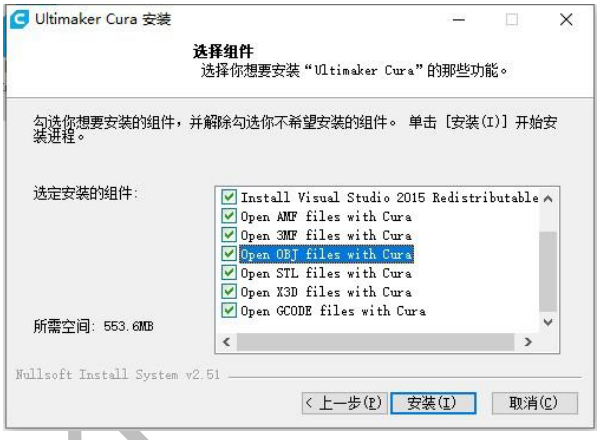
SLUBAN CURA Configuration tutorial

Cura is one of the earliest open source slicing software, slicing speed is fast, user-defined parameters are rich, strong development, here only teach basic operation, if you want to know more, please study by yourself.

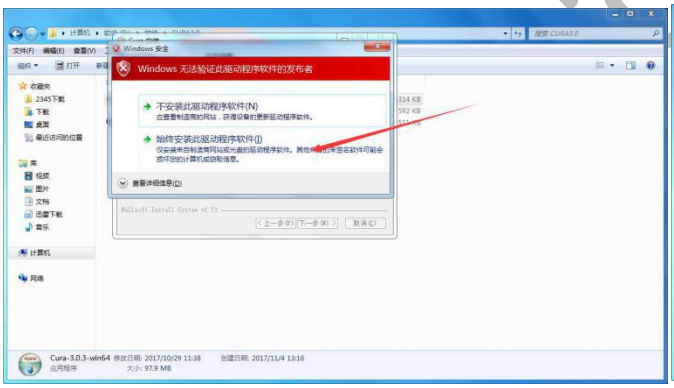
First download the software and configuration file, please install and configure according to the tutorial.

1Install Cura first

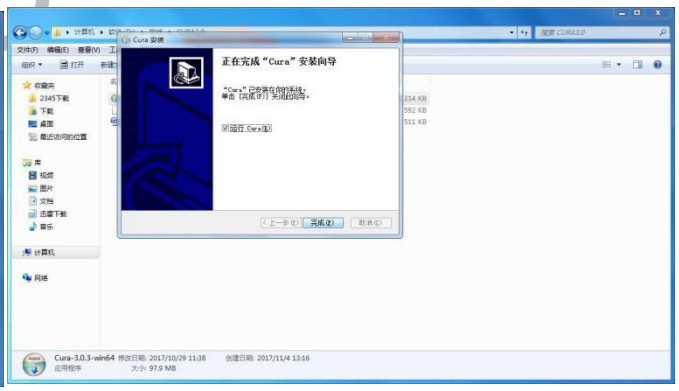
Associated format, Check all the options



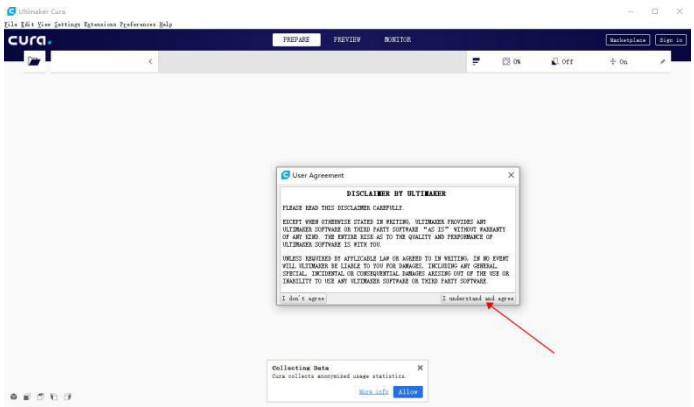
Prompt for driver installation, Select always install



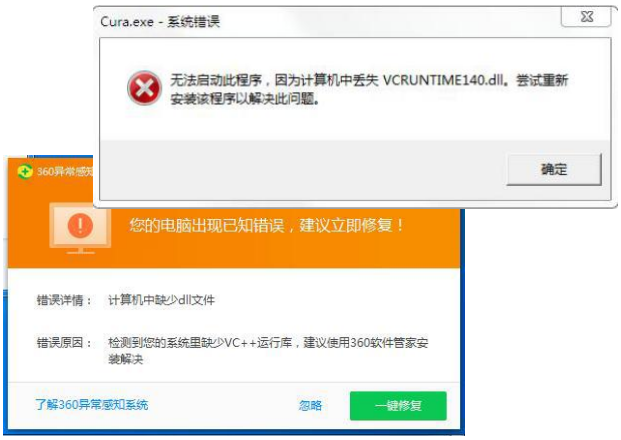
Open after installation, If no shortcut icon is generated, find it in the installation directory



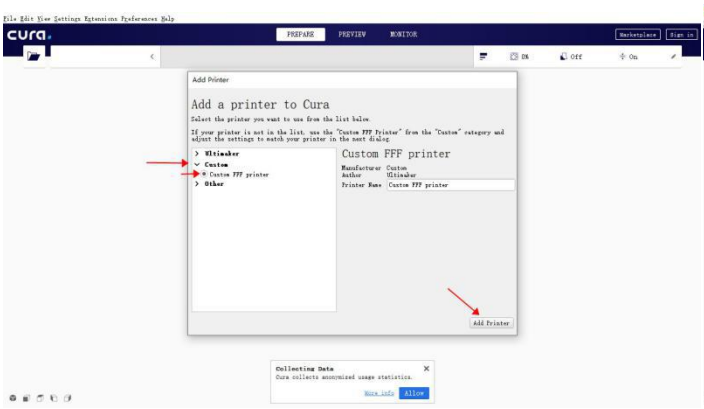
Pop up the dialog box and click "agree"

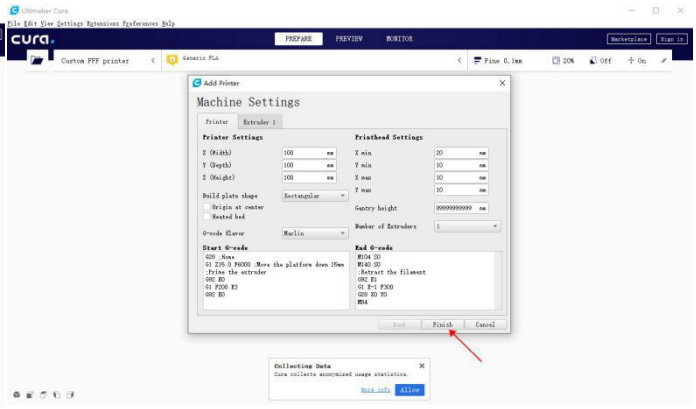


If this prompt appears, install the corresponding runtime

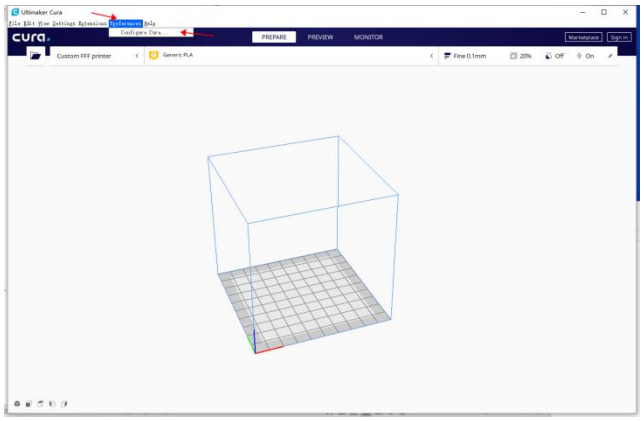


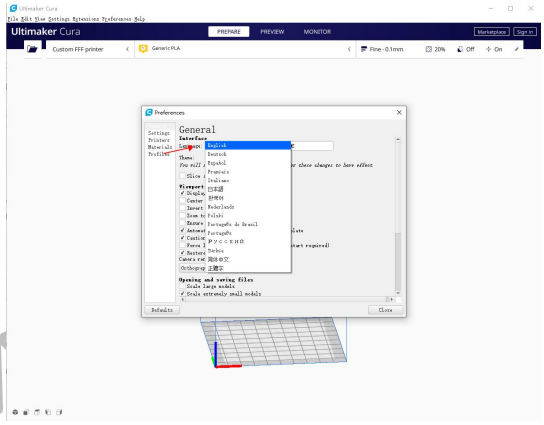
2 Configure the model and select the middle custom FFF





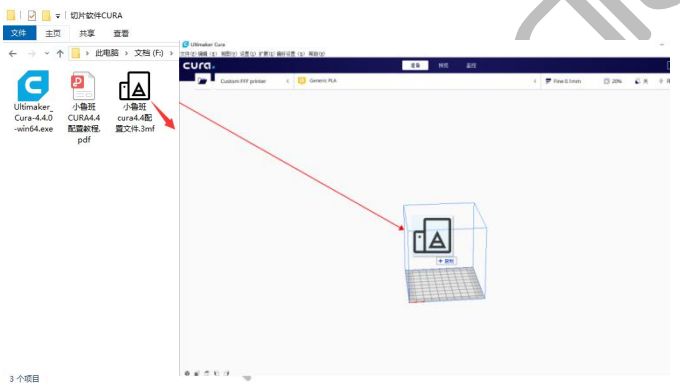
Choose the language, choose the language you want, go back to the interface, turn off the software and restart

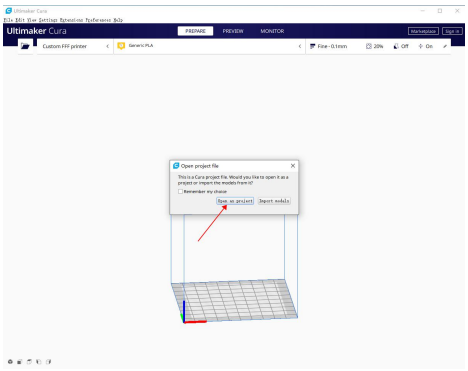




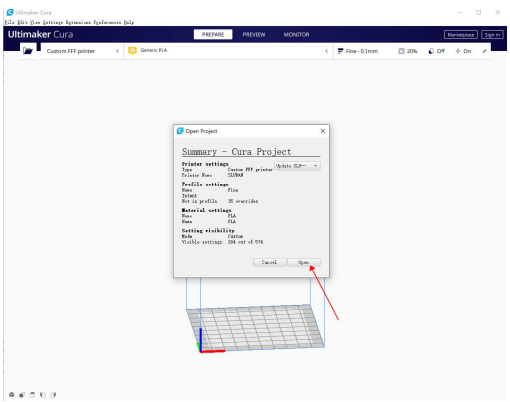
3 configuration parameter

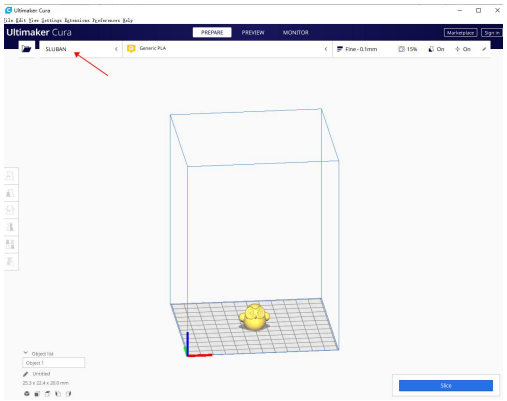
Drag the configuration file of the folder (suffix. 3mf) directly into the software Open as project





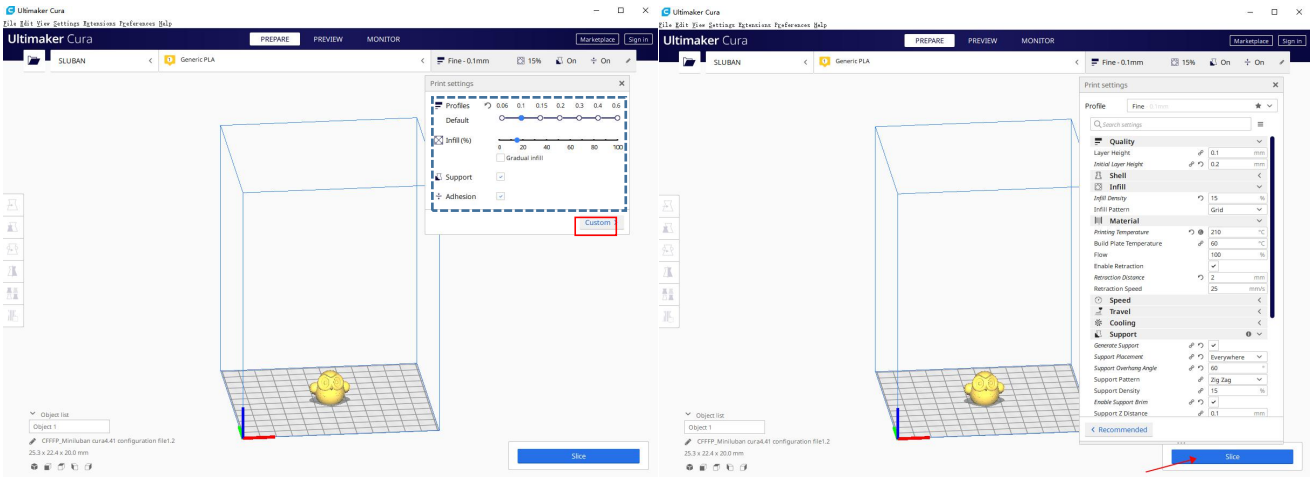
Open If the machine name shows "SLUBAN", it means the configuration is ready





4 Basic parameter setting

Click on the upper left status bar, （Don't change the settings in the dotted box）, Click "custom" directly, You can modify various parameters, There are comments on the parameters



Common parameters

Quality-Layer Height It refers to the height of each layer printed, Usually 0.2, exquisite 0.1, fast0.3

Infill-Infill Density Usually 10-20% is suitable, 100% is solid

Material-Printing Temperature Generally, PLA uses 190-210℃, Different materials from different manufacturers have different optimal temperature, and they can be tested according to the situation

Support After checking, support can be generated automatically, Models with overhanging structure or inclination greater than 45 degrees need to be √

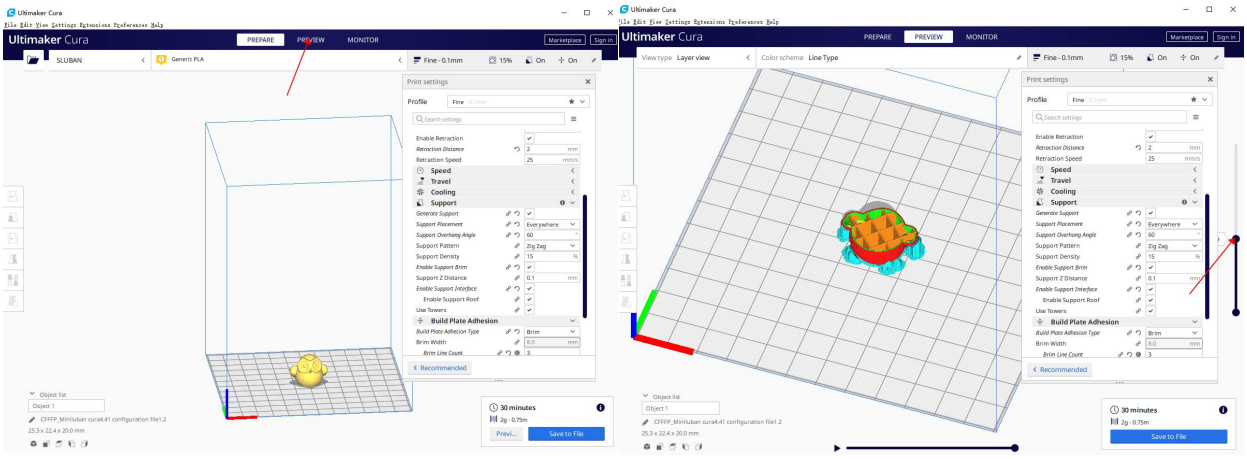
Build Plate Adhesion 'Brim' can increase model adhesion, 'Raft' can generate pedestal

After setting the parameters, it will slice automatically (Or click "Slice" in the lower right corner), After slicing, Click 'Save to file', Then import the saved file into SD card. Printing time and material consumption are shown at the bottom right (The time deviation is about 20%, The deviation of consumables is about 5%)

(The file name can only be English letters and Arabic numerals)

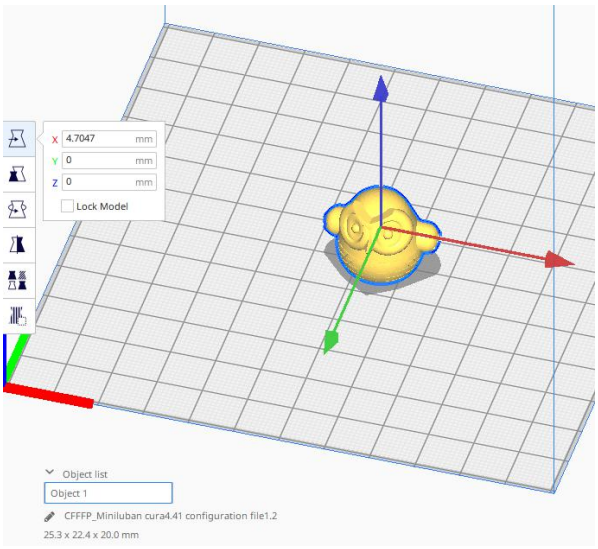
5 Slice Preview

Pull this point to check each layer of the model. Pull it to the first layer to check whether the model is on the platform. If it is not on the platform, printing will fail

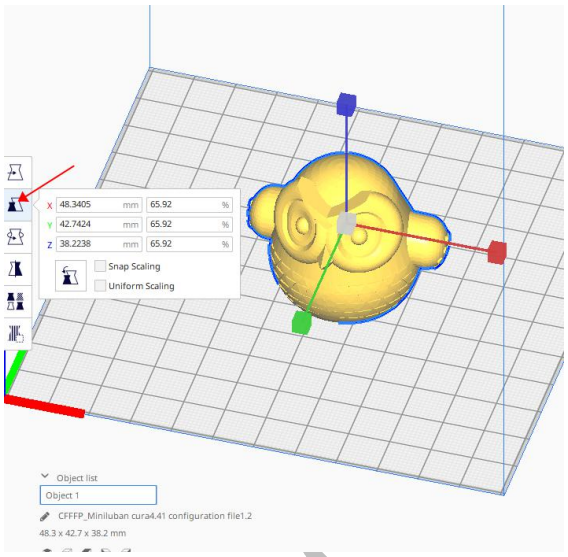


6 Model modification

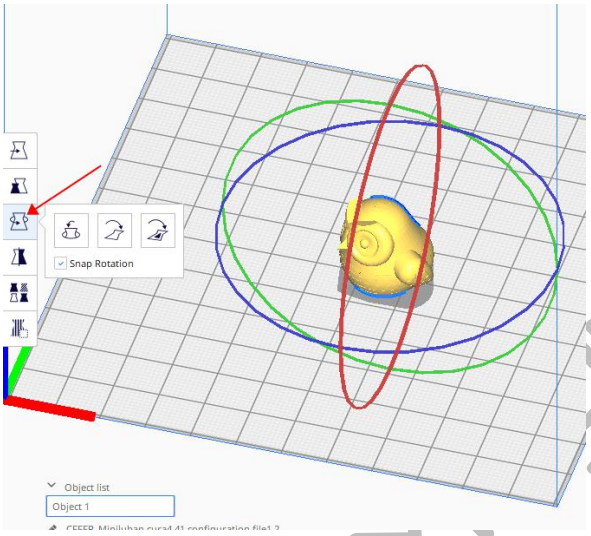
Move, change the position of the model



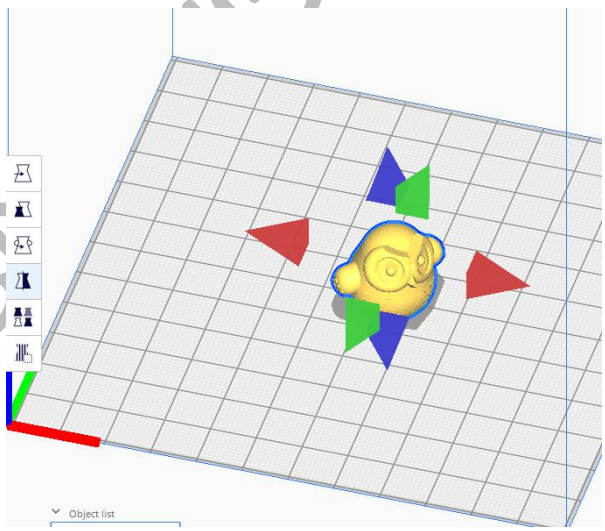
Zoom, change model size



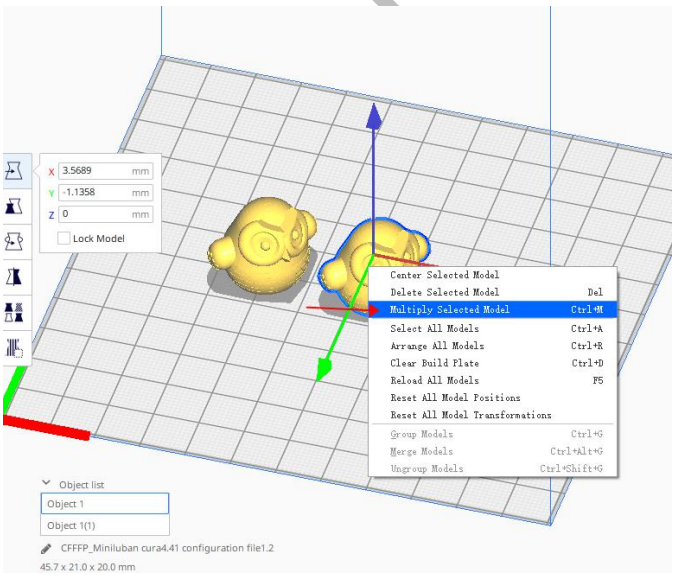
Rotating model



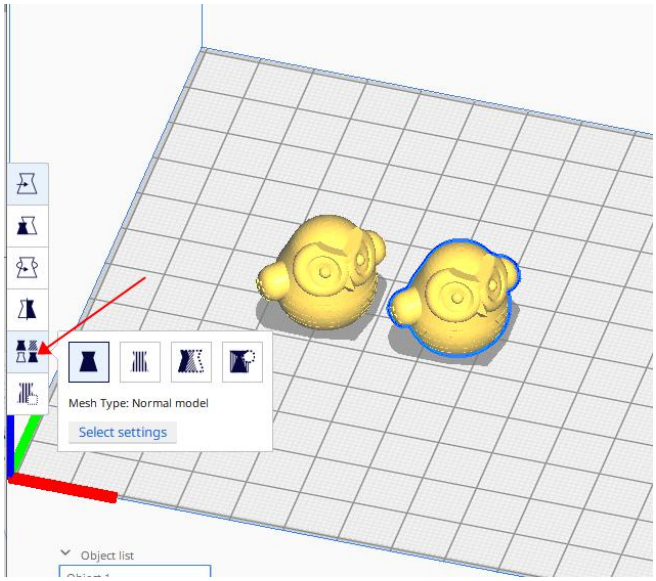
Mirror image, model symmetry transformation



Select the model and right-click to copy it



Parameters are set for different models



Introduction of common parameters

Please adjust the main parameters that affect the printing model according to the demand

Layer Height

The lower the thickness of each layer of the model, the finer the lamination and the longer the printing time,Usually 0.1-0.3mm

Filling density

It is the mesh density inside the model. The higher the density is, the higher the hardness and strength of the model are, the longer the printing time is, and the more consumables are consumed, It is usually set between 5% and 30%, 100% is solid

Printing temperature

Temperature of hot head, Match the temperature setting according to the printing of the material, See the material recommendation for details

PLA Hot head 190-210 Hot bed 60

TPU Hot head 210-220 Hot bed 60 (Slow down)

ABS Hot head 230-240 (The cover of the fan hood needs to be opened) Hot bed 95 (brim 5-10)

Before printing TPU, it is recommended to print "soft material printing auxiliary frame" and install it.Download it in the link below.

https://github.com/magicmaker3/magicmaker/tree/master/MiniLuban/Related%20model%20files

Other materials directly look at the recommended parameters of selling materials, hot head below 245, hot bed below 95, SLuban can printing.

Printing speed

The faster the speed is, the less time is needed, and the printing effect will decline, usually 50mm/s, (After printing, you can accelerate or decelerate by directly turning the knob beside the screen)

Support

Due to gravity, when printing the suspended part of the model, the material will fall off. Setting the support will automatically generate the support to hold the suspended part. After printing, the material will fall off manually

Retraction (Influence on wire drawing)

Because of gravity, the melted material will flow down during printing. When the nozzle prints the model, it will jump directly from one place to another and form wire drawing on the surface of the model

After retraction is set, the machine will return the material when it is idling, so that the material will not flow down. However, the length of retraction will be different according to different idling lengths. Generally, the retraction distance is set to 2mm. The larger retraction distance of the model can be set longer, and the smaller retraction distance of the model can be set smaller. Each model has different optimal parameters,So you don't have to spend too much time debugging, It can be manually processed after printing

Build Plate Adhesion

The first layer adhesion is the core of successful printing. Different adhesion can help different models stick better.

For models with large bottom contact surface, choose 'skirt' (routing, extruding nozzle air), for models with small contact surface, choose 'brim' (Increase bottom area), 'raft' (base, strong fault tolerance)

The flat side of the model is facing down to ensure that the platform has a larger contact area, and the printing will stick more firmly

For more parameters, see slicing software, in which each parameter has comments

Particular attention

1 Confirm that the configuration is imported correctly, and check whether the machine name on the left is SLUBAN

2 Don't change the red box. Change the material and change the parameters on the left. If you move the box, the configuration will be disordered and need to be reconfigured