User Guide

Creator Pro Desktop 3D Printer

FLASHFORGE

FLASHFORGE 3D PRINTER

Create a Joyful 3D Life with You!

Contents

Acknowledgment

Our sincere gratitude goes to you for your great support to Flashforge Corporation. Even if you are familiar with earlier Flashforge machines or 3D printing technology, we still recommend that you read through this guide, as there is lots of important information about the Creator pro for you to get a better 3D experience. Following this guide will ensure your successful first print journey.

Acknowledgment

Commitment

Each member of Flashforge team is definitely passionate about 3D printing and willing to share this new technology with each individuals of the world. Flashforge is committing to providing the best 3D printers and best service to each clients and customers.

If you can imagine it, you can make it. Together with Flashforge team, let's turn the sky into limit.

PRECAUTIONS

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

⚠ **E**nsure that your Creator pro is earthed lest electrostatic interference.

⚠ **T**urn off the machine and unplug the power cord before any maintenance or modifications.

⚠ **A**lways avoid touching the heating nozzle and build plate during operation

⚠ **O**nly use your Creator pro in a well-ventilated area. (When printing with ABS, small concentrations of Styrene vapor can be released. This may (in some cases) make you uncomfortable.)

⚠ **D**o not wear gloves when operating or repairing, as entanglement may occur and injury may cause .

⚠ **D**o not leave the machine unattended when during operation.

Chapter 1: What's in the Box?

Along with your Creator Pro 3D printer, the box also contains the followings:

On the top of the PE foam sheet, you'll find:

- Filament guide tube*2
- Quick start*1
- Build tape*2
- Leveling card*1

Within the accessory PE foam

- Power cable *1
- USB cable*1
- Spool holder*2
- Dual extruder*1
- Tool bag(Allen Wrench/Leveling knob/Bolt)
- Extruder's accessory kit(Bolt/Turbofan baffle)
- 4G SD card *1

Under Creator Pro's build platform:

- ABS filament*1kg (2.2pounds),
- PLA filament*1kg (2.2pounds),
- Lid*1

Chapter 2: Unpacking

The Creator Pro was carefully packaged at FLASHFORGE manufacturing facility. Please follow the unpacking steps laid out below.

> **CAUTION**
>
> ⚠️　Handle the package and its contents with extra care; do not use any unnecessary force.
>
> ⚠️　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.

Place your Creator Pro on a clean and stable surface, open the carton and then take the foam sheet out. Within the PE foam sheet are lying **two filament guide tubes, one piece of build tape, one piece of Flashforge leveling card and one Quick Start Guide**.

Firmly grasp the two side handles of Flashforge Creator Pro. Lift it out of the box and place it on a stable surface. Then open and remove the transparent covering.

The Flashforge Creator Pro and its accessories shall be displayed in front of you. Take the dual extruder out and place it onto a clean ground.
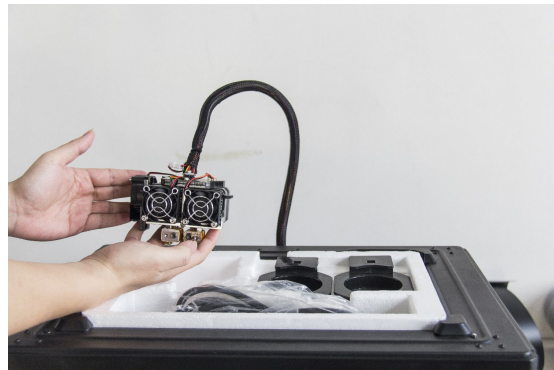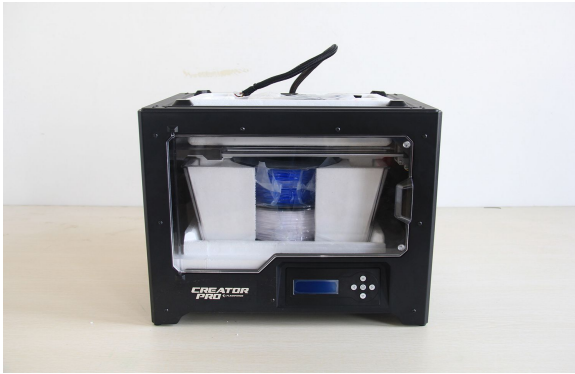
Take the accessory foam container out, within which are lying **a power cable, a USB cable, two spool holders, a bag of tools, a SD card, a bag of extruder accessories and turbofan baffle.**
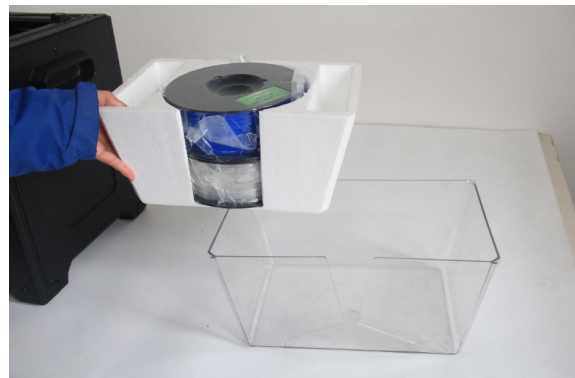

Slide the X-axis rods to the very back. Take the transparent lid together with two rolls of filament out from the printer. Take the two rolls of filament(**PLA filament * 1, ABS filament * 1**) out from the enclosed lid.
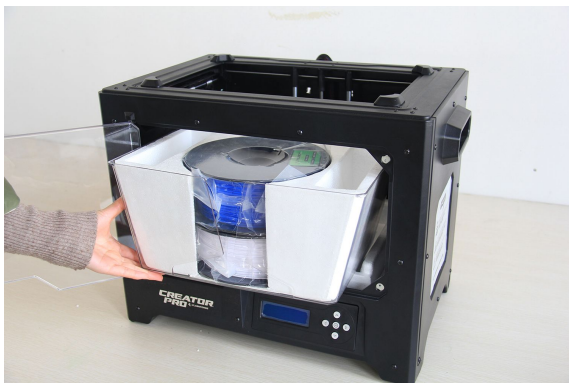
Elevate the build plate to its limit. Take the side protective foam sheets from the printer. Take the bottom protective foam from the printer.
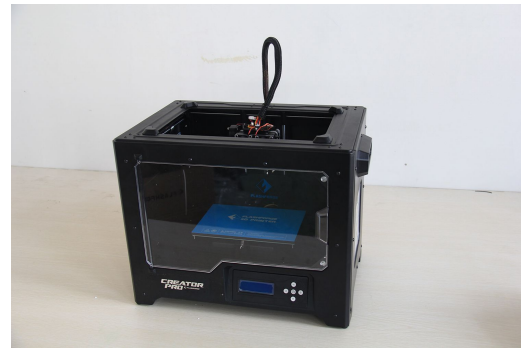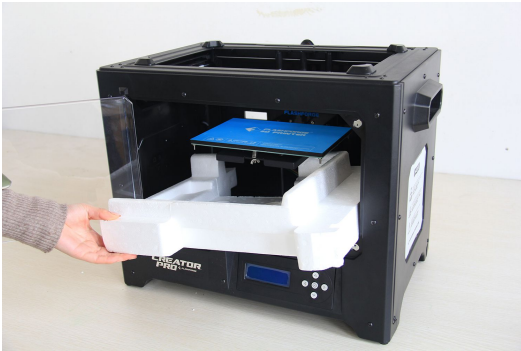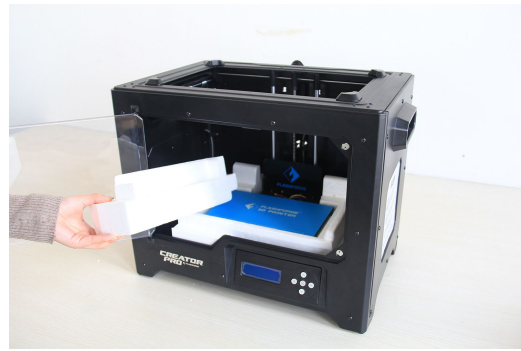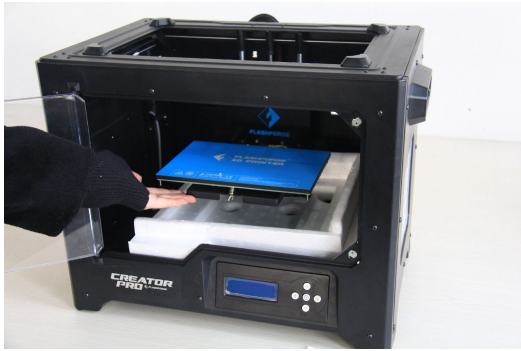
Congratulations! You've finished the unpacking process. (Tip: Save your Flashforge Creator Pro packaging for future transportation and storage.)

Chapter 3: Hardware Assembly

First, you need two M3*8 bolts from the extruder's accessories kit(including bolts and turbofan baffle), and the 2.5mm Allen Wrench. Lower the build plate as much as possible, hold 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 bolts.

Unscrew the two turbofan bolts and take out the turbofan baffle to install.(Note: Beside the turbofan bolt hole locates a small hole. You need to in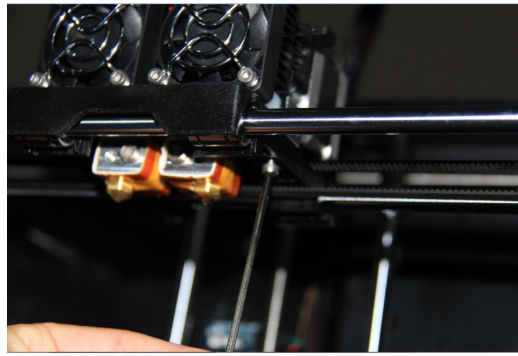sert the bump of turbofan baffle to this hole) Take out a M3 x 6 bolt and a 2.5mm Allen Wrench to secure the turbofan baffle onto the extruder. Finally, screw the two turbofan bolts.
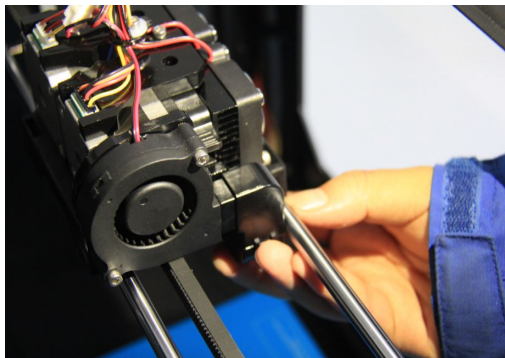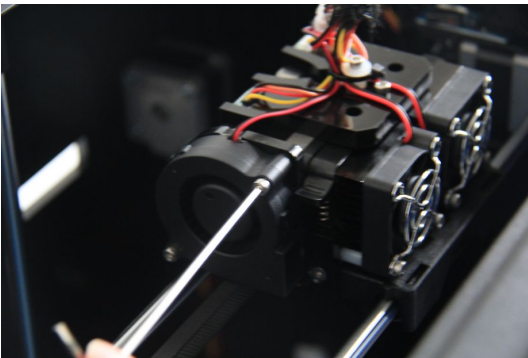
The next step goes to the installation of the spool holders. Install one on each side.

To install a spool holder, level it and insert the end into one of the openings. Turn down the spool holder to make the holder bottom cleave to the printer back.
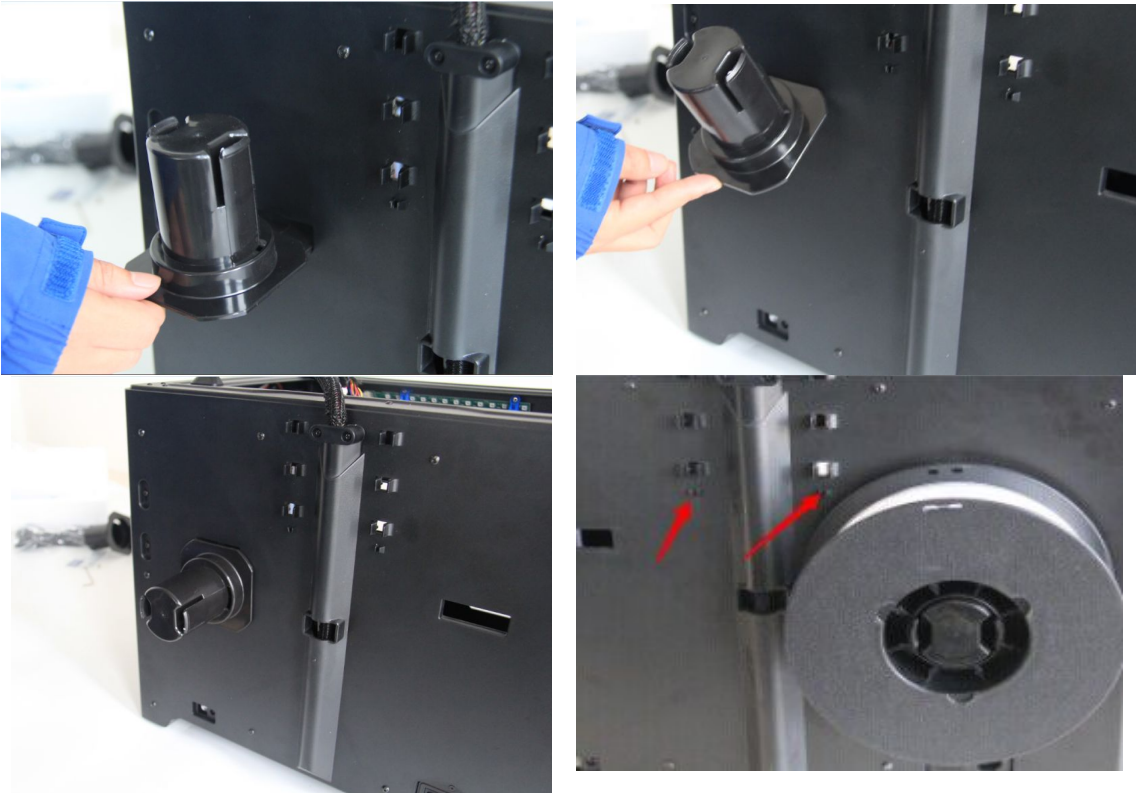
To lock the filament guide tube with R-shape buckles and insert another end to the filament intake. To avoid filament jams, always ensure that any filament spool you mount on the Creator pro feeds from the bottom of the spool toward the top. Filament mounted on the right spool holder when viewed from the back should always unspool clockwise and filament loaded onto the left spool holder should always unspool counterclockwise.
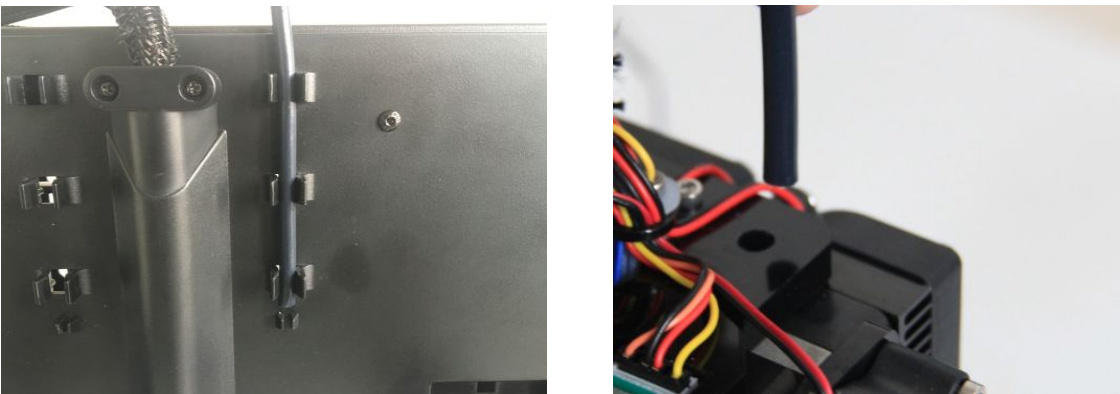
Congratulations! You've finished the hardware assembly process.

Chapter 4:Software Installation

SPECIAL NOTES TO WIN 8 and WIN 10 USERS BEFORE SOFTWARE INSTALLATION

Win 8 and Win 10 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

Steps for installing your software:

Method 1: (1) Download the appropriate version of ReplicatorG suitable for your system from: www.ff3dp.com

Steps：Support--Downloads--Creator Pro FDM 3D Printer--Software(ReplicatorG&Python)

(2) After completing loading, please decompress the RAR. files.

(3) First, run Python-2.7.8.amd64 and the Python 2.7.8 and the double click the Replicator G icon to start the software.

Method 2: From the SD card shipped with your printer, browse to the " operational software " folder and run Python-2.7.8.amd64 and the Python 2.7.8.

After completing the Python component installation, click Replicator-0040r24-Sailfish-windows to install the ReplicatorG software.

To run the ReplicatorG software, double click the shortcut on your desktop or in your start menu.
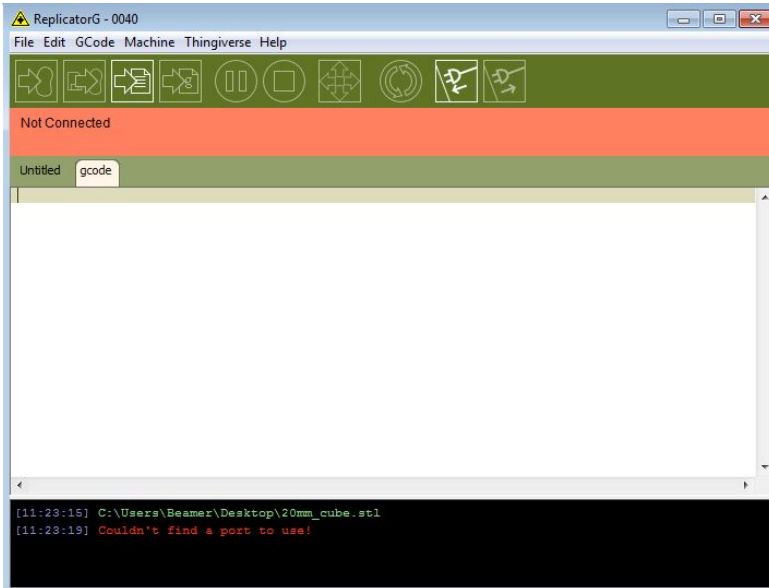
After opening ReplicatorG software, click on **Machine** to select your **Machine Type(Driver).** Choose **The Replicator Dual** for Creator Pro dual extruder machine.
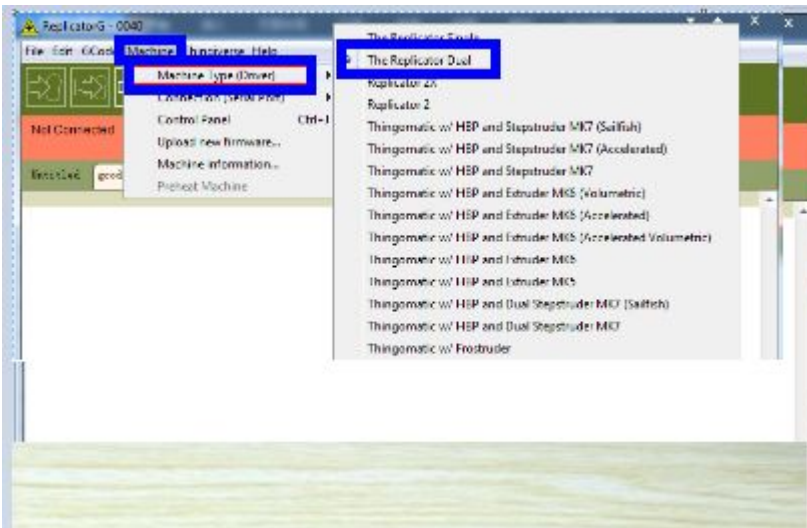
After selecting the appropriate machine type, click on **GCode** on the top navigation bar, and under **GCode Generator**, select **Skeinforge(50)**.
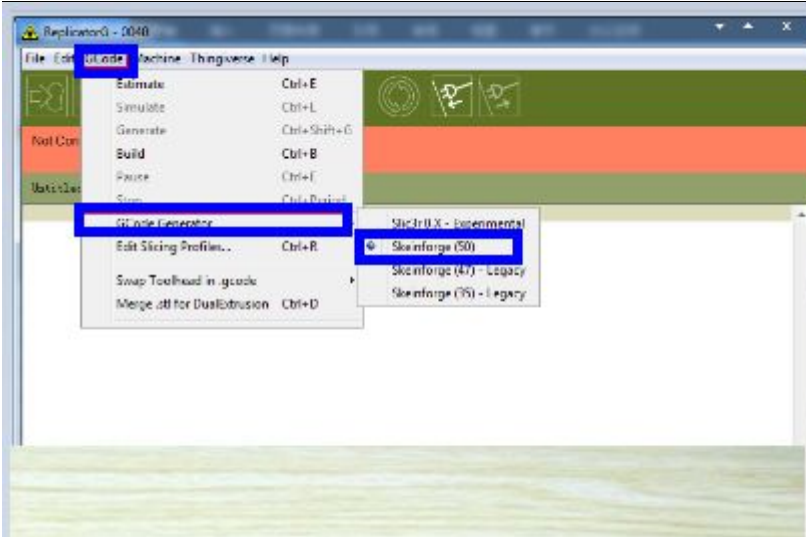
The following gives an introduction on how to import files (.STL) into the Replicator G software and then generate Gcode to print your creation.

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.
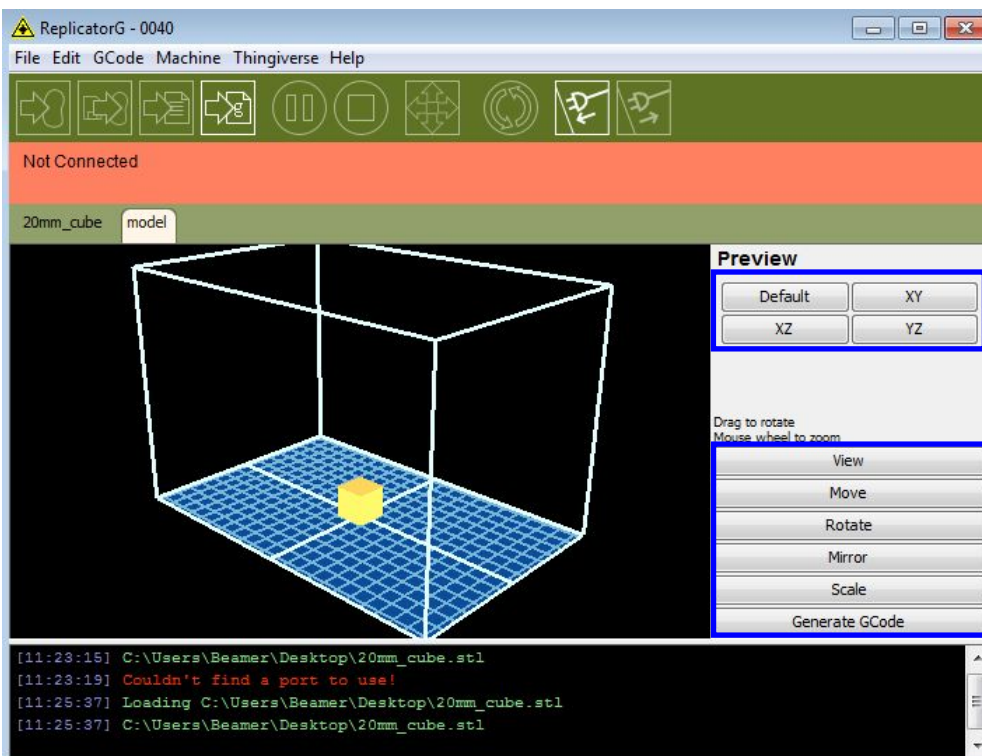
When the object is imported you may find that it is not on the virtual build platform or even on the screen, using the function keys indicated by the blue boxes in the illustration above, you can change the camera angle and click [Move]--[Center] to re-position the object onto the center of the build 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.

Note   **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.**

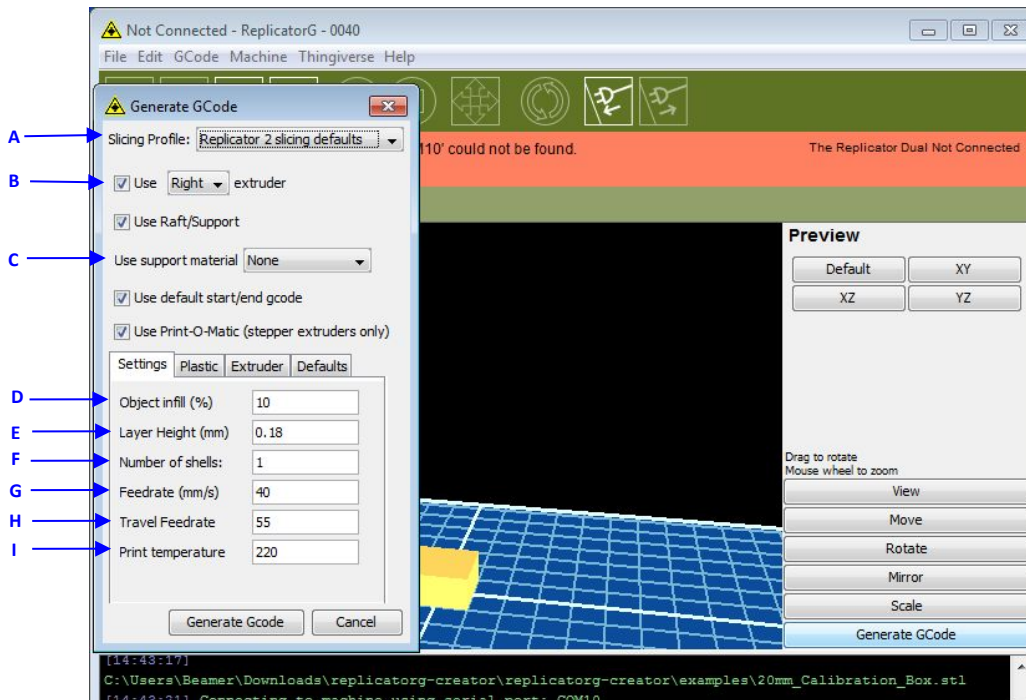**B.** This tells the printer which extruder to use for a dual-extruder head printer, either the **left** or **right** head can be selected.

   Use Raft/Support: Raft helps the model stick to the build plate firmly and the supports works as the bridge pillars. (If raft or supports are necessary, you need check the box)

C. If your sample will have any hanging surfaces, it is recommended to have support. Select **None** indicates that there is no support. **Exterior support** 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: the thickness of each layer. The recommended thickness is **0.27mm**.

F. Number of shells: this is the wall thickness; it's usually set at **1. (Default thickness is 1.2mm)**

G. Feedrate（mm/s） is the extruder speed while extruding filament, and this is usually set between **30** and **100**. For **ABS printing**, **60** is recommended; for **PLA printing**, 80℃ is recommended.

H. Travel feedrate is the extruder speed at while no filament is extruded, and it's usually set between **30** and **120**. For **ABS printing, 80℃** is recommended; for **PLA printing, 100℃** 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.

As for the" plastic", "extruder" and "Defaults", you just need to maintain the default settings. After you finish inputting in the settings, click **Generate Gcode** and the progress bar will appear.
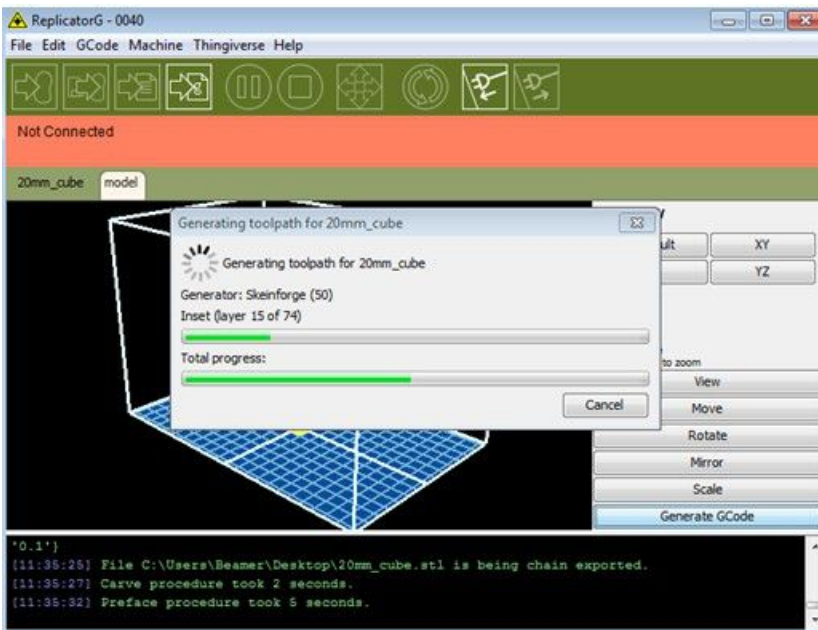
For users who did not choose the default installation path in the installation of Python, clicking the **Generate Gcode** button in Replicator G will result in dialogue box popping up alerting that the executable Python file cannot be found.
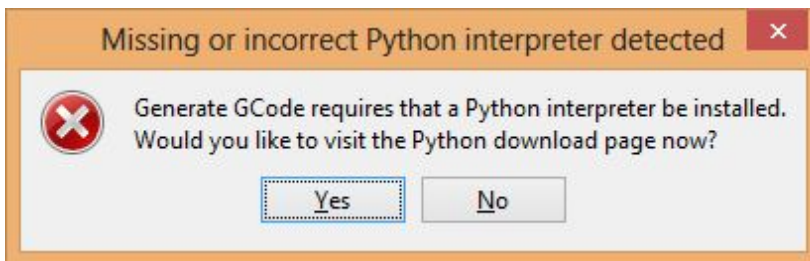
In order to solve this, click the **No** button to dismiss the dialogue box. Next, we need to configure the corresponding menu.
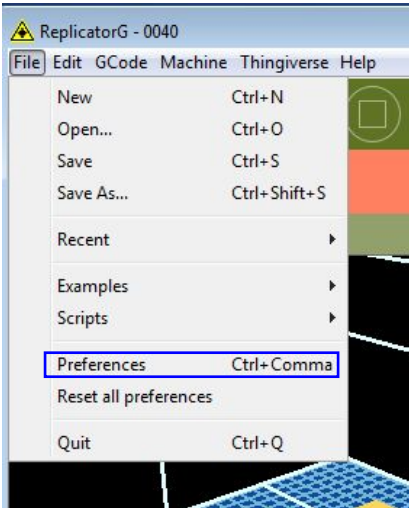
Click **File** > **Preferences**, then click **Select Python interpreter** under the **Advanced tab**. A window will pop up. Navigate to the Python installation directory, and select **python.exe** and click **Open**.
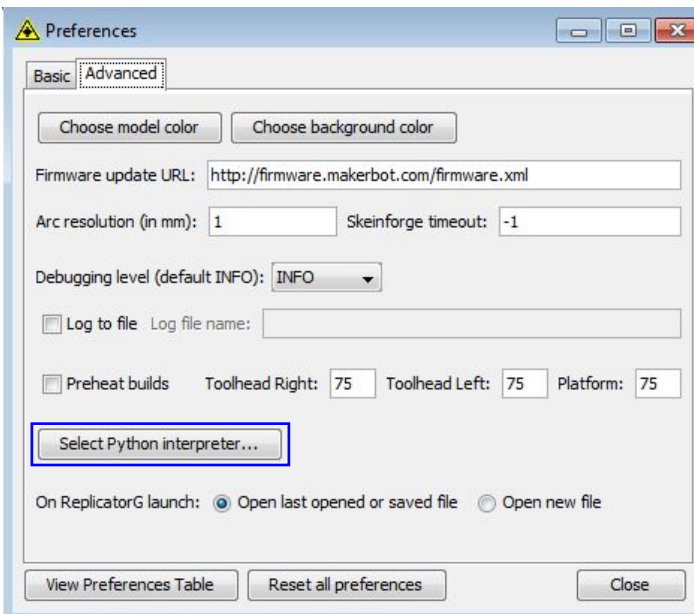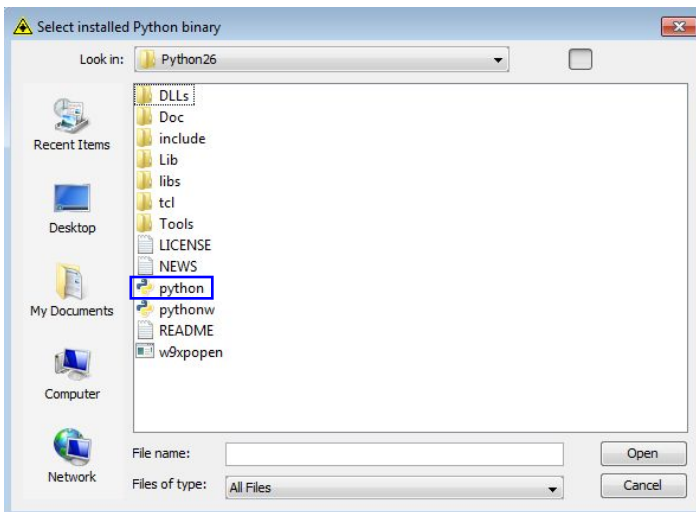
Last, click **Close** on the Preferences menu and you are done!

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.

Chapter 5: USB Connection & Temperature Setting

First, connect the machine and computer with the provided **USB 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.
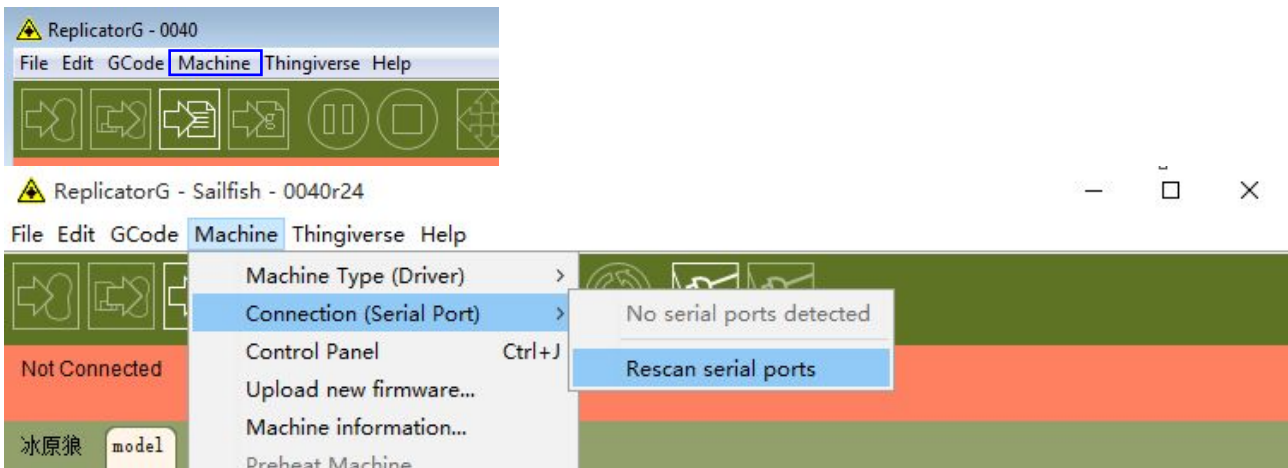
Inside the ReplicatorG software, click **Machine** > **Connection (serial port)** > **Rescan serial ports**, if no new ports appear then the software driver has not been installed. To install the driver manually, go to **My computer** and right click on **Properties.** The basic system parameters appear. Next select **Device Manager**.
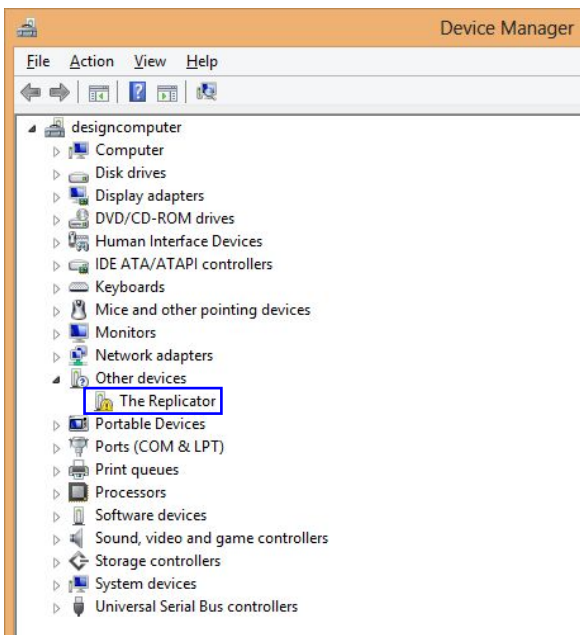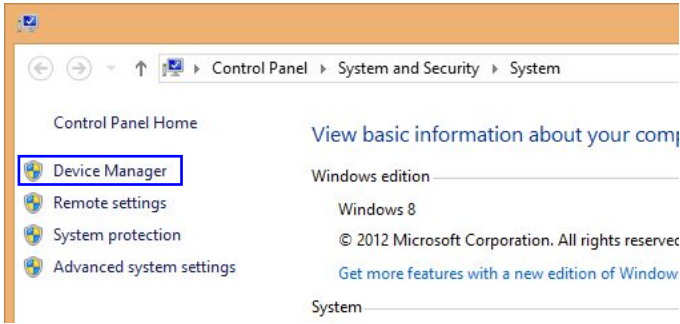
Locate the software driver shown in the blue box above. Double click **The Replicator**, then a dialog box will pop up. Click **Driver Software** on the top and then and click **Update Driver Software**.
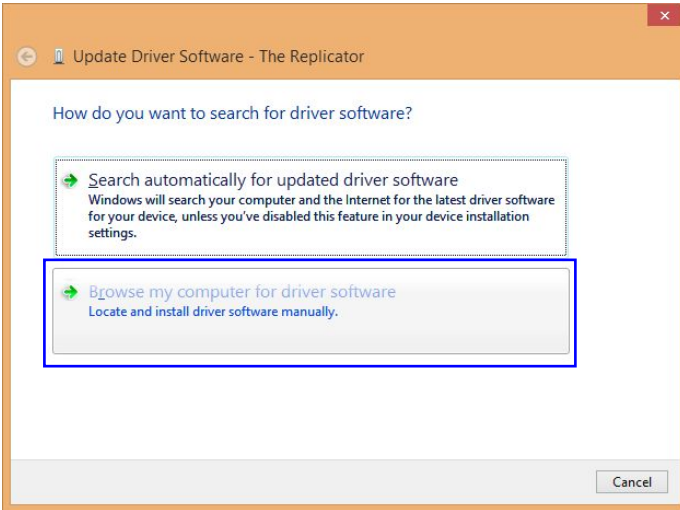
Click **Browse my computer for driver software** to find the location of **drivers** under ReplicatorG0040 on your system.
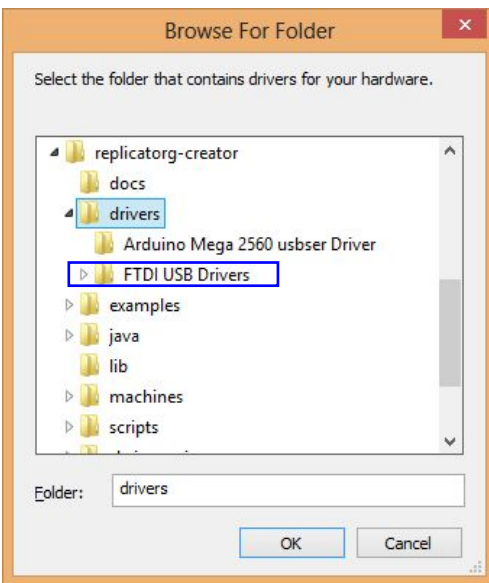
Left click **drivers**. And then click **FTDI USB Drivers** in the driver folder before confirmation and click **OK**. The drivers will then be installed.

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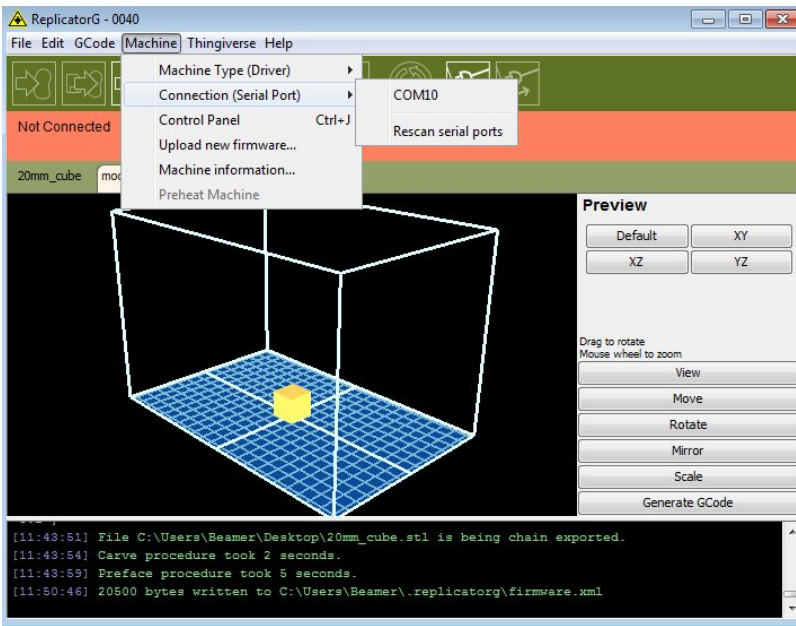
Now we can link the machine to the computer. Click on the printer connect icon highlighted in blue :

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 plate.
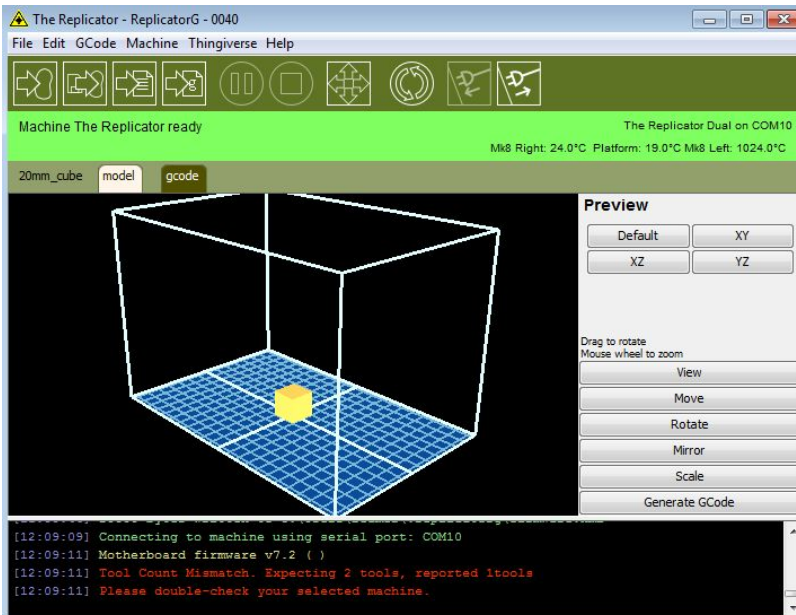
In the same set of menu, click on the cross-shaped icon and a new interface will appear.
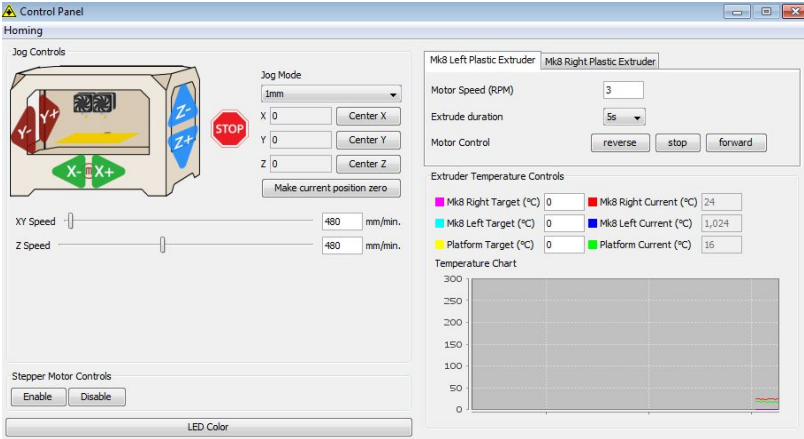
Input the following target values: 220°C for the extruder and 110°C for the build plate, type **Enter** to confirm. After entering the values, the plate will start to preheat. When the extruder temperature reaches 50°C, the cooling fan will be activated and the current temperature value will display to the right as shown below.
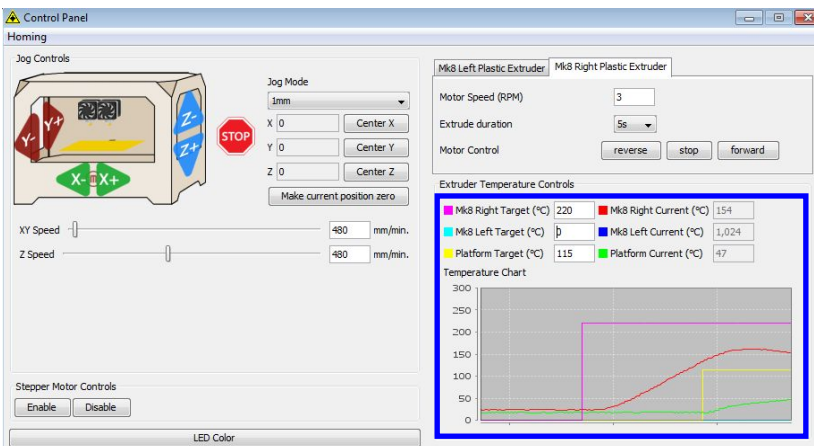
Chapter 6: Filament

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

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

6.1 Installing Filament

First, remove the filament guide tube from the extruder.
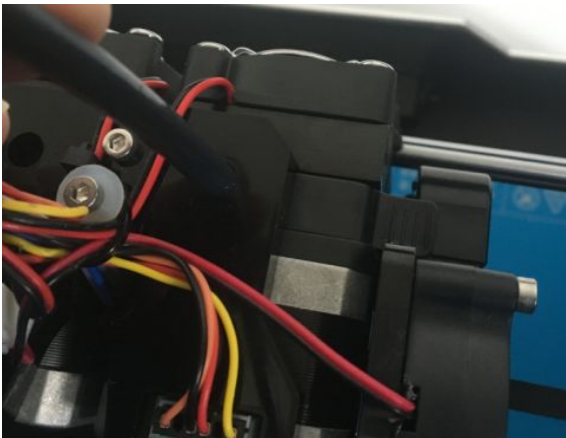
To avoid filament jams, always ensure that any filament spool you mount you mount on the Creator pro feeds from the bottom of the spool toward the top. Filament mounted on the right spool holder [When viewed from the back] should always unspool clockwise and filament loaded onto the left holder should always unspool counterclockwise.

Moreover, when you insert the filament into the filament intake, you should press down the **spring presser** first.
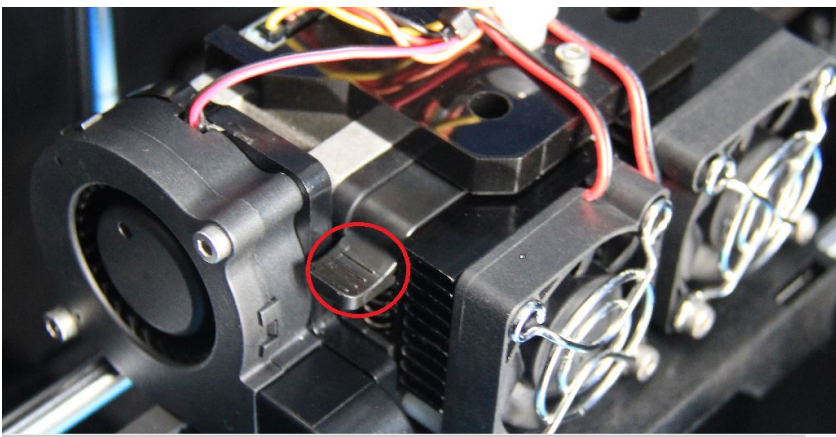
Note: The left extruder is suggested to print with PLA while the right extruder is suggested to print with ABS. Because the left extruder locates beside the turbofan baffle, which is more beneficial for cooling down the PLA models.
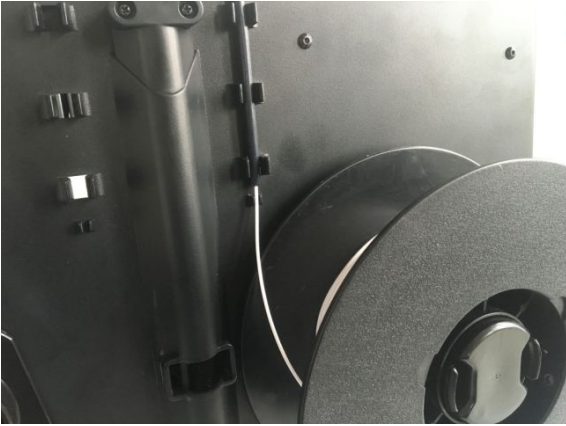
There are two ways of feeding the filament into the extruder head, one is using the LCD screen on the printer itself and the other is by using the control panel on the Replicator G software.

6.2 Feeding the Filament Using the LCD Screen

1. Turn on the Creator Pro; the display will indicate:
   - ▶ Print 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:
   Monitor Mode
   Filament loading
   Preheat Setting
   General Settings

3. Select **Utilities**; press the OK key of the keypad. The display will show:
   Monitor Mode
   - ▶ Filament loading

Preheat Setting

4. Select **Filament loading**; 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** unit 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 Creator Pro, 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 Preheating

    Right Extruder   OFF

    Left Extruder   OFF

    Platform   OFF

3) Press **page down key** to select **Left Extruder (or Right Extruder)**, click **OK key**, you will

see：

    ▶ Start Preheating

    Right Extruder   ON

    Left Extruder   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.
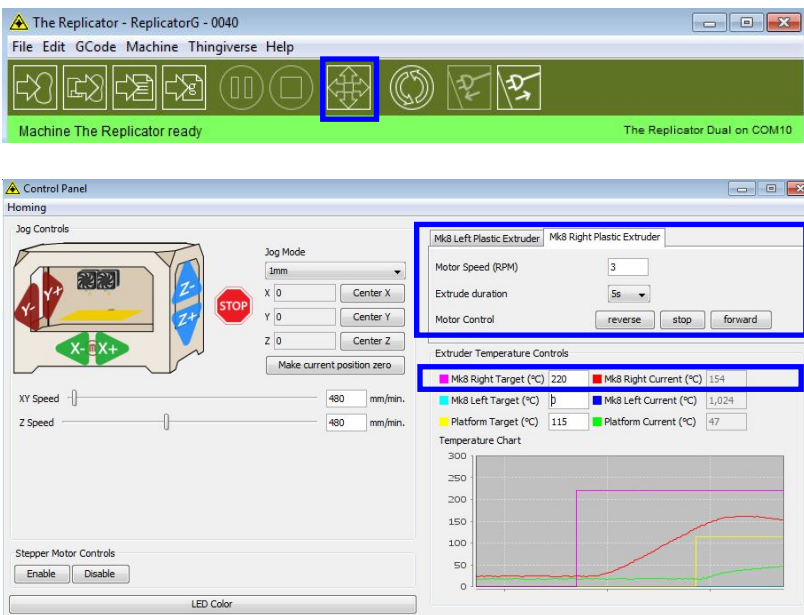
To heat the right extruder and feed material, click the right extruder on the upper right corner of the control panel and manually modify the temperature in the "Right Target" setting for the extruder temperature control changing the temperature to 220°C. A red line will be plotted in the temperature

map. When the actual (Current) temperature reaches 220°C, then click the **Forward** button, so 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**

Chapter 7: First Print

First, make sure that you've completed all the steps in the **Unpacking and Hardware Assemby section**.

> **CAUTION**
>
> ⚠️ Creator Pro 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.

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 Creator Pro.

Every printer will be leveled before shipped out, but we can't ensure that the platform won't move during delivery, so it will be better to level platform before you begin to print. We should tighten the three knobs under the build platform, then put one SD card within the 3D printer，there is a file named **PlateLeveling.x3g.**Put the SD card into the slot,choose **Print from SD ,**click **OK,**then choose **PlateLeveling.x3g**and click it, you will see the content below:**Find the 3 knobs under the build platform and tighten four or five turns. Nozzles are at the right height when you can just slide a sheet of paper between the nozzle and the plate. Grab a sheet of paper to assist us. Please wait. (**During the process, what you should do is only click **OK** continuously, the screen will go to next page.**)**

(1) Then the build plate and the nozzle start moving, once they suspend, you need to adjust the corresponding knob under the build plate individually. Use the leveling card or a thin piece of paper to check the distance between the build plate and the nozzles.

(2).As you adjust the knob, make sure the leveling card just slides between the nozzle and build plate. You should feel some friction on the card but still be able to easily pass the card between the nozzle and build plate.

(3) When you finish leveling the first point, you should repeat the steps above to finish leveling the rest three points.

How to Level the Build Plate

Three knobs lower and raise the build plate

• Tightening the knobs[turning them counterclockwise] moves the build plate away from the extruder nozzles.

• Loosening the knobs[turning them closkwise] moves the build plate away from the extruder

nozzles.

• The distance between the extruder nozzles and the build plate should be about the thickness of the leveling card.

Note: To view a video of the build plate leveling process, please go to the FF Creator pro video page on: https://www.youtube.com/playlist?list=PLWfXP01hqgWsEfzZ5Z-sZzifa05iIZb0Q
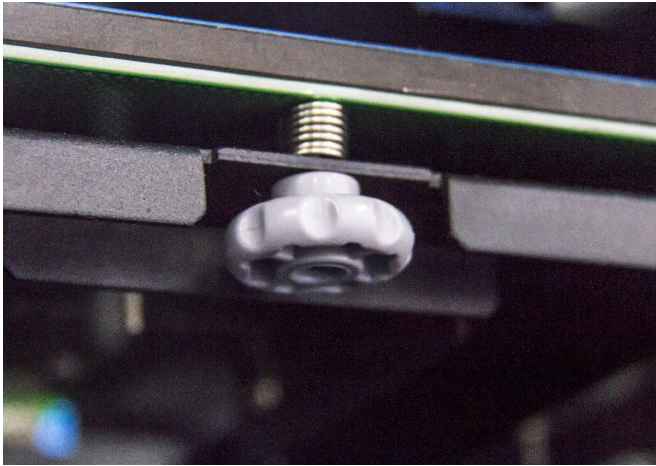
Now, let's start the **first print**. Click **File** > **Examples** and then select **the 20mm_Calibration_Box.stl.** The preview interface will then appear along with a virtual impression of the 20mm cube on the virtual printer bed. On the preview interface click **Move** > **Center** and **Put on platform** so that the sample will be printed on the center of the build plate
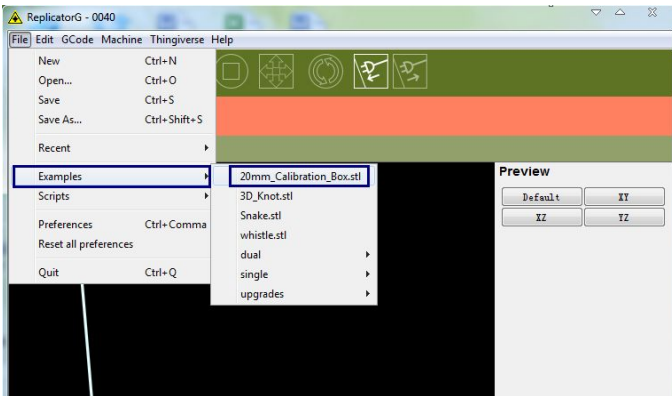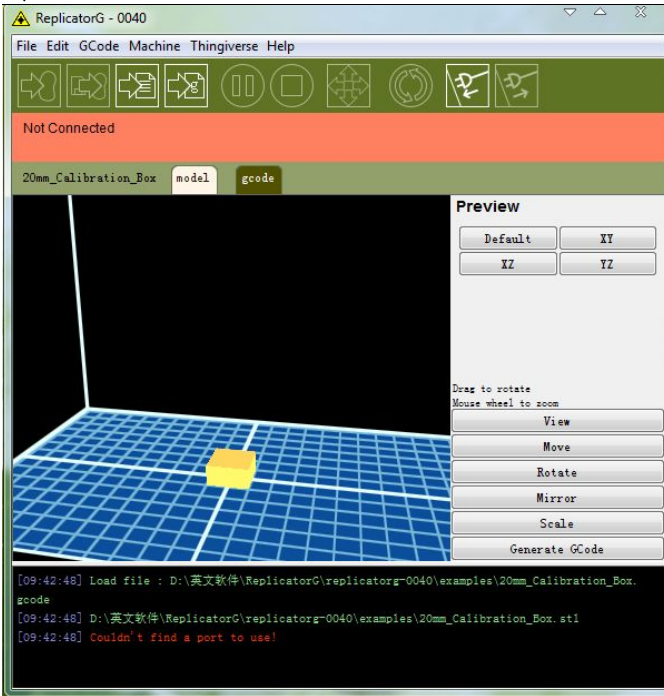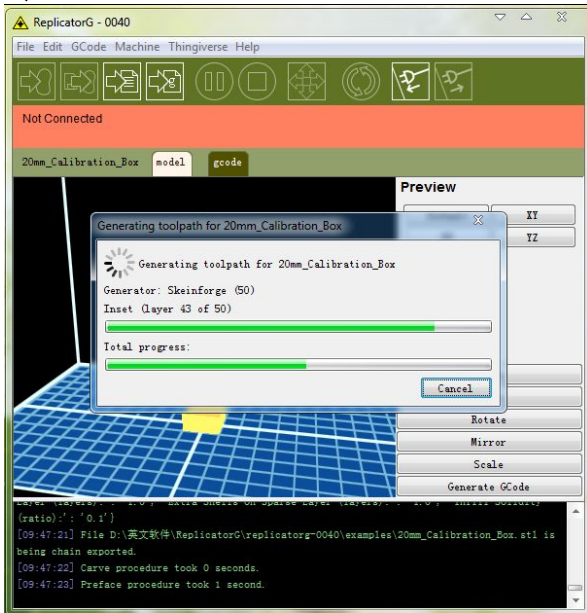
Next click **Generate GCode**, a dialogue box will pop up, after inputting your chosen parameters the Gcode will be generated. Please refer to **EXPLANATION OF GCODE SETTINGS.**
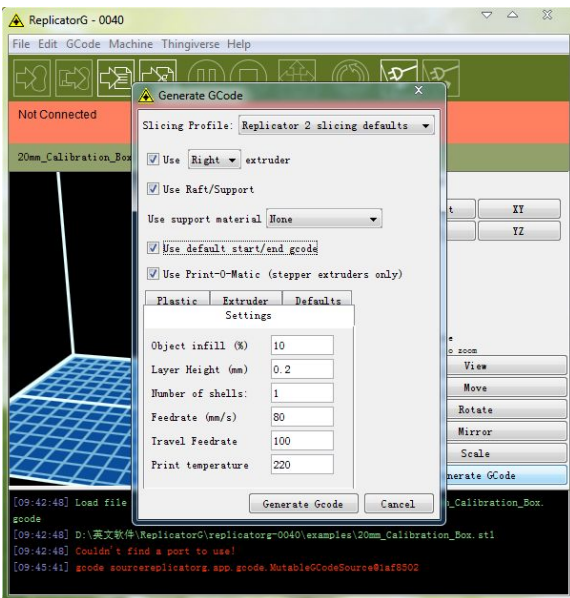

A loading bar will then appear showing the progress of the Gcode generation.

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.
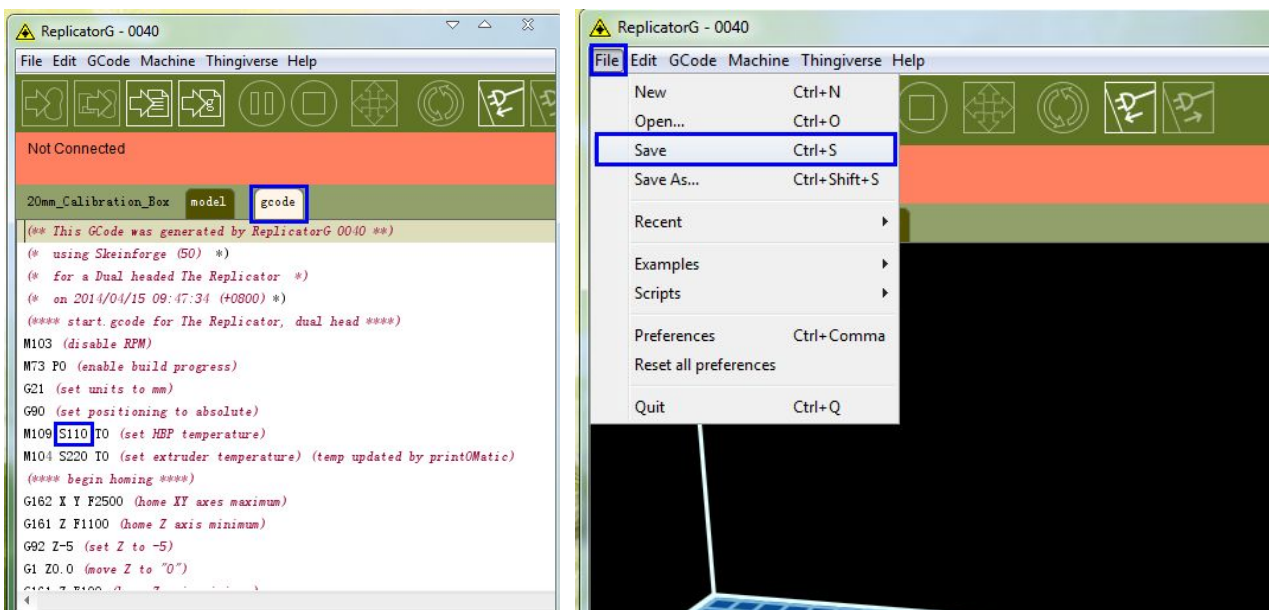

If you have connected Replicator G Creator Pro printer, click the left most button on the green bar near the top of the screen. The Gcode has now been sent to the printer and the object will start printing shortly. If not, please refer to **USB Connection & Temperature Setting** to connect.

The Replicator - ReplicatorG - 0040

File Edit GCode Machine Thingiverse Help
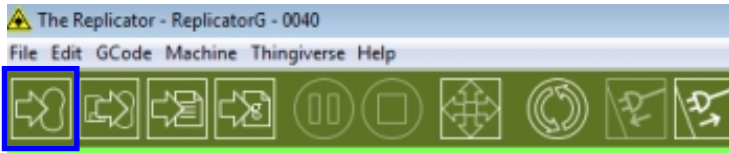
Note：**The printer won't turn on the turbofan automatically if you print the model not sliced by RepliactorG.**

Chapter 8: Dual Extruder Print

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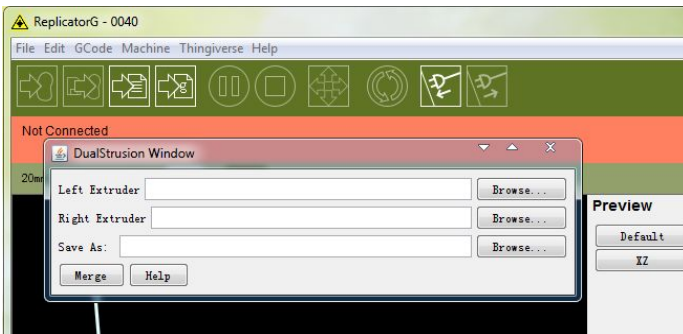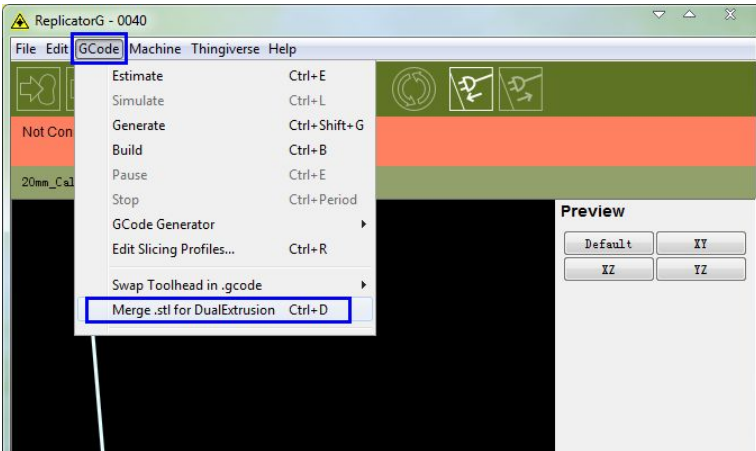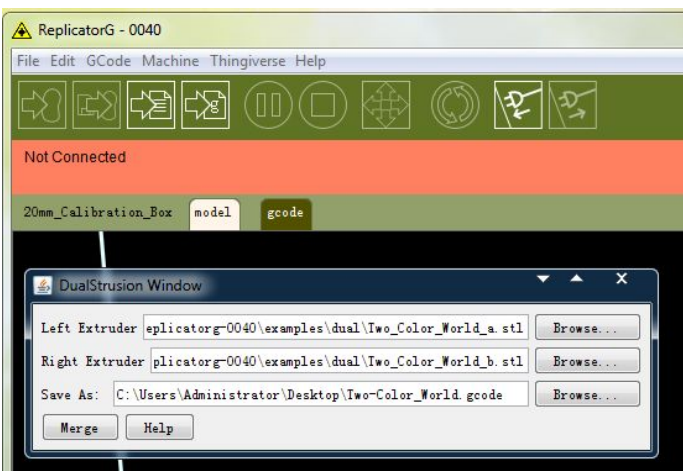

Click on the first **Browse** button to locate and select the file for left extruder; and then click on the second **Browse** button for right extruder. For example, you can find the **examples** folder in its installation path. Open the folder **examples** select **Two_color_world_a.stl for** left extruder 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.

NOTE: Do not check "the 'use raft/support' option. If you want to print dual colors with PLA filament, then perform the following settings:
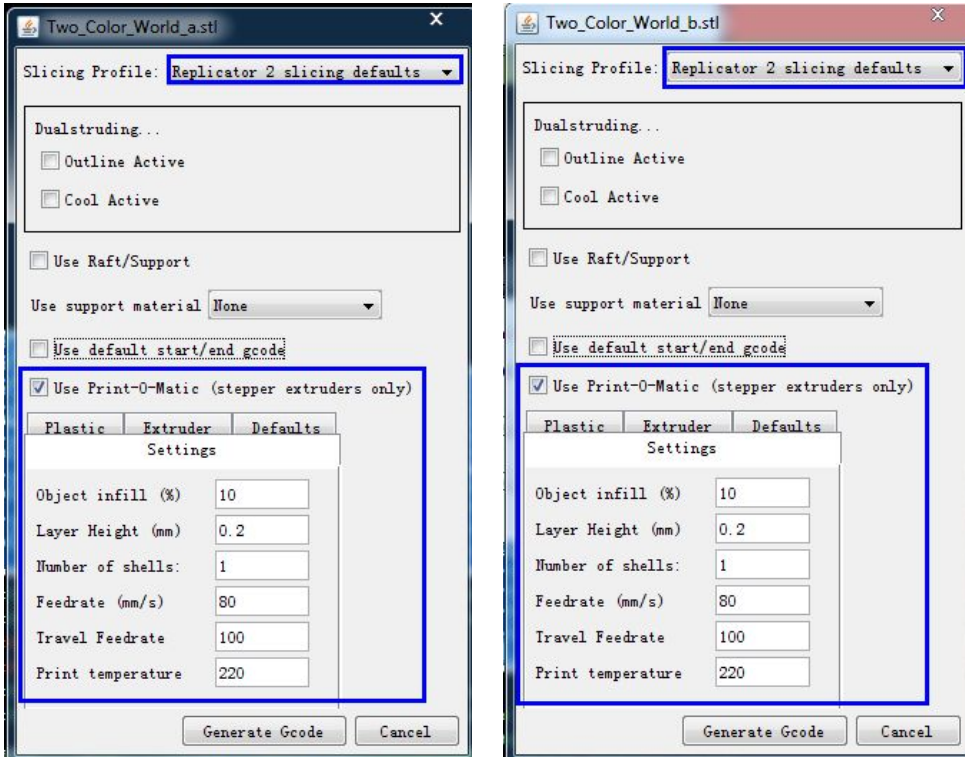

If you want to print dual colors with **ABS filament**, select **Replicator slicing defaults** under the **Slicing Profile** column, and lower the **Feedrate** to **60** and **Travel Feedrate** to **80**.

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**.
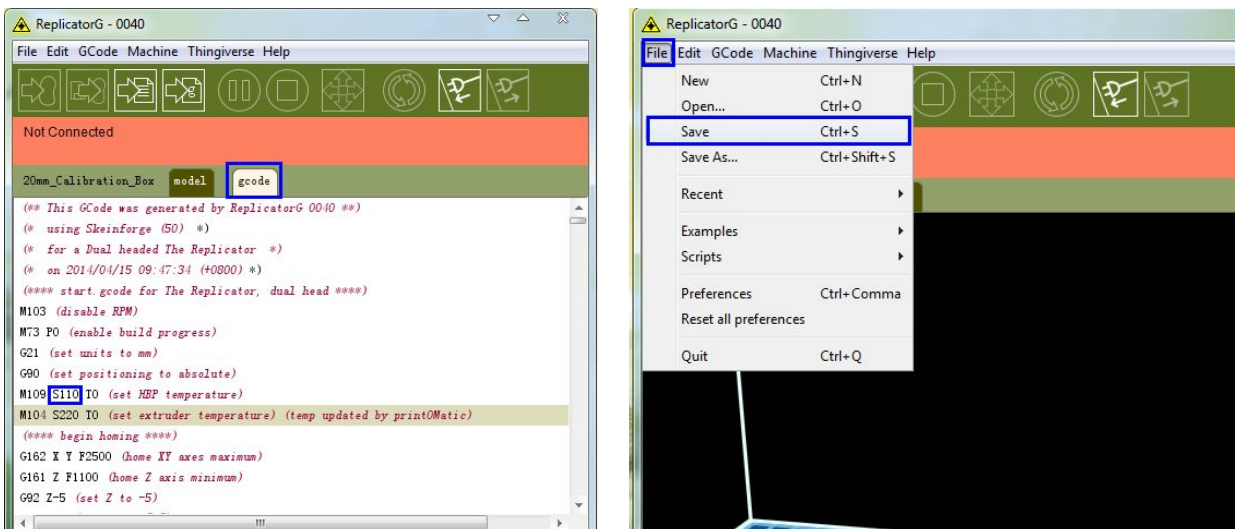
Click on the following button to export **.x3g** file for SD card print.
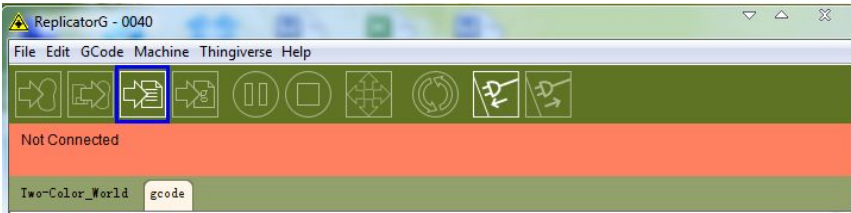
Note: If you meet some problems, you can go to Flashforge official website：www.ff3dp.com for initial help.

Chapter 9: Other Information

9.1.Supports and Service

Flashforge team is on standby and ready to help you with any challenges you may have with your Guider. If the issues or questions are not covered in this User Guide, you can seek for solutions on our office website or contact us via telephone.

There are solutions and instructions to common issues that can be found in our knowledge base. Have a look first as most basic questions are answered there.

http://www.ff3dp.com

The FlashForge support team can be reached by e-mail or phone between the hours of 8:00 a.m. to 5:00 p.m. PST Monday through Saturday. In case you contact us during after-hours, your inquiry will be answered the following business day.

Tel：**86+579-82273989**

ADD：**No. 518, Xianyuan Road, Wucheng, Jinhua, Zhejiang**

*When contacting support, please have your serial number ready. The serial number

is a bar code on the back of your Creator pro."

9.2 References

Name	Creator Pro
Number of Extruder	Dual
Print Technology	Fused Deposition Modeling (FDM)
Screen	LCD
Build Volume	230×150×155mm
Layer Resolution	0.1 - 0.2mm
Build Precision	±0.2mm
Positioning Precision	Z axis 0.0025mm; XY axis 0.011mm
Filament Diameter	1.75mm
Nozzle Diameter	0.4mm
Build Speed	24CC/hr
Software	Replicator G/Makerware
Support Formats	stl/obj/X3G
OS	Windows/Mac OS/Linux
Device Size	480*338*385mm
AC Input	100-240V, 50-60Hz, 300W
Connectivity	USB cable,SD card