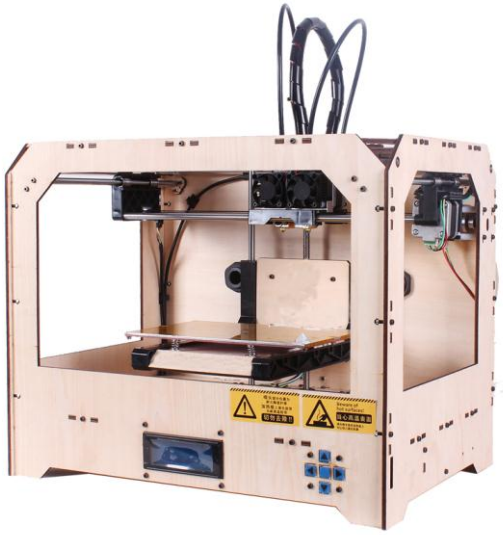
Desktop 3D Printer Start-up Guide

[Includes unpacking, setup, usage, and troubleshooting information]



Contents

Precautions……...3
What's in the box……..3
Unpacking……4
Hardware installation……...8
Software installation……11
USB connection & setting temperature ……..20
Feeding and withdrawing filament……..27
Setting parameters……...32
Initial print setup……..33
Dual-Extruder print……..37

Precautions

[Important: Read this user guide and heed all warnings]

! Because the Desktop 3D Printer is sensitive to static electricity, make sure to release your own body's static electricity by touching a grounded object before any operation of the Desktop 3D Printer. If you want to repair your Desktop 3D Printer by yourself, the power source must be shut off and the power cord must be unplugged.

! Beware of high temperatures. After operating, do not move the unit until it has cooled to the touch. Before attempting to repair, the heating plate should be cool to the touch. The extrusion material is very hot as it comes out of the extruder—allows cooling before handling.

! Operate the Desktop 3D Printer in a ventilated area. The machine may release a strong odor.

! Do not wear gloves when operating or repairing—entanglement may occur and cause injury.

! Do not leave the machine unattended when it is in operation.

What's in the box

Along with your Desktop 3D Printer, this package contains the following…

The accessory box on the top of the machine includes:

--One or two extruder
--1x or 2x filament holders
--bolt tool plate
--hex wrench tool box

Under the build platform in the Desktop 3D Printer framework, there are:

--1kg primary color ABS filament
--1kg black ABS filament (only for dual extruders)

Under the Desktop 3D Printer you'll find:

--power supply
--USB A to B cable
--1x or 2x filament guide tubes

Unpacking

The Desktop 3D Printer was carefully hand-packed.

Please follow the unpacking steps laid out below and heed all warnings.

! Handle with extra special care. Do not use any unnecessary force.
! Do not remove the thin yellow film from the heating plate. It is a heat-resistant adhesive tape intended to improve the adhesion of the materials to the plate.
! Do not remove the wrapping around the nozzle. It consists of ceramic fiber fabric and heat-resistant adhesive tape which helps the temperature remain constant in the heating zone and the filament is yielded uniformly.

First, put the box on the floor in a clean and flat work area. Remove the top carton , and then pull out the cardboard packing material that encases the Desktop 3D Printer.

Now, you can see the top of the printer, as well as some cardboard boxes. The large box with the black wire is the **accessory box**, which includes **extruders** and other important components. Don't remove the accessory box or its contents, yet. Note: The black cable is not a handle! Do not use it to lift the Desktop 3D Printer.

Now take the Desktop 3D Printer out of the box by grasping the frame. Be sure to grasp only the frame. Gently lift and transfer the printer to your work surface.

With the Desktop 3D Printer removed, you will find at the bottom of the box the **power supply and cable**, as well as a **USB A to B cable** and a **filament guide tube**. Take them out of the box and set them aside. We will now focus on the **accessory box**. Open the accessory box and remove the accessory sleeve.

You will find the **extruder** in protective packaging along the black cable. Handle it

with care as you take it from the box and place it on your work surface.

Remove the cardboard packing material and take the **accessory box** from the printer. Set the **accessory box** to the side for now.

The build platform should now be visible. It is an aluminium plate covered in a thin polyamide film. This is the platform that objects will print on. Remember: do not remove the film.

The next step is to raise the build platform. There are two ways to do this: 1) Turn the screw which is behind the rotating platform. 2) Grasp the printing platform with one hand on each side, and raise it slowly. Try to keep it level as you raise it up just shy of the black **nozzle**.

Under the build platform you'll find the filament -- either one or two rolls, depending on whether your Desktop 3D Printer has one or two extruders. It's easiest to remove the filament by first removing all of the remaining packing material.

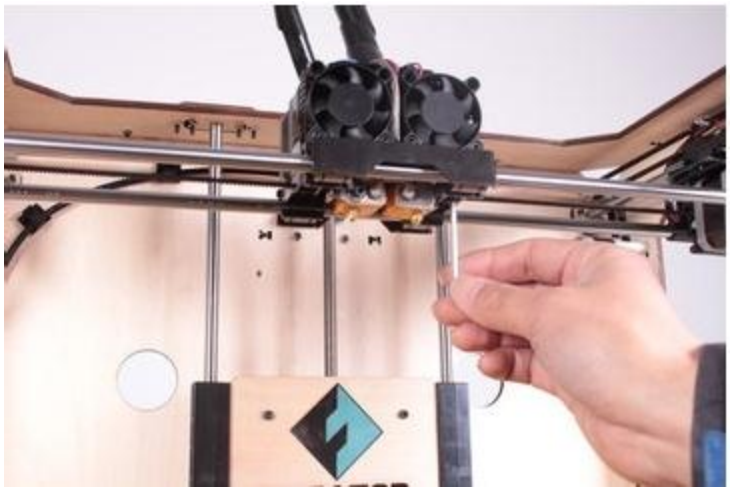
To do this, first take out the long box in the front, then the small box on the right, and finally the two wire trays.

You have now finished the unpacking job! Next: set up the hardware.

Initial hardware installation

Start by installing the extruder. You'll need two black screws from the bolt tool box found in the accessory box, and the appropriate hex wrench.

First, lower the build platform using one of the methods described above (either turning the screw or gently pushing down with both hands). Holding the extruder by its sides, take it out of the accessory sleeve and position it on the extruder seat with the fan facing forward. Align with the screw holes and fasten with the black screws.

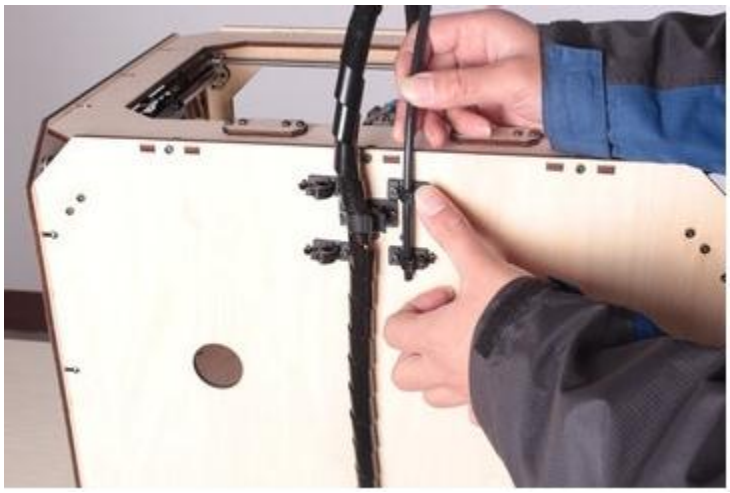


Next is the installation of the filament bracket. If you have two brackets, install one on each side; if you have only one, install it on the left hand side (when viewing the Desktop 3D Printer from the front).

The installation of the filament bracket is very simple -- just insert it into the circular opening and tighten the nut behind.

Then install the **filament guide tube** to the vacancy upon the extruder.

Put one end of the guide tube into the vacancy, buckle with the thumb and index finger to put in the vacancy on the top of the extruder.

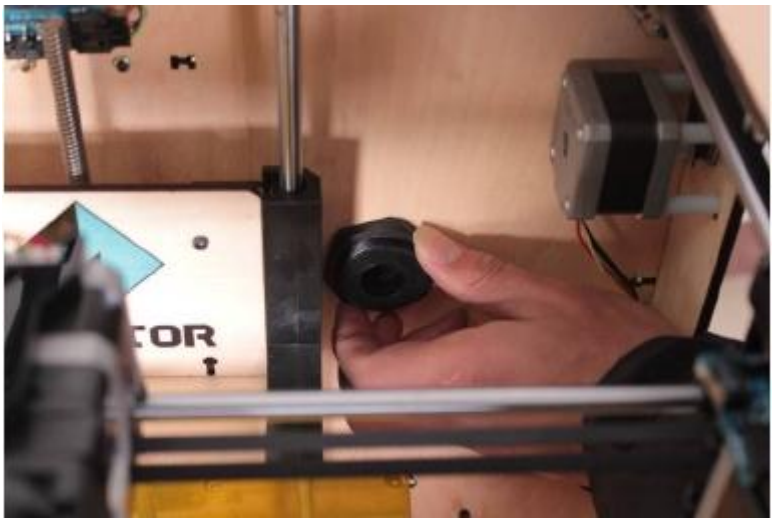


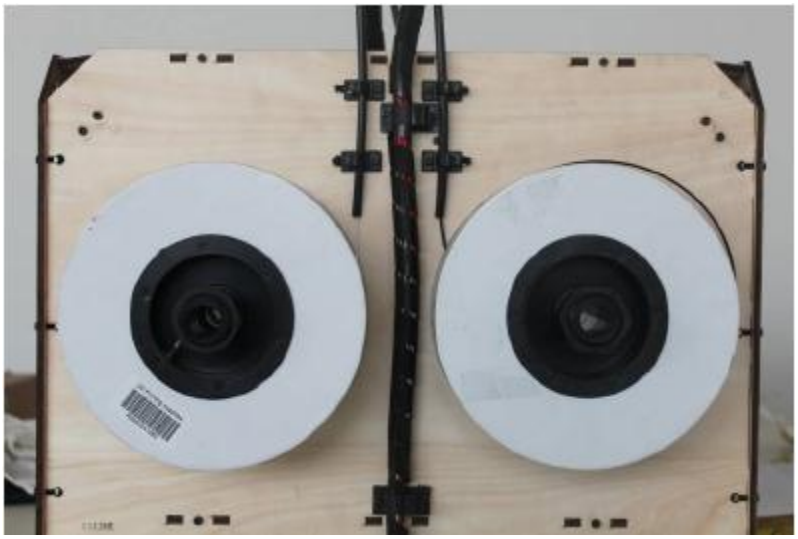
The hardware installation is almost completed.

Next, with the power switch in the "OFF" position, confirm that the power cord is plugged in to the power outlet next to the power switch.

Now plug the USB A TO B cable into the USB B-type port -- do not plug the other end in yet.

Finally, take the filament out of the box, install it on the bracket, and screw the retaining nut. The nut should not be over-tightened.





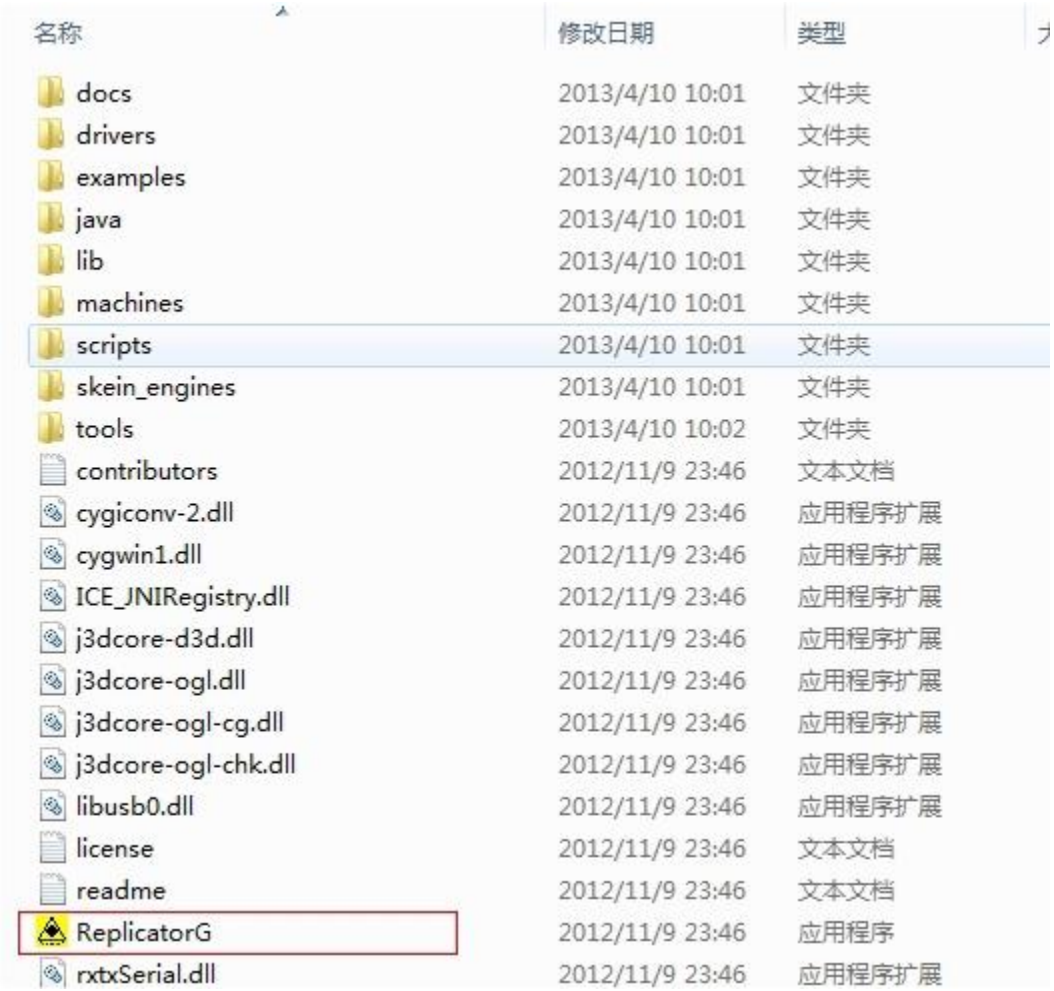
Congratulations, you have completed the initial hardware installation! If you're ready to get started printing, proceed to the next step: software installation!

Software installation

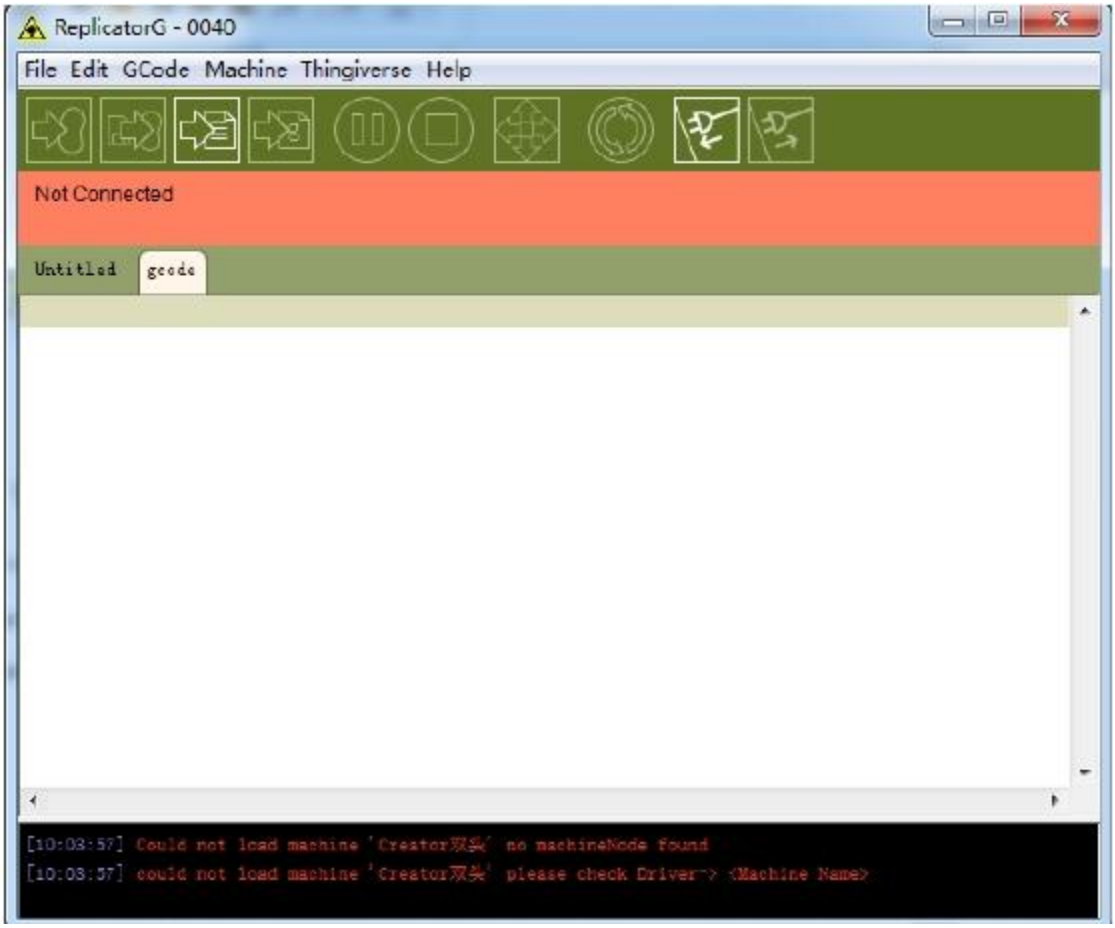
ReplicatorG0040 is the ideal software to drive your dual extruder Desktop 3D Printer. It can be downloaded from our official website:

Download drivers and softwares of the Desktop 3D Printer

Download the 3 extracting packages in the red box and decompress them one by one. Then install the python installation file and python acceleration components. Then click replicatorg-0040 which is the decompressed file of replicator G.



名称	修改日期	类型
docs	2013/4/10 10:01	文件夹
drivers	2013/4/10 10:01	文件夹
examples	2013/4/10 10:01	文件夹
java	2013/4/10 10:01	文件夹
lib	2013/4/10 10:01	文件夹
machines	2013/4/10 10:01	文件夹
scripts	2013/4/10 10:01	文件夹
skein_engines	2013/4/10 10:01	文件夹
tools	2013/4/10 10:02	文件夹
contributors	2012/11/9 23:46	文本文档
cygiconv-2.dll	2012/11/9 23:46	应用程序扩展
cygwin1.dll	2012/11/9 23:46	应用程序扩展
ICE_JNIRegistry.dll	2012/11/9 23:46	应用程序扩展
j3dcore-d3d.dll	2012/11/9 23:46	应用程序扩展
j3dcore-ogl.dll	2012/11/9 23:46	应用程序扩展
j3dcore-ogl-cg.dll	2012/11/9 23:46	应用程序扩展
j3dcore-ogl-chk.dll	2012/11/9 23:46	应用程序扩展
libusb0.dll	2012/11/9 23:46	应用程序扩展
license	2012/11/9 23:46	文本文档
readme	2012/11/9 23:46	文本文档
ReplicatorG	2012/11/9 23:46	应用程序
rxtxSerial.dll	2012/11/9 23:46	应用程序扩展

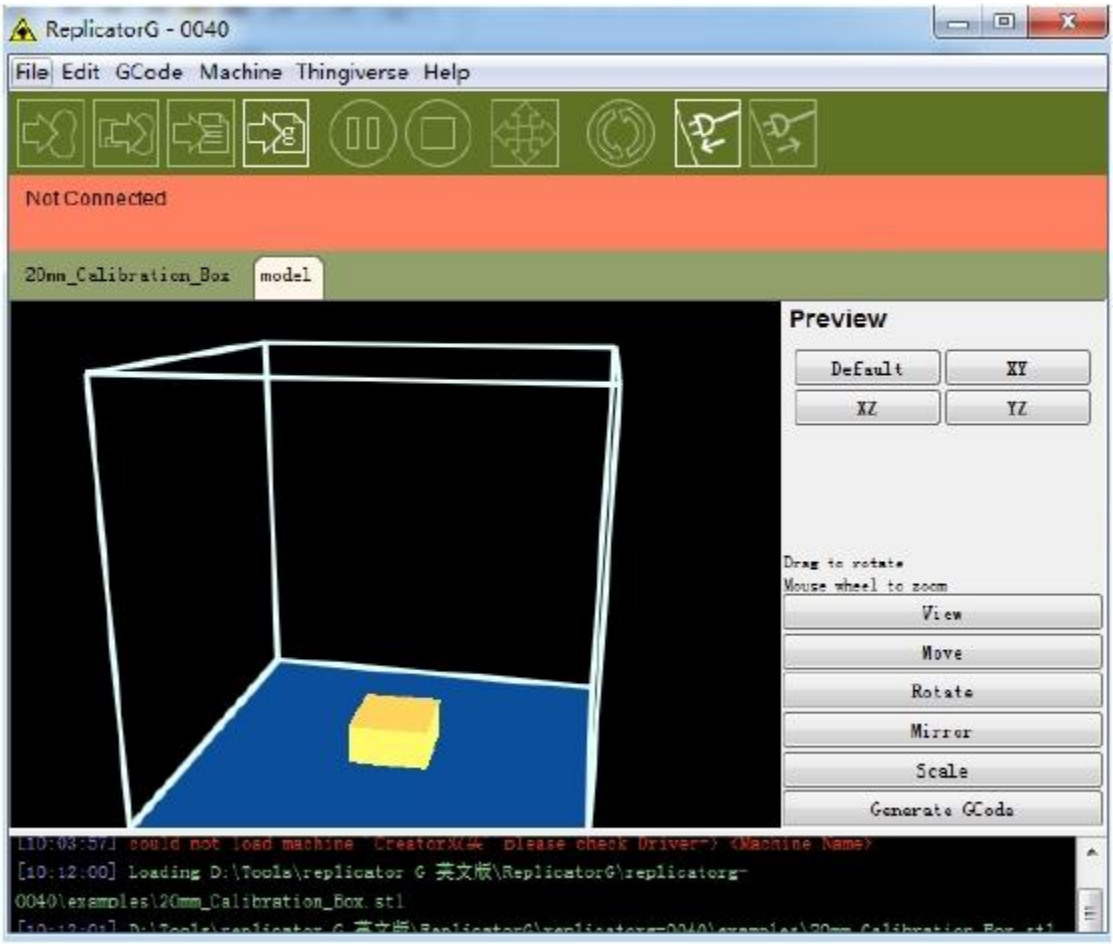


（the text and green code on the above image can not be seen ）

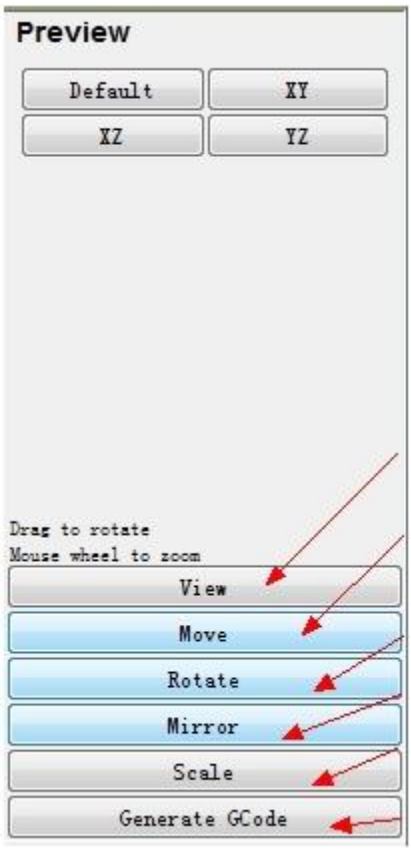
Then we will give a detailed introduction on how to import files and generate G-code .

Click on the **open** option in **file** 

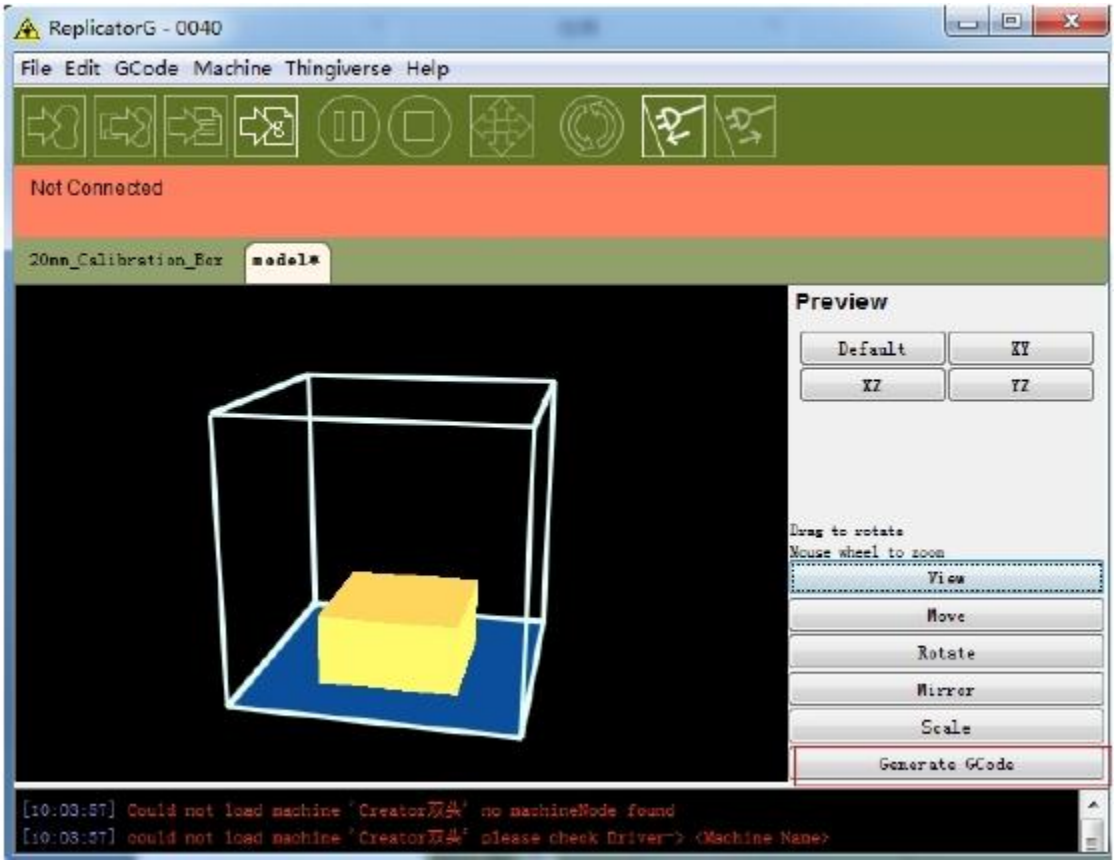
, click the file you want to print (STL format), and import it by double-clicking. Then the work piece of the drawing design will appear in the interface.



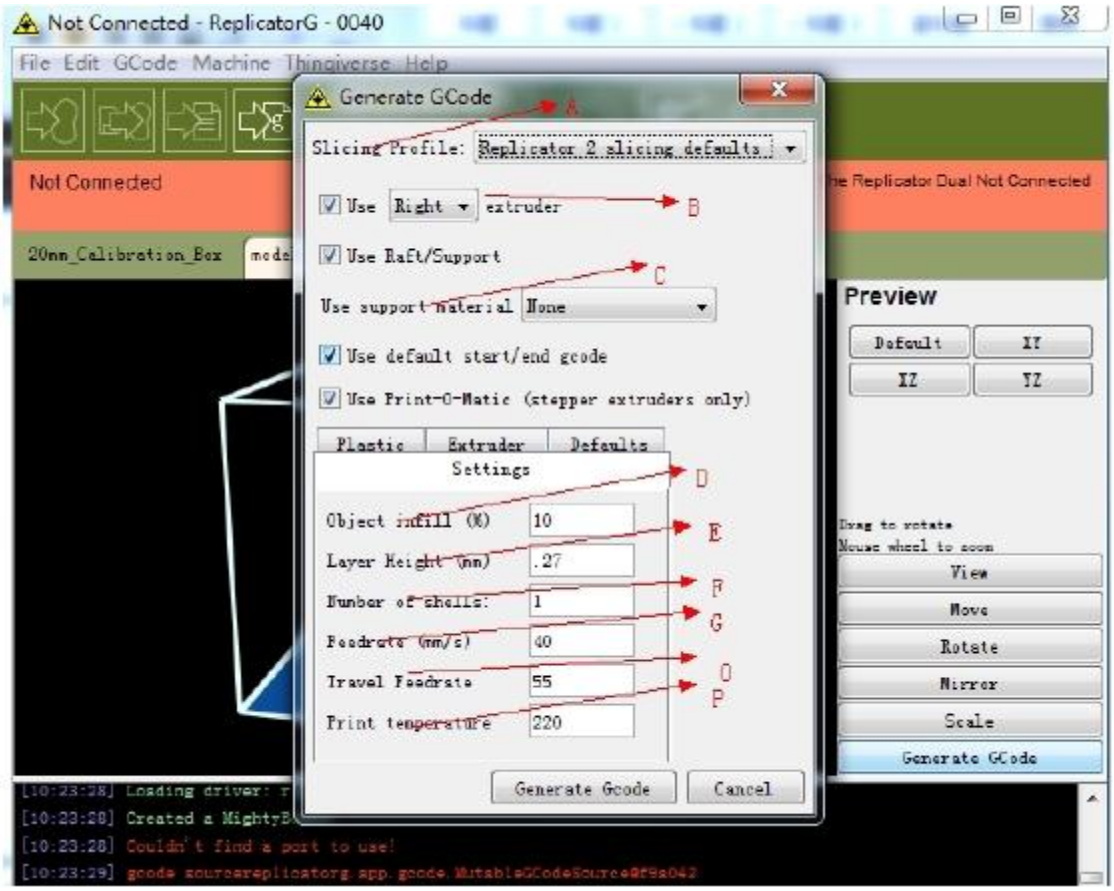
After the image is imported, there may be some problems, such as the diagram is unseen or the position is wrong. In that case, the following function keys can be used:



When the position is in the center, which is the ideal printing position. The next step is to generate G-code.

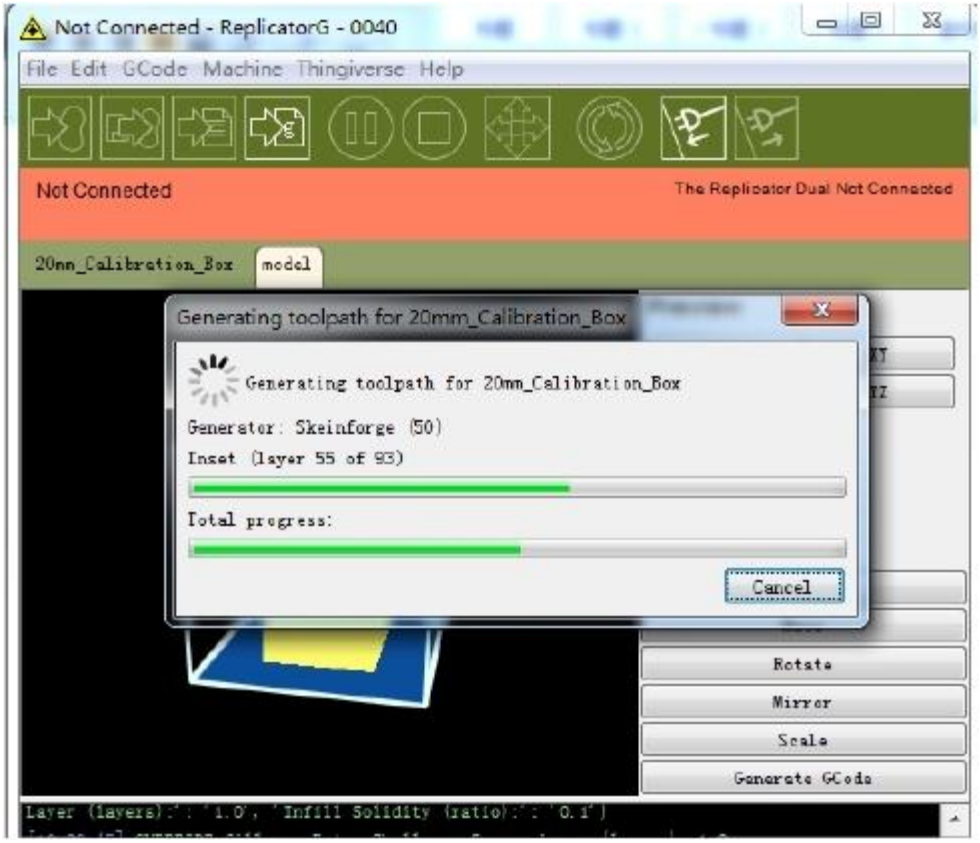


You can get the G-code by clicking the button in the red box.



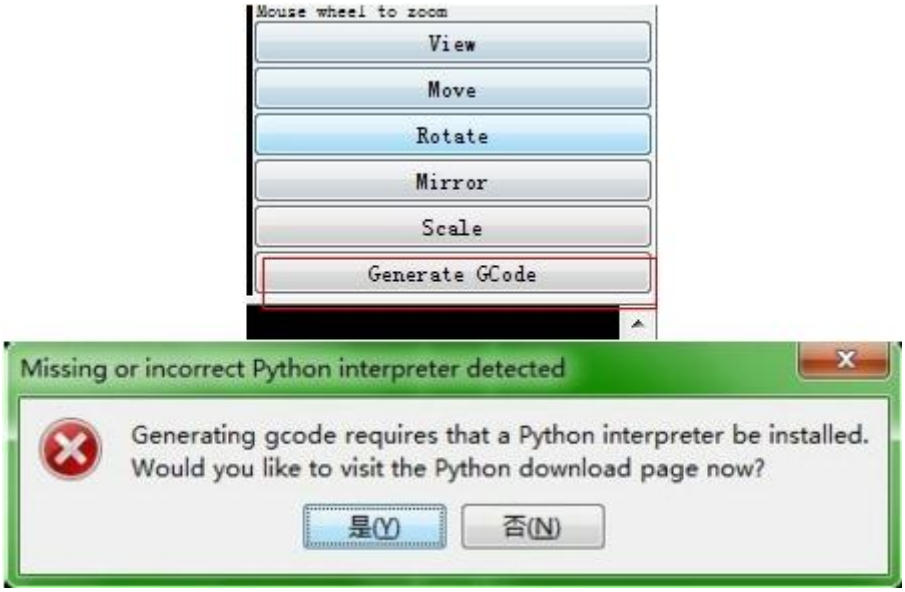
A: Slicing profile. Select replicator slicing defaults
B: Left and right extruder print selecting. Left means printing with the left extruder, while right means printing with the right extruder.
C: If your sample is in dangling structure, print support is needed. None means no support. Exterior means surface support. Full support means all support.
D: Object infill. 100% is a solid print. We usually choose 10% fill in order to save filament.
E: Layer Height. Layer thickness is related to precision. Generally, the minimum print thickness is 0.18mm. 0.27mm is most commonly used.
F: Number of shells is the wall thickness -- usually set at 1
G: Feedrate is generally in the range of 70-30
O: Travel feedrate is generally in the range of 70-30
P: Print temperature is 220 degrees

Click G code Generation, a progress bar will appear

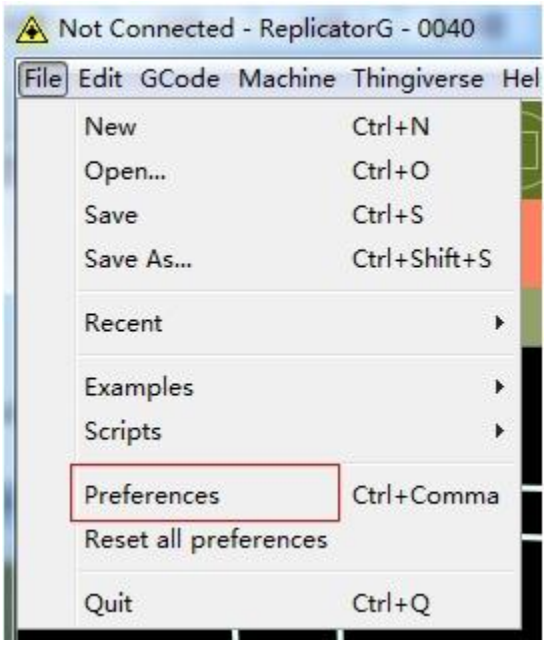


And there is another possibility:

Users who do not choose the default installation path in the installation of python, can click "G-code generation" button in Replicator G, and a dialog box will pop up alerting that the executable file python can not be found.

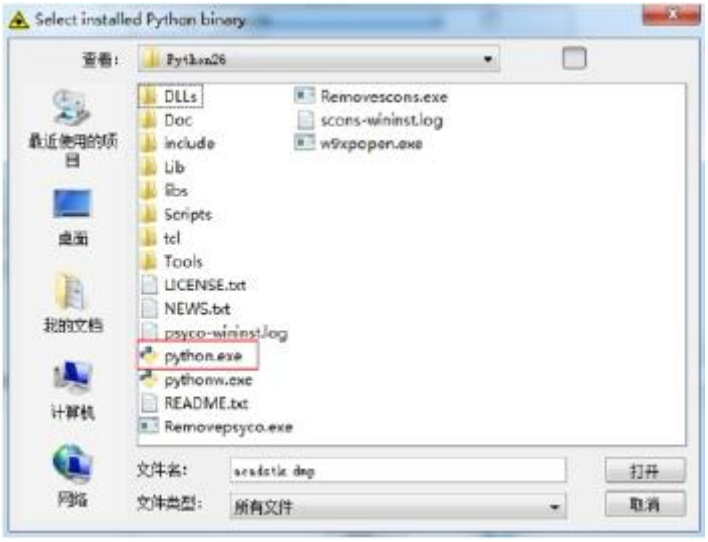


First click "N" button in the above dialog box to close the dialog box. To solve this problem, we need to configure the corresponding menu.

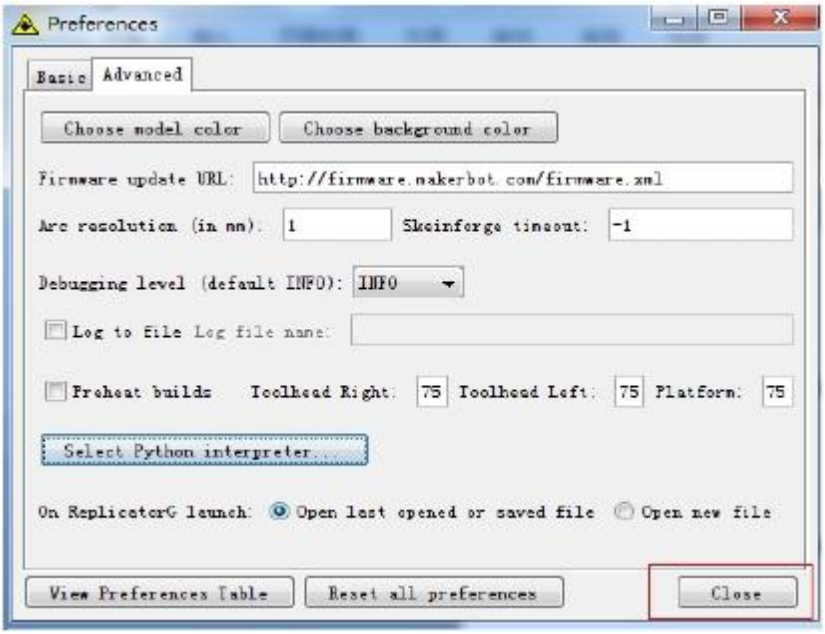


Select the "parameter setting" option in the "File".

Click the button “select Python interpreter”.



Find the Python installation directory in the dialog box, select python.exe and click "Open".

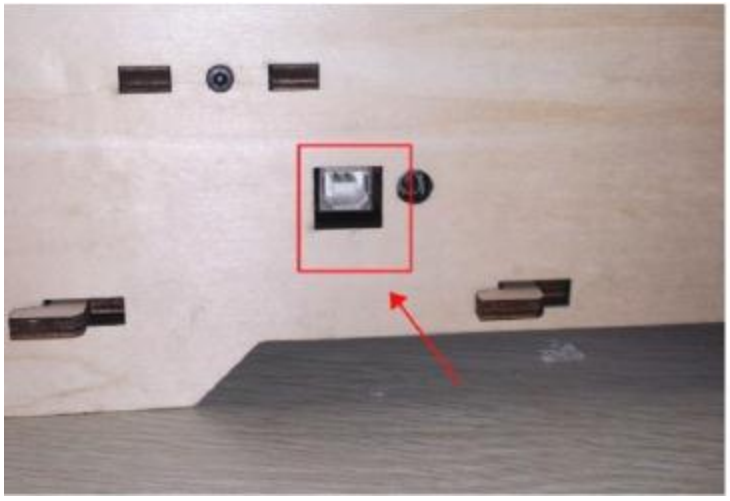


After clicking the "Close" button, the settings are done!

The machine will work normally by clicking the “G-code generation” button. Next we will start a preliminary test on the machine’s connections and we will heat the platform and extruder.

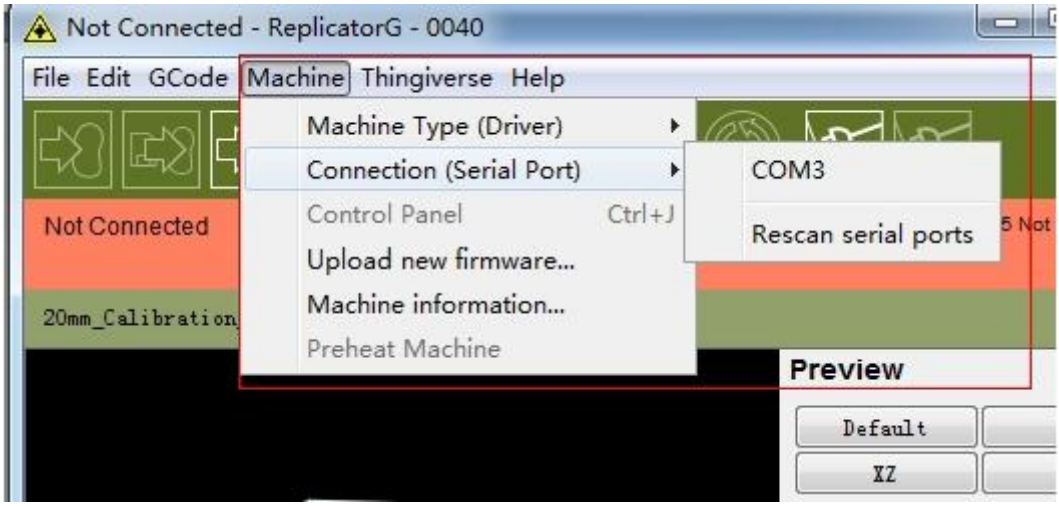
USB connection & setting the extruder and platform temperature

First, connect the machine and computer with the provided USB cable.

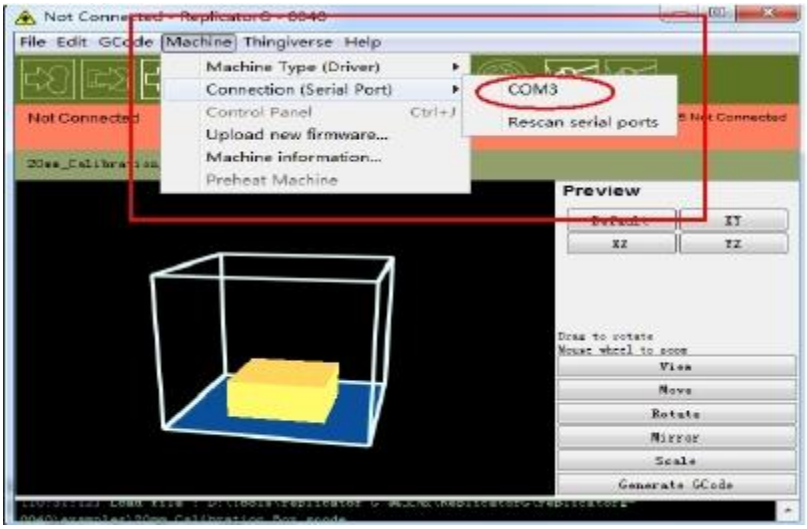


This is the USB port on the machine. After connecting the cable, open the software -- we are going to connect the computer and printer.

Click the **Machine**,



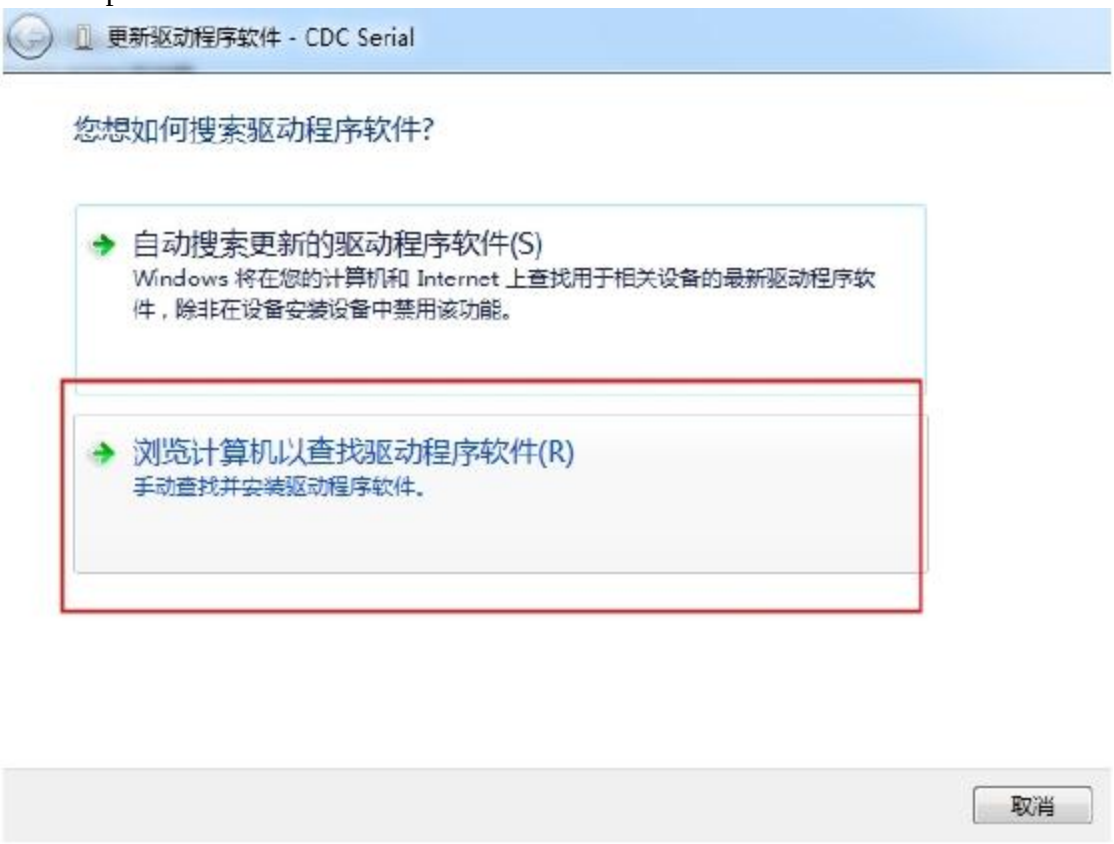
Choose the second option (Serial Port) on the menu, and we will find no port connected. Click rescan serial port, and the port will appear.



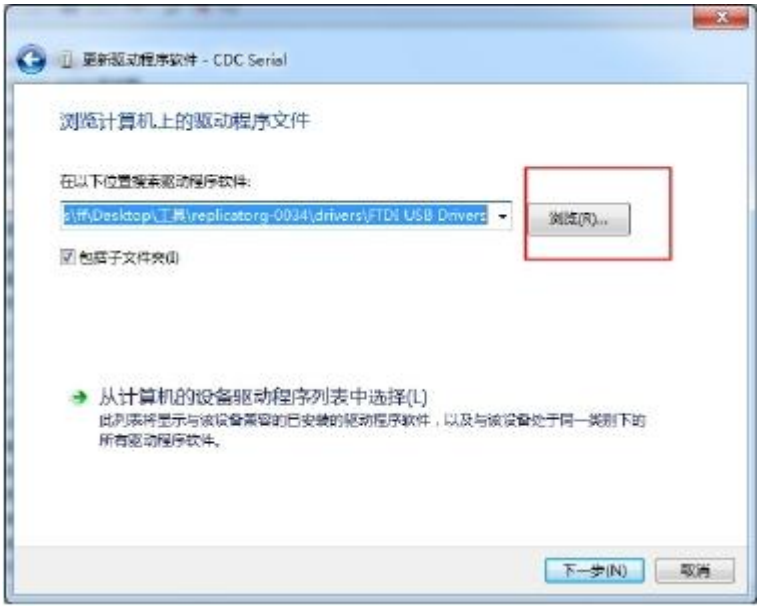
The software driver has not been installed if no port appears; let's install the driver. Click My Computer; right click select properties, then the basic system parameters appear. Then select Device Manager.

This software driver in the red box is not installed. Right click the Replicator, and

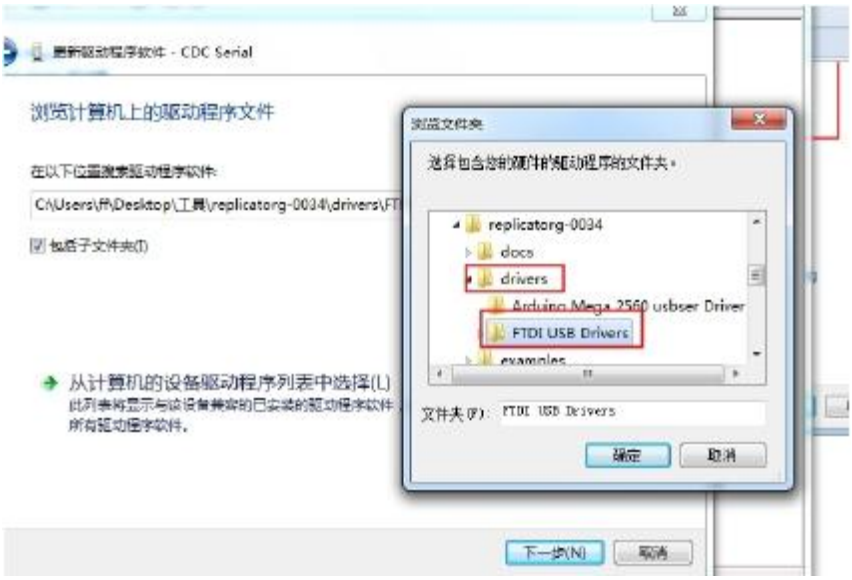
select Update Driver Software.



Select browse my computer to search for driver software.

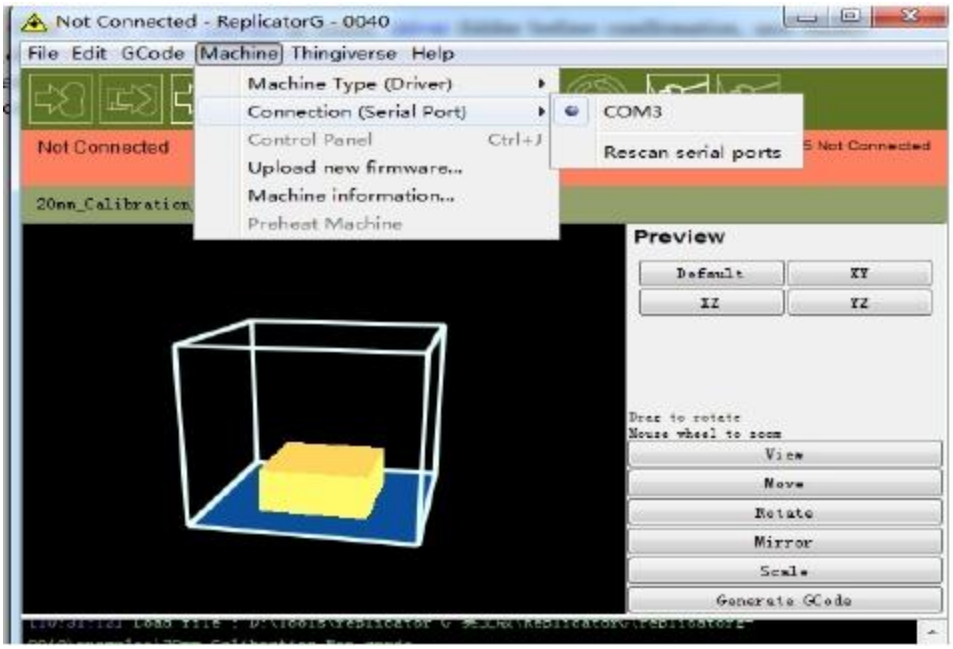


Click Browse to find the location of software 0040.

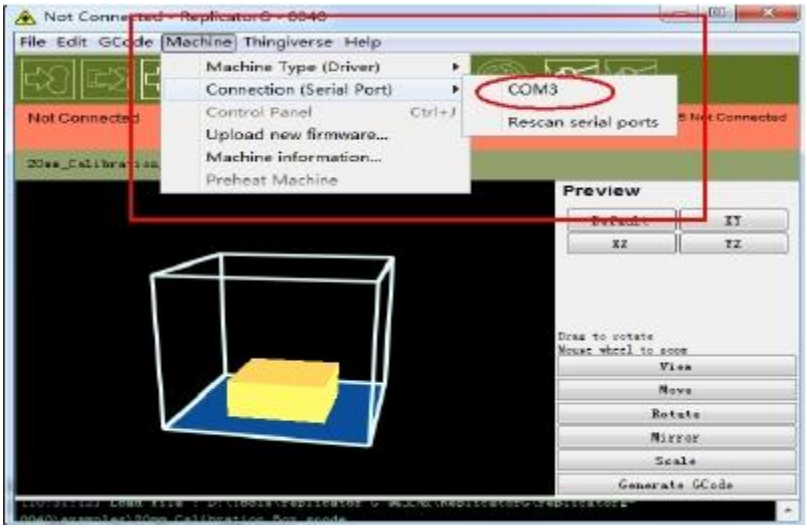


Click FTDI USB Drivers in the driver folder before confirmation, and finally click next. The driver is now installed.

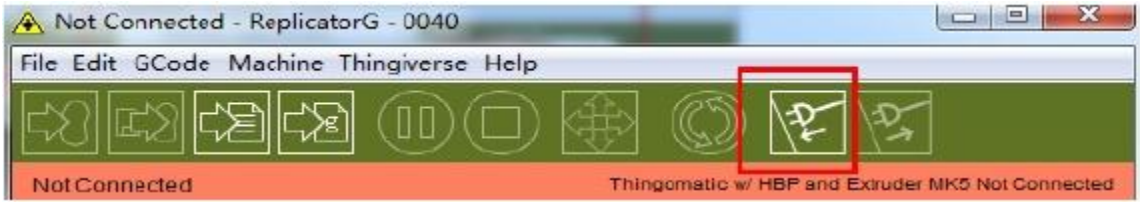
Next is to connect to the printer. The steps are the same. Click printer; select the second connection (serial port).



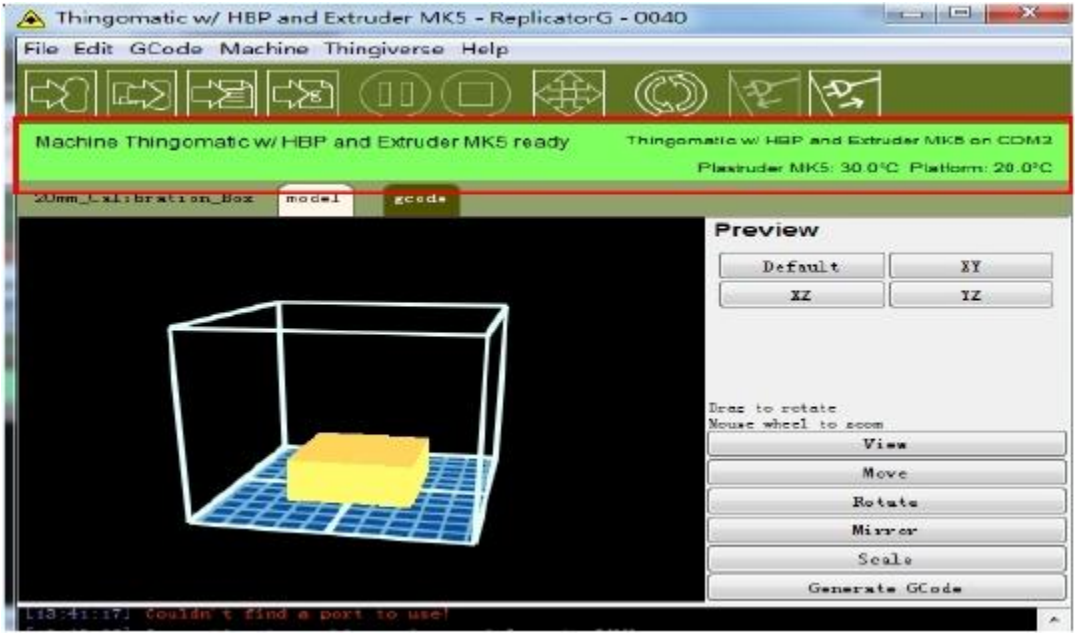
The port COM3 (this is the port for our test machine, it is different in each machine) appears. Click COM3.



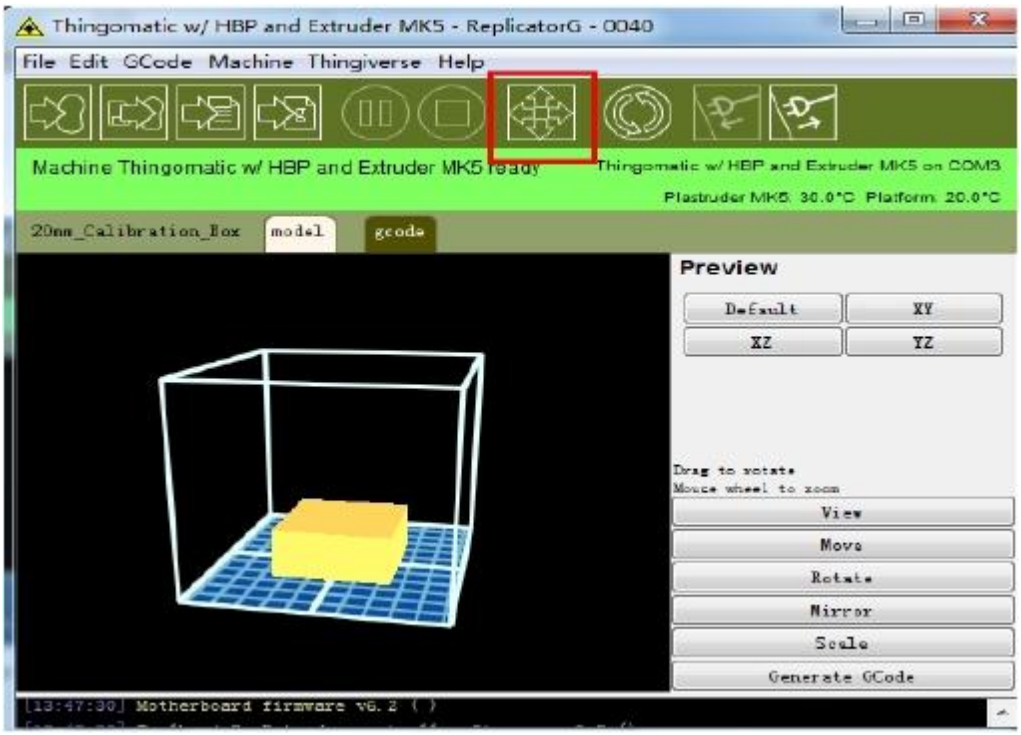
Next, we can connect the machine.



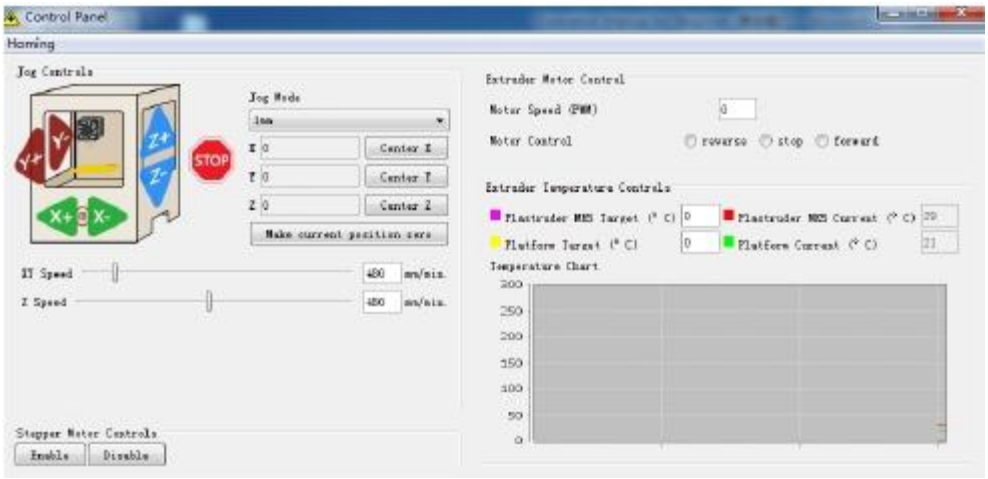
Click the function keys in the red box.



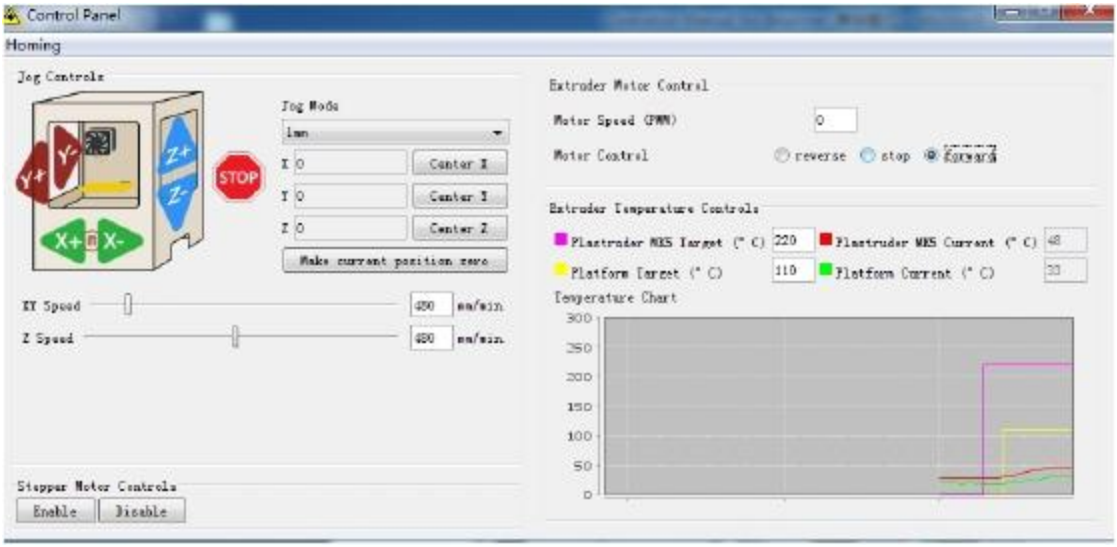
If red turns to green, the machine is connected with the computer. The next step is to heat the extruder and build platform.



An interface will appear by clicking the cross-shaped function key.



Input the target values. 220 degrees for the extruder (maximum 230 degrees); 115 degrees for the heating platform (maximum 120 degrees). After input, the platform will start to warm up. When the extruder temperature reaches 50 degrees, the cooling fan will activate and the current temperature value will display to the right, as shown below.



It shows that the heating of the sprinkler and bottom plate is normal.

Feeding and withdrawing filament

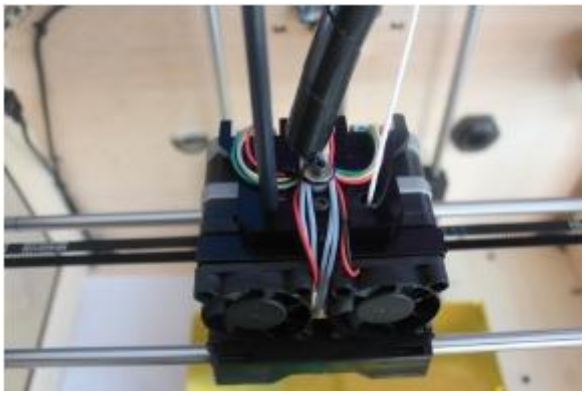
Some users report that it’s easy to feed filament, while others report that it’s hard -- what would explain this? Sometimes, if the location is wrong (trying to operate at different angles), the filament feeding wheel is not secure and the material cannot be fed. To prevent this, please do the following:

After putting the filament into the feeding hole, do not push it further until the extruder temperature reaches 200 degrees or more, and not until you can feel that the filament is being drawn in.

Withdrawing of filament is the same; the material cannot be withdrawn until the temperature reaches 200 degrees or more. Put your hands on the filament near the feeding hole and withdraw it only once you feel that the material is being discharged.

How to set the filament

First, remove the filament Guide Tube by pulling out.



When the guide sleeve is disengaged, you can keep pulling the material from the guide sleeve behind the device until the thread passes through from the front.

To avoid blocking in print, please ensure that two threads are loaded from the middle. There are two wire trays -- one runs clockwise and the other one runs counter-clockwise, as shown below:

After the materials are inserted into the extruder, do not insert the guide wire tube into the extruder before the machine has started. There are two ways of feeding: one is to feed by using the LCD screen, the other is to feed using the control panel of the Print software.

Feeding the material by LCD option

1. When you start your machine, the display indicates:

    Build from SD
    Preheat
    Utilities

2. Click the button board on the right, and then click the page down key to advance to the second page. The display indicates:

    Preheat
    Utilities
    Info and Settings

3. Select Utilities; click the M key in the middle of the button board. The display indicates:

    Monitor Mode
    Change filament
    Level Build Plate
    Home Axes
    Feed operation with LCD screen

4. Select Change filament. Click the M key in the middle of the button board. The display indicates:

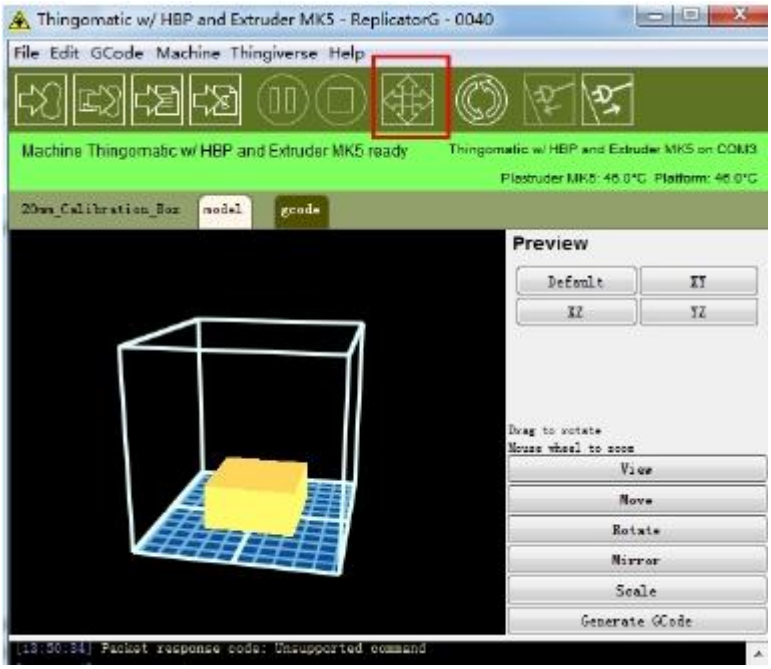
    Load Right
    Unload Right
    Load Left
    Unload Left

5. Select Load right. Click the M key on the button board. The display indicates: I'm heating up my extruder! At this time, the temperature of the right nozzle is being heated up. When the temperature reaches 220 degrees, click the M key on the button board. The nozzle should spit material. If it does not, keep clicking the M key until it is done.

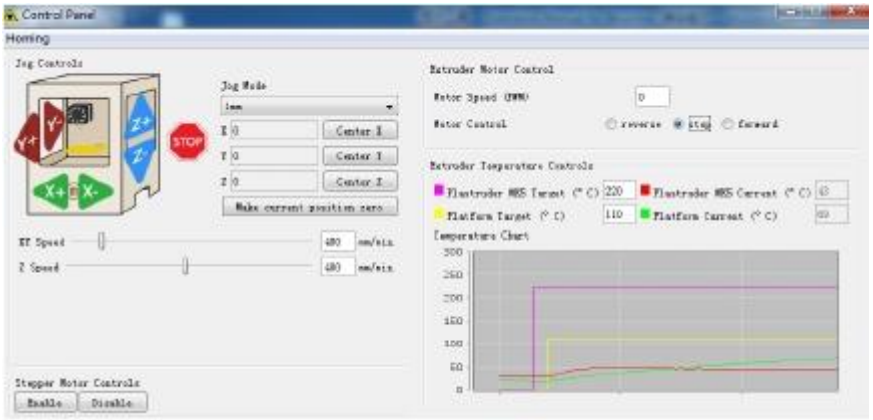
Feed operation using Replicator G control panel

This operation cannot be done until your machine has been connected to your computer.

If the print software is successfully connected, a dialog box will pop up:



Click the icon in the red box -- this is the Control Panel icon. The dialog box shown below will then pop up.



To heat the right extruder and feed material: click the right extruder in red box on the upper right corner of the control panel -- manually modify the temperature in the right

target of extruder temperature control -- and change the temperature to 220 degrees. A rose red line will move around in the temperature map. When the actual temperature reaches 220 degrees, you can pull out the filament out.

How to withdraw the filament

1. When you start your machine, the display indicates:

    Build from SD
    Preheat
    Utilities

2. Click the button board on the right, and click the page down key to advance to the second page. The display indicates: Preheat

    Utilities
    Info and Settings

3. Select Utilities. Click the M key in the middle of the button board. The display indicates: Monitor Mode

    Change filament
    Level Build Plate
    Home Axes

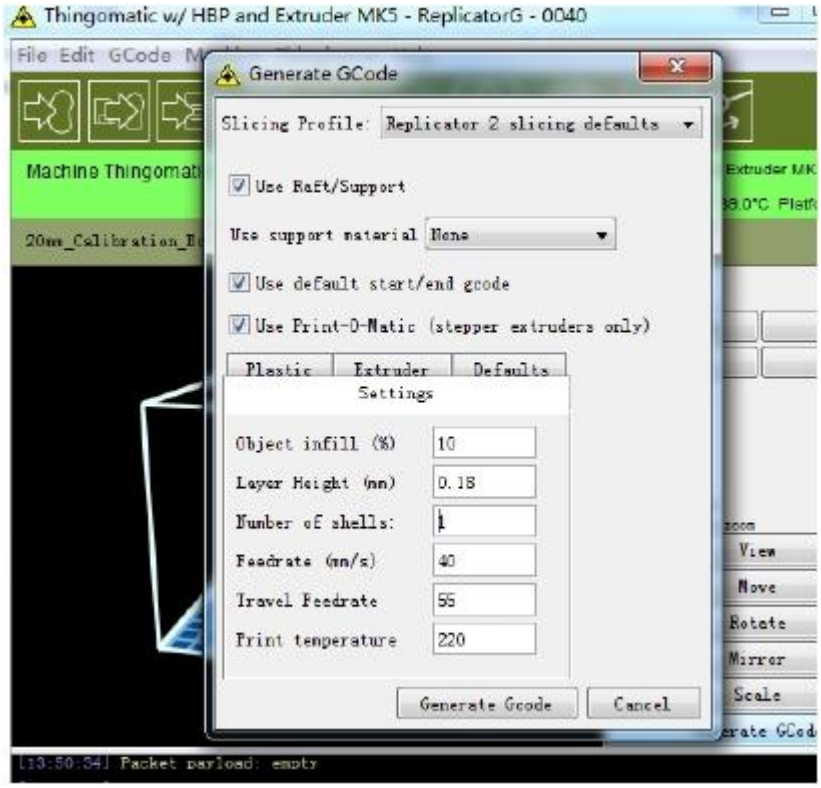
4. Select Change filament. Click the M key in the middle of the button board. The display indicates:

    Load Right
    Unload Right
    Load Left
    Unload Left

5. Select Unload right. Click the M key on the button board. The display indicates: I'm heating up, my extruder! At this time, the temperature of the right nozzle is being heated. When the temperature reaches 220 degrees, click the M key on the button board. The nozzle will discharge material. If it does not discharge, continue to click the M key until it is done.

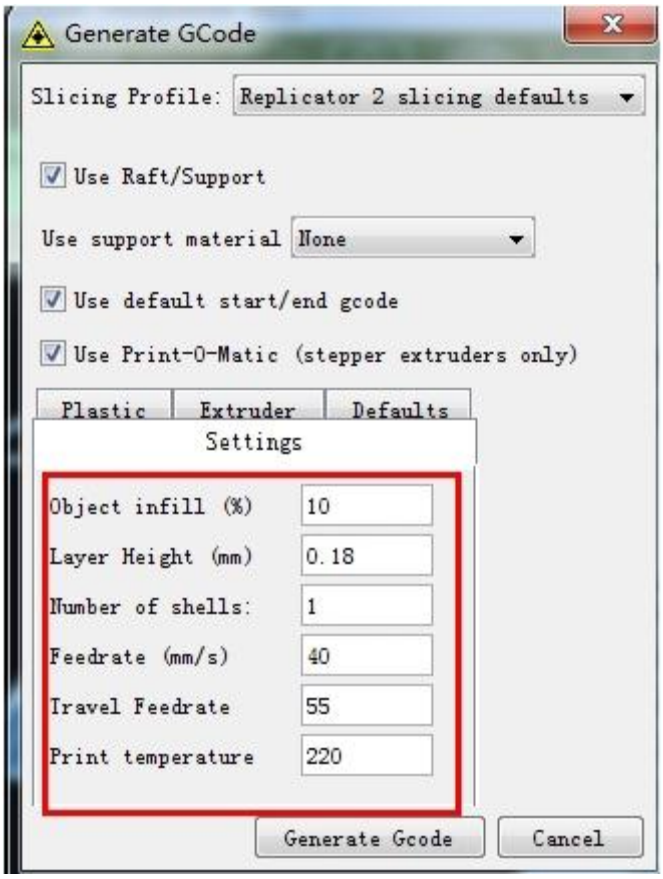
Setting parameters

3D printer parameters include fill ratio, layer thickness, wall thickness, wiring speed and idling speed. The parameters are shown below.



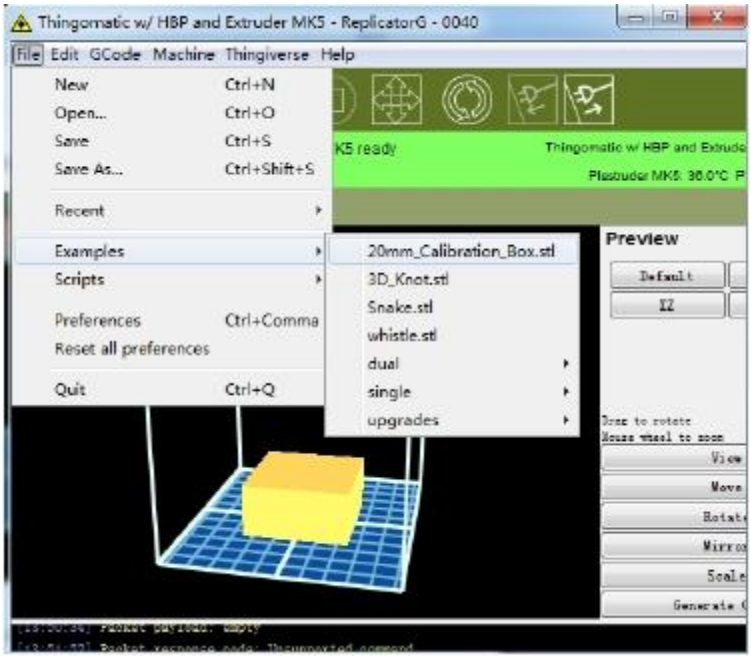
Filling ratio: the rate to fill the object -- 100% percent is solid.
Layer thickness: the wiring thickness of each layer.
Wall thickness: the outer thickness of the printed article.
Wiring speed: the extruder speed in print.
Idling speed: the nozzle operation speed when in nonprinting state.

These parameters affect the printing quality. The default settings are relatively stable. You can also find your own parameters, as shown below.

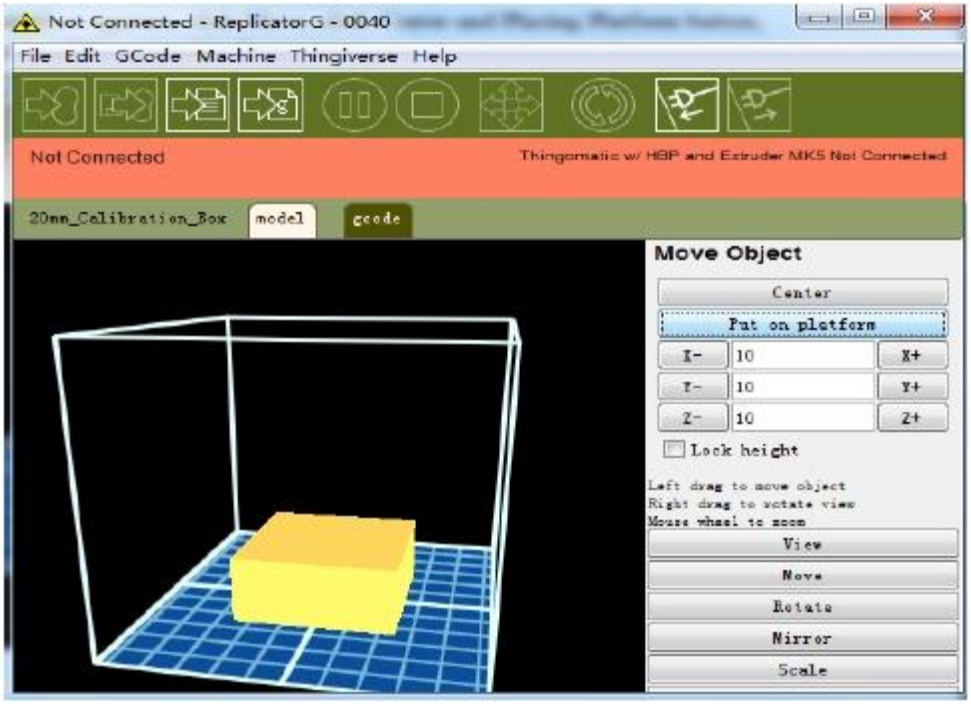


Initial print

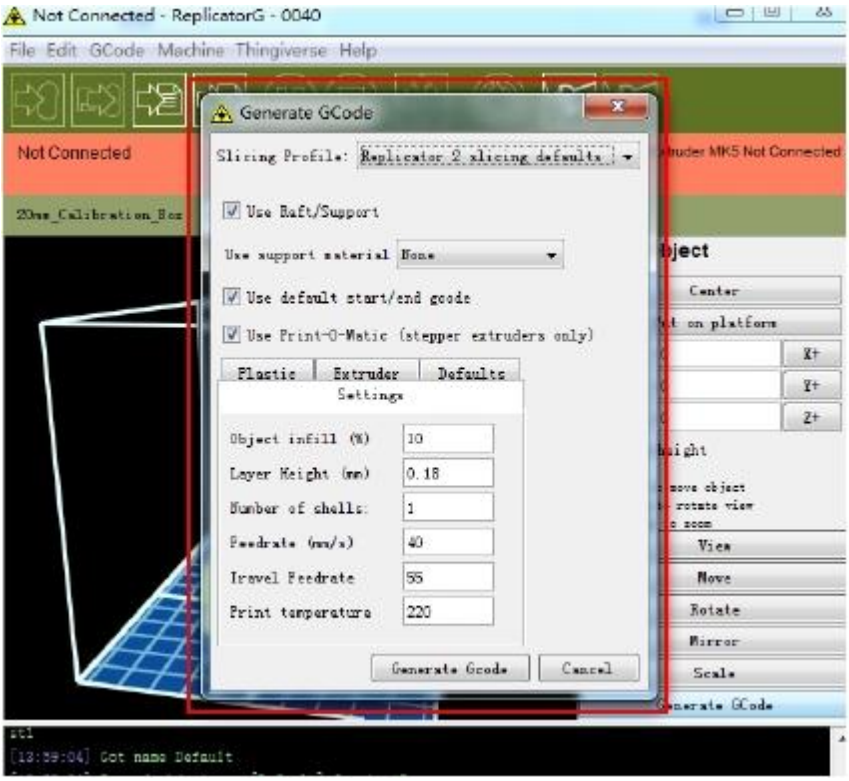
First click file column to select example, print a small cubic of 20mm and check the finished product, then this picture pops up.



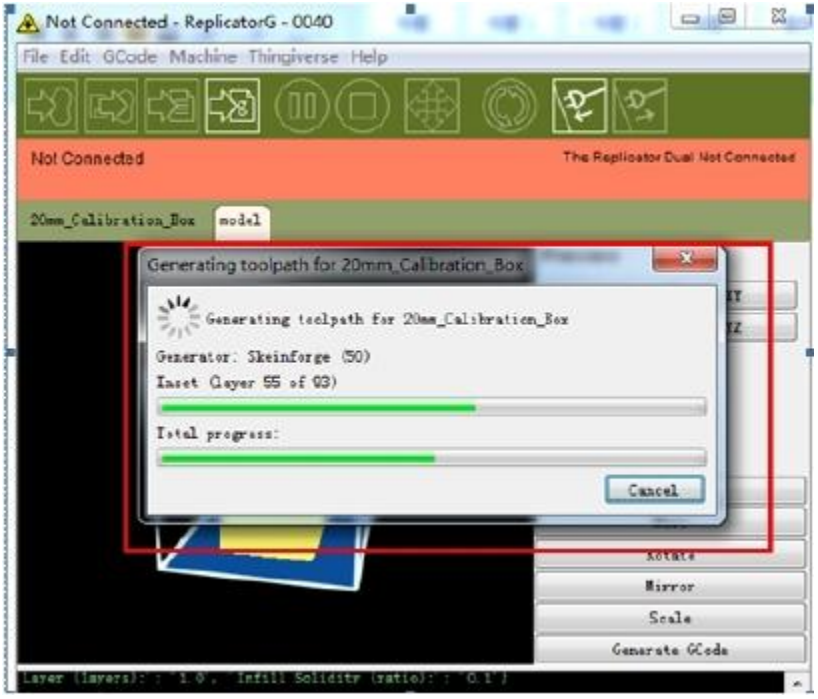
When this interface appears, click Move, click Center and Placing Platform button, so that the sample will be printed in the center.



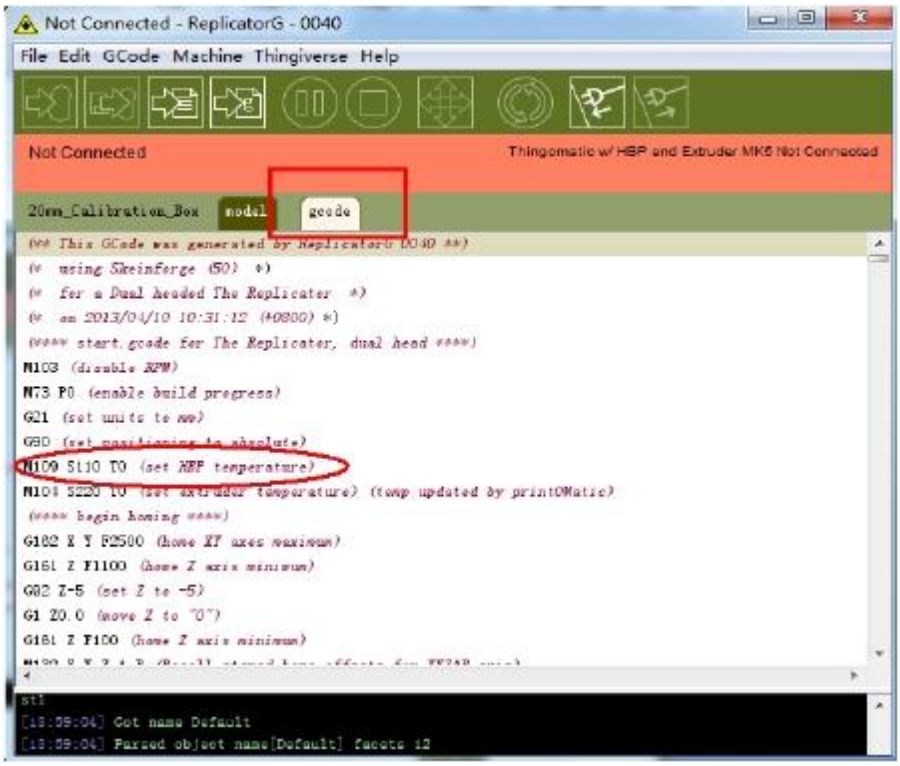
After clicking the two buttons, click G Code Generation button, a Generation G Code dialog box pops up. Parameters will be set according to the data setting in the picture, as shown in Figure



Click G Code Generation, the following picture appears



After G code is generated, the temperature of the bottom plate should be changed, please check the following picture for modification. First click G code in red box, and then change M109 S110 T1 to M109 S115.



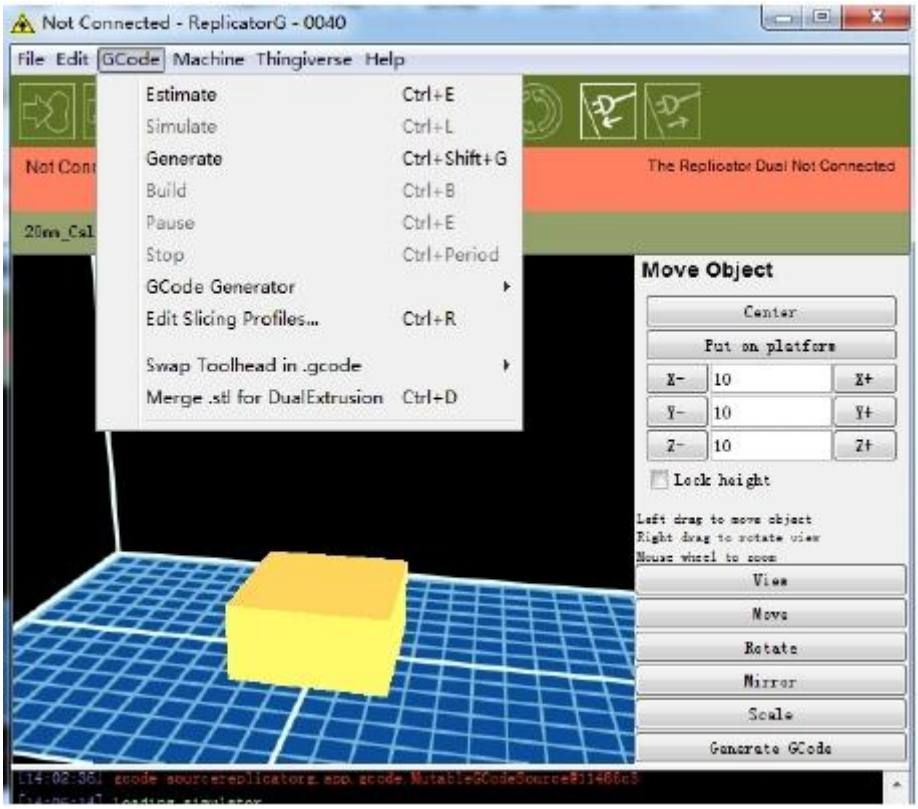
After modifying, click the File button to save this sample, and click button to build the sample.

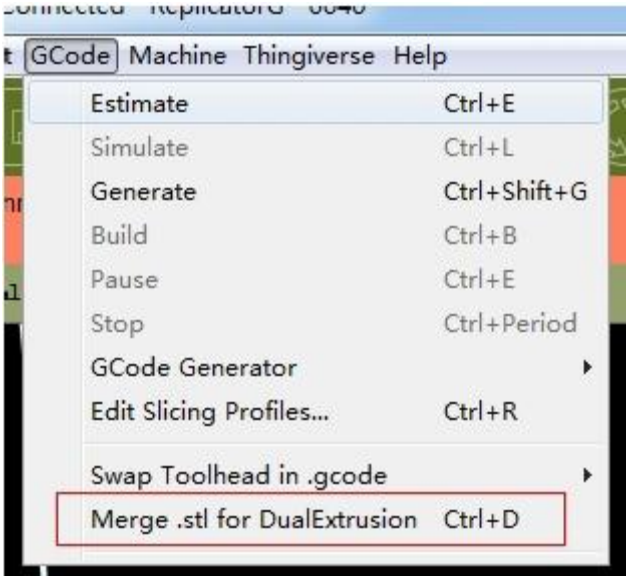
This concludes the initial print instructions.

Dual-Extruder print

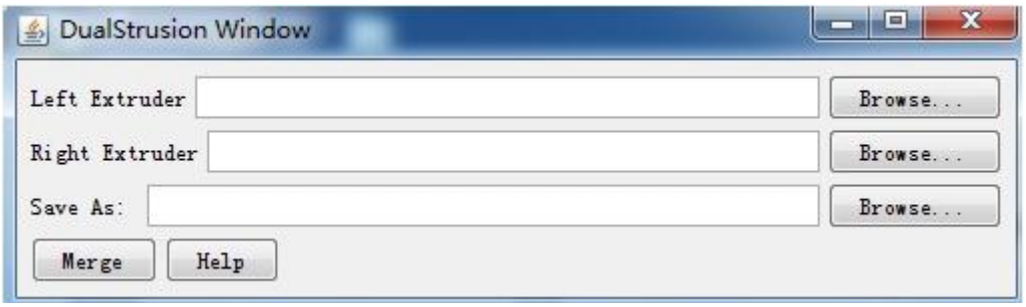
First try the two-color printing coming with our software

Open operating software replicator g, and then click the G code option in red box. Select the last column to Merge .stl for Dual Extrusion

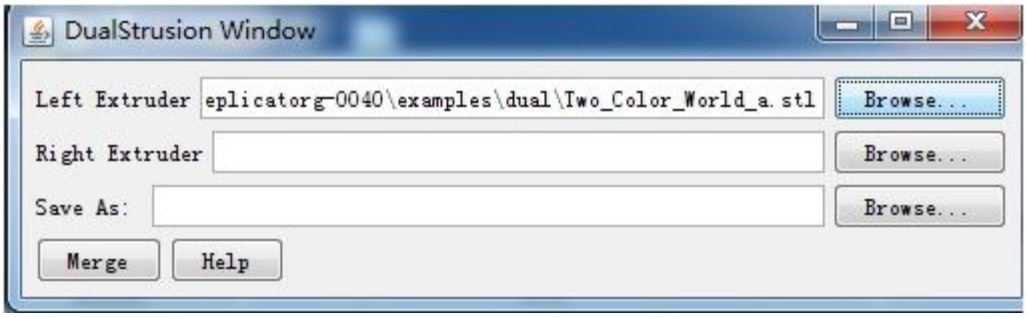




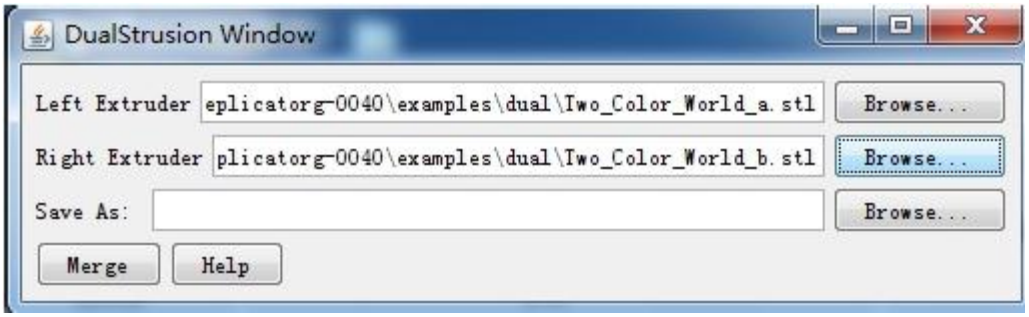
Click Merge .stl for Dual Extrusion, dialog box will pop up as below:



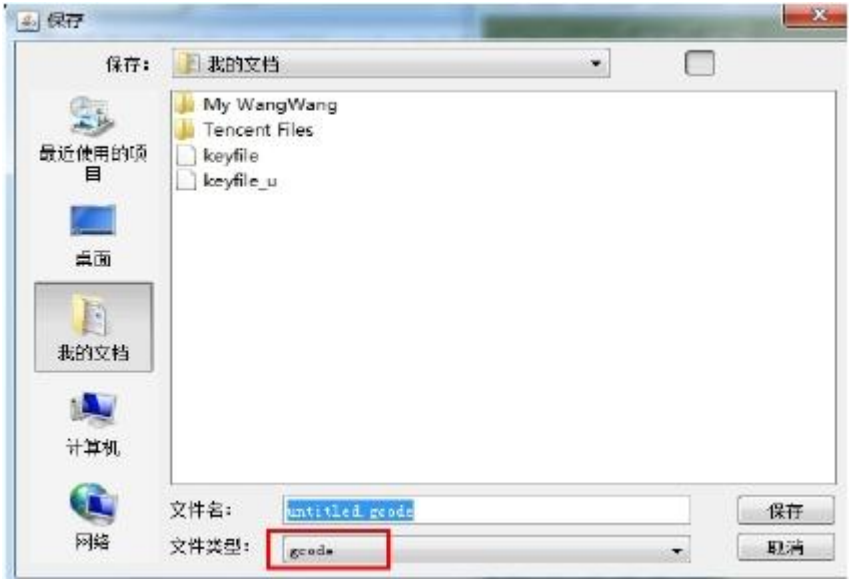
Click Browse for left Extruder, find folder finished version replicatorg-0040, click it, find examples and select double-head choice Two_color_World _a.stl, as shown:



Click Browse for right Extruder, find folder finished version replicatorg-0040, click it, find examples and select Dual-Extruder choice Two_color_World _b.stl, as shown in figure:



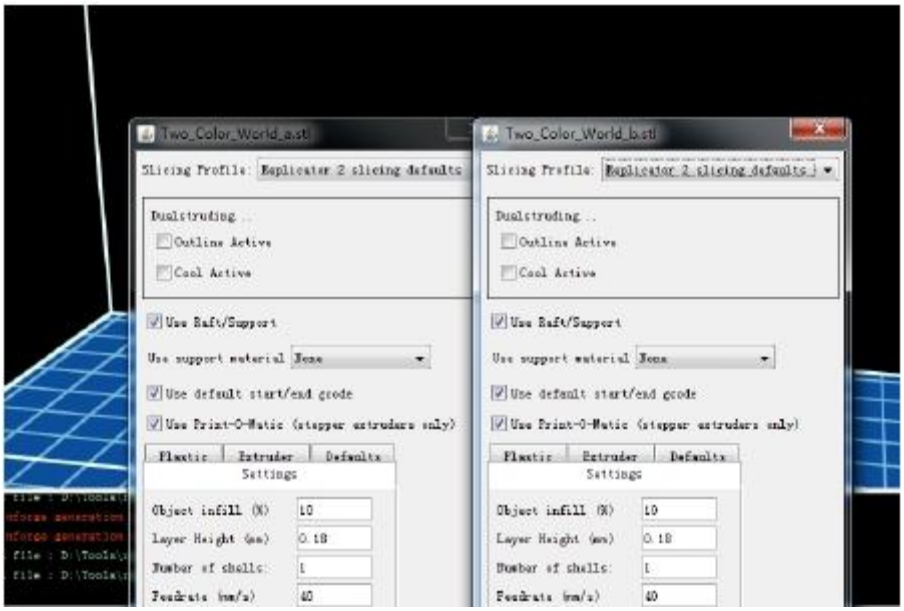
It is saved on the desktop with suffix name. G-code.



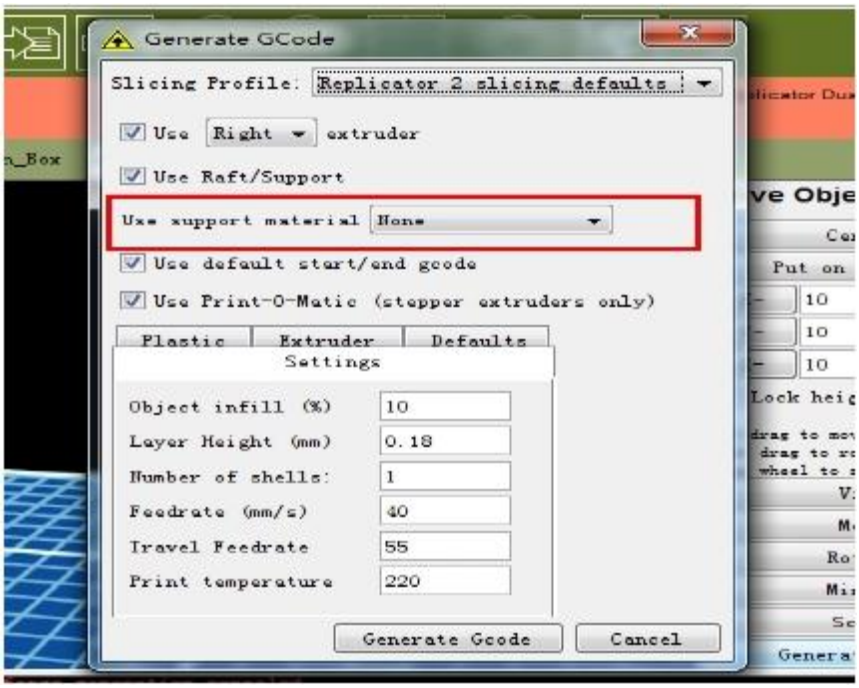
The suffix name must be. G-code

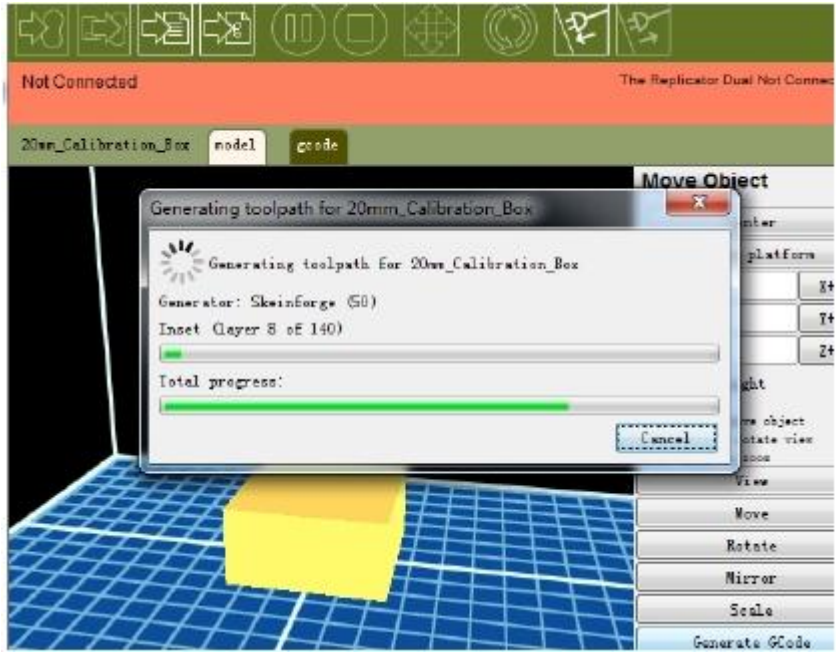
Save it to the desktop.

Then click Right G-code Generation button, two G-code Generation dialog boxes pop up. As shown below:



After G-code dialog boxes pop up, remove √ before use base / support in the front, and then generate G-code one by one to.





After G-code is generated, click the G-code in red box and modify the temperature of the bottom plate

Then modify M109 S110 in red box TO (set HBP temperature) M109 S115 (set HBP temperature)

As shown below:

After modification, save the file, click Build button for two-color printing.

Shown as below: