kywoo3D

Ultimaker Cura 4.5

Installation & Operation Manual

Contents
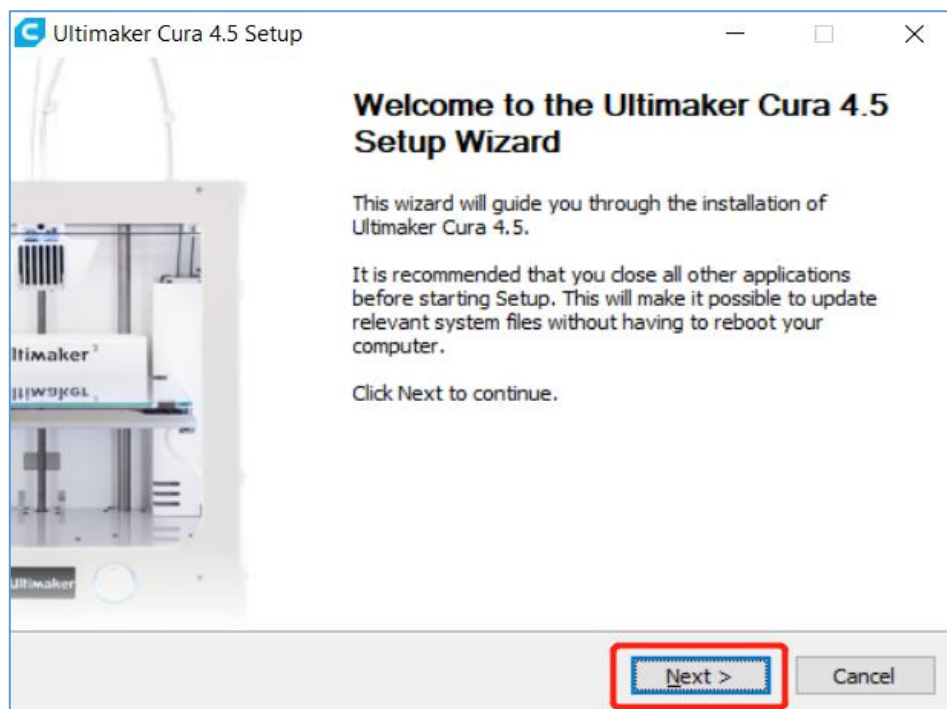
Chapter 1: Installation and Settings

1.1 Install Cura
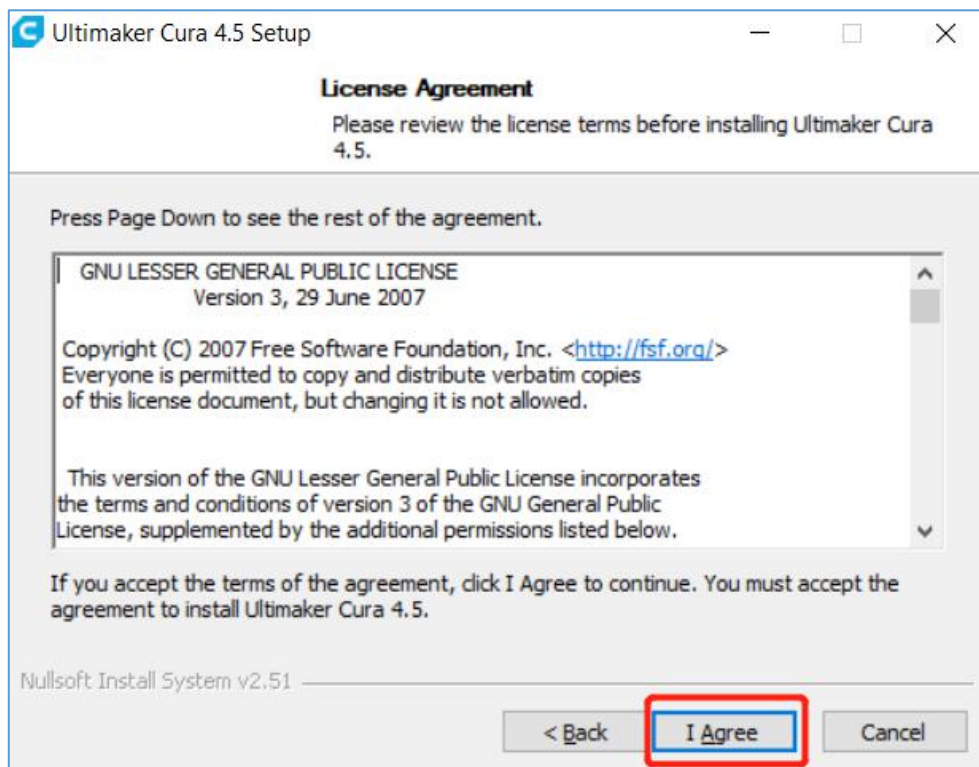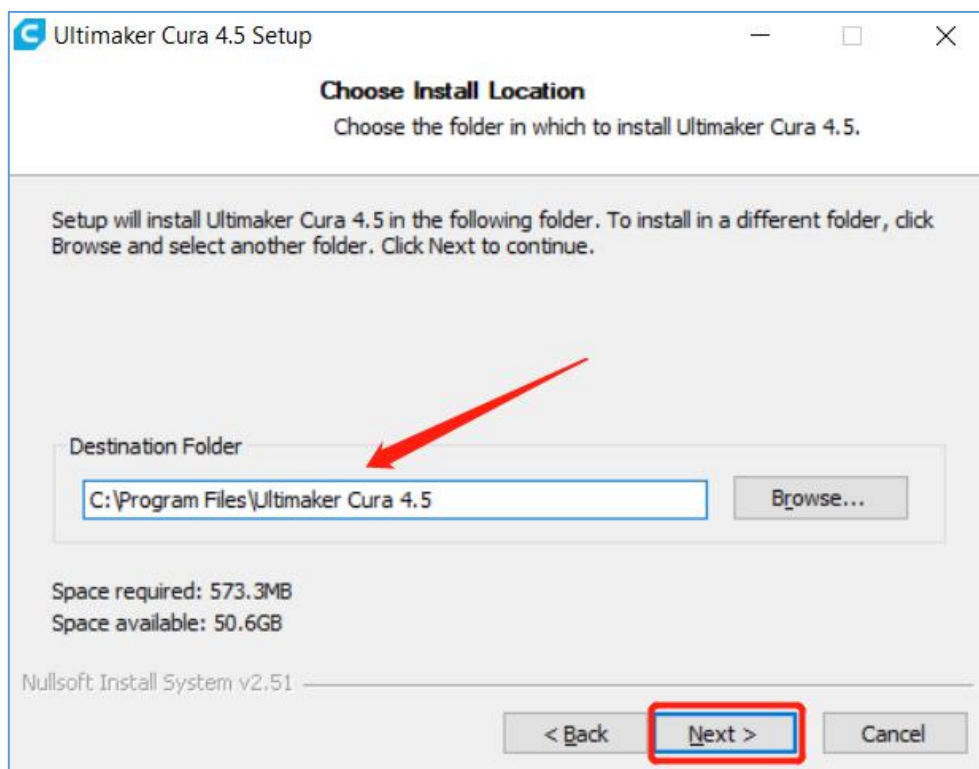
ⅰ. Double click to open "Ultimaker.Cura-4.5.0" package.
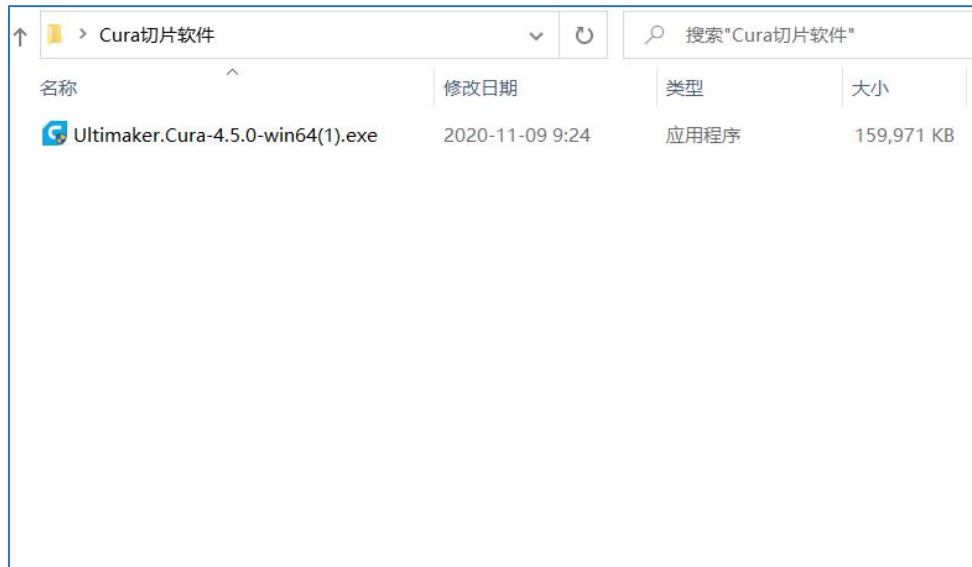
ⅱ. Click "Next"→ "I Agree".

iii . Select the installing address. You can use the default address or custom the address as you wish.

ⅳ. Select related components. It's recommended to select by default and click "install".
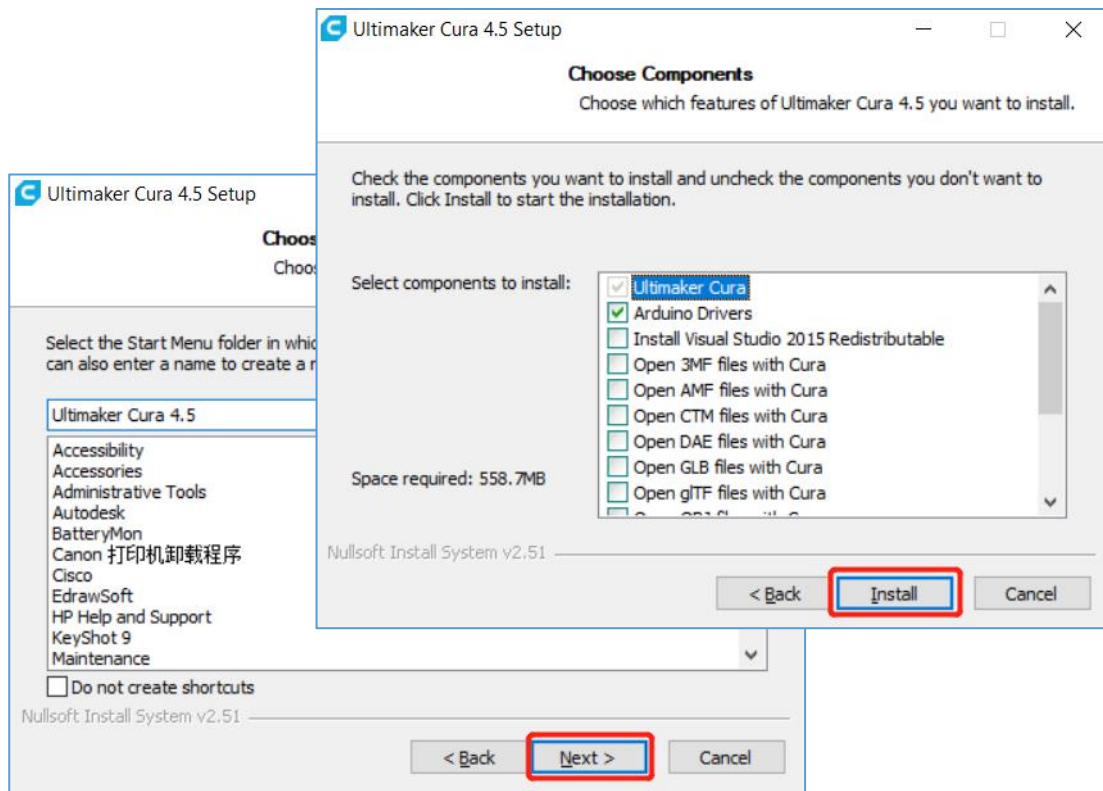
ⅴ. Install completed.

1.2 Language & Printer Setting

ⅰ. Double click to start Ultimaker Cura 4.5.
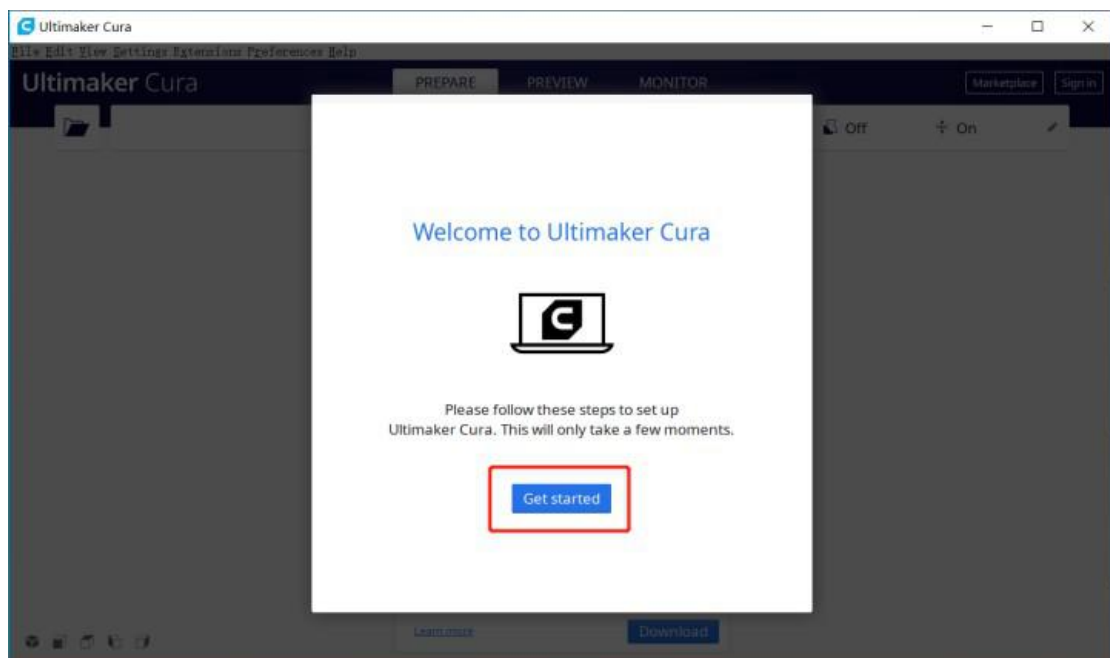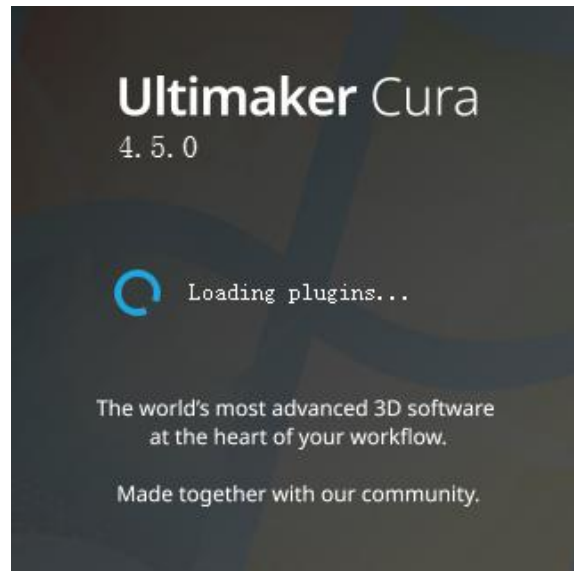
ⅱ. Click "Agree"→"Next".
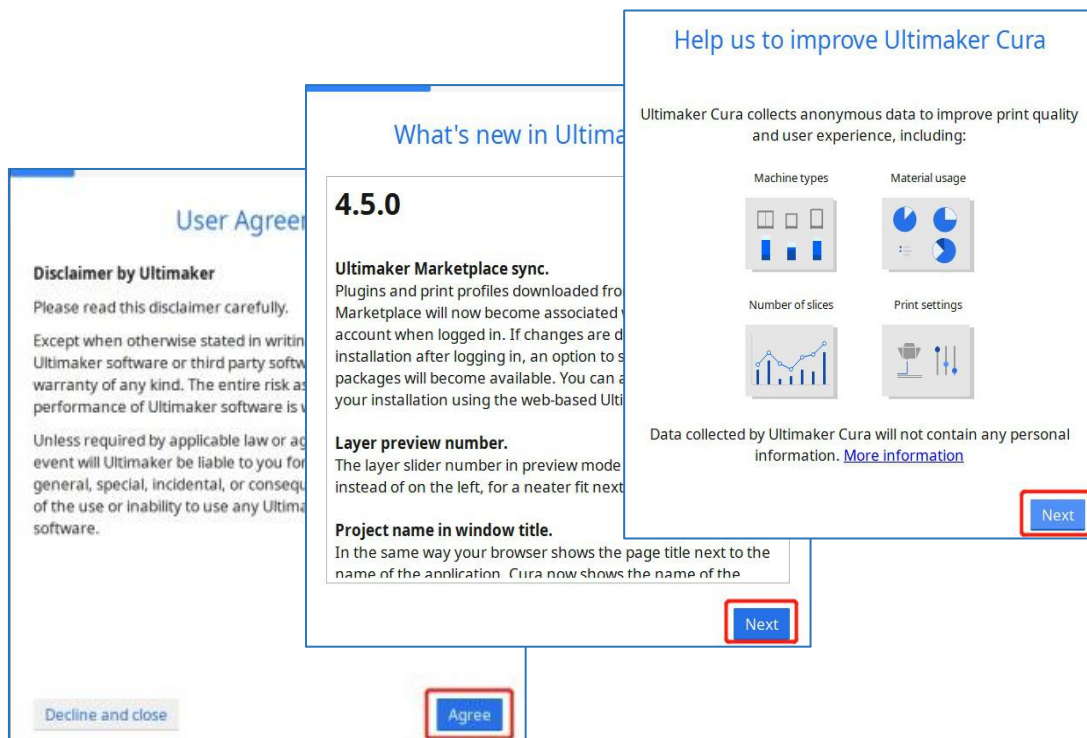
ⅲ. Select custom printer

Choose "Add a non-networked printer"→"Custom"→"Custom FFF printer", name the printer as "Kywoo Tycoon" and click "Next".
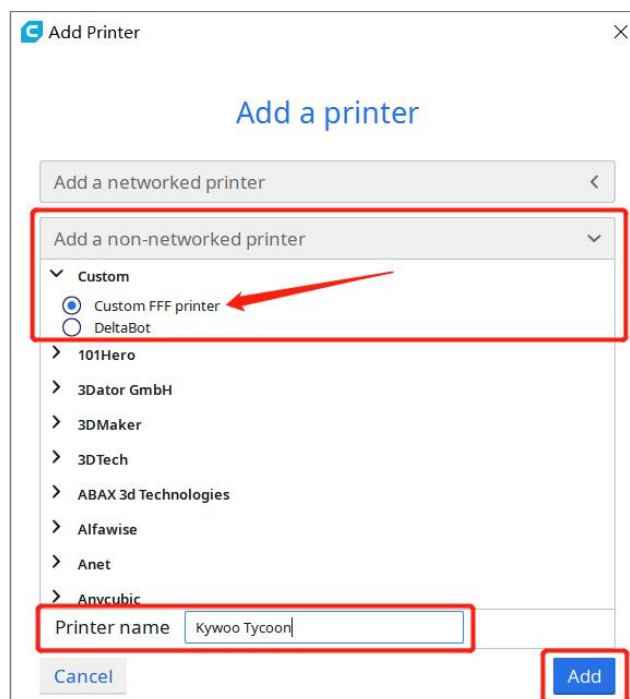
Set the the size of the printer: 240mm*240mm*230mm, tick "Heated bed" , then set the Nozzle size: 0.4mm, the Compatible material diameter: 1.75mm, and click "Next".
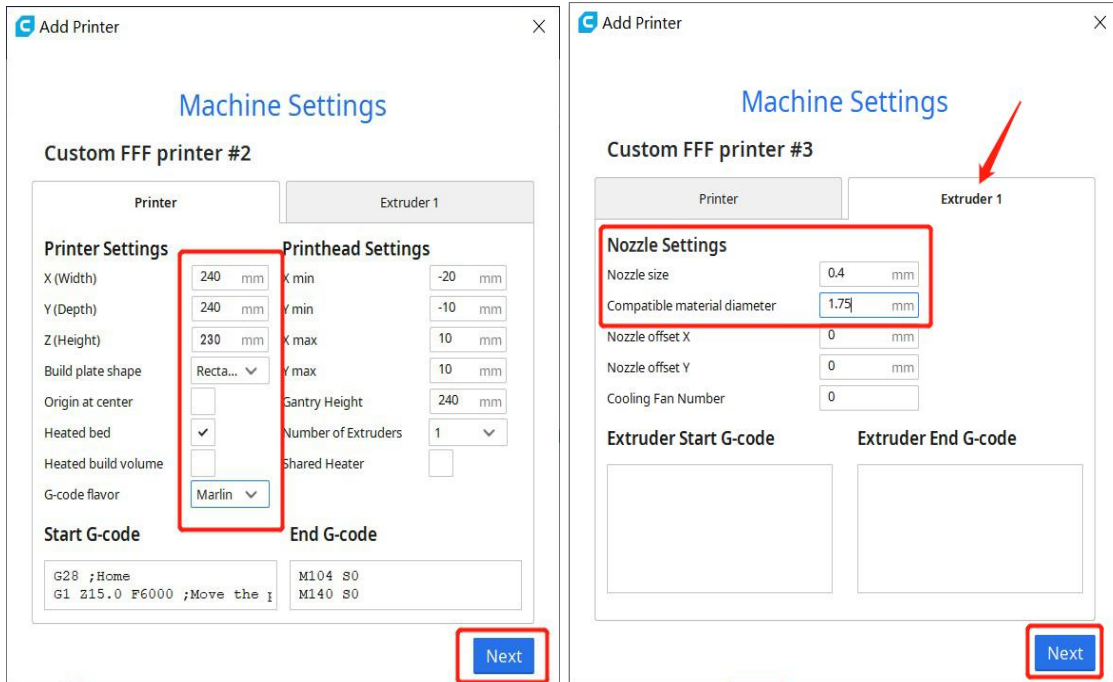
The main screen of Cura is as shown in the following picture.

8
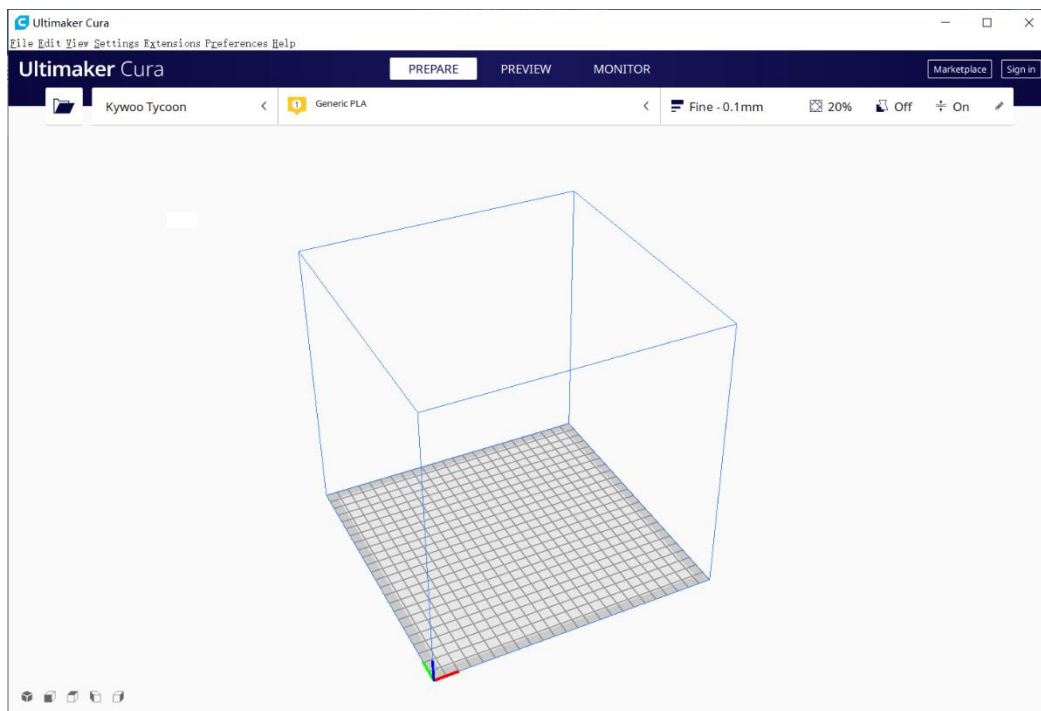
ⅳ. Language Setting

Click "Preferences" in the upper-left corner of the screen, then click "Configure Cura…" to next page, then language can be set in "General" tab.
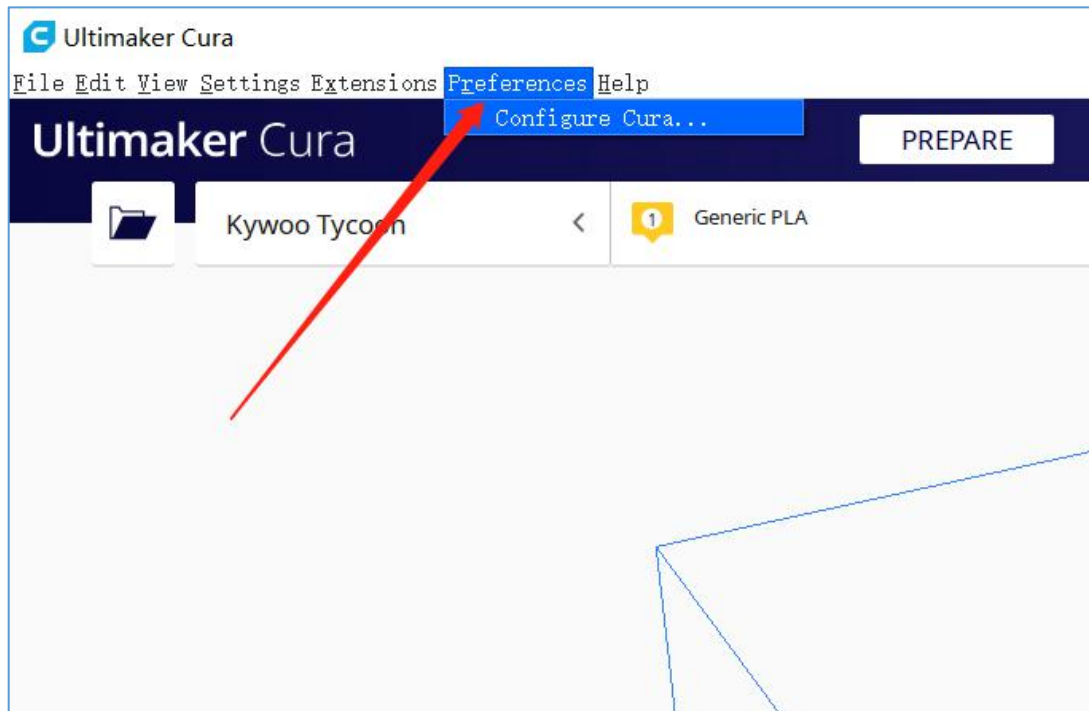
Chapter 2: Operation

2.1 Open File

Open a 3D model file. Cura supports STL, 3MF, AMF, OBJ files, etc.
①: You can click "file"→ "Open File"→select the file.
②: Or drag and drop the file in the screen of Cura.
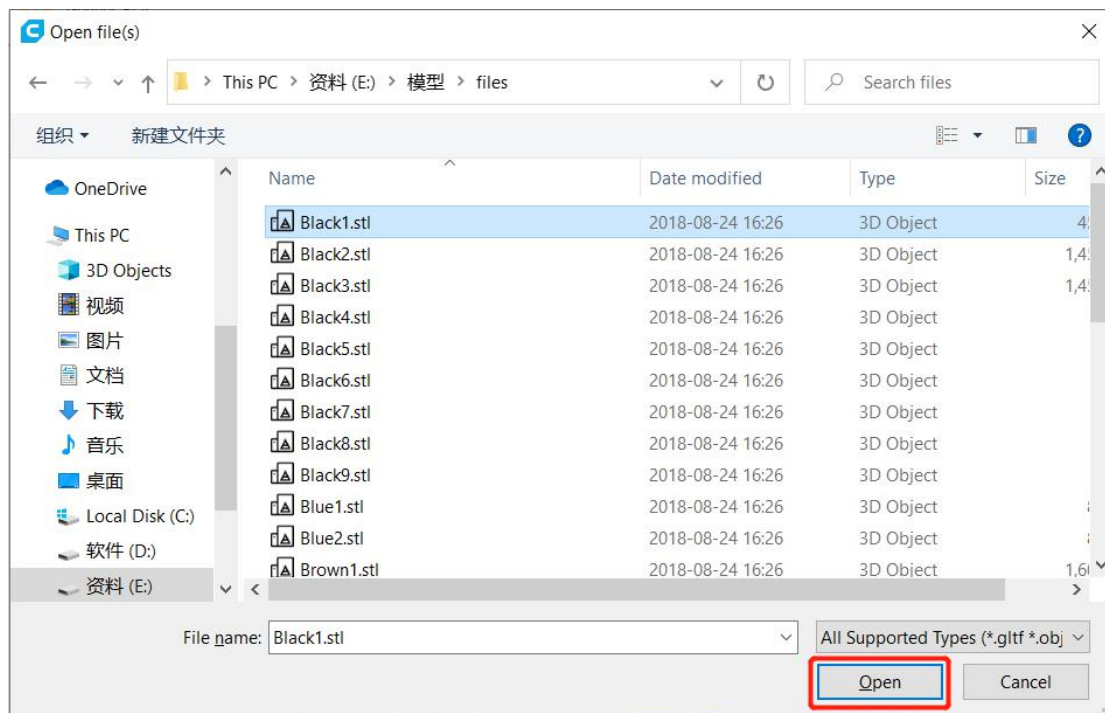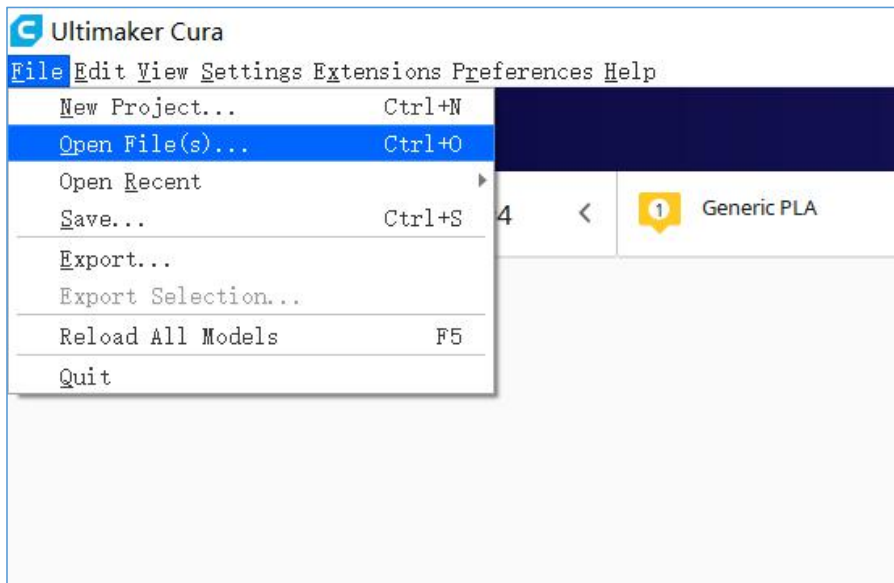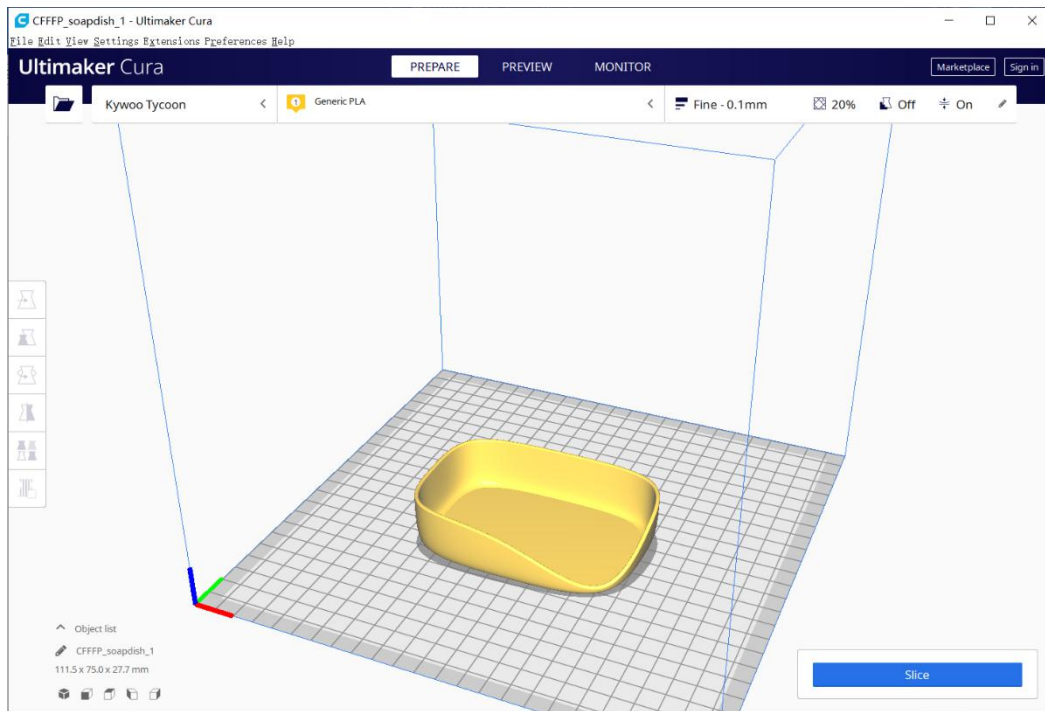
2.2 Edit the model

*You need to click and check the model before editing it.

ⅰ. Move the model. Click the first icon on the left, then the model can be dragged to from top to bottom and move around, or you can input a value to X, Y, and Z.
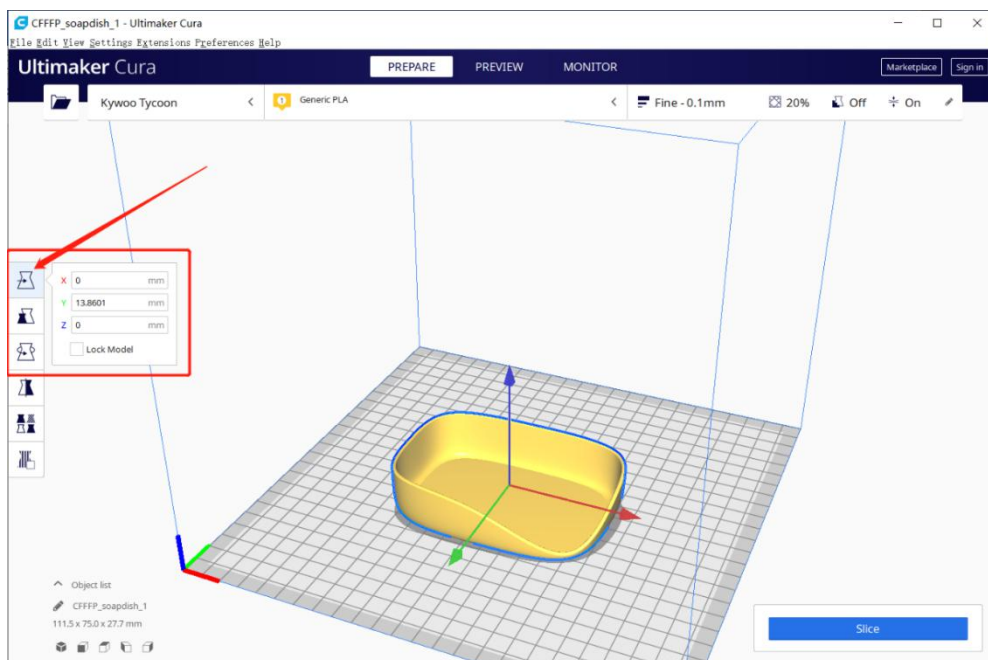
ⅱ. To scale the model, click the second icon on the left to assign a value to one of X/Y/Z or fill in the scale ratio. You can select "Snap Scaling" or "Uniform Scaling".
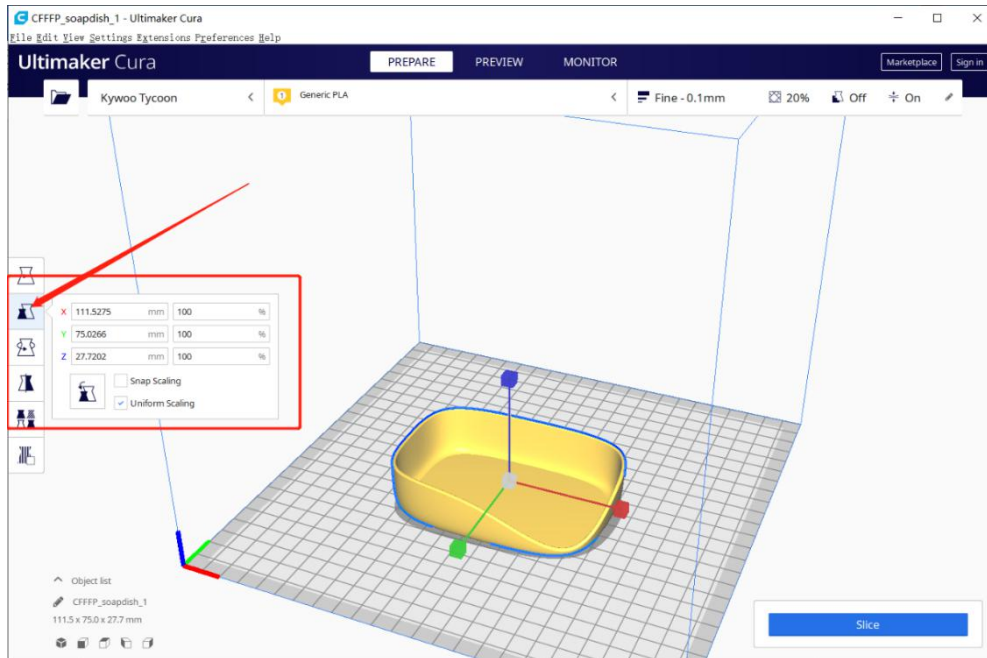
ⅲ. Rotate the model. Click the third icon on the left, and drag the red, green and blue circles to rotate the model around X-axis/Y-axis/Z-axis. Besides, you can also rotate the model to make its one side fit the platform.
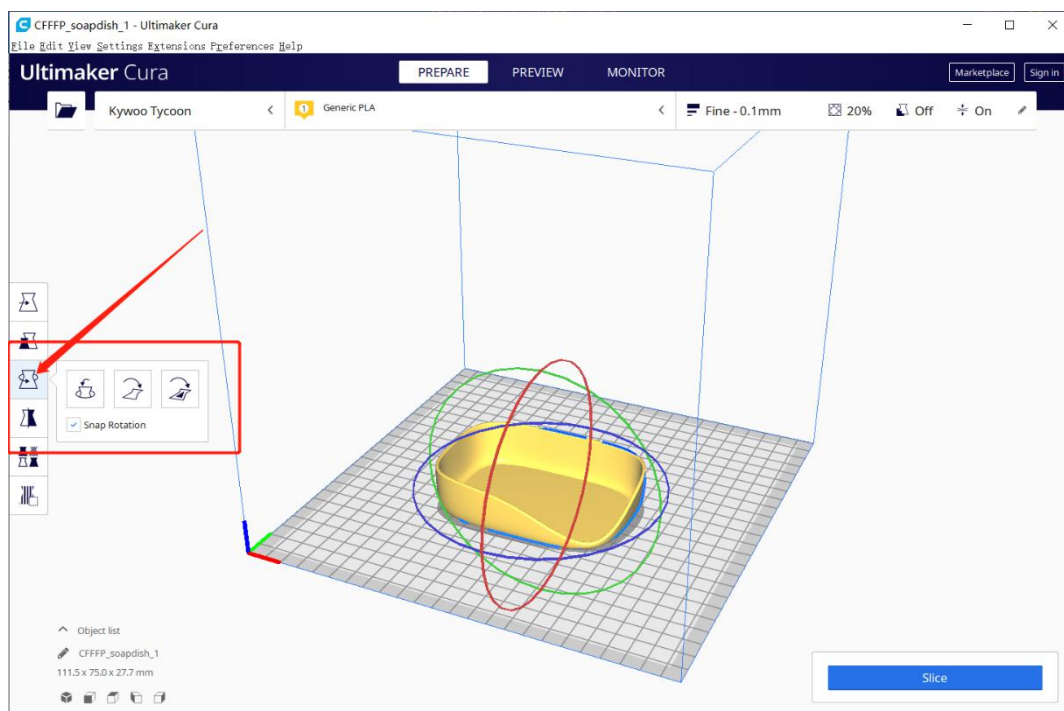
ⅳ. Mirror the model and change the model's display. Click the fourth icon on the left to mirror the model around a certain axis and the fifth icon to change the model's display.
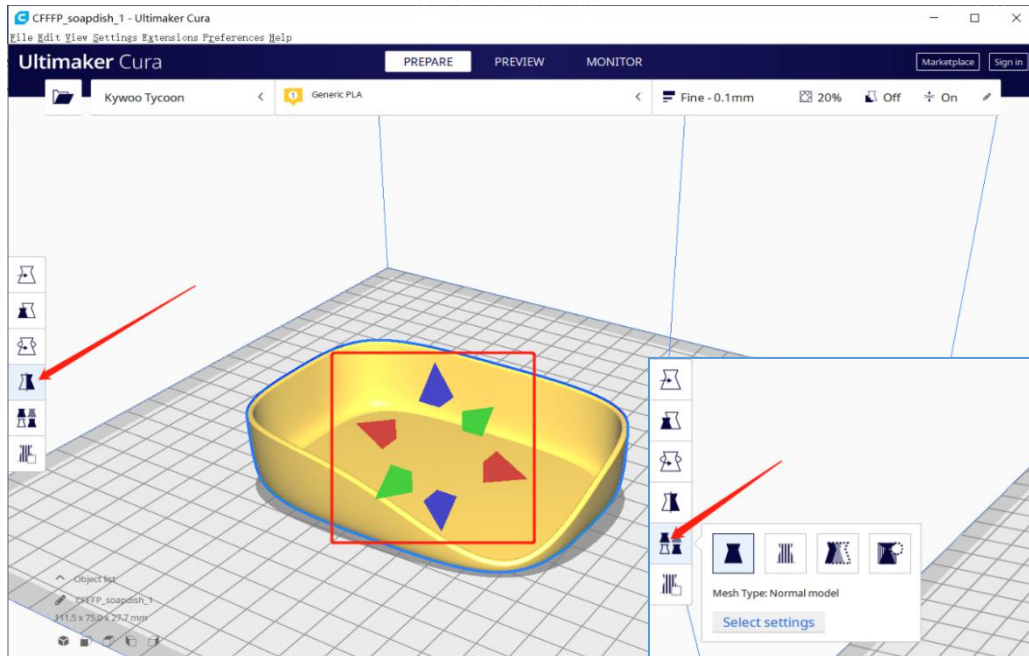
2.3 Print Settings

ⅰ. The icons on the upper-right corner are for a quick setting to layer height and infill. Click "Custom" to enter the complete custom parameter settings screen.
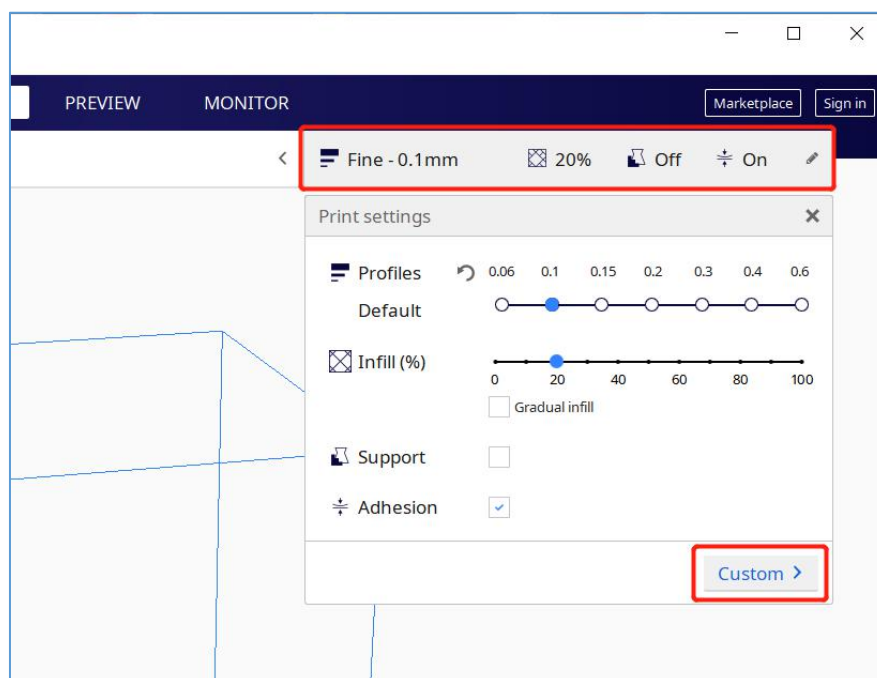
In the custom page, you can choose one default profile, Cura will automatically match other corresponding settings, or manually set in the listed parameters.

ⅱ. The introduction of listed parameters

Quality: also layer height, the height of every printed layers. It is the parameter that most affects the printing quality. The general setting range is 0.1mm-0.3mm. The smaller the layer height, the better the surface quality and the longer the printing time.
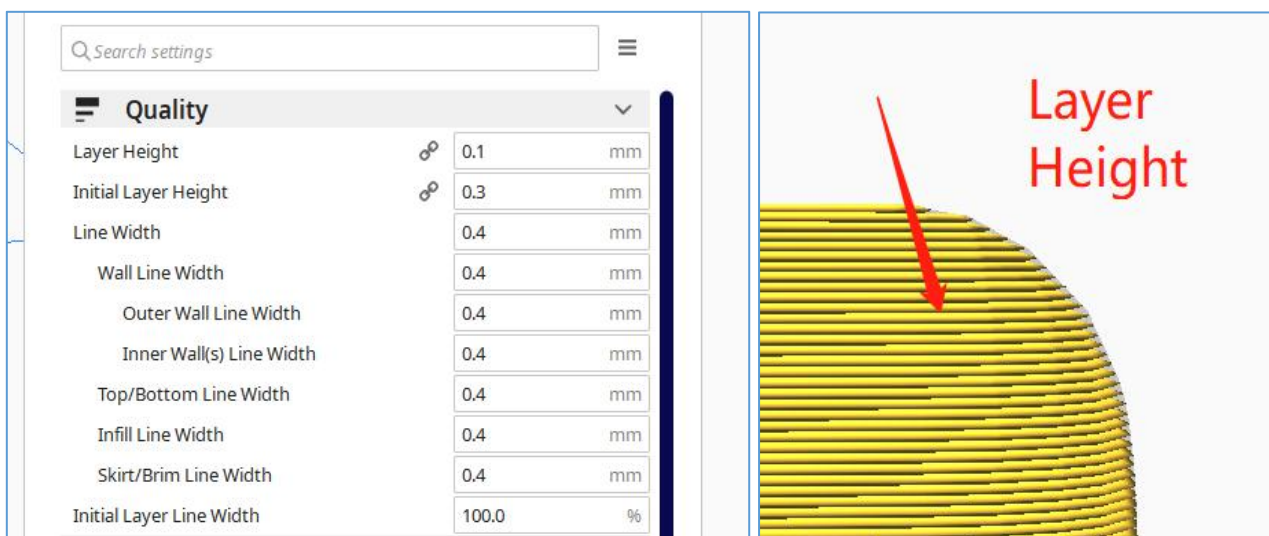
Shell: the setting of the outer wall of the model, mainly the wall thickness, which affects the strength and impact resistance of the 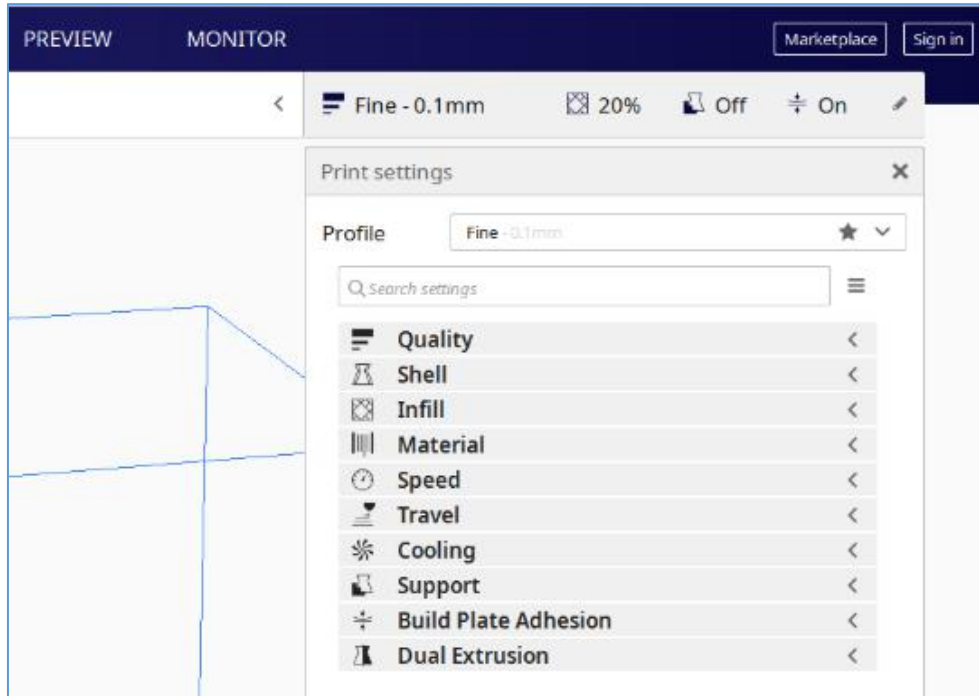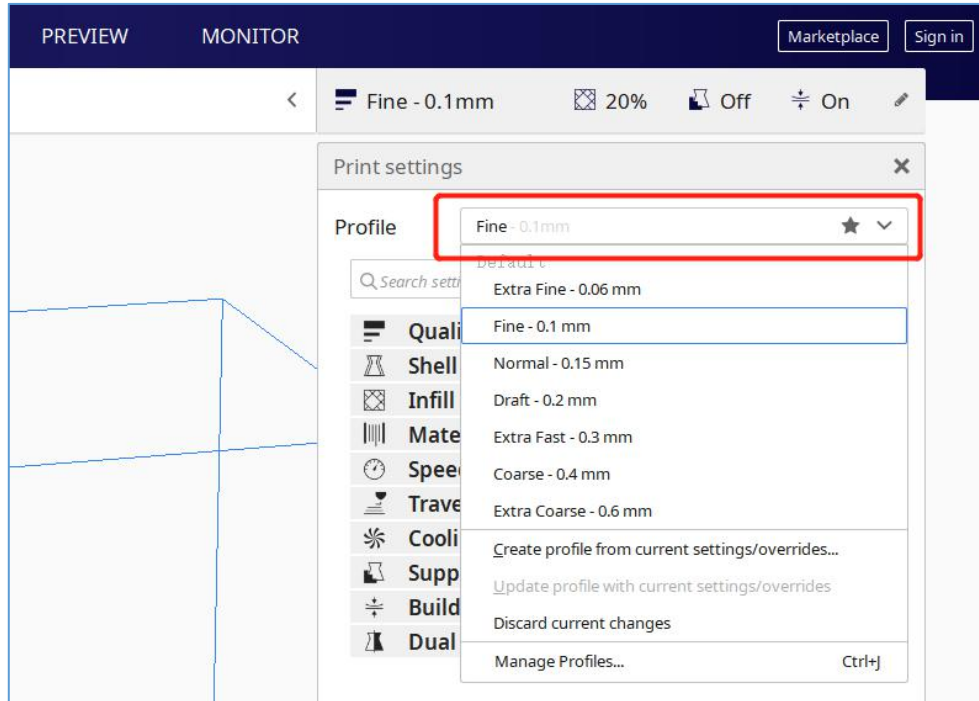model. The greater the wall thickness, the higher the strength, and the longer the printing time. Generally, it is set as a multiple of the nozzle diameter, such as 0.8mm; 1.2mm.
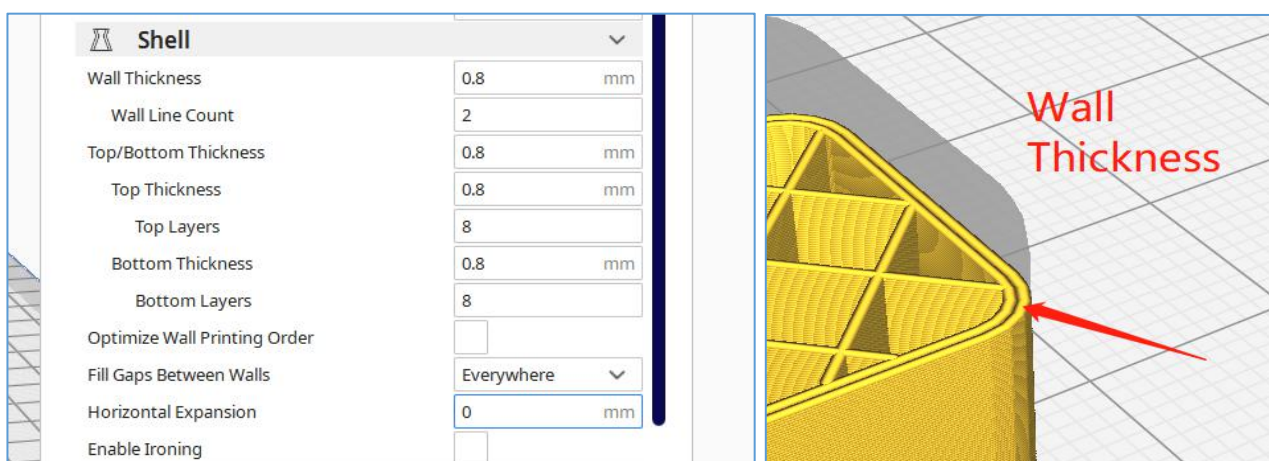
Infill: the infill density inside the model. 0% means hollow and 100% means solid. It also determines the strength of the model. The more the infill, the higher the

strength. It's generally set as 10%-30%. You can also set the infill pattern, generally, the grid is used.
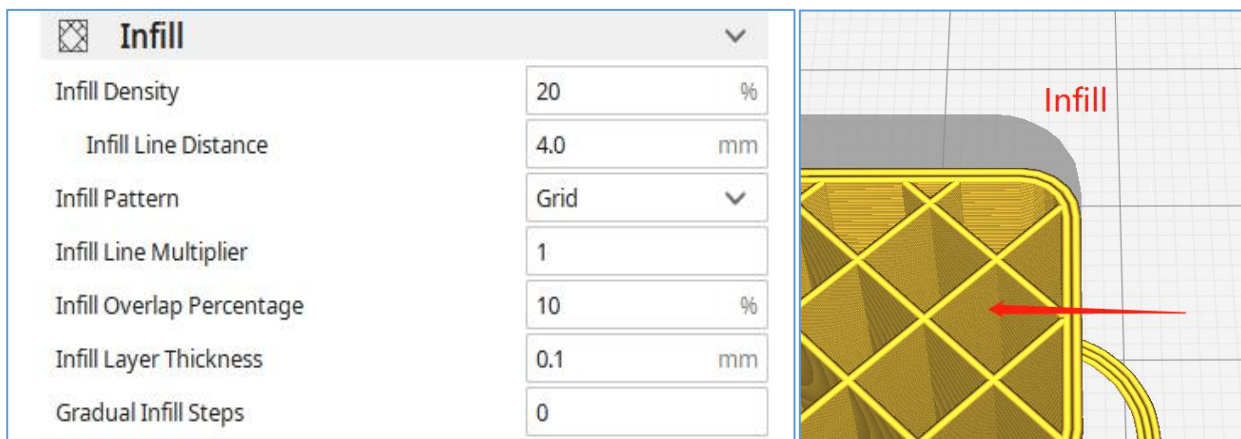
Material: mainly the temperature of nozzle and hotbed need to be set. They are often set as:

PLA: Nozzle 195℃-210℃, hotbed 50℃-55℃

ABS: Nozzle 230℃-240℃, hotbed 95℃-105℃

Speed: the parameter adjustment of various speeds in the printing process, mainly the print speed, generally 50-80mm/s.

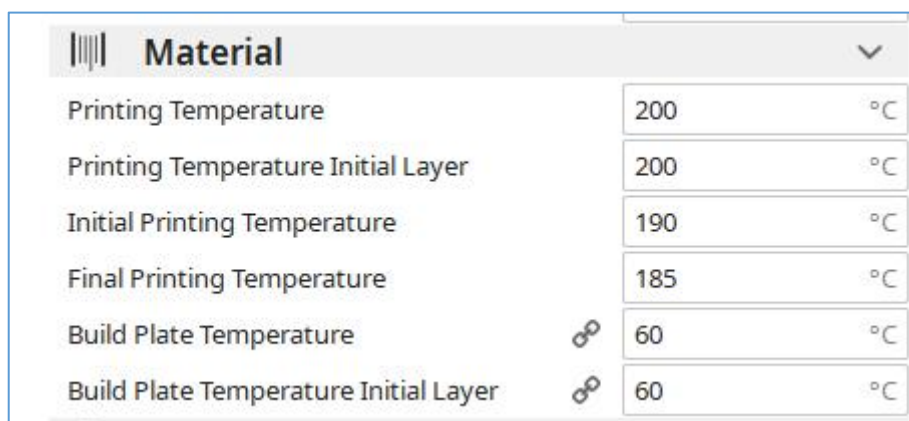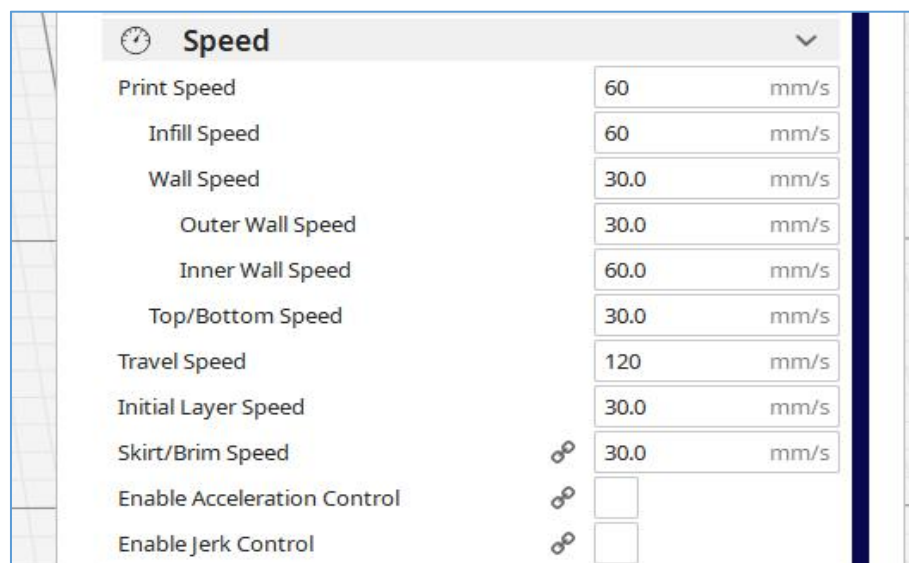*Travel:* mainly the retraction needs to be set. Please make sure to tick "Enable Retraction", because setting retraction can effectively avoid stringing and ensure the printing quality. Generally, the retraction speed is set to 90mm/s and the retraction distance to 9mm.

Travel		
Enable Retraction	✓	
Retract at Layer Change		
Retraction Distance	9	mm
Retraction Speed	90	mm/s
Retraction Retract Speed	90	mm/s
Retraction Prime Speed	90	mm/s
Retraction Extra Prime Amount	0	mm³
Retraction Minimum Travel	0.8	mm
Maximum Retraction Count	90	
Minimum Extrusion Distance Window	9	mm

Support: If the overhang angle of the model is greater than 60°, it is recommended to choose support. If some part of the model is completely overhung, you must add support. This option can edit the support density/pattern/angle. The placement can be set to touching build plate or everywhere support.

| Support		
Generate Support		✓
Support Placement		Everywhere
Support Overhang Angle		50 °
Support Pattern		Zig Zag
Support Density		15 %
Support Horizontal Expansion		0 mm
Support Infill Layer Thickness		0.1 mm
Gradual Support Infill Steps		0
Enable Support Interface		
Enable Support Roof		✓
Enable Support Floor		

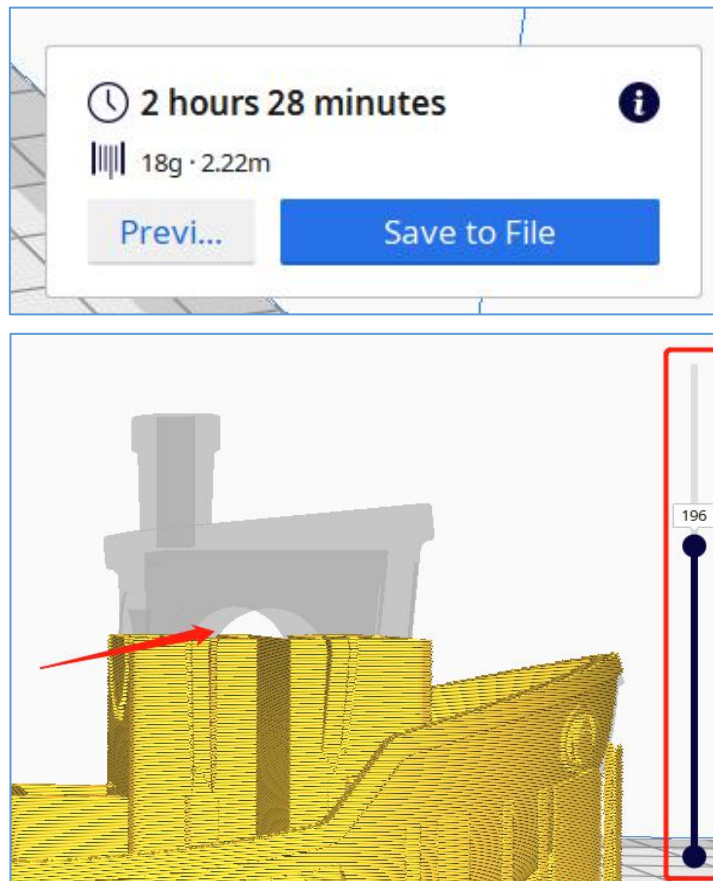***Build Plate Adhesion:*** Users can set Skirt, add Brim or Raft.
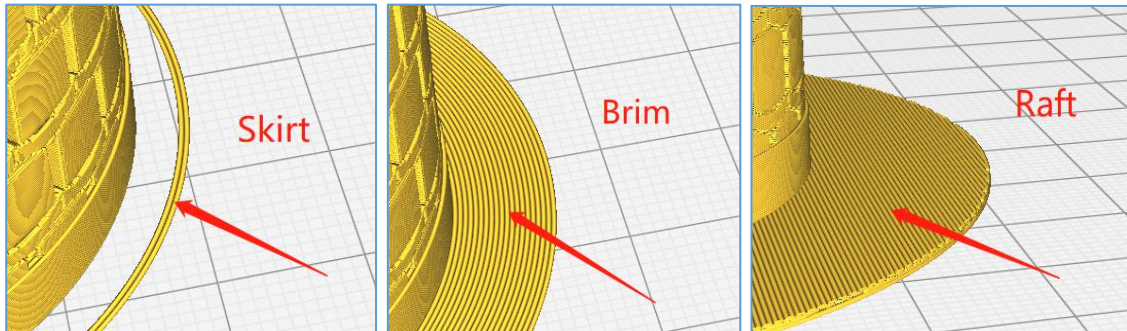
2.4 Export Sliced File

ⅰ. Start Slicing

After the printing parameters are set, click "Slice" button in the lower-right corner of the screen to slice the model. After the slice is completed, you can see the estimated printing time. Click "Preview" and drag the slider on the right to preview the effect of each ayer.
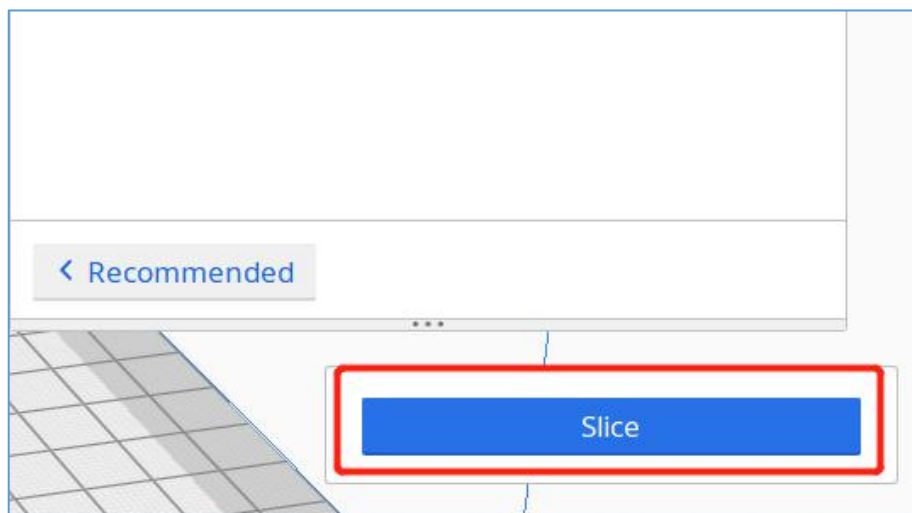
ⅱ. Save Gcode File

After all parameters are set, the file can be exported for printing. Click "Save to File" in the lower-right corner to export. In the pop-up dialog box, you can modify the file name and save.