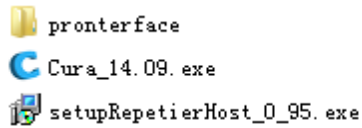


There are three software in the printer software directory



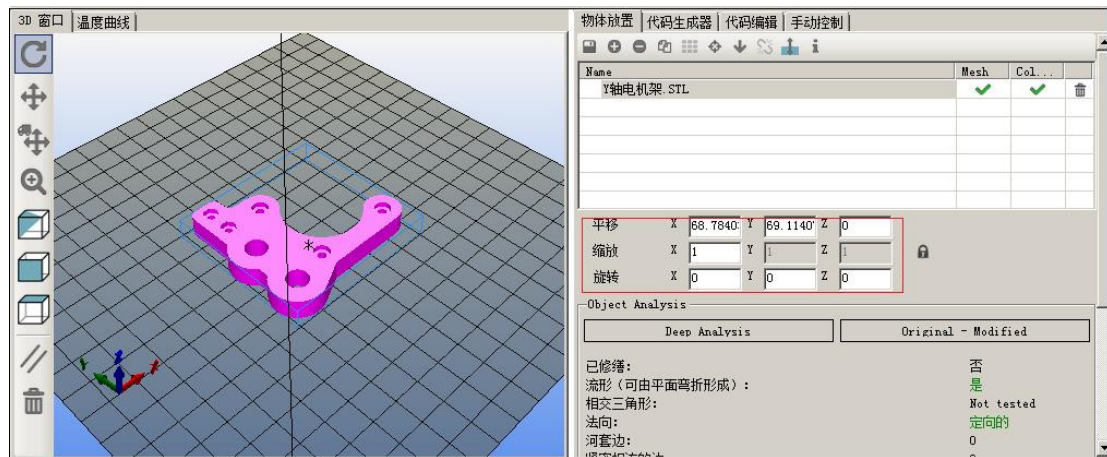
First one, Pronterface is a Green software, which is mean no installing when use it. and relatively stable when connect a computer to print something. But there also a shortcoming due to its single-function. We need to meet a additional slicing software.

The most important characteristic of CURA is slice speed, even a low configuration computer. It also can dispose complex model. But the quality of slice is not enough good.(personal opinion)

Repetierhost is a software that can switch language into Chinese. With the advantage of more powerful functional, it very suitable for testing printer, the disadvantage is that the software is not perfect, there may be all sorts of problem when printing.

In conclusion, person recommend to use the Repetierhost for debugging the machine. And use pronterface to print. In this essay, i use the repetierhost to do a explain for you.

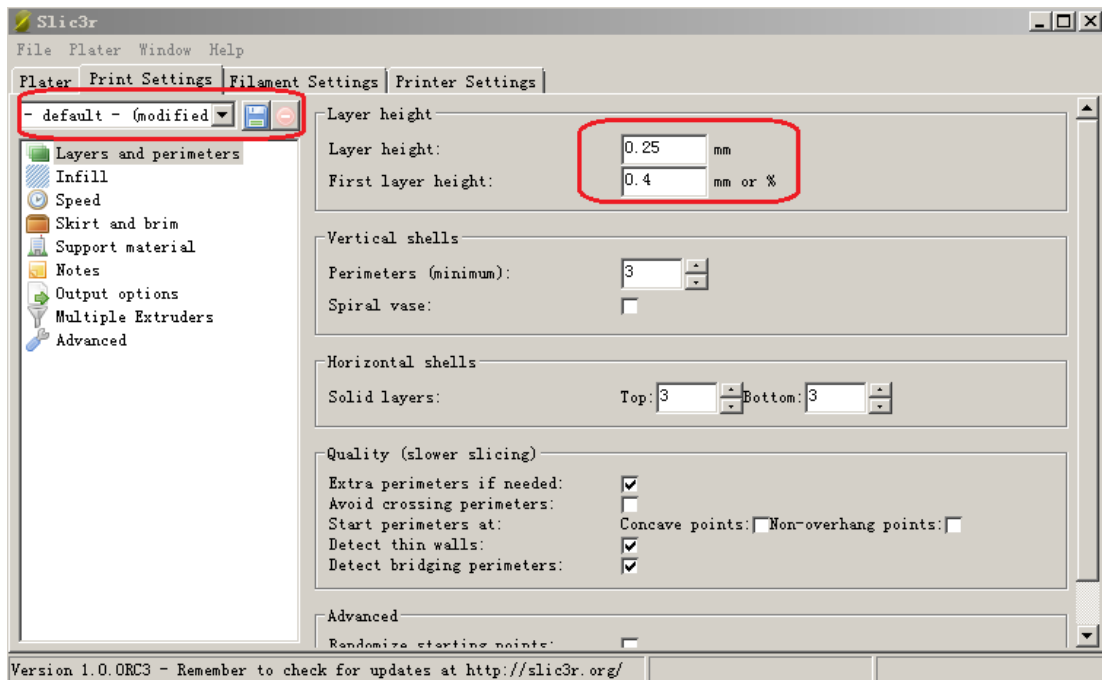
Open the Repetier and load in. there is a model of axis motor frame in the specification documents.



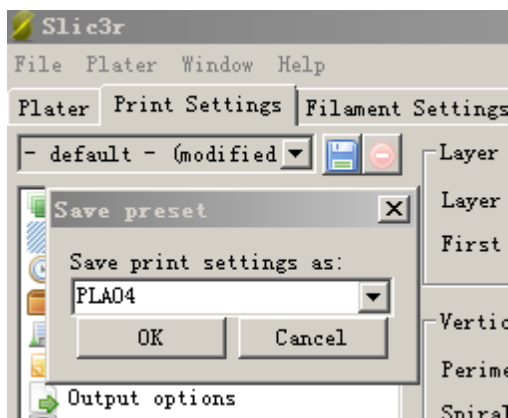
On the left side of the toolbar, we can adjust the view angle, and change the model size and place angle in the circled part. click the code generator. What in red circle are the configured parameters. And I'll introduce the step of configuring these parameters.



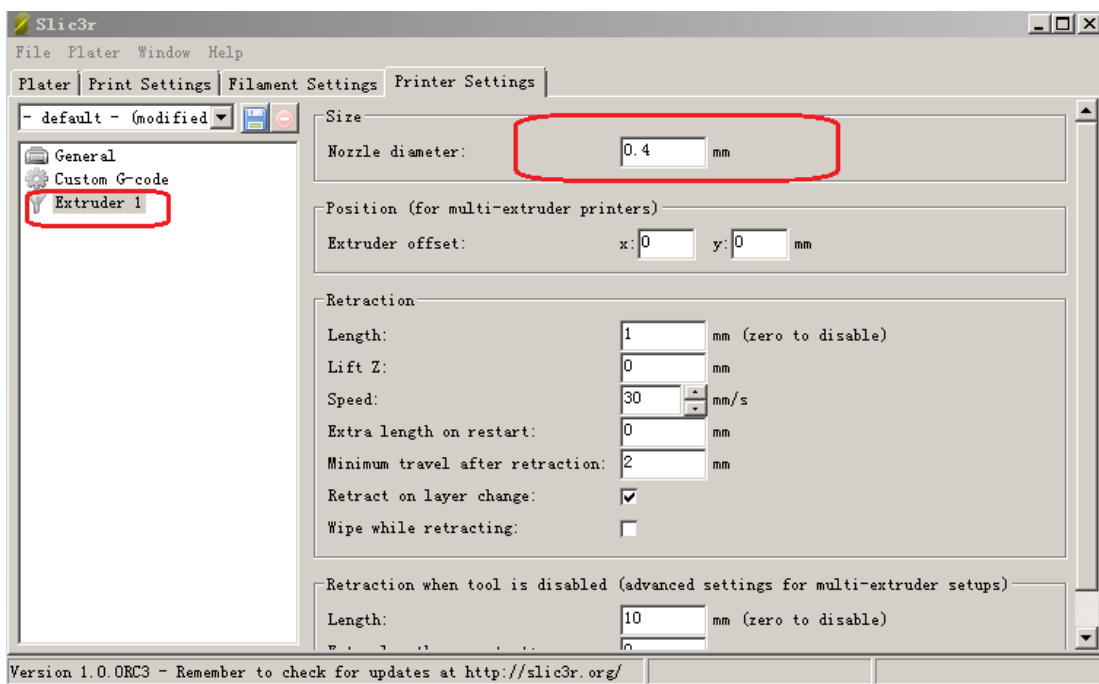
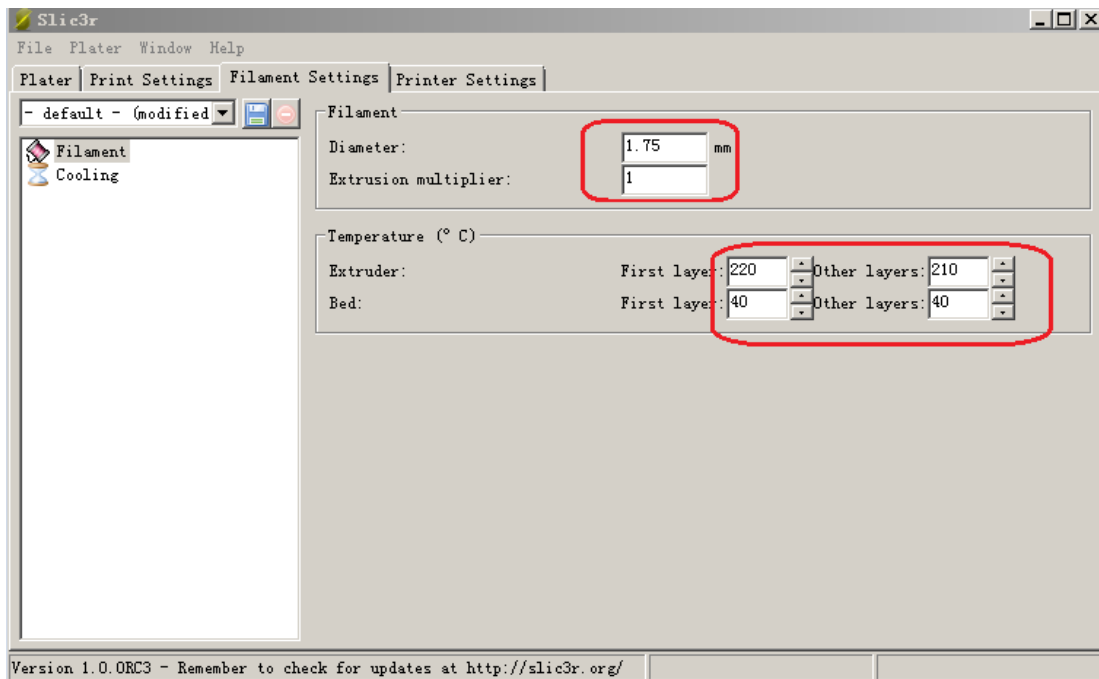
1: Click the “Configure”, wait for a while ,it start the slicing software now. According to your computer’s configuration, the startup speed is also different. After starting, it will pop-up a wizard prompts, you can choose all the default. As below picture shows:



Change the layer height and first layer height According to the parameters in the red circle. Then save it and rename it.



Each page need to save separately after modifying. No matter the name same or not.



Check it again to make sure that the parameters had been changed yet.

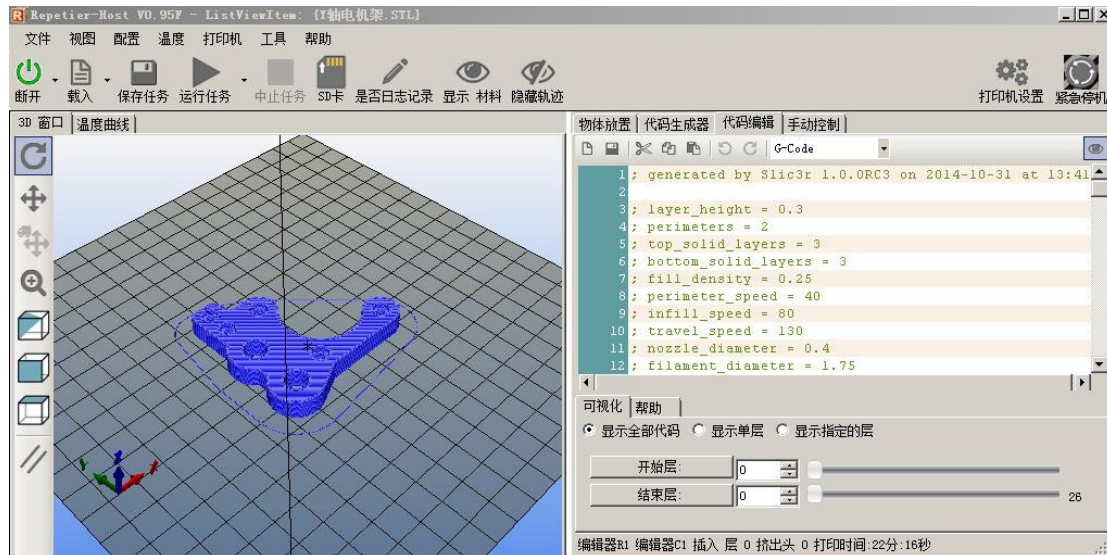
Close it and back to just interface you'll see the parameter group options as shown below.



Here you can save multiple sets of parameters, after selecting, click the “generate codes slic3r”

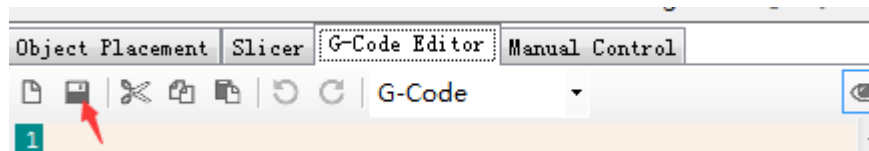


When reading progress bar is completed, we can check the print path in the code edit window.



Now we have finished the slice of model. And it's time to start printing:

You could save the G code in this page, and print it by Pronterface



This passage shows how to start print, but not enough for parameter description, please reference within the explanation documentation of SLIC3R. By modifying the parameters and then observed the results of print.

If strange errors, please use the Pronterface to print. Use as follows:

Run this software, choose the port and baud rate, click the connect first, then click “load file” to read the previous step saved G-code. At last ,click the “print” start print. The first thing is to observe if there is some dislocation around or not. Then remove it. We can modify the parameter according to preceding article.

Printer Interface

File Settings

Part: COM39 @ 250000 [Connect] [Reset] [Load file] [Compose] [SD] [Print] [Pause] [Recover]

Motors off XY: 2000 mm/min Z: 3000

Heat: Off 230 (abs) Set Check temp

Bed: Off 60 (pla) Set Watch

Extrude 5 mm 2

Reverse 5 mm/min 1

0 Ext Bed Ex0

-1

-2

2222 RRRRR Send

Connecting...
ok
Printer is now online.
T:22.15 B:21.34 B@:0 @:0
Disconnected.

Not connected to printer.