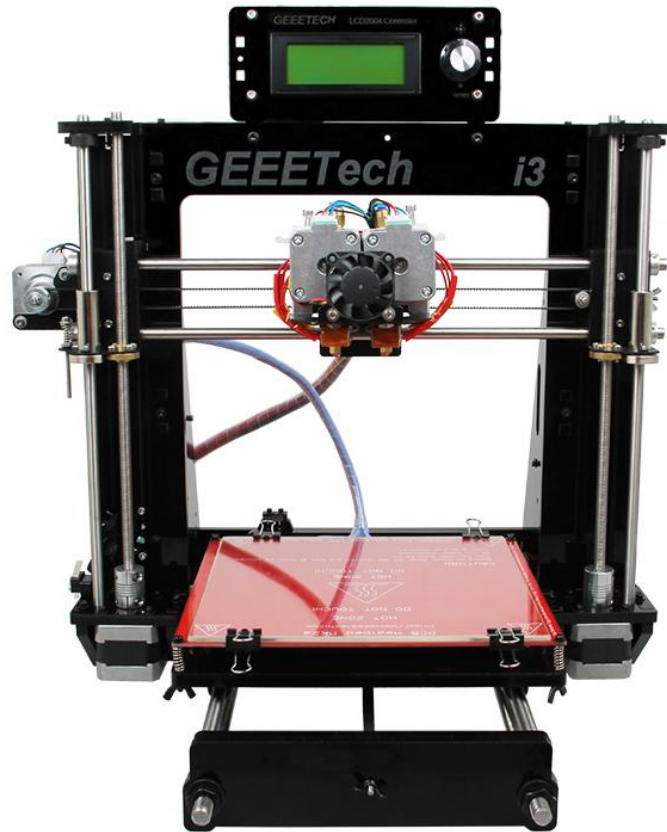


Geeetech Prusa I3 pro C

User Manual



Safety Instructions

Building the printer will require a certain amount of physical dexterity, common sense and a thorough understanding of what you are doing. We have provided this detailed instruction to help you assemble it easily.

However ultimately we cannot be responsible for your health and safety whilst building or operating the printer, with that in mind be sure you are confident with what you are doing prior to commencing with building or buying. Read the entire manual to enable you to make an informed decision.

Building and operating involves electricity, so all necessary precautions should be taken and adhered to, the printer runs on 12V supplied by a certified power supply, so you shouldn't ever have to get involved with anything over 12V but bear in mind there can still be high currents involved and even at 12V they shouldn't be taken lightly.

Check the power input of the power supply unit. You can choose either 110V or 220V according to the Power regulation in your country.

High temperatures are involved with 3D Printing, the Extrusion nozzle of the hot end can run about 230 °C, the heated bed runs 110 °C and the molten plastic extruded will initially be at around 200 °C, so special care and attention should be made when handling these parts of the printer during operation.

We wouldn't recommend leaving your printer running unattended, or at least until you are confident to do so. We cannot be held responsible for any loss, damage, threat, hurt or other negligent result from either building or using the printer.

Contents

Safety Instructions	2
1. Software Resources	1
1.1 Repetier-Host.....	1
1.2 Driver.....	1
1.3 Arduino IDE	5
2. Connect the Printer	5
3. Printer Setting	8
3.1 Printer Speed.....	8
3.2 Number of Extruder and Nozzle Diameter	9
3.3 Printer Shape.....	11
4. Printer Function Test.....	14
4.1 Use Repetier Host to Test	14
4.1.1 Motor Direction Test.....	14
4.1.2 Heating Test	15
4.1.3 Extruder Test.....	17
4.2 Use LCD Control Panel to Test	20
4.2.1 Motor Direction Test.....	20
4.2.2 Heating Test	23
5. Hotbed Leveling	25
5.1 Homing	25
5.2 Coarse Tuning of distance between the nozzle and the hotbed.....	26
5.3 Fine Tuning.....	27
6. Dual-extruder leveling	28
7. Slice Settings	28
7.1 Printing Setting.....	30
7.2 Filament Setting.....	31
7.3 Printer Setting	32

7.4 Other Settings	36
8. Begin Printing	39
8.1 print with one extruder.....	39
8.1.1 Load Print Model.....	40
8.1.2 Model Slicing.....	41
8.2 Print with Dual-extruder	44
8.3 Stand-alone print with SD Card.....	50
8.3.1 Save	51
8.3.2 Generate file.....	51
8.3.3 Print	52
9. FAQ	54
9.1 How to upload firmware?	54
9.2 How to change the motor direction in firmware?	55
9.3 User manual of Repetier Host.....	56
9.4 Possible causes for motor's non-action.....	56
9.5 Extruder does not work normally	56
9.6 The hotbed or extruder can not be heated.....	57
9.7 LCD error: MAX/MIN TEMP error	57
9.8 No information or displaying black square on LCD.....	57
9.9 Time.h/endstop hit or other abnormalities	57

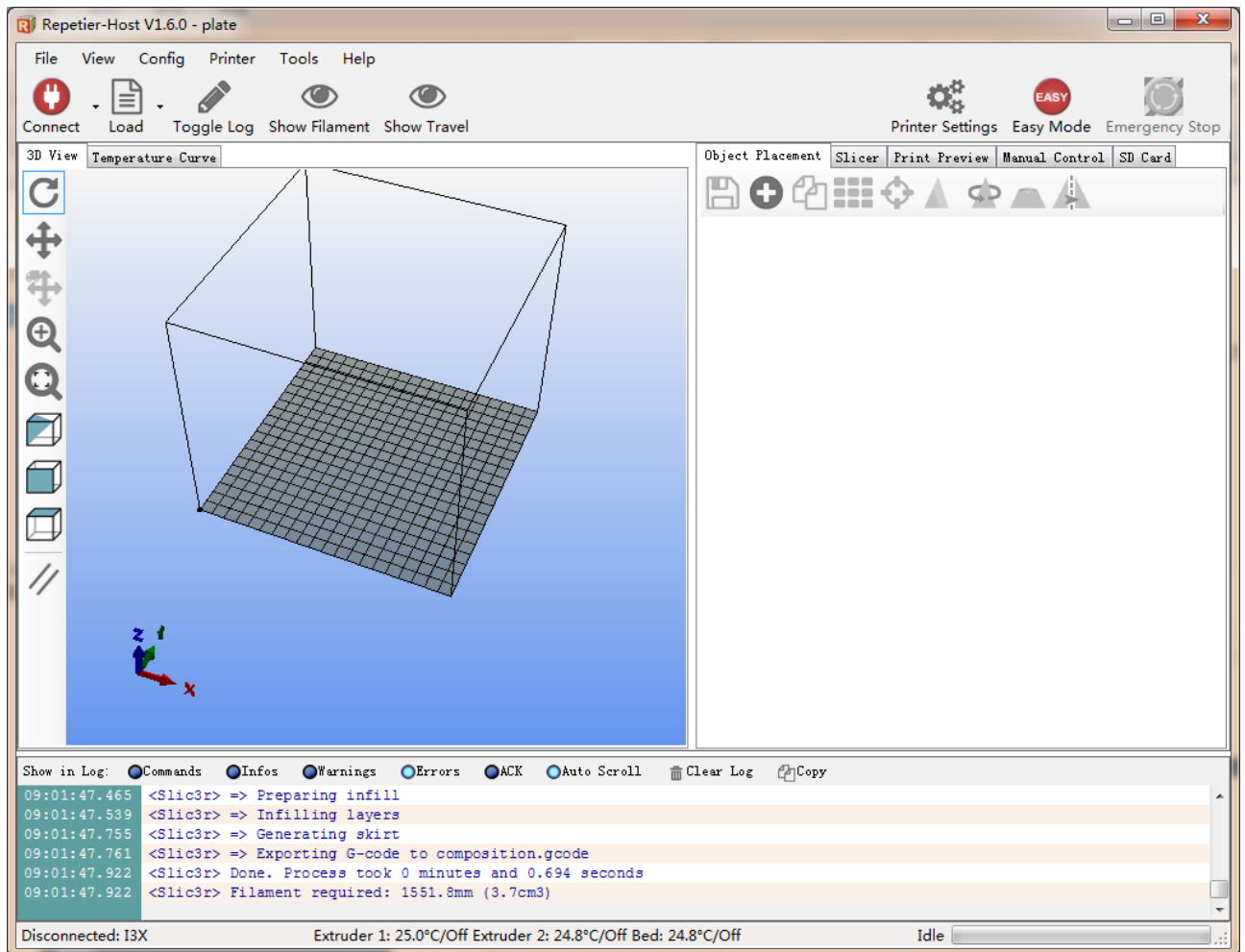
1. Software Resources

1.1 Repetier-Host

We use the latest Repetier host 1.6.0 as the printer control software, here is the link for download:

<http://www.geeetech.com/wiki/index.php/Repetier-Host#Download>

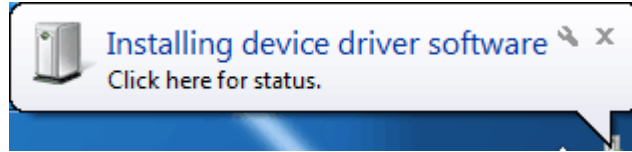
After downloading and installation, it can be used.



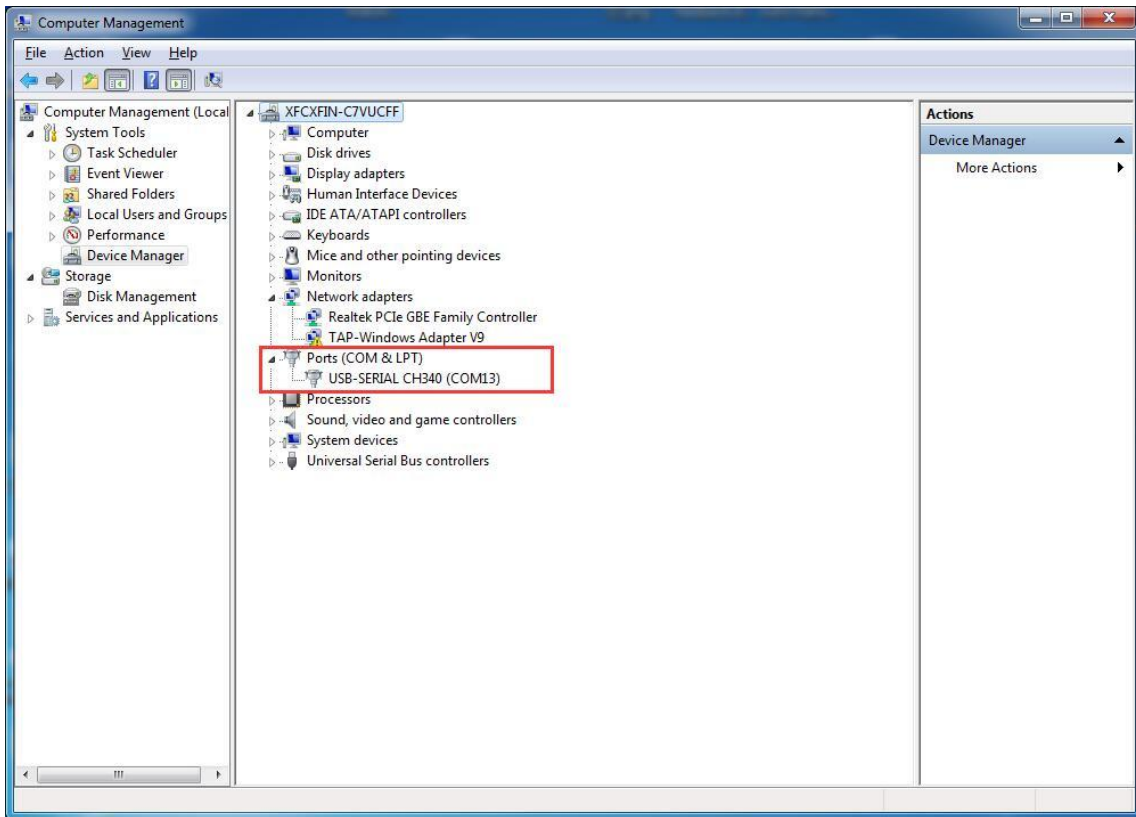
1.2 Driver

Connect the printer to computer with USB cable. The motherboard driver installer will automatically appear. Please check the following steps.

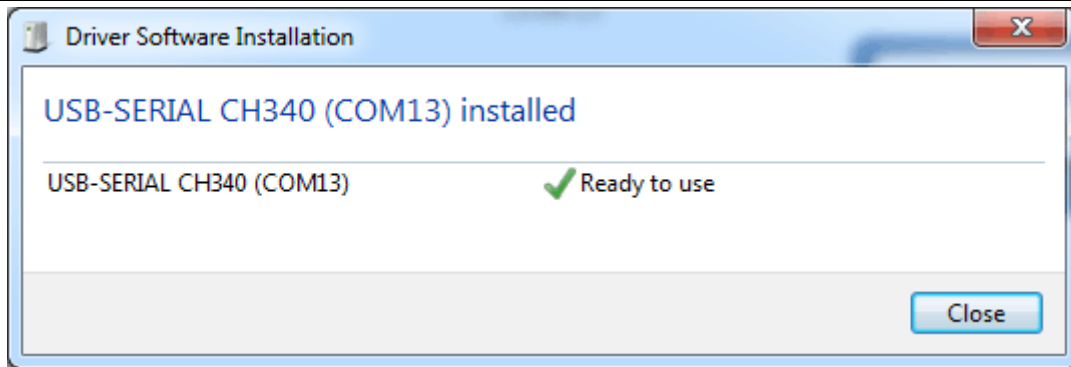
1. When you connect your printer to computer with a USB cable. The following dialog box will appear.



2. In the Arduino IDE when the CH340C is connected you will see a COM Port in [Device manager]. The COM number for your device may vary depending on your system.

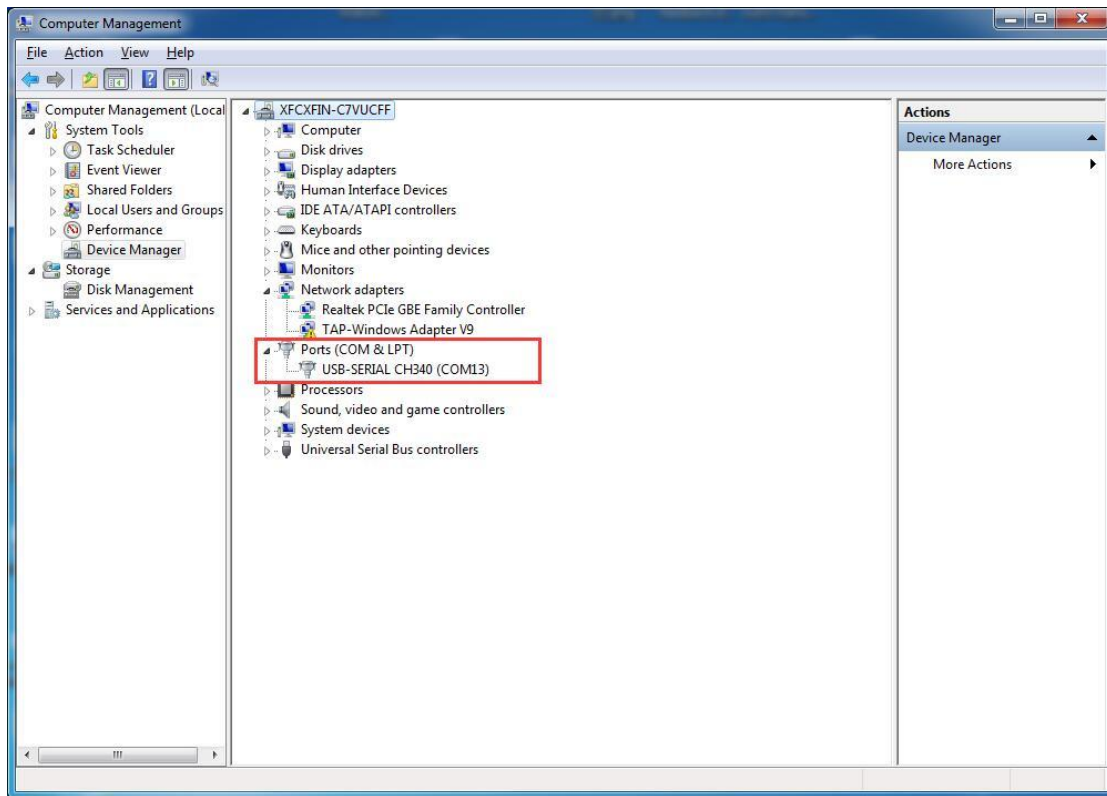


3. After successful installation, a pop-up box will appear as follows.

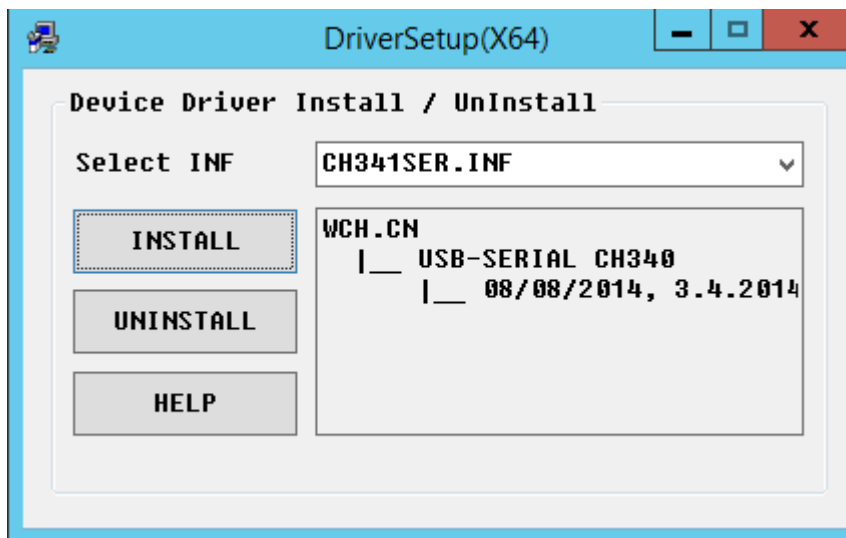


Note: If the driver fails to install automatically, you can finish the installation job manually. Here are the detailed steps.

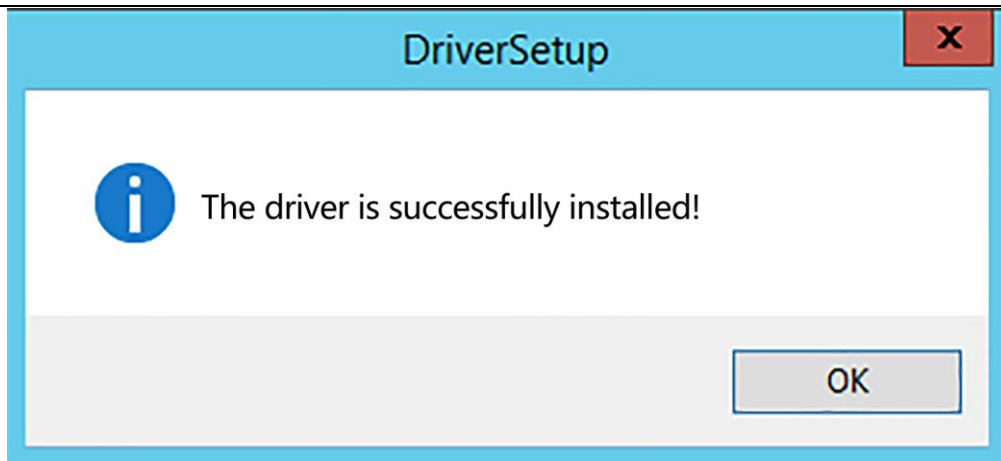
1. Download CH340C here, which is the driver of GT2560:
http://www.wch.cn/download/CH341SER_EXE.html (Windows OS)
http://www.wch.cn/download/CH341SER_MAC_ZIP.html (MAC OS)
http://www.wch.cn/download/CH341SER_LINUX_ZIP.html (LINUX OS)
2. Unzip the file.
3. Run the installer which you unzipped.
4. In the Arduino IDE when the CH340C is connected you will see a COM Port in [Device manager]. The COM number for your device may vary depending on your system.



5. Click [Install] to continue.



6. After successful installation, a pop-up box will appear.



1.3 Arduino IDE

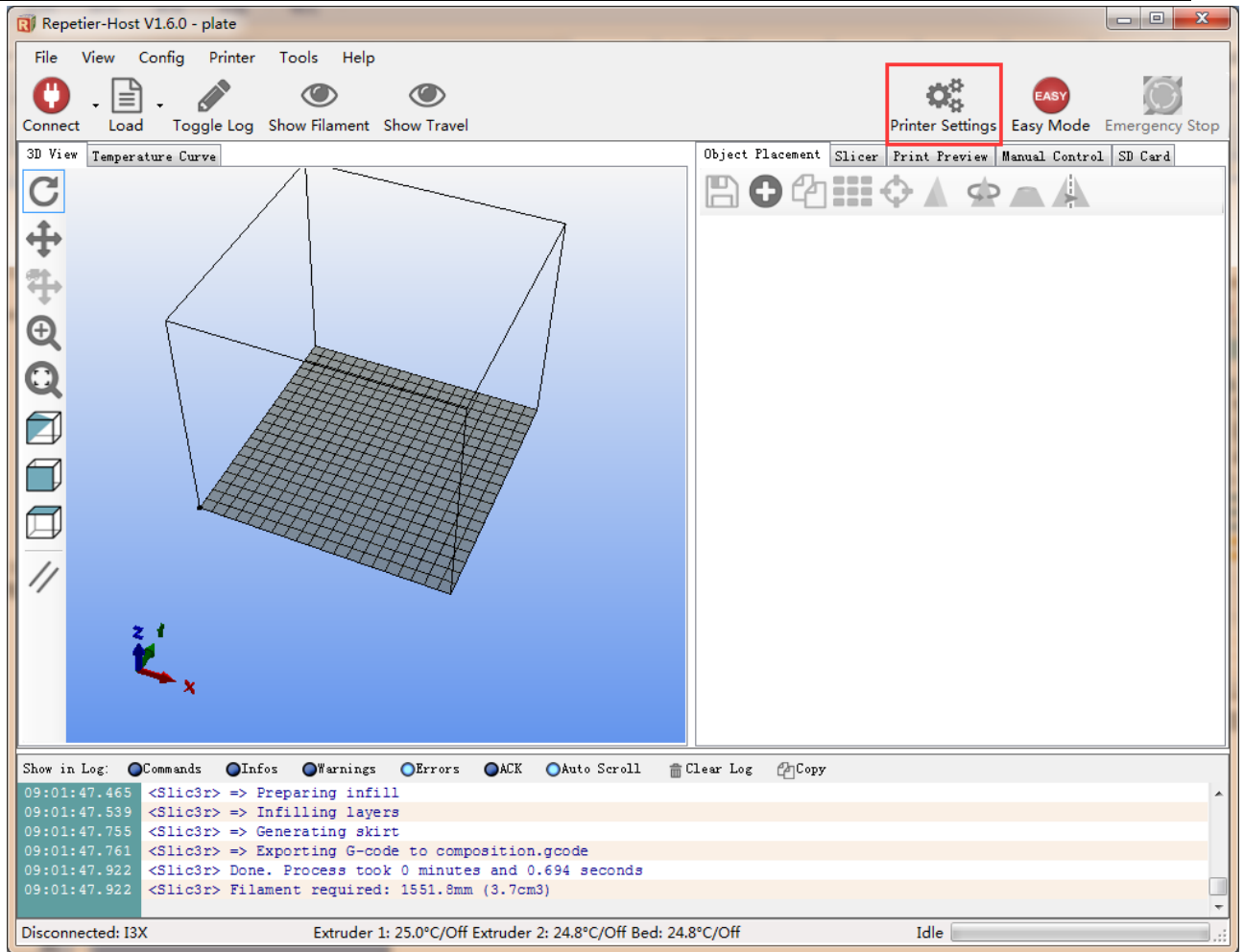
In the process of using the printer you may need to re-upload the firmware, and the software you need to use is Arduino. About how to upload firmware [please refer to FAQ](#).

2. Connect the Printer

After the above software is installed, we can connect the printer to the computer. Plug the USB into

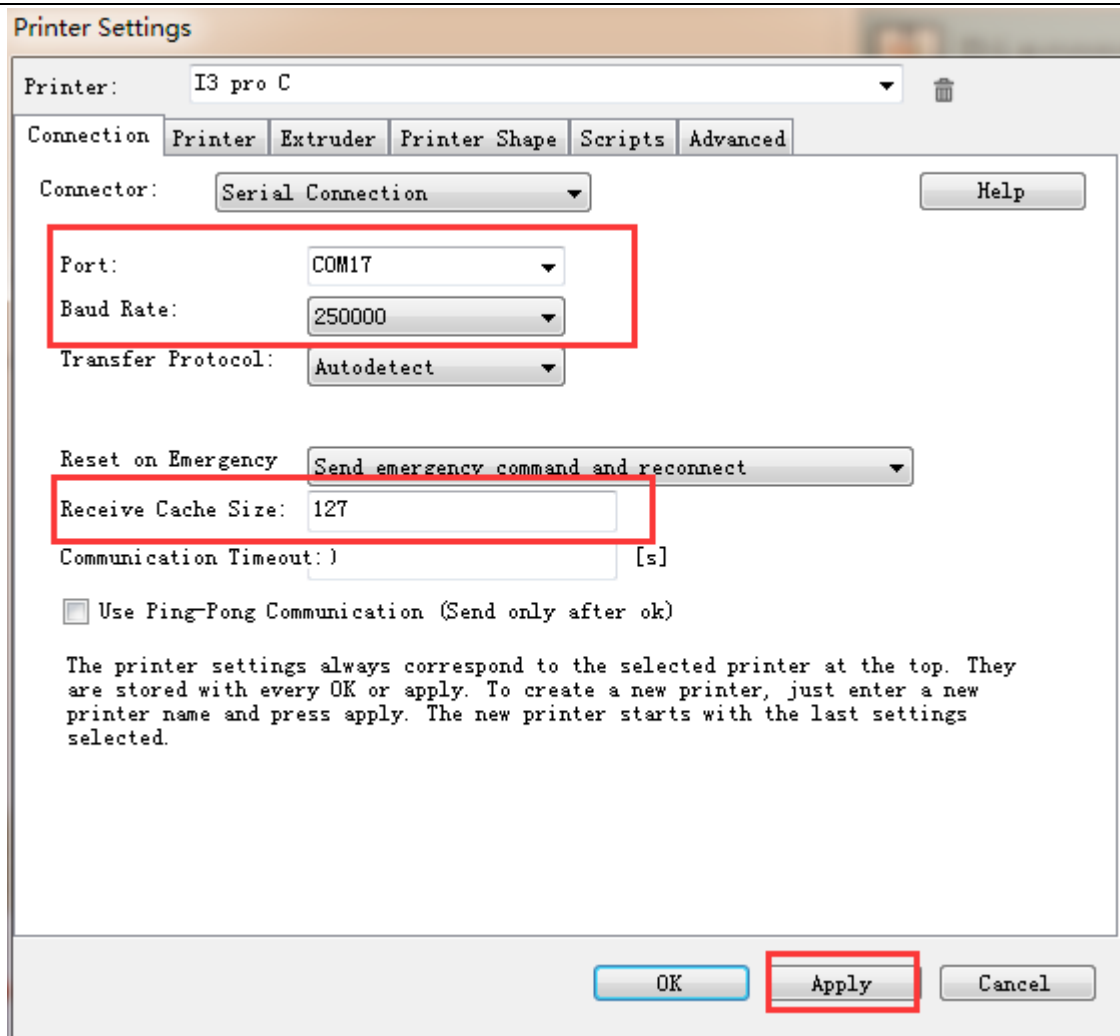



the computer, open Repetier host, click the [Printer Settings](#) icon in the top right corner, and the dialog box of printer settings will appear.

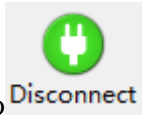


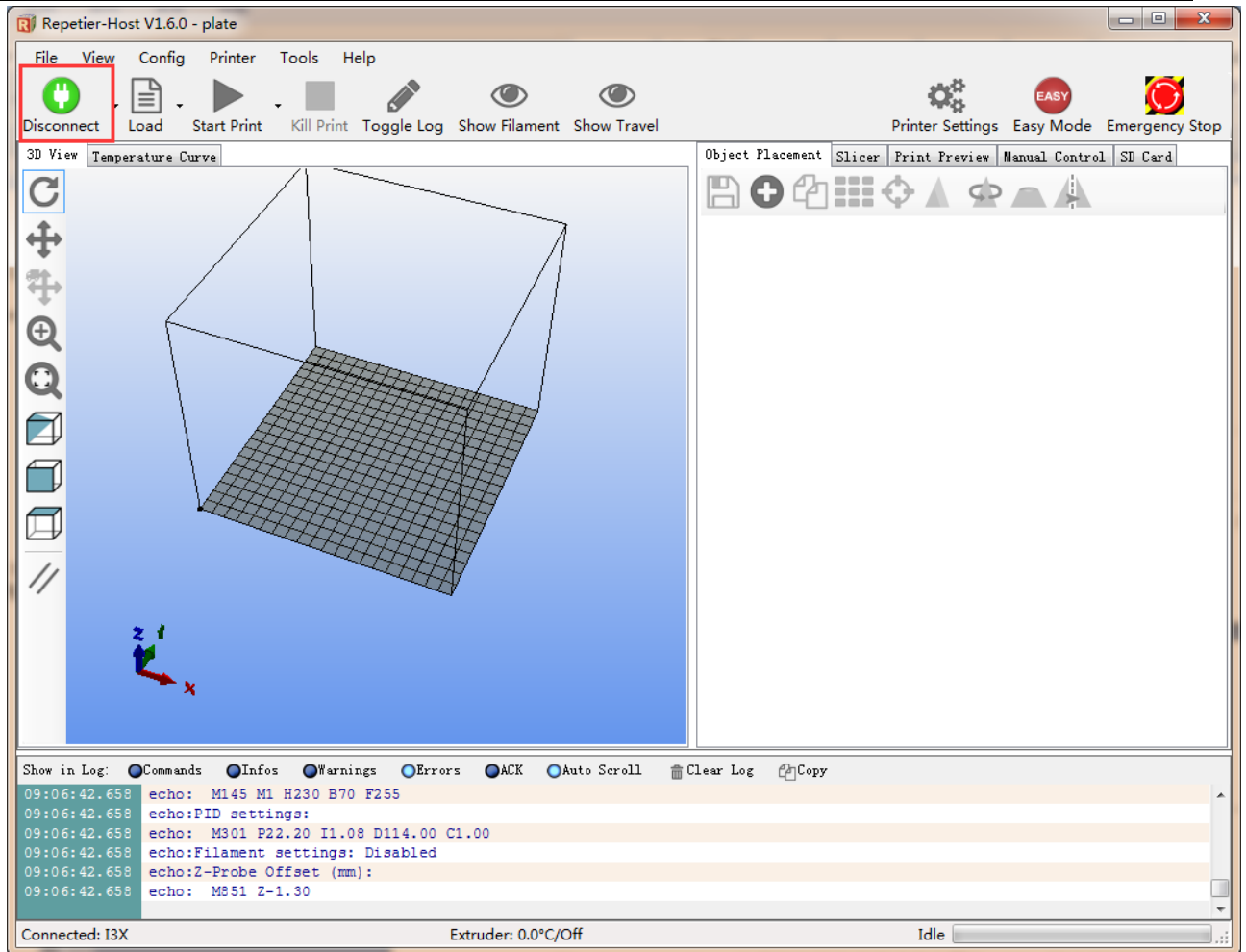
Select corresponding COM port of the printer and baud rate as 250,000 in the drop-down menu.

Click Apply after the setting. Shown as below.



After setting, click the  button in the top left corner of Home Page. When the button turns

green and changes into , the printer connection is successful.



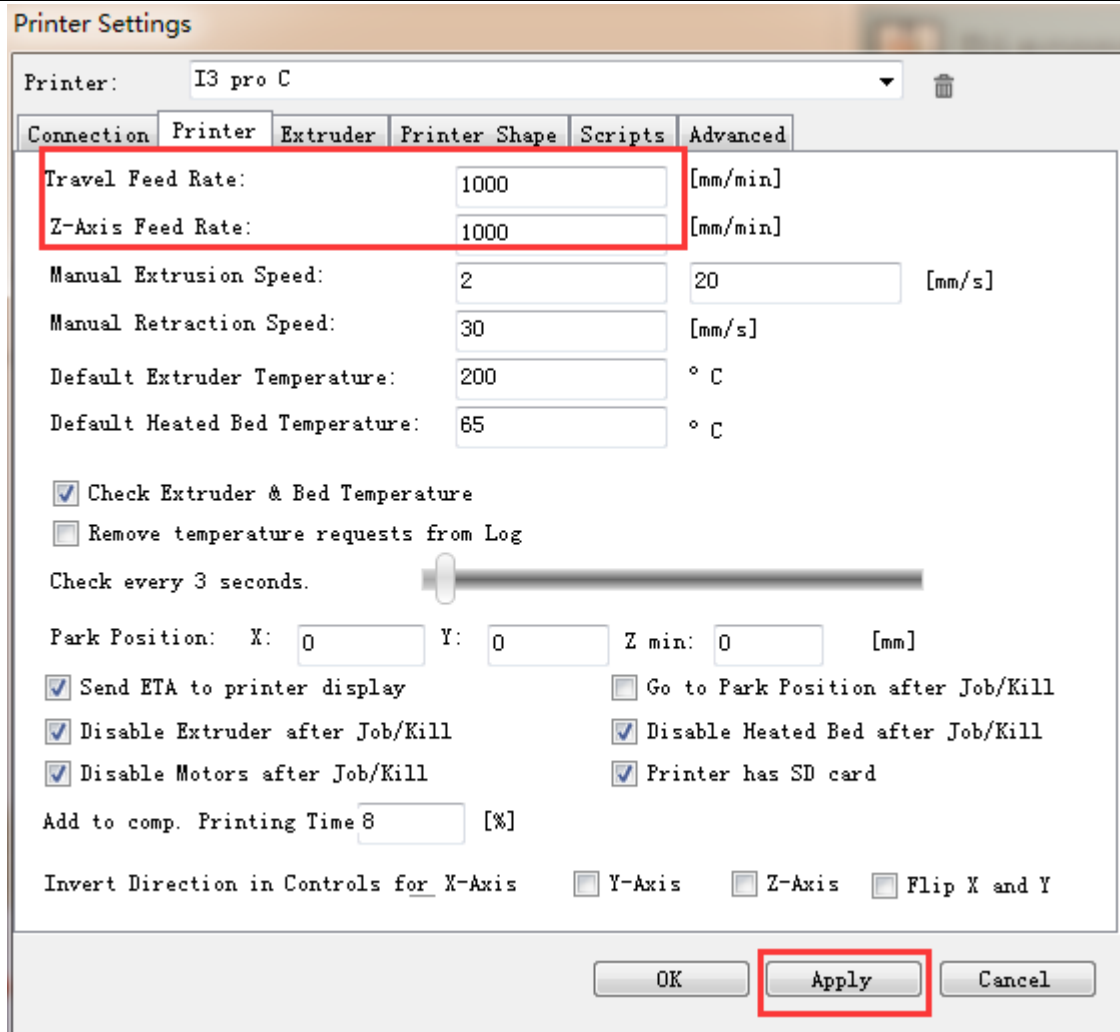
3. Printer Setting

3.1 Printer Speed

Set the running speed of printer as 1000 in the printer settings in order to avoid collision because of high speed during the test. After follow-up tests are completed it can be changed back to the original value. (Recommend speed of Z-axis should not be too high, 1000 is OK)

Travel feed rate: 1000mm/min

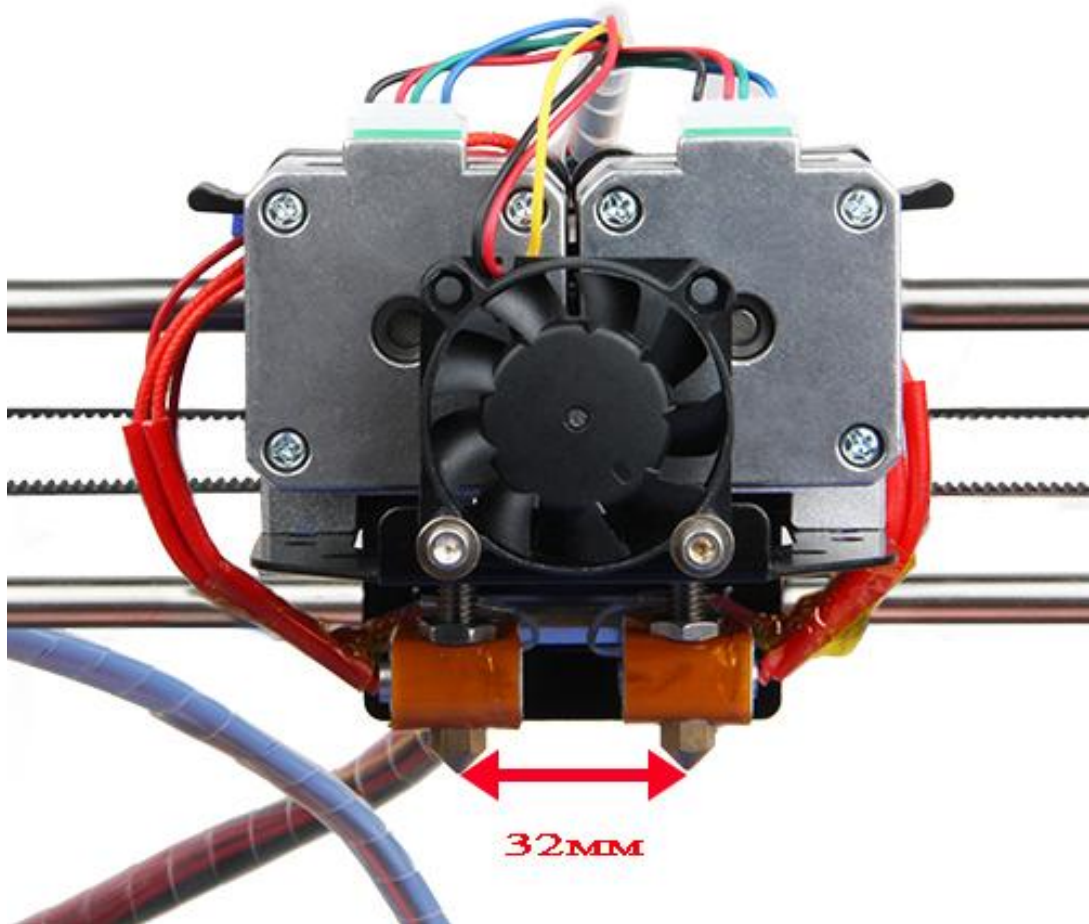
Z-Axis feed rate: 1000mm/min

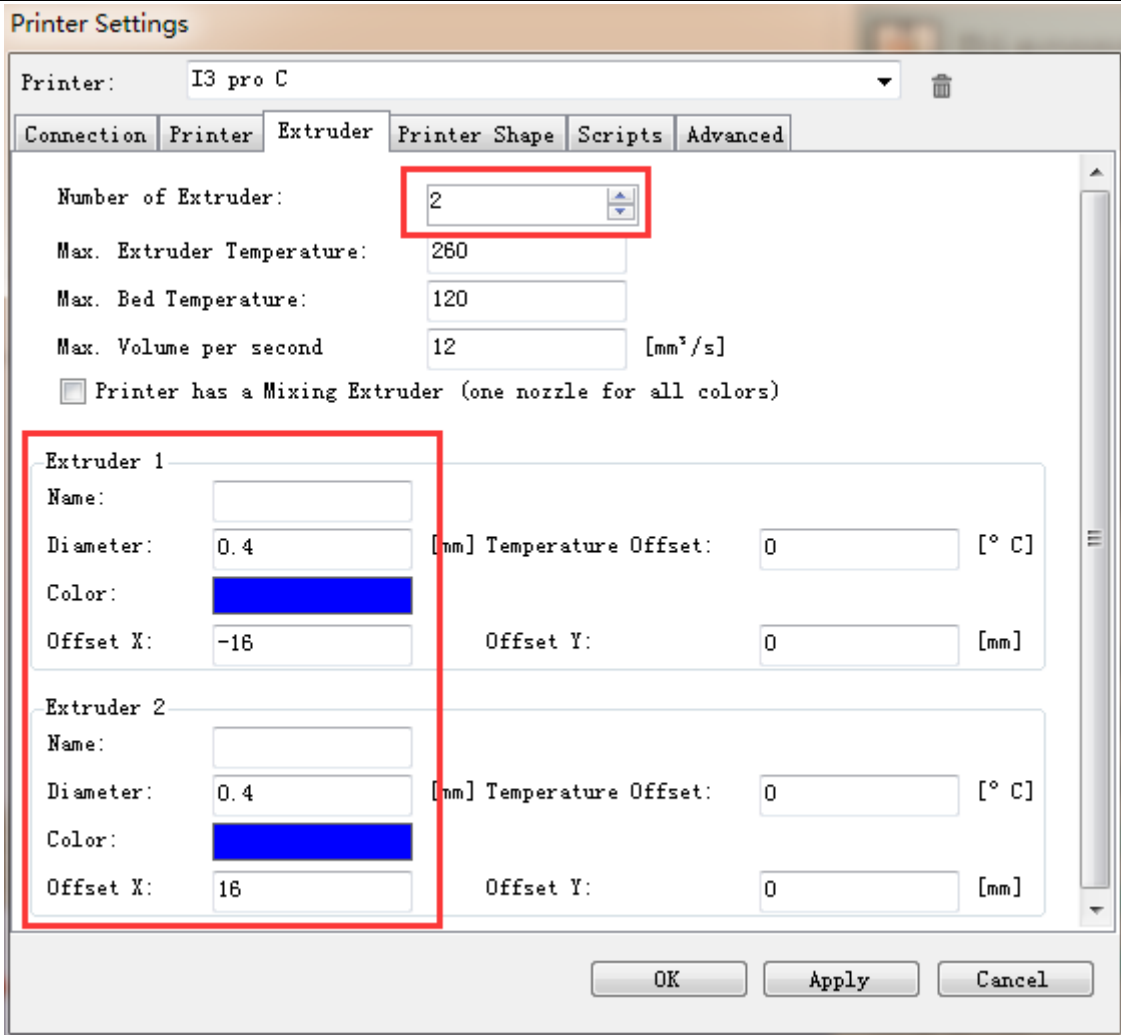


3.2 Number of Extruder and Nozzle Diameter

The number of extruder is set as 1. The diameter of print head is set according to the actual size, here is 0.4mm.

The distance between two extruders on X direction is 32mm, then set offset X -16 and 16mm. (You can also set it depending on actual measurement)



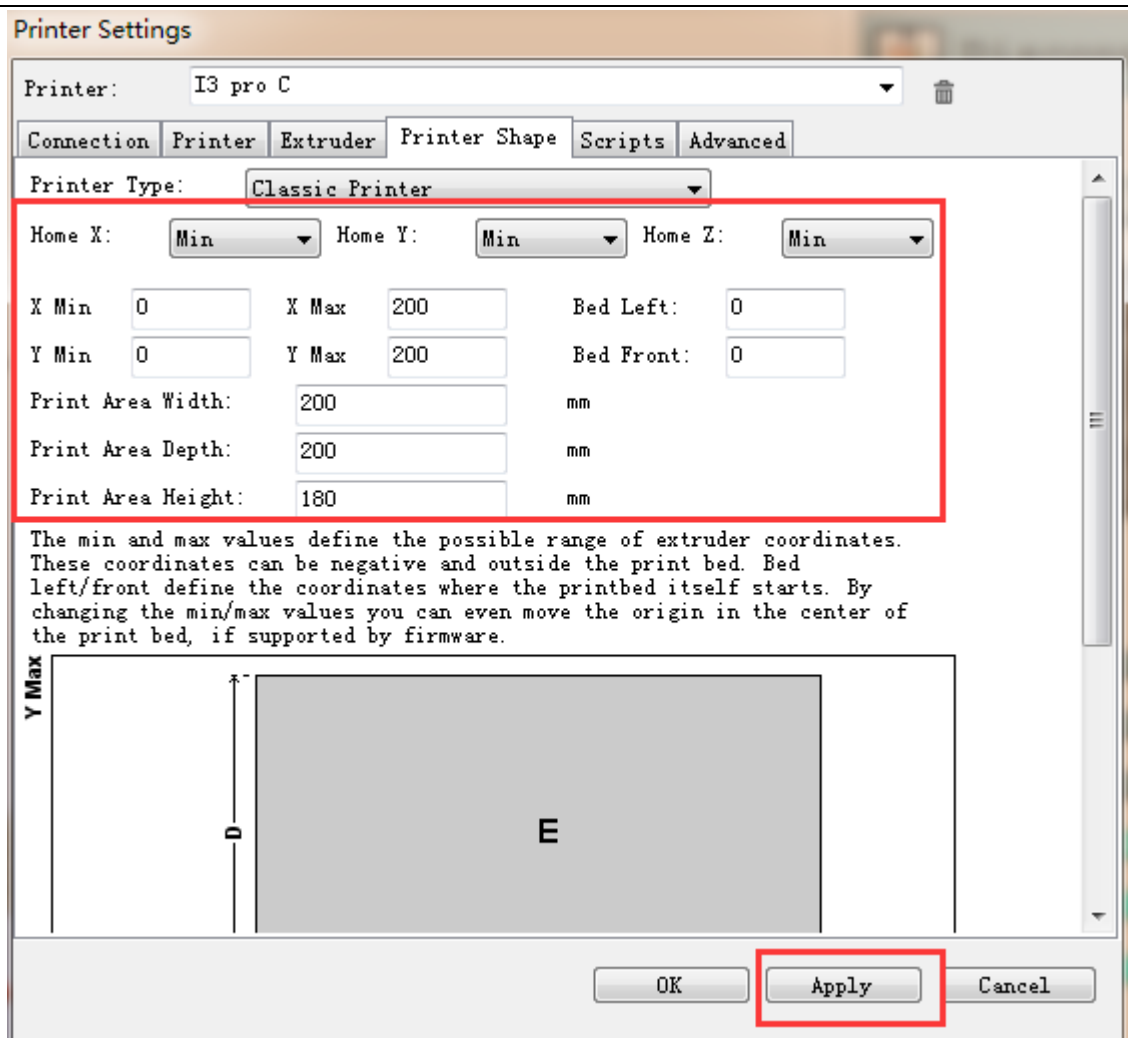


3.3 Printer Shape

Here the printer shape of I3 pro C is Classic Printer;

For home position, choose Min;

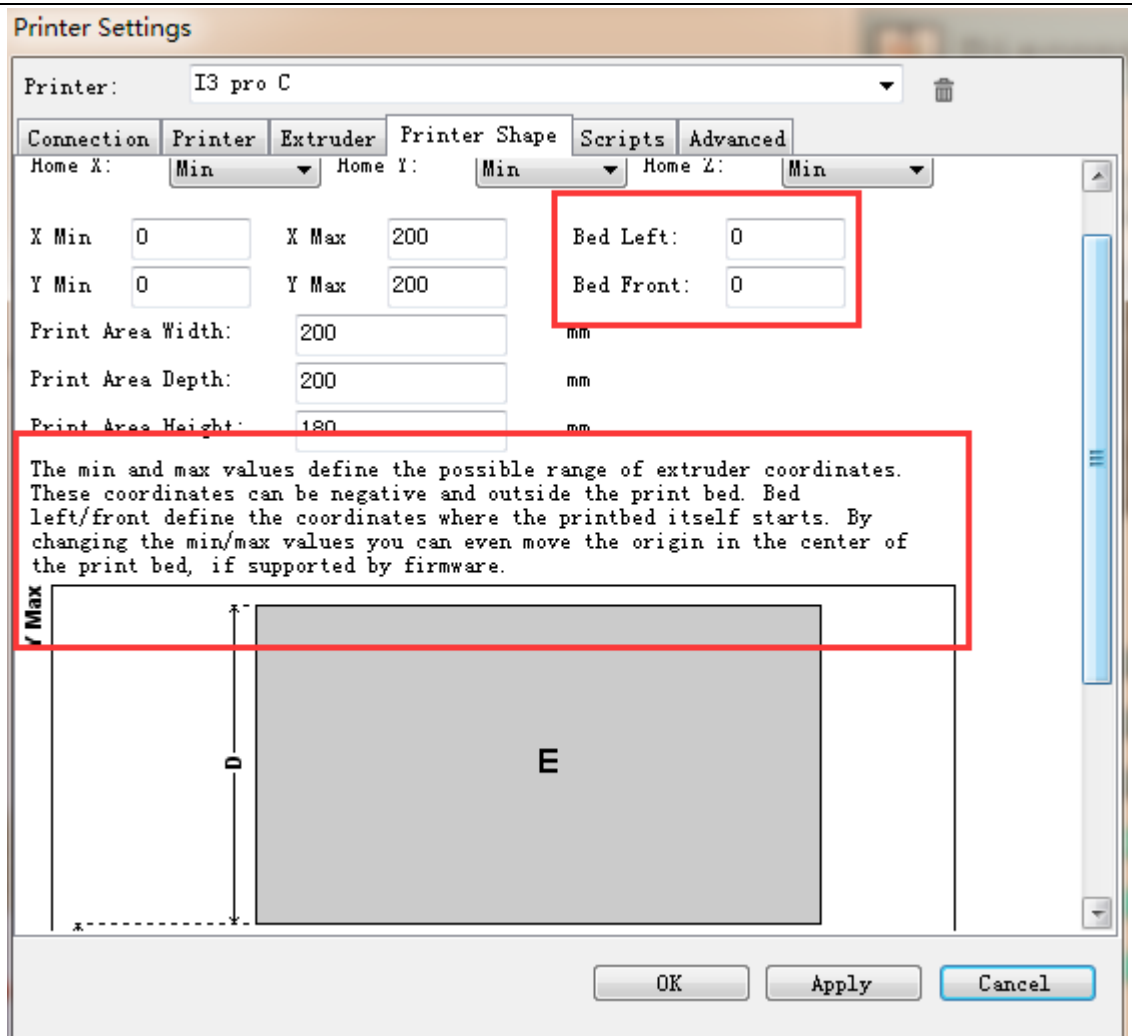
The printer's maximum printing volume 200 * 200 * 180 (length / width / height)



If the object which is being printed is not in the center of the hot bed, you can adjust the following two values to compensate for the distance deviation of X / Y direction:

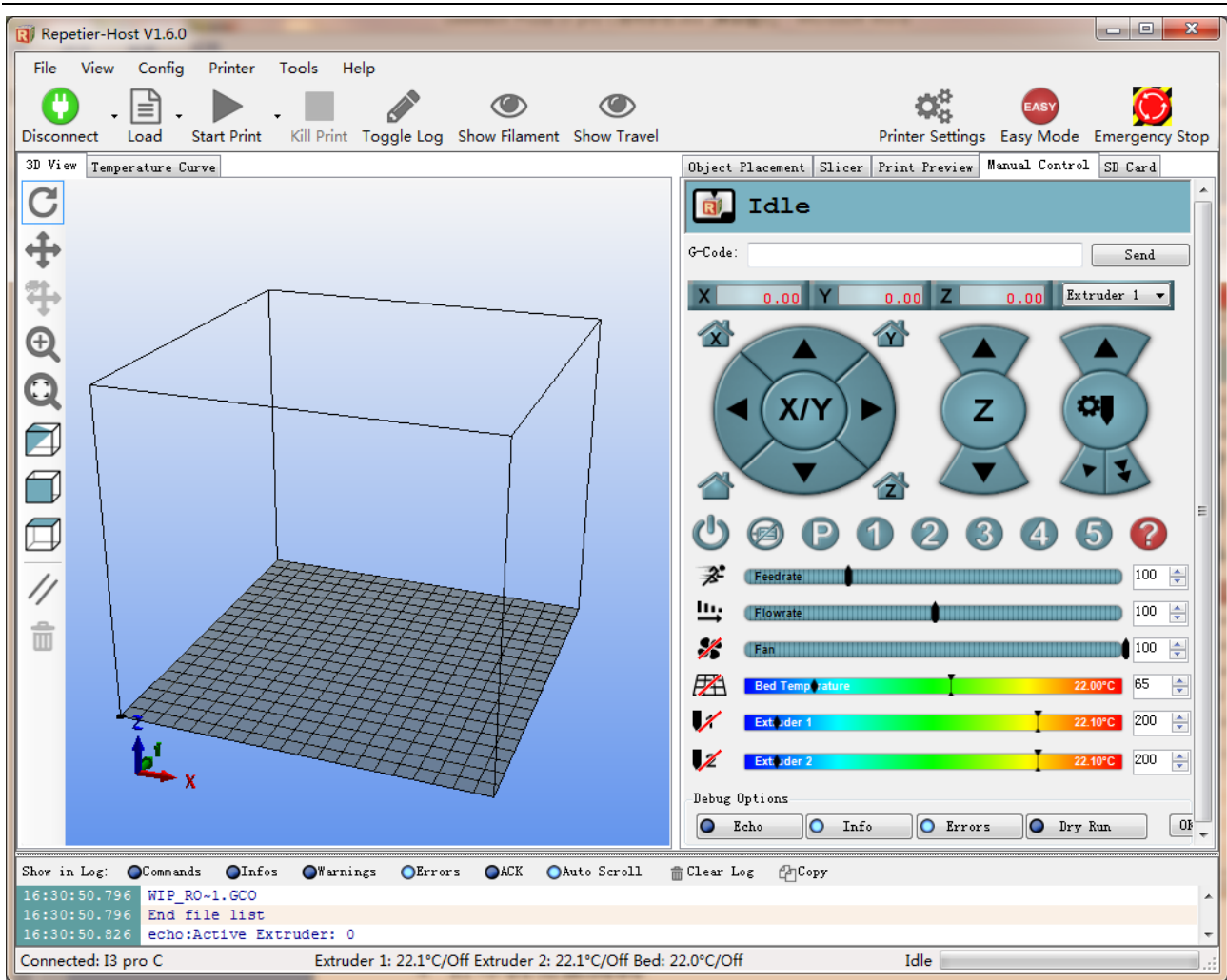
Bed Left

Bed Right



After setting, click Apply>OK

The interface after setting is as following:



4. Printer Function Test

When we are testing, we can test through LCD control panel or we can also do the test on Repetier Host. The methods are as followings.

4.1 Use Repetier Host to Test

4.1.1 Motor Direction Test

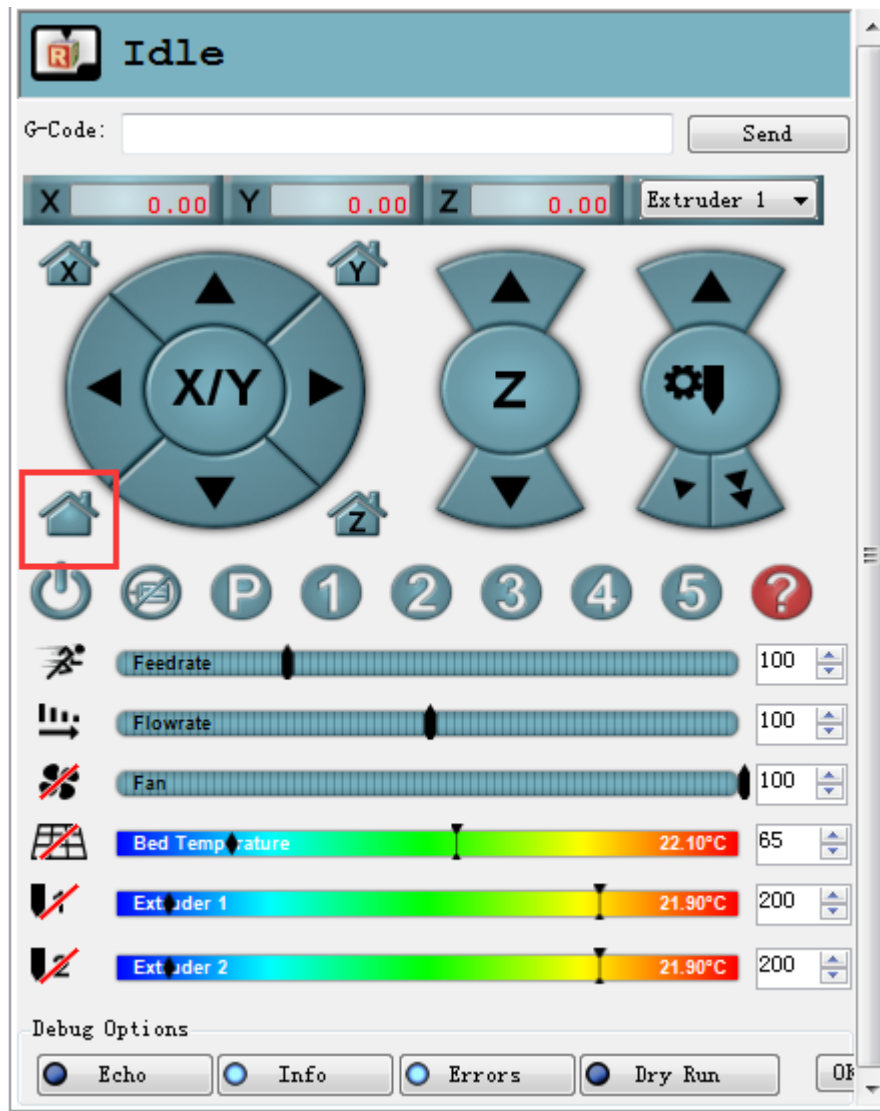
Before the test, manually set the motors of X / Y / Z axis at the intermediate position of each axis to avoid accidental collision occurring during the test. There is emergency stop button at the top right





corner, or you can cut off the power. Always be prepared for emergency stop.

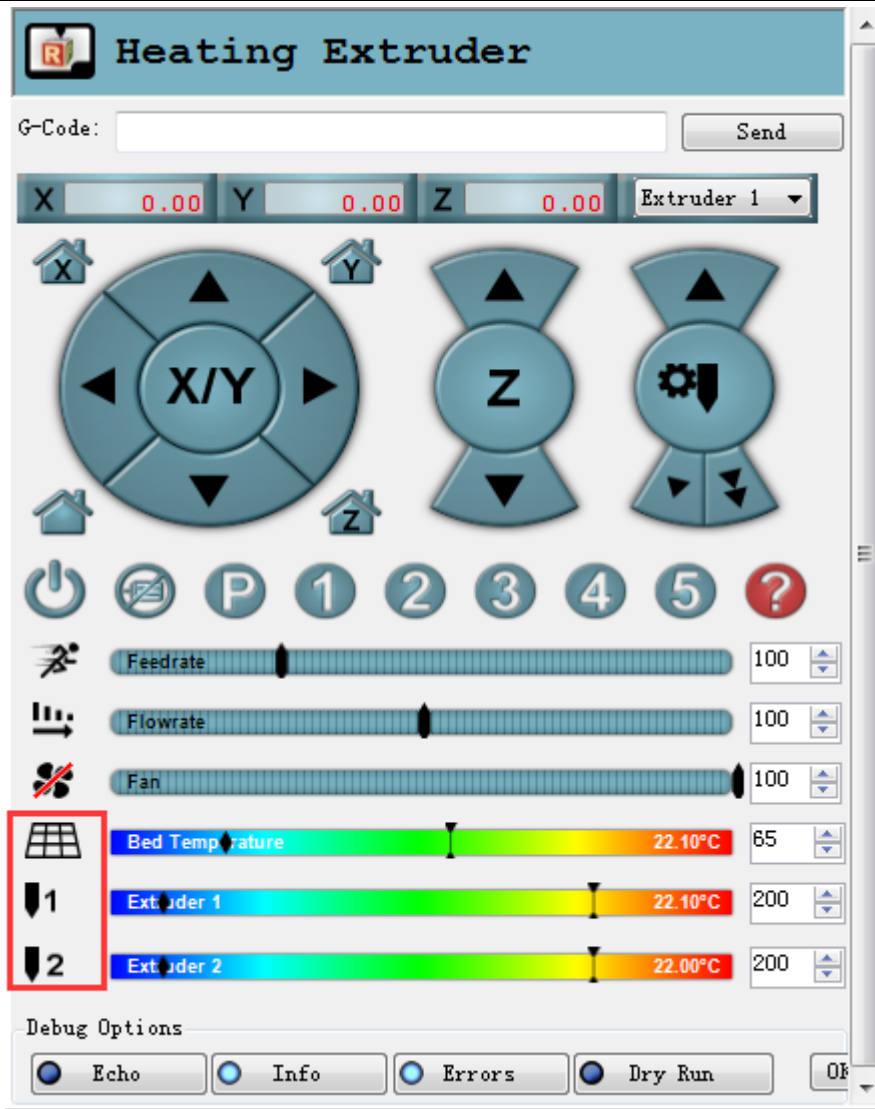
Open Repetier host and connect the printer. Click Home button , then all three axes will move

towards the endstop. After they hit the endstop they will move back for a short distance and then stop the movement. The origin location should be in the right front corner of the printer platform. If the moving direction is reversed, you can change the motor direction in firmware. ([please refer to FAQ](#)).



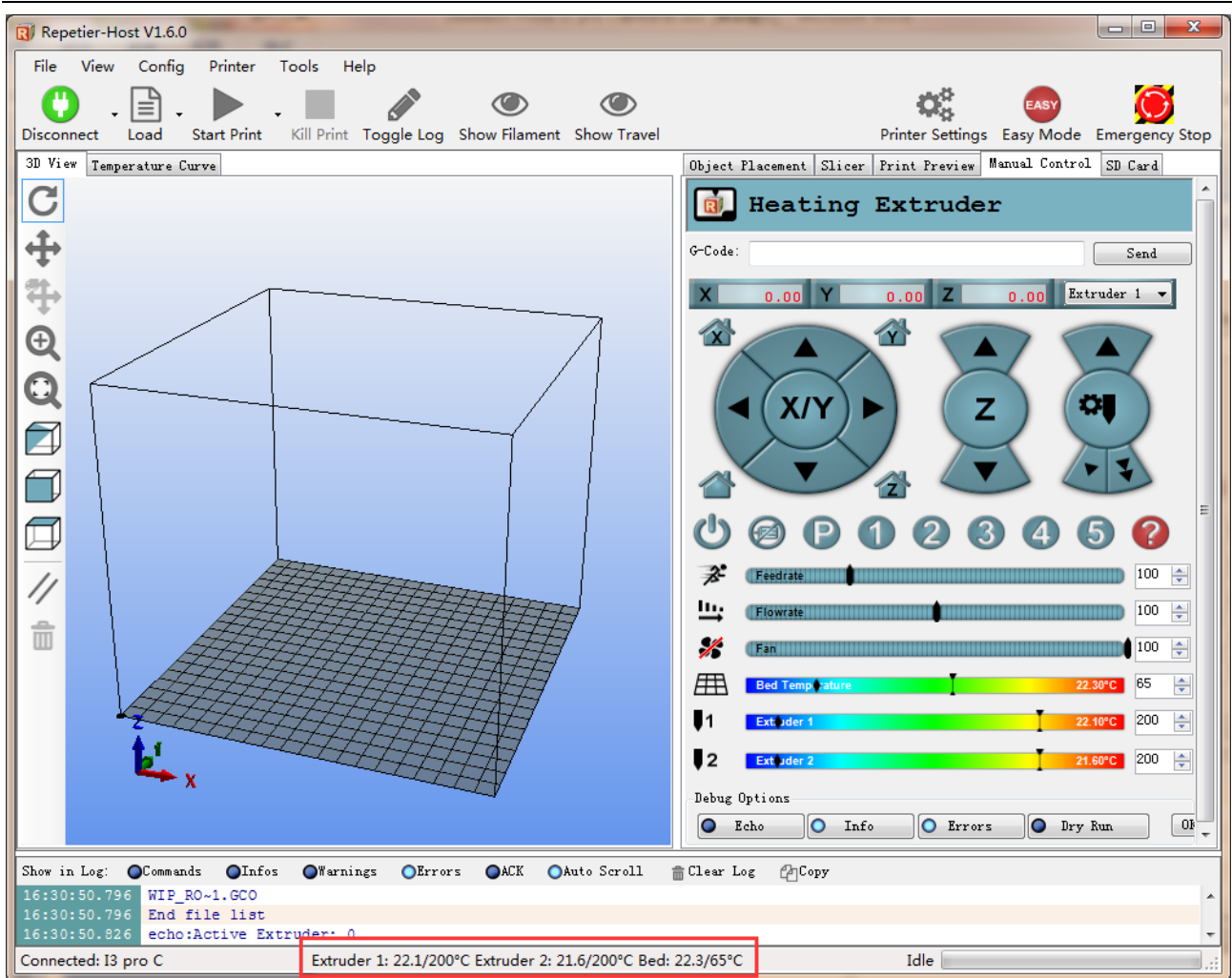
4.1.2 Heating Test

Click the heating button of the hot bed  and the extruder . The slash will disappear and you will see the temperature rising up.



Meanwhile you can see the temperature changing at the bottom of Repetier.

If the hot bed or extruder can not be heated, please refer to [FAQ](#).

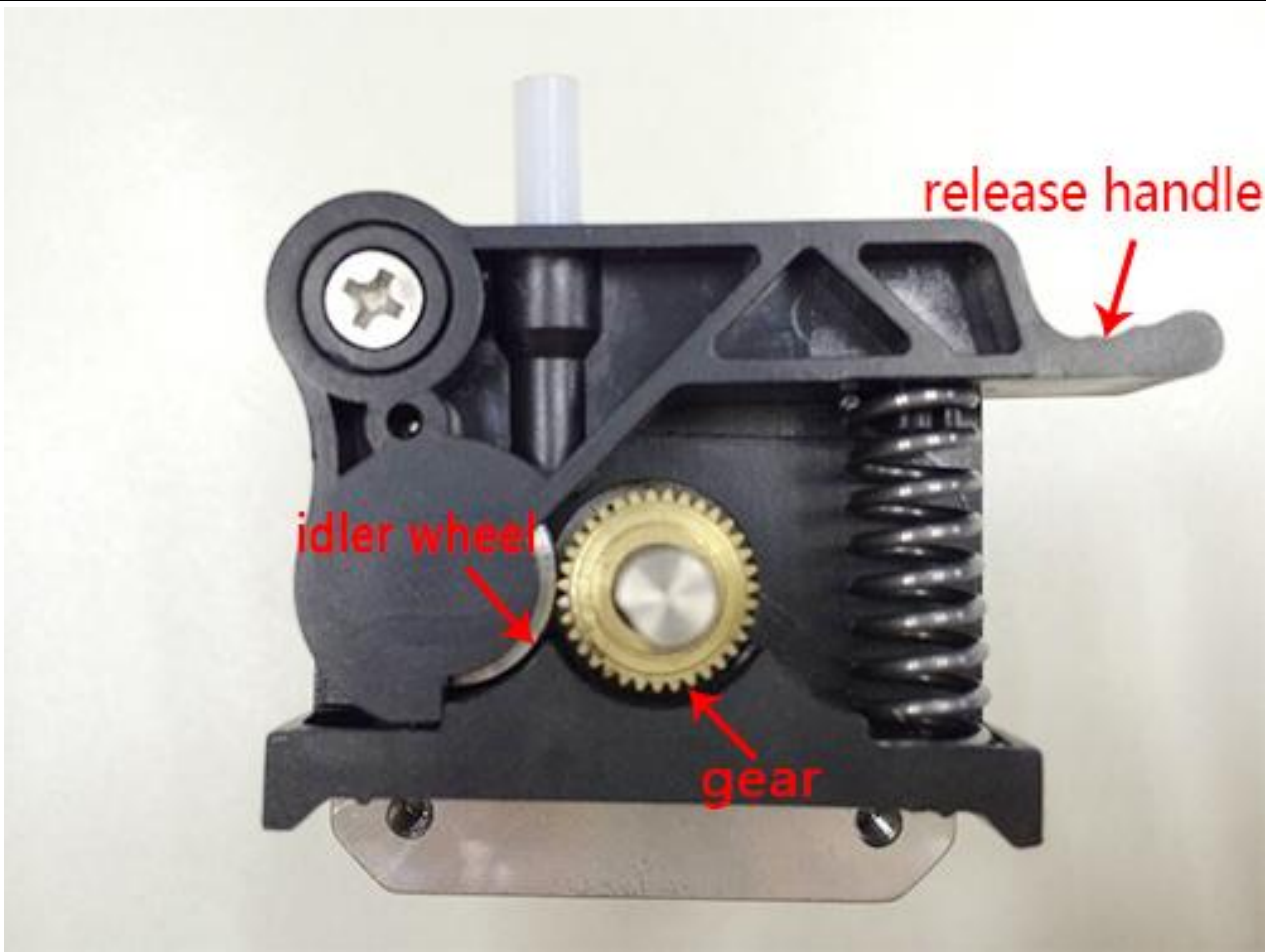


4.1.3 Extruder Test

Please note: The extruder will not begin working until the temperature reaches above 170 °C.

Heat the extruder to 200 °C, and then insert the filament into hotend via push-in fitting. When you are pushing the filament, press the handle with one hand and push the filament with the other hand.

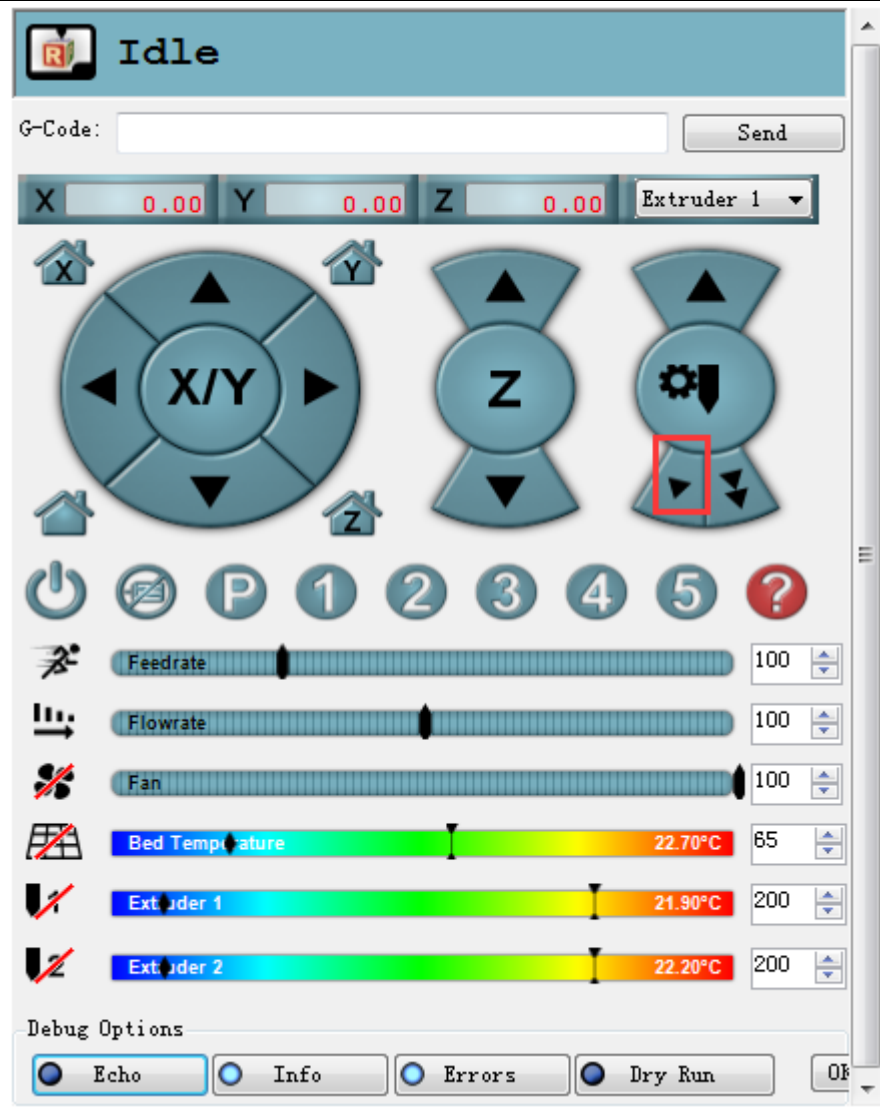
Please note that the filament must be in the slot of the idler wheel, otherwise feeding will not be fluent.





After pushing the filament into the bottom of hotend, use the feeding button on Repetier (as shown in following picture) to do manual feeding. If the nozzle can extrude filament fluently, the extruder is working well. If not, please refer to [FAQ](#).

Note: If the moving direction of the motor is reversed, which means you found the filament going backwards when you click to feed downwards, change the motor direction of the extruder in the firmware to solve the problem. This method is the same with that for X/Y/Z axis motor.



4.2 Use LCD Control Panel to Test

4.2.1 Motor Direction Test

Functions of the knob on LCD:

1. Press the knob: OK or enter the sub-menu
2. Turn the knob: to scroll through the options, or change parameters

Note: Turn the knob during printing, you can change the Feed rate in real time.



After learning about the functions of the knob on the LCD, press the knob to enter sub-menu and choose Prepare:



Choose Move axis:



Choose Move 1mm:

(Note: the extruder can only support two options: Move 1mm and Move 0.1mm. The 10mm option is disabled.)



Choose the axis that you want to move by using Move X/Y/Z options. Turning the knob can make each axis move left and right or back and forth.



After testing each axis, you can choose Prepare>Auto home in the menu to make it back to home position.



If you want to manually move the motor, you can unlock the motor. Unlocking method:

Prepare>Disable steppers:



4.2.2 Heating Test

In the Control menu, you can select temperature setting of hotbed and extruder.

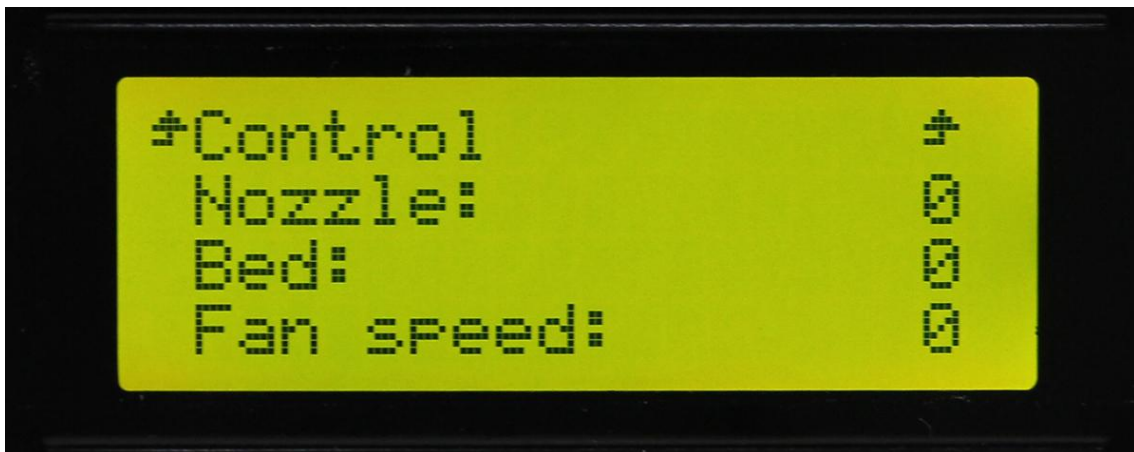
Choose control:



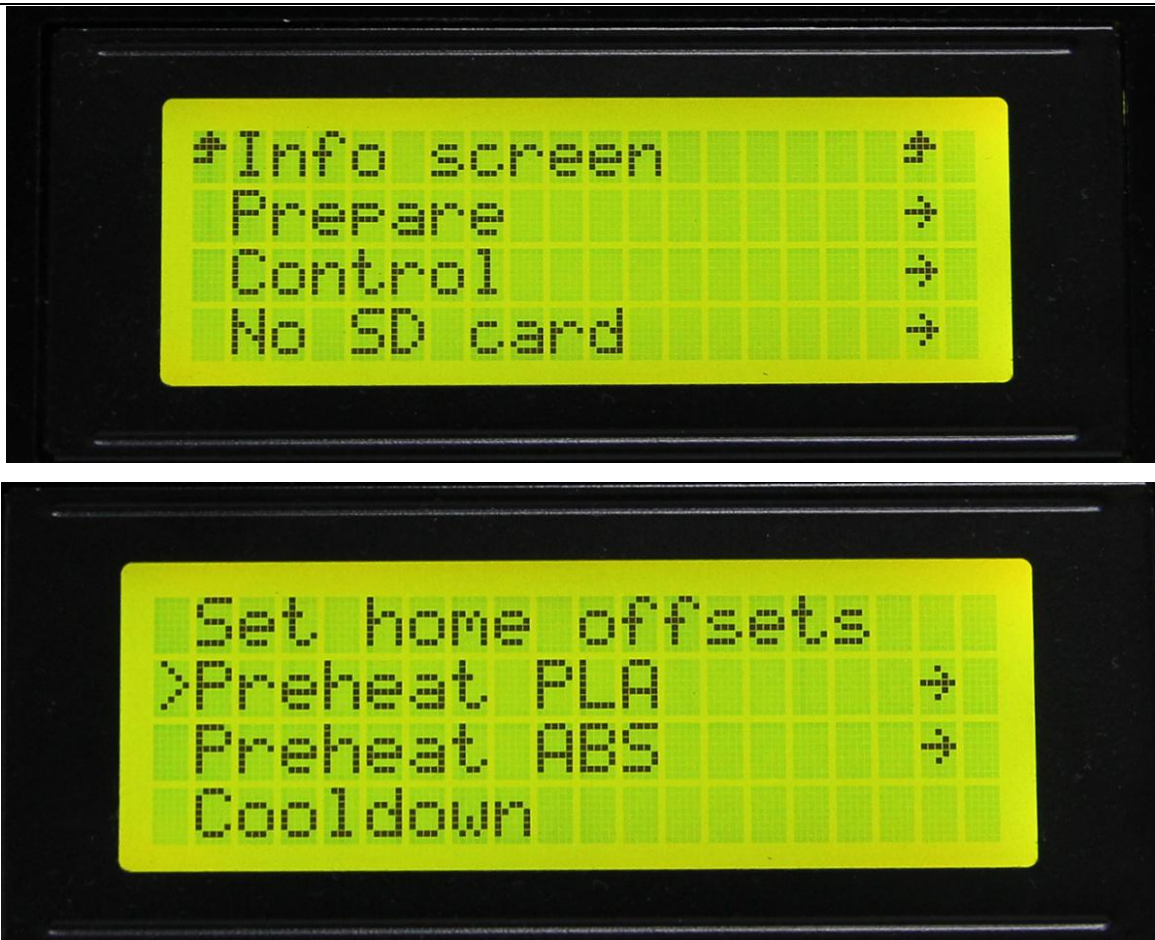
Choose temperature:



Select the temperature you want to set:



After setting, you can choose Prepare> Preheat PLA to preheat hot bed and extruder.



So far, functional tests of the printer are done.

5. Hotbed Leveling

Before printing, we need to do hotbed leveling. Only after this step can quality of printed parts get improved. Otherwise the un-leveling gap between the hotbed and extrusion head will affect the filament adhering to the hotbed thus cause poor print quality or even not printing.

5.1 Homing

Home the printer.

5.2 Coarse Tuning of distance between the nozzle and the hotbed

For dual-extruder printer, you can do leveling by taking the distance between hotend 1 and hotbed as a reference, and then do leveling for two extruders in subsequent steps.

*If the nozzle is far from the surface of hotbed, loosen the M3x50mm screw to extend the bottom part.

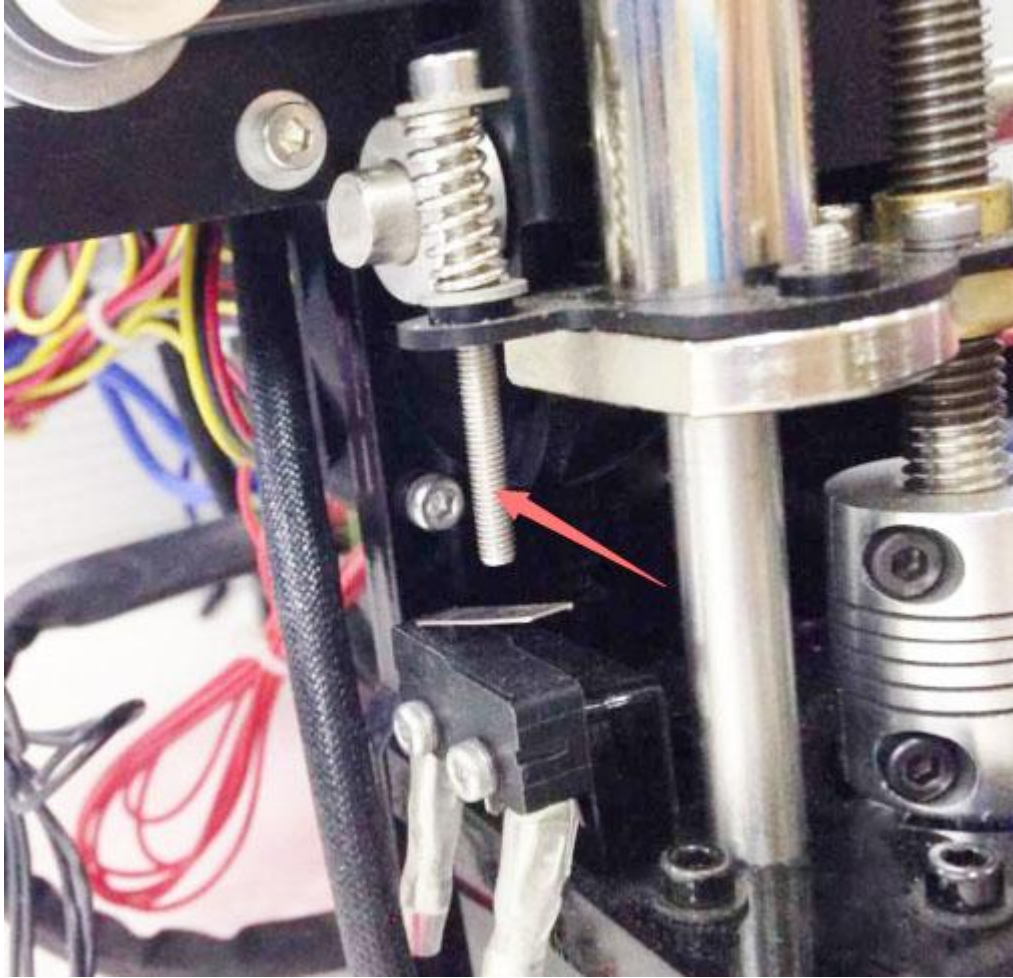
Then the printer will home at a relatively lower place, the distance between the nozzle and the hotbed will be decreased.

*If the nozzle is too close to the surface of hotbed, tighten the M3x50mm screw to shorten the bottom part.


Then the printer will home at a relatively higher place, the distance between the nozzle and the hotbed will be increased.

You may need to calibrate for many times at this step.

The recommended distance between the nozzle and the hotbed is 0.5-1mm.



5.3 Fine Tuning

After the initial adjustment, home the printer. Disable the stepper motors(), manually move the extruder to the four corners of the hot bed and fine-tuning the wing nut at each corner to level the distances between the extrusion head and four corners. During adjustment, you can put a piece of A4 paper(half fold it) between the nozzle and hotbed. If you can feel a bit friction when you pull the paper back and forth, the distance is proper.

6. Dual-extruder leveling

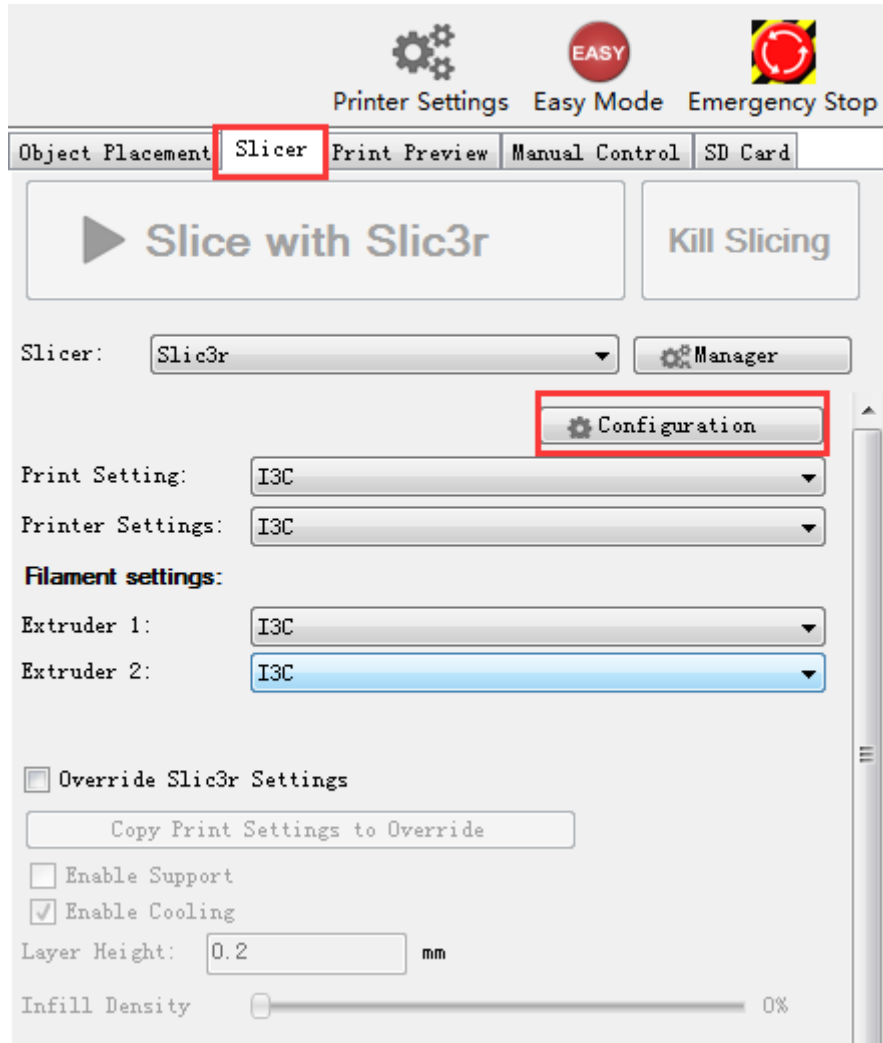
I3 pro C uses MK8 dual-extruder, so you only need to adjust the two screws when leveling the extruder which are marked in following picture. You can take hotend 0 as a reference when you are doing the adjustment. Loosen the nut clockwise to increase or decrease the height of hotend 1.



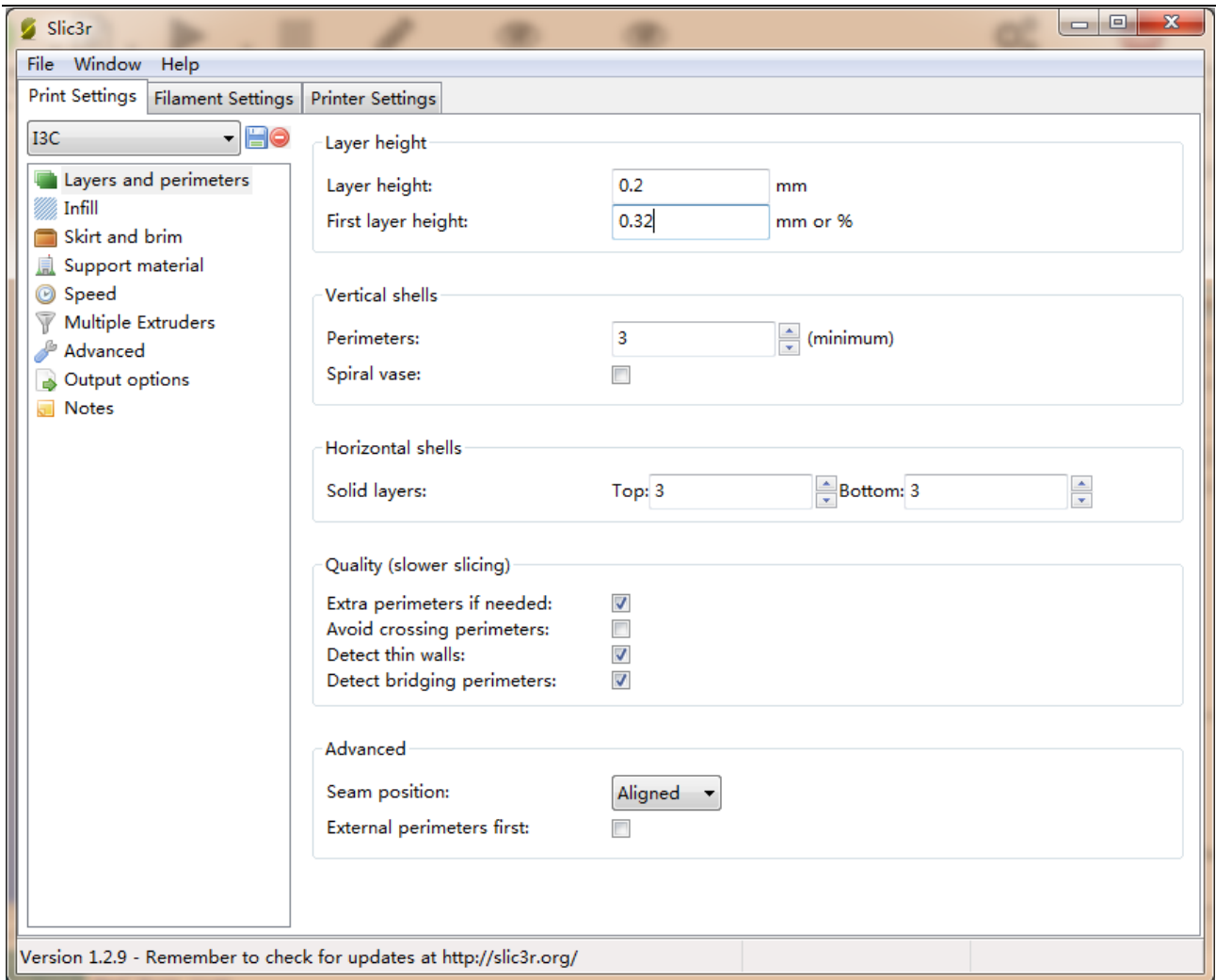
7. Slice Settings

With one more step, we can start printing. Set the slice parameters such as print material diameter, the speed, the layer height, etc. to achieve better print quality. Open slic3r, which is a slice software

coming with Repetier host. As shown below:

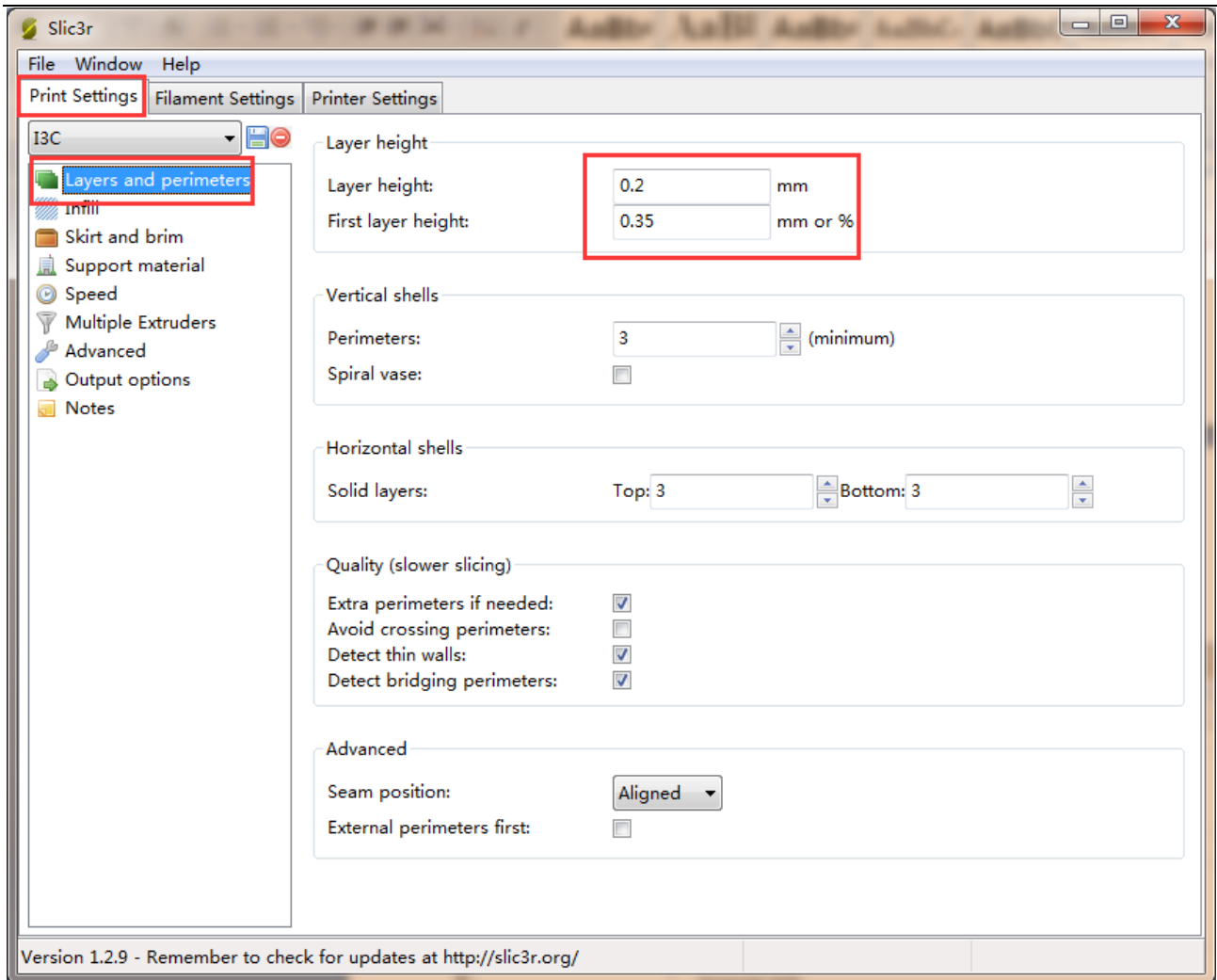


You will be presented with the following interface:



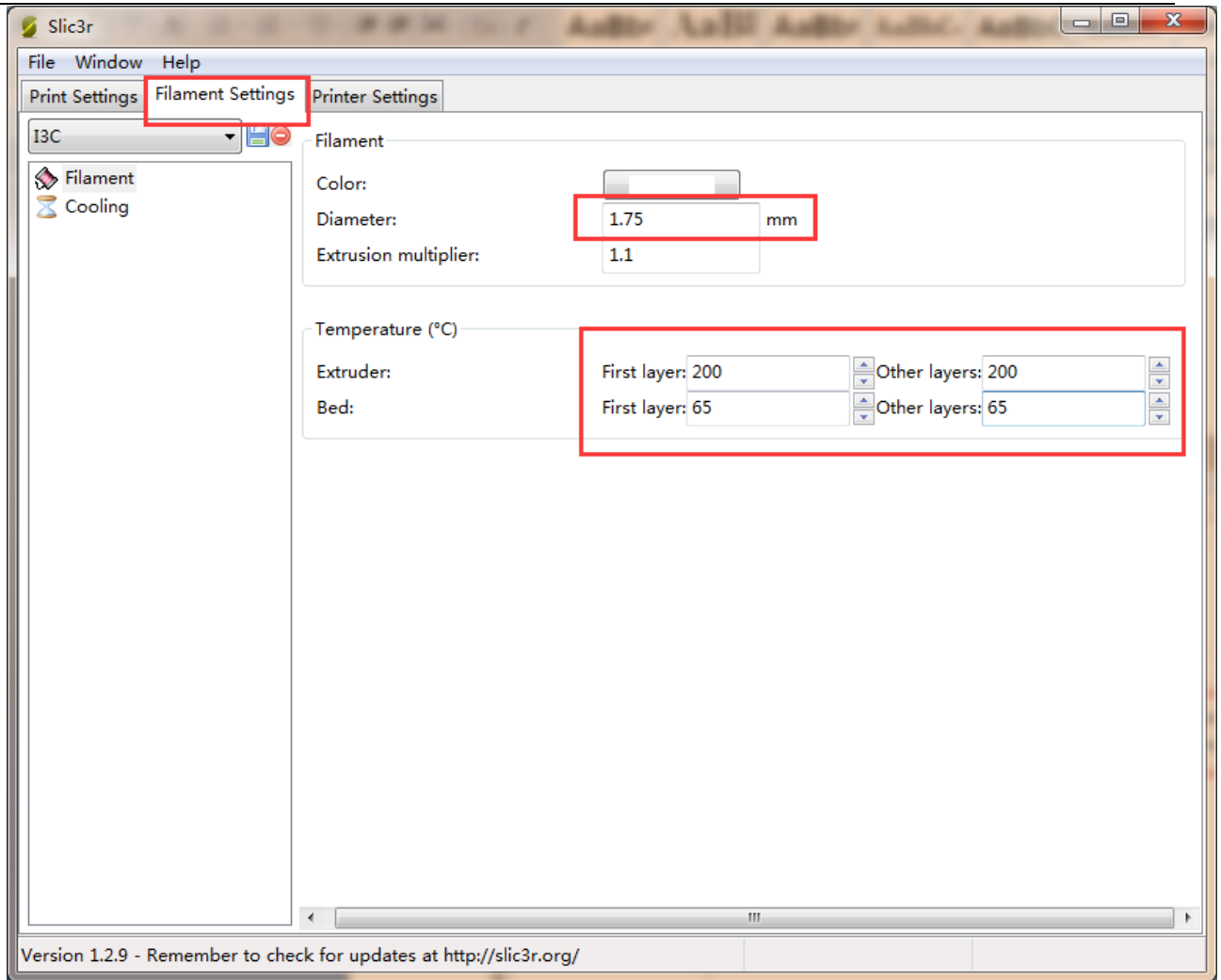
7.1 Printing Setting

Set layer height and the first layer height in the Print setting tab. Generally speaking, layer height can be set as 0.1-0.3mm. Considering of accuracy and speed, 0.2mm is the most appropriate layer height. The layer height is set as 0.35mm by default.



7.2 Filament Setting

Set filament diameter and print temperature in the Filament setting tab. Filament we use here is 1.75mm PLA. Usually the temperature of the extrusion head is 195-210°C, and the temperature of hotbed is 60-70°C. Here we separately set them as 200°C and 65°C.



7.3 Printer Setting

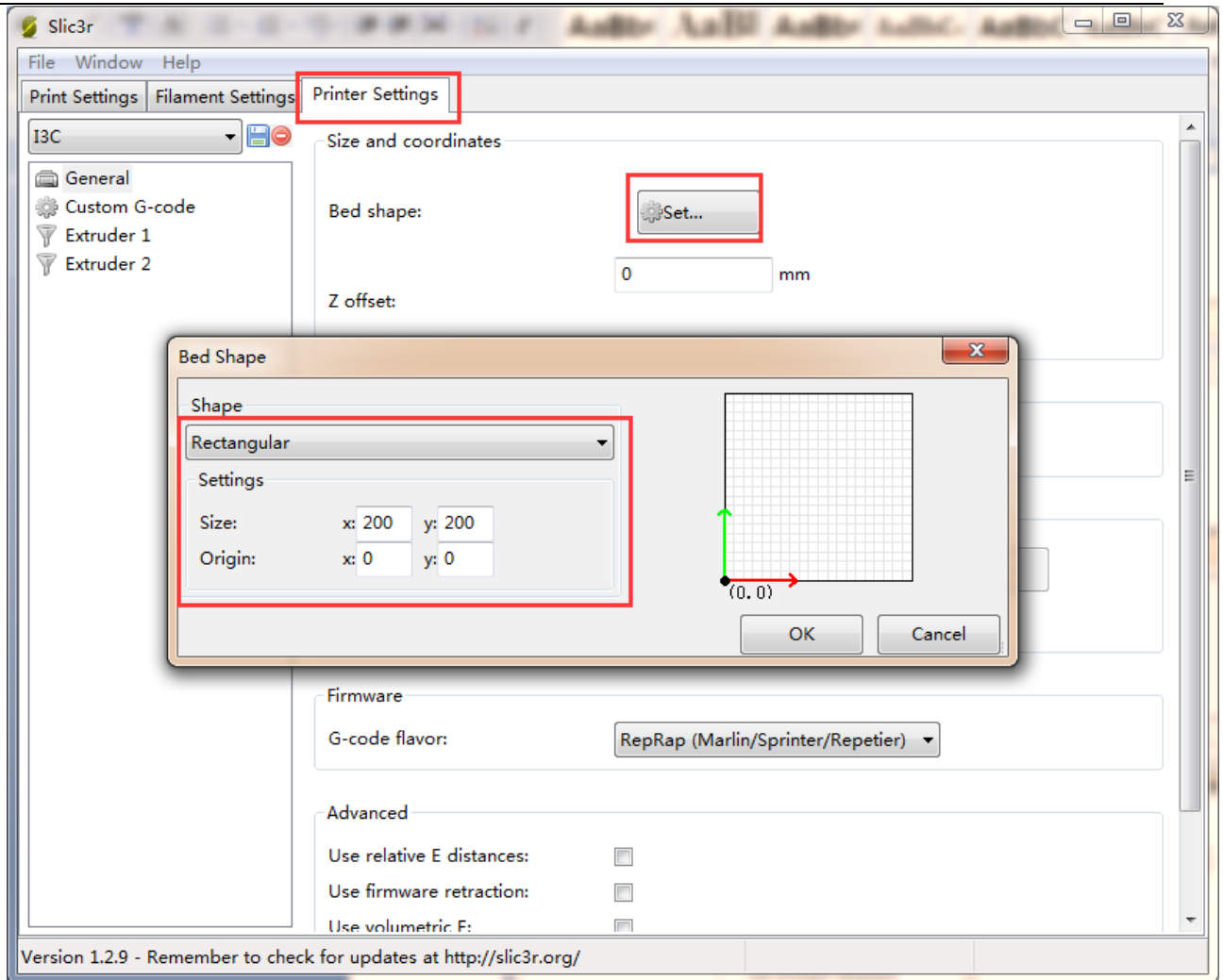
Set printer's hardware parameters in the Printer setting> General tab.

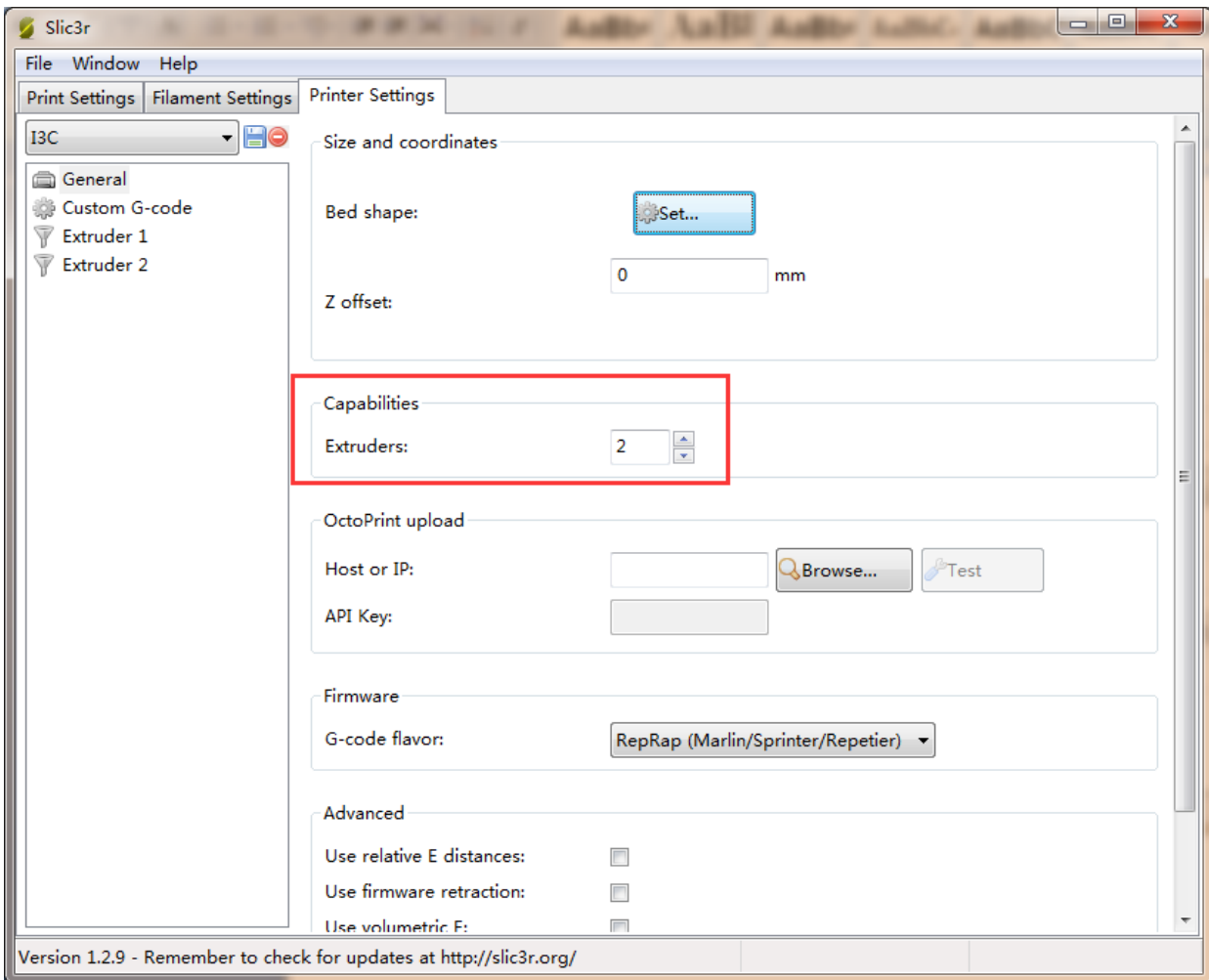
Shape of hotbed: Rectangular

Size: 200*200

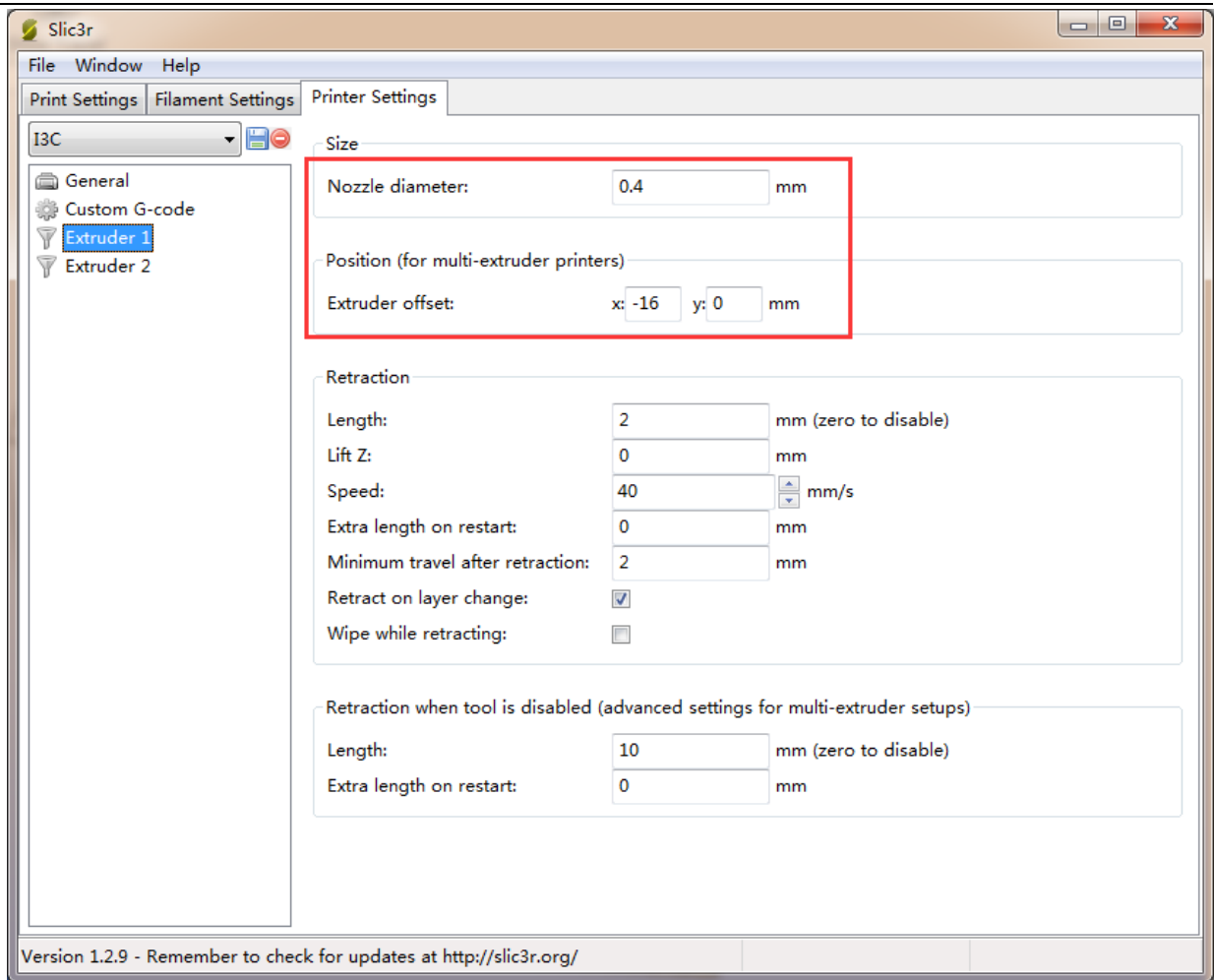
Number of extruders: 2

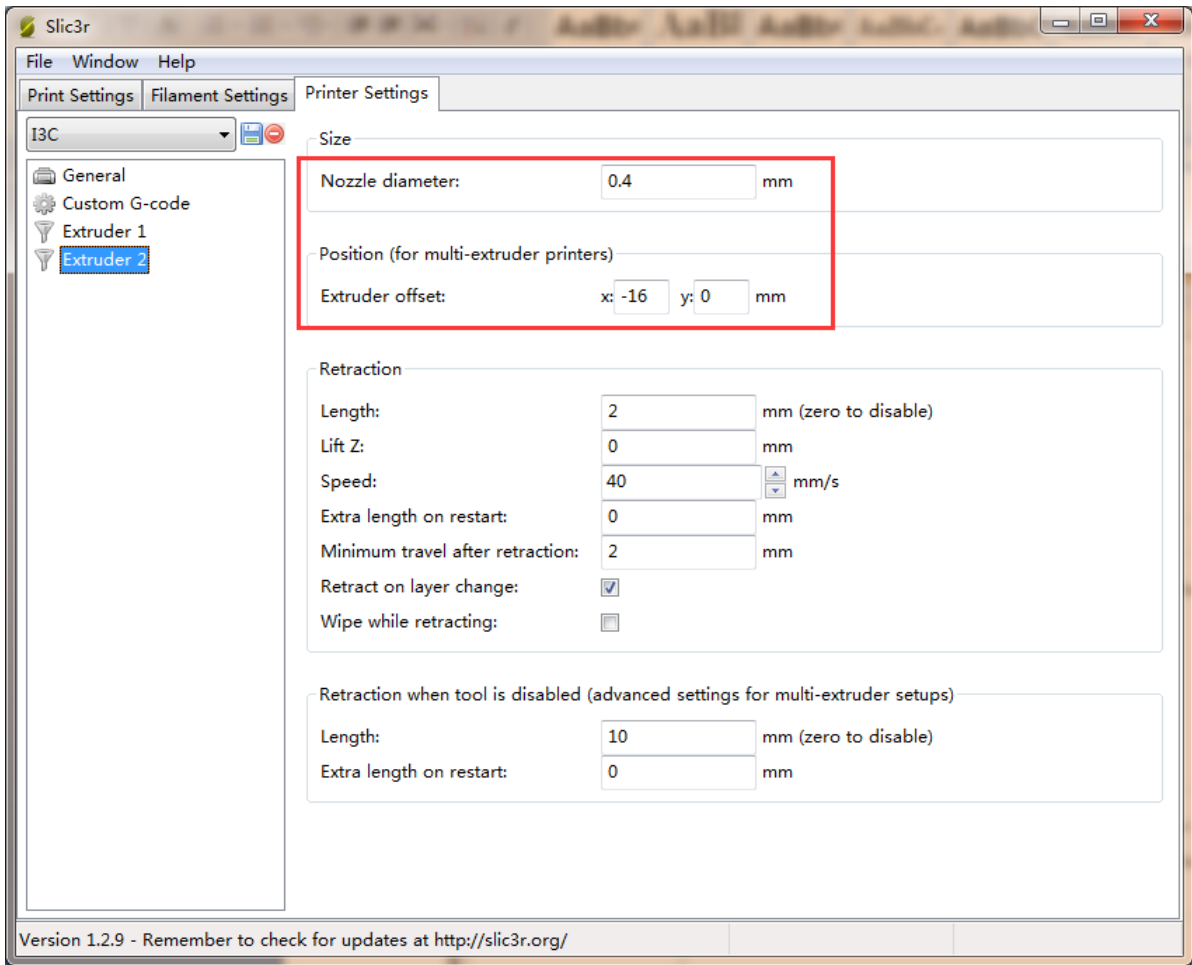
Offset on X direction: Extruder 1: -16mm, Extruder 2: 16mm





Meanwhile set the extrusion head diameter as 0.4mm (input the actual diameter of your own printer):

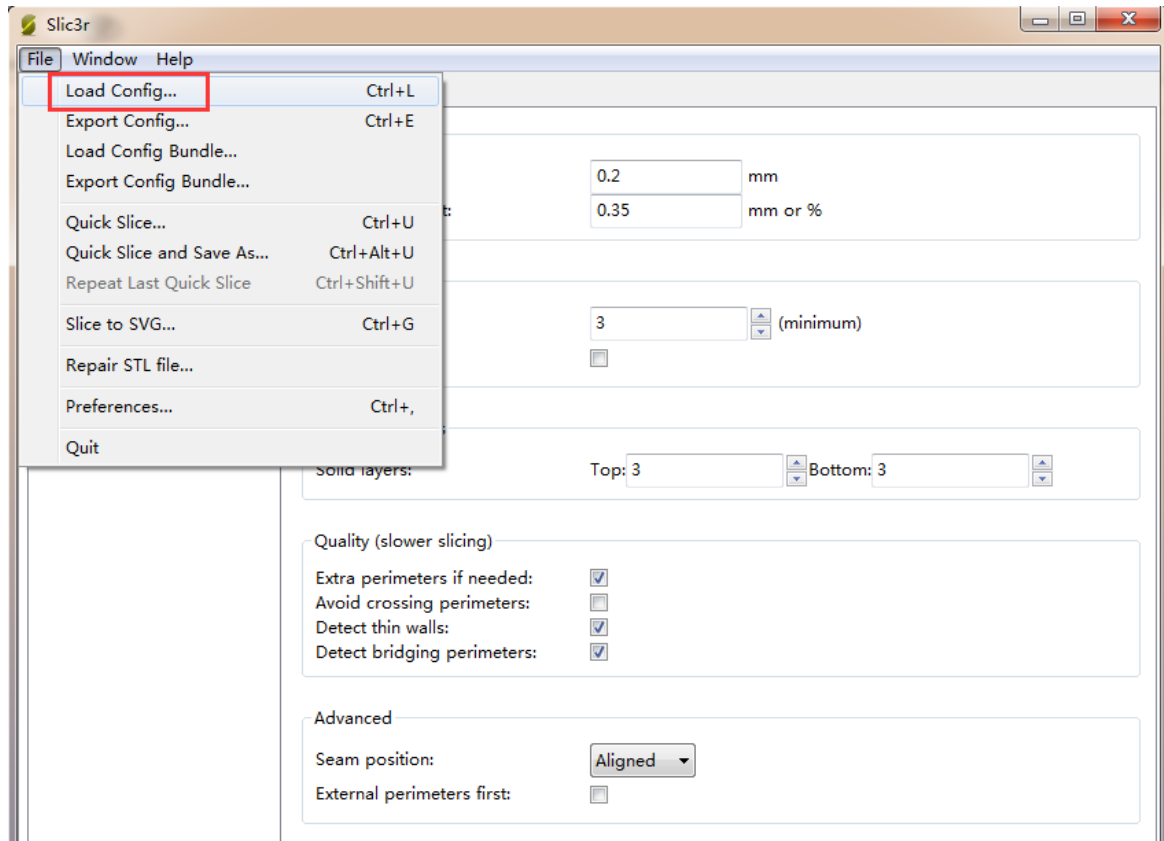




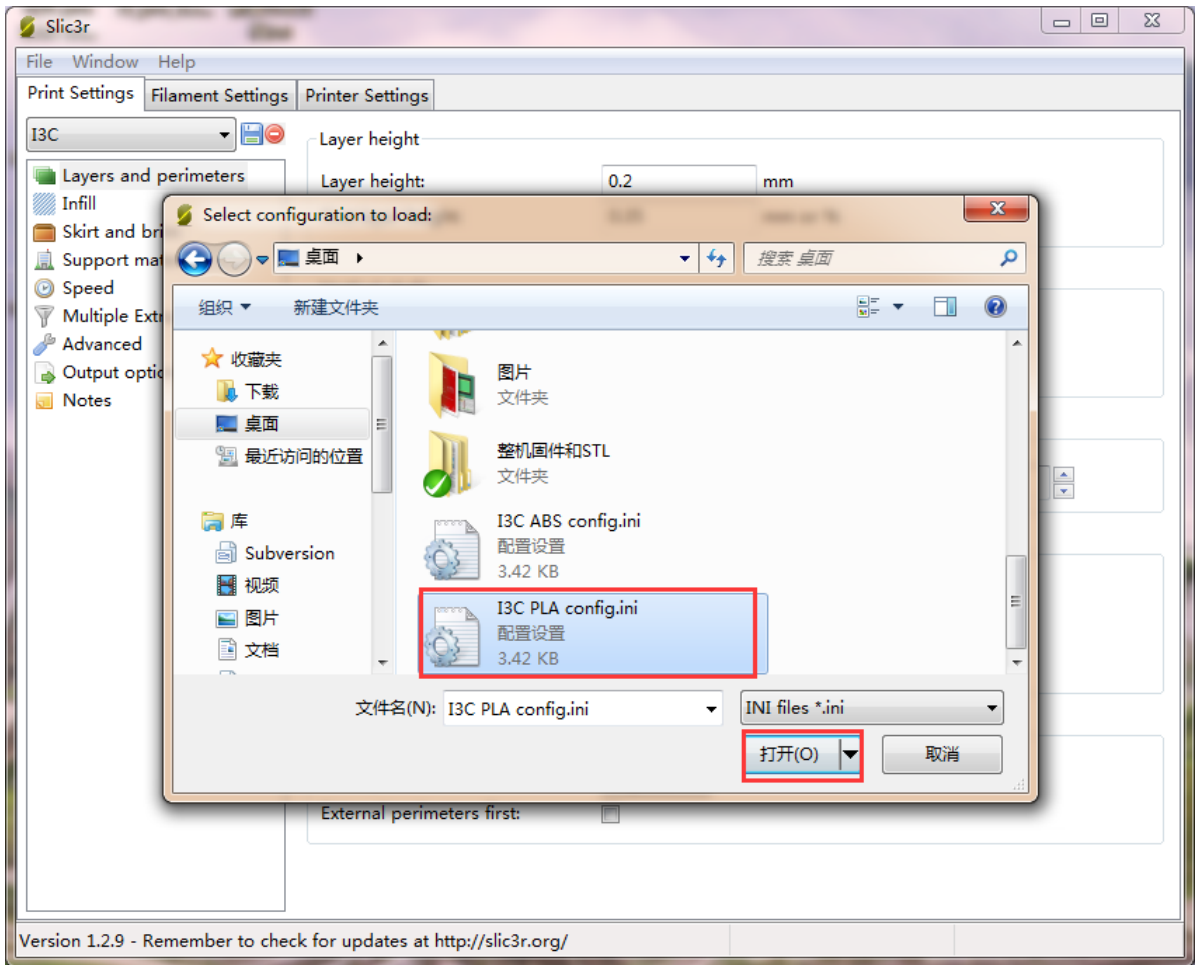
7.4 Other Settings

In addition to the above parameter settings, parameters like the speed of the printer are also very important to print quality. It requires user's long-term using experience and exploration. Here we'd like to give you a setting for reference, click to download the [config.ini](#). You can import it into slic3r to view it through the following steps.

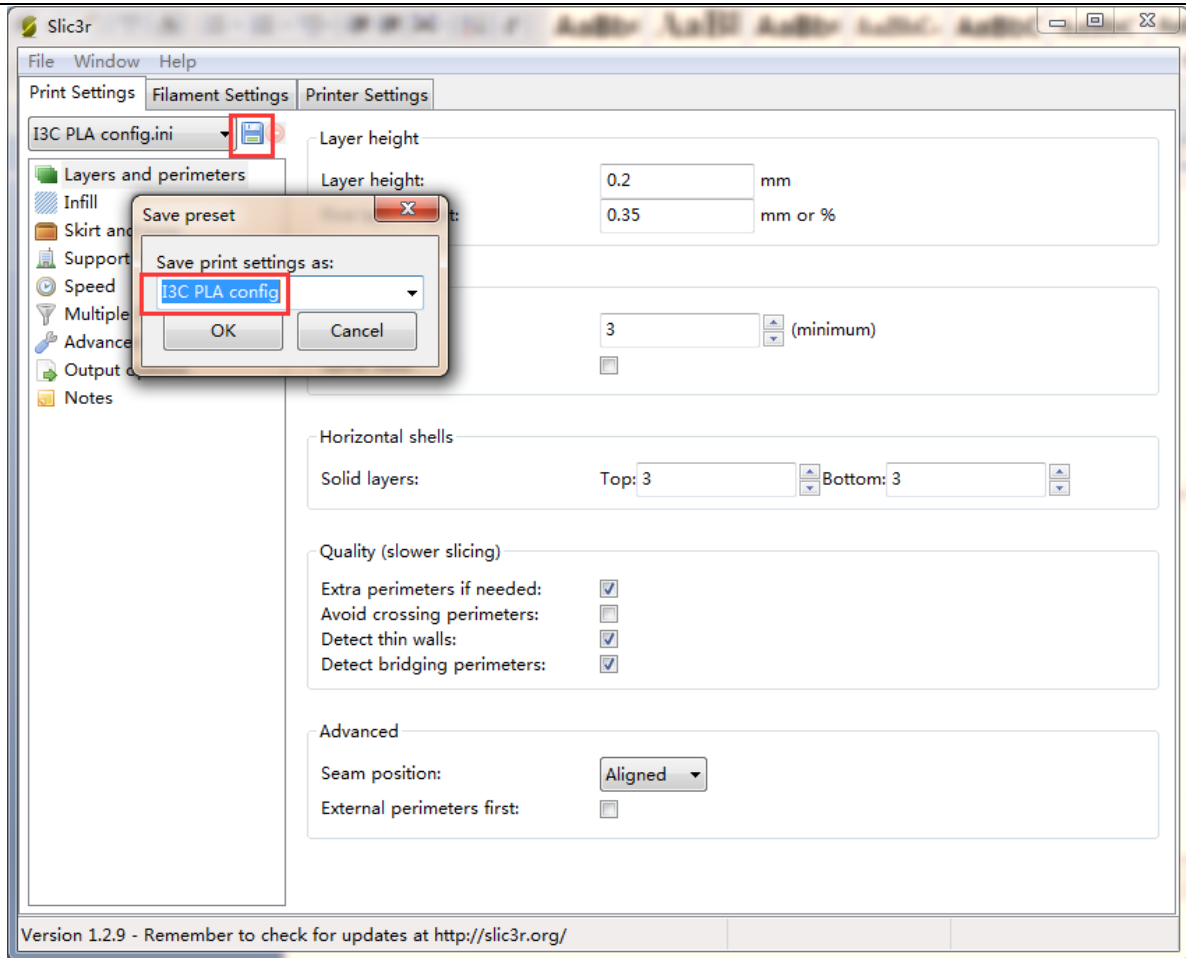
Open slic3r>File>Load Config:



Find "I3C PLA config.ini" file in the dialog box and open it.



Then you can successfully import the config.ini file. Click the Save button to rename and save it.



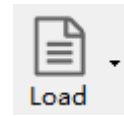
8. Begin Printing

So far, all the preparatory work is completed. The next step is to import the model file for slicing and printing. For 3D printer, the file format of the model is generally .stl file. We can download for free and print those models which are shared online , and you can also design your own creative objects for printing.

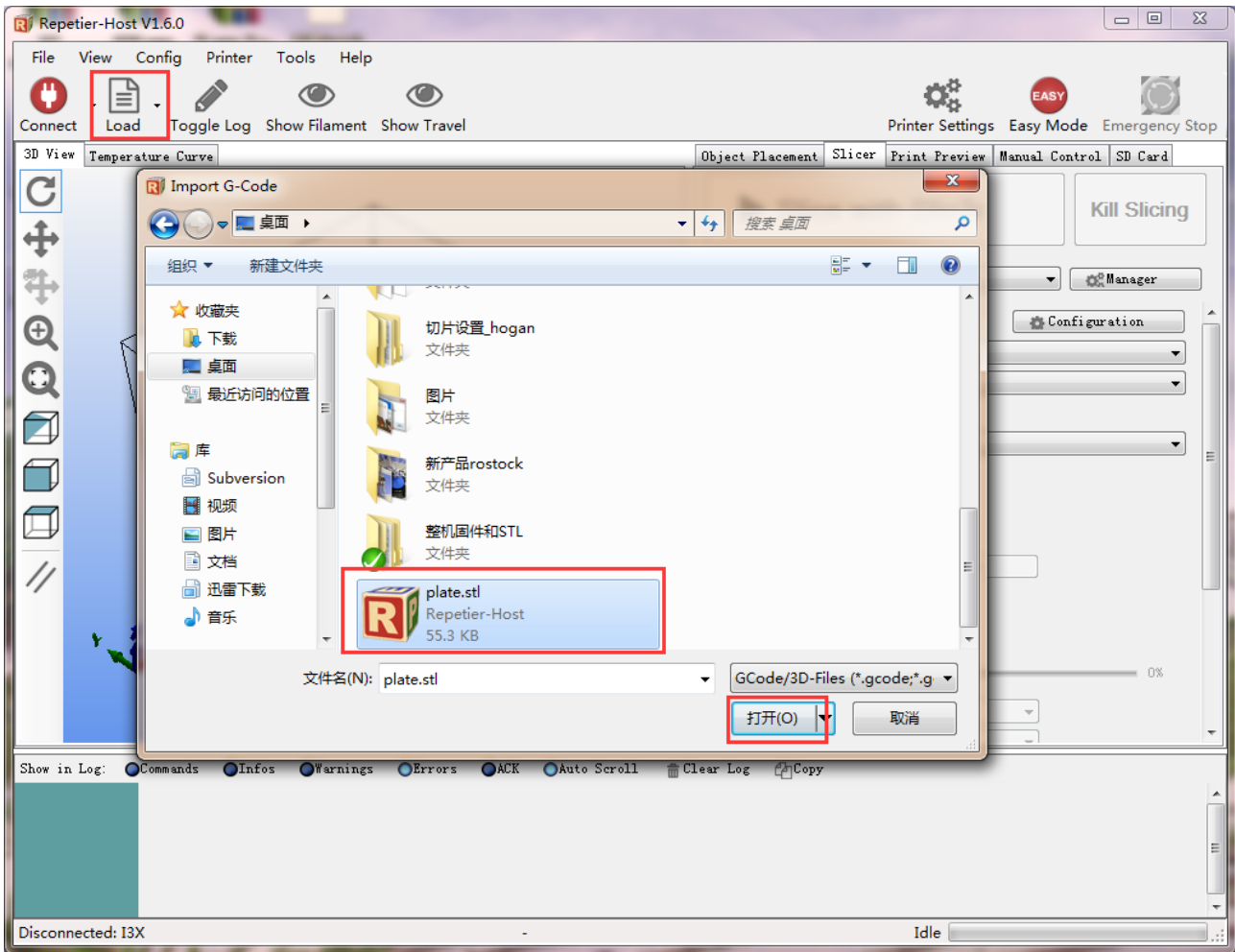
8.1 print with one extruder

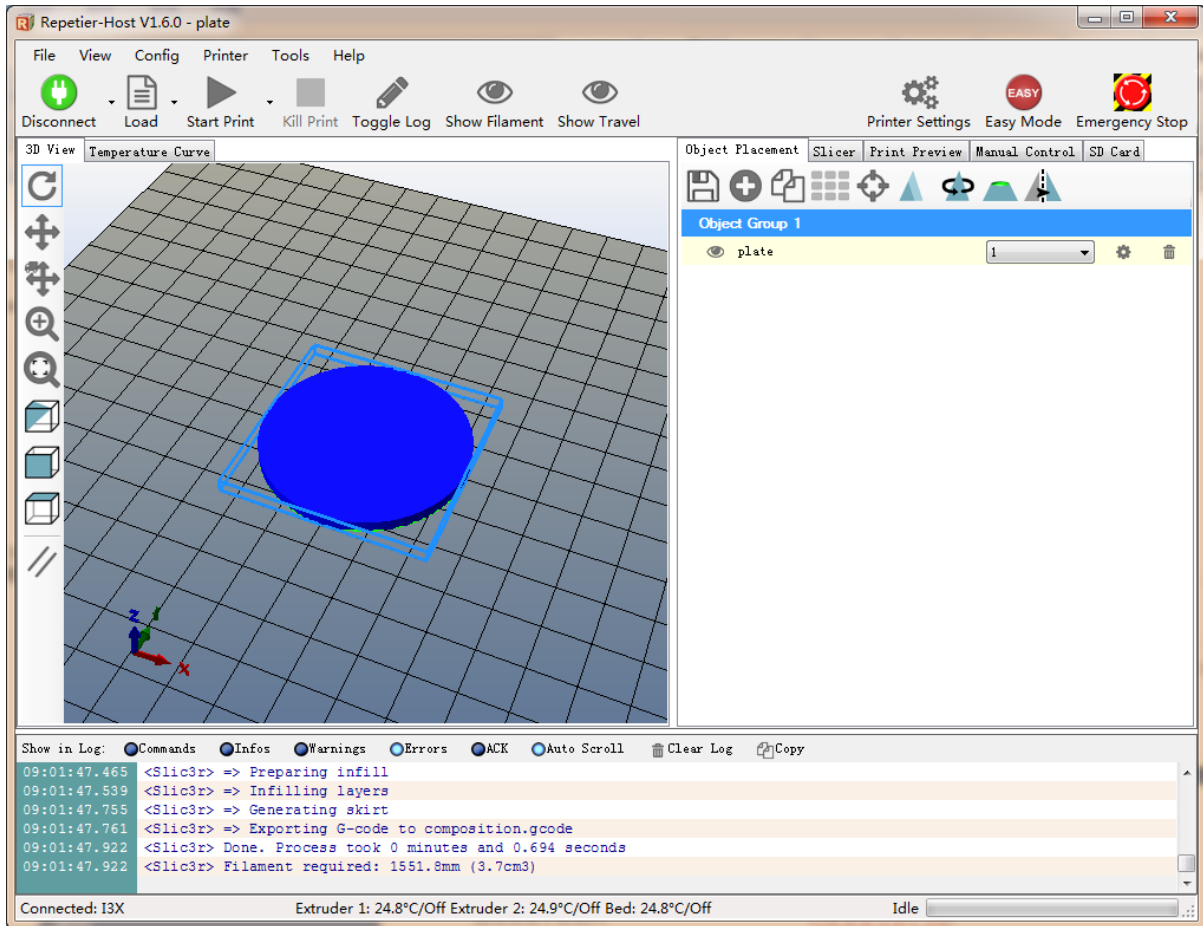
If you are new to 3D printing, you can try to print with only one extruder first. After you got familiar with the whole process you can try to use dual-extruder. Here we print a simple plate. You can download the model file here: [plate.stl](#).

8.1.1 Load Print Model



In the main interface of Repetier host, click the Load button . Choose the file you download and open it.

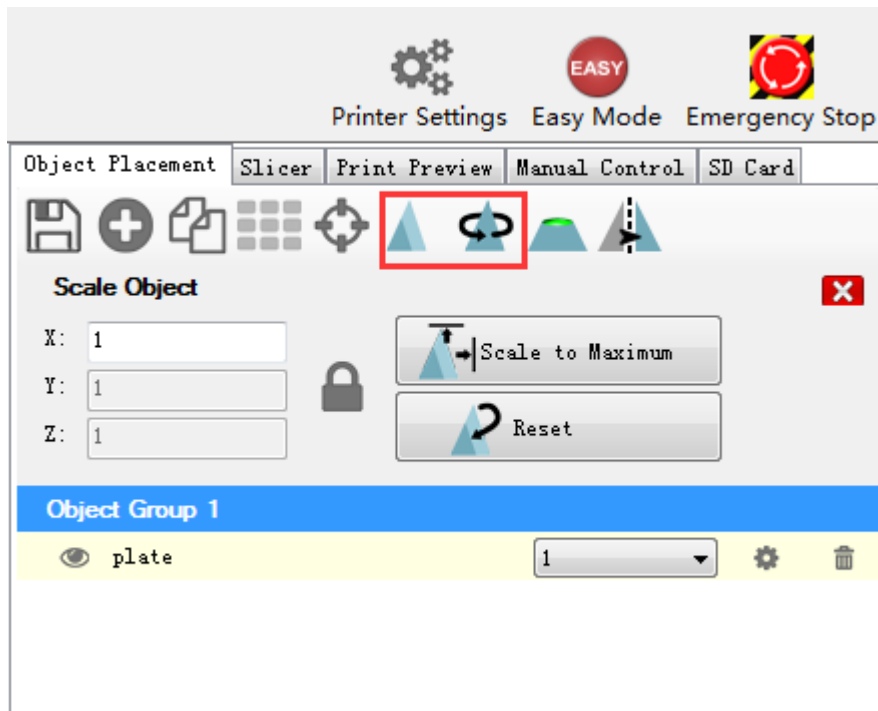




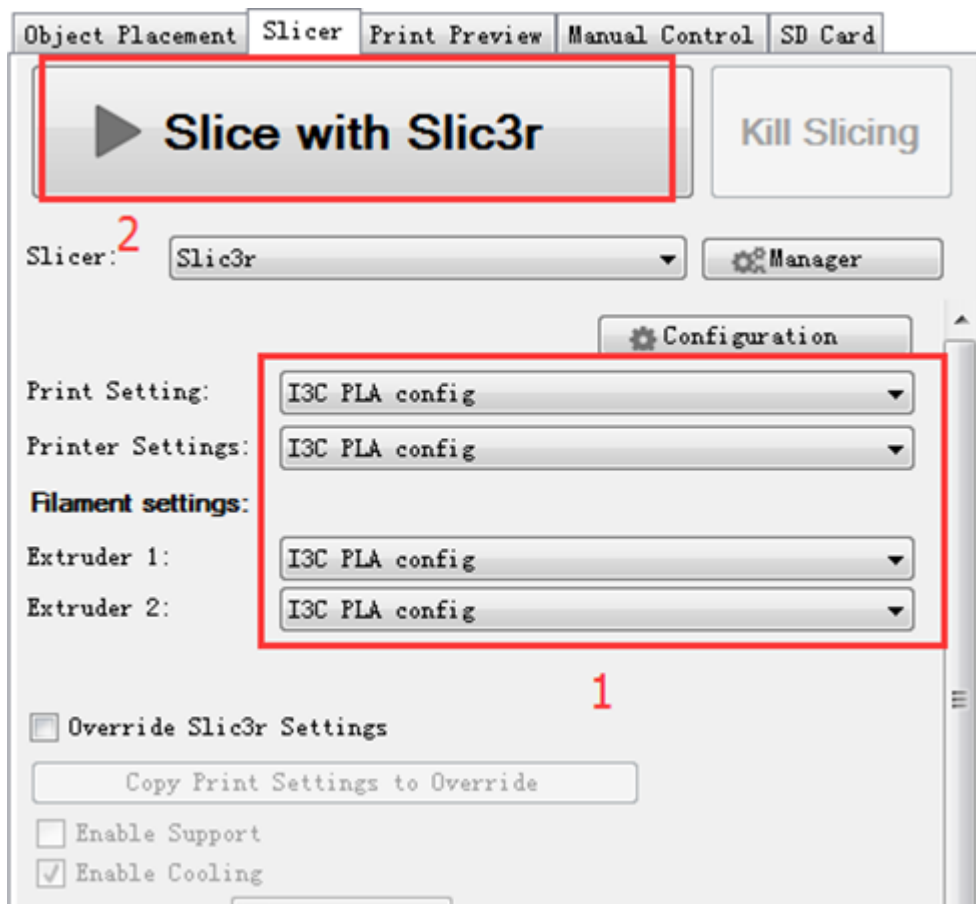
8.1.2 Model Slicing

After the file is loaded, you can use the following buttons to enlarge, shrink or rotate it.

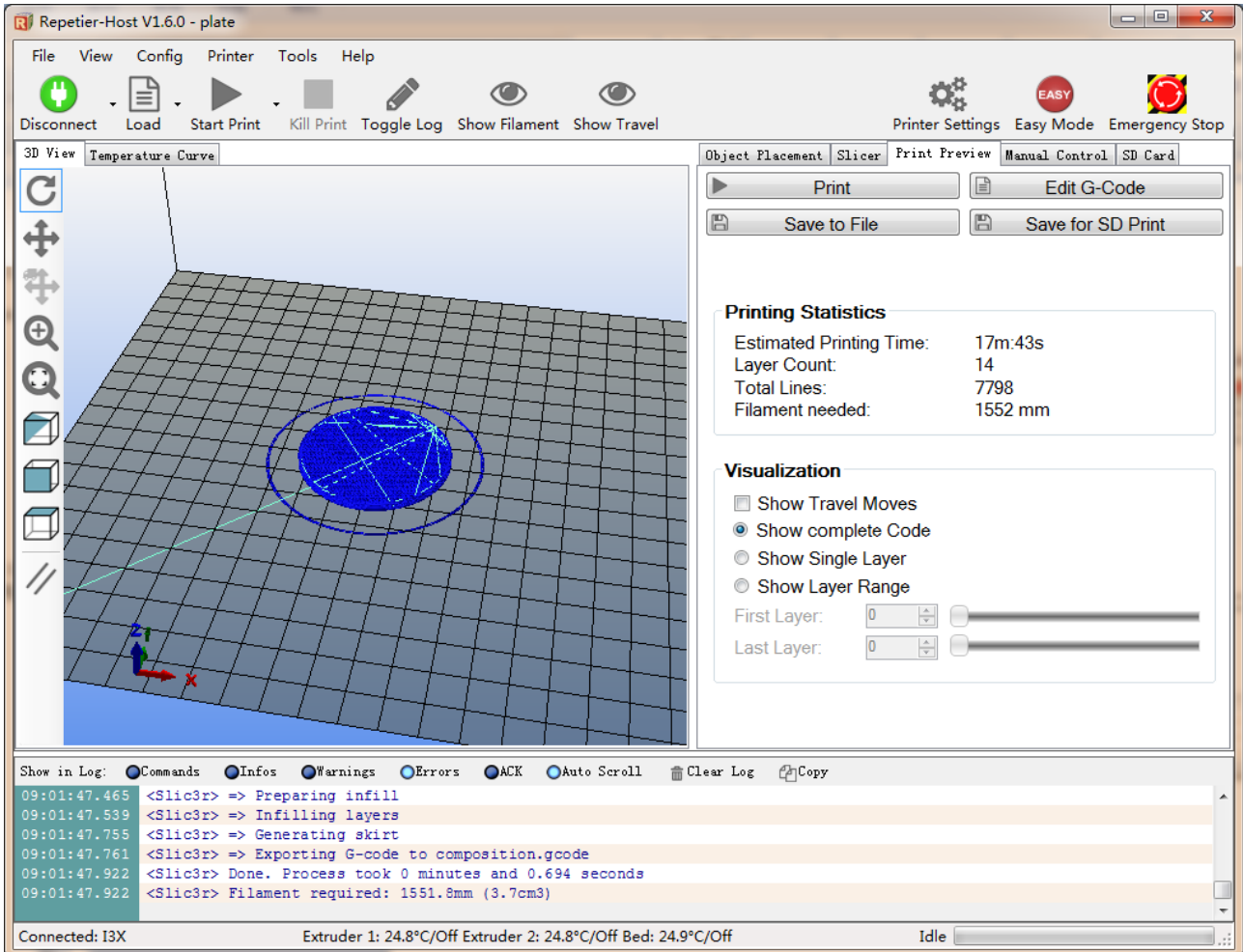
choose the extruder.

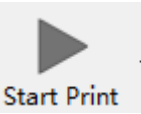


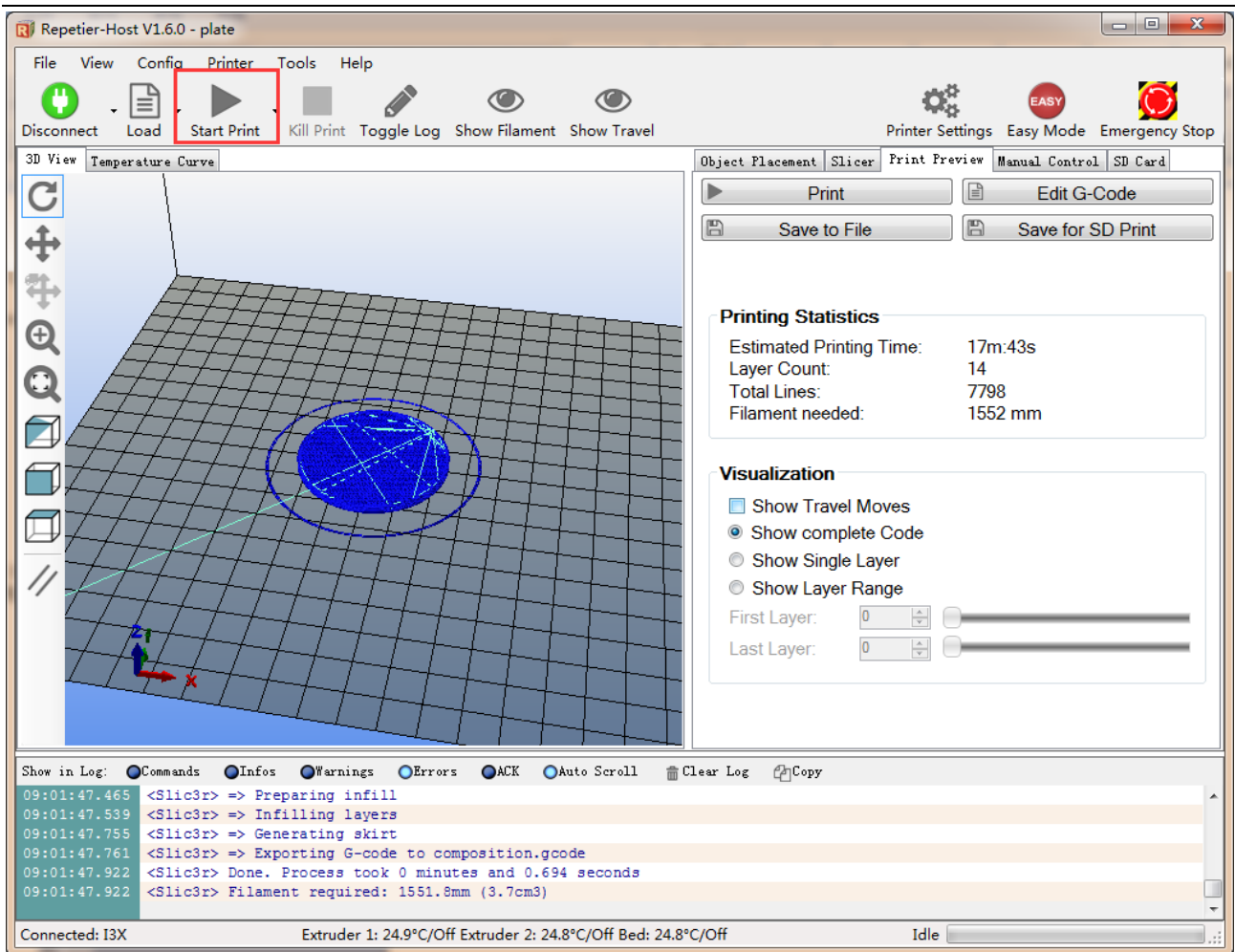
After the size setting, choose the slicing parameters which were imported previously in the drop-down menu, and click **slice with slic3r**.



In this way, .gcode file which can be recognized by the printer is generated.



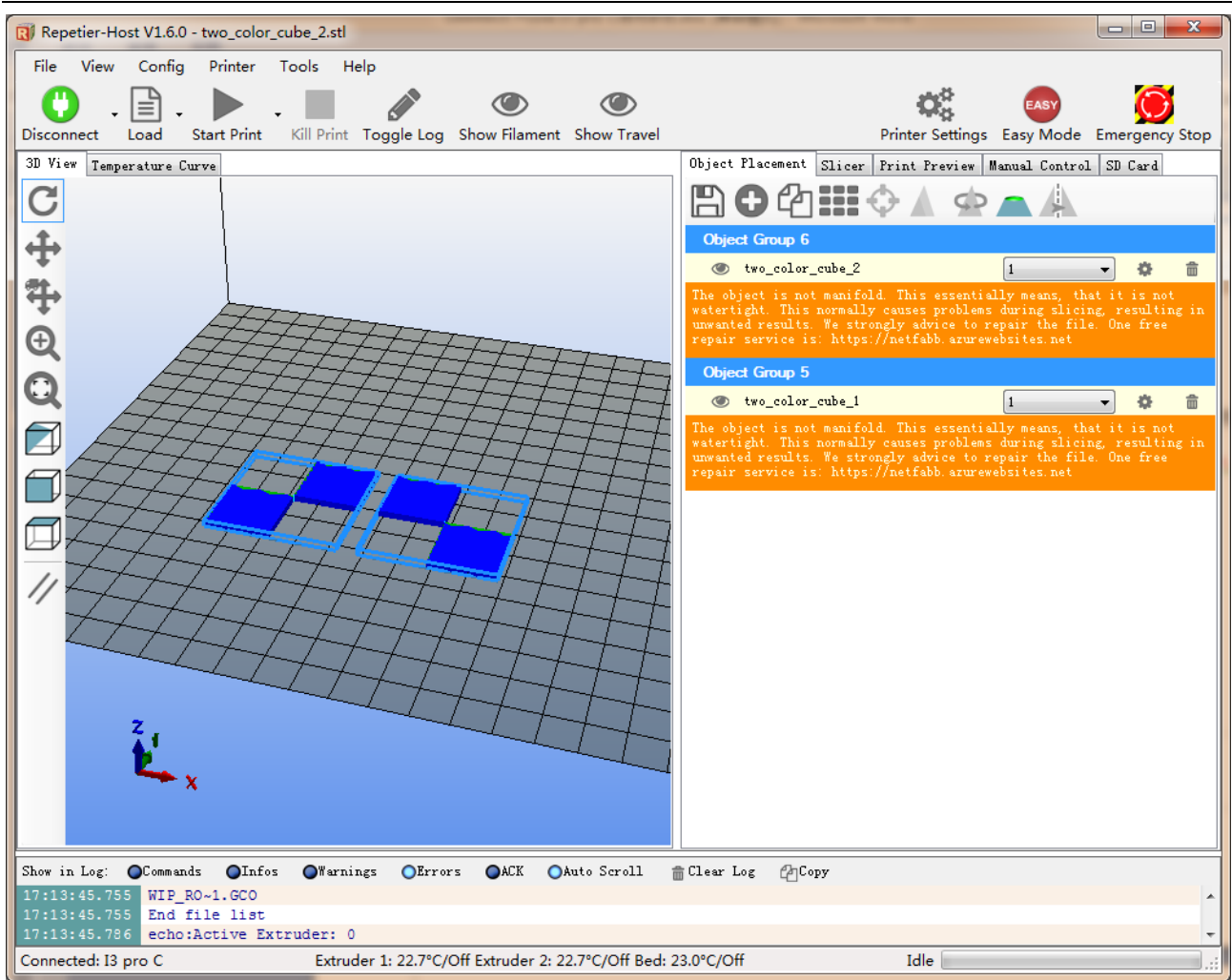
Last step, click  button to print.




8.2 Print with Dual-extruder

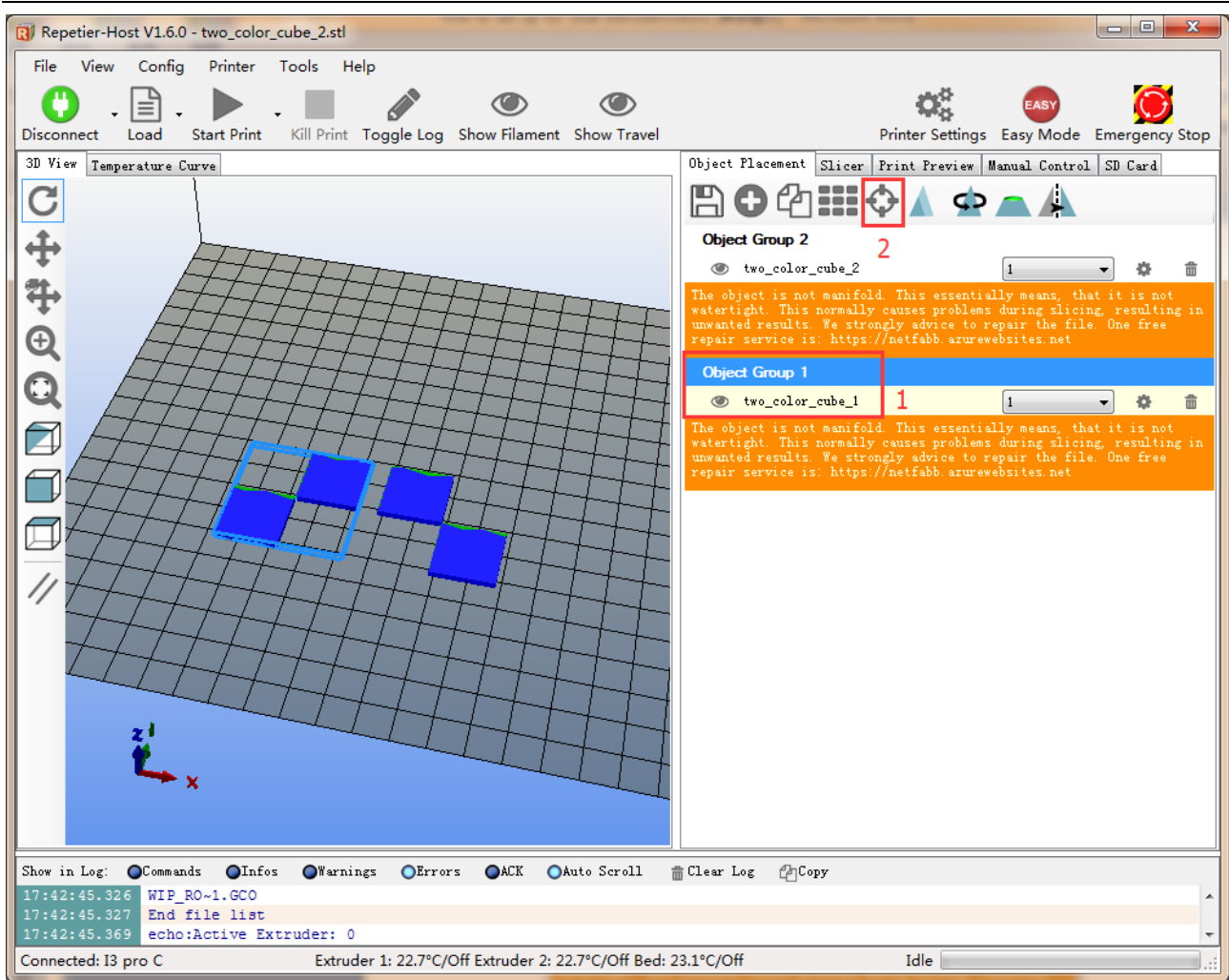
Actually dual-extruder print is to load two separate objects, and then separately print them with the assigned extruder. Thus you can print objects with two colors. Click [here](#) to download stl files which are used for dual-extruder printing: [two colour cube.stl](#).

After downloading, load the model, which is shown as below:

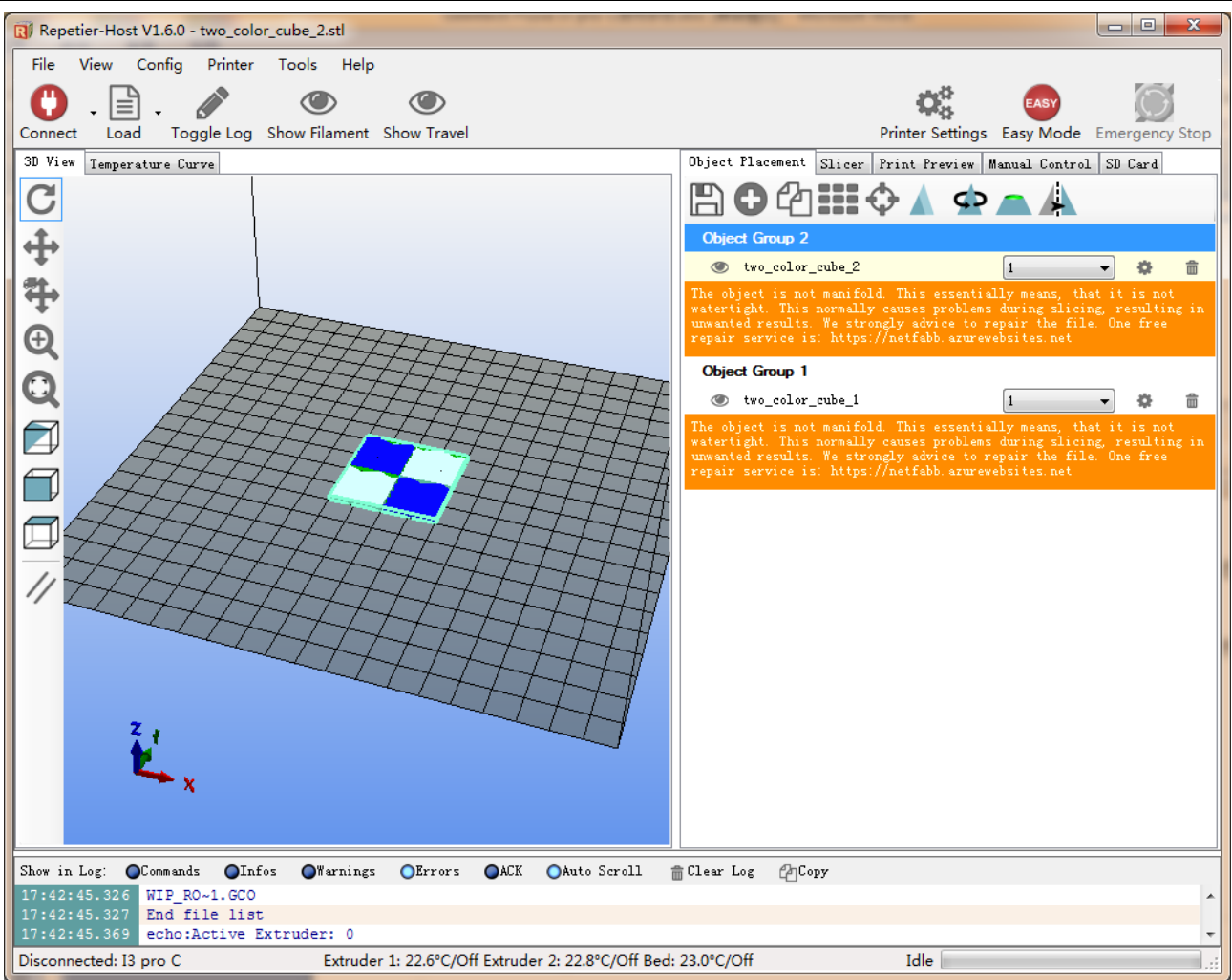


The two models loaded are separate. First we adjust their locations to combine them together. Click

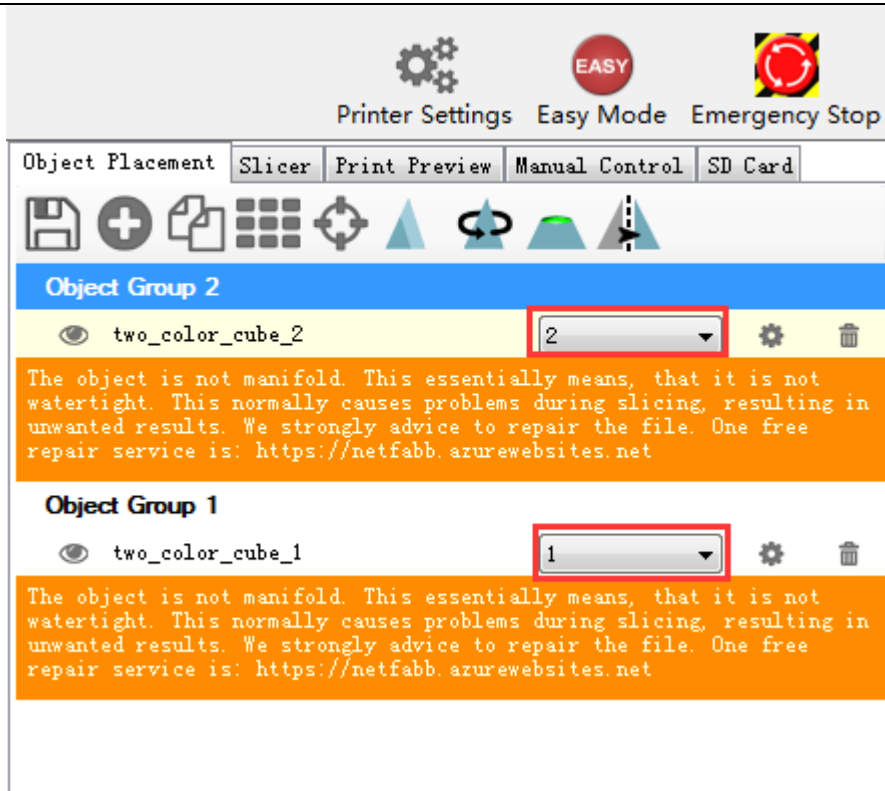
Object Group 1, choose object 1, and then click centering button .



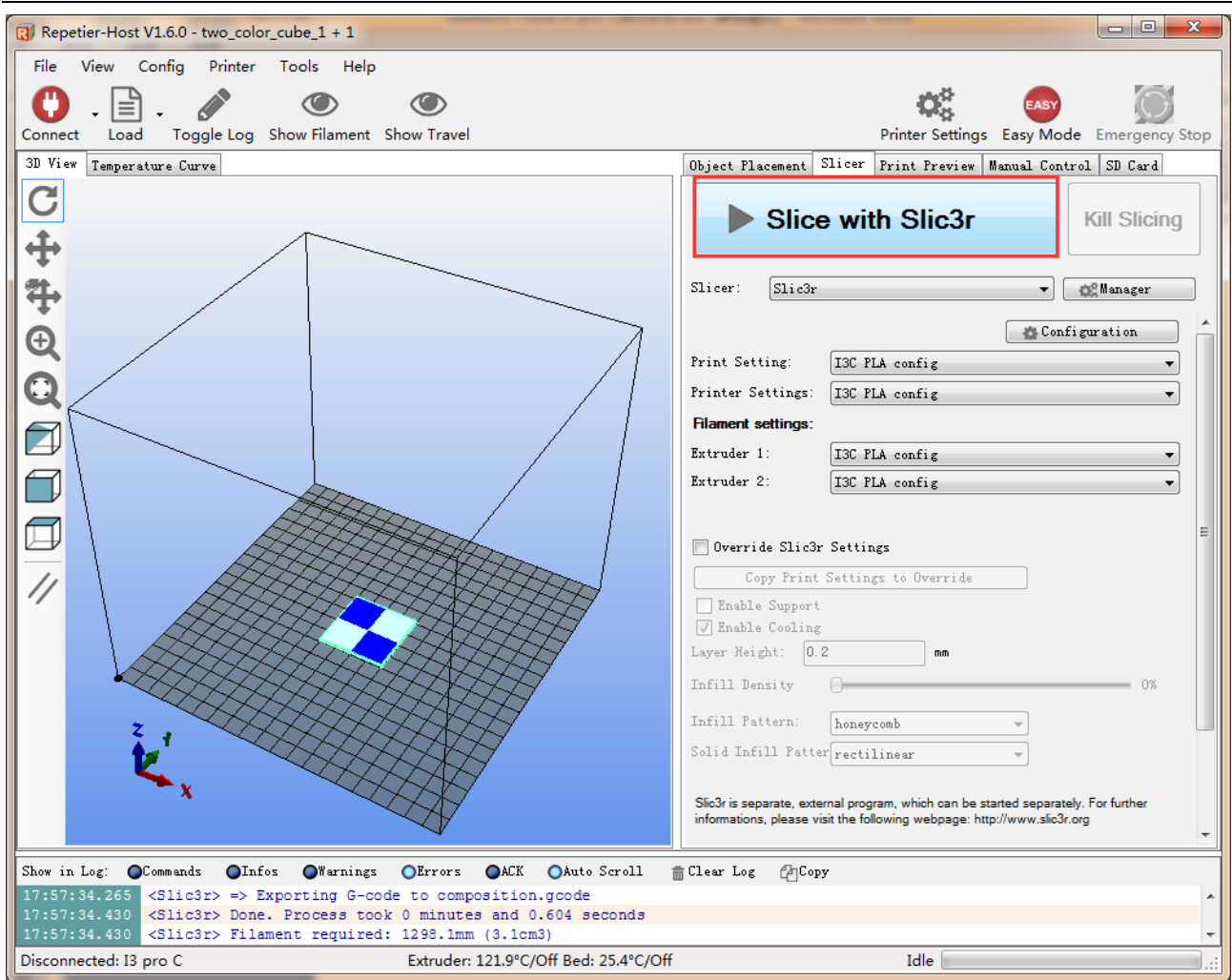
The object is now in the center of the hotbed. Put object 2 in the center of hotbed in the same way. After the setting, what you will see is shown as below:



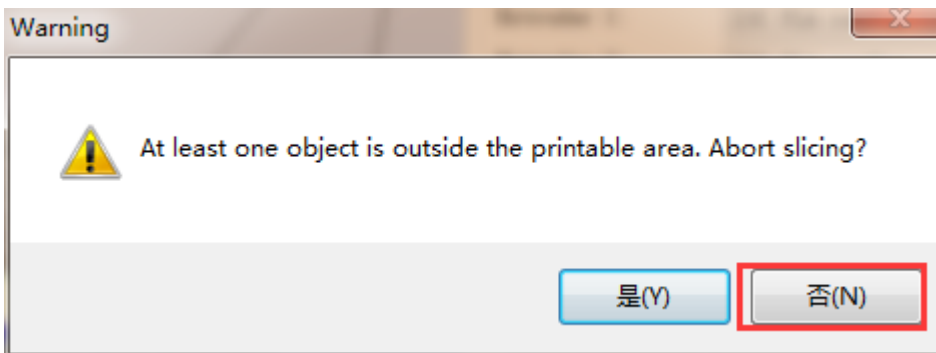
Then assign extruder for each object:

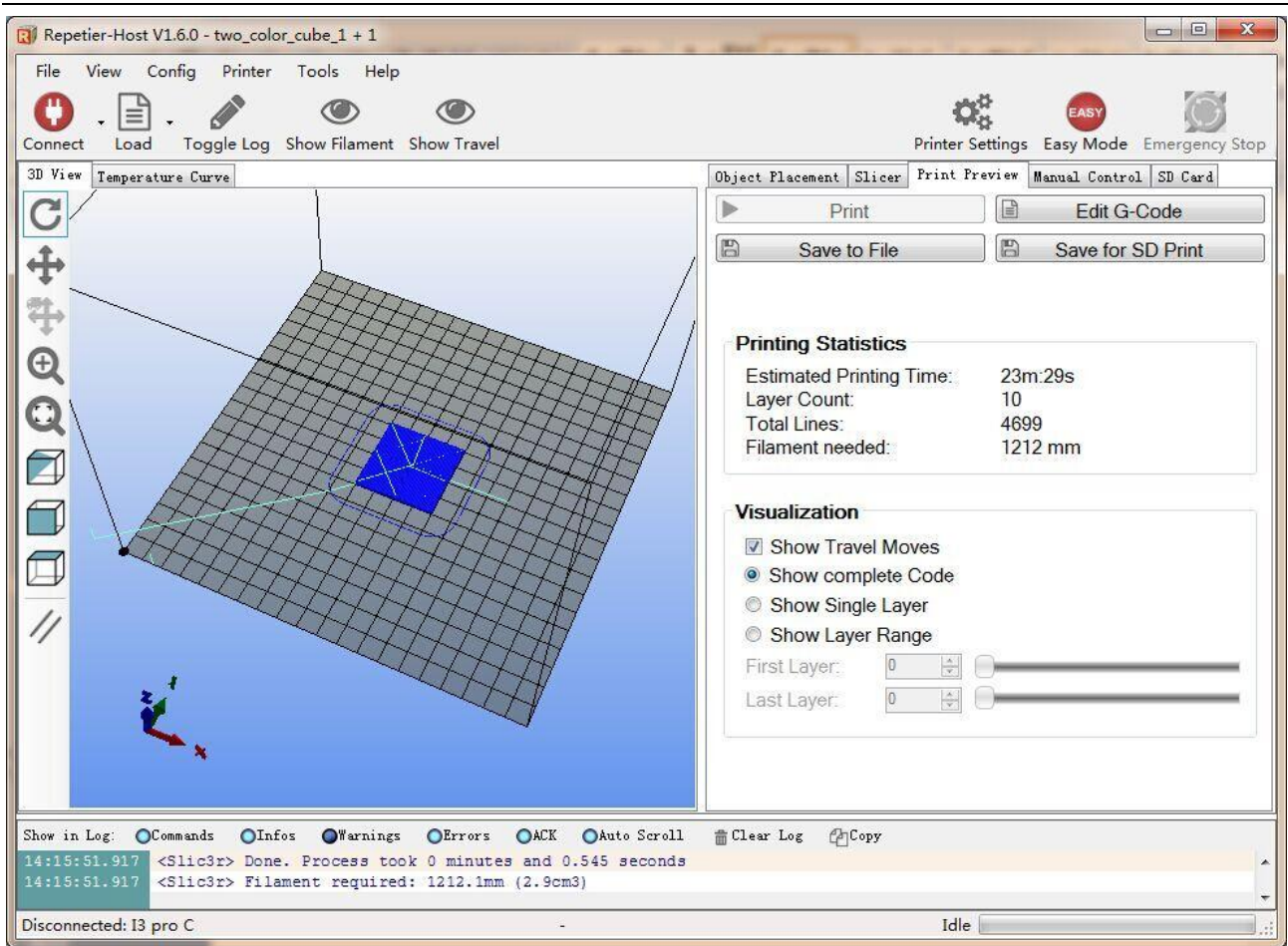


Then click **Slice with Slic3r**:

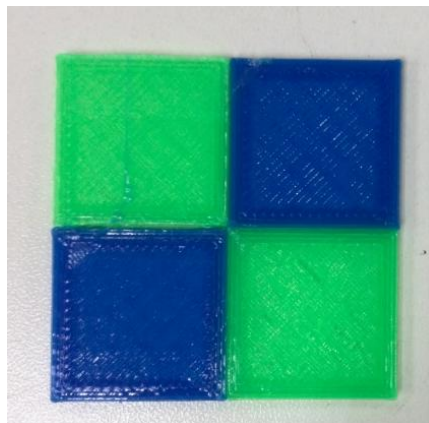


If there is a warning, you can neglect it and choose no.





Click **Start Print** to begin printing. The finished object is shown as bellow:



8.3 Stand-alone print with SD Card

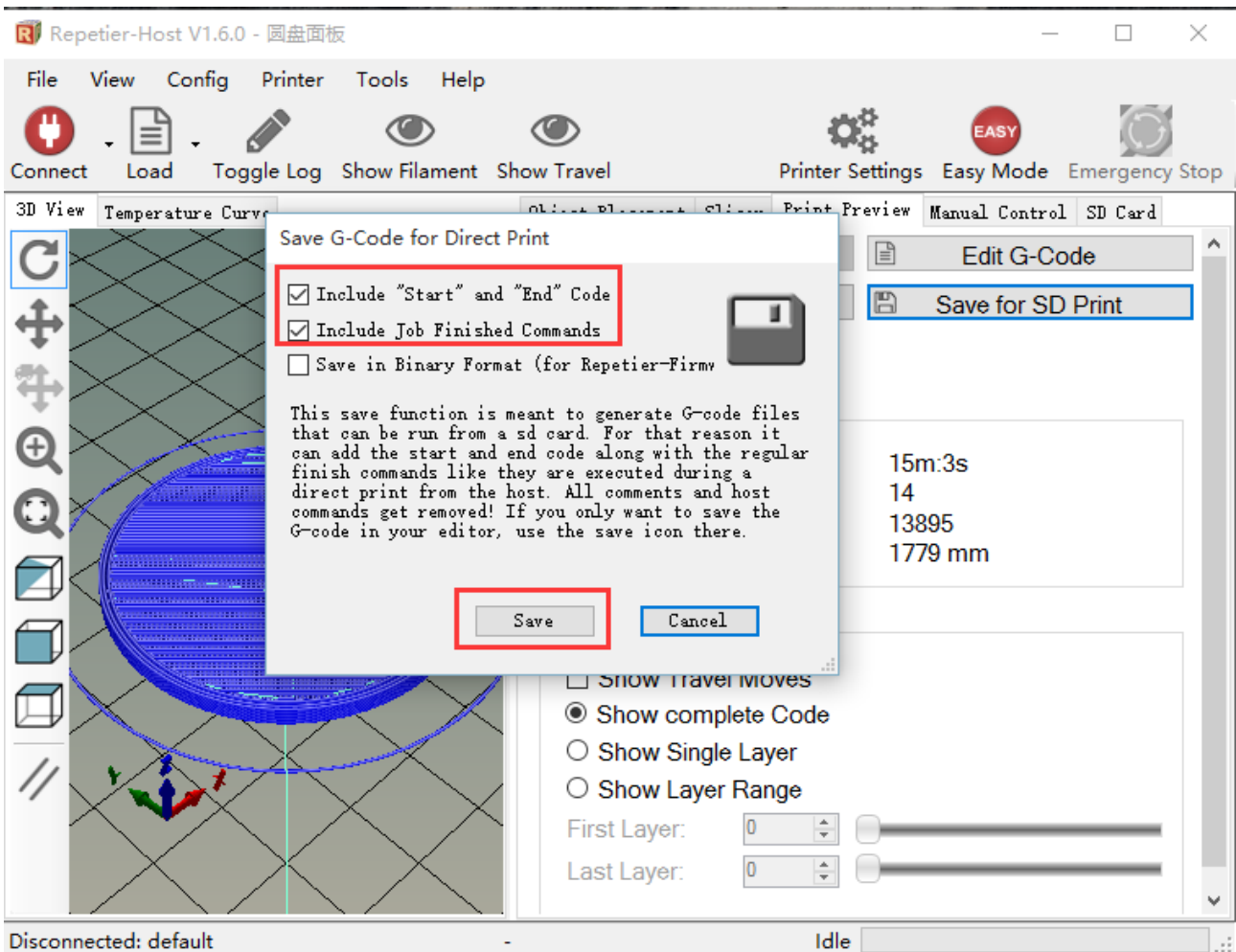
If you want to print Stand-alone , you can save the .gcode file in SD card for printing.

Note: The printer only recognizes .gcode file in the SD card, and the file can not be placed in any folder!

Steps are as follows:

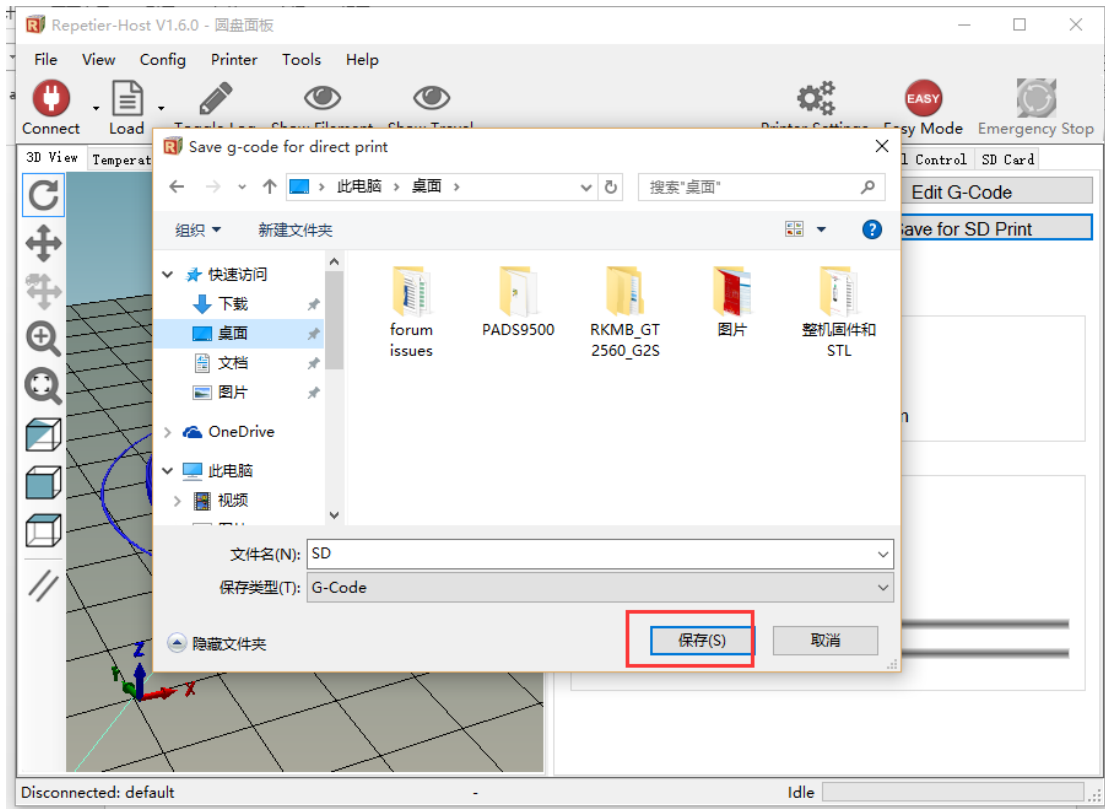
8.3.1 Save

Click Save for SD Print, and select the save button in the dialog box.



8.3.2 Generate file

Choose a save path to save the generated .gcode file which can be used for SD card printing.



8.3.3 Print

Insert the SD card into the SD card slot at the back of the LCD control panel, and choose the corresponding .gcode to print.

Press the knob on LCD, rotate the knob to enter the main menu and choose Print from SD option.



Choose corresponding .gcode file to begin printing.



Heating



When heating is completed, it begins printing automatically.

9 FAQ

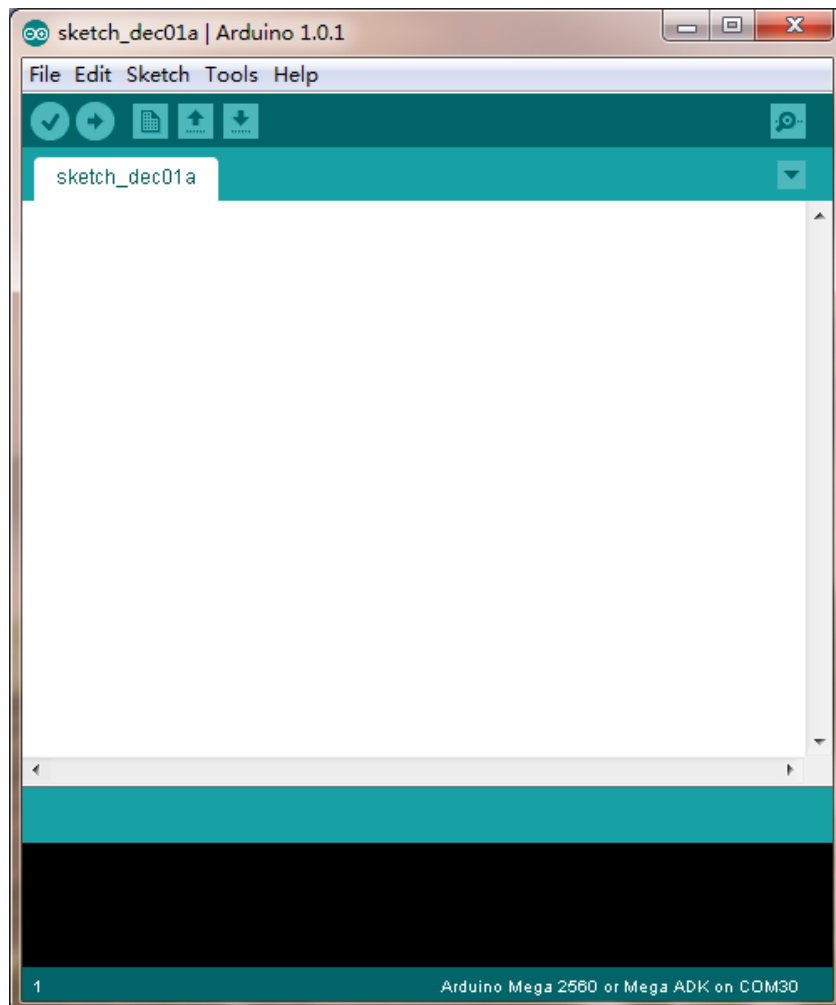
If you encounter any problems during use, you can come to our forum:

<http://www.geeetech.com/forum/>, and there are detailed solutions in the forum. Common problems are as follows:

9.1 How to upload firmware?

The motherboard of the printer is already burned with firmware when it leaves the factory. If you have firmware problem when you are using the product and need to upload firmware, you need to use the Arduino IDE to burn. Arduino1.0.1 is recommended, and the download address is:

<https://www.arduino.cc/en/Main/OldSoftwareReleases#1.0.x>



The way of uploading firmware, please refer to:

<http://www.geeetech.com/forum/viewtopic.php?f=13&t=17181>

Firmware download address, please refer to:

<http://www.geeetech.com/forum/viewtopic.php?f=10&t=17046>

Common parameters settings of firmware, please refer to:

<http://www.geeetech.com/forum/viewtopic.php?f=13&t=17194>

9.2 How to change the motor direction in firmware?

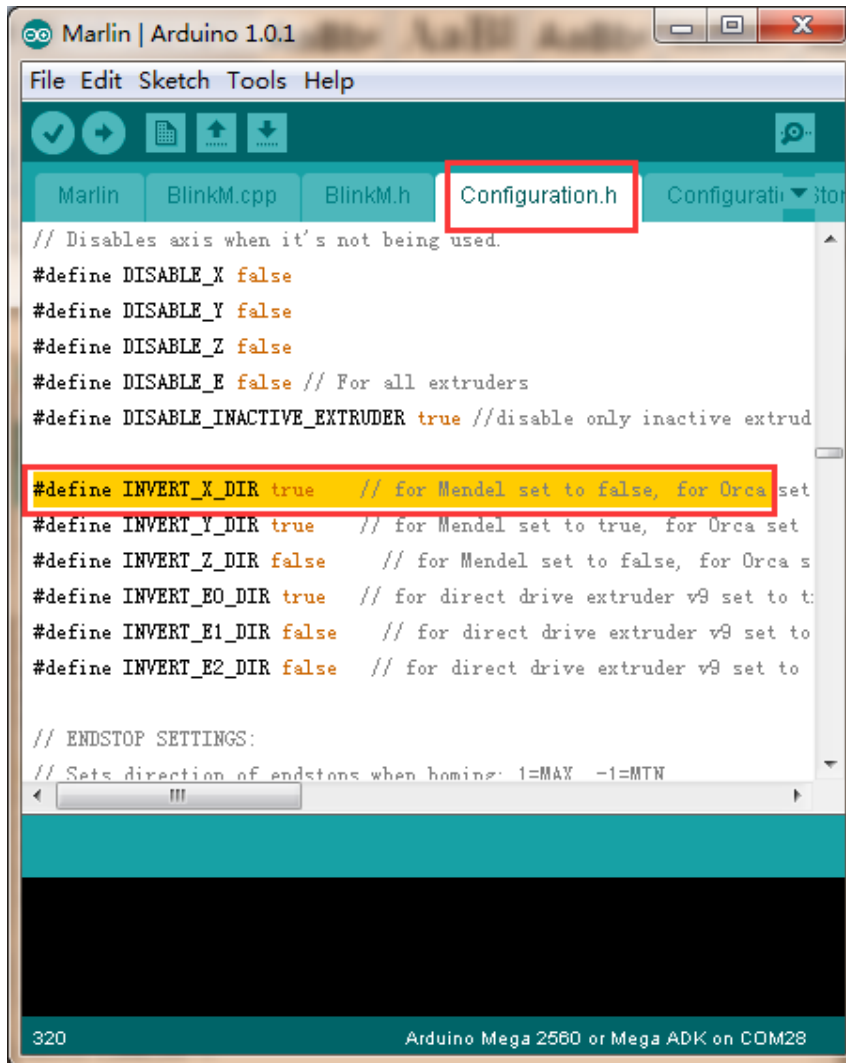
In the configuration.h tab of firmware, find the following codes. Change the true of corresponding axis into false or false into true, and save the firmware and burn into the printer. (Because you don't know the firmware which has been burned by mother board is true or false, it may be necessary to burn both of the two parameters.)

```
#define INVERT_X_DIR true
```

```
#define INVERT_Y_DIR false
```

```
#define INVERT_Z_DIR true
```

```
#define INVERT_E0_DIR false
```



9.3 User manual of Repetier Host

For detailed user manual of Repetier Host, please refer to Wiki:

<http://www.geeetech.com/wiki/index.php/Repetier-Host>

9.4 Possible causes for motor's non-action

<http://www.geeetech.com/forum/viewtopic.php?f=13&t=17038>

9.5 Extruder does not work normally

<http://www.geeetech.com/forum/viewtopic.php?f=13&t=17097>

9.6 The hotbed or extruder can not be heated

<http://www.geeetech.com/forum/viewtopic.php?f=13&t=17039>

9.7 LCD error: MAX/MIN TEMP error

<http://www.geeetech.com/forum/viewtopic.php?f=13&t=17030>

9.8 No information or displaying black square on LCD

<http://www.geeetech.com/forum/viewtopic.php?f=13&t=17040>

9.9 Time.h/endstop hit or other abnormalities

<http://www.geeetech.com/forum/viewtopic.php?f=13&t=17037>