

EasyThreed self-developed slicing software is a simple and easy to use slicing software, suitable for beginners and children. If you want better printing effect, you should learn how to use Cura slice.



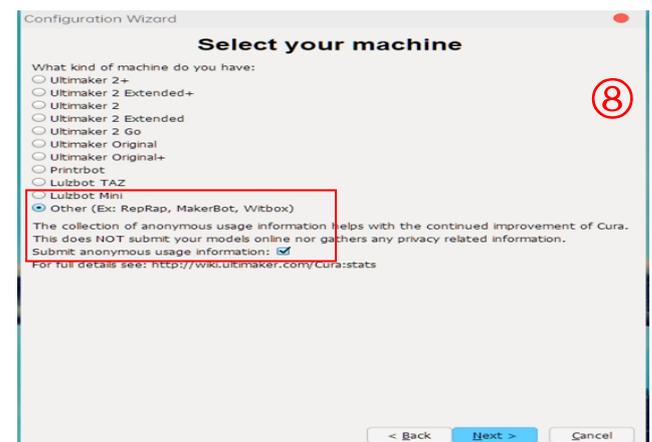
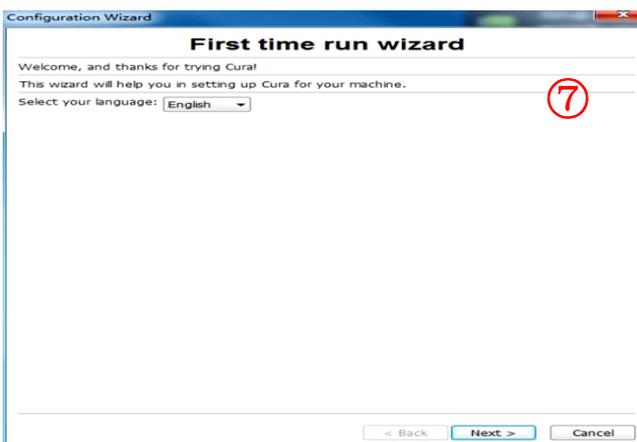
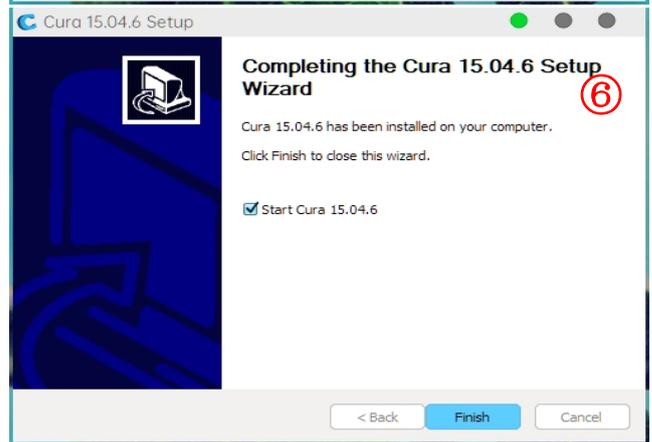
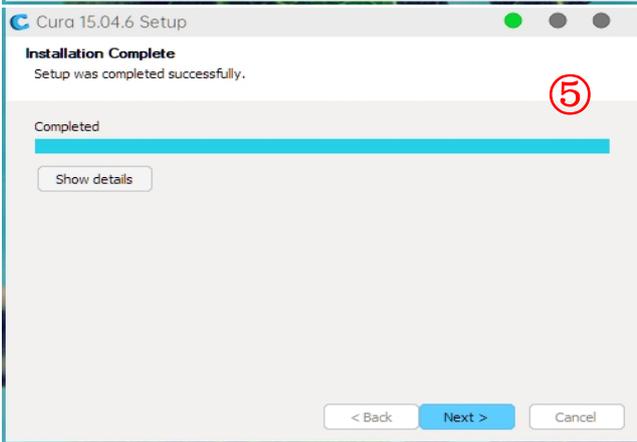
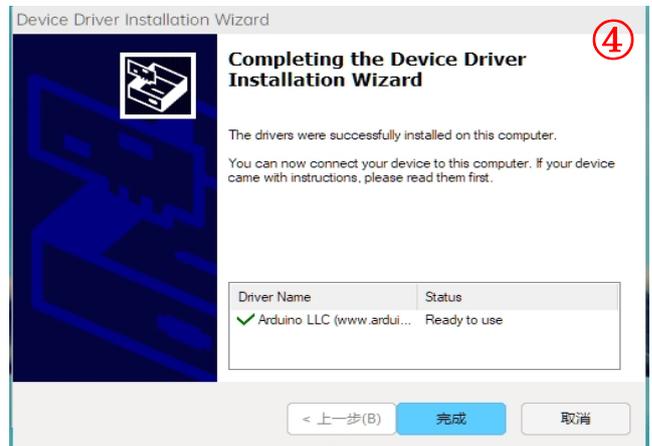
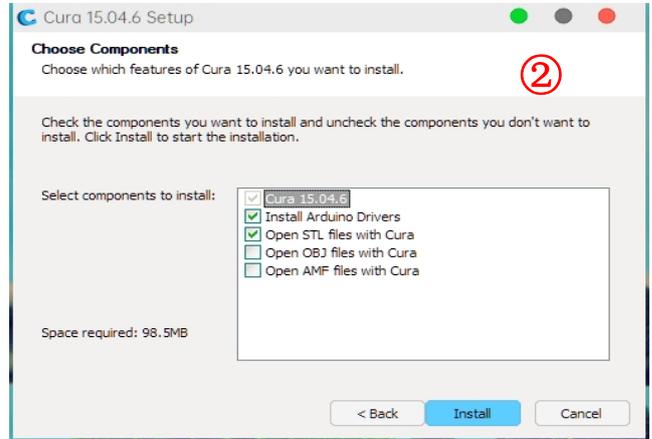
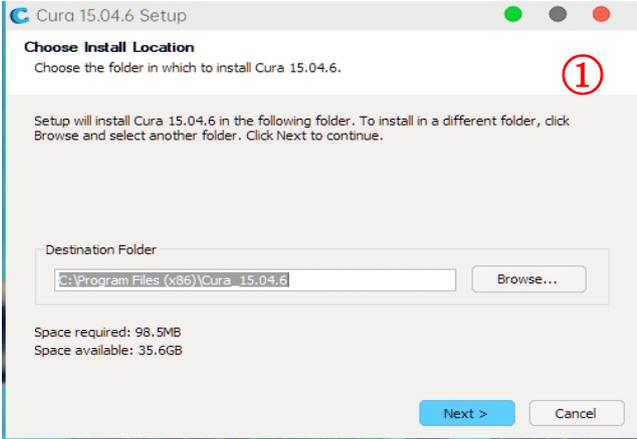
CURA 15.04 user's instruction

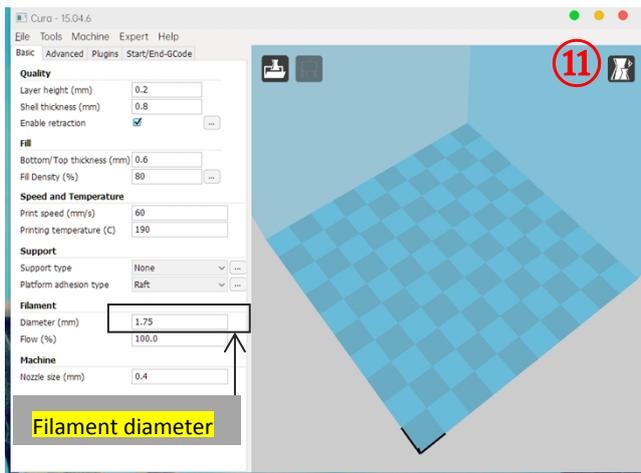
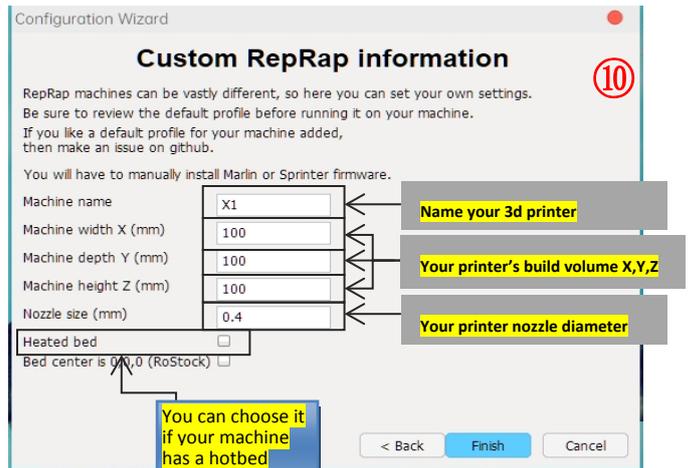
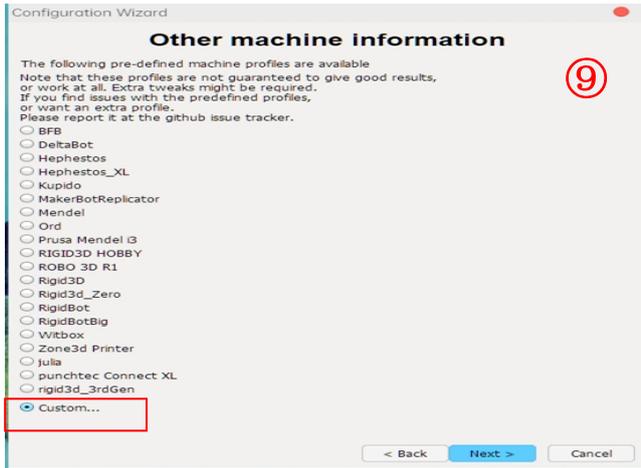
Please read the setting instruction carefully before application,

The picture is for reference only, and the product is subject to the real object

1: install CURA software

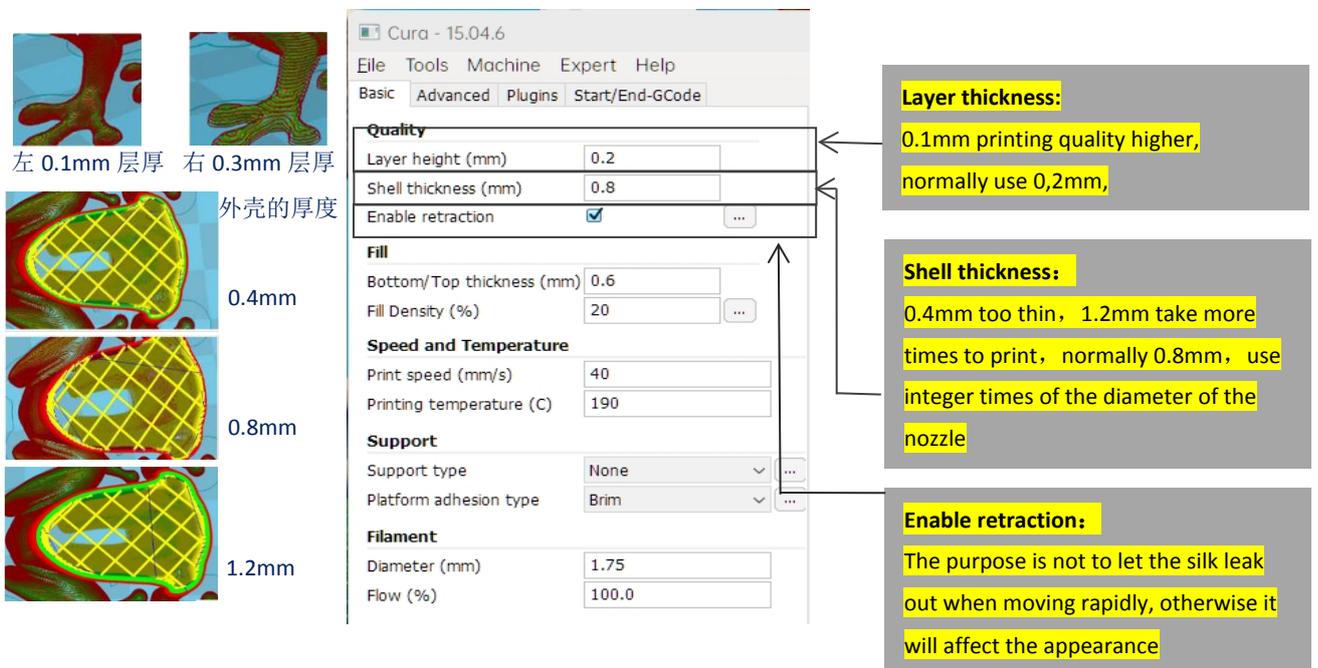
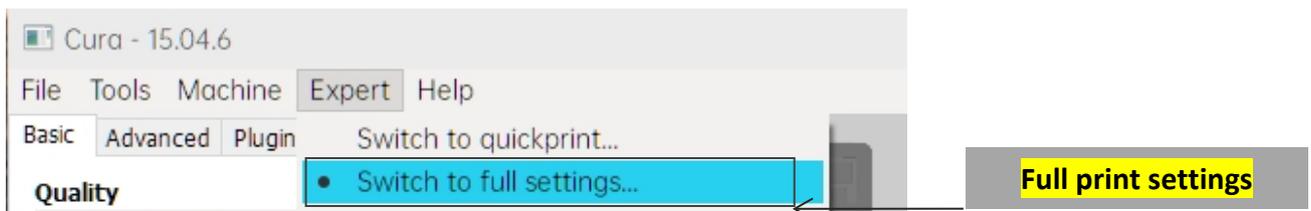
After clicking the Cura installation program, the page will pop up, and you can always click the next step to complete the installation.(set the required path in the preinstallation window: it is better to install it in English directory)

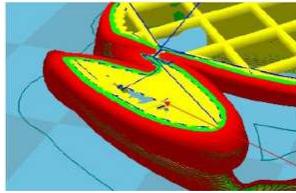




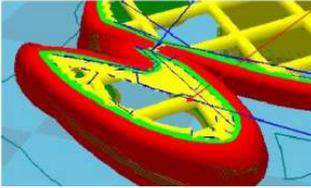
2: slice setting for printer

Open Cura's interface to modify all the control parameters of the printing process.





Bottom/Top thickness:1mm



Bottom/Top thickness:0.6mm

File Tools Machine Expert Help

Basic Advanced Plugins Start/End-GCode

Quality

Layer height (mm) 0.2

Shell thickness (mm) 0.8

Enable retraction

Fill

Bottom/Top thickness (mm) 0.6

Fill Density (%) 20

Speed and Temperature

Print speed (mm/s) 40

Printing temperature (C) 190

Support

Support type None

Platform adhesion type Brim

Bottom/top thickness:

If the filling density is less than 20%, the thickness of 0.6 mm is very easy to cause holes on the top, and the value of 1 mm is generally better

Fill density:

If high strength is not required, 10% is enough. For high strength, increase the filling ratio, but the printing time will increase

Print speed:

This is the default speed. If the shell and filling speed are not set separately, the printing time is not directly proportional to the speed. If the speed is too fast, printing will easily cause quality problems. Generally speaking, 40mm / S is a better speed, According to the speed specifications of your printer

File Tools Machine Expert Help

Basic Advanced Plugins Start/End-GCode

Quality

Layer height (mm) 0.2

Shell thickness (mm) 0.8

Enable retraction

Fill

Bottom/Top thickness (mm) 0.6

Fill Density (%) 20

Speed and Temperature

Print speed (mm/s) 40

Printing temperature (C) 190

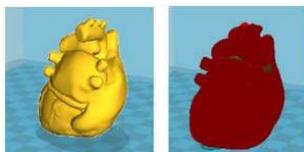
Support

Support type None

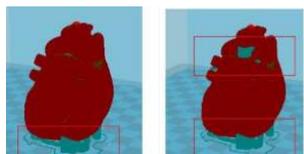
Platform adhesion type Brim

Print temperature:

PLA starts to melt at 180 °C, but the viscosity is relatively large, so it is difficult to squeeze. It is recommended to set the temperature at 190 °C -- 210 °C, printing speed faster or layer thickness thicker, and set the printing temperature a little higher



none



touching buildplate

everywhere

Bottom/Top thickness (mm) 0.6

Fill Density (%) 20

Speed and Temperature

Print speed (mm/s) 40

Printing temperature (C) 190

Support

Support type None

Platform adhesion type Brim

Filament

Diameter (mm) 1.75

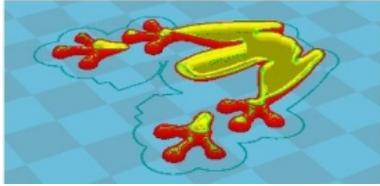
Flow (%) 100.0

Machine

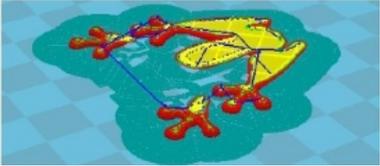
Nozzle size (mm) 0.4

Support type: (None, touching, everywhere).

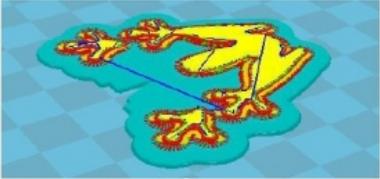
For the model with complex structure, it is usually necessary to add support. Usually choose Everywhere



none



Brim



Raft

Quality

Layer height (mm)

Shell thickness (mm)

Enable retraction ...

Fill

Bottom/Top thickness (mm)

Fill Density (%) ...

Speed and Temperature

Print speed (mm/s)

Printing temperature (C)

Support

Support type ...

Platform adhesion type ...

Filament

Diameter (mm)

Flow (%)

Platform adhesion type: (Brim, Raft, None.)

Normally choose Brim help the model stick platform well.

You can also choose None if the platform leveling very well.

Default is OK

0.3mm It's to make the model easier to peel off

The minimum printing time of each layer, When the real printing time of the layer less Than 3 sec, the speed will be lower than the set speed.

File Tools Machine Expert Help

Basic **Advanced** Plugins Start/End-GCode

Retraction

Speed (mm/s)

Distance (mm)

Quality

Initial layer thickness (mm)

Initial layer line width (%)

Cut off object bottom (mm)

Dual extrusion overlap (mm)

Speed

Travel speed (mm/s)

Bottom layer speed (mm/s)

Infill speed (mm/s)

Top/bottom speed (mm/s)

Outer shell speed (mm/s)

Inner shell speed (mm/s)

Cool

Minimal layer time (sec)

Enable cooling fan ...

Configuration according to the 3d printer

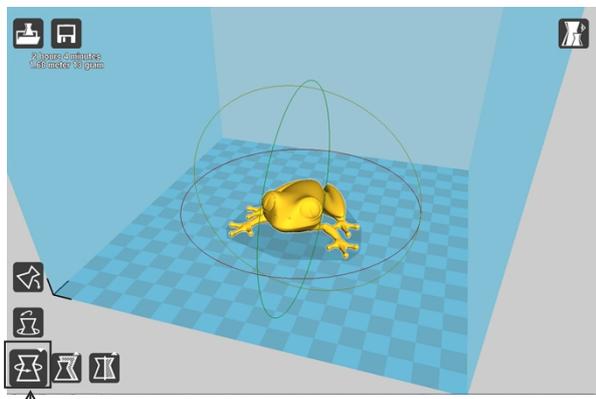
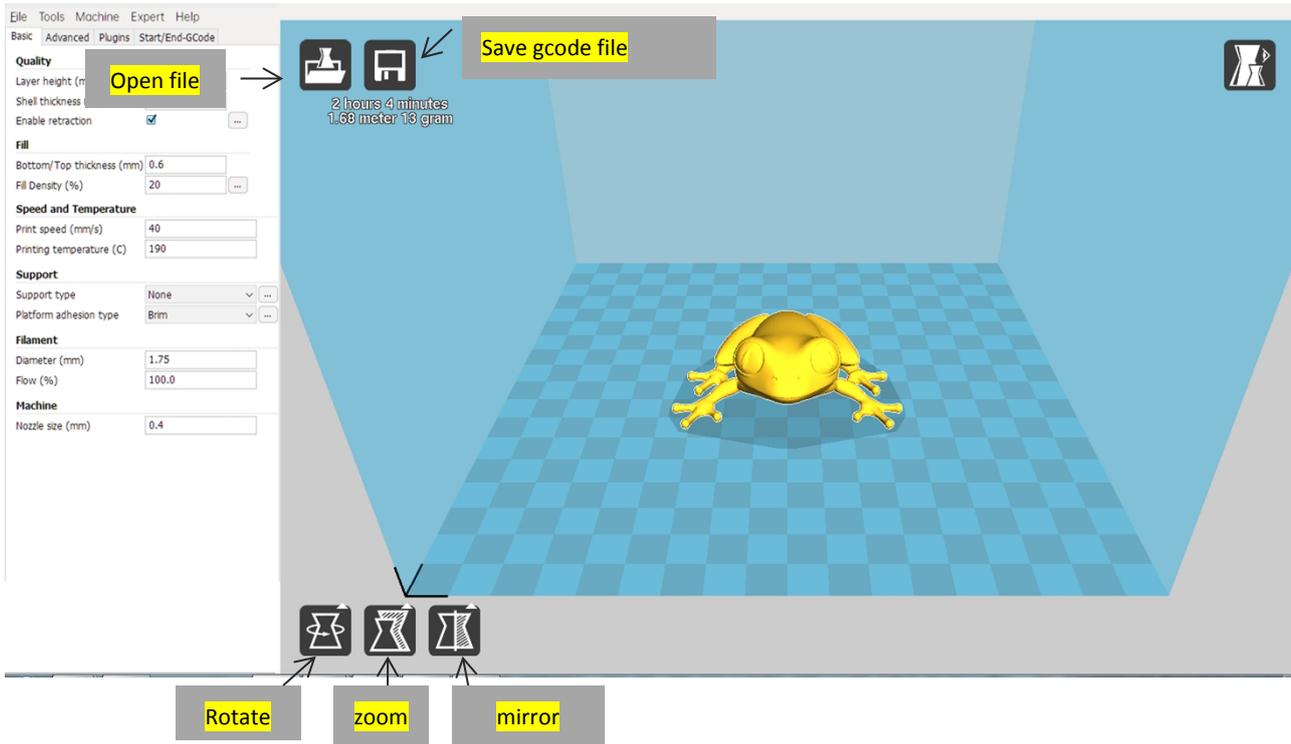
Default 100%

0 Indicates that the global default speed is used, If the global speed is > 60mm / s, the shell speed should be less than 60mm / s to ensure the printing quality

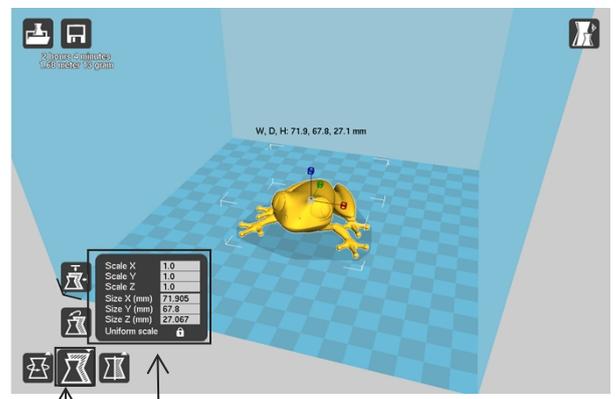
Use cooling fan

3: slice and save

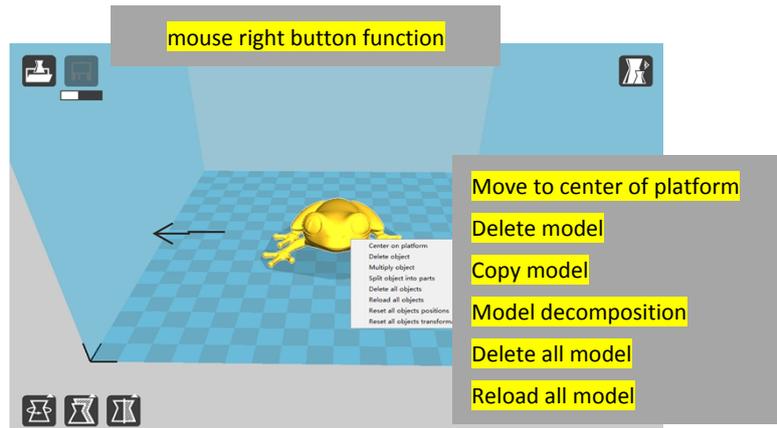
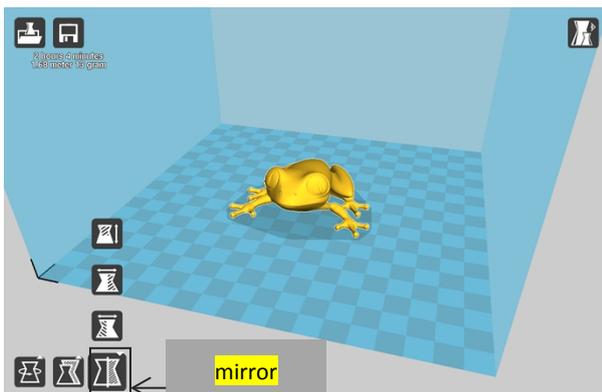
After all the parameters are set, load a 3D model file, save the sliced gcode file to the SD card, and then start printing.



1. Click to rotate
2. Pull the rotation control circle. The unit is 15 ° by default



- zoom By scale or size



The image shows a vertical list of three save options in a 3D printing software interface. Each option consists of two icons, a callout box with an arrow pointing to one of the icons, and a text box explaining the option. Below each option, estimated printing time, filament length, and weight are displayed.

- Option 1:**
 - Icons: A printer and a computer monitor.
 - Callout: "Save the gcode file to computer when SD card not available"
 - Text: "1 hour 24 minutes", "8.87 meter 26 gram", "1.32", and "保存按钮" (Save button).
 - Explanation: "1hour24minutes: means the Estimated printing time" and "8.87meter 26ggram: means filament need 8m, weight 26g"
- Option 2:**
 - Icons: A printer and an SD card.
 - Callout: "Save the gcode file to SD card."
 - Text: "2 hours 1 minutes", "10.40 meter 31 gram"
- Option 3:**
 - Icons: A printer and a printer with a network symbol.
 - Callout: "Online print mark"
 - Text: "2 hours 1 minutes", "10.40 meter 31 gram"

Thanks for choosing EasyThread, we will try our best to serve you

ShenZhen Easythread technology CO.,LTD

Address : No.A2 , 2nd Floor , Avant Low Carbon Innovation Park , No1034 of Longgang Road , Pingdi Street , Longgang District , ShenZhen , China

Website : www.easythread.com

Email: info@easythread.com