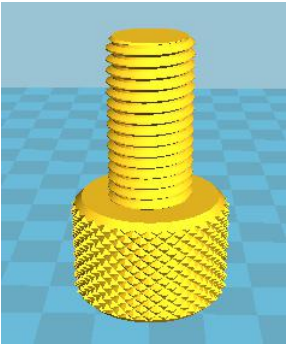Bottom layer speed (mm/s) 25
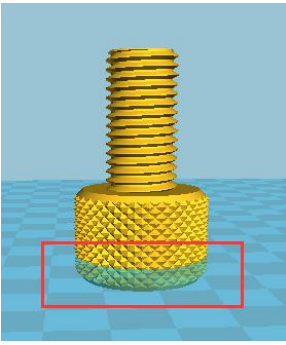Infill speed (mm/s) 0.0
Outer shell speed (mm/s) 30
Inner shell speed (mm/s) 0.0

Cool

Minimal layer time (sec) 5
Enable cooling fan ☑ ...
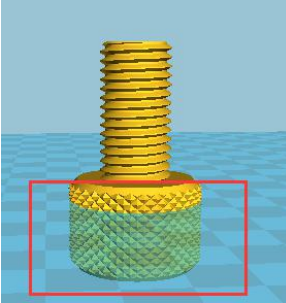
Cut off:0mm

Cut off: 5mm
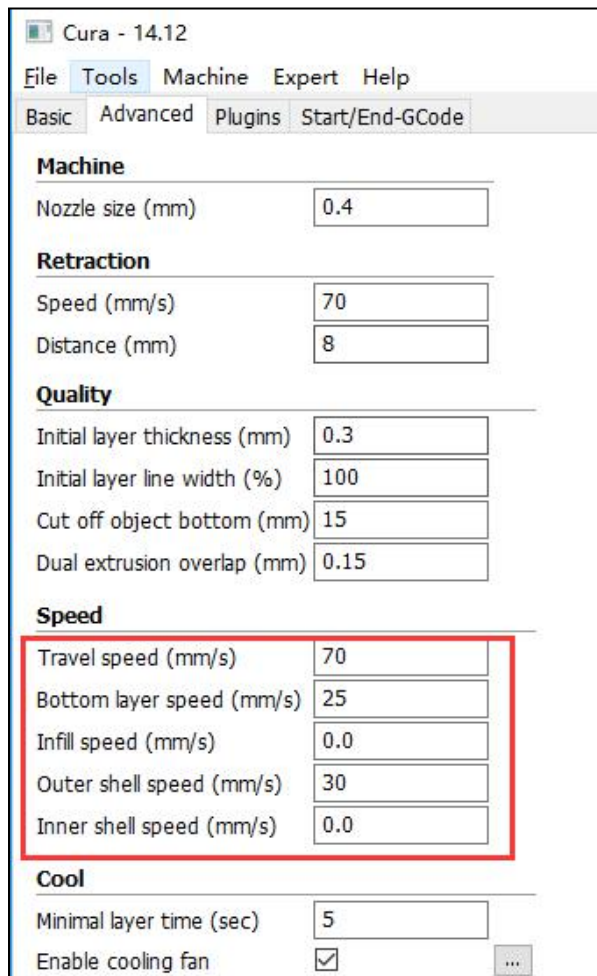
Cut off:15mm

Explanation： Sinks the object into the platform, this can be used for objects that do not have a flat bottom and thus create a too small first layer.

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 - 14.12

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 (%)
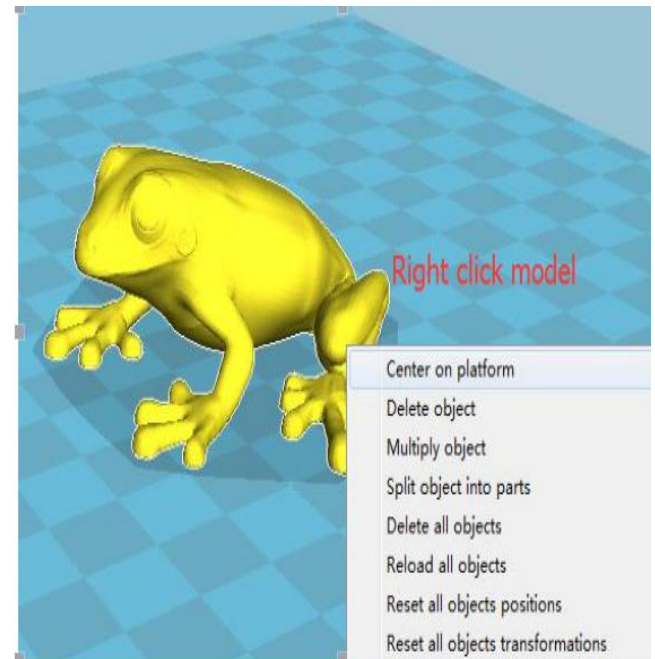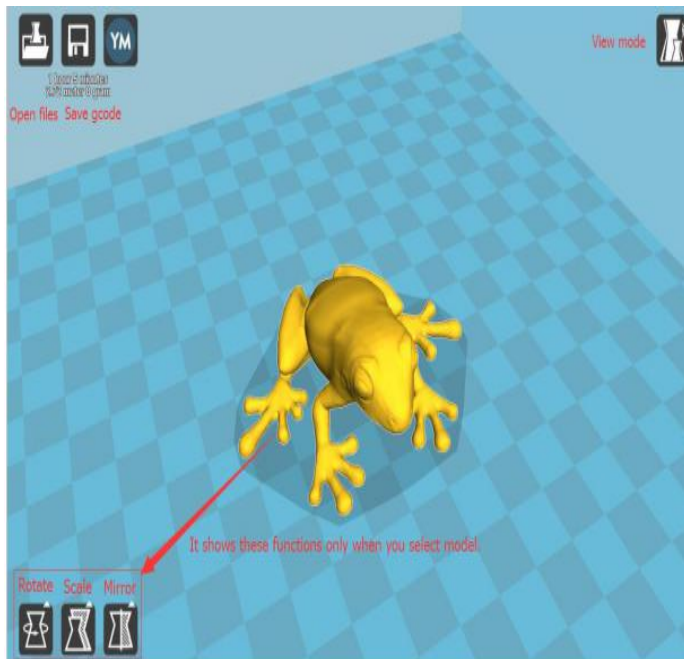
New General Flow Rate (%)

New Flow Rate Extruder 1 (%)

New Flow Rate Extruder 2 (%)

New Bed Temp (deg C)

New Extruder 1 Temp (deg C)
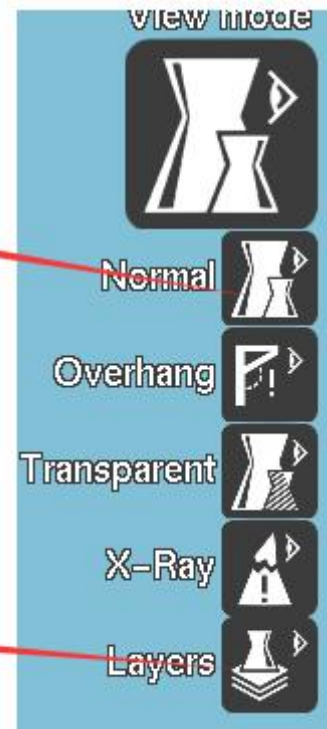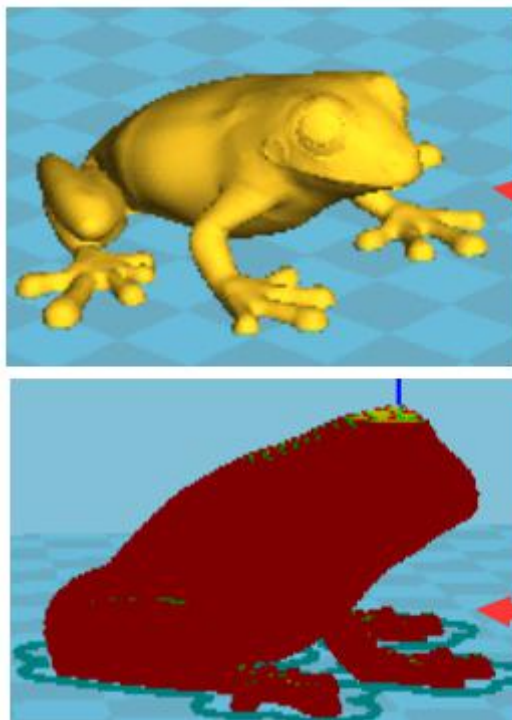
Open files Save gcode

View mode

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

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	☐
Only follow mesh surface	☐

Brim

Brim line amount	8

Raft

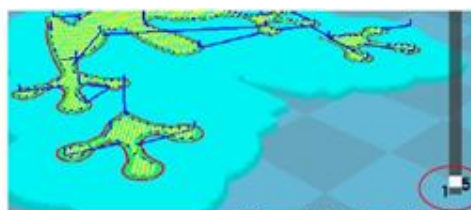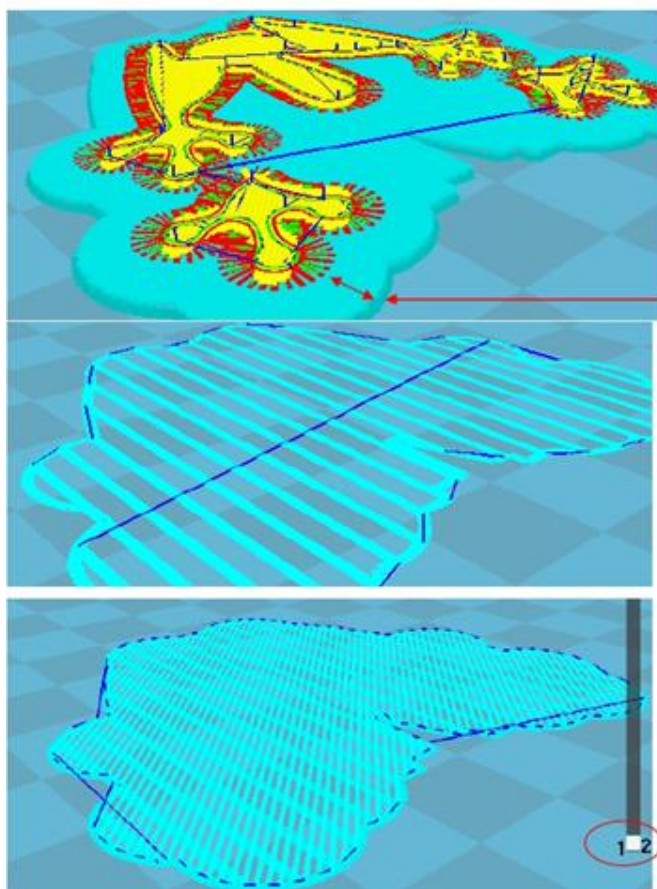|	
Extra margin (mm)	5.0
Line spacing (mm)	3.0
Base thickness (mm)	0.3
Base line width (mm)	1.0
Interface thickness (mm)	0.27
Interface line width (mm)	0.4
Airgap	0.0
First Layer Airgap	0.22
Surface layers	2
Surface layer thickness (mm)	0.27
Surface layer line width (mm)	0.4

Fix horrible

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

Ok

degree: 75

degree: 60

degree: 45

第一层与raft的间隙

Brim line amount	20

Raft

Extra margin (mm)	5.0
Line spacing (mm)	3.0
Base thickness (mm)	0.3
Base line width (mm)	1.0
Interface thickness (mm)	0.27
Interface line width (mm)	0.4
Airgap	0.22
Surface layers	2

Fix horrible

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

Ok

base line

Interface line

surface