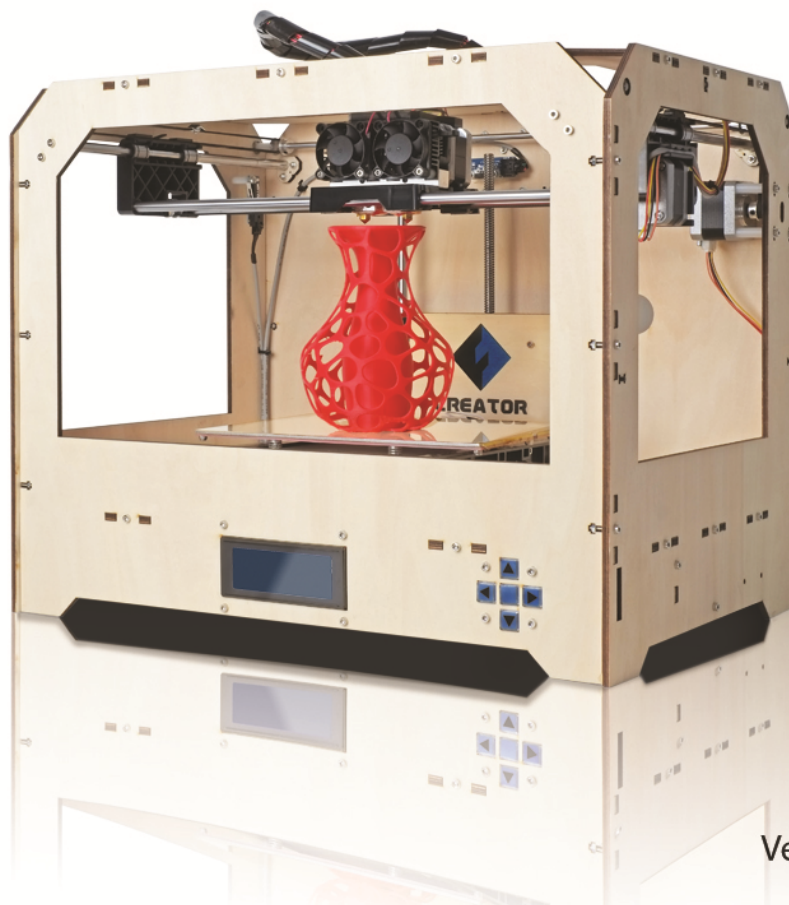




**FLASHFORGE USA™**

[www.flashforge-usa.com](http://www.flashforge-usa.com)

# Creator Startup Guide



Version 2.0 (January 2014)

---

## Contents

4	<b>Section 1: What's Included in the Box?</b>
5	<b>Section 2: Unboxing</b>
8	<b>Section 3: Initial Hardware Installation</b>
11	<b>Section 4: Software Installation</b>
17	<b>Section 5: USB Connection &amp; Temperature Setting</b>
22	<b>Section 6: Filament</b>
22	Installing the Filament
23	Feeding the Filament Using LCM Screen
24	Withdrawing the Filament Using LCM Screen
25	Setting the Parameter
26	<b>Initial Print</b>
28	<b>Dual Extruder Print</b>

**WARNING:**

Prior to powering on the Creator, make sure the power supply switch is set to 115v if you are located in the United States. For other countries, please refer to your country standards. **Failure to do so will damage the motherboard.** For instructions on how to do this, please refer to page 8.

**Precautions:**

Please make sure to read this page carefully prior to setting up and operating the Creator.



The Creator 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 Creator, it is essential that the machine is turned off, and the power cord is unplugged.



The Creator operates at a very high temperature; allow the nozzle, extruded plastic and heating plate to cool before touching.



Some plastic filaments may give off a slight 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 Creator 3D printer, the box also contains the following.

**An accessory box on top of the Creator:**

- Single or dual extruder heads,
- 1 or 2 filament holders,
- SD Card,
- Bolt tool plate, hex wrench toolbox.

**Under the Creator's build platform, there are 2 filaments:**

- 2.2 pounds (1 kg) ABS filament,
- 2.2 pounds (1 kg) PLA filament.

**Under the Creator unit, you'll find:**

- Power supply,
- USB A to B cable,
- 2 filament guide tubes

---

## 2 Unboxing

The Creator is carefully packaged at the FlashForge 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, place the box on the floor in a clean and flat surface. Remove the top carton and pull out the cardboard packing that encloses the Creator.



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 **extruder(s), SD card, and other important components**. Do not remove the accessory box and its contents yet. **Note:** Do not lift the box by the black cable. Doing so could cause damage.

Next, take the Creator 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, as shown on the next page.



With the Creator removed, you will find the **power supply and cable, USB A to B cable, and a filament guide tube**. Remove them from the box and set them aside. Now, open the accessory box and remove the accessory sleeve.



You will find the **extruder** in the protective packaging along 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 to remove this film.**

The next step is to raise the build platform. There are two ways to do this:

1. Turn the screw that 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.



Next, you should be able to see underneath the build platform. Here, you'll find the filament (either one or two rolls depending on whether your Creator has single or dual extruders). It's easiest to remove the filament by setting aside the remaining packaging material.

To do this, remove the long box in the front, then the small box on the right, and finally the two wire trays. You have now finished unboxing! The next task is to setup the hardware.



---

## 3 Initial Hardware Installation



### WARNING:

Prior to powering on the Creator, make sure the power supply switch is set to 115v if you are located in the United States. For other countries, please refer to your country standards. **Failure to do so will damage the motherboard.**

Prior to installation, please make sure that the red power supply switch at the bottom of the Creator is set to 115V (for United States).

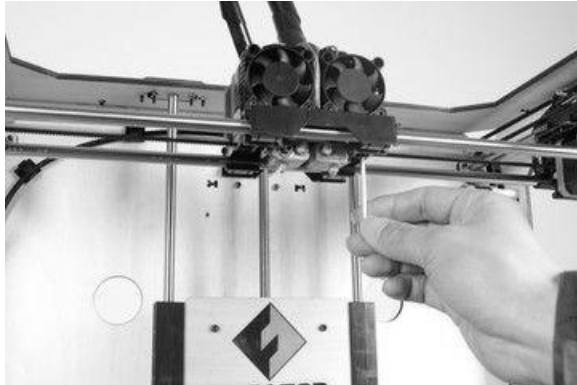
To do this, begin by carefully placing the Creator on one of its sides so that the bottom is exposed. You will see the Creator's power supply. Using a flathead screwdriver, switch the red power supply switch so that it displays 115V, as shown below. Now you are ready to install the extruder.



You'll need two silver screws from the bolt tool plate found in the accessory box and the appropriate hex wrench.

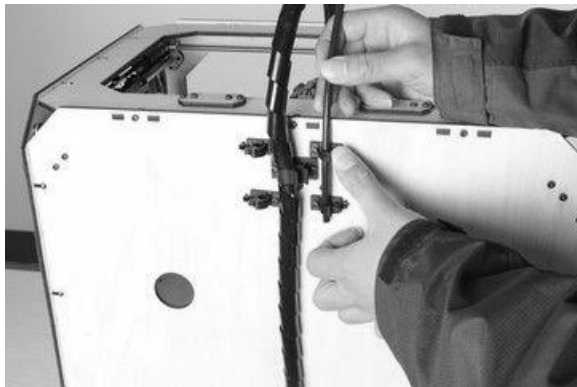
First, lower the build platform using one of the methods described in the previous section. Holding the extruder by both sides, take it out of the accessory sleeve and position it on the extruder seat with the fan facing forward. Align the screw holes and fasten with the shortest silver screws.





Next is the installation of the filament bracket. If you have two brackets, install one on each side; If you only have one, install it on the right hand side (when viewing the creator 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 empty spot on the extruder. Place one end of the guide tube into the hole.



The hardware installation is almost complete.

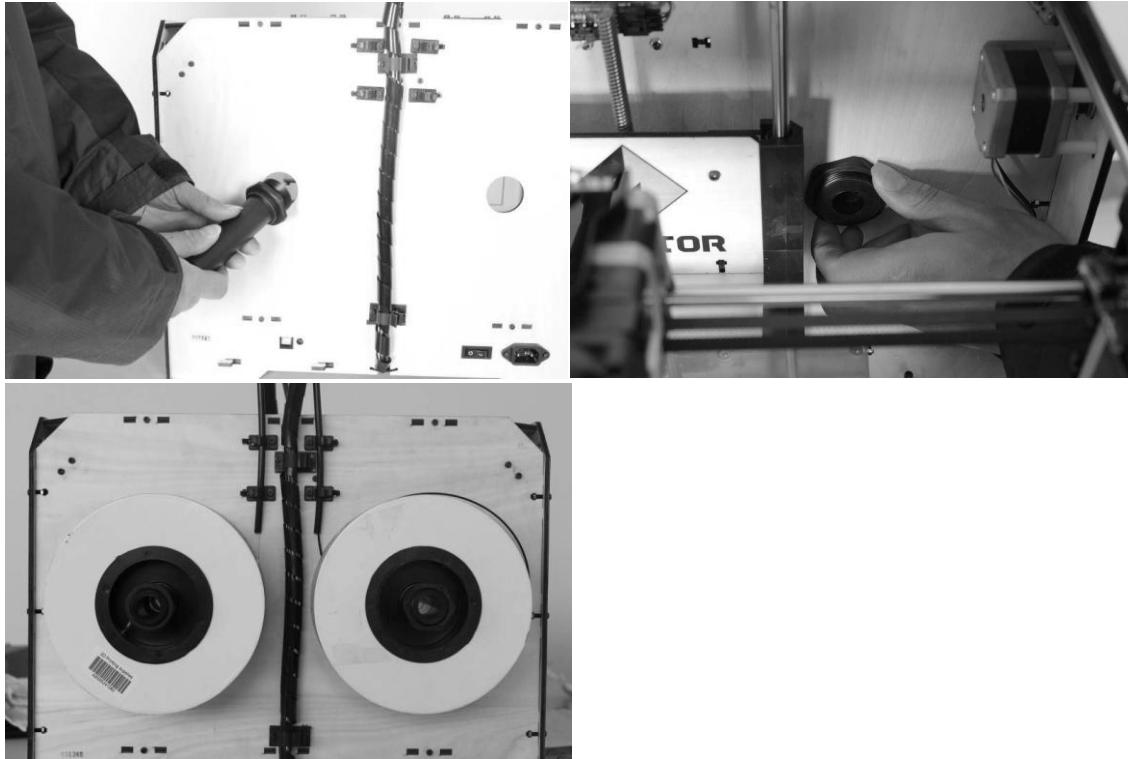
Next, with the power switch in the 'OFF' position, confirm that the power cord is plugged into the power outlet next to the power switch.



---

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

Finally, take the filament out of the box, install it onto the bracket 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.

---

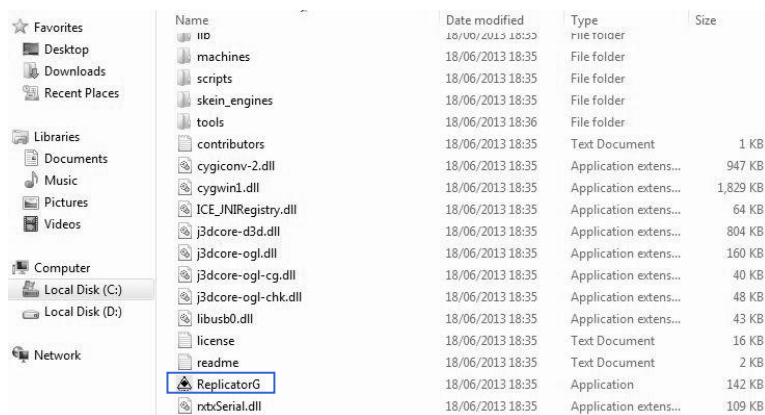
## 4 Software Installation

ReplicatorG0040 is the ideal software to use with the single or dual extruder Creator. The latest version can be downloaded from our official website by following the link below and clicking on “Download Replicator G 0400 Installer”.

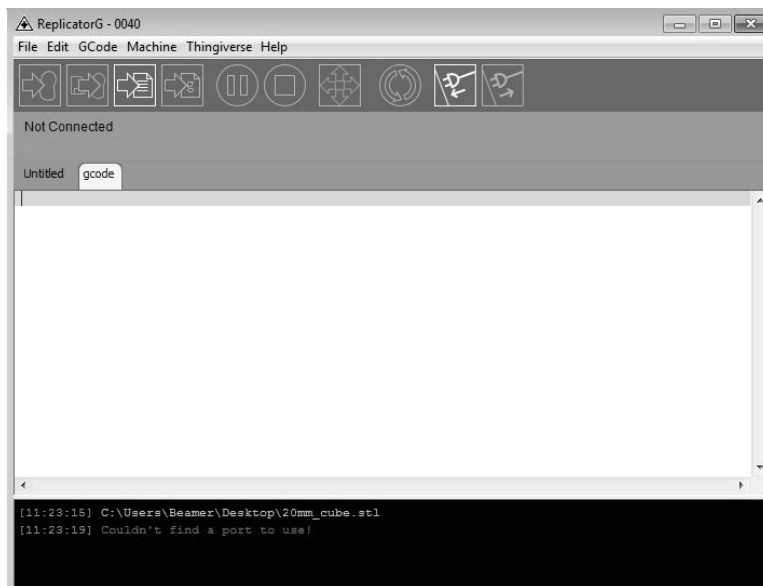
<http://www.flashforge-usa.com/printer-support/downloads/>

Download the files suitable for your system and decompress/extract them. For this guide, we will be installing on a Windows PC. For Mac users, please e-mail FlashForge USA at [supports@flashforge-usa.com](mailto:supports@flashforge-usa.com) if you encounter any problems or setbacks throughout this guide.

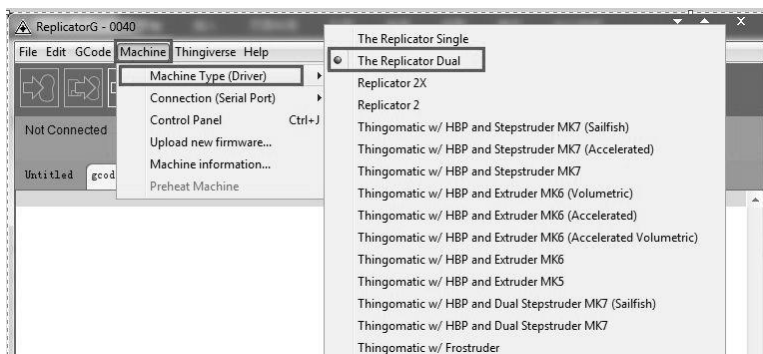
Run the Python installation file and the Python acceleration components. Click Replicator-0040-Installer to install the Replicator G software. To run the Replicator G software, double click on the ReplicatorG icon highlighted below.



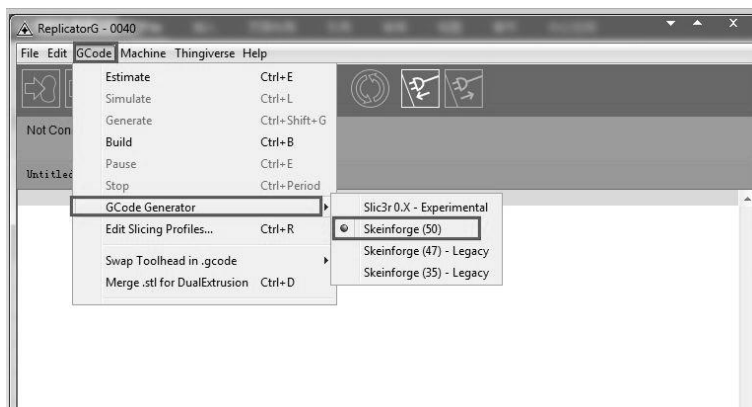
Name	Date modified	Type	Size
11D	18/06/2013 18:35	File folder	
machines	18/06/2013 18:35	File folder	
scripts	18/06/2013 18:35	File folder	
skein_engines	18/06/2013 18:35	File folder	
tools	18/06/2013 18:36	File folder	
contributors	18/06/2013 18:35	Text Document	1 KB
cygiconv-2.dll	18/06/2013 18:35	Application extens...	947 KB
cygwin1.dll	18/06/2013 18:35	Application extens...	1,829 KB
ICE_JNIRegistry.dll	18/06/2013 18:35	Application extens...	64 KB
j3dcore-d3d.dll	18/06/2013 18:35	Application extens...	804 KB
j3dcore-ogl.dll	18/06/2013 18:35	Application extens...	160 KB
j3dcore-ogl-cg.dll	18/06/2013 18:35	Application extens...	40 KB
j3dcore-ogl-chk.dll	18/06/2013 18:35	Application extens...	48 KB
libusb0.dll	18/06/2013 18:35	Application extens...	43 KB
license	18/06/2013 18:35	Text Document	16 KB
readme	18/06/2013 18:35	Text Document	2 KB
ReplicatorG	18/06/2013 18:35	Application	142 KB
ntxSerial.dll	18/06/2013 18:35	Application extens...	109 KB



After opening Replicator G software, click on **Machine** to select your **Machine Type**. Choose **The Replicator Dual** for Creator dual extruder machines, and choose **The Replicator Single** for Creator II single extruder machines. In this guide, we will select The Replicator Dual.



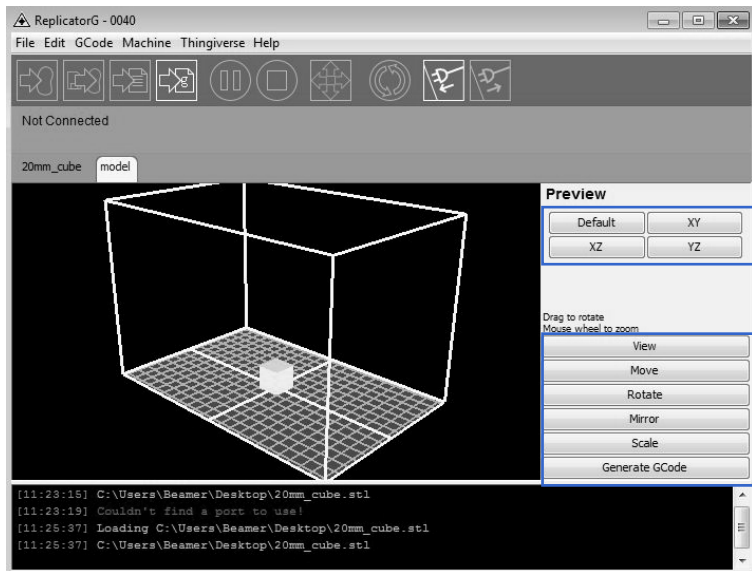
After selecting the appropriate machine, click on **GCode** on the top navigation bar, and under **GCode Generator**, select **Skeinforge (50)**.



---

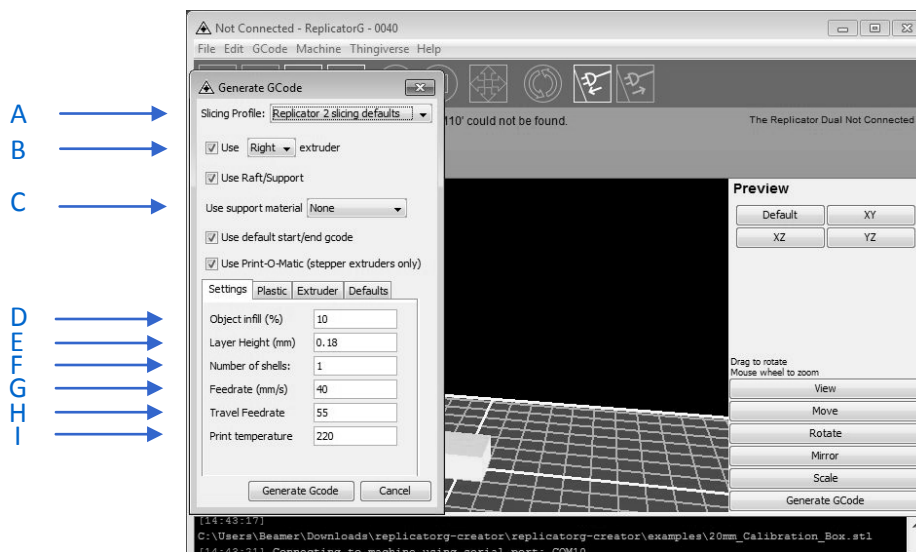
The following gives an introduction on how to import files (.STL) into the Replicator G software and generate Gcode in order to print.

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



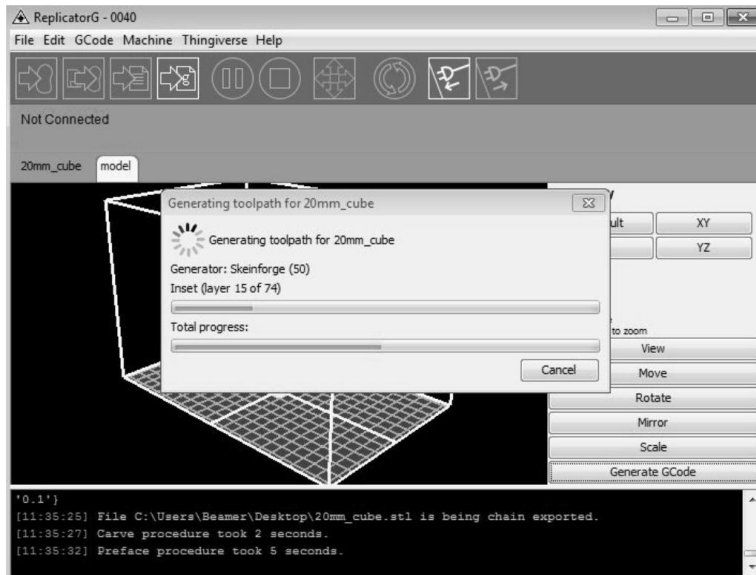
When the object is imported, you may find that it is not on the virtual build platform or even on the screen. Using the function keys indicated by the blue boxes in the illustration above, you can change the camera angle and reposition the object onto the center of the build platform. Once you've done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

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

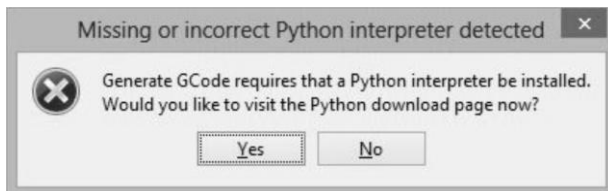


- A. Slicing Profile: select **Replicator slicing defaults for ABS printing**, then select **Replicator 2 slicing defaults for PLA printing**.
- B. This tells the printer which extruder to use for a dual extruder-headed printer — either the left head or the right head can be selected.
- C. If your sample will have any hanging surfaces, it is recommended to have support. Selecting **'None'** indicates that there is no support; **"Exterior"** means surface support; and **'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.
- 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: this is the speed at which the filament is fed into the extruder. This is usually set between **30** and **70 mm/s**. For ABS printing, **60 mm/s** is recommended; for PLA printing, **100 mm/s** is recommended.
- H. Travel Feedrate: this 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: this is the temperature at which the nozzle is heated. This varies between filament types. Set it to **220** for the default filament.

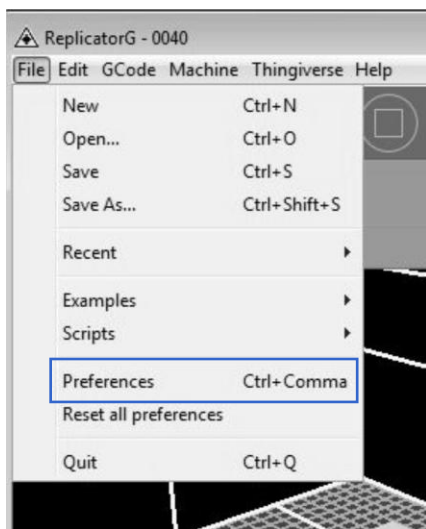
After you are finished inputting in the settings, click **Generate Gcode**, and the progress bar will appear.



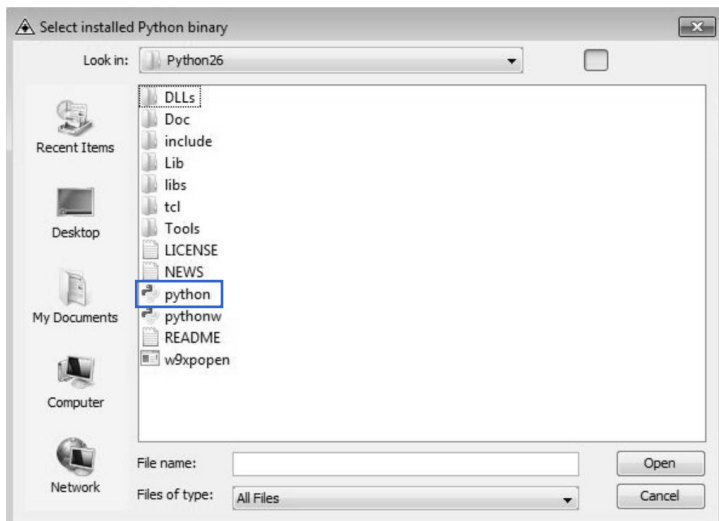
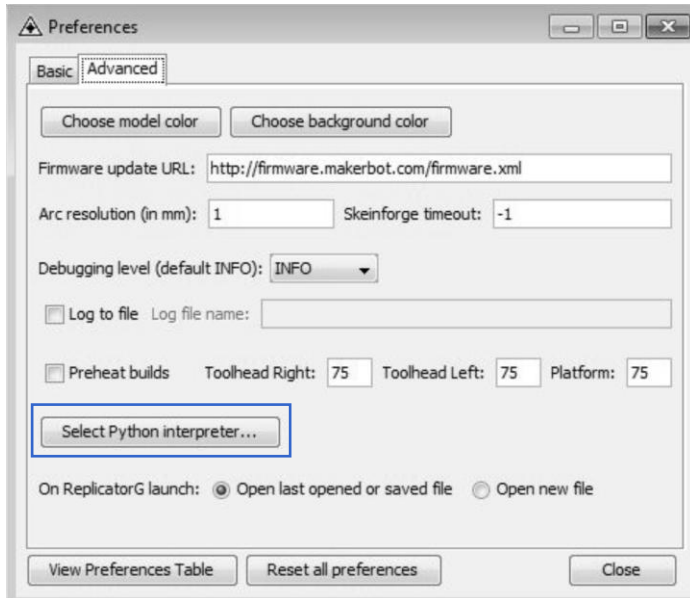
For users who did not choose the default installation path in the installation of Python, clicking the **Generate Gcode** button in Replicator G will result in dialogue box popping up alerting that the executable Python file cannot be found.



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



Click **File > Preferences** on the top navigation bar, then click **Select Python Interpreter** under the **Advanced tab**. A new 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!

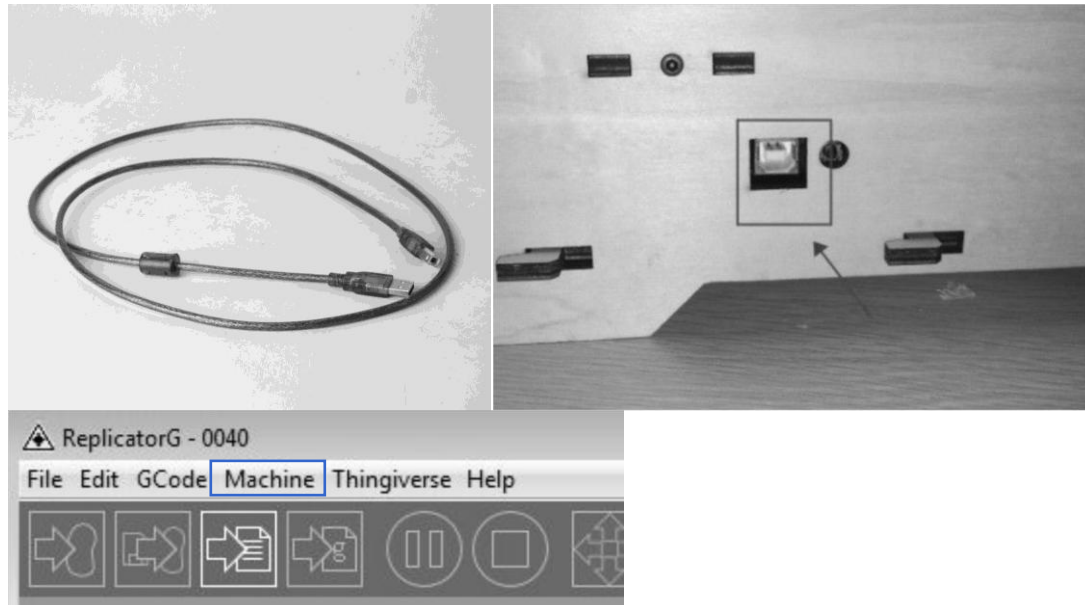
The machine will now work as normal when generating Gcode, and the Python error message will not pop up again. In the next section, we will start a preliminary test on the machine's connections and heat the platform so that the extruder is ready for its first print!



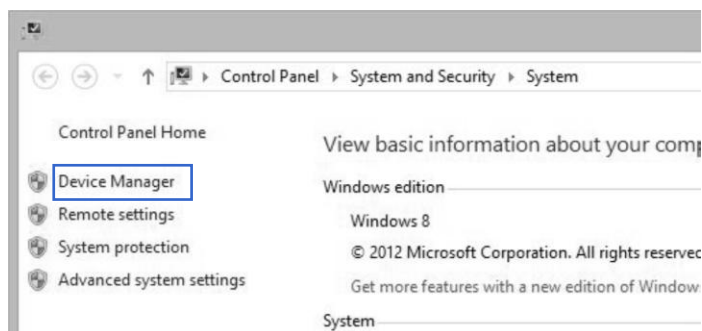
---

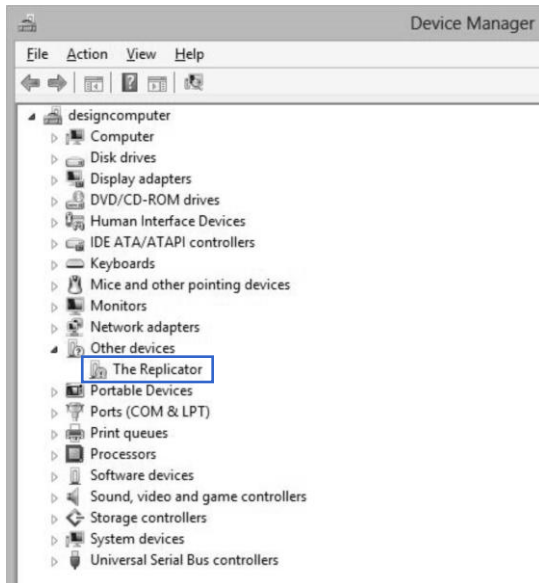
## 5 USB Connection & Temperature Setting

First, connect the machine and the computer using 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 Replicator G software as we are going to link the computer and printer.

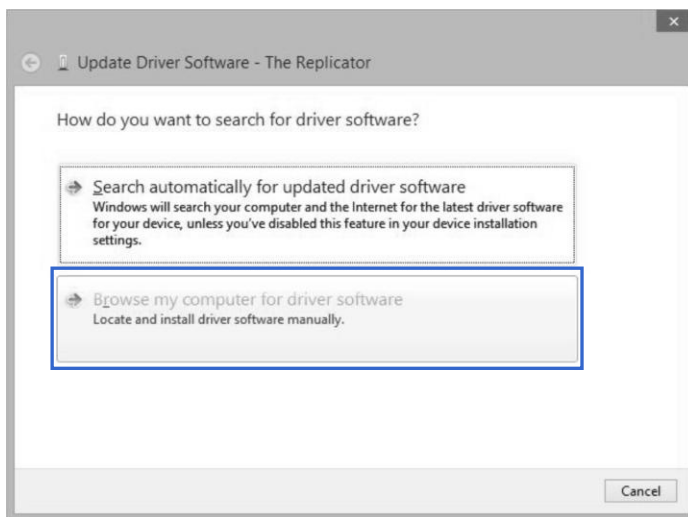


Inside the Replicator G software, click **Machine** > **Connection (serial port)** > **Rescan serial ports**, if no new ports appear, then the software driver has not been installed. To install the driver manually, go to '**My Computer**' and right click on '**Properties**'. The basic system parameters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

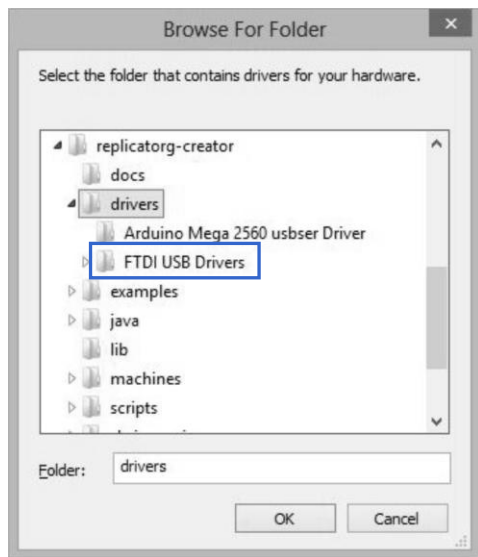




Locate the software driver shown in the blue box above. Right click and select '**Update Driver Software**'.

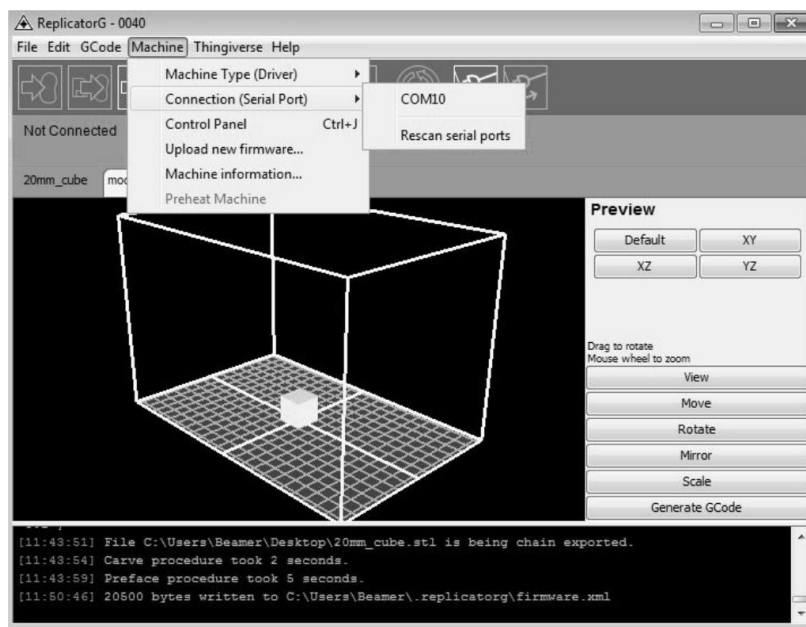


Click '**Browse my computer for driver software**' to find the location of ReplicatorG0040 on your system.



Click '**FTDI USB Drivers**' in the driver folder and click '**OK**'. The drivers will then be installed.

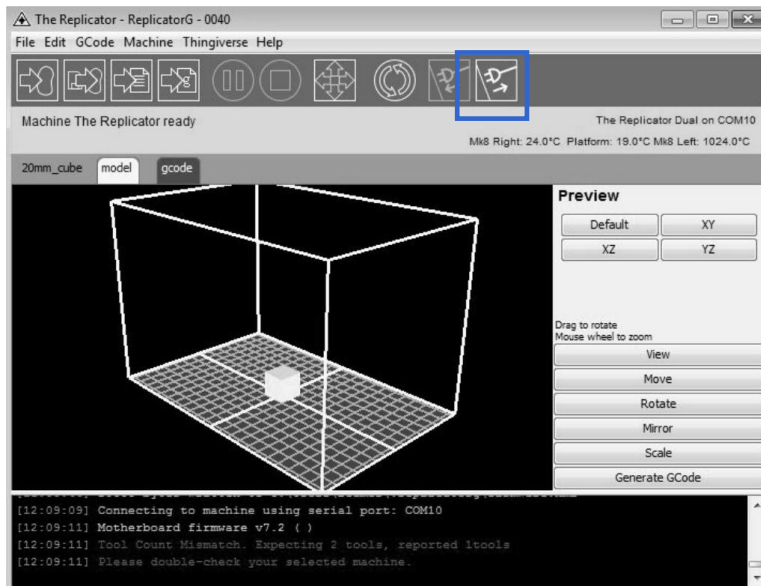
The next step is to link the printer. Rescan the serial ports and select the one that appears on your machine (on our test machine the port was COM10).



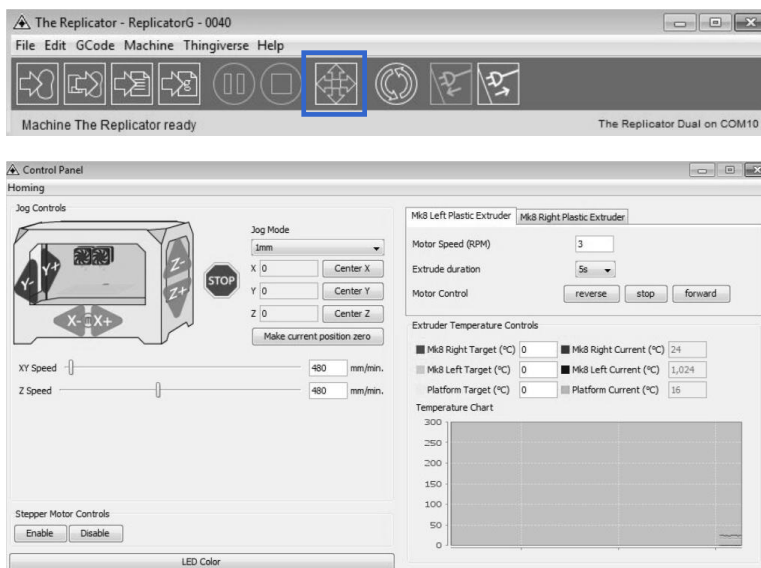
Now we can link the machine to the computer. Click on the printer connect icon highlighted in blue.



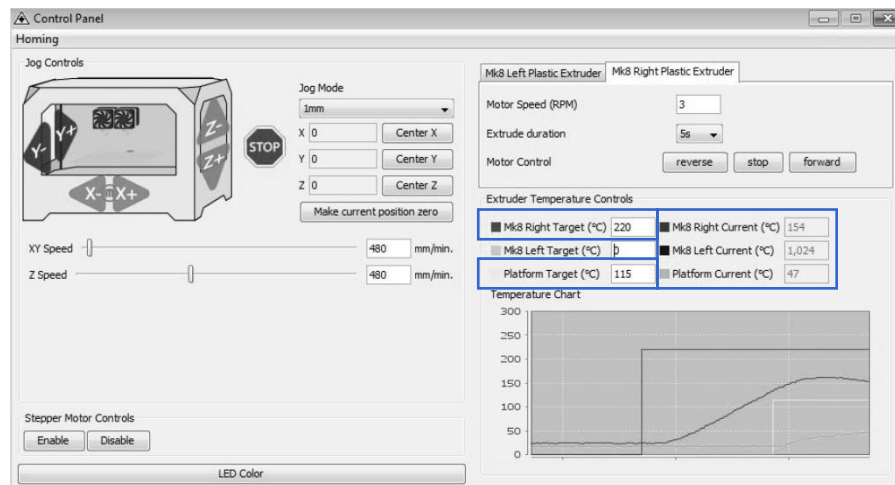
If the printer connect icon turns dim, as shown in the illustration below, the printer has successfully connected to the computer. Next, we need to preheat the extruder and build platform.



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 activate and the current temperature value will display to the right as shown on the following page.



---

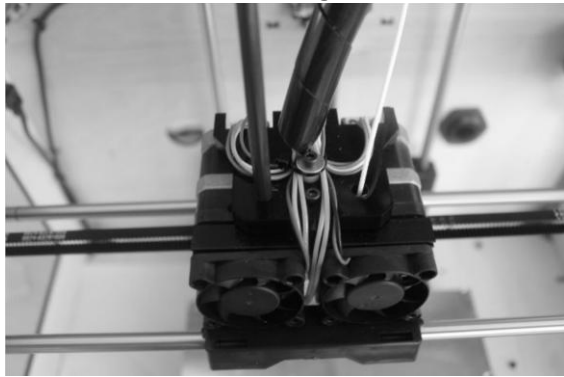
## 6 Filament

To make the process of feeding the filament into the printer 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 Installing the 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:



---

## 6.2 Feeding the Filament Using LCD Screen

1. Turn on the Creator; 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 **middle key** in the middle 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 **middle key** in the middle 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 **middle key** again on the keypad. The display will indicate: "I'm heating up my extruder!" At this time, the temperature of the nozzle is being heated up. When the temperature of the nozzle reaches its target temperature, press the **middle key** on the keypad, and the nozzle should start extruding material. If not, keep pressing the **middle key** until it does.

---

### 6.3 Withdrawing the Filament Using LCD Screen

The procedure to remove filament from the extruder is simple. We will first load the filament so that it can extrude the material. Afterwards, we will unload the filament and remove it from the extruder. This is so that the removal process is smooth and the filament does not become stuck.

1. Turn on the Creator; the display will indicate:
  - Build from SD
  - Preheat
  - Utilities
6. 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
2. Select **Utilities**; press the **middle key** in the middle of the keypad. The display will show:
  - Monitor Mode
  - Change Filament
  - Level Build Plate
  - Home Axes
  - Feed operation with LCD screen
3. Select **Change Filament**; press the **middle key** in the middle of the keypad. The display will show:
  - Load Right
  - Unload Right
  - Load Left
  - Unload Left
4. Select the appropriate side of the extruder you wish to load (Left or Right for models with dual extruders). Press the **middle key** again on the keypad. The display will indicate: "I'm heating up my extruder!" At this time, the temperature of the nozzle is being heated up. When the temperature of the nozzle reaches its target temperature, press the **middle key** on the keypad, and the nozzle should start extruding material. If not, keep pressing the **middle key** until it does. Allow the FlashForge Creator to extrude material for approximately 10 seconds, and proceed to the next step.
5. Select **Exit** to go back to the previous menu. Using the keypad, select **Unload (Side)**, depending on which extruder you wish to remove filament from (for dual extruder model only). The Creator will now begin to heat up again to its target temperature.
6. After it is finished heating up, the extruder will begin to reverse its gears and unload the filament. Assist in the removal process by pulling the filament out very gently, making sure the angle of the filament is straight up and not crooked. Failure to do so may cause the removal process to fail.



---

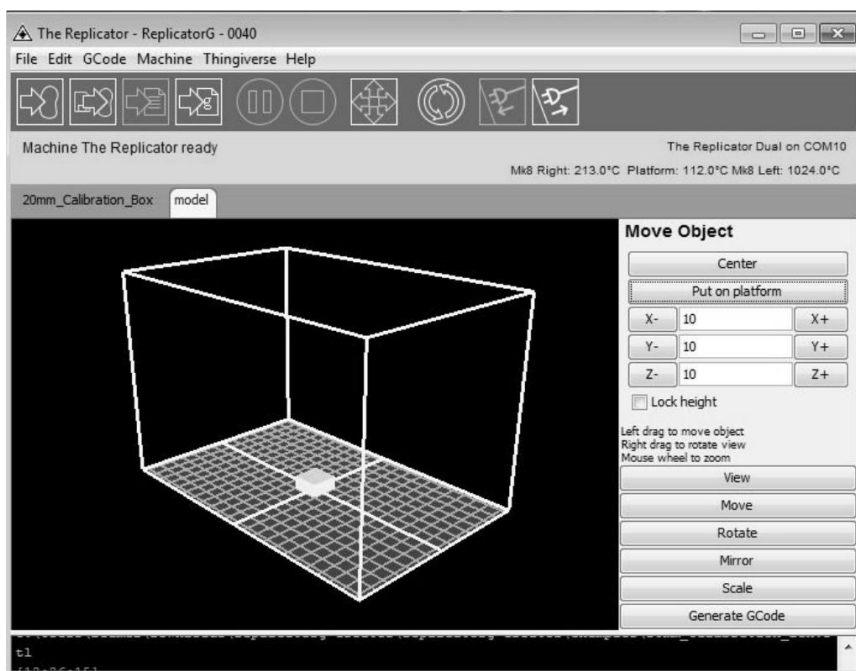
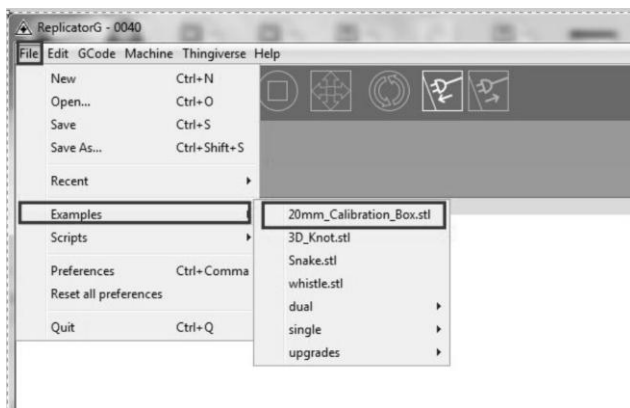
In the event that a grinding noise occurs and the filament stop unloading, select **Exit** on the LCD screen and repeat steps 1 to 6. If the error persists after several attempts, please call FlashForge USA technical support at **626-322-3855** (M-F 8:30 a.m. PST – 5:00 p.m PST) or e-mail [supports@flashforge-usa.com](mailto:supports@flashforge-usa.com).

### 6.5 Setting the Parameters

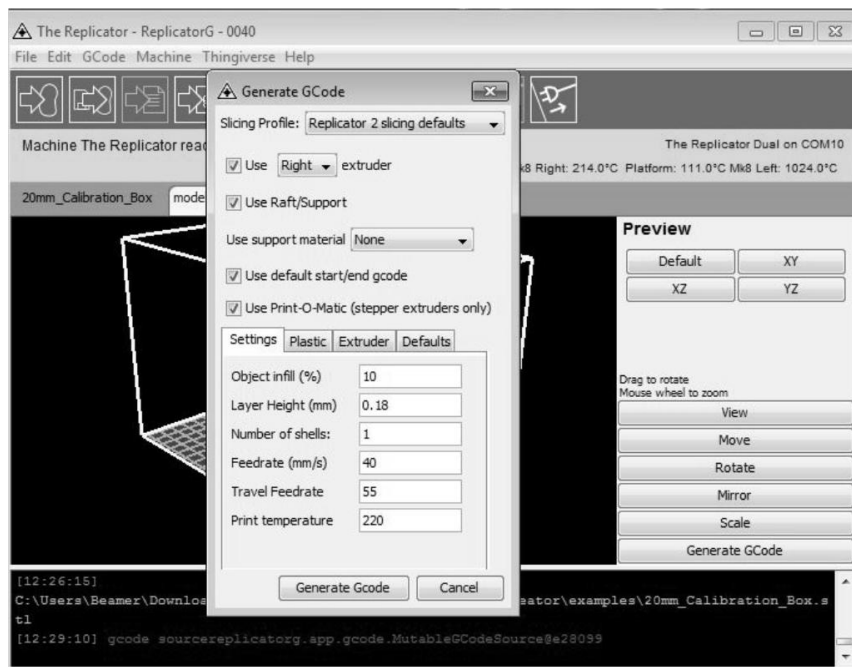
Before generating Gcode, specific parameters can be set up to customize various aspects of the print. An explanation to each of the settings can be found in the previous section (pages 13-14).

## 7 Initial Print

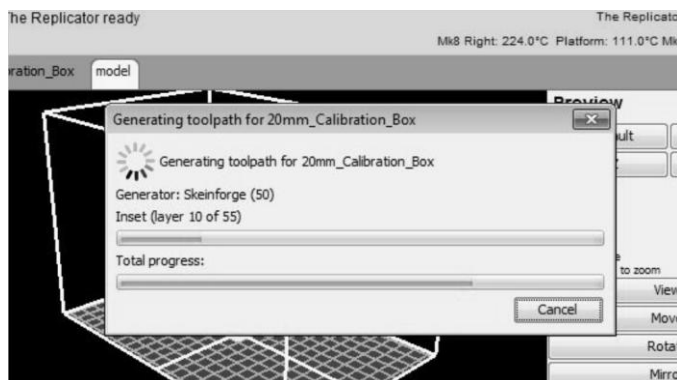
Click **File > Examples** and then select “**20mm\_Calibration\_Box.stl**”. The preview interface will then appear along with a virtual impression of the 20mm cube on the virtual printer bed. On the preview interface click **Move > Center** and **Put on platform** so that the sample will be printed on the center of the build plate.



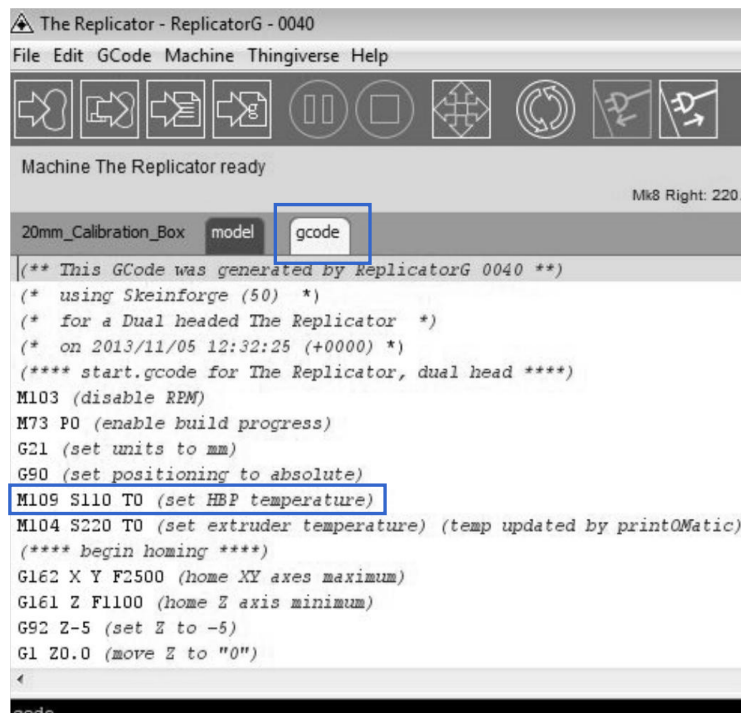
Next, click **Generate GCode**; a dialogue box will pop up. After inputting your chosen parameters, the Gcode will be generated.



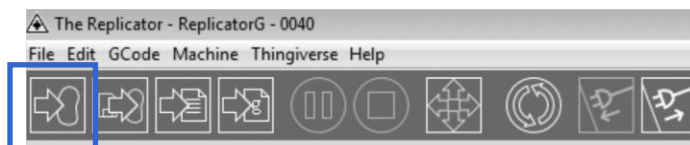
A loading bar will then appear showing the progress of the Gcode generation.



After the Gcode is generated, the temperature of the bottom plate needs to be changed. See below for the required modification. First, select Gcode as shown highlighted in the illustration below and change **'M109 S110 T0'** to **'M109 S115 T0'**.

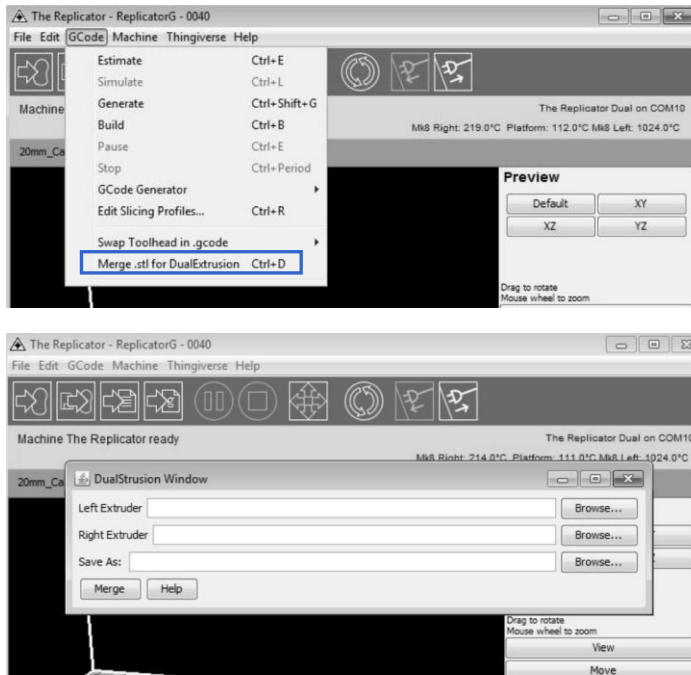


After modifying, click on file and save the sample. Next, click on the left most icon on the green bar, as highlighted below. The Gcode has now been sent to the printer and the object will start printing shortly.

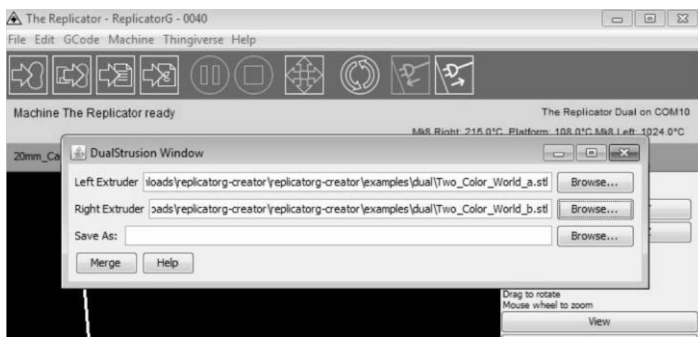


## 8 Dual Extruder Print

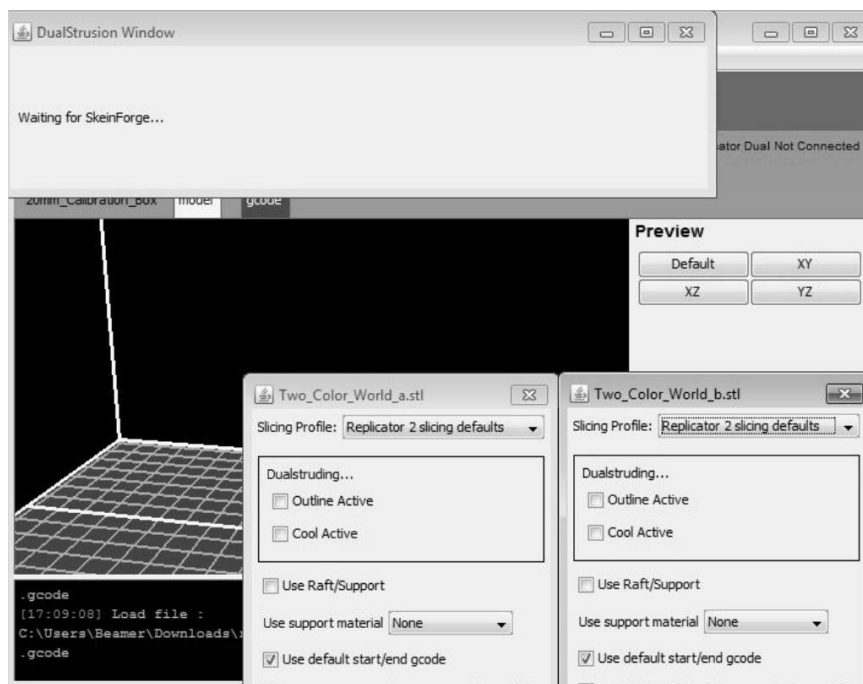
Open Replicator G and select **Gcode > Merge .stl for Dual Extrusion**. You will see the following:



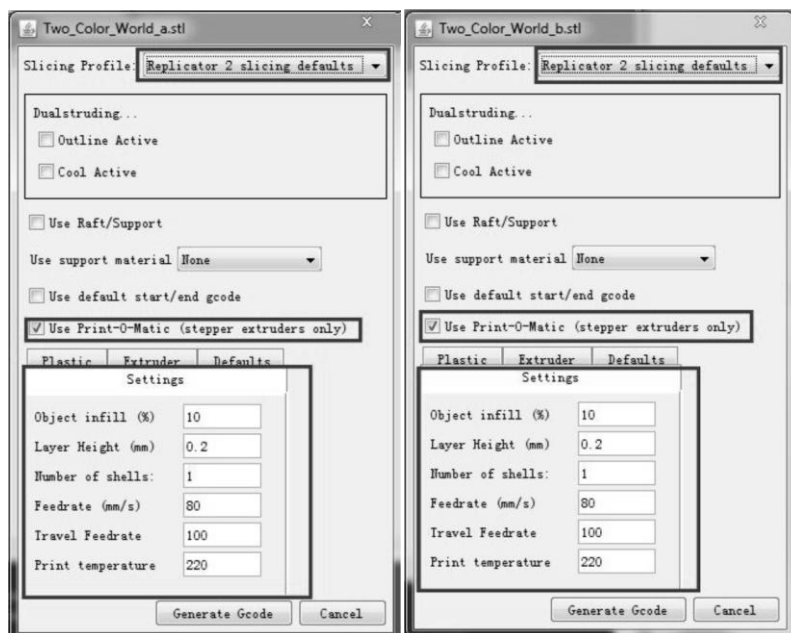
Click on the first **Browse** button to locate and select the file for the left extruder, and then click on the second **Browse** button for right extruder:



Click on the third **Browse** button to save these two files as one .gcode file on the desktop; now click **Merge** button, you will see two dialogue boxes pop up:

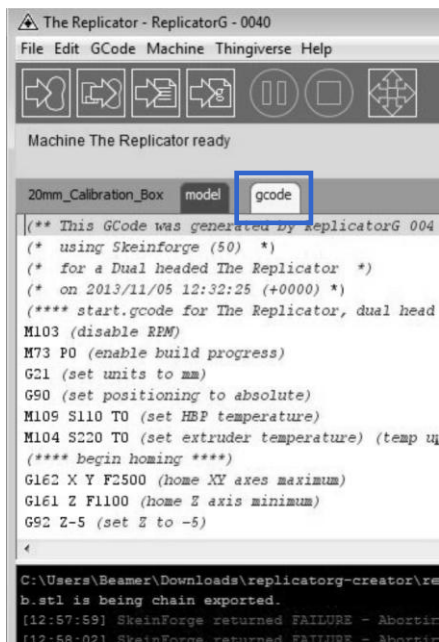


If you want to print dual colors with PLA filament, then perform the following settings:



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.

