# Easy use of Repetier-Host software

Examples base on version Repetier-Host 1.06

1. Repetier-Host overview

Repetier Host is an easy to use software for 3D printing. The main functions include manual debugging and model slicing. Repetier Host is one of the most widely used printing software for 3D printing. Below is an interface picture of successful installation of Repetier - Host 1.06.
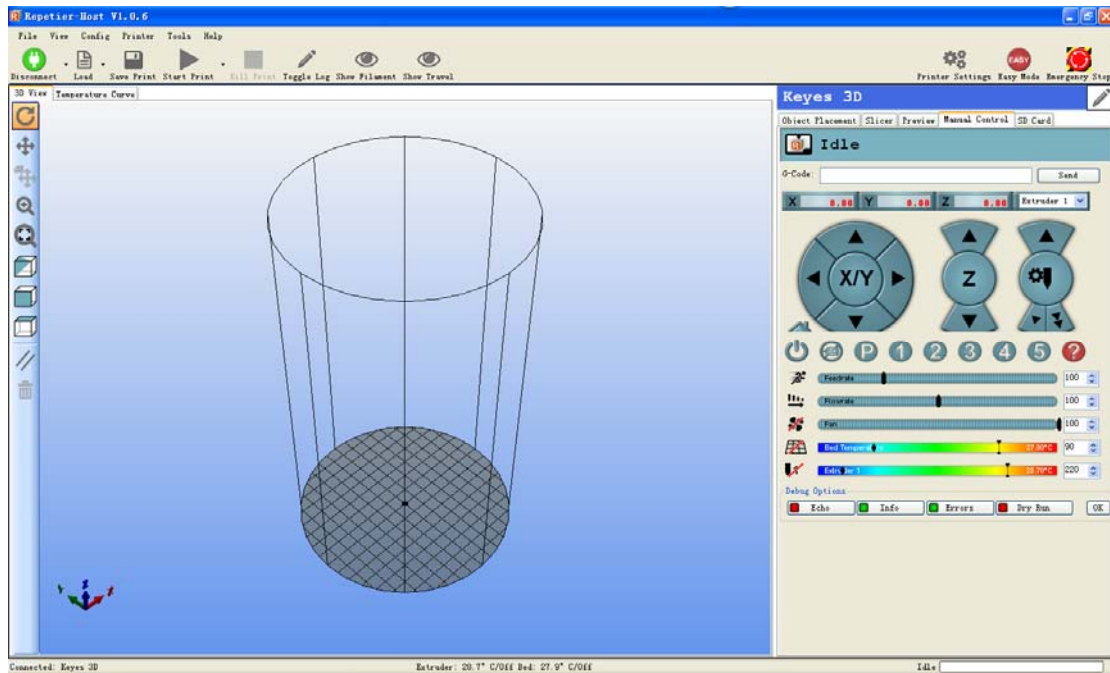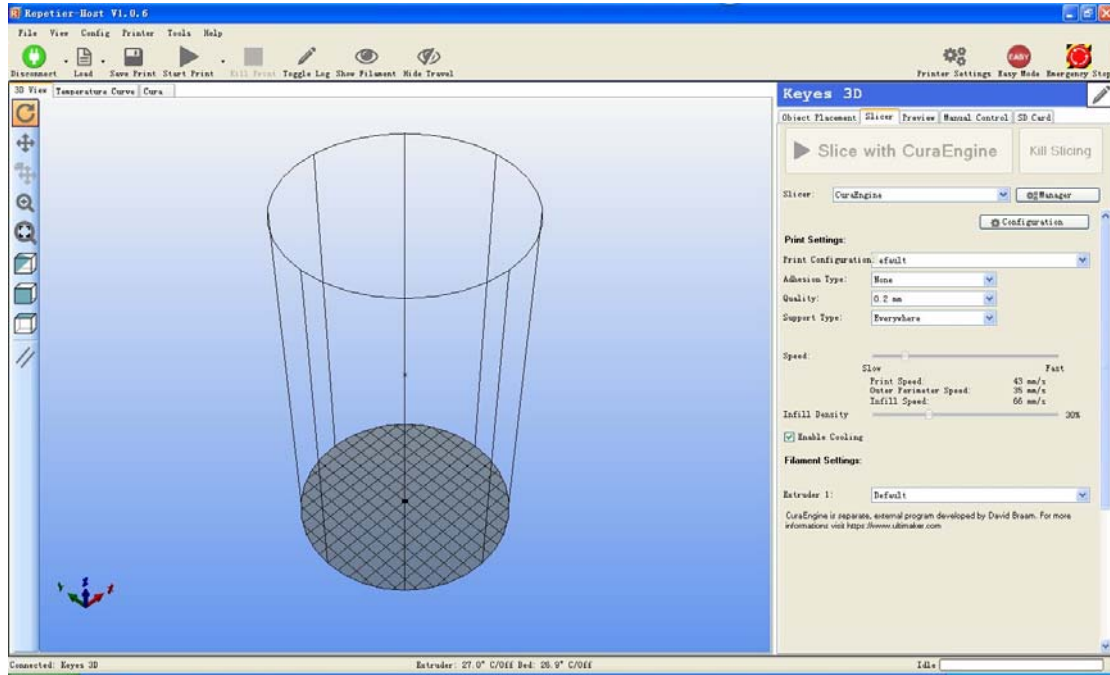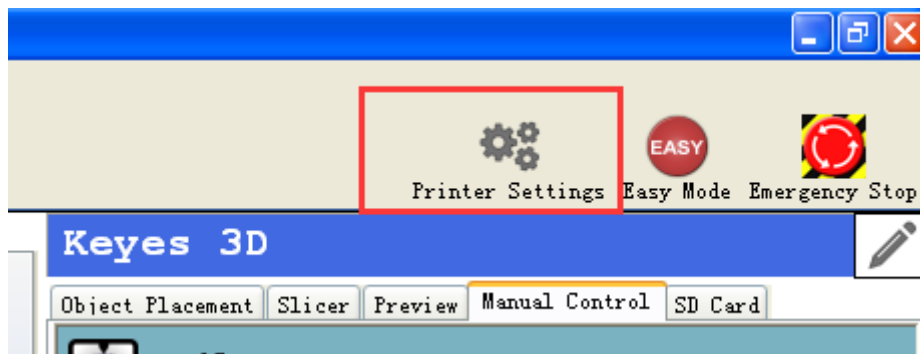

1) user interface language setup

Click Config ⟶ Language ⟶ English
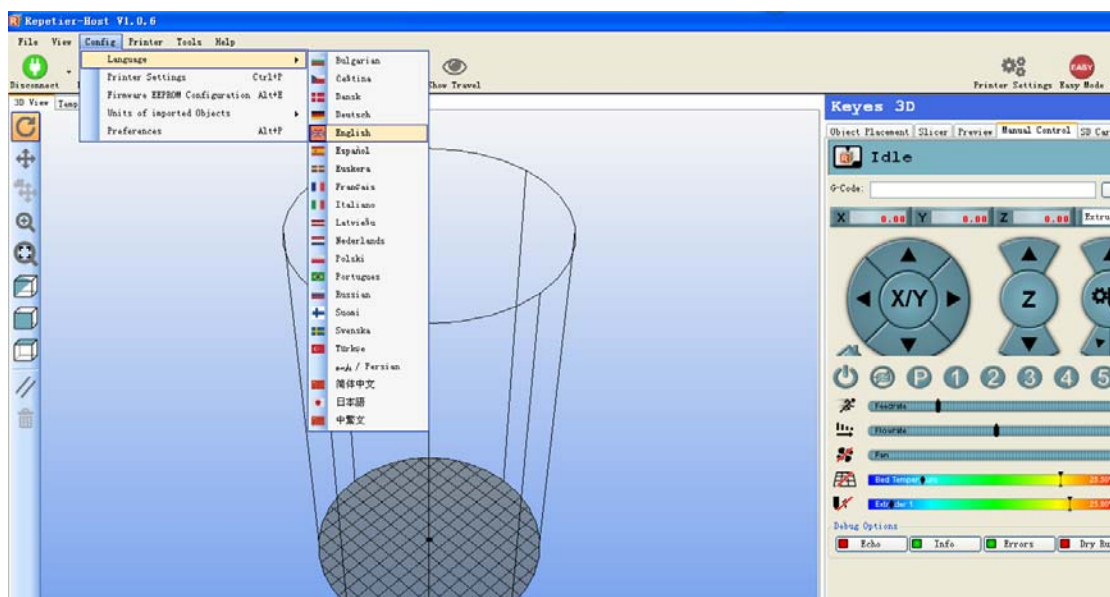
After the setup, restart the software. You can see below English interface.

2)after setting up the language, we can set up configuration linking the printer and the software.

① Click printer settings button as above picture shows; you can see interface as below picture. First one is connection option:

A. For connection port, select serial port connection;

B. For COM port, please select corresponding port number of your printer.Mine is COM925;

C. Baud rate set up as 250000. If yours is 115200, please change accordingly. For other setup, just use the default ones.
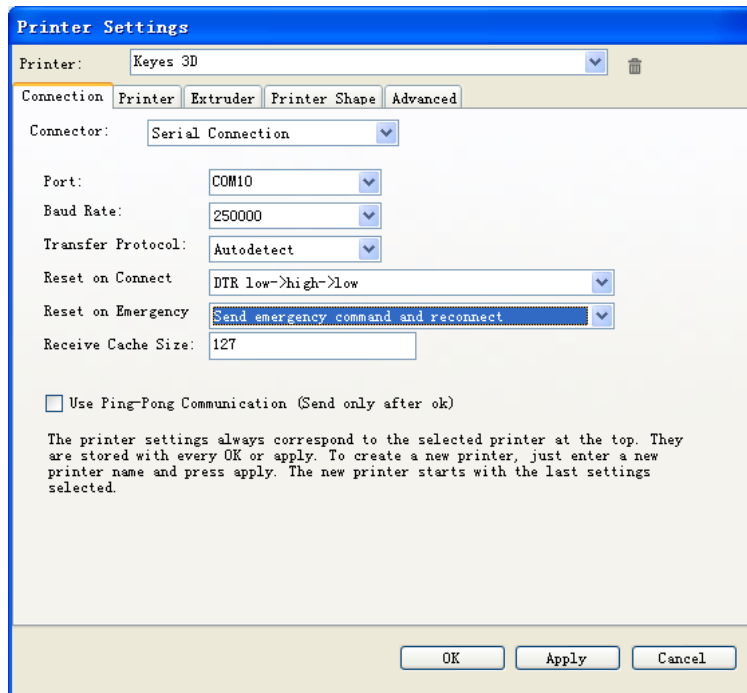
② Printer setup

A. Extruder travel feed speed: 8000mm/min

B. Z axis feed speed: 8000mm/min
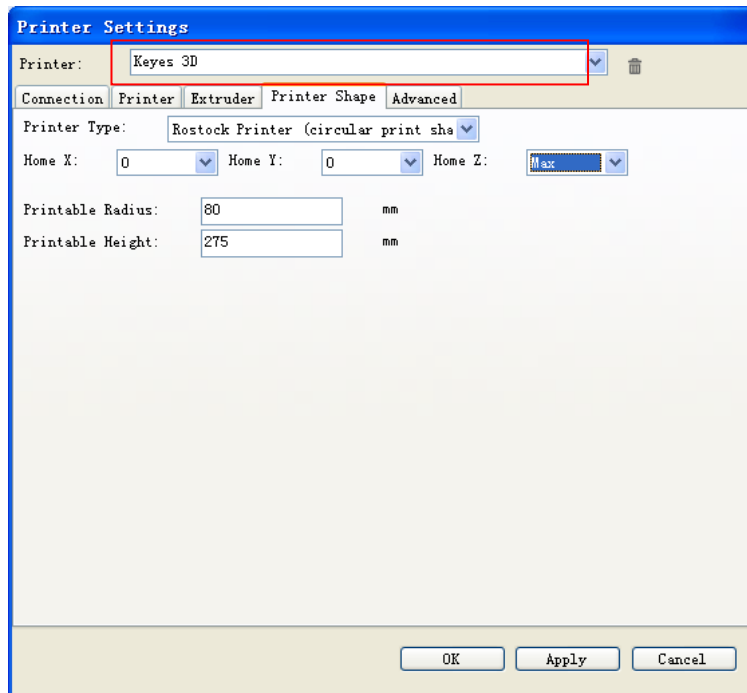
C. For other setup, just use the default ones.
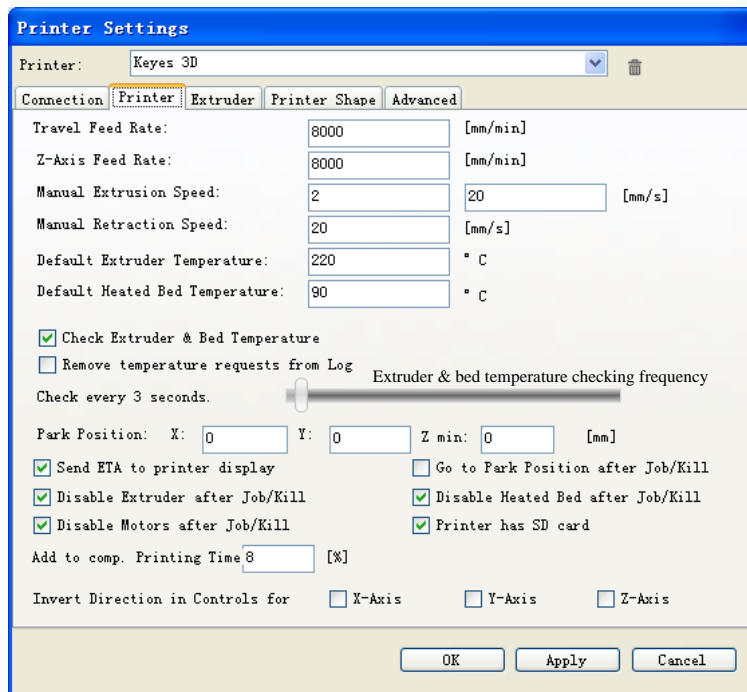
③ For Extruder, all use the default ones.
④ Printing shape setup, according to below picture.

A. In the red box, you can enter your printer's name. Mine is Keyes 3D

B. Select Rostock printer for printer type.

C. Home X：0　　　Home Y：0　　Home Z：Max

D. Printable radius: 80mm
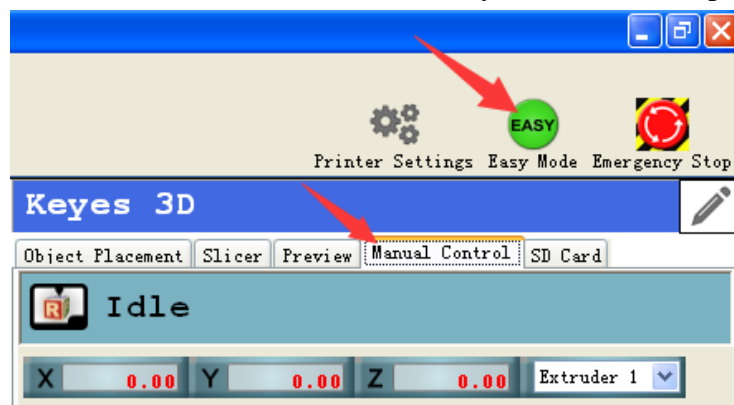
E. Printing height: 275mm

1. After the above setup, we can link the printer to our computer. Click the connect button as below picture shows.
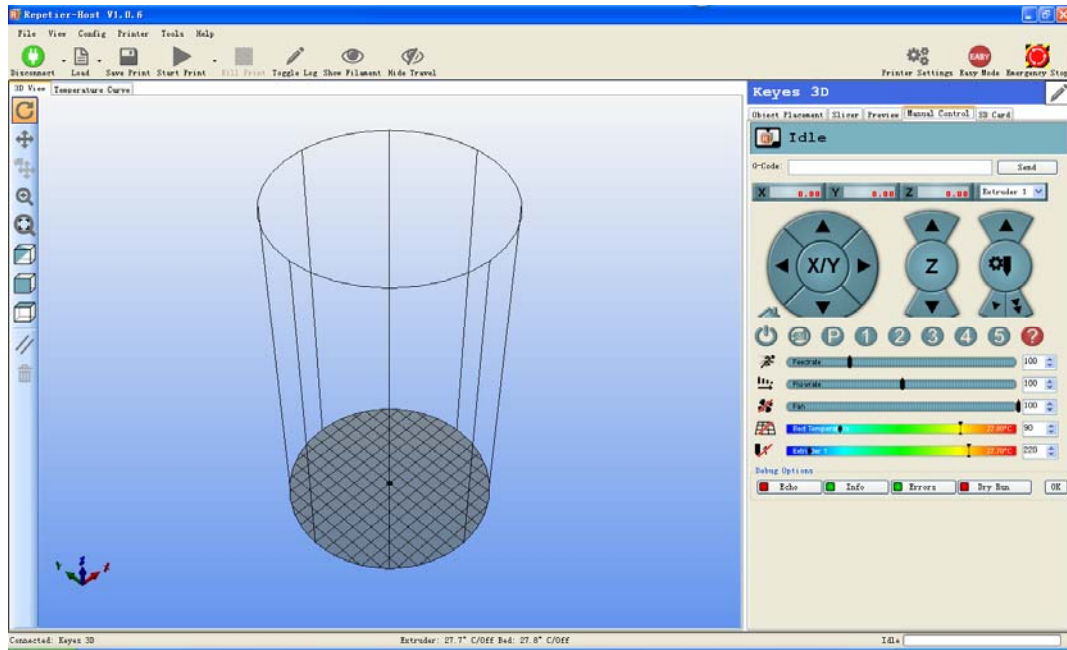

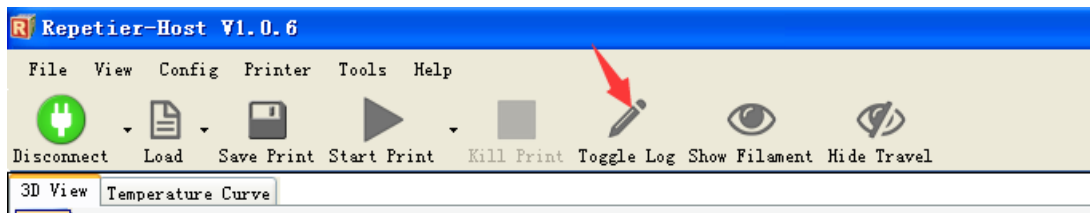If you see the button as below picture, the connection is succeeded.

2. After the connection, click manual control and Easy Mode as below picture.
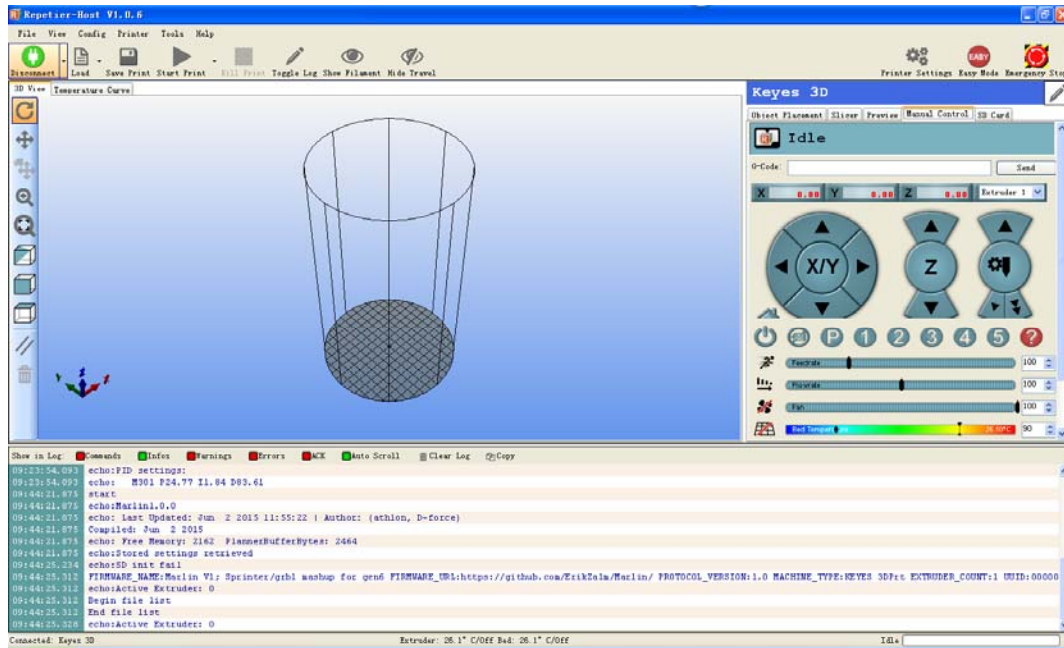

You can see below interface:

Then, click the "Toggle log"button

You can see below interface


3). Manually control the motor movement

    1）See if the installation is correct by manually control the motor rotate direction using buttons of X,Y,Z and extruder 1.

Click X on the right, extruder moves to the right; move to the left if click the left one.

Click Y on the above, heat bed moves backward; moves forward if click the below one.

Click Z on the above, nozzle moves upward; moves downward if click the below one.

Click nozzle on the above, nozzle motor moves reversely; moves forward(feeding) if click the below one. This action requires the nozzle temperature to reach the melt temperature of the filament.
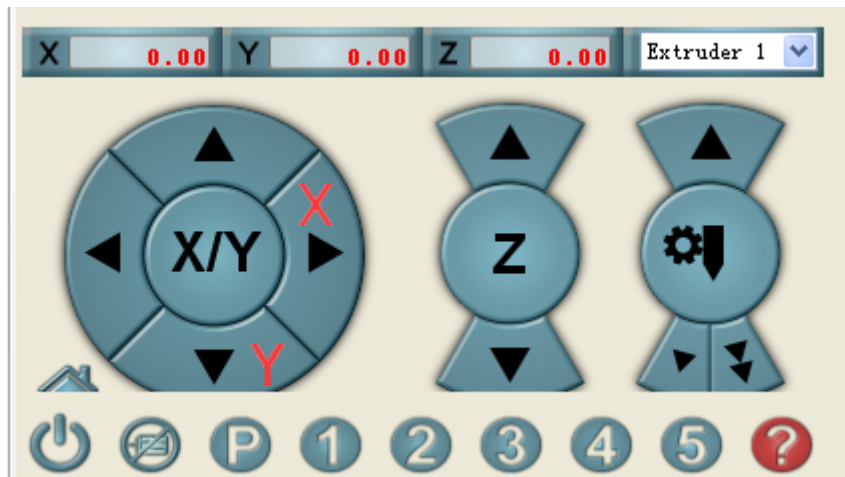
2）Control the motor rotation through the display.

In standby mode interface, click the knob button to enter the next level menu.

Click Prepare ➝ Move axis ➝ Move 0.1mm ➝ Move X、Move Y、Move Z or Move E

For X axis, it moves to the right with increase of value; for Y axis, it moves backwards with increase of value;for Z axis, it moves upwards with increase of value; for E axis,it rotates forward(feeding) with increase of value.

Above directions are true when you directly face your display.

After the above, we use G28 to home the machine and check if everything is working correctly.

4) for height adjustment of 3D printing nozzle and heat bed, please refer to another file.
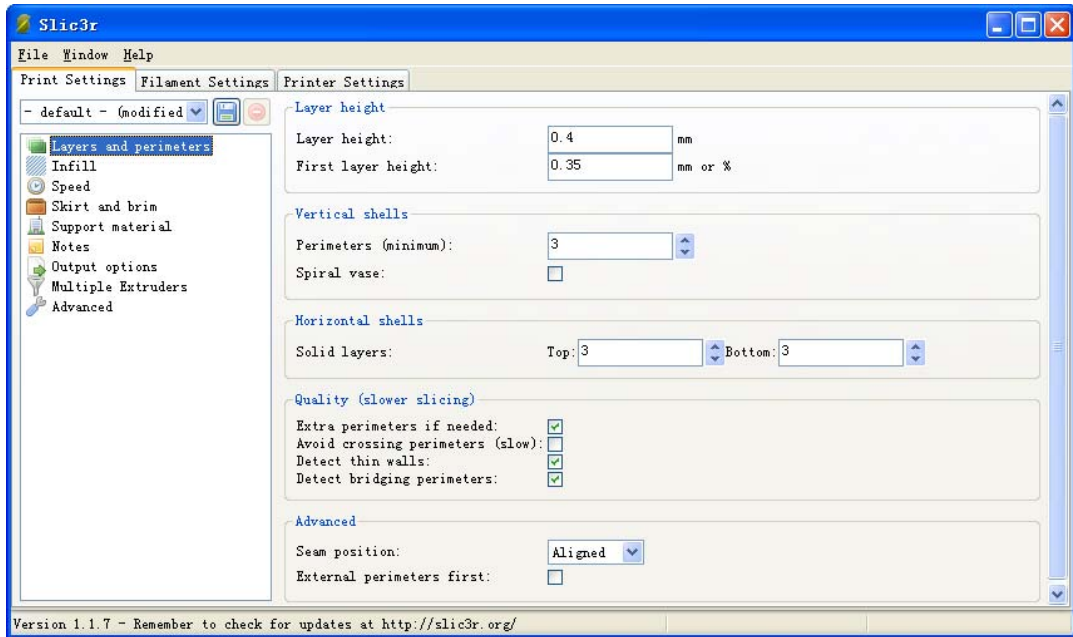
2. slicing

1）parameter setup for Slic3r

Please save each setup.

A. Print Settings

1)layers and perimeters

A. In layer setting, there are layer height and first layer height. Layer height means height for each layer during the printing. It's generally base on nozzle diameter. The higher the value, the coarser the product. The lower the value, the finer.

The height of the first layer will be lower than the other layers because first layer will be pressed down and if too high, material will pile up, not good for other layer printing.

B. Then it's the perimeters. Perimeter means the minimum thickness of the outer layer, normally no less than 3.

C. Horizontal shells means layers on the top and bottom, often with solid layer. You can define the top and bottom layer here.

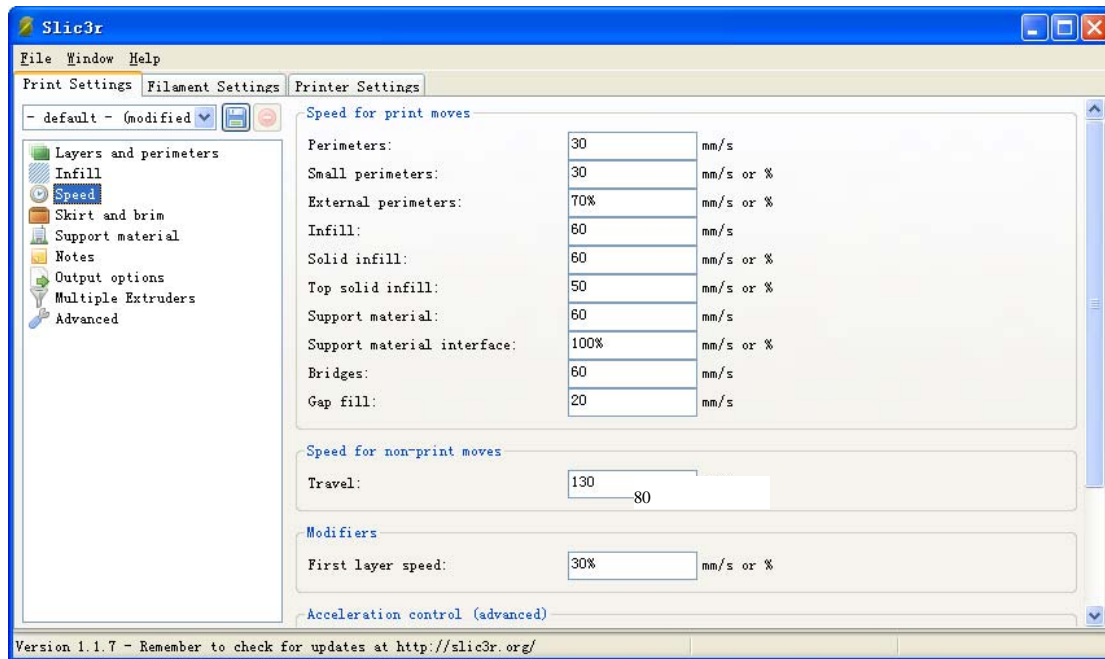D. For advanced, just use the default ones.

   2)infill settings
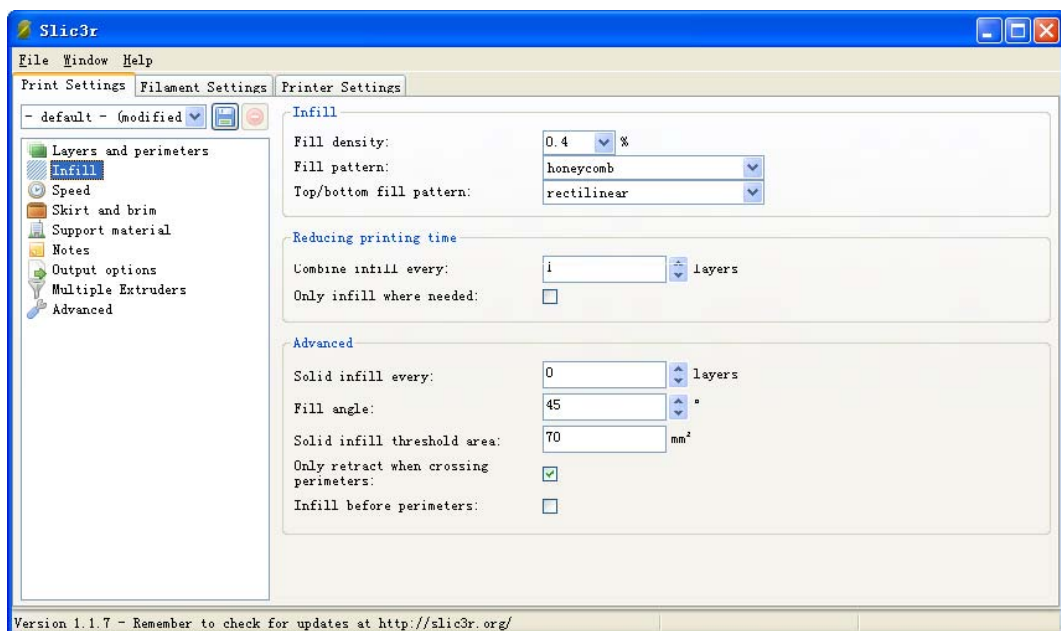
A. In the infill settings. One option is fill density, which means density of filament. For fill pattern and top/bottom fill pattern, you can choose freely as they don't affect product much.

B. For advanced, just use the default ones.

3）speed setup:

Special attention: in the first printing, you need to set all speeds at 20mm/s. When the first printing is a success, you can try to set the speed higher but in reasonable values.

For speed setup, there are many items, but all very clear. Among them, speed for non-print moves means moving without load. Note that for first layer speed, don't sep too high or filament cannot stick to heat bed.

4）skirt and brim setup:
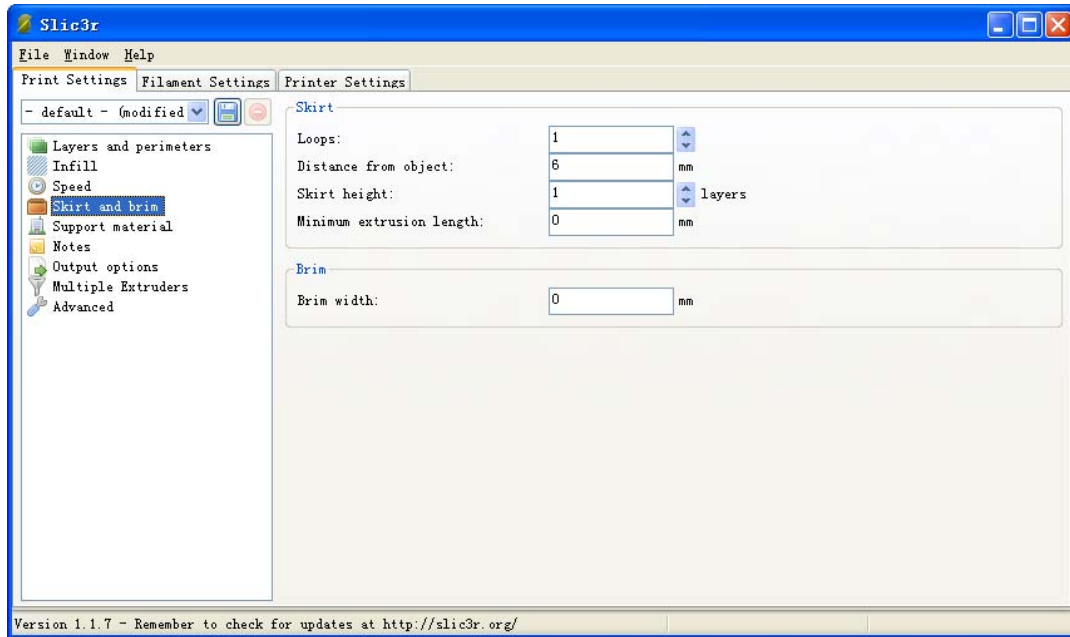
This setup is for the nozzle action in the beginning. It will a circle of lines around the product. We suggest you keep this setup because in the beginning, the nozzle will be in short of filament. After one circle. The filament feeding will be normal.

5）support material setup:
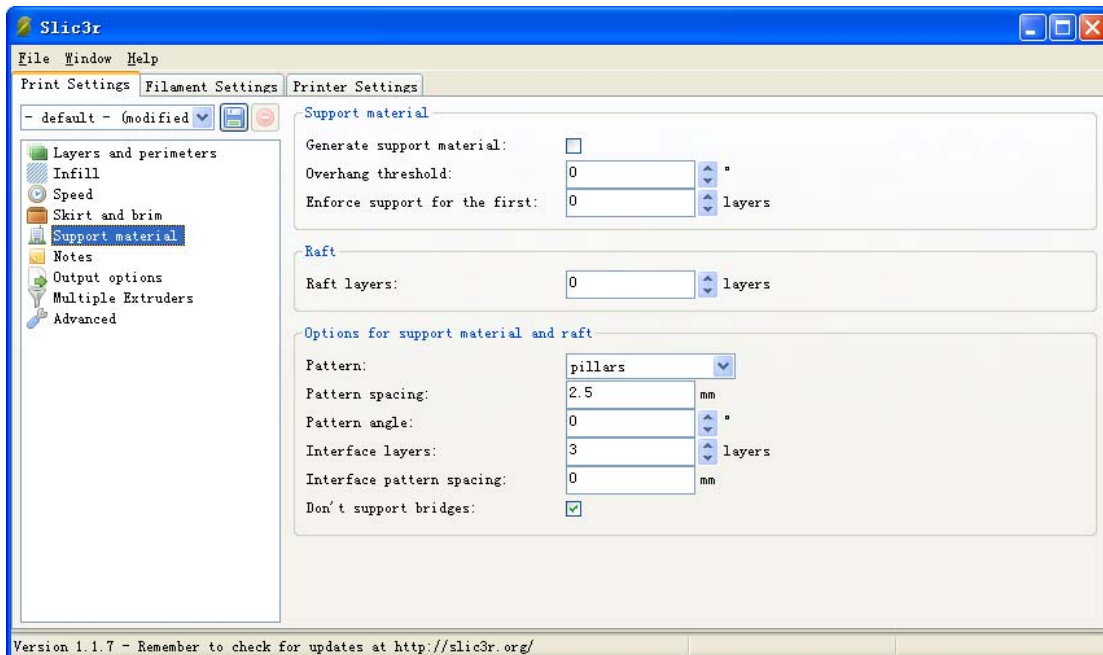
During the printing, there may be hollow parts or connection using bridge. If the bridge is too long, support is needed. Support is extra part, you can remove after the the printing is complete. Raft is the base layer during the printing. When printing on glass heat bed, raft is not recommended.

6）notes
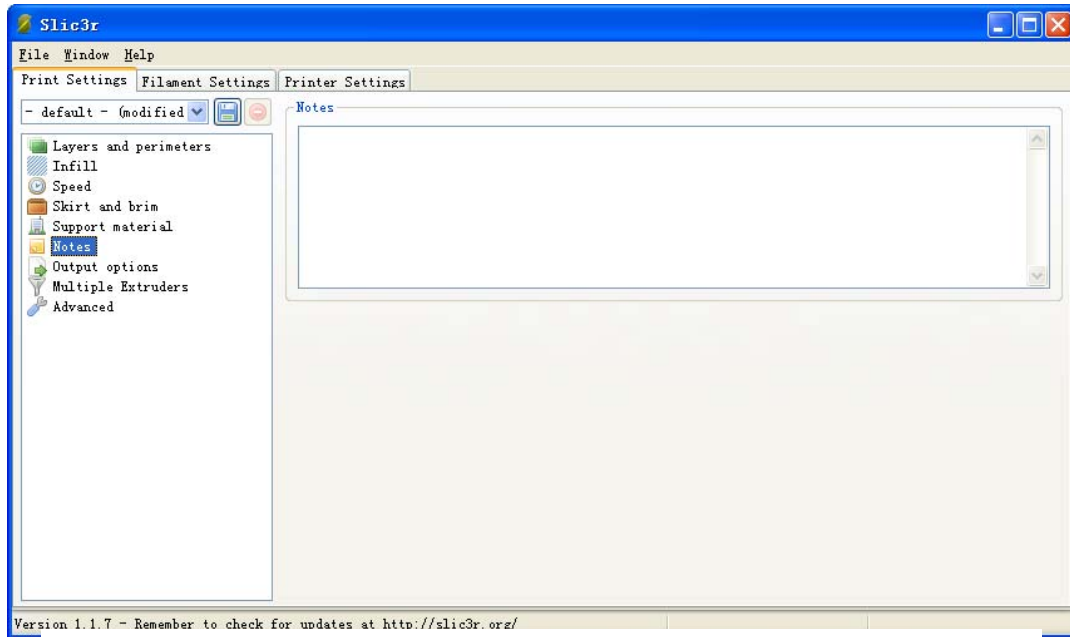
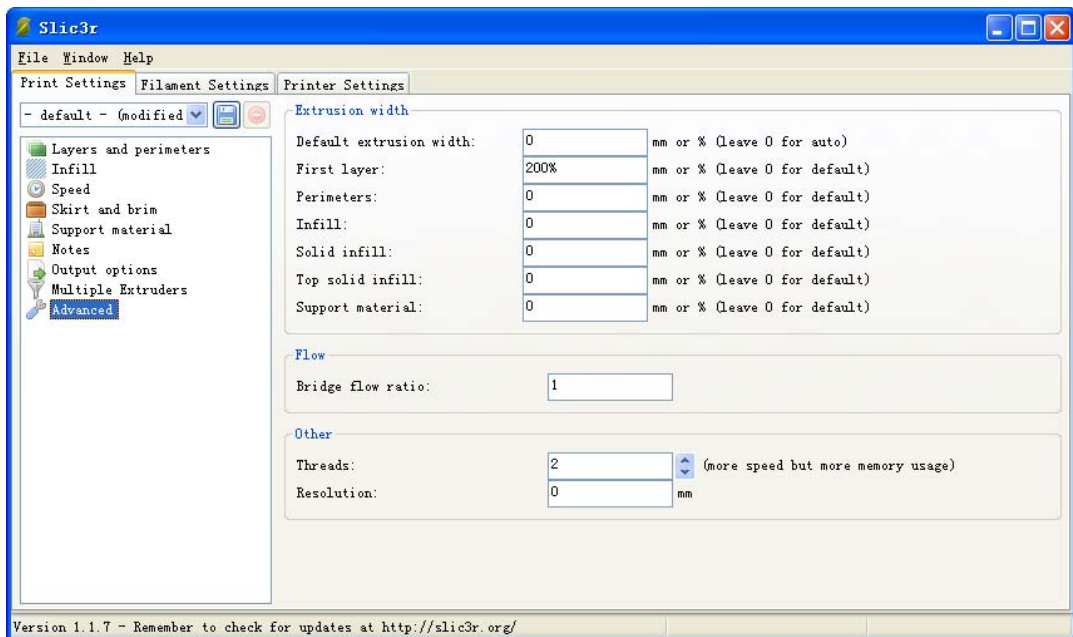You can add things you want to write down, just like a notebook.
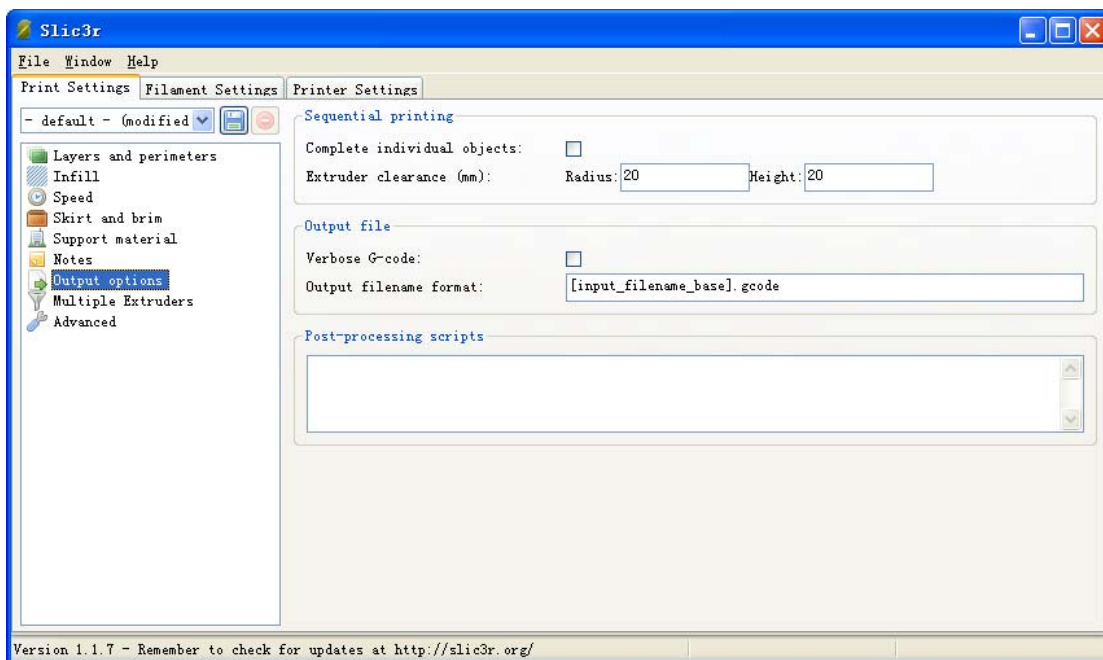
This option includes sequential printing and output file etc. Just use the default ones.

8）extruder setup:
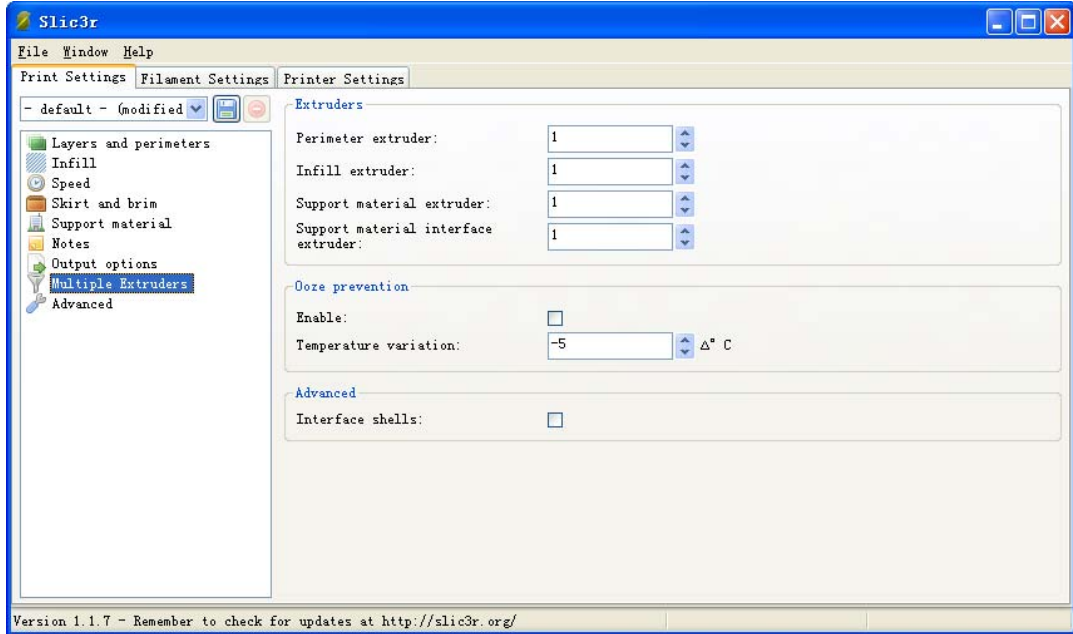
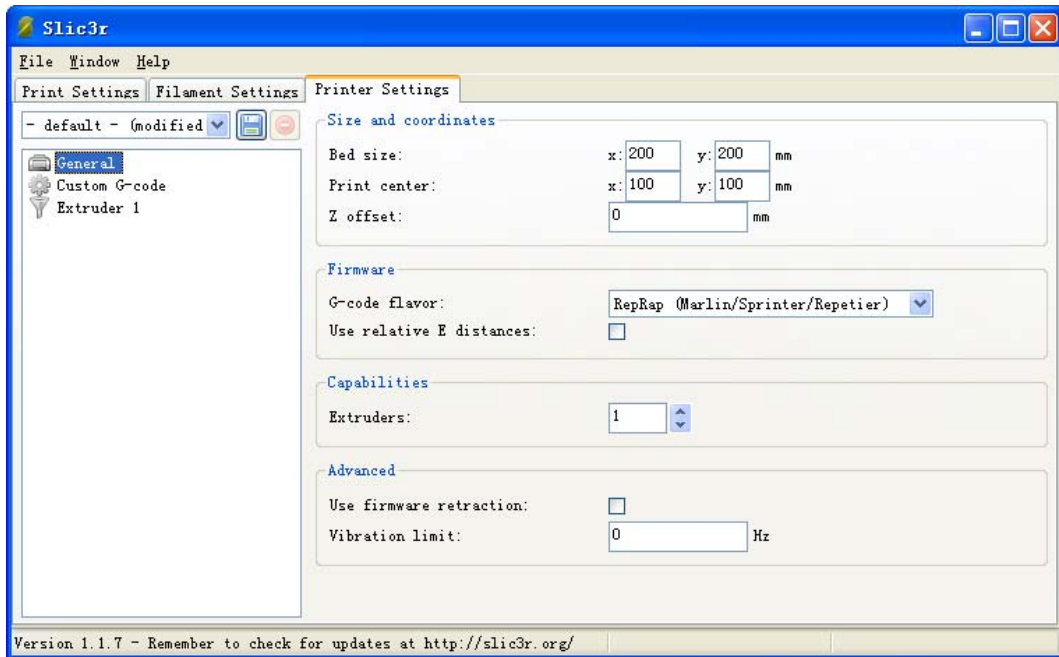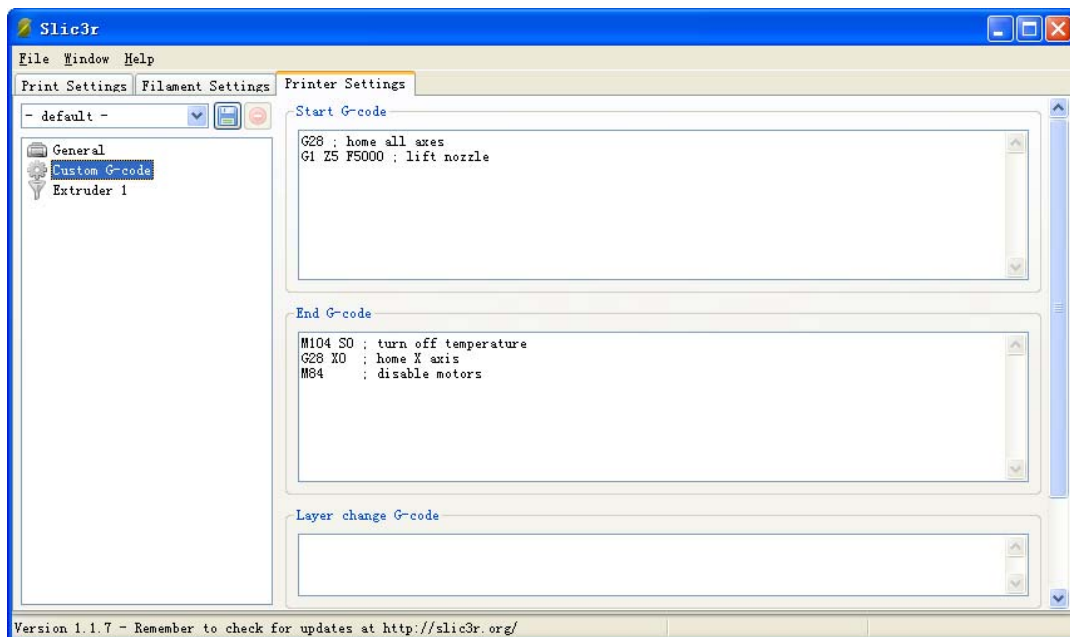This option is for multiple extruders. Not needed for 1 extruder.

9）advanced:

Suggest not to change any parameters in this option.

B. Printer setting

1）for this option, just setup as below picture.

2）custom G-code

; Default start code

M140 S95

M104 S220

G28 ; Home extruder

G29

G1 Z15 F1500

;G28 X0 F2000

;G28 Y0 F2000
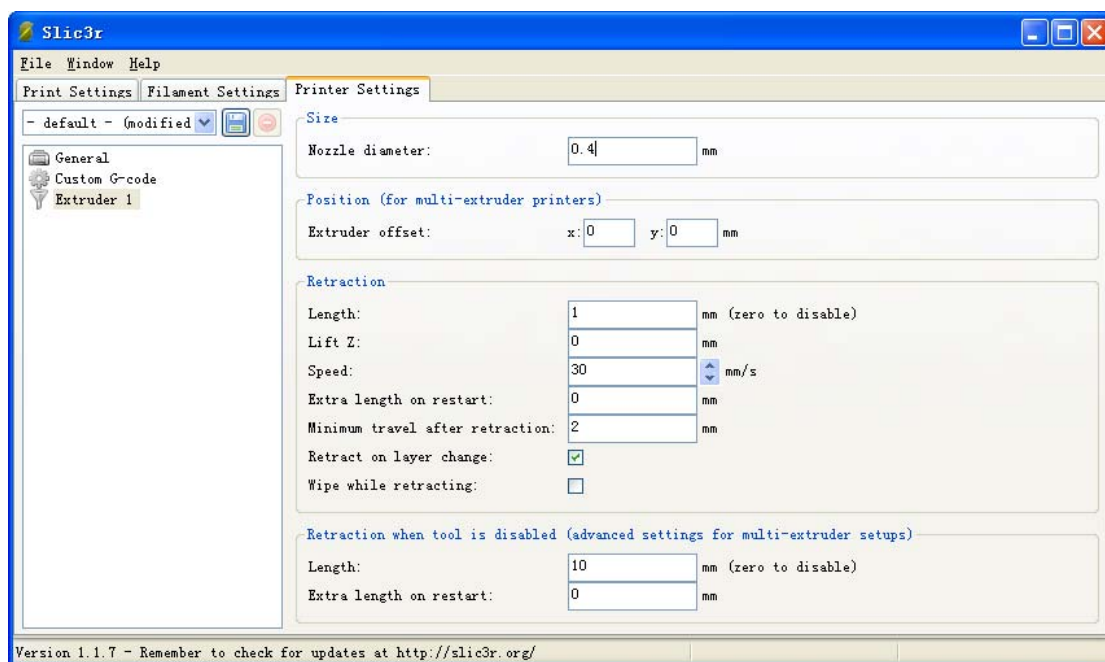
M107 ; Turn off fan

G90 ; Absolute positioning

M82 ; Extruder in absolute mode
{IF_BED}M190 S100
; Activate all used extruder
{IF_EXT0}M104 T0 S225
G92 E0 ; Reset extruder position
; Wait for all used extruders to reach temperature
{IF_EXT0}M109 T0 S225
G92 E0
G1 E15 F500
G92 E0
M300 S1000 P3000
M0
以上是供参考的开始 G 代码。

3） **Extruder setup**

For extruder setup, because we only need one extruder. That means we only need to have one setup.

Size: According to actual situation. For Pri3, nozzle diameter is 0.4mm.

Position: stop position when printing is complete. You can set up freely but do not exceed the printing area.

Retraction: after printing out 1 filament, the extruder will have a retraction. You can set up here though we suggest we go with the defaults.

Advanced is for multiple extruders, you don't to set up for 1 extruder.

C. filament setting

1）filament setup
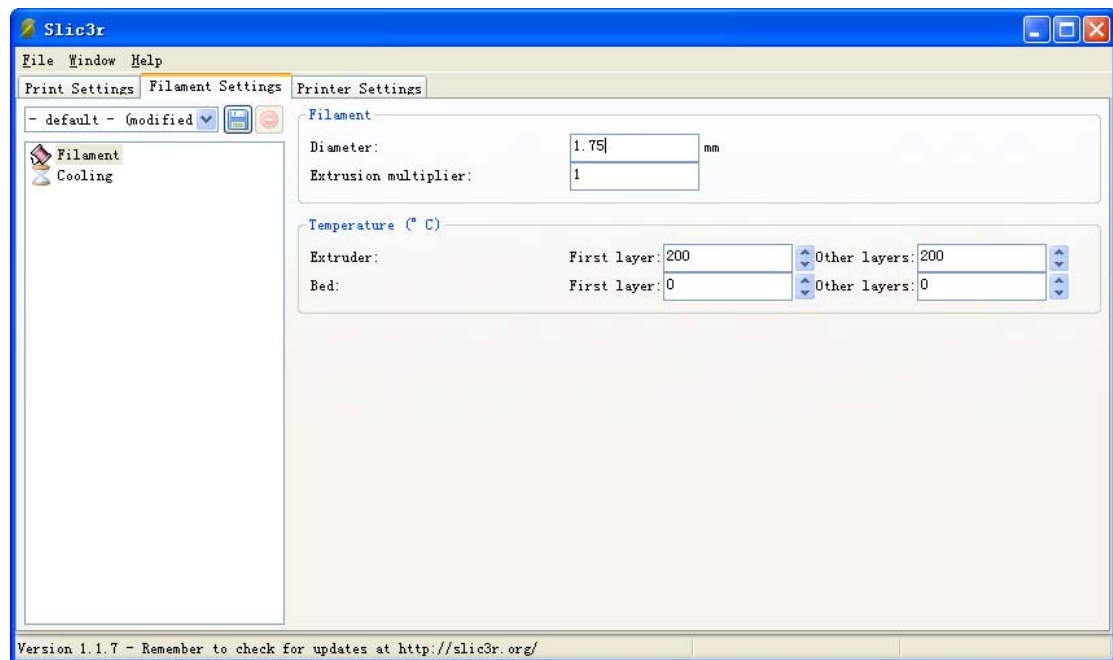
- Filament setup: diameter according to actual diameter, usually are 3mm and 1.75mm.
- Temperature setup: this setting is important. There are two kinds of materials. One is ABS; another one is PLA. For ABS, temperature settings are: extruder 220℃, heat bed 110℃; for PLA: extruder 185℃, heat bed 55℃.
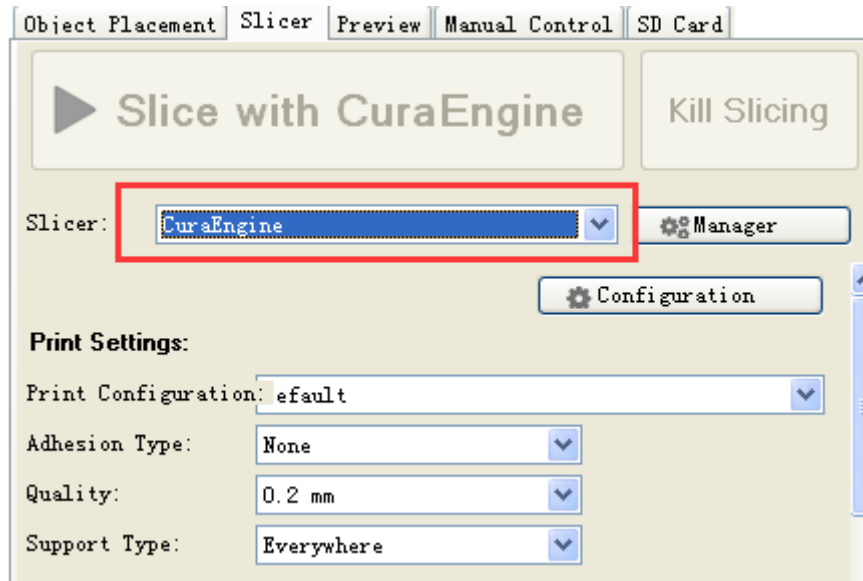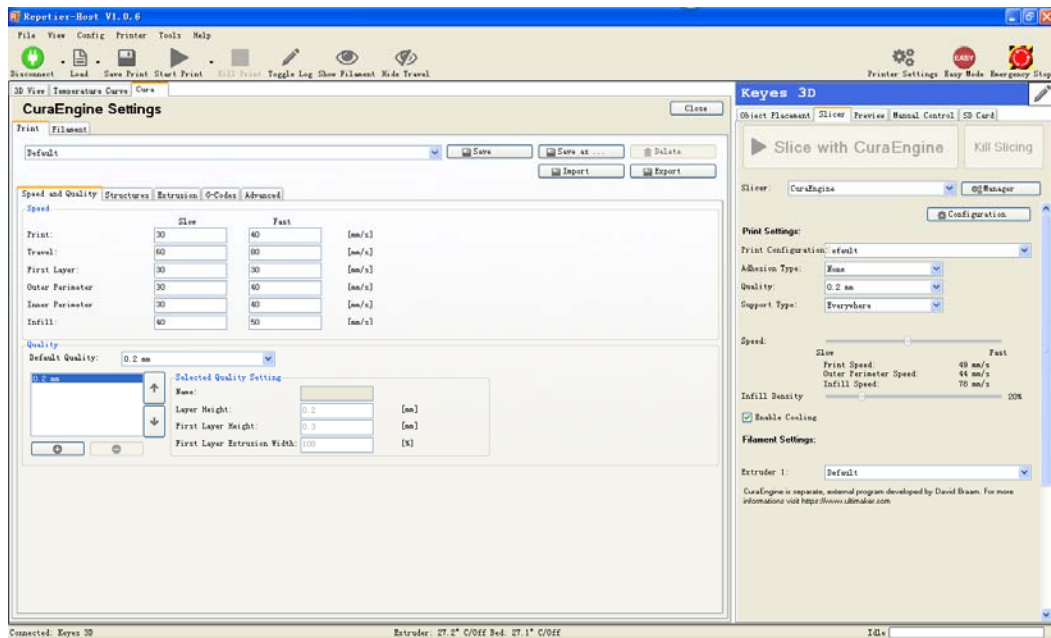
2）cooling setup
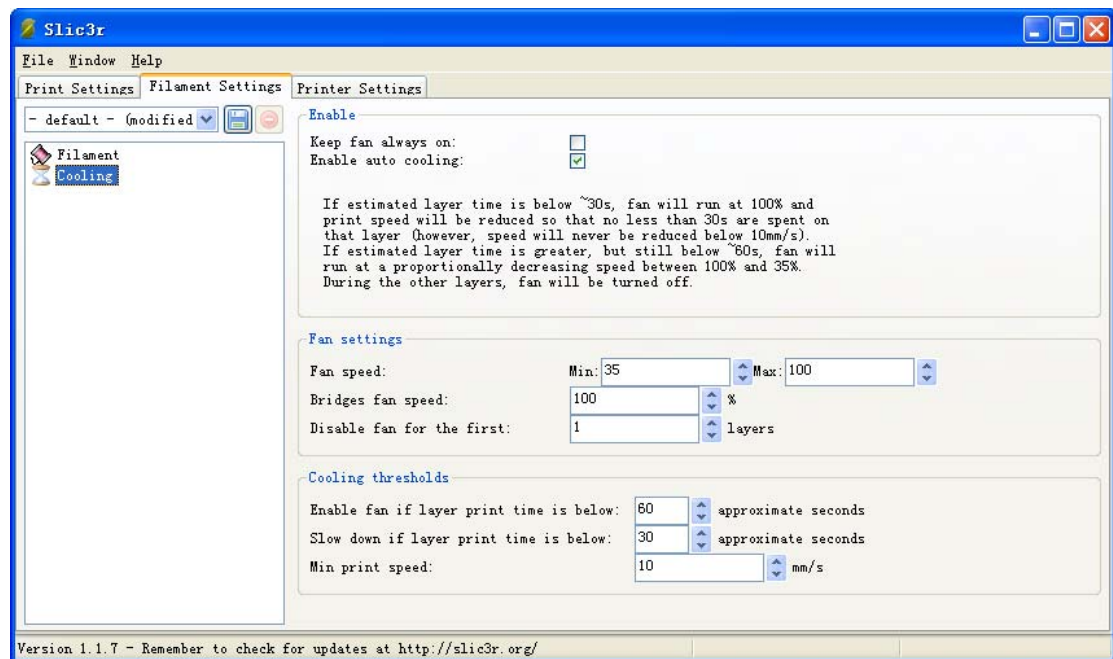
For cooling option, it's mainly used in printing small objects or printing hole parts. Otherwise, warpage will happenin printing big objects.

2）CuraEngine Settings

Open Repetier-Host software; click slicing software; select CuraEngine as red circle shows.

Click configuration, you can see CuraEnginesetup interface as below.
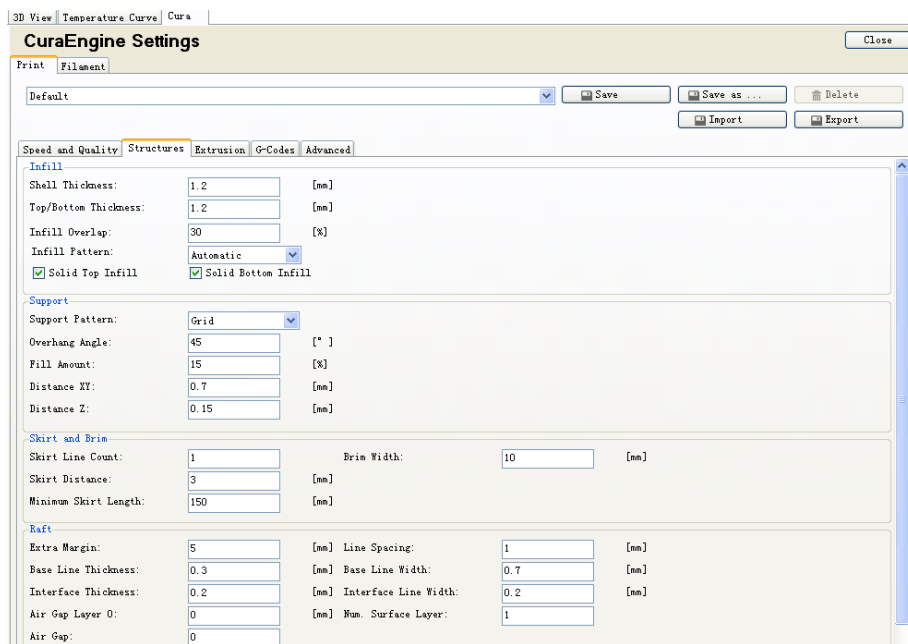

1）printing setup

A. Speed and quality

For the first time printing, please refer to below setup. After the setup, click "save as"and enter a name in the prompt box to save as your own configuration. For the first layer, extrusion width can set up as 200% to increase bonding between the printing object and the heat bed to avoid warpage.
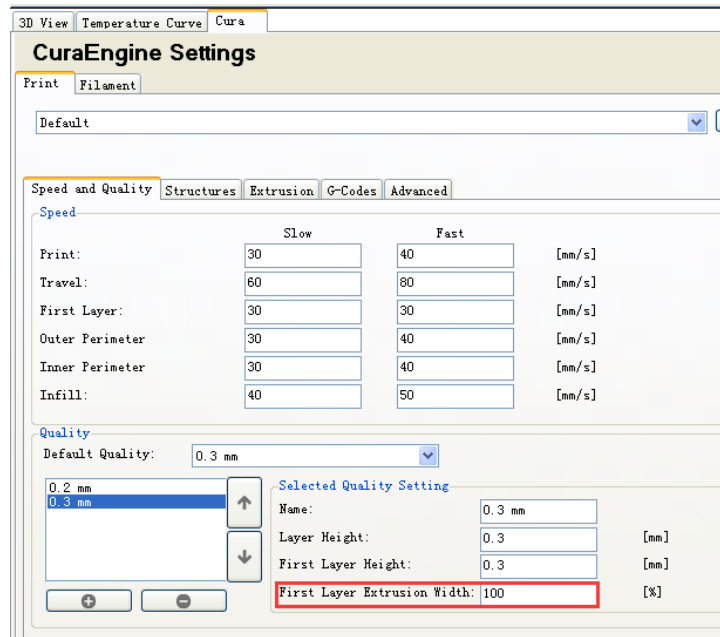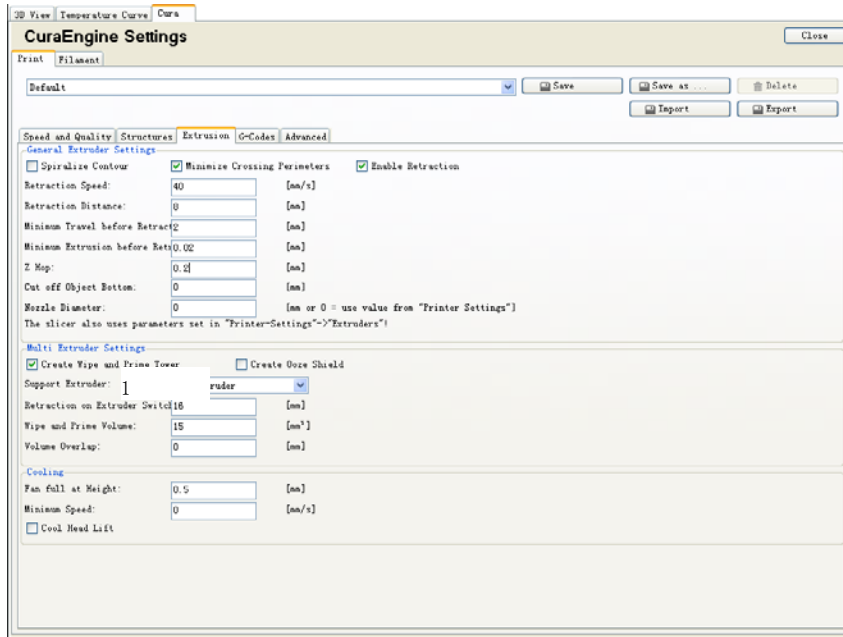
B. structure

For the first time printing, set up as below picture. For different object, set up different parameters for better effect.

C. extrusion

Set up as below picture. If there is only one extruder, no need to set up multiple extruder option. If you are printing spiral object, you need to select option "spiral perimeter".
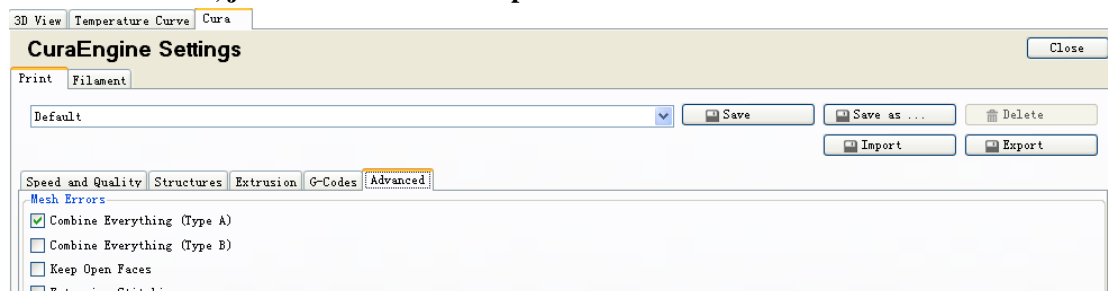
D. G-code
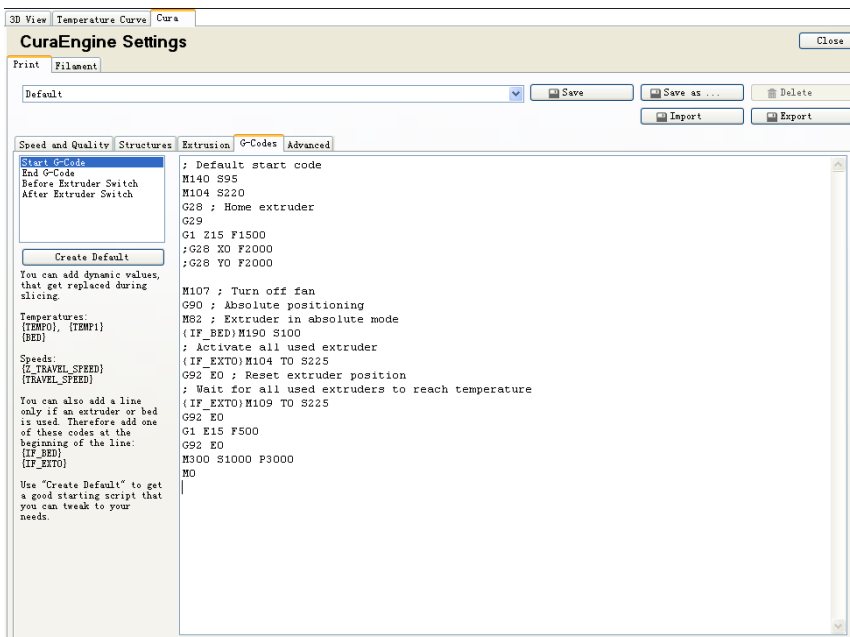
E. advanced

For advanced, just use the default setup.

Before moving on to setup for next option, save the last setup first.

2）material

For ABs, temperature settings are: extruder 220℃, heat bed 100℃; for PLA: extruder 185℃, heat bed 55℃.