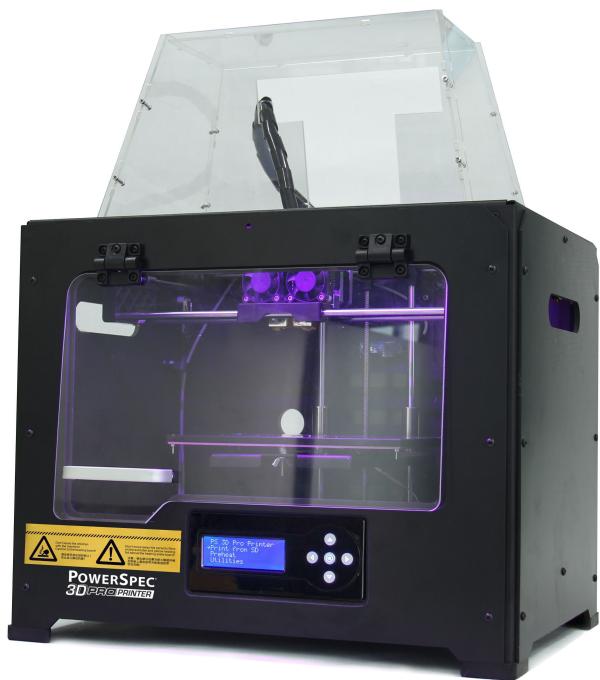


---

# PowerSpec Pro 3D Printer Start-up Guide



---

# Contents

**1 What's Included in the Box? 2**

**2 Un-boxing 2**

**3 Initial Hardware Installation 6**

**4 Software Installation 8**

**5 USB Connection & Temperature Setting 12**

**6 Filament 16**

**6.1 Install Filament 16**

**6.2 Feeding the Filament Using the LCD Screen 17**

**6.3 Withdrawing the Filament Using LCD Screen 17**

**6.4 Feeding the Filament Using Replicator G Control Panel 18**

**6.5 Setting Parameters 19**

**7 Initial Print 19**

**8 Dual Extruder Print 23**

**TROUBLESHOOTING 25**

---

## **PRECAUTIONS:**

**Please make sure to read this page carefully prior to setting up and operating the 3D Printer.**

-  **No** need to switch the power voltage as we use universal AC input/full range power supply.
-  The 3D Printer is very sensitive to static electricity, so please make sure you contact a grounded object before operating the machine.
-  Before repairing or making any alterations to the 3D Printer, it is essential that the machine is turned off and the power cord is unplugged.
-  The 3D Printer operates at very high temperature; allow the nozzle, the extruded plastic and heating plate to cool before touching.
-  Some plastic filaments may give off a little odor when heated. Because of this, the machine should always operate in a well-ventilated area.
-  Do not wear gloves when operating or repairing, as entanglement may occur and cause injury.
-  Do not leave the machine unattended when in operation.

# 1 What's Included in the Box?

Along with your 3D printer, the box also contains the following:

**In the two long boxes on the top of the package, you'll find:**

- Power cord x 1pc,
- USB A to B cable x 1pc,
- Sensor Line x 1pc,
- Filament guide tube x 2 pcs.
- Filament spool holder x 2 pcs.

**An accessory box on the top of the 3D Printer:**

- Dual extruder heads x 1pc,
- 4G SD card x 1 pc,
- Bolt, hex wrench and screws kit x 1 bag.

# 2 Un-boxing

The 3D Printer was carefully packaged at manufacturing facility. Please follow the unboxing steps laid out below.

## CAUTION

-  Handle the package and its contents with extra care; do not use any unnecessary force.
-  Do not remove the [thin yellow film](#) from the heating plate. It is heat resistant tape that improves the adhesion of the extruded plastic to the plate.
-  Do not remove the wrapping around the nozzle. It consists of a ceramic fiber fabric and heat resistant tape that helps to keep the nozzle at a constant temperature.

First, put the box on the floor in a clean and flat surface. Remove the top EPE foam piece and put the two long boxes aside. You will find **1x power cord, 1x USB A to B cable, 2x filament guide tubes, and 2x spool holders** in these two long boxes.





Then pull out the EPE foam packing that encloses the 3D Printer. Now you can see the top of the printer along with more boxes inside. The large box with the black wire is the **accessory box**. This contains the **dual extruder heads, SD card and other important components**. Don't remove the accessory box or its contents yet. [Note: Do not lift the box by the black cable! Doing so could cause damage.](#)

Next, take the 3D Printer out of the box by grasping the outer frame. [Be sure to grasp only the frame.](#) Gently lift and transfer the printer to your work surface.



Gently lay down the printer and open the bubble film as shown in next page.



Now, open the accessory box and remove the accessory sleeve. You will find the **dual extruder heads** in the protective packaging along with the black cable, carefully remove it and place it on your work surface. Remove the cardboard packing material and take the **accessory box** from the printer, set it aside for later.



The build platform should now be visible. It is an aluminum plate covered in a thin polyamide film. This is the surface that your objects will be printed on. **Note: Remember not remove the film.**

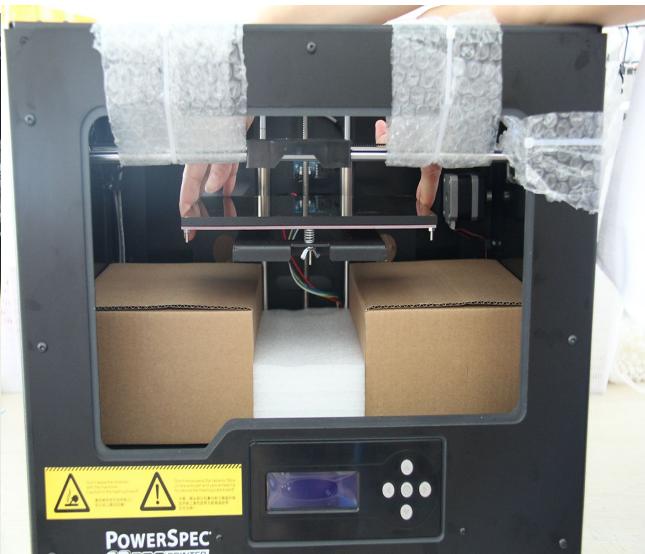
The next step is to raise the build platform; there are two ways to do this:

1. Turn the screw which is behind the rotating platform.
2. Grasp the printing platform with one hand on each side, raising it slowly and keeping it level.  
Stop once the platform is just shy of the bronze nozzle.

Now you should be able to see underneath the build platform. It's easy to remove the remaining packaging material.



To do this, remove the EPE foam in the front, and then in the middle and take out the two boxes. Finally cut down ties between the X axis and front metal frame. You have now finished unboxing! The next task is to setup the hardware.

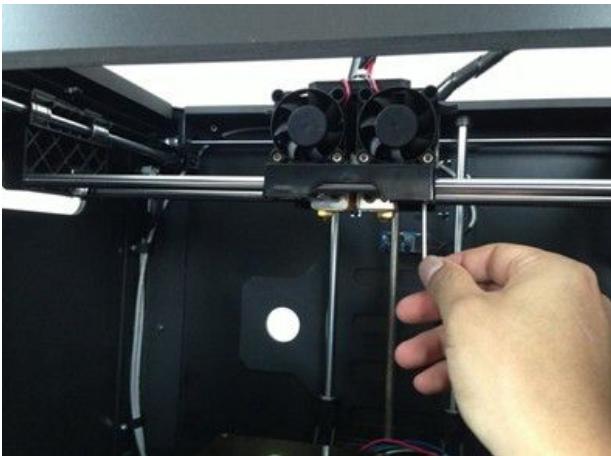




## 3 Initial Hardware Installation

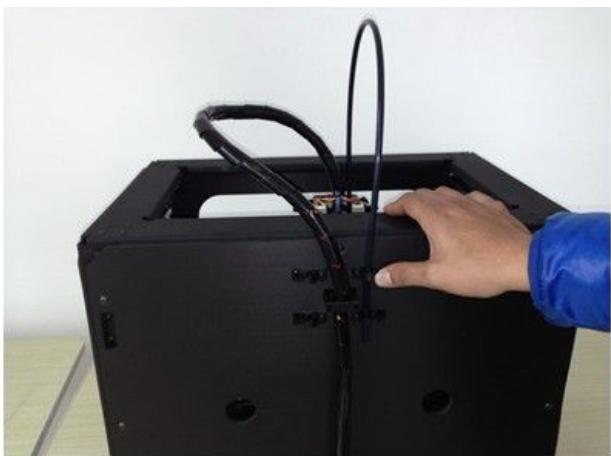
You need the two shortest silver screws from the kit found in the accessory box, and the appropriate hex wrench.

First, lower the build platform by using one of the methods described in the previous section. Holding the extruder by both sides, take it out of the accessory sleeve and position it on the extruder seat with the fan facing forward. Align the screw holes and fasten with the two short silver screws.



Next is the installation of the spool holder. Install one on each side.

The installation of the spool holder is very simple – just insert it into the circular opening and tighten the nut behind. Then install the [filament guide tube](#) to the empty spot on the extruder. Place one end of the guide tube into the hole. The hardware installation is almost complete.



---

Next, with the power switch in the **OFF** position, confirm that the power cord is plugged into 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 install the filament on the **spool holder** and screw in the nut. **Do not over tighten.**



Congratulations! You have completed the initial hardware installation! If you're ready to start printing, proceed to the next step: Software Installation.

## 4 Software Installation



### SPECIAL NOTES TO WIN 8 USERS BEFORE SOFTWARE INSTALLATION

Win 8 users please make sure that you have disabled the Driver Signature Enforcement setting before installing the ReplicatorG software. Below is a video link showing you how to accomplish this:

<http://www.youtube.com/watch?v=NM1MN8QZhnk>

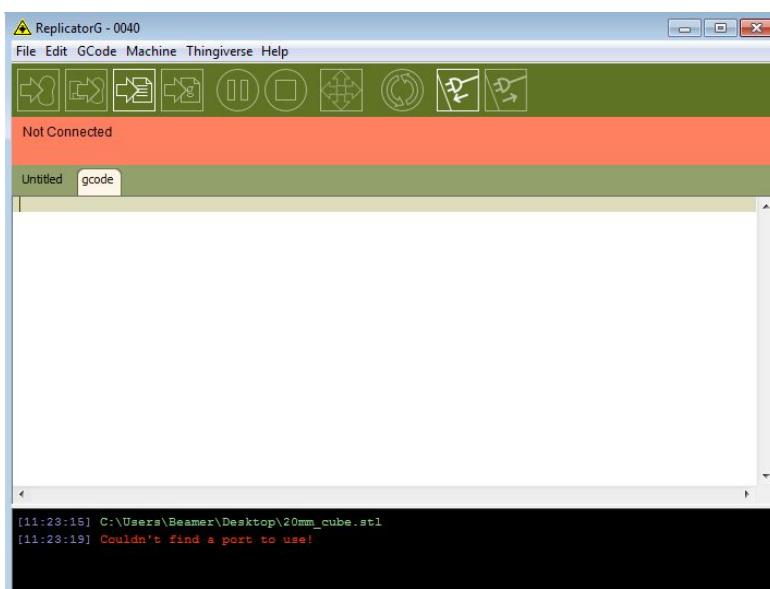
ReplicatorG0040 is the ideal software to use with the dual extruder 3D Printer. It can be downloaded from replicat official website by following the link below and click on Download.

<http://replicat.org/download>

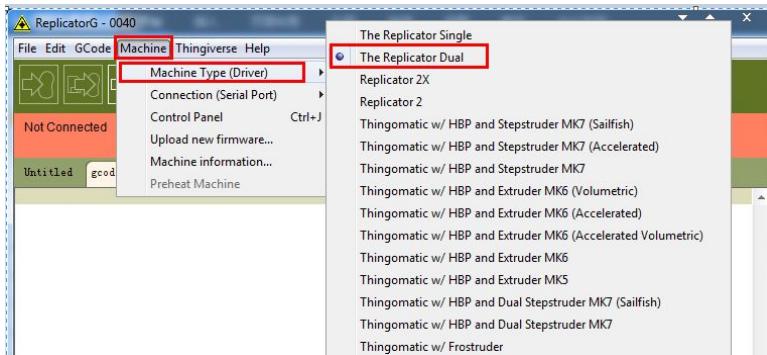
ReplicatorG0040 for Windows users can also be found in the folder of **information of 3D printer** on the SD card, which shipped together with the 3D Printer.

Steps for installing your software:

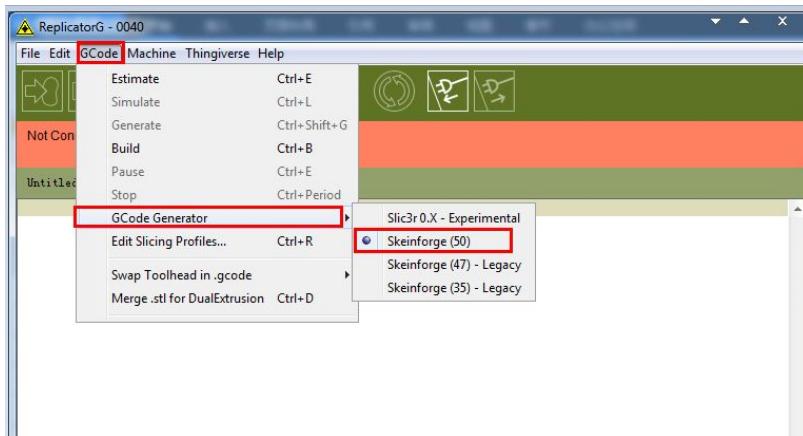
1. Download the appropriate version of ReplicatorG suitable for your system from <http://replicat.org/download>.
2. From the SD card shipped with your printer, browse to the “information of 3D printer” folder and run the Python installation file and the Python acceleration components.
3. After completing the Python component installation, click Replicator-0040-Installer to install the ReplicatorG software.
4. To run the ReplicatorG software, double click the shortcut on your desktop or in your start menu.



After opening ReplicatorG software, click on **Machine** to select your **Machine Type**. Choose **The Replicator Dual** for 3D Printer dual extruder machine.

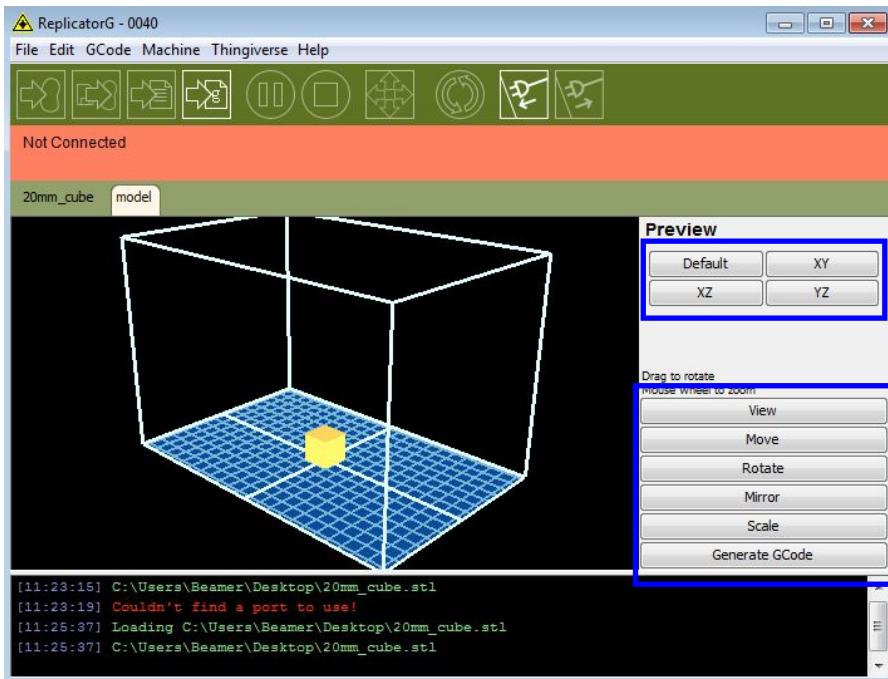


After selecting the appropriate machine type, click on **GCode** on the top navigation bar, and under **GCode Generator**, select **Skeinforge(50)**.



The following gives an introduction on how to import files (.STL) into the Replicator G software and then generate Gcode to print your creation.

Click **File > Open**, then browse and select the file (.STL) that you would like to print. Import the file by double-clicking on it. Then the drawing will appear on the Replicator G interface.

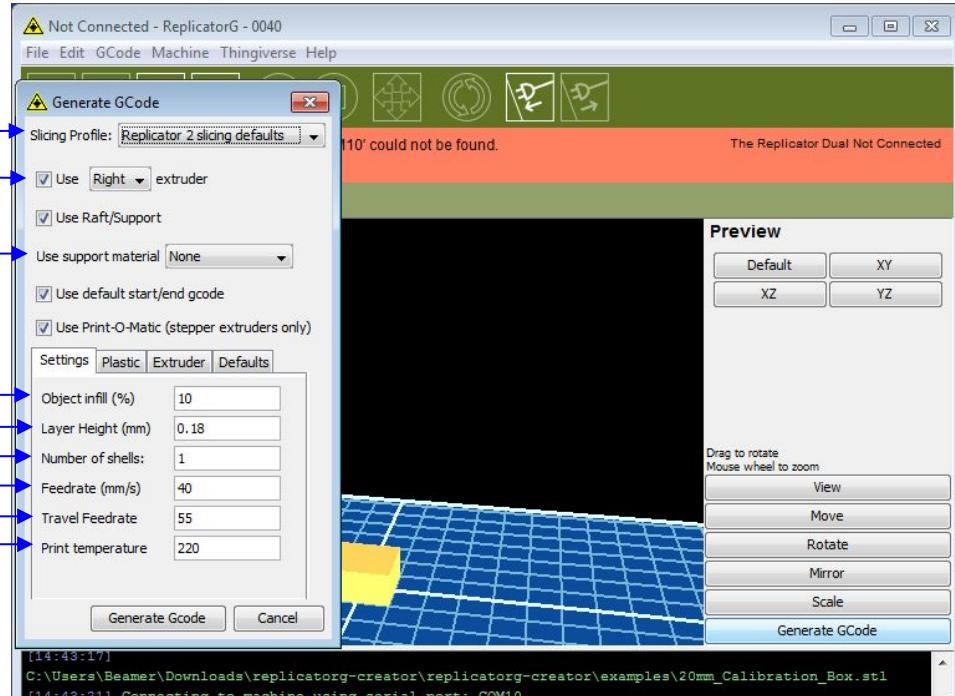


When the object is imported you may find that it is not on the virtual build platform or even on the screen, using the function keys indicated by the blue boxes in the illustration above, you can change the camera angle and reposition the object onto the center of the build platform. Once you've done this, the next step is to generate the Gcode. This is achieved by clicking on the button (**Generate GCode**) at the bottom of the panel.



### EXPLANATION OF GCODE SETTINGS

A new window will pop up, giving you several options on how the Gcode will be generated:

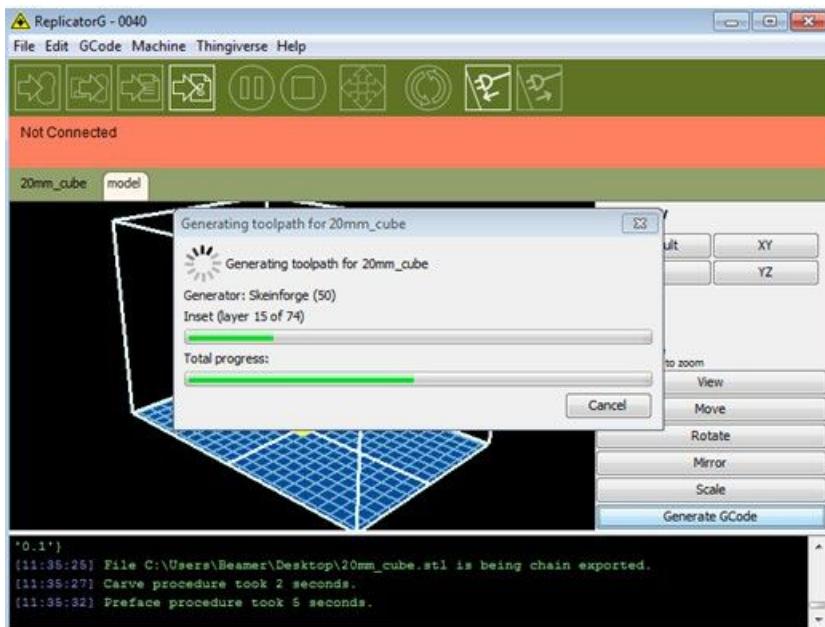


- A. Slicing Profile:** select **Replicator slicing defaults** for ABS printing, then select **Replicator 2 slicing defaults** for PLA printing.
- B.** This tells the printer which extruder to use for a dual-extruder head printer, either the **left** or **right** head can be selected.
- C.** If your sample will have any hanging surfaces, it is recommended to have support. Select **None** indicates that there is no

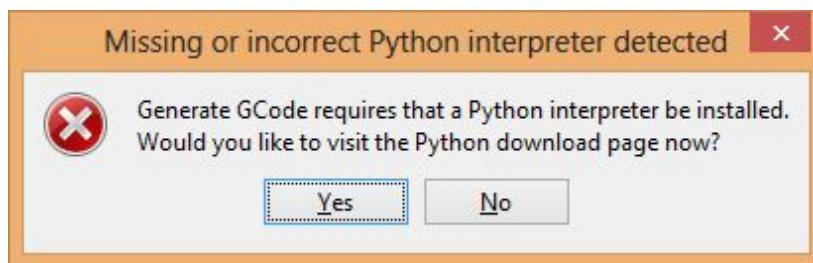
support. **Exterior** means surface support. **Full support** means all support.

- D. **Object infill:** where **100%** is a solid print, **0%** is a hollow object. The recommended setting is **10%**; this will save time and filament. Low infill also can reduce the corners lift up issue during ABS printing.
- E. **Layer Height:** this controls the vertical resolution of the print. The recommended thickness is **0.27mm**.
- F. **Number of shells:** this is the wall thickness; it's usually set at **1**.
- G. **Feedrate** is the speed at which the filament is fed into the extruder. This usually set between **30** and **70**. For **ABS printing**, **60** is recommended; for **PLA printing**, **100** is recommended.
- H. **Travel feedrate** is the speed at which the printer head moves over the base, it's usually set between **30** and **100**. For **ABS printing**, **80** is recommended; for **PLA printing**, **120** is recommended.
- I. **Print Temperature:** is the temperature to which the nozzle is heated. This varies between filament types. Set it to 220 for the default filament.

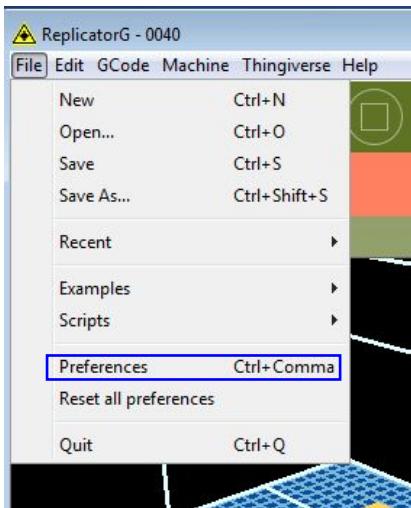
After you finished inputting in the settings, click **Generate Gcode** and the progress bar will appear.



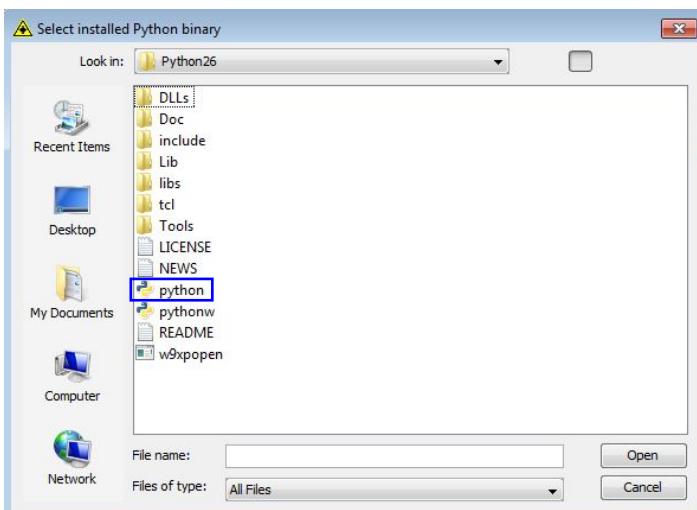
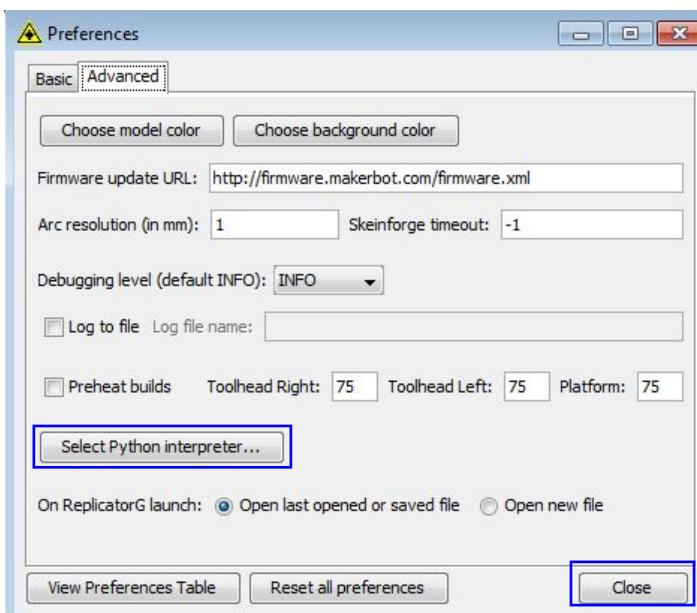
For users who did not choose the default installation path in the installation of Python, clicking the **Generate Gcode** button in Replicator G will result in dialogue box popping up alerting that the executable Python file cannot be found.



In order to solve this, click the **No** button to dismiss the dialogue box. Next, we need to configure the corresponding menu.



Click **File > Preferences**, then click **Select Python interpreter** under the **Advanced tab**. A window will pop up. Navigate to the Python installation directory, and select **python.exe** and click **Open**.



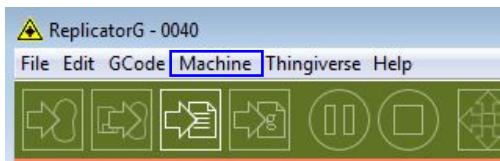
Last, click **Close** on the Preferences menu and you are done!

---

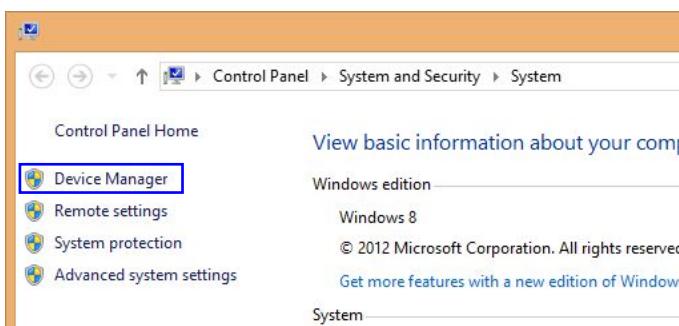
The machine will now work as normal when generating Gcode, and the Python error message will not pop up again. In the next section, we will start a preliminary test on the machine's connections, and heat the platform and extruder so that the extruder is ready for its first print.

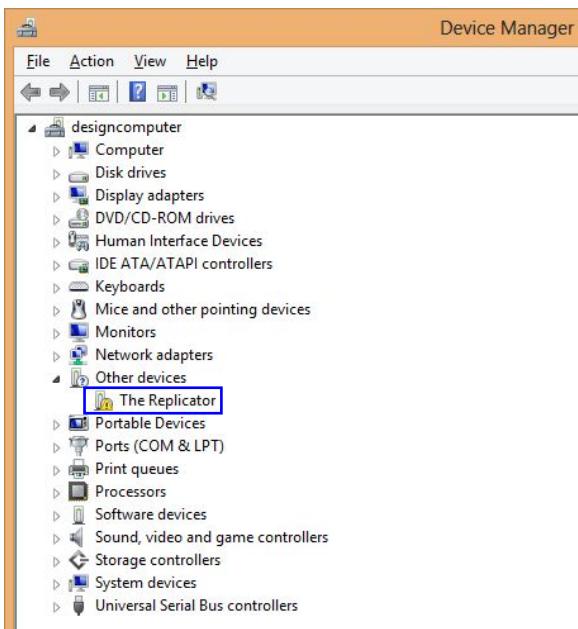
## 5 USB Connection & Temperature Setting

First, connect the machine and computer with the provided [USB A to B cable](#). The USB port on the machine is bound by the box in the below image. After connecting the cable, open the ReplicatorG software as we are going to link the computer and printer.

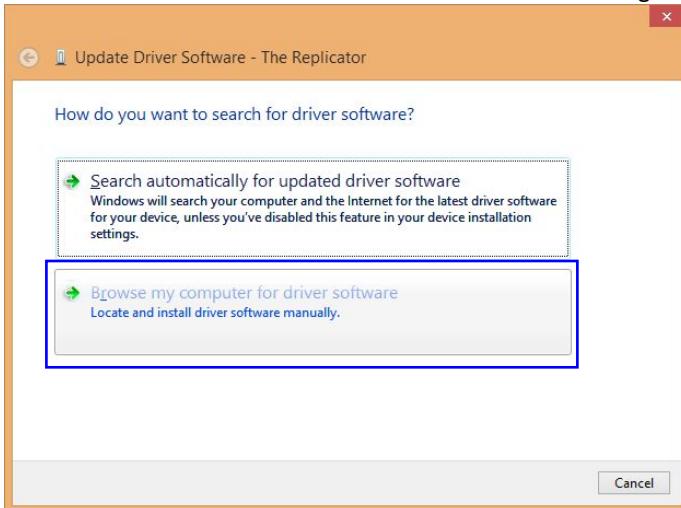


Inside the ReplicatorG software, click **Machine** > **Connection (serial port)** > **Rescan serial ports**, if no new ports appear then the software driver has not been installed. To install the driver manually, go to [My computer](#) and right click on **Properties**. The basic system parameters appear. Next select **Device Manager**.

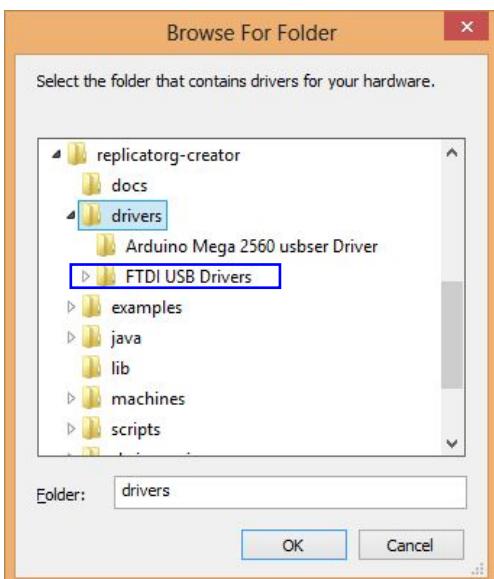




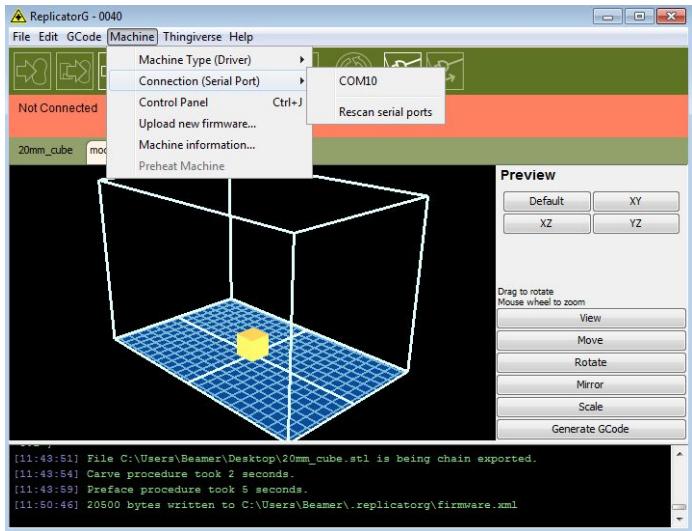
Locate the software driver shown in the blue box above. Right click and select [Update Driver Software](#).



Click [Browse my computer for driver software](#) to find the location of ReplicatorG0040 on your system.



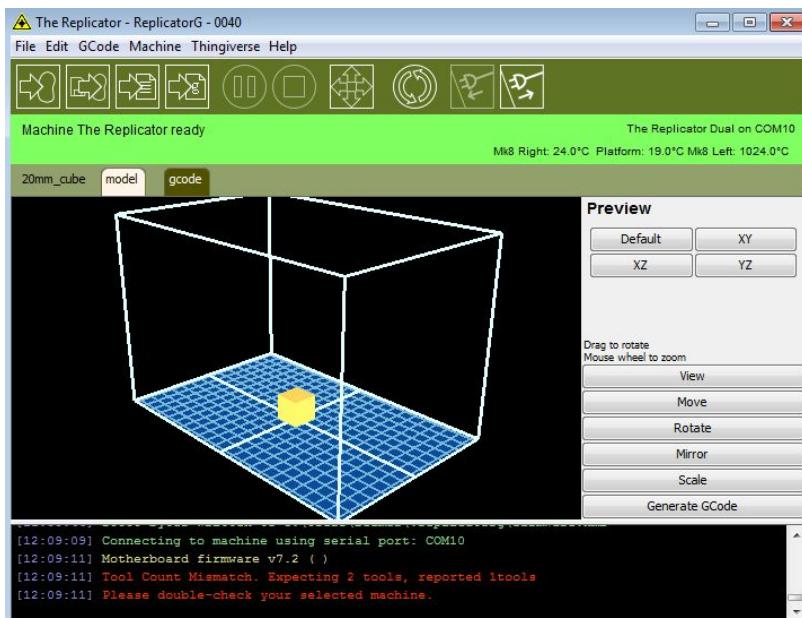
Click **FTDI USB Drivers** in the driver folder before confirmation and click **OK**. The drivers will then be installed. The next step is to link the printer. Rescan the serial ports and select the one that appears on your machine (**on our test machine the port was COM10, but different computers vary**).



Now we can link the machine to the computer. Click on the printer connect icon highlighted in blue :

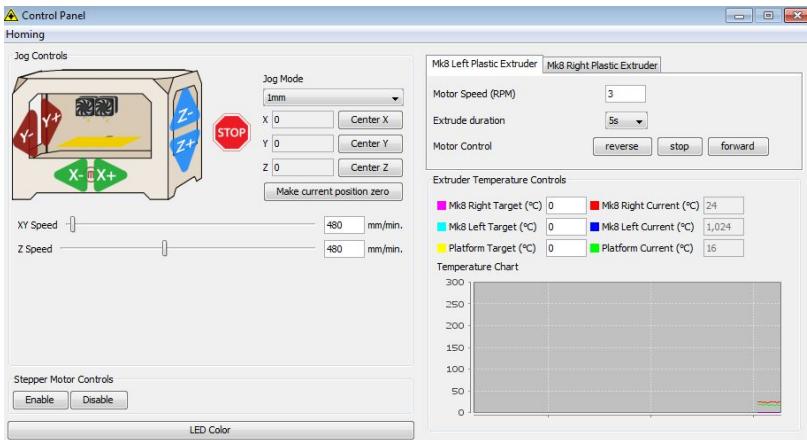


If the printer connect icon turns dim, as shown in the illustration below, the printer has successfully connected to the computer. Next we need to preheat the extruder and build platform.

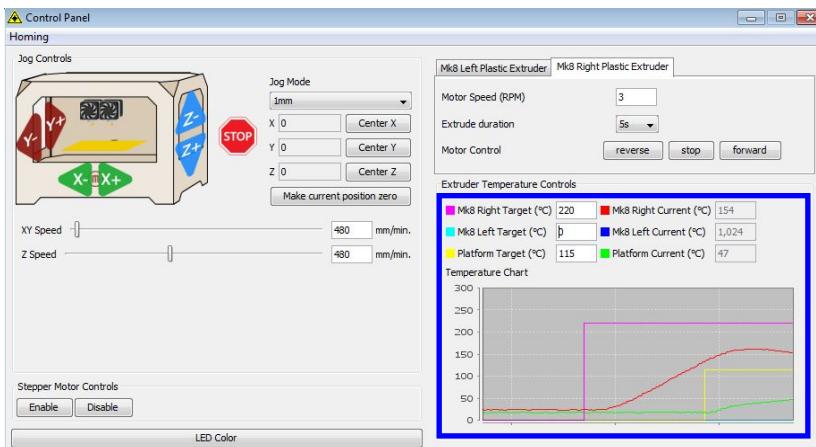


In the same set of menu, click on the cross-shaped icon and a new interface will appear.





Input the following target values: 220°C for the extruder and 115°C for the heating platform. After entering the values, the platform will start to warm up. When the extruder temperature reaches 50°C, the cooling fan will be activated and the current temperature value will display to the right as shown below.



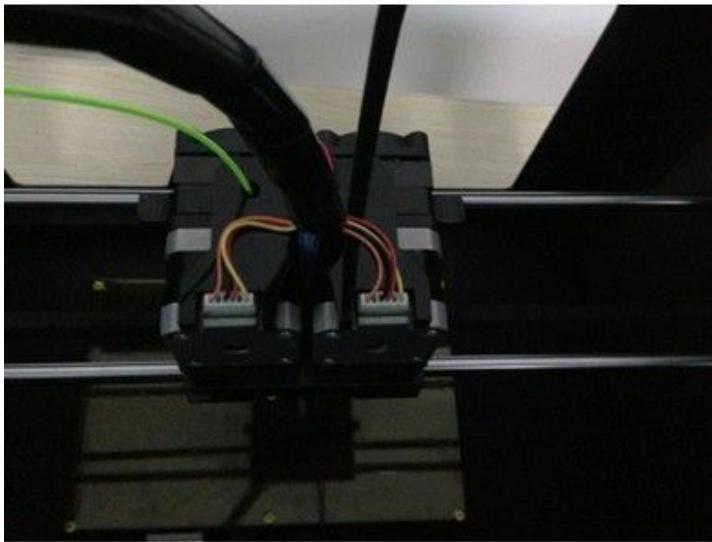
## 6 Filament

To make the process of feeding or withdrawing the filament easy, please follow the next few steps carefully:

After inserting the filament into the feeding hole, do not push it further until the extruder temperature reaches 200°C or more. Once the machine reaches this point, you will feel the filament being pulled into the extruder head.

### 6.1 Install Filament

First, remove the filament guide tube from the extruder head.



When you have removed the guide tube you can remove the filament that is inside the guide tube. To avoid any blockages during printing, please ensure that the 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:



There are two ways of feeding the filament into the extruder head, one is using the LCD screen on the printer itself and the other is by using the control panel on the Replicator G software.

## 6.2 Feeding the Filament Using the LCD Screen

1. Turn on the 3D Printer; the display will indicate:
  - ▶ Build from SD
  - Preheat
  - Utilities
2. Using the directional arrows to the right of the screen, press the page down key to scroll to the next page. The display will show:
  - Preheat
  - ▶ Utilities
  - Info and Settings
3. Select **Utilities**; press the OK key of the keypad. The display will show:
  - Monitor Mode
  - ▶ Change filament
  - Level Build Plate
  - Home Axes

---

#### Feed operation with LCD screen

4. Select **Change Filament**; press the **OK key** of the keypad. The display will show:
  - ▶ Load Right
  - Unload Right
  - Load Left
  - Unload Left
5. Select the appropriate side of the extruder you wish to load(Left or Right for models with dual extruders). Press the **OK key** again on the keypad. The display will indicate: "I'm heating up my extruder!" At this time, the temperature of the right nozzle is being heated up. When the temperature of the nozzle reaches its target temperature, pressing the **OK key** on the keypad, and the nozzle should start extruding material. If not, keep pressing the **OK key** until it does.

### 6.3 Withdrawing the Filament Using LCD Screen

#### CAUTION

- ⚠ If you just finished your printing or already extruded the filament and want to withdraw the filament, at this moment, the extruder is still over 200°C, first push the filament in a little further, and then directly pull it out.
- ⚠ If you want to change another color filament, first you need to withdraw filament and then load in. Please do as following shown to avoid filament jam.

- 1) Turn on the 3D Printer, the display will indicate:

Build from SD  
▶ Preheat  
Utilities

- 2) Using the directional arrows to the right of the screen, press **page down key** to scroll to the next page and select Preheat, Press **OK key** on the keypad, then you will see:

▶ Start Preheat  
Right Tool OFF  
Left Tool OFF  
Platform OFF

- 3) Press **page down key** to select **Left Tool (or Right Tool)**, click **OK key**, you will see:

▶ Start Preheat  
Right Tool ON  
Left Tool ON  
Platform OFF

- 4) Press **page up key** back to Start Preheat, and Press **OK key**, you will see:

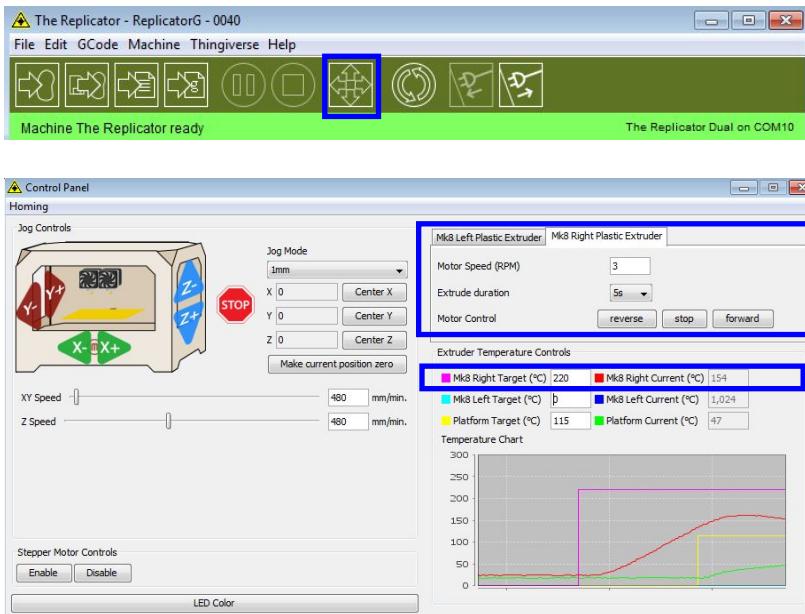
Heating:  
R Extruder: 033/230C  
L Extruder: 033/230C  
Platform: 024

This means the left extruder is heating up, when it reaches 220°C. First **push in the filament a little bit until you see filament come out of the nozzle**, then **pull it out quickly**. This will ensure you have withdrawn the filament inside the nozzle successfully.

### 6.4 Feeding the Filament Using Replicator G Control Panel

The following can only be performed while the printer is connected to the computer.

Open the Replicator G software, click the icon in the red box, this is the control panel icon. The dialogue box shown below will then pop up.



To heat the right extruder and feed material, click the right extruder on the upper right corner of the control panel and manually modify the temperature in the “Right Target” setting for the extruder temperature control changing the temperature to 220°C. A red line will be plotted in the temperature map. When the actual (Current) temperature reaches 220°C you can pull the filament out.

## 6.5 Setting Parameters

Before generating Gcode, specific parameters can be set up to customize various aspects of the print. An explanation to each of the settings can be found in the [EXPLANATION OF GCODE SETTINGS](#).

# 7 Initial Print

## CAUTION

**⚠️** 3D Printer has been leveled perfectly before leaving factory. However, it will get unleveled during long distance shipping, please do level the build plate again before starting a print. See below in instruction.

**⚠️** Please firstly heat the build plate to 50°C (PLA) or 110°C (ABS) to make this printing start quickly. Please refer to 6.3(Page 18).

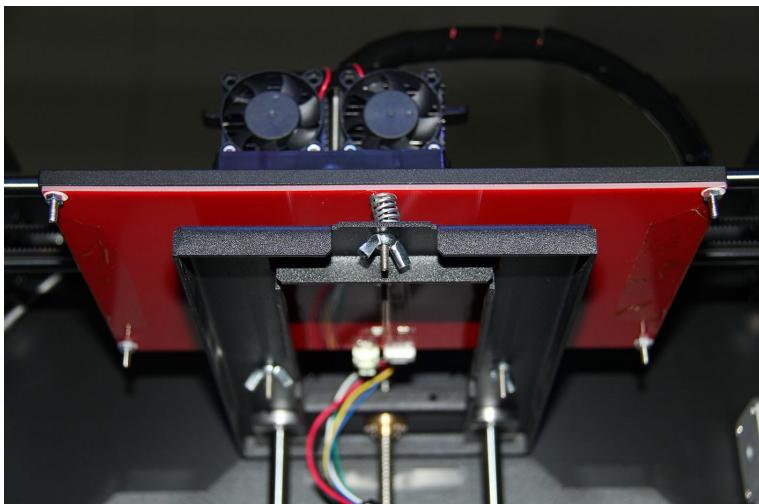
First, make sure that you've completed all the steps in the [Un-boxing and Hardware Setup section](#).

1. Your dual extruder heads should be bolted in place,
2. Your filament guide tubes is connected,
3. Your filament is mounted on the spool holders.

Once all these have been completed, you can plug in your power supply. If everything is ready, then turn on the power switch on the back of your 3D Printer.

To level the build plate, you must adjust the three knobs under the build plate.

1. Tighten the knobs(turning them to the right) moves the build plate away from the extruder nozzle.
2. Loosening the knobs(turning them to the left)moves the build plate closer to the extruder nozzle.



If your machine has factory firmware, Go to [Utilities>Level Build Plate](#) , Leveling Your Platform. **Pay close attention: leveling your platform is very important to print quality!** Here's the first screen.

Find the 4 knobs on the bottom of the Platform and tighten four or five turns.

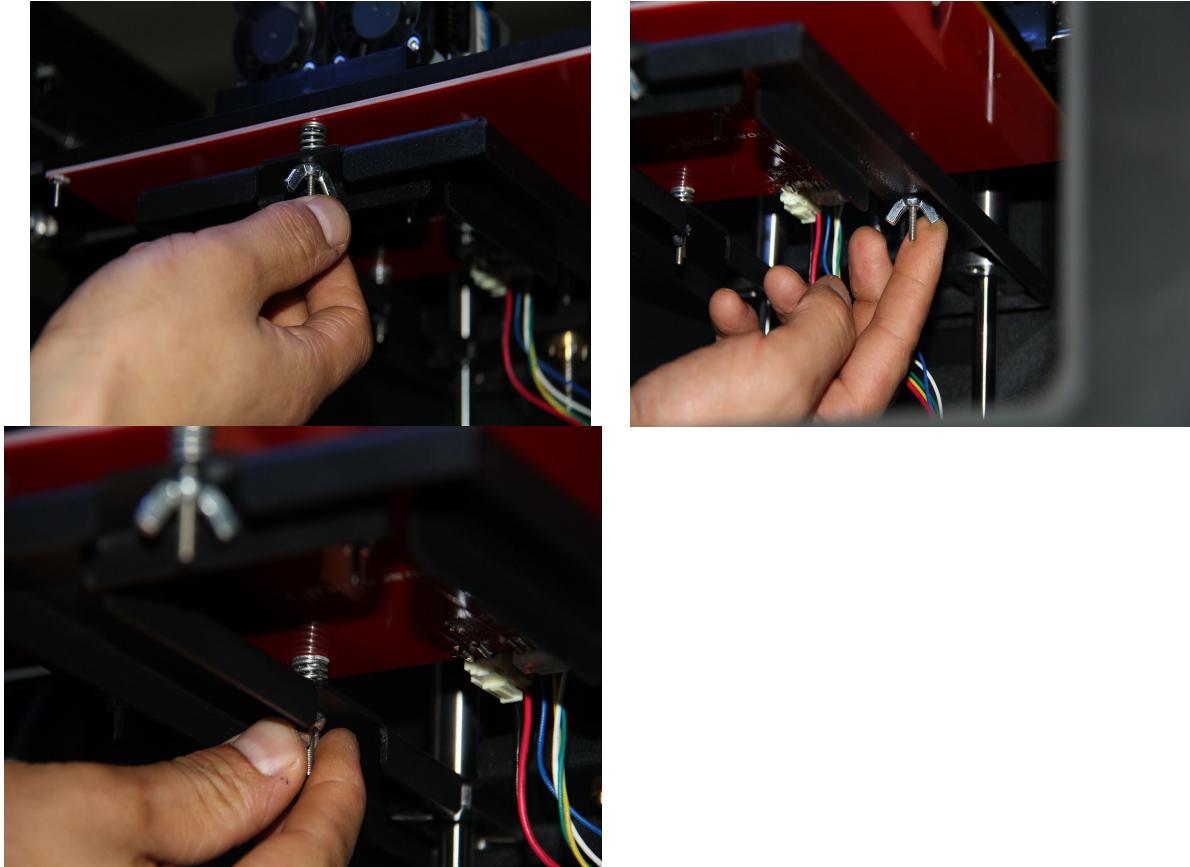
Actually you will find the printer only has three knobs on the bottom of the platform, just inside the plastic arms that hold it up. This doesn't affect leveling operation, please ignore the quantity of knobs and process it. The printer has been leveled perfectly before leaving factory. However, it will get unleveled during long distance shipping. Please do level the build plate again before starting a print.

[If your machine loaded with sailfish firmware, need you to adjust the build plate manually.](#)

Now, go to [Utilities > Level Build Plate](#), you will see

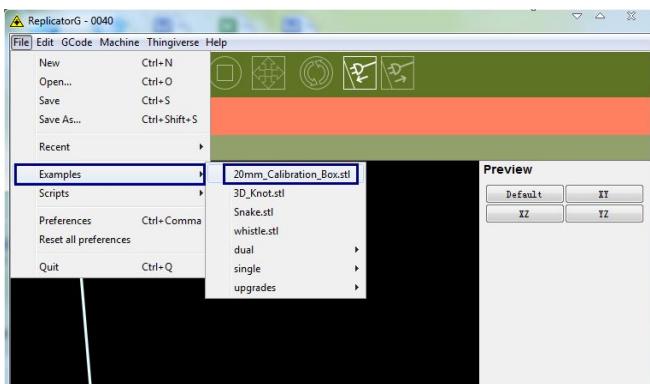


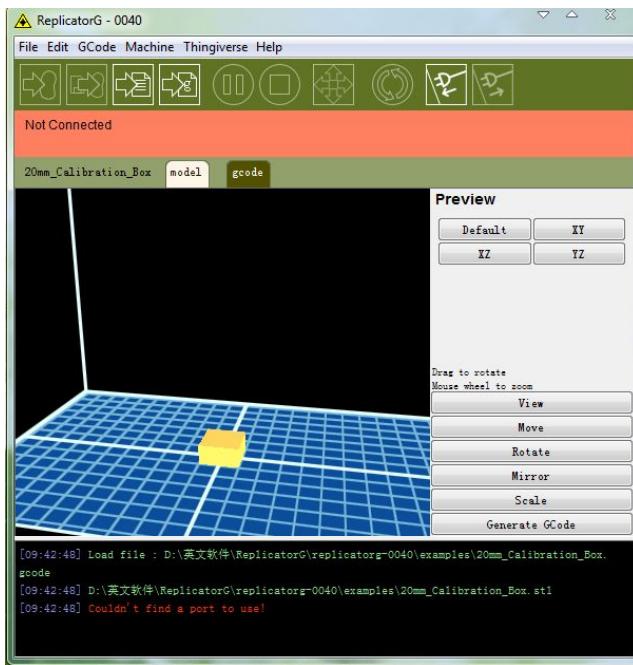
Next, Press **OK key** to level build plate, you will see the extruder start to move to the build plate. Now, **manually move** the extruder to four corners and center to adjust the build plate. Use a sticky note, sticky side up, to test the platform height one last time. If the note can back and forth freely underneath the nozzle, your platform is too low; loosen the knobs to make it closer. If it can slide around a little bit with some resistance, you're ready to go.



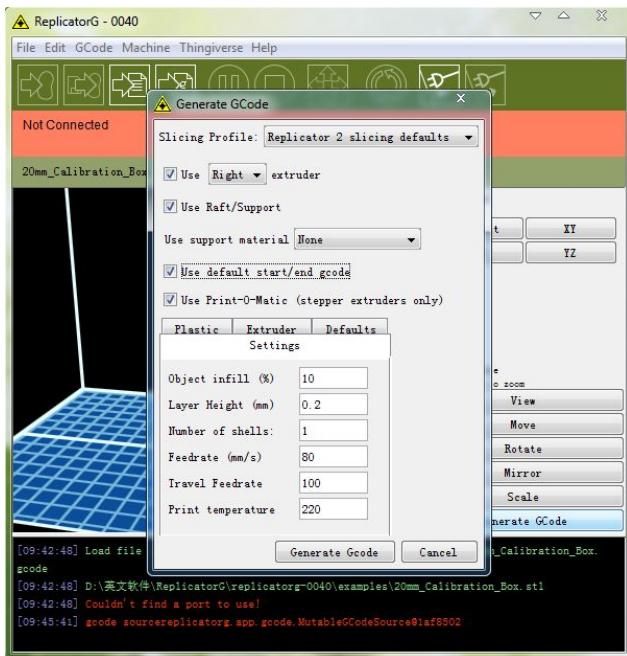
If your platform is too low, your prints might not stick to the surface, and if it's too high, the nozzles could tear the Kapton tape on the platform surface. If you're having trouble, or if you just need to level your platform again, you can always get back to the leveling script in the menu by going to [Utilities > Level Build Plate](#).

Now, let's start the [initial print](#). Click [File > Examples](#) and then select the [20mm\\_Calibration\\_Box.stl](#). The preview interface will then appear along with a virtual impression of the 20mm cube on the virtual printer bed. On the preview interface click [Move > Center](#) and [Put on platform](#) so that the sample will be printed on the center of the build plate.

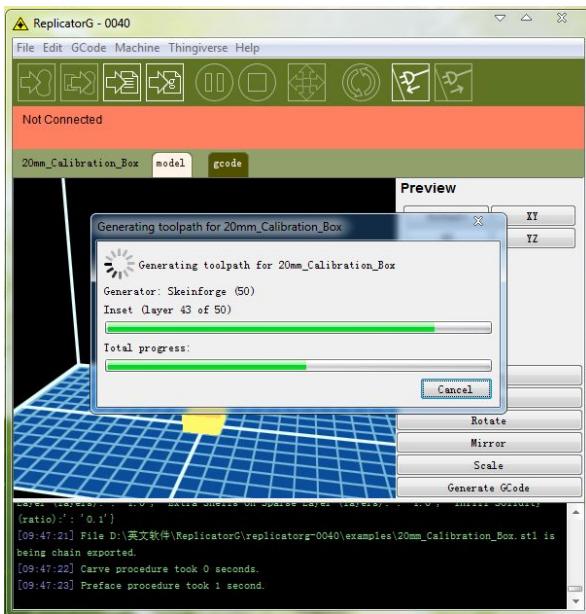




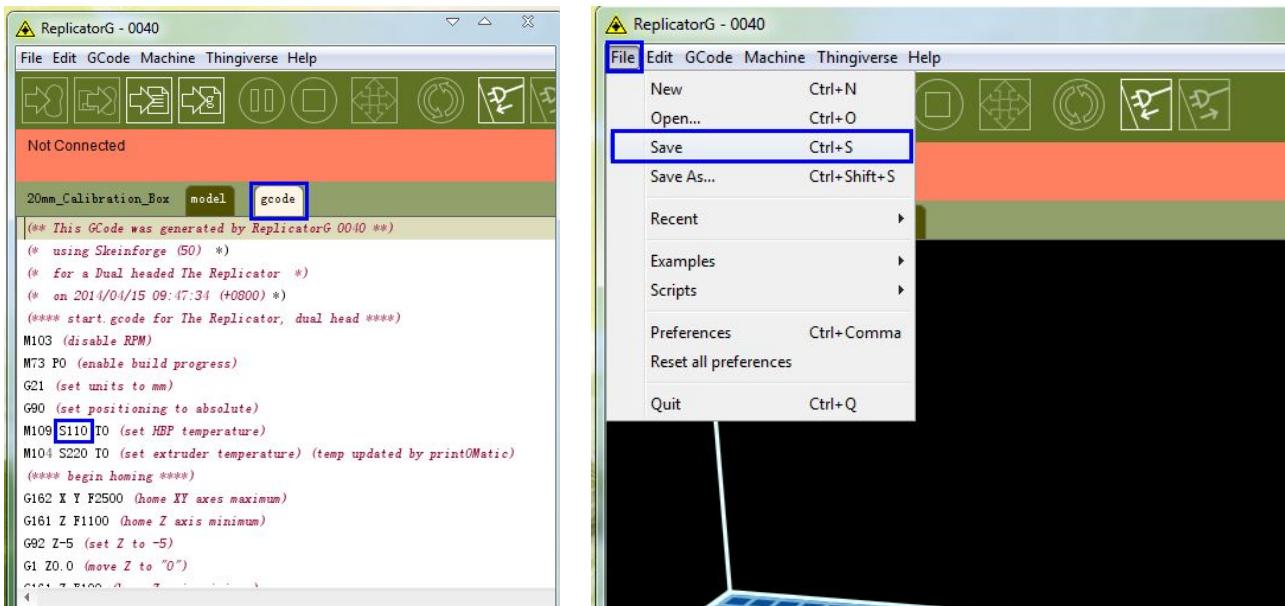
Next click **Generate GCode**, a dialogue box will pop up, after inputting your chosen parameters the Gcode will be generated. Please refer to [EXPLANATION OF GCODE SETTINGS](#).



A loading bar will then appear showing the progress of the Gcode generation.



After the Gcode is generated, the temperature of the bottom plate needs to be changed if you print PLA filament. See below for the required modification. First, select Gcode as shown highlighted in the illustration below and change **M109 S110** to **M109 S50** to set the HBP temperature at 50°C. Then click **File > Save** to save this modification. If you print ABS filament, then no HBP temperature modification is needed.

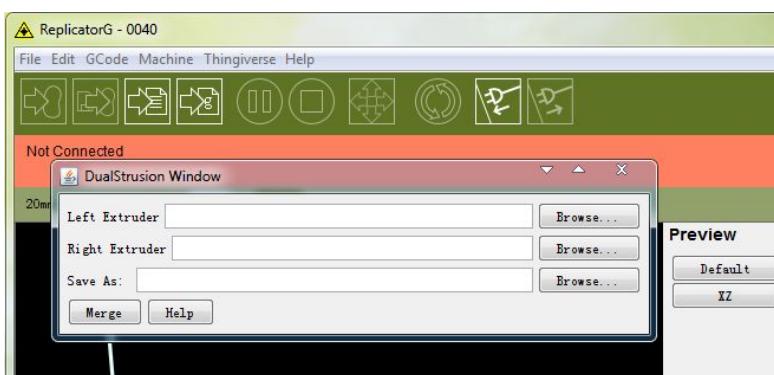
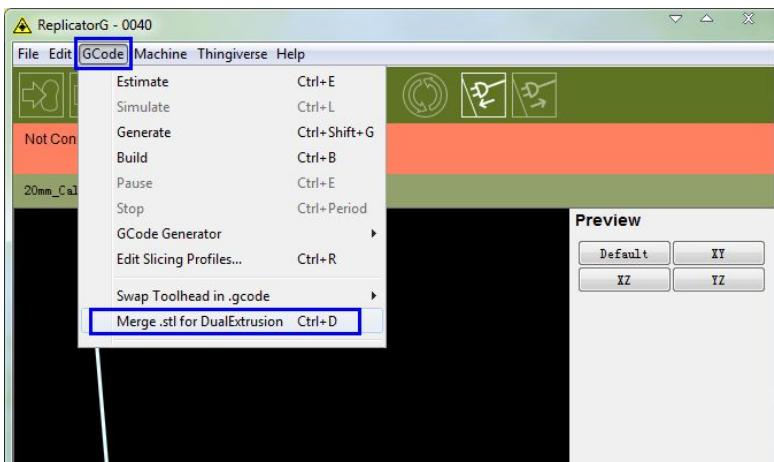


If you have connected Replicator G 3D Printer printer, click the left most button on the green bar near the top of the screen. The Gcode has now been sent to the printer and the object will start printing shortly. If not, please refer to [USB Connection & Temperature Setting](#) to connect.

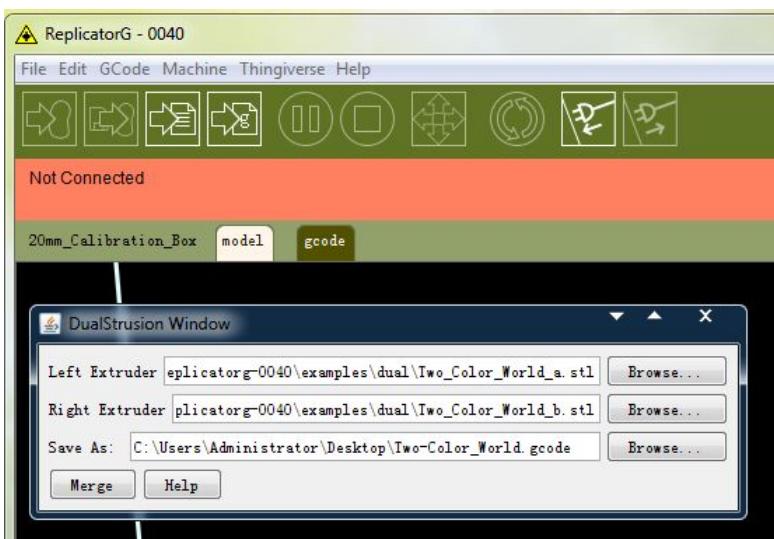


## 8 Dual Extruder Print

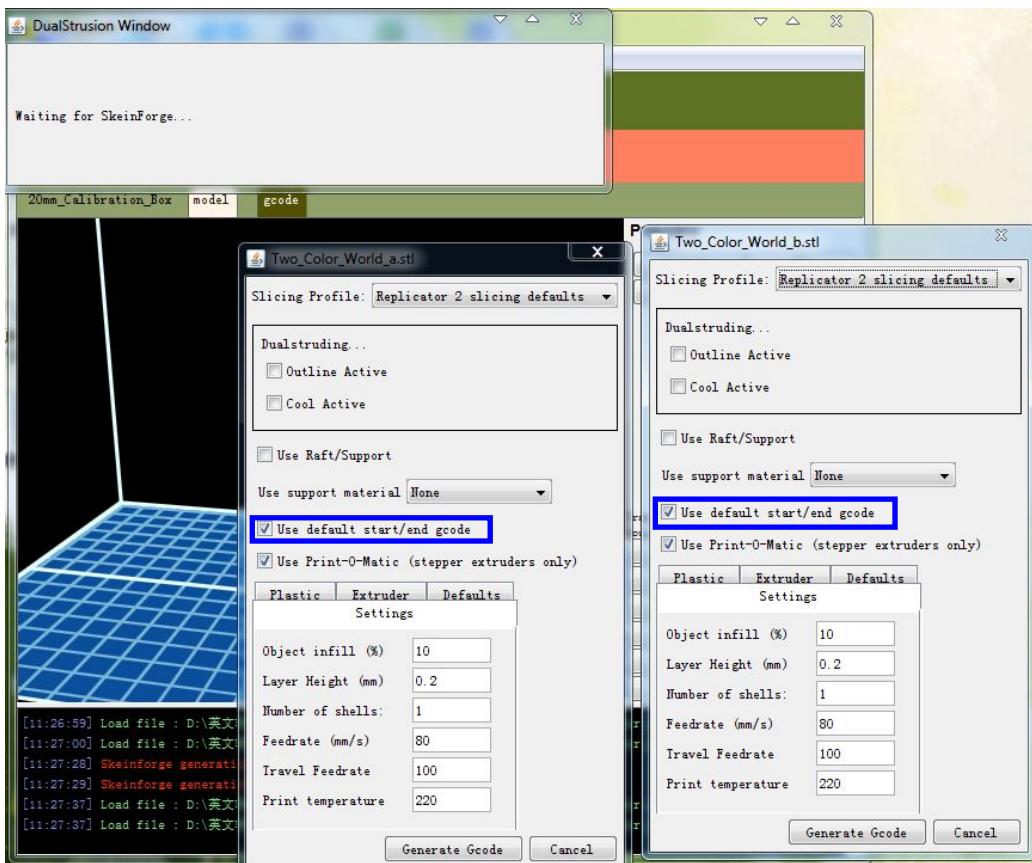
Open ReplicatorG and select **Gcode > Merge .stl for Dual Extrusion**. You will see following:



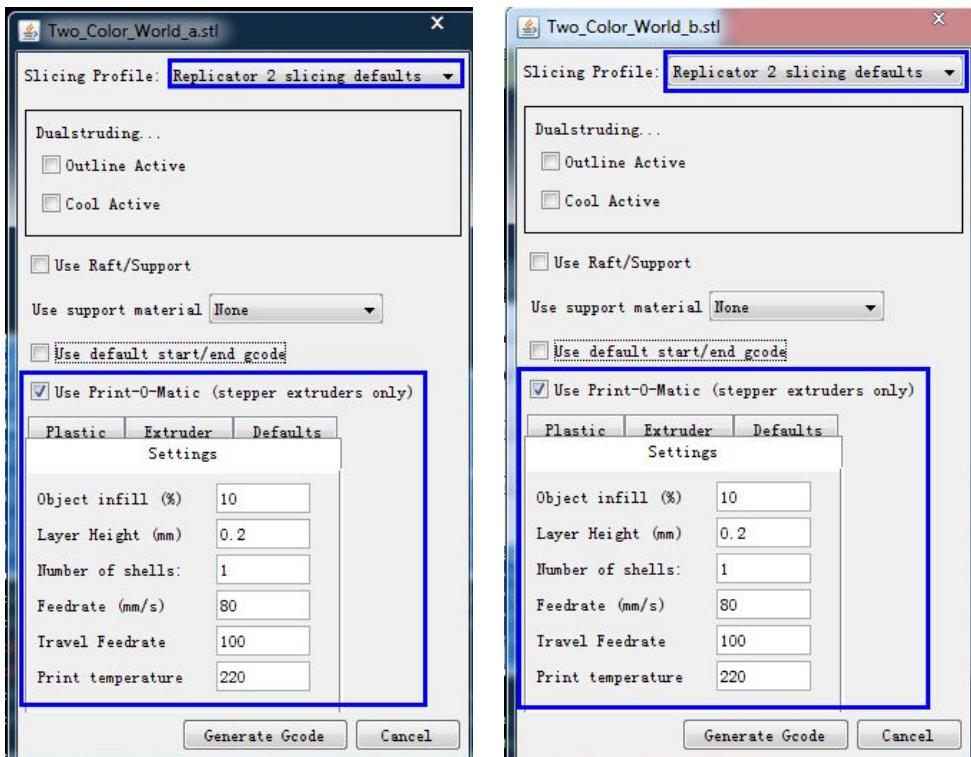
Click on the first **Browse** button to locate and select the file for left extruder; and then click on the second **Browse** button for right extruder. For example, you can find the [examples](#) folder in its installation path. Open the folder [examples](#) select [Two\\_color\\_world\\_a.stl](#) for left extruder and repeat the process for the right extruder as shown below:



Now you need to save it to the desktop with the suffix format [.gcode](#). Click **Merge**, two dialogue boxes will then pop up.



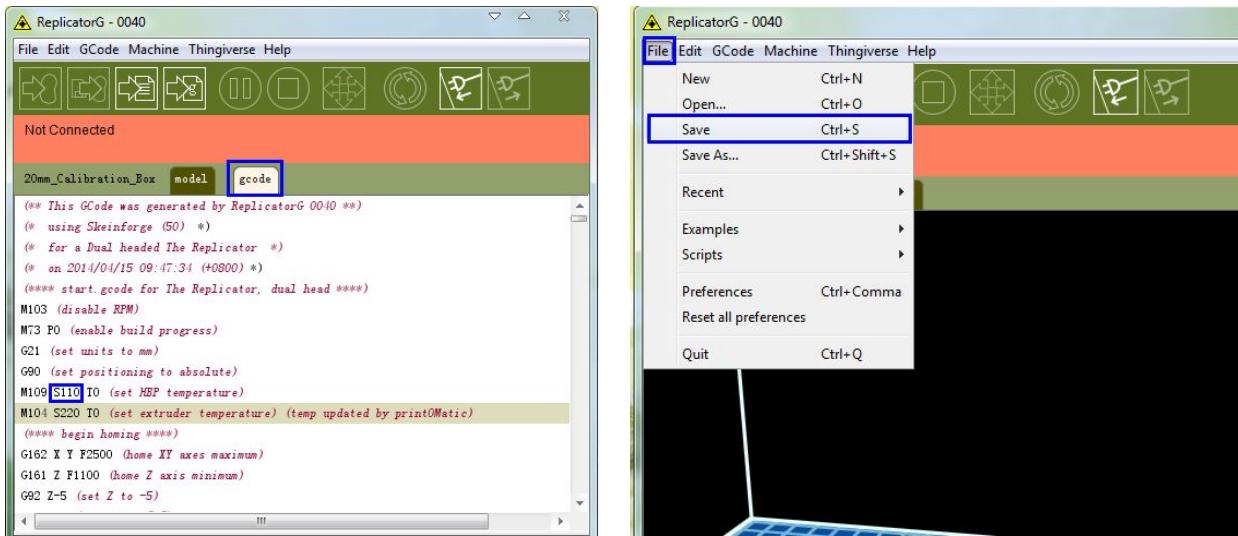
In both boxes you need to remove the tick in front of the 'use raft/support' option. If you want to print dual colors with PLA filament, then perform the following settings:



If you want to print dual colors with **ABS filament**, select **Replicator slicing defaults** under the **Slicing Profile** column, and lower the **Feedrate** to **60** and **Travel Feedrate** to **80**.

Click the **Generate Gcode** option for both boxes. After the Gcode is generated, click on the **gcode** tab as highlighted

below and change HBP temperature to **50** if you are printing with PLA and save this modification by going to [File > Save](#).



Click on the following button to export .x3g file for SD card print.

