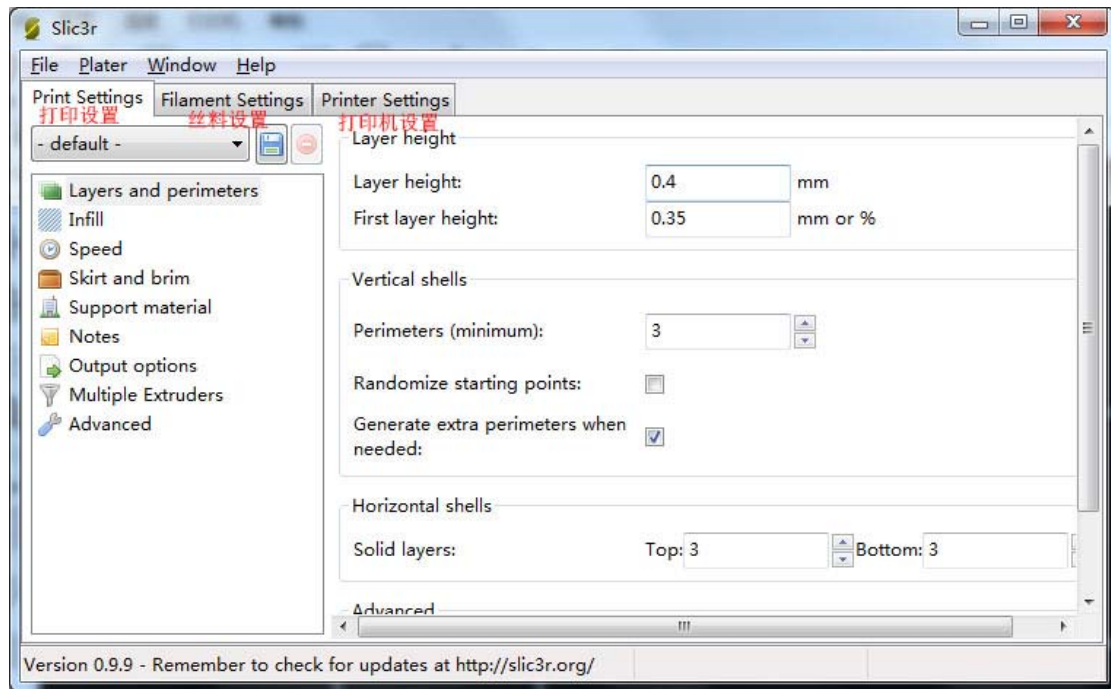



Parameter settings

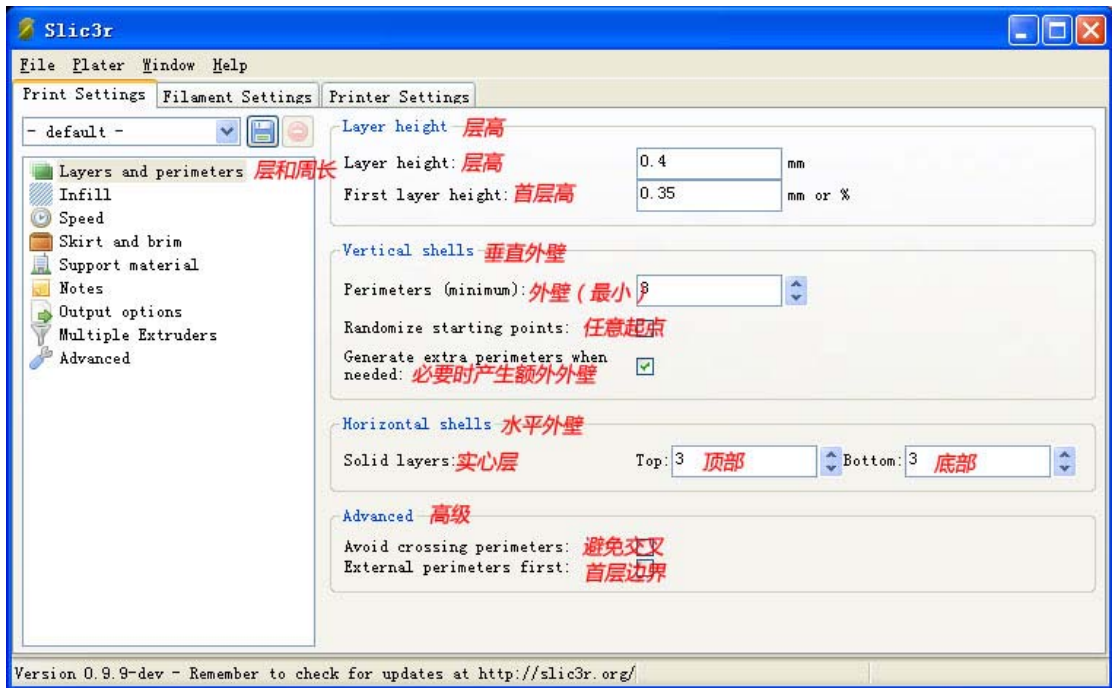
Firstly,click the “SETTING” button to set the parameters



Here, you can set “PRINT” , “FILAMENT” , “ PRINTER” . After you set every one, you have to press  button to save your settings. Next, please read the functions of the above 3 settings carefully.

1. Print setting

The first item in the “PRINT SETTING” was Layers and perimeters, as you can see in the picture below.



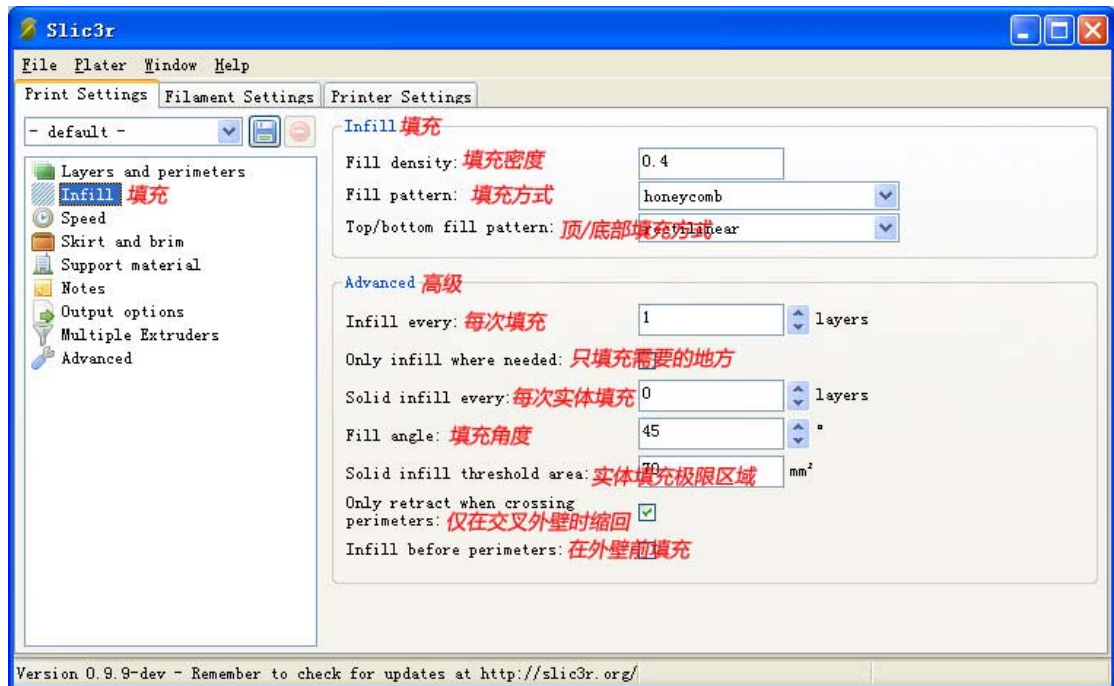
a. Here you can set Layer height and First layer height. Layer height is the height of every layer when printing. The value of the layer height should not be larger than the value of the nozzle's diameter. If you set the value larger, the product you print is rougher, so it is better to set it smaller. The value of the First layer height should be smaller than that of other layers, because the first layer would be flattened, if too higher, the filament would accumulated to have a bad influence on printing.

b. Setting the "Perimeter", which is the minimum thickness of the outer ring, the value should be less than "3".

c. The Horizontal shells include the layers of the bottom and top. Generally they should be solid layers. You can define the the solid layers here.

d. The 2 options in the "ADVANCE" could be just setted the default.

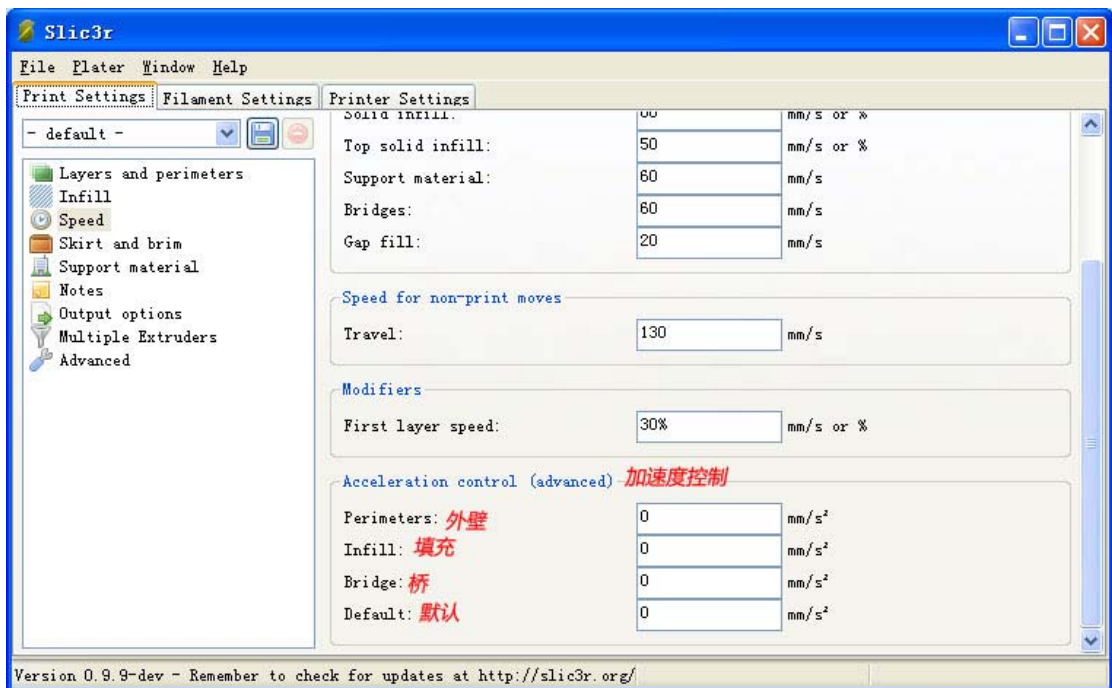
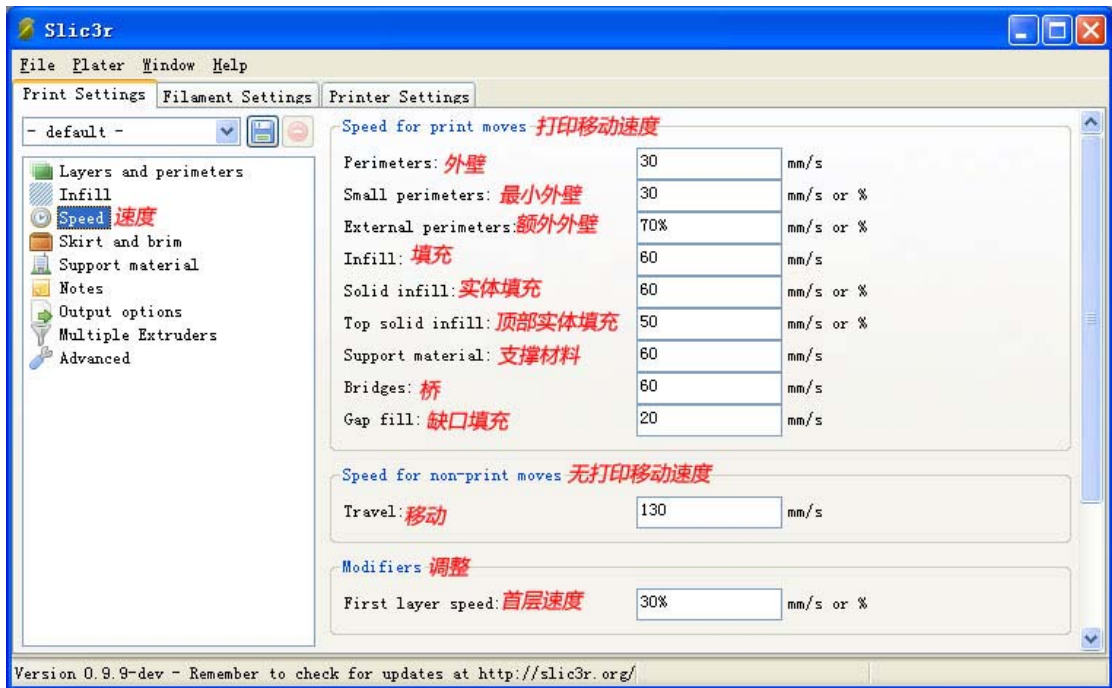
The second item is "Infill"



A: You can see Fill density, which is the filament density. You can choose the Fill patterns and the Top/bottom fill pattern as you like.

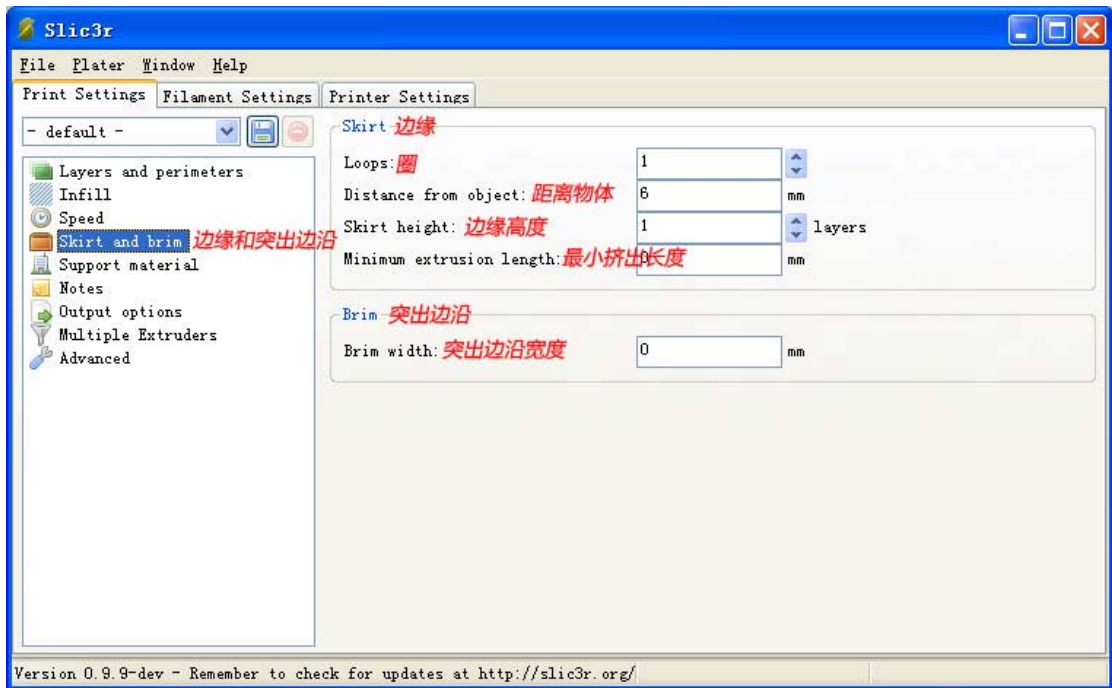
B: The items in the "ADVANCE" can be setted default

The third item "SPEED" :



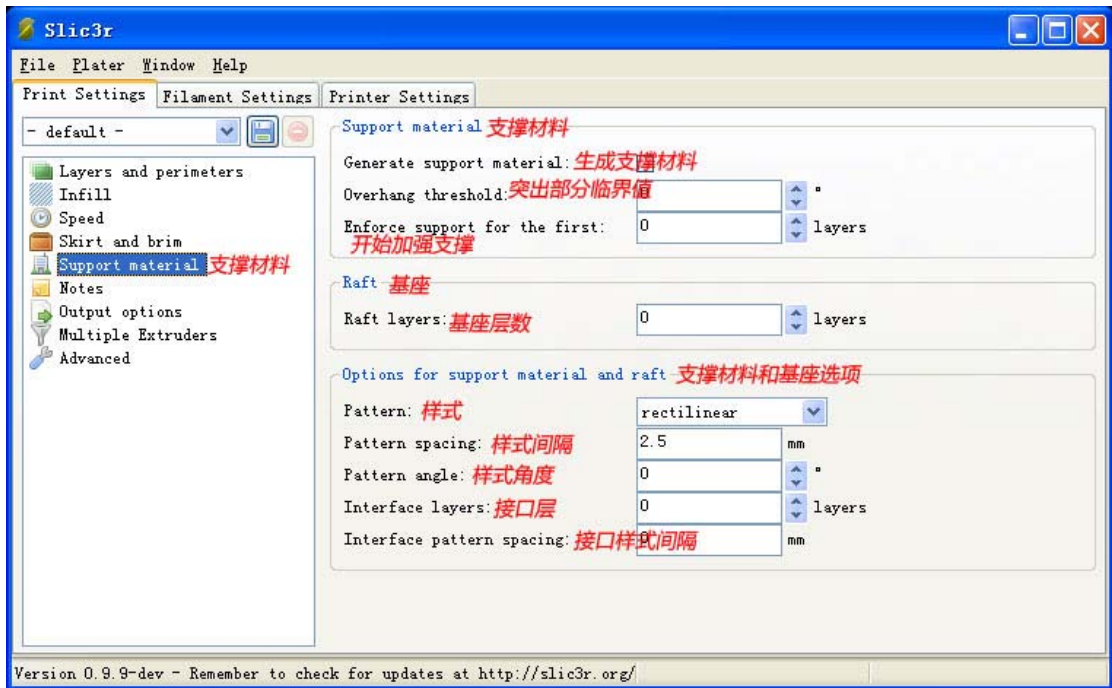
There are many options in the "SPEED" item, but you can recognize them easily. The value of the First layer speed cannot be set too high, otherwise the filament would not adhere on the hotbed, this is the most important one.

The fourth item is Skirt and Brim



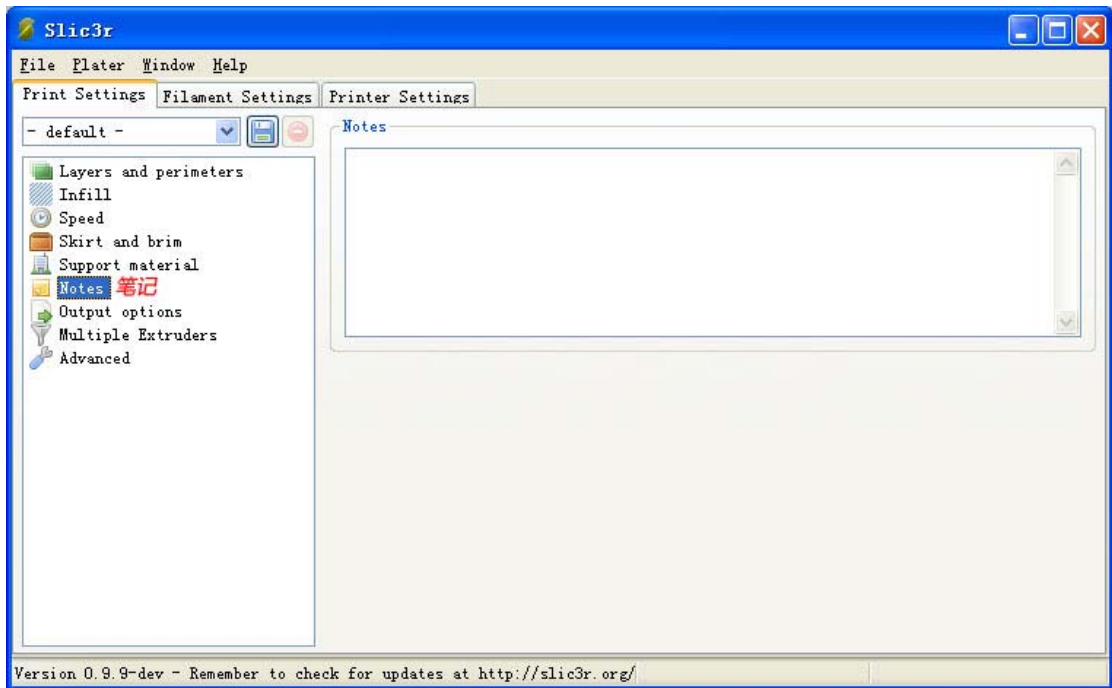
This option is to set the moves of the nozzle at the beginning of printing, when the nozzle would print a circle of filament around the printed parts. It is better to save this setting, because at the beginning, the nozzle is empty, but after print that circle, the nozzle could extrude the filament normally.

The fifth item is Support material



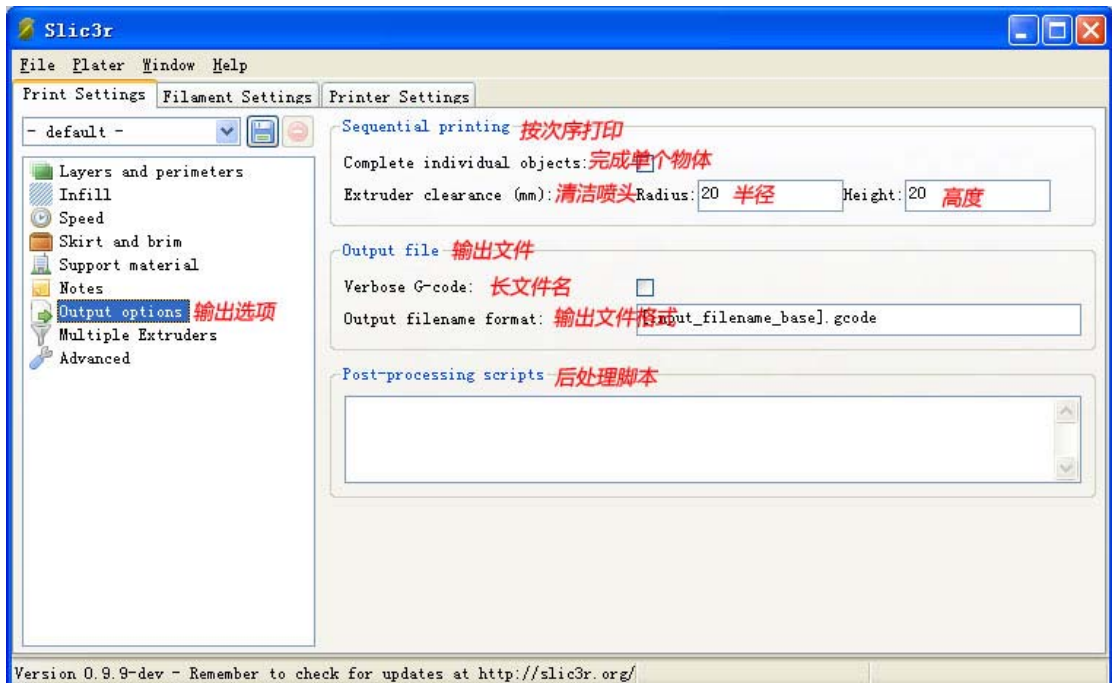
When printing, there might be bridging or cavity, if the bridging is too long, some supports is needed, which is extra, could be deleted when the print is finished. Raft is the base of your printed parts, you don' t need to set it when you are printing on the glass hotbed.

The sixth item is NOTE



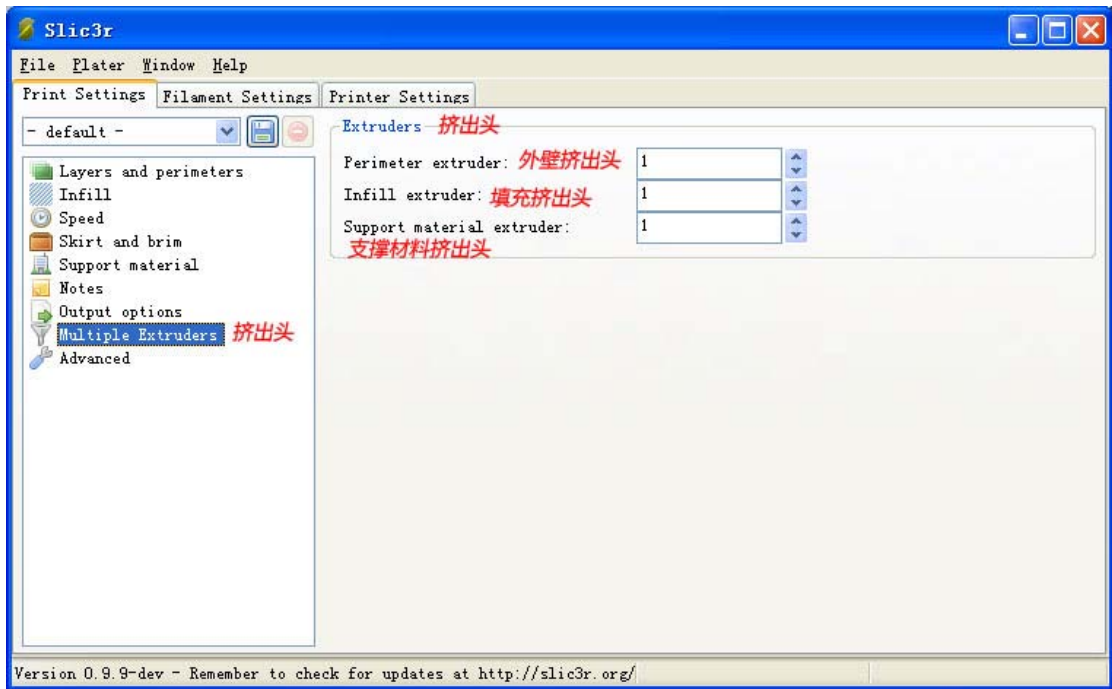
You can write anything here, just write down your diaries.

The seventh item is OUTPUT OPTIONS



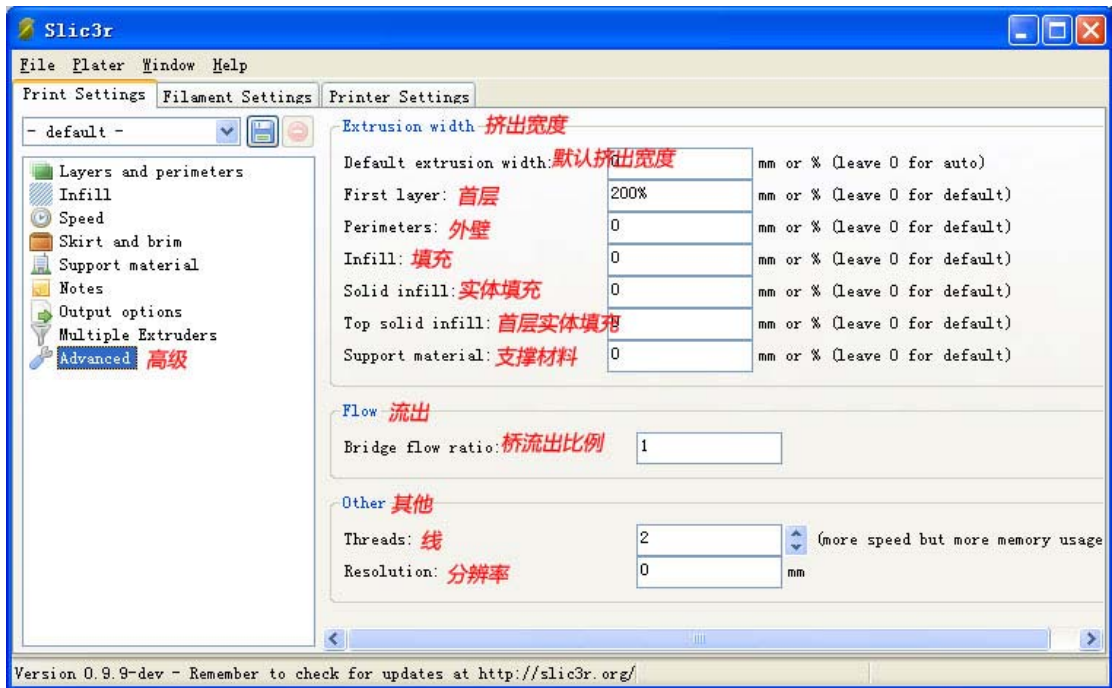
This item include SEQUENTIAL PRINTING and OUTPUT FILE, etc. You can just use the default vale.

The eighth item is MULTIPLE EXTRUDERS



If you have 2 or more extruders, set this.

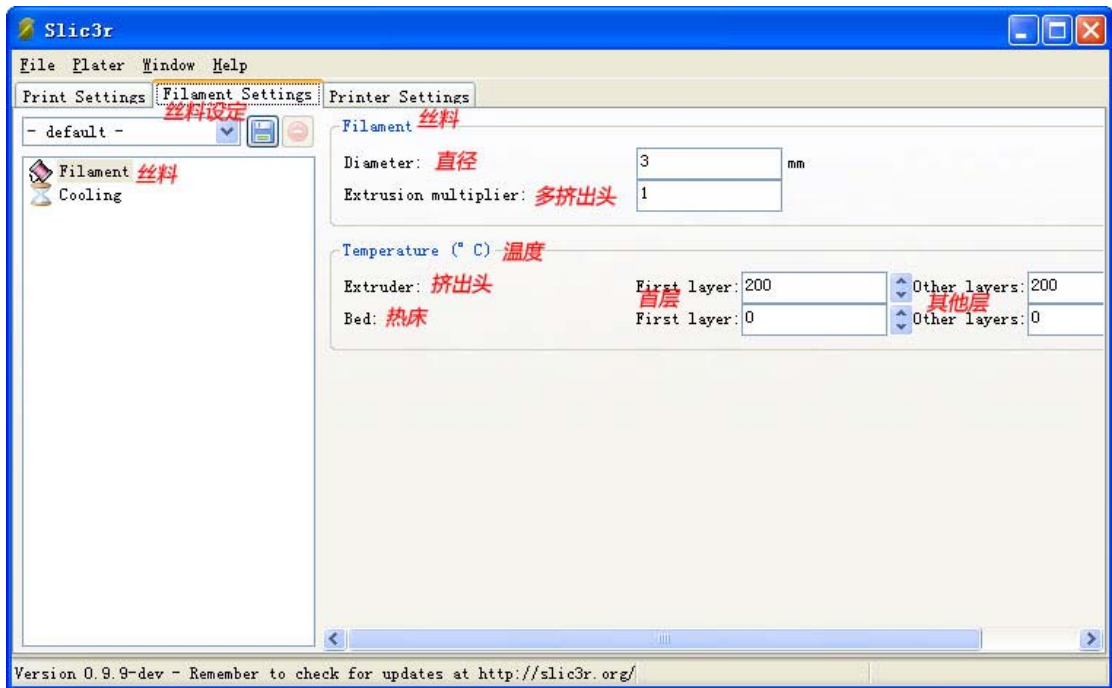
The ninth item ADVANCE



Here, we don't suggest you change anything, we have already set all for you.

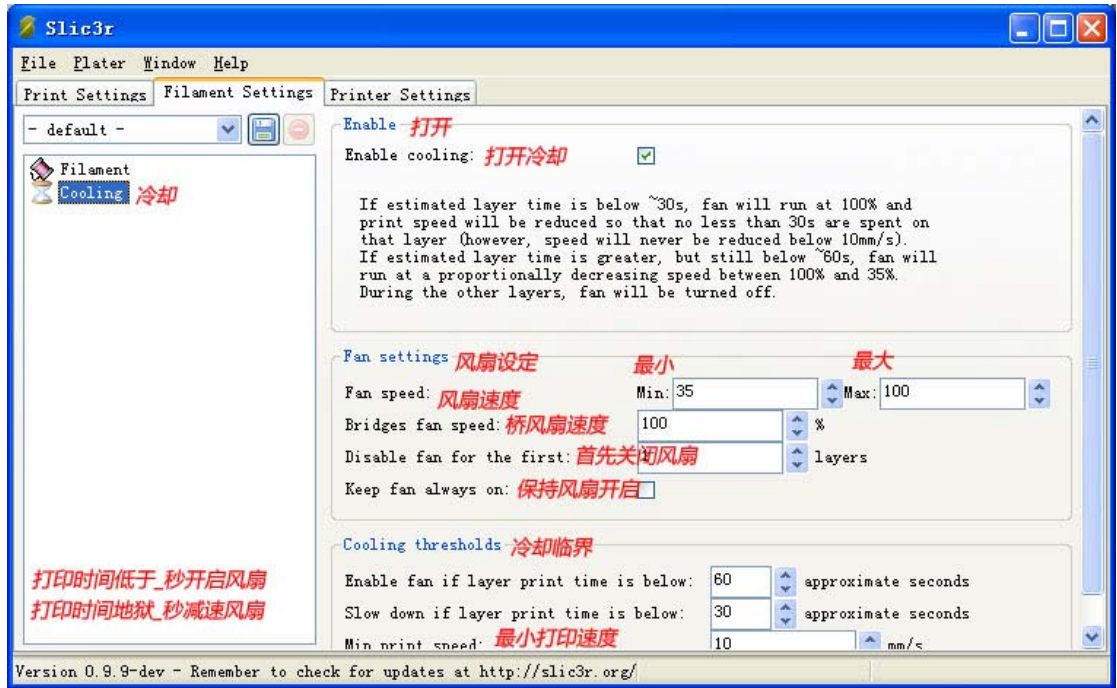
2. FILAMENT SETTINGS

The first item FILAMENT



- A、 Set the diameter as the filament diameter, generally 3mm and 1.75mm
- B、 TEMPERATURE, this set is important, because different filament, different colors. ABS should be setted like this, EXTRUDER 230°C, BED 110°C. PLA like this, EXTRUDER 185 °C , BED 55 °C . If you print the GCODE file, the machine would not print until the temperature is to the setted.

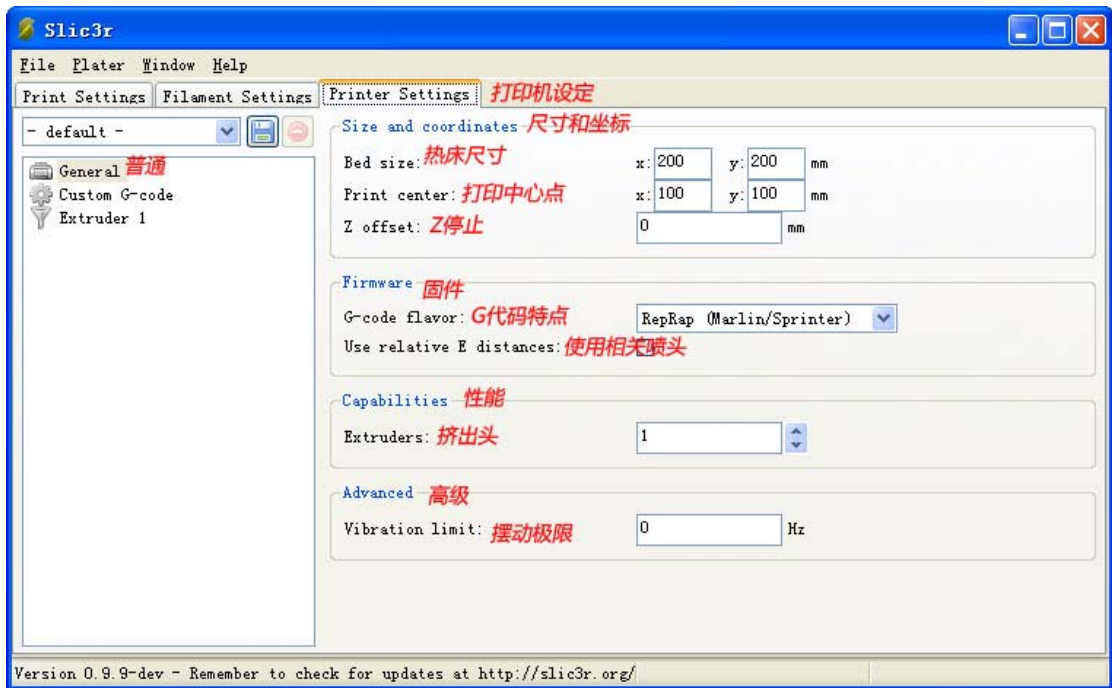
The second item COOLING



This item is better used when printing small object or print something with holes, otherwise, your printed parts would warp.

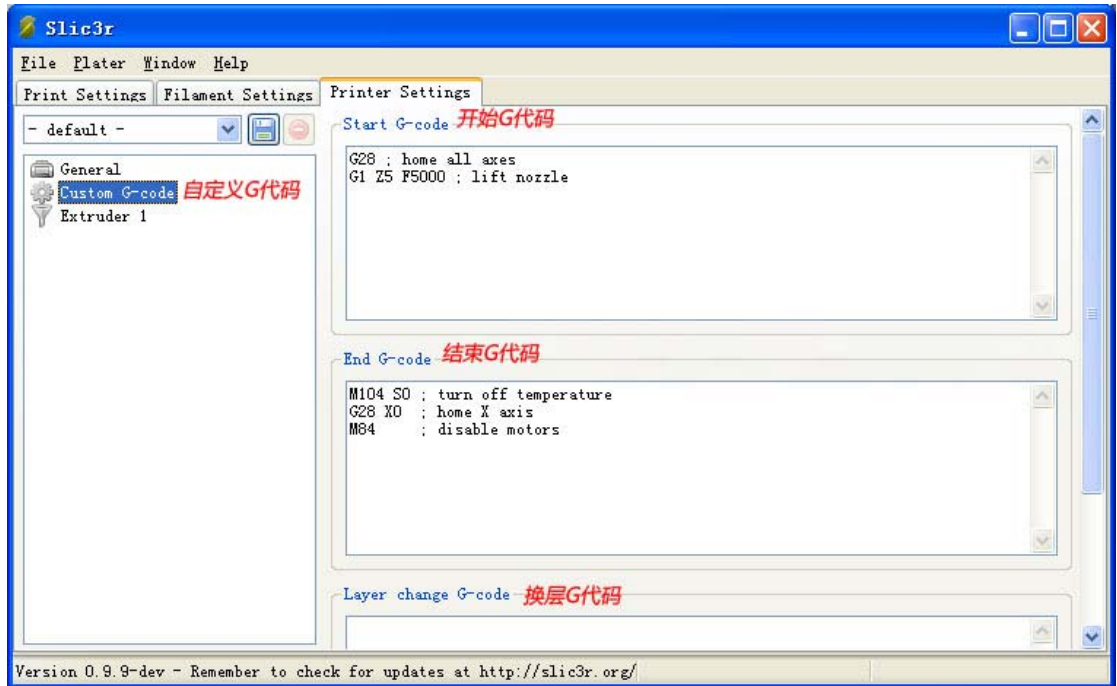
3. Printer setting

The first item is GENERAL



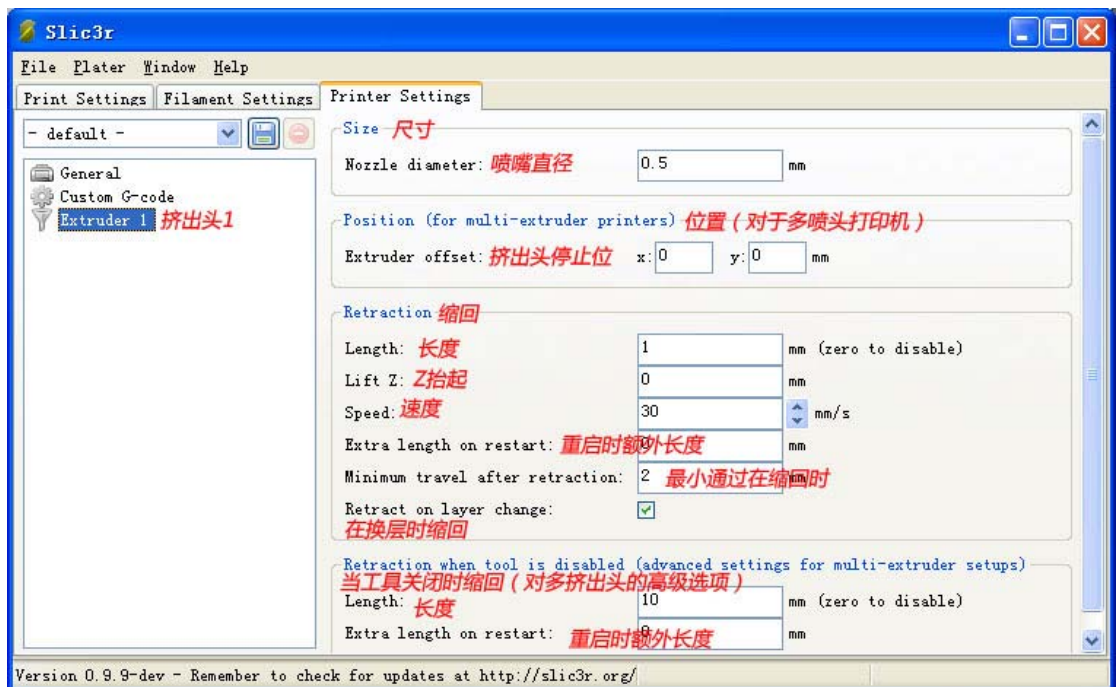
- A、 The SIZE AND COORDINATES could be setted as the practical setting
- B、 FIRMWARE, Gcode is similar to the current existed machine style, don' t change it.
- C、 CAPABILITIES AND ADVANCE, use the default value.

The second item is CUSTOM G-CODE



You mainly set the START, END and LAYER CHANGE here.

The third item is EXTRUDER



Since we need only one extruder, so set one only.

- A、 Size: filling the value according to practical situation.
- B、 POSITION: the offset after print, self-determined, but don't beyond the print limit.
- C、 RETRACTION: after print one side, the extruder will retract. You can set here, but it is better to use our default value.
- D、 ADVANCE: For multiple extruders.