

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.

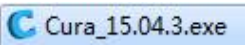
5. Note that in the printing process

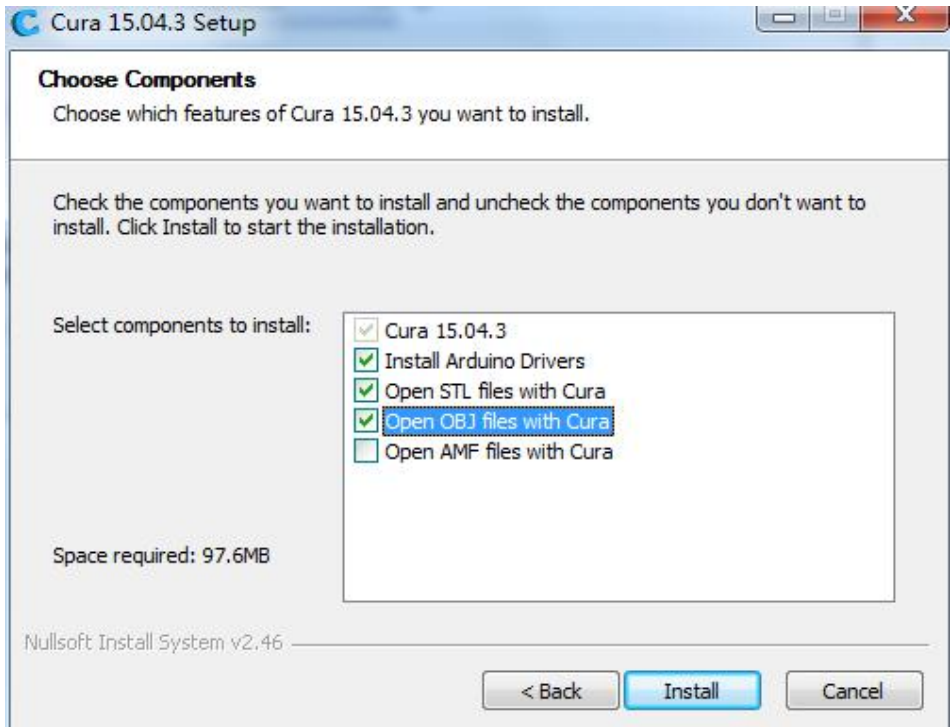
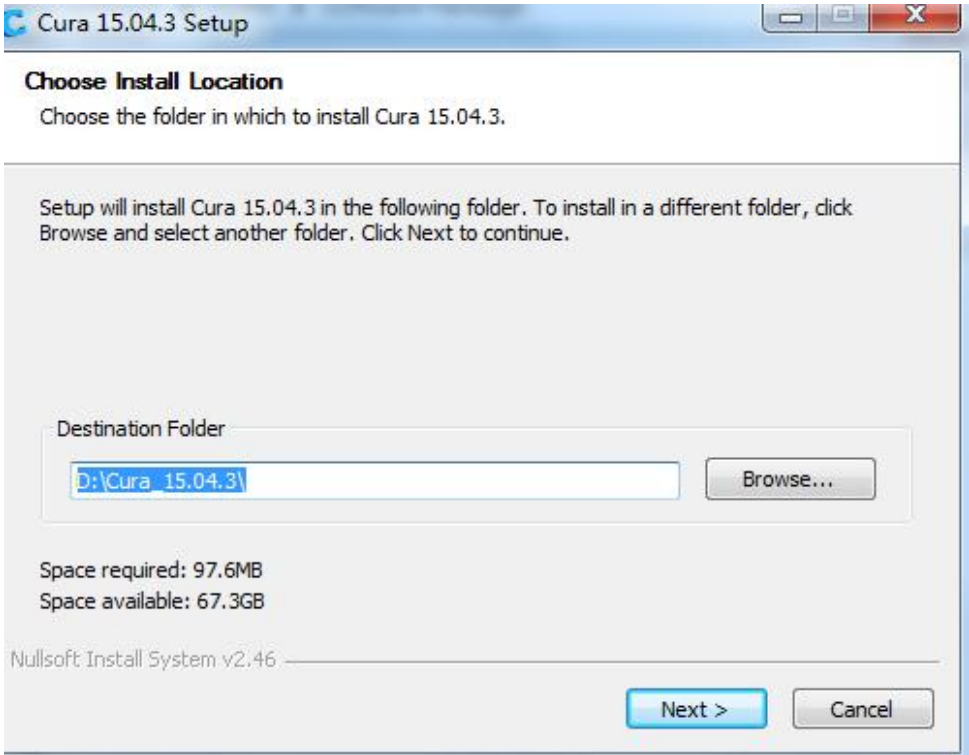
- Do not use the machine without the supervision of staff.
- Printing process and print just completed, avoid touching the printer's internal structure and prints, to prevent burns.
- If occur when printing printer smoke, abnormal noise, immediately turn off the power switch, the printer stops working, 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

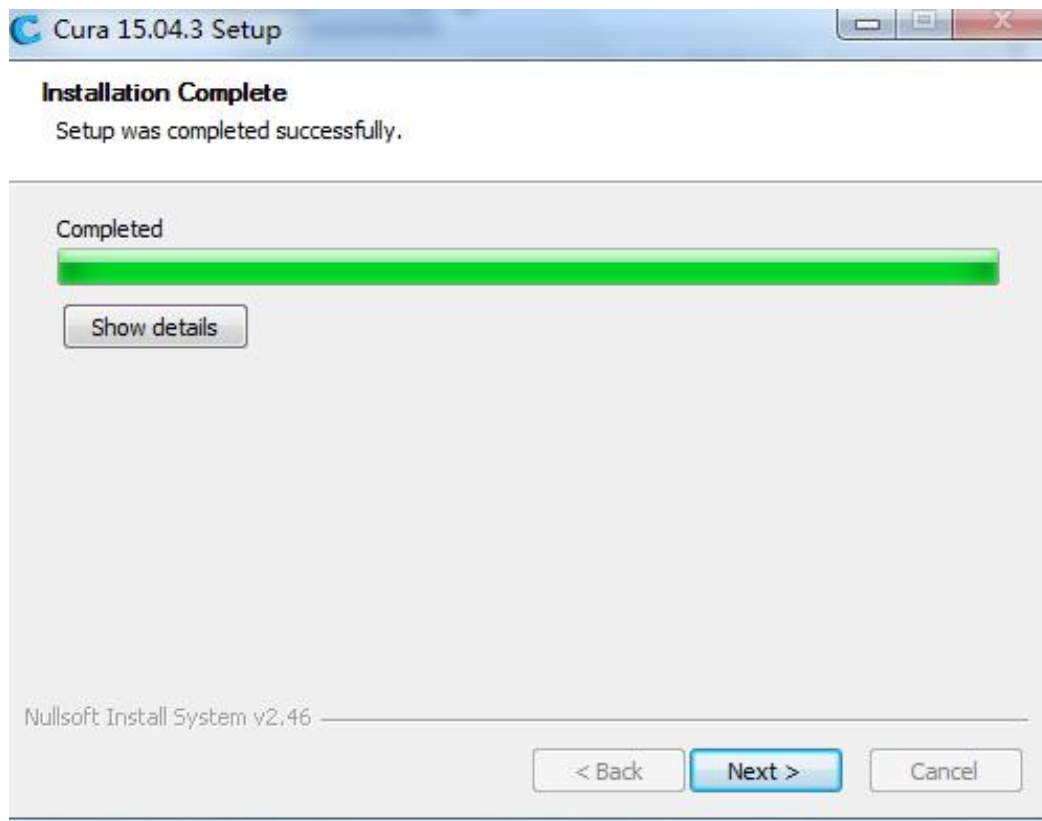
1. Slice software installation

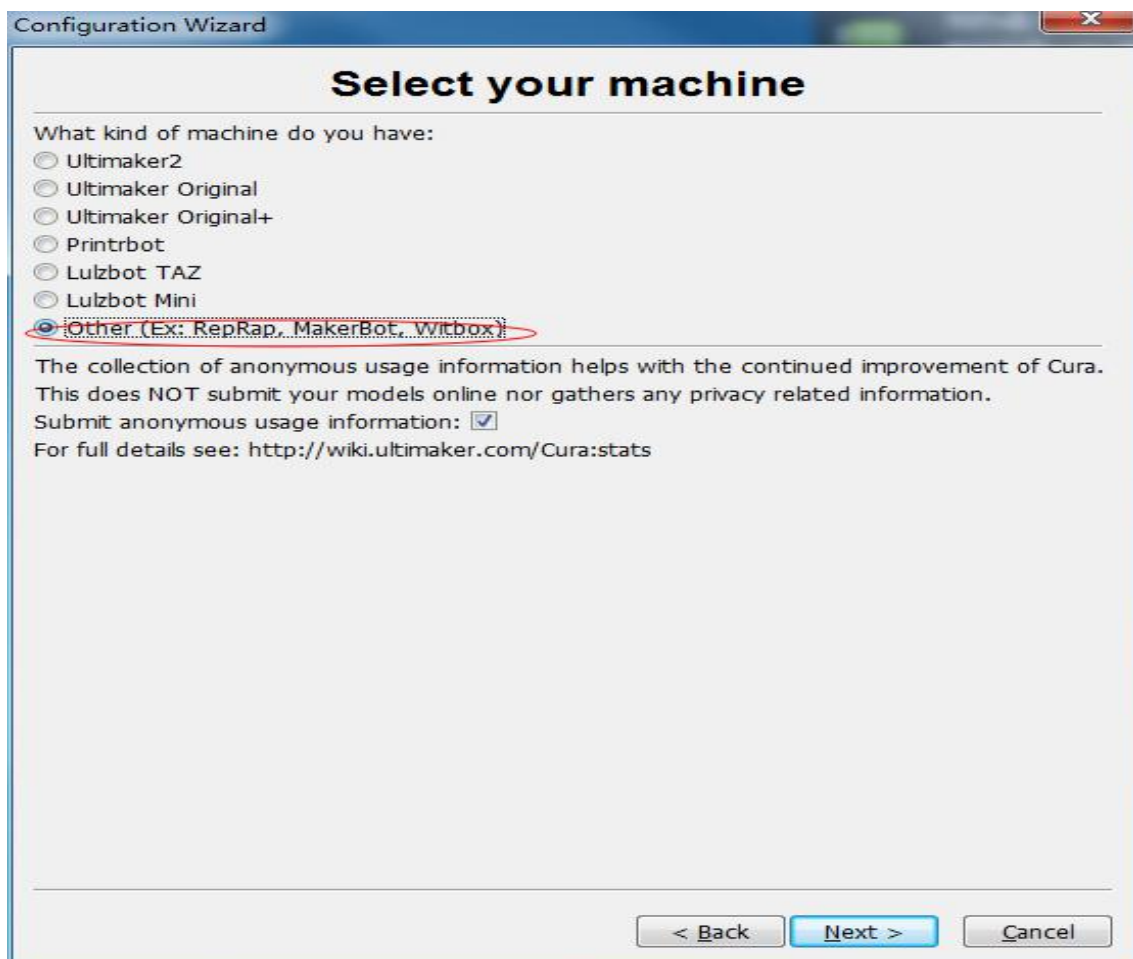
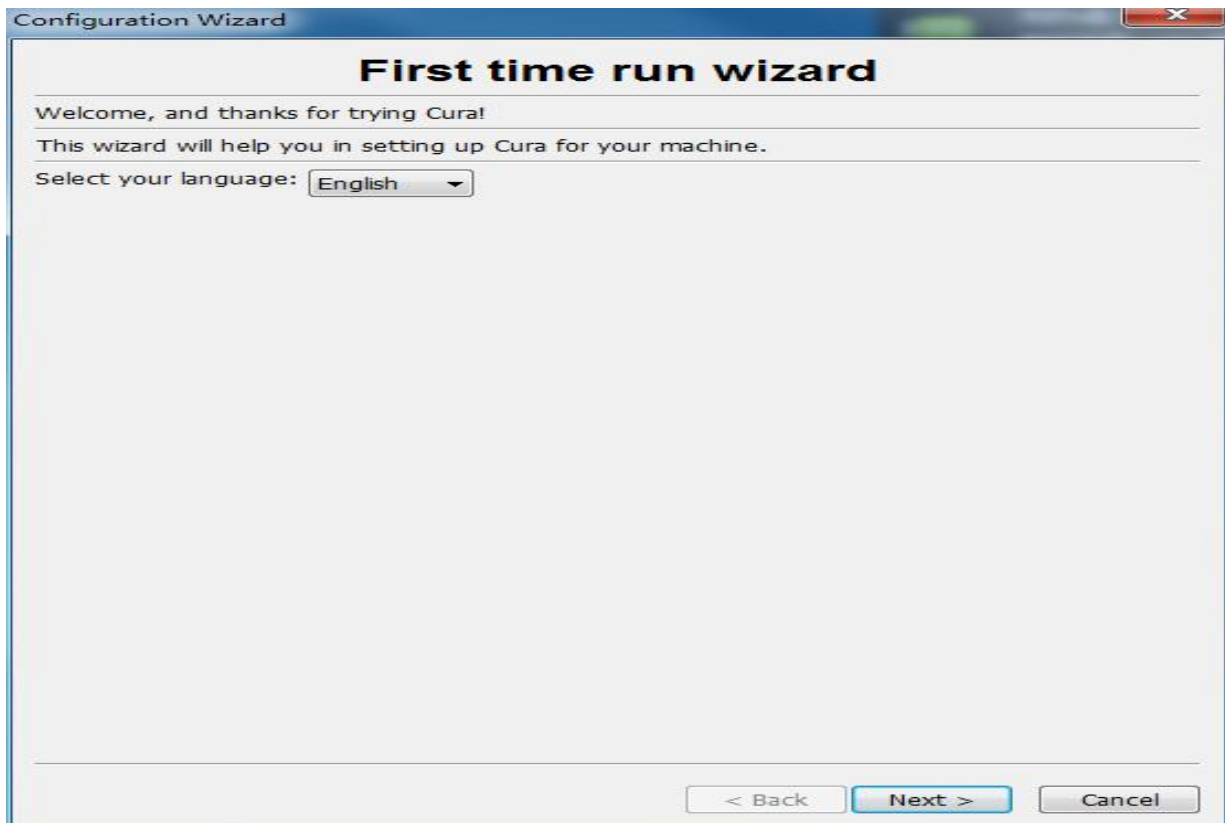
a. Install the software

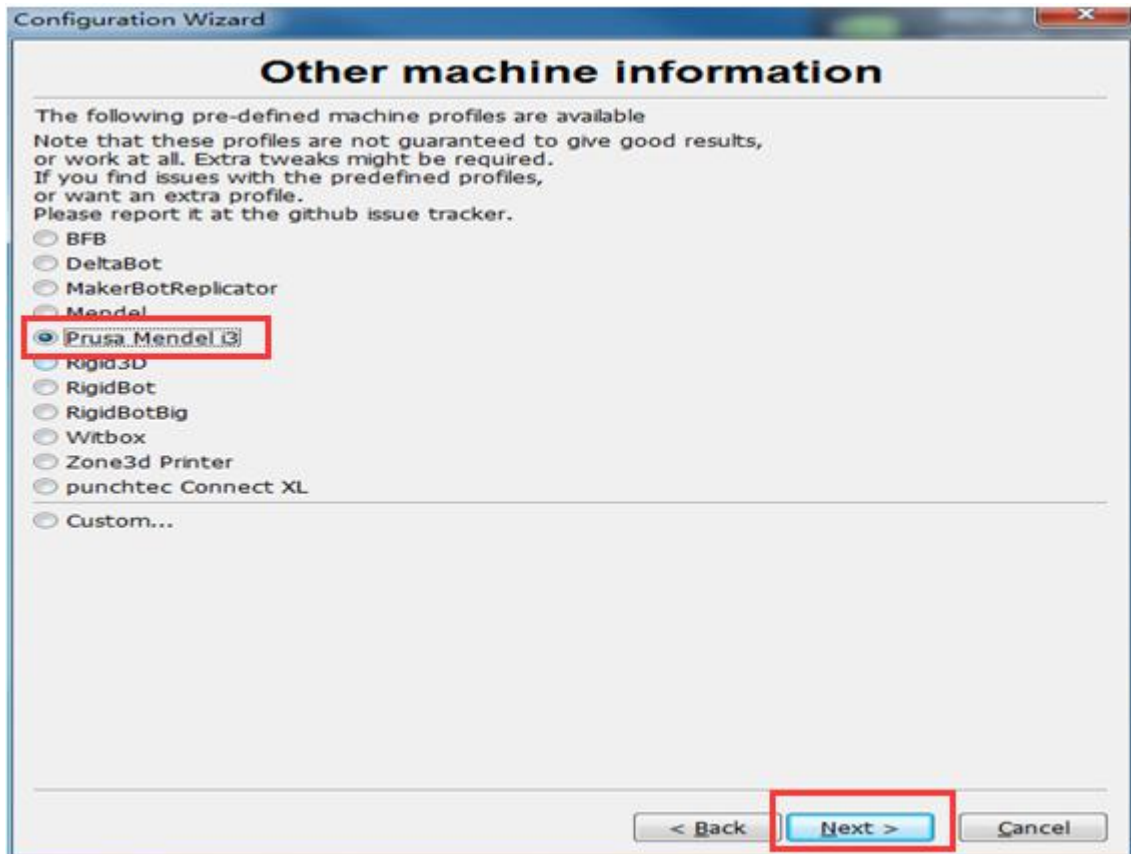
Click Setup  , pop-up installation window, always click Next to complete the installation.



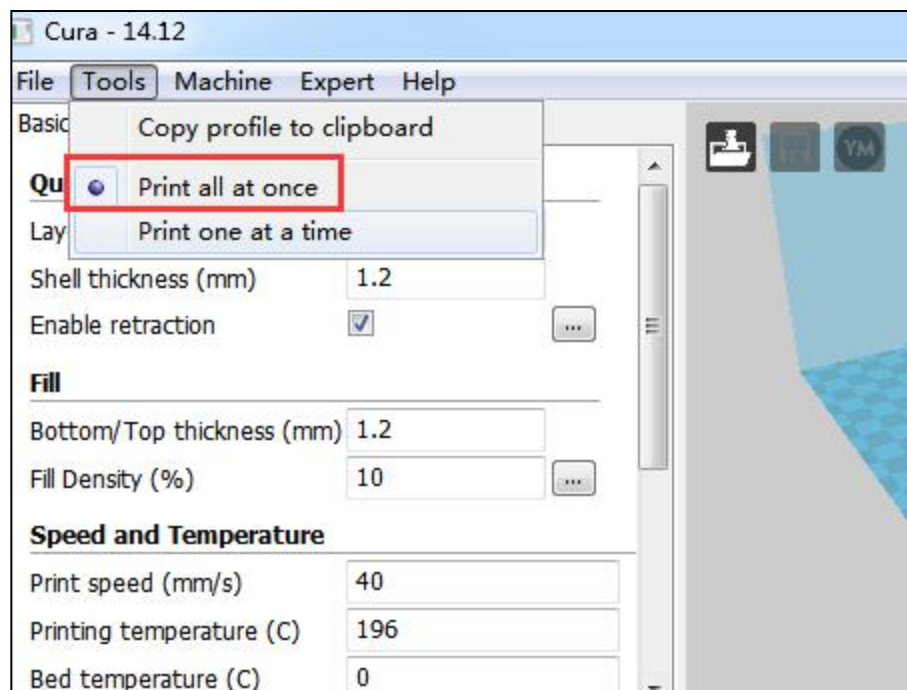


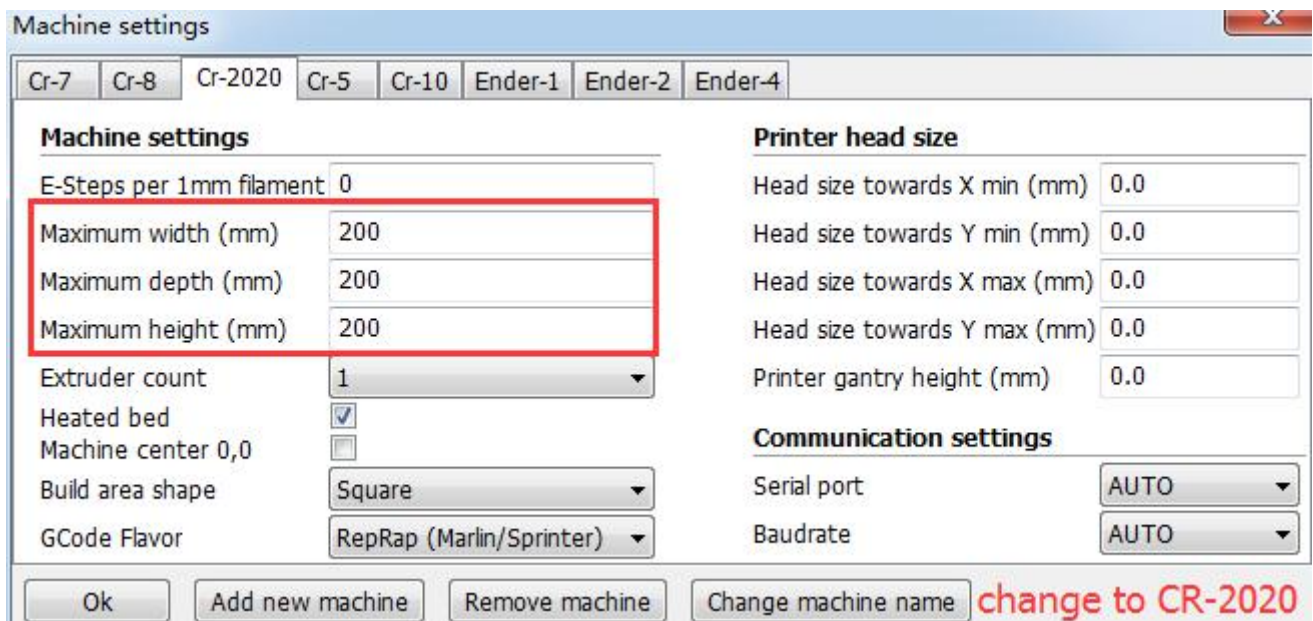
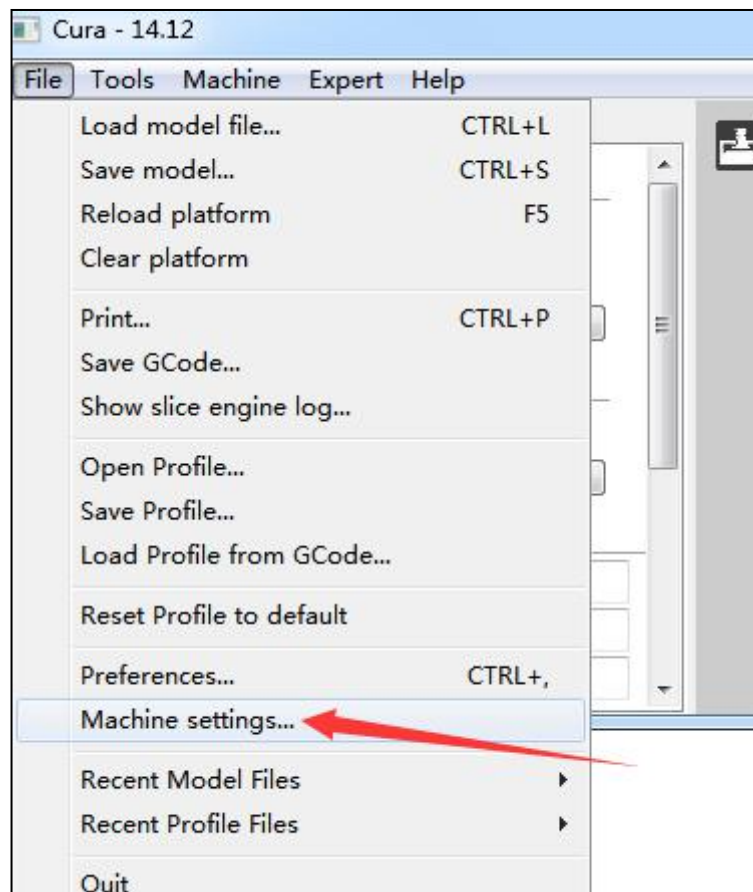






b. Select "print multiple model" from the "Tools"

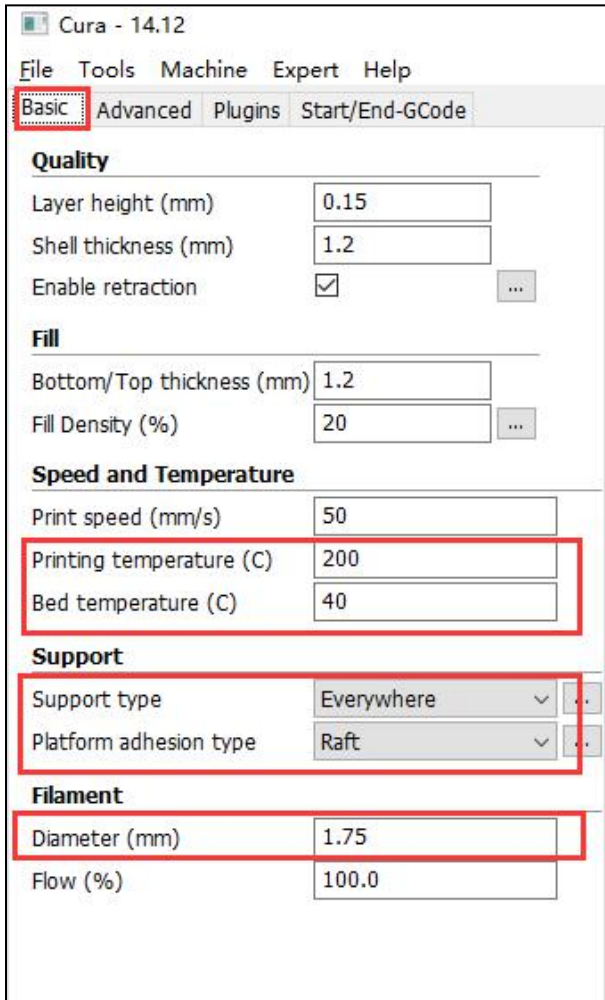




2. Software parameter setting

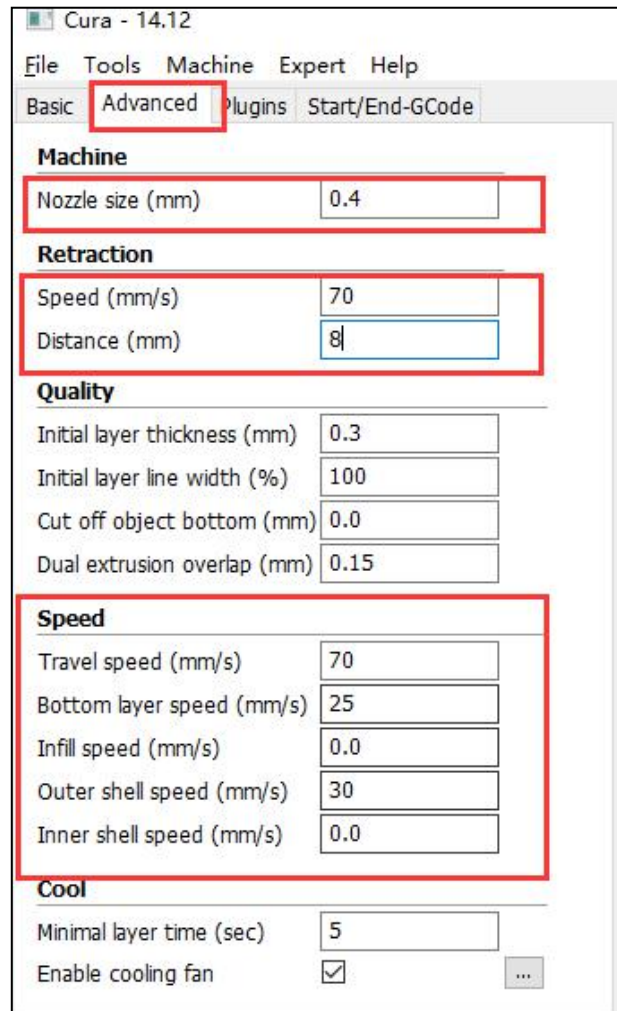
Open [Cura_15.04.3.exe](#), you can modify all of the printing process control parameters display screen.

a. Recommended parameter settings:



The screenshot shows the 'Basic' settings tab in Cura 14.12. The 'Basic' tab is selected and highlighted with a red box. The settings are organized into several sections: Quality, Fill, Speed and Temperature, Support, and Filament. The 'Printing temperature (C)' and 'Bed temperature (C)' fields are highlighted with a red box. The 'Support type' and 'Platform adhesion type' dropdowns are also highlighted with a red box. The 'Filament' section, including 'Diameter (mm)' and 'Flow (%)', is highlighted with a red box.

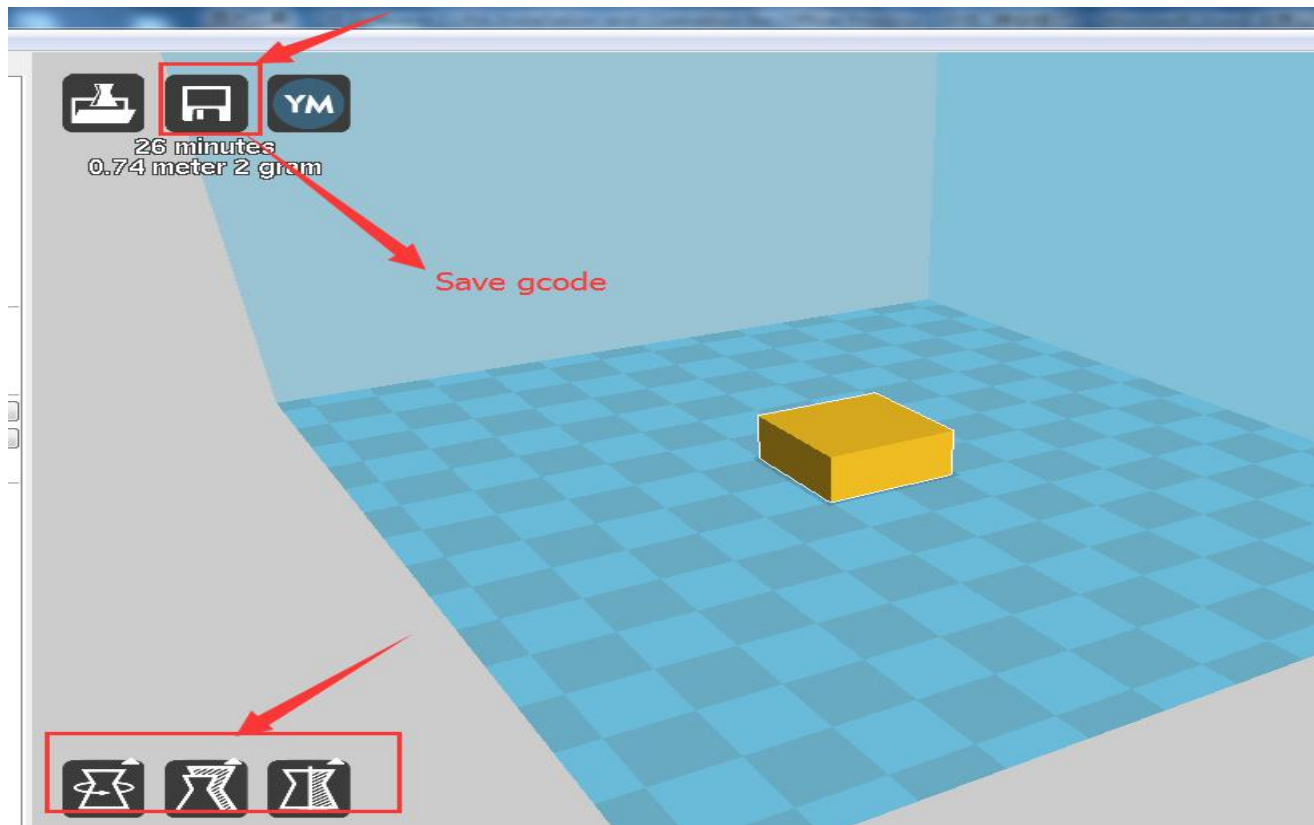
| Section | Parameter | Value |
|-----------------------|---------------------------|-------------------------------------|
| Quality | Layer height (mm) | 0.15 |
| | Shell thickness (mm) | 1.2 |
| | Enable retraction | <input checked="" type="checkbox"/> |
| Fill | Bottom/Top thickness (mm) | 1.2 |
| | 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 |



The screenshot shows the 'Advanced' settings tab in Cura 14.12. The 'Advanced' tab is selected and highlighted with a red box. The settings are organized into several sections: Machine, Retraction, Quality, Speed, and Cool. The 'Machine' section, including 'Nozzle size (mm)', is highlighted with a red box. The 'Retraction' section, including 'Speed (mm/s)' and 'Distance (mm)', is highlighted with a red box. The 'Speed' section, including 'Travel speed (mm/s)', 'Bottom layer speed (mm/s)', 'Infill speed (mm/s)', 'Outer shell speed (mm/s)', and 'Inner shell speed (mm/s)', is highlighted with a red box. The 'Cool' section, including 'Minimal layer time (sec)' and 'Enable cooling fan', is highlighted with a red box.

| Section | Parameter | Value |
|------------|------------------------------|-------------------------------------|
| 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 | <input checked="" type="checkbox"/> |

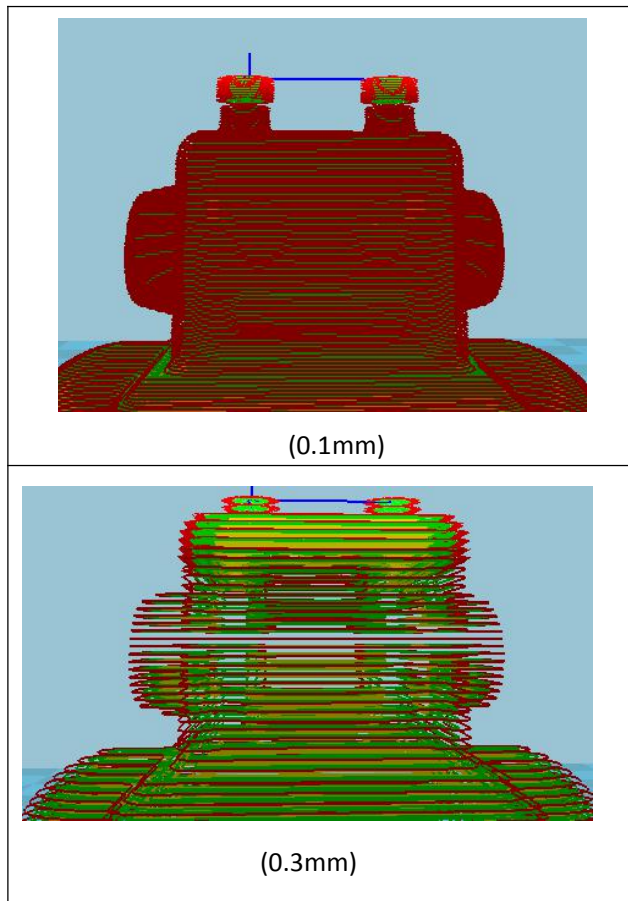
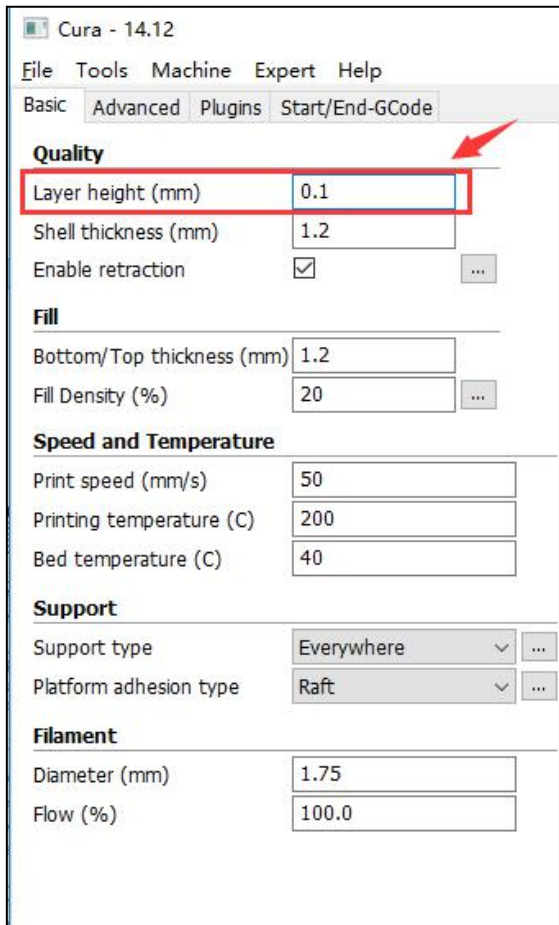
b. For in-depth study, refer to the following notes:



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



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

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

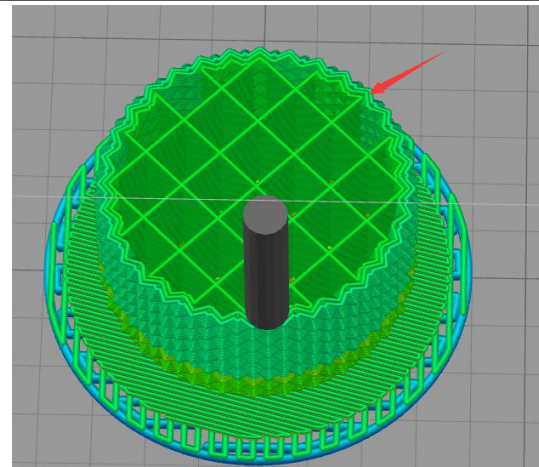
Support type Everywhere

Platform adhesion type Raft

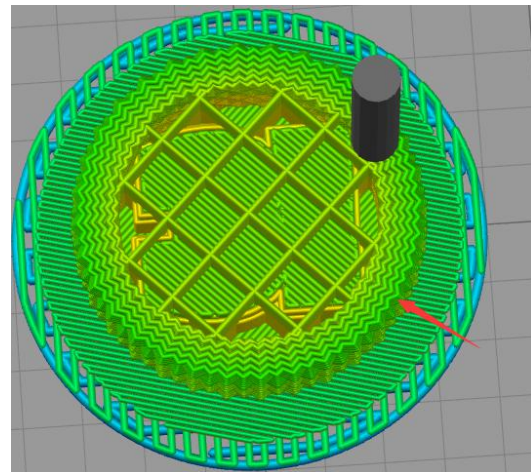
Filament

Diameter (mm) 1.75

Flow (%) 100.0



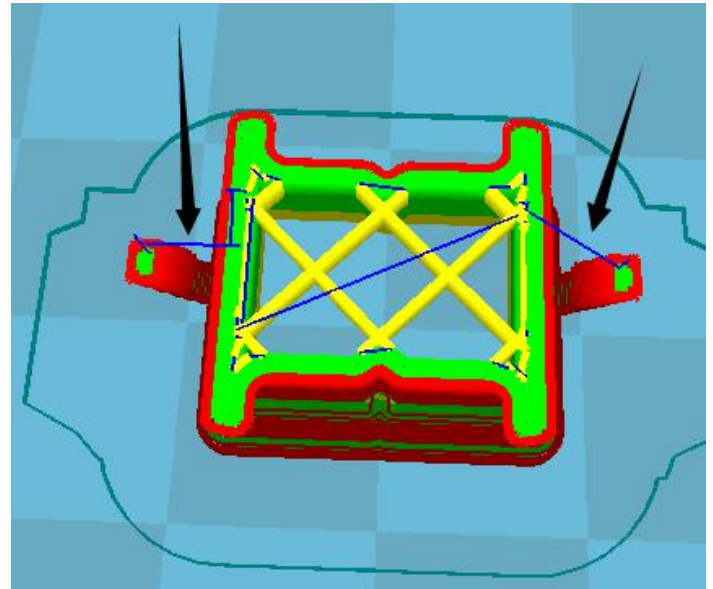
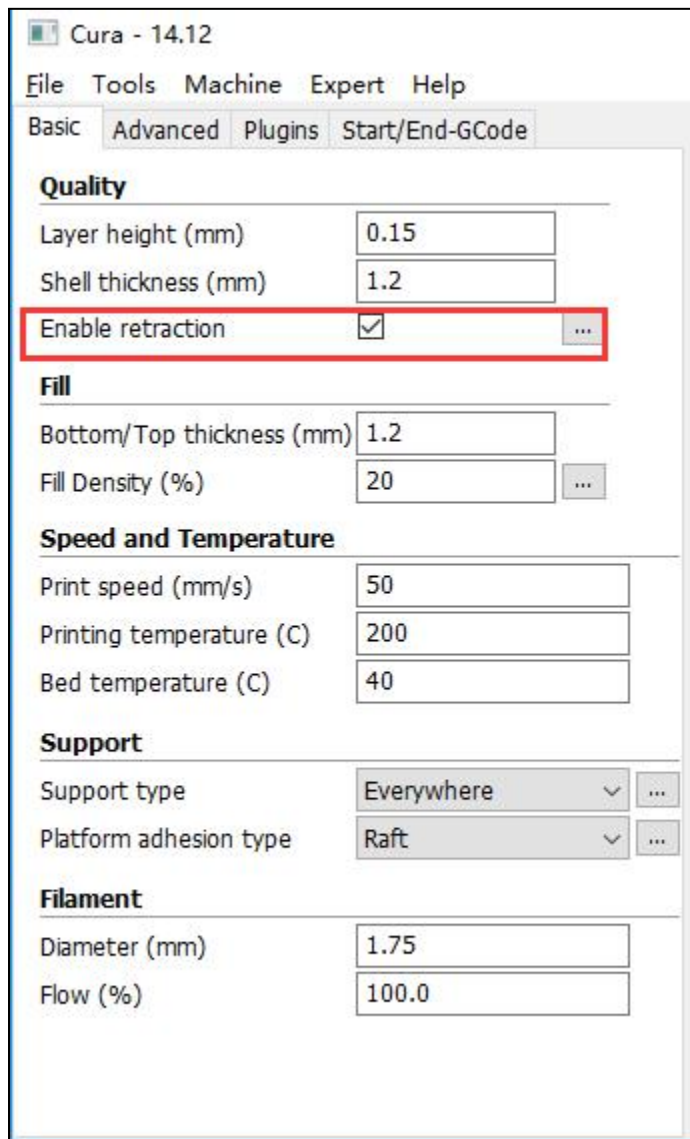
0.8mm(If the nozzle is 0.4mm)



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.

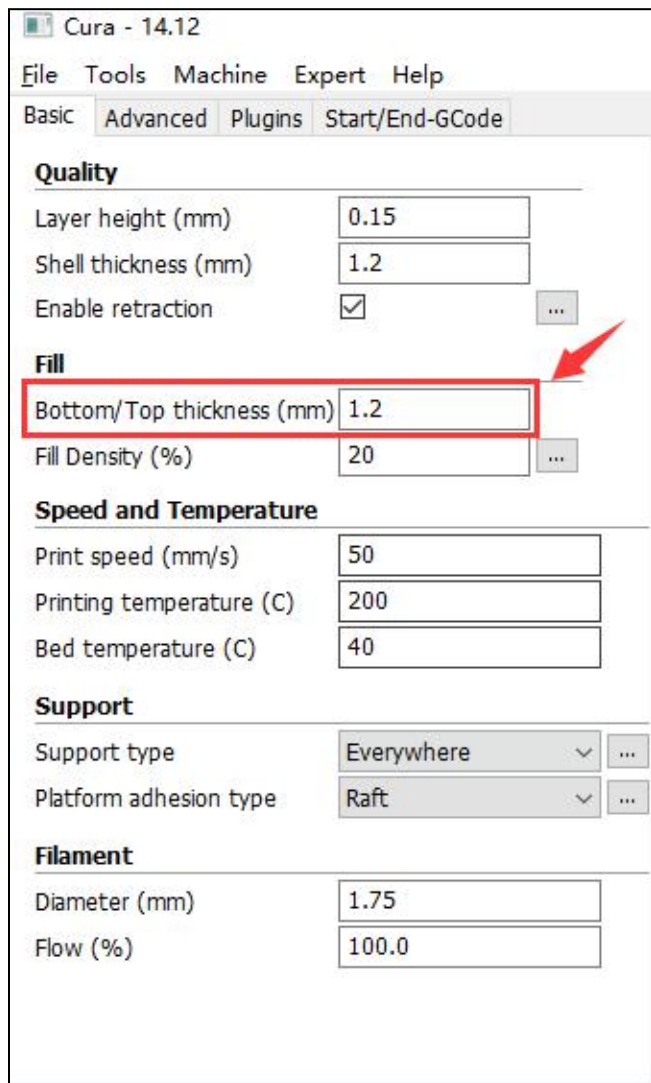
3、 Enable retraction



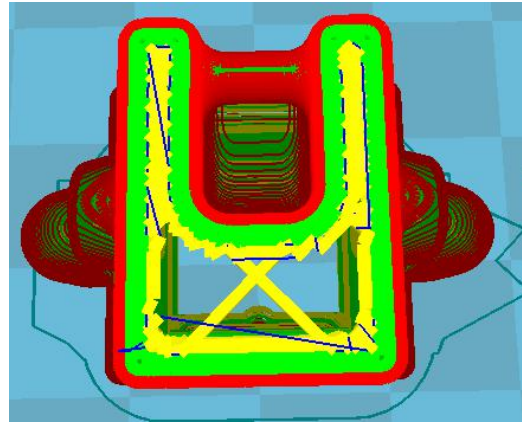
Explanation:

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.

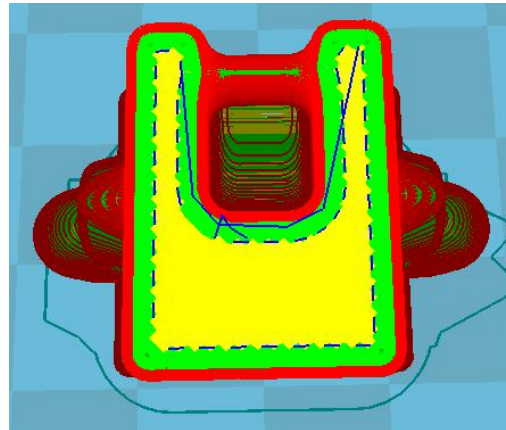
4、 Bottom/Top thickness



Under the same layers



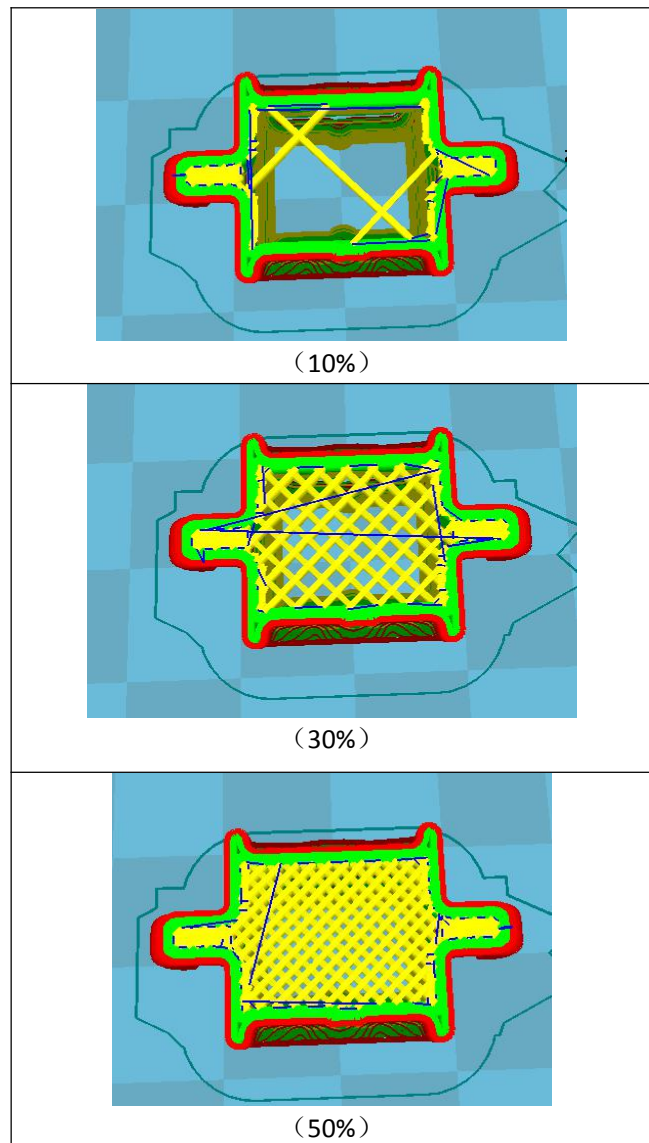
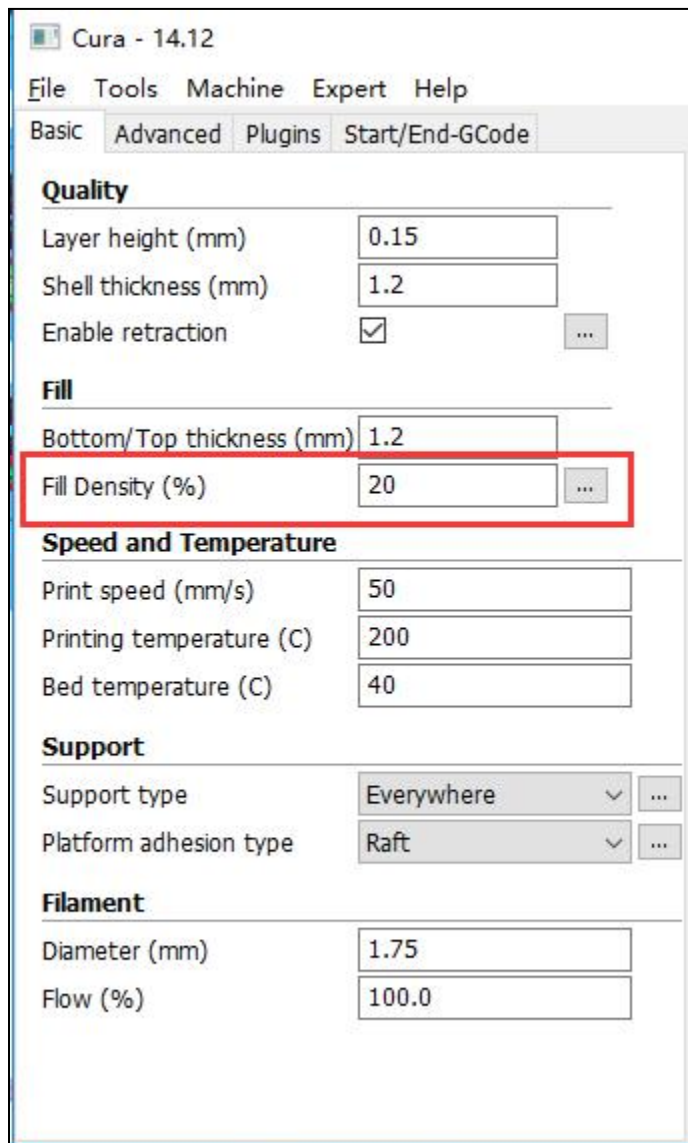
(0.6mm)



(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.

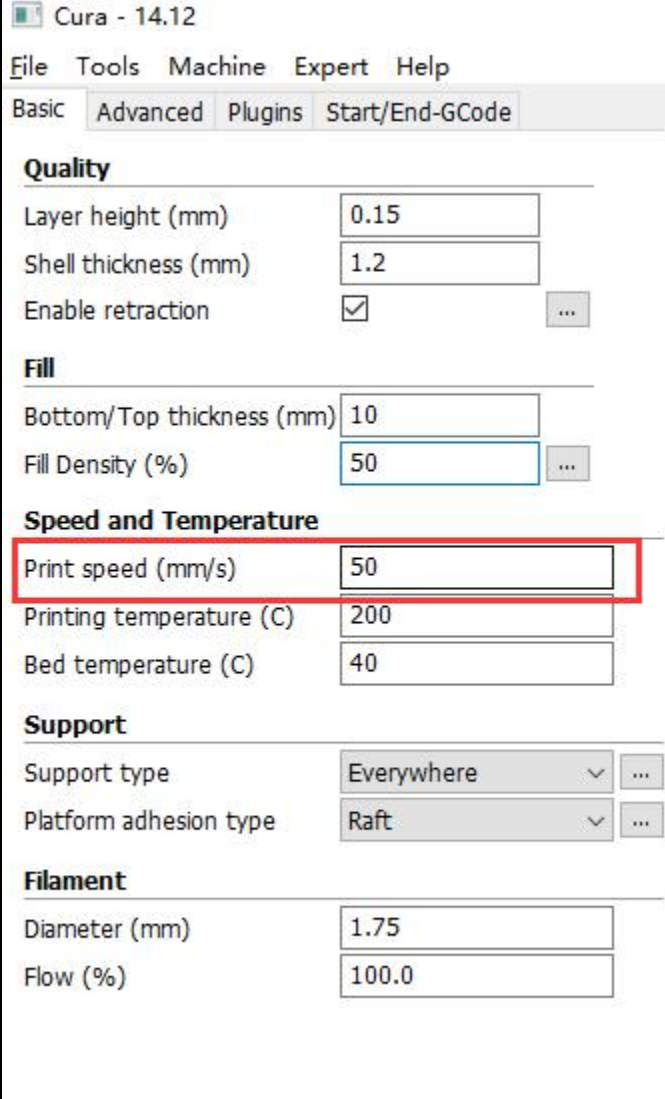
5、 Fill Density(%)



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)



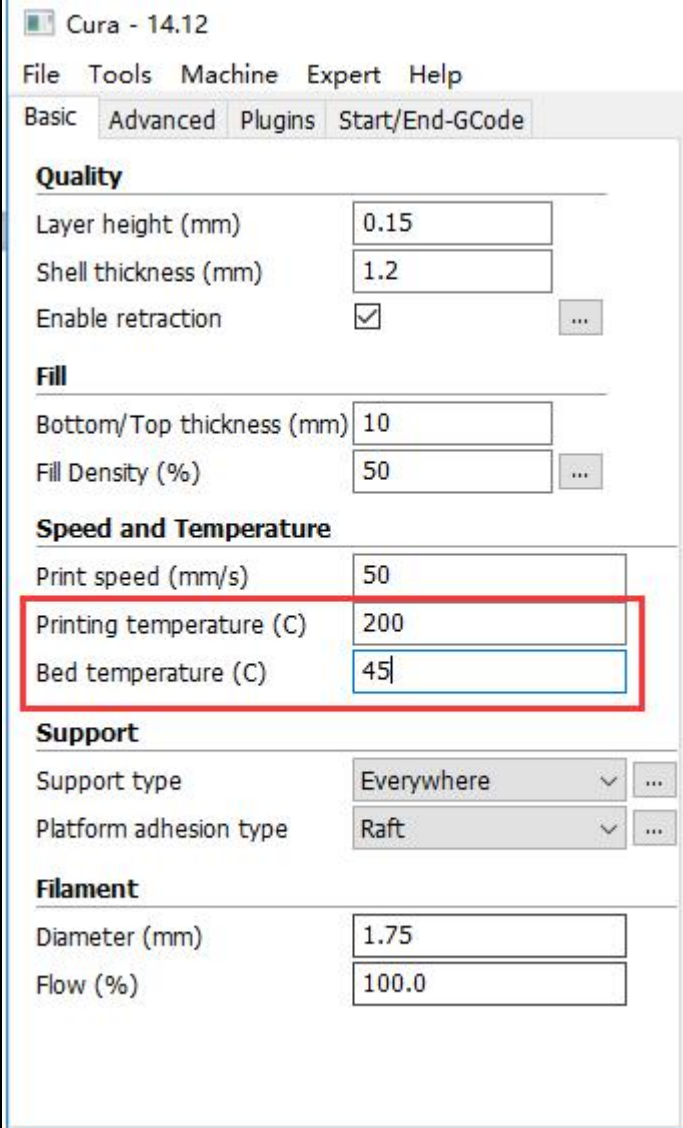
The image shows the Cura 14.12 software interface, specifically the 'Basic' settings tab. The 'Print speed (mm/s)' field is highlighted with a red rectangle. The settings are organized into sections: Quality, Fill, Speed and Temperature, Support, and Filament.

| Section | Setting | Value |
|-----------------------|---------------------------|-------------------------------------|
| Quality | Layer height (mm) | 0.15 |
| | Shell thickness (mm) | 1.2 |
| | Enable retraction | <input checked="" type="checkbox"/> |
| 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(°C)



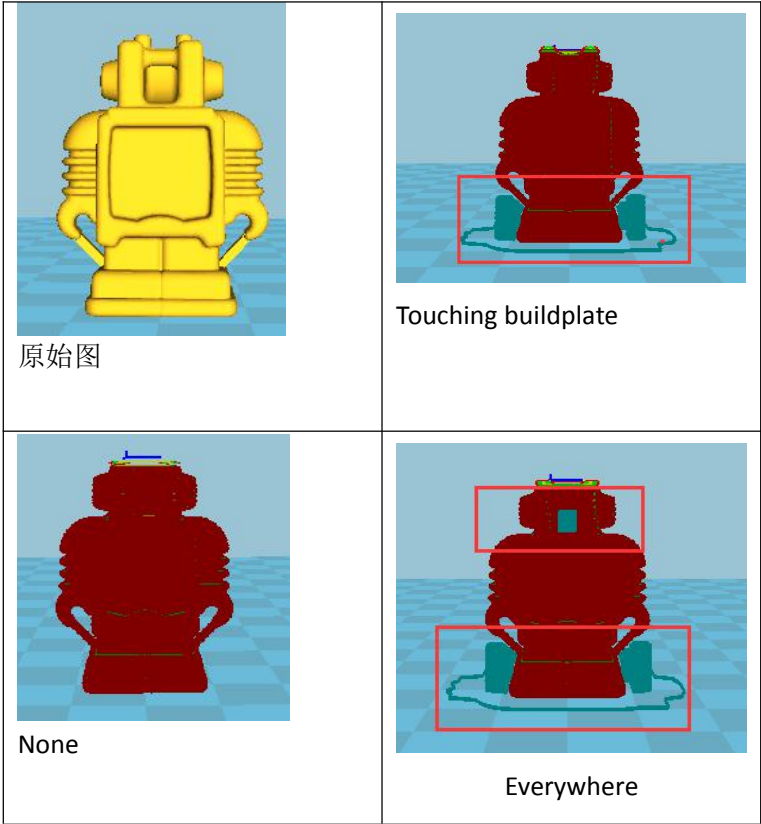
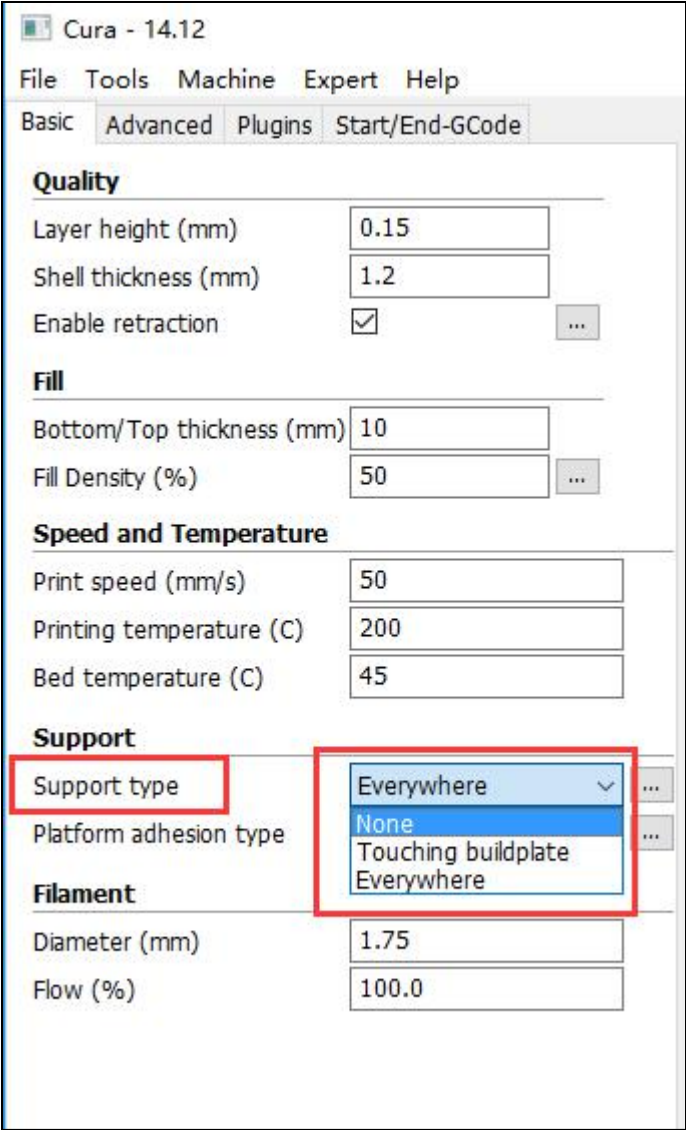
The image shows the 'Basic' settings tab in Cura 14.12. The 'Speed and Temperature' section is highlighted with a red box. It contains three input fields: 'Print speed (mm/s)' with a value of 50, 'Printing temperature (C)' with a value of 200, and 'Bed temperature (C)' with a value of 45. The 'Printing temperature (C)' field is also highlighted with a blue border.

| Section | Parameter | Value |
|-----------------------|---------------------------|-------------------------------------|
| Quality | Layer height (mm) | 0.15 |
| | Shell thickness (mm) | 1.2 |
| | Enable retraction | <input checked="" type="checkbox"/> |
| 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



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

FileToolsMachineExpertHelp

BasicAdvancedPluginsStart/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 typeEverywhere

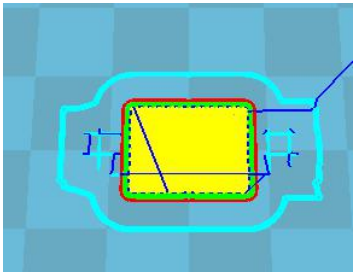
Platform adhesion type

RaftNoneBrimRaft

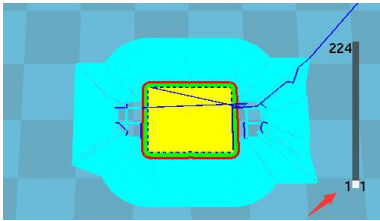
Filament

Diameter (mm)

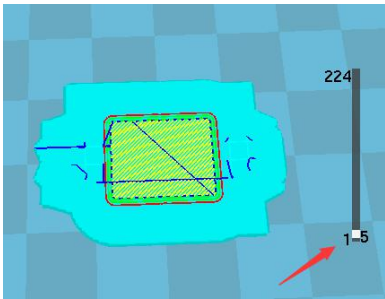
Flow (%)100.0



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

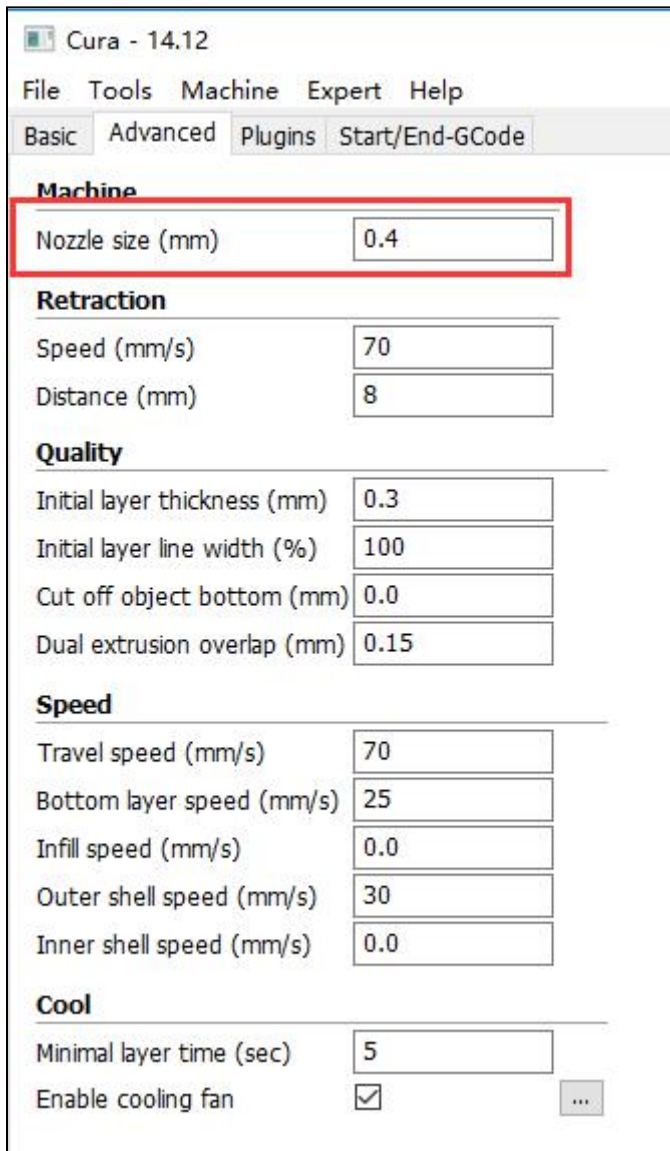
Diameter (mm)

1.75

Flow (%)

100.0

11、 nozzle size (mm)



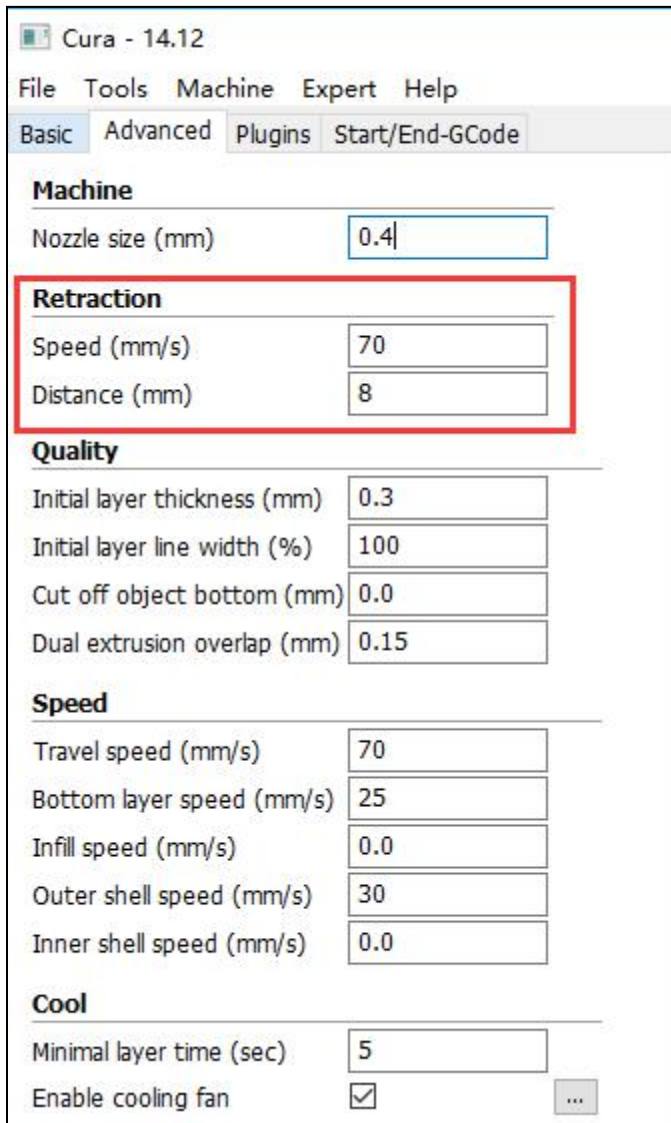
The screenshot shows the Cura 14.12 software interface. The 'Machine' tab is selected, and the 'Nozzle size (mm)' is set to 0.4. Other settings visible include Retraction (Speed: 70 mm/s, Distance: 8 mm), Quality (Initial layer thickness: 0.3 mm, Initial layer line width: 100%, Cut off object bottom: 0.0 mm, Dual extrusion overlap: 0.15 mm), Speed (Travel speed: 70 mm/s, Bottom layer speed: 25 mm/s, Infill speed: 0.0 mm/s, Outer shell speed: 30 mm/s, Inner shell speed: 0.0 mm/s), and Cool (Minimal layer time: 5 sec, Enable cooling fan: checked).

| Section | Parameter | Value |
|------------|------------------------------|-------------------------------------|
| Machine | Nozzle size (mm) | 0.4 |
| | Retraction | |
| 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 | |
| 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 | <input checked="" type="checkbox"/> |

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



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 ☒ ...

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

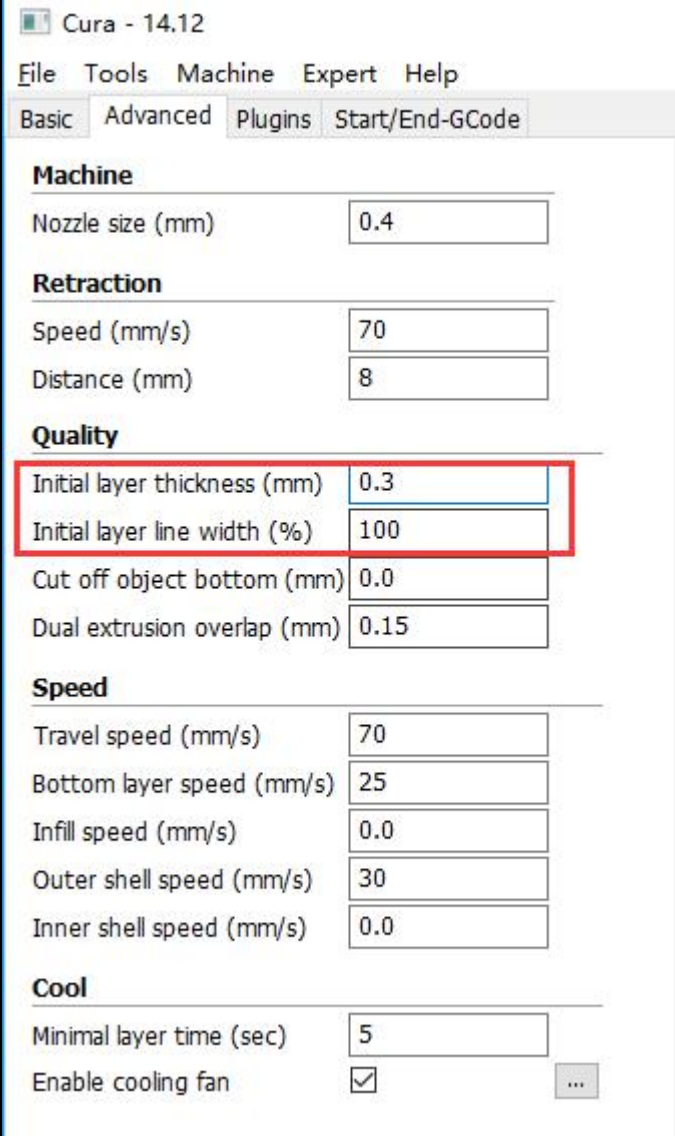
1、speed(mm/s)

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



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 ☒ ...

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

FileToolsMachineExpertHelp

BasicAdvancedPluginsStart/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

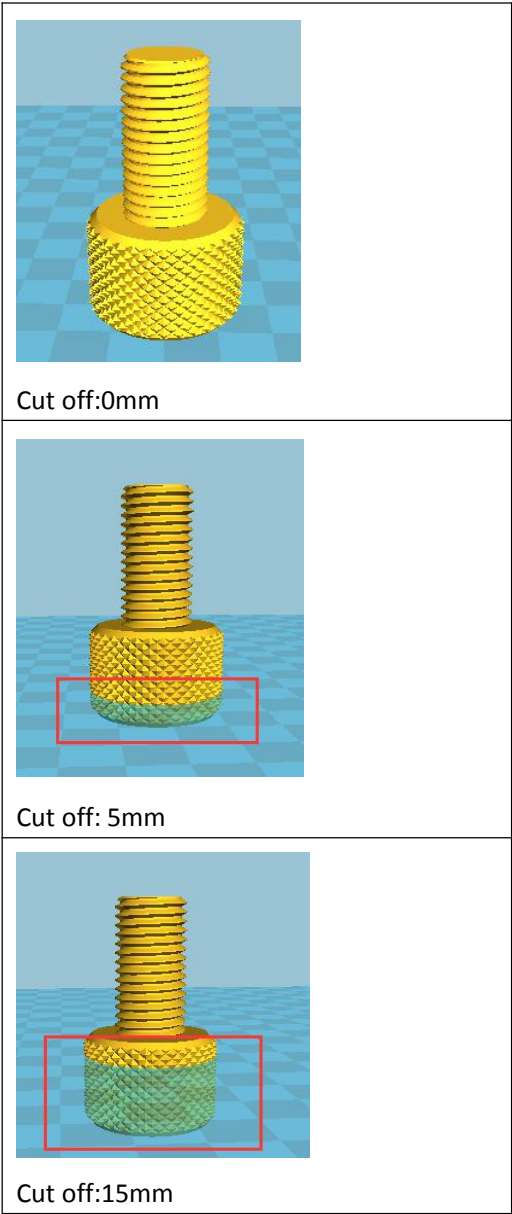
Outer shell speed (mm/s)30

Inner shell speed (mm/s)0.0

Cool

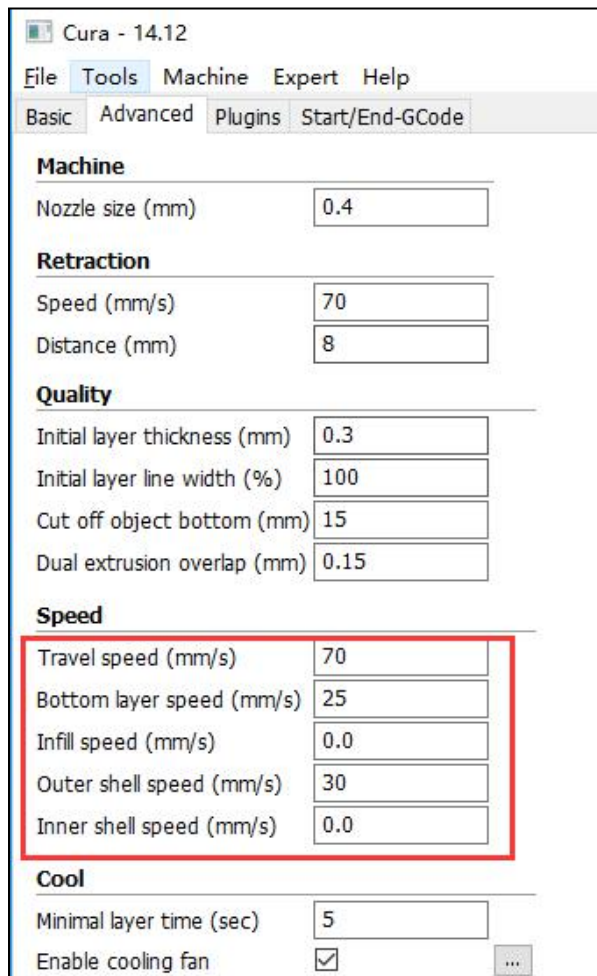
Minimal layer time (sec)5

Enable cooling fan☒



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



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) 15

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 ☒

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

FileToolsMachineExpertHelp

BasicAdvancedPluginsStart/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 heightX

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 3.2X

Change printing parameters at a given height

Z height to tweak at (mm)5.0

(ALT) Layer no. to tweak at

New Speed (%)

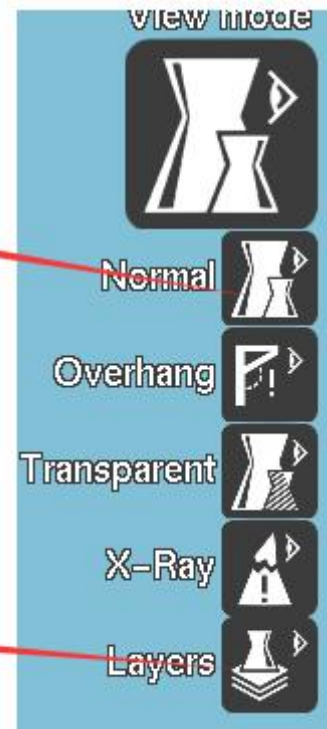
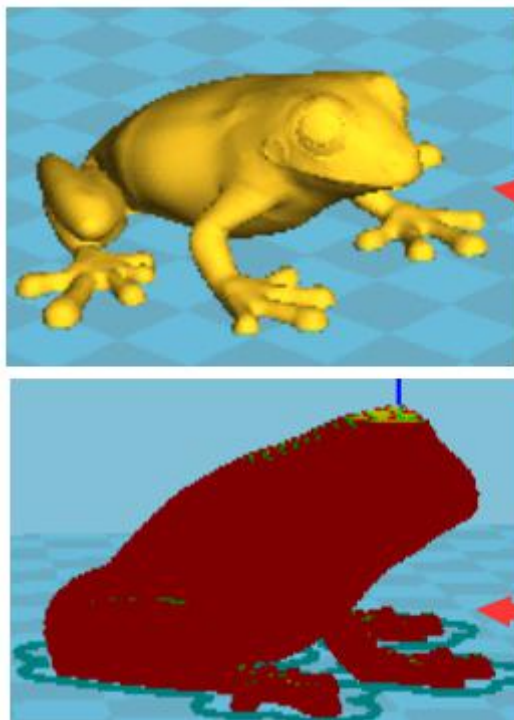
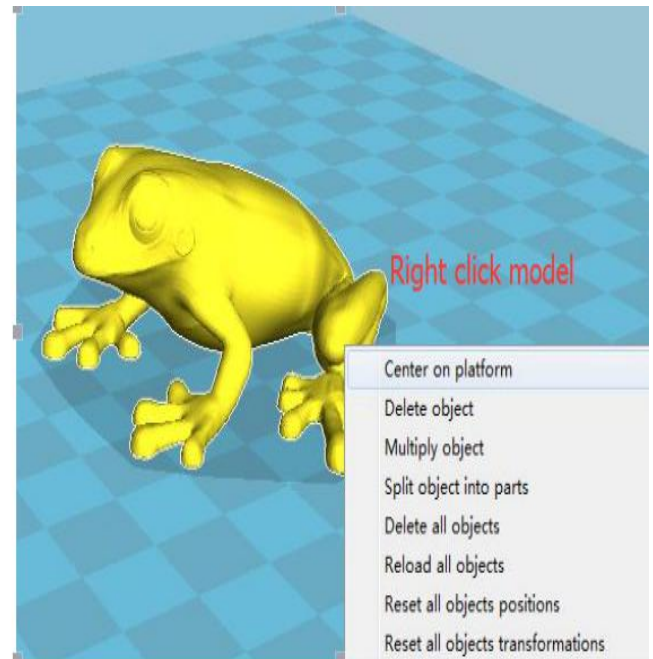
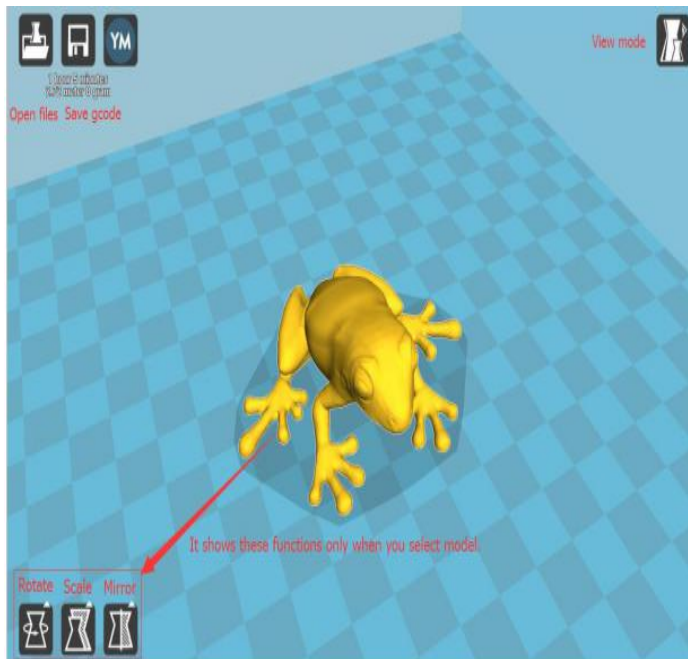
New General Flow Rate (%)

New Flow Rate Extruder 1 (%)

New Flow Rate Extruder 2 (%)

New Bed Temp (deg C)

New Extruder 1 Temp (deg C)



Expert settings

Cura - 14.12

FileToolsMachineExpertHelp

BasicAdvancedPlugin

Switch to quickprint...

Switch to full settings...

Quality

Layer height (mm)

Shell thickness (mm)

Enable retraction

Fill

Bottom/Top thickness (mm) 10

Open expert settings...CTRL+E

Run bed leveling wizard...

Run head offset wizard...

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.

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 count1

Start distance (mm)3.0

Minimal length (mm)150.0

Cool

Fan full on at height (mm)0.5

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 typeLines

Overhang angle for support (deg)45

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 amount20

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

Airgap0.0

First Layer Airgap0.22

Surface layers2

Surface layer thickness (mm)0.27

Surface layer line width (mm)0.4

Fix horrible

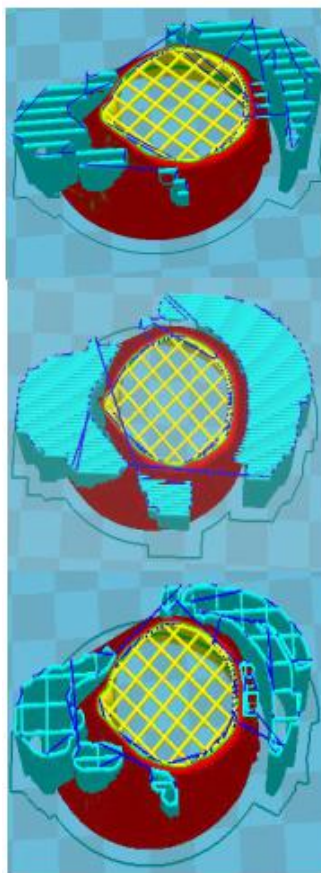
Combine everything (Type-A)☒

Combine everything (Type-B)☐

Keep open faces☐

Extensive stitching☐

Ok



type:Line

Degree:60

Fill amount:15

Distance X/Y :0.7

type:Line

Degree:60

Fill amount:50

Distance X/Y :3

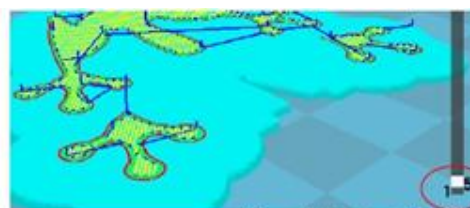
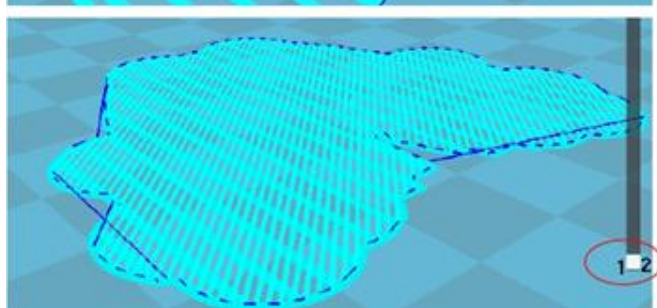
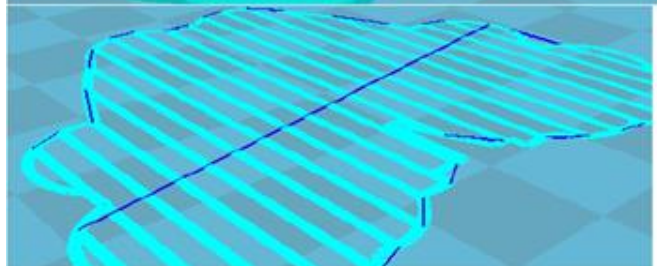
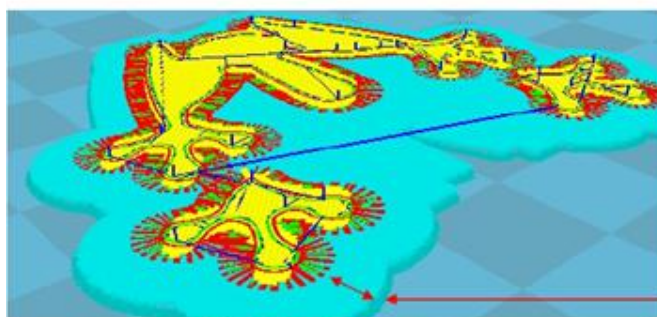
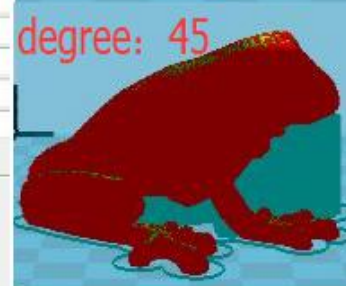
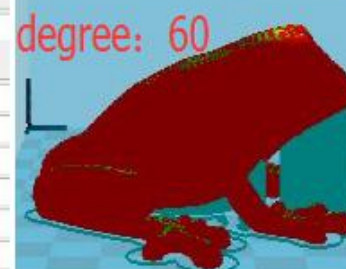
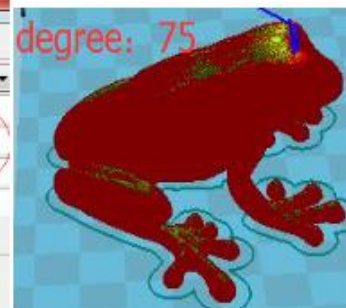
type:Grid

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 | <input type="checkbox"/> |
| Only follow mesh surface | <input type="checkbox"/> |
| 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) | <input checked="" type="checkbox"/> |
| Combine everything (Type-B) | <input type="checkbox"/> |
| Keep open faces | <input type="checkbox"/> |
| Extensive stitching | <input type="checkbox"/> |



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

base line

Interface line

surface

