There are three software in the printer software directory

pronterface
Cura\_14.09.exe
distance
interface
inte

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.

物体放置(代码生成器)代码编辑)手动控制)		
▶ 开始	中止操作	
代码生成软(Slic3r		• ©©Manager
		Configure
Print Setting:	PLA04_hspeed	<b>•</b>
Printer Settings:	PLA04_hspeed	•
Printer Settings: Filament settings:	PLA04_hspeed	•
-	PLA04_hspeed PLA04_hspeed	<b>•</b>
Filament settings:		× ×

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:

💋 Slie3r			
File Plater Window Help			
Plater Print Settings Filament	Settings   Printer Settings		
- default - (modified Layers and perimeters Infill Speed Skirt and brim Support material	Layer height Layer height: First layer height:	0.25 0.4 mm or %	<b>^</b>
Notes Output options Multiple Extruders Advanced	Perimeters (minimum): Spiral vase:	3	
	-Horizontal shells Solid layers:	Top: 3 * Bottom: 3 *	
	Quality (slower slicing) Extra perimeters if needed: Avoid crossing perimeters: Start perimeters at: Detect thin walls: Detect bridging perimeters:	▼ Concave points:  Non-overhang points:  ▼ ▼	
	Advanced Rendomize starting noints:	<b>-</b>	
Version 1.0.0RC3 - Remember to ch	eck for updates at http://slic3r.	org/	

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

💋 Slic3r	
File Plater Window Help	
Plater Print Settings Filament	Settings
- default - (modified 💌 📔 🔵	Layer
Save preset 🗙	Layer
Save print settings as:	First
FLA04	
DK Cancel	Vertic
	Perime
output options 📄 🚽	Sniral

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

💋 Slic3r			
File Plater Window Help			
Plater   Print Settings   Filament	Settings Printer Settings		
- default - (modified 🗾 🔚 🔾	Filament		
S Filament	Diameter:	1.75 mm	
Cooling	Extrusion multiplier:		
	Temperature (°C)		
	Extruder:	First layer: 220 • Other layers:	210
	Bed:	First layer: 40Other layers:	40
Version 1.0.0RC3 - Remember to che	ck for updates at http://slic3r	. org/	

💋 Slic3r			- 🗆 🗡
File Plater Window Help			
Plater   Print Settings   Filament	Settings Printer Settings		
- default - (modified 💌 🔚 🔵	Size		
General	Nozzle diameter:	0.4 mm	
Custom G-code			
Fxtruder 1	Position (for multi-extruder prin	nters)	
	Extruder offset:	x: 0 y: 0 mm	
	Retraction		
	Length:	1 mm (zero to disable)	
	Lift Z:	0 mm	
	Speed:	30 • mm/s	
	Extra length on restart:	0 mm	
	Minimum travel after retraction:	2 mm	
	Retract on layer change:	<b>v</b>	
	Wipe while retracting:	Г	
	Retraction when tool is disabled	(advanced settings for multi-extruder setups)	
	Length:	10 mm (zero to disable)	
Version 1.0.0RC3 - Remember to che	ck for updates at http://slic3r.org	z/	

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.

· · · · · · · · · · · · · · · · · · ·		
		🚓 Configure
Print Setting:	PLA04_hspeed	•
Printer Settings:	PLA04_hspeed	•
Filament settings:		
挤出头 1:	PLA04_hspeed	•

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.

