



Start Up and Instruction Guide

Creator X Start-up Guide

Contents

Precautions:.....	1
Please always check power setting before plug in.....	1
What's in the box?.....	1
Unboxing.....	2
Initial Hardware Installation.....	3
Special Notes to Win8 users before Software Installation.....	6
Software Installation.....	6
Explanation of GCode Settings.....	7
USB connection & setting the extruder and platform temperature.....	10
Feeding and withdrawing filament.....	14
Setting the filament up.....	15
Feeding using the LCD screen.....	15
Withdrawing using the LCD screen.....	16
Feeding using the Replicator G control panel.....	17
Setting parameters.....	18
Level Build Plate before initial print.....	18
Initial print.....	19
Dual Extruder print.....	21

Precautions:

[★★★ Make sure you read and understand the steps detailed in this guide]

- ! The Creator X is very sensitive to static electricity, so make sure you contact a grounded object before operating the machine.
- ! Before repairing or making any alterations to the Creator X it is essential that the machine is turned off and the power cord is unplugged.
- ! The Creator X operates at very high temperatures; 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 be set up 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.

Please always check power setting before plug in

FlashForge sets the power supply to default to 230V before leaving the factory. If voltage in your location is between 90V to 132V, please switch the power setting from 230V to 115V before you plug in the power. You will find the power source at the bottom of The Creator X.

The red box is where the power setting is located:



What's in the box?

Along with your Creator X 3D printer, this package contains the following:

In two long boxes on the top of the package, you will find:

- Power cable,
- USB A to B cable,

- 2 filament guide tubes,
- 2 spool holders.

The accessory box on top of the machine includes:

- 2 extruder heads,
- SD card,
- Bolt, Hex wrench and screws kit.

Under the build platform in the Creator X framework, there are:

- 2 spools of filament

Un-boxing

The Creator X was carefully packed by our staff at the FlashForge factory. Please follow the un-boxing steps laid out below.

- ! 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 which helps to keep the nozzle at a constant temperature.

First, put the box on the floor in a clean and flat area. Remove the top two long boxes, set them aside and then pull out the cardboard packing that encloses the Creator X. Inside the two long boxes, you will find a power cable, USB A to B cable, 2 filament guide tubes, and 2 spool holders.



Now you can see the top of the printer, as well as some other boxes inside it. The large box with the black wire is the **accessory box**; this contains the **extruder(s)**, **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 damage it.

Now take the Creator X out of the box. Gently lift and transfer the printer to your work surface. You will find the **extruder** in the protective packaging along with the black cable, carefully remove it and place it on your work surface.



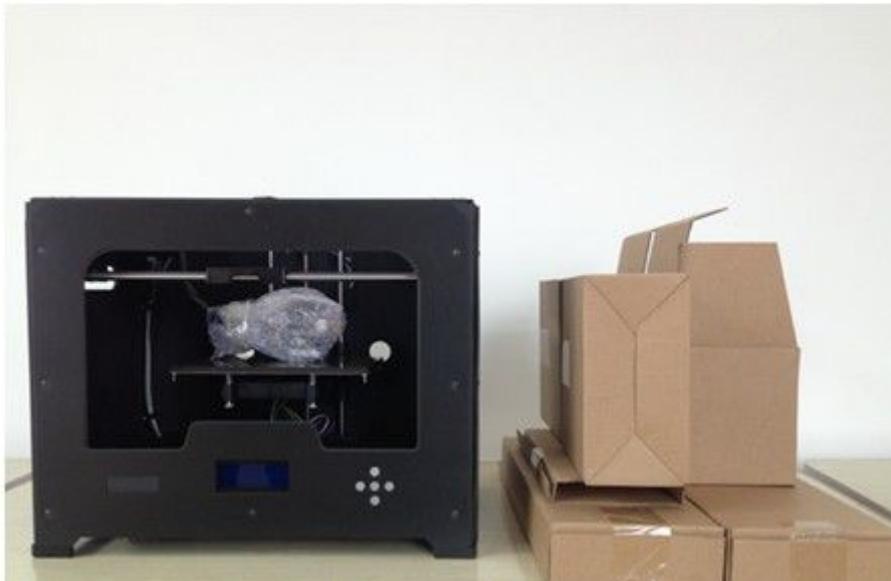
Remove the cardboard packing material and take the **accessory box** from the printer, set it aside for later.

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. **Remember: Do not remove the film.**

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

- 1) Turn the screw which is behind the rotating platform.
- 2) Grasp the printing platform with one hand on each side, 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, you'll find the filament here; it's easiest to remove the filament by setting aside the remaining packaging material. After that, cut off the ties between the X axis and front metal frame.



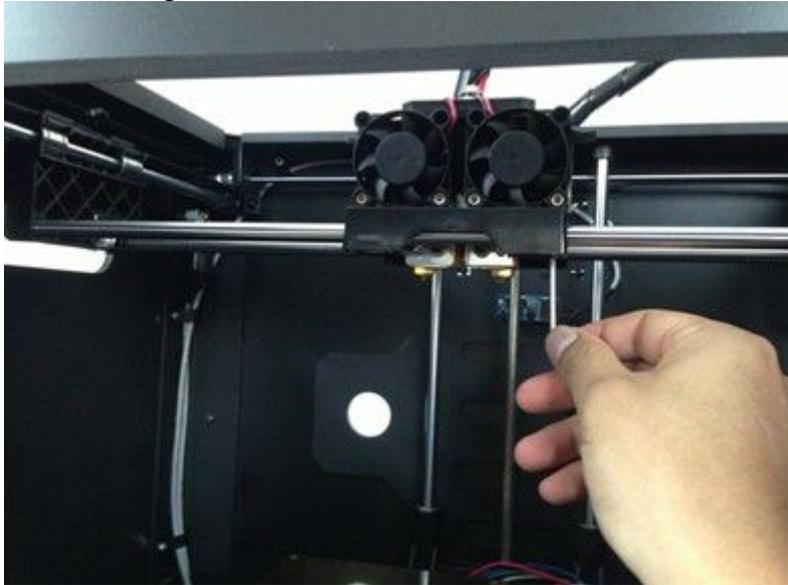
You have now finished un-boxing! The next task is to set up the hardware.

Initial Hardware Installation

Start by installing the extruder. You'll need the two shortest silver screws from the kit found in the

accessory box, and the appropriate hex wrench.

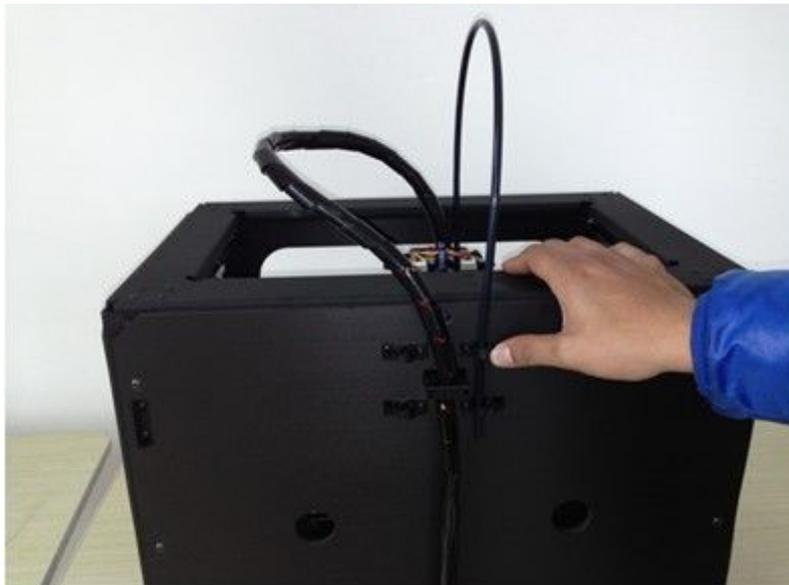
First, lower the build platform using one of the methods described in the previous section. Holding the extruder by both sides, take it out of the accessory sleeve and position it on the extruder seat with the fan facing forward. Align the screw holes and fasten with the two short silver screws.



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, put 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. *****Remember to verify your power supply is set to the proper voltage for your location.**



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

Finally take the filament out of the box, install it on the **spool holder** and screw in the nut, however the nut should not be over tightened.



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

Special Notes to Win8 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=NM1MN8QZhmk>

Software Installation

ReplicatorG0040 is the best software to use with the dual extruder Creator X. It can be downloaded from its official website at: <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 Creator X.

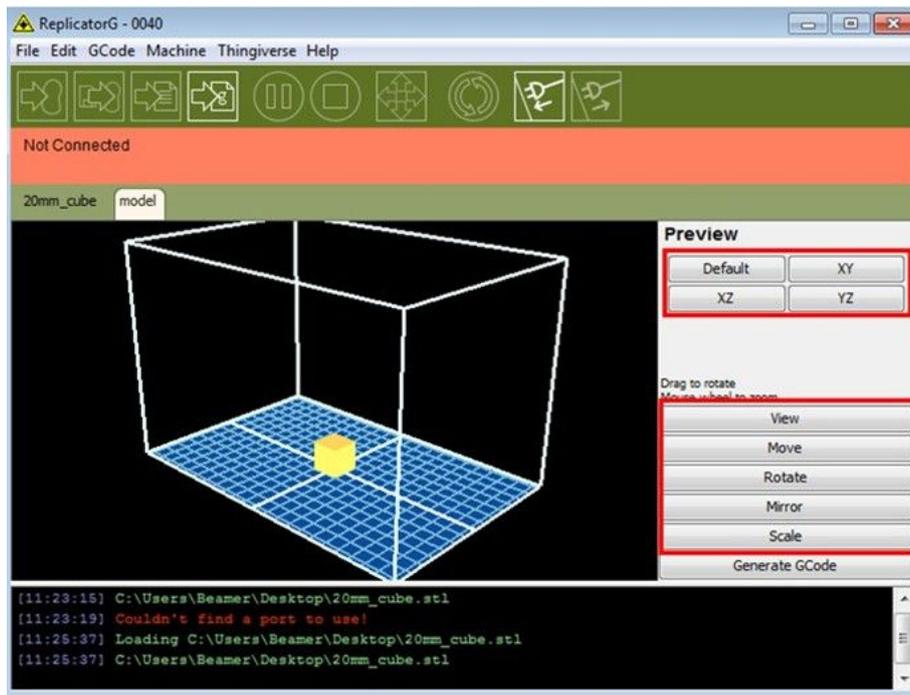
Steps for installing your software:

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 Replicator G software.
4. To run the Replicator G software, double click the shortcut on your desktop or in your start menu.



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

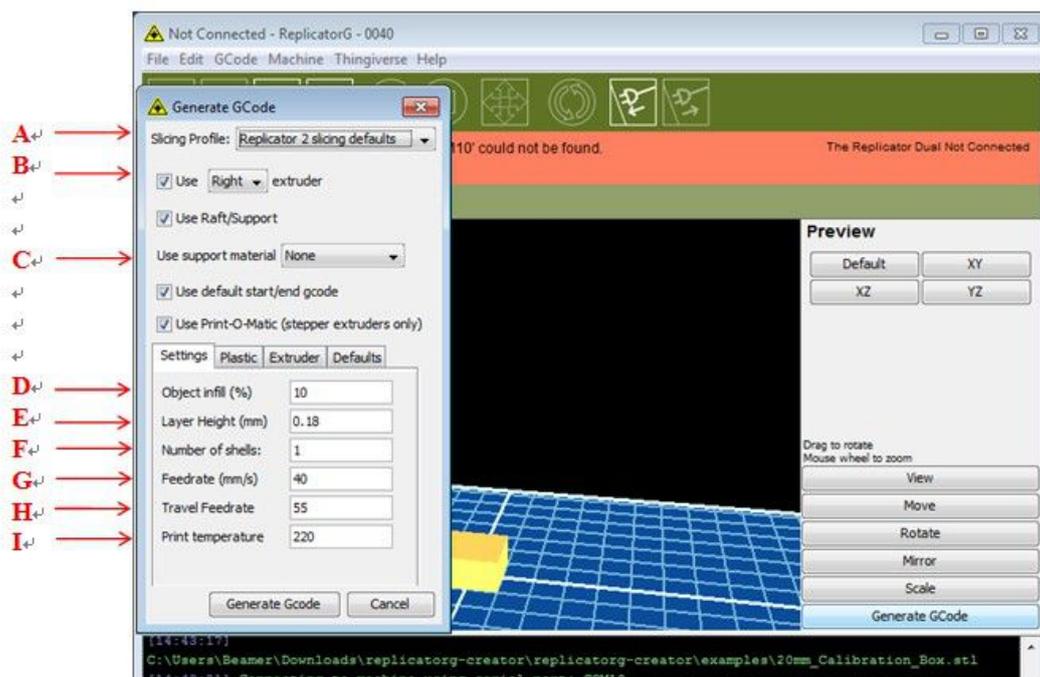
Click **File > Open**, then browse and select the file (.STL) that you would like to print, import the file by double-clicking. Then the drawing will appear on the Replicator G interface.



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 red boxes 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 open up, giving you several options on how the Gcode will be generated:

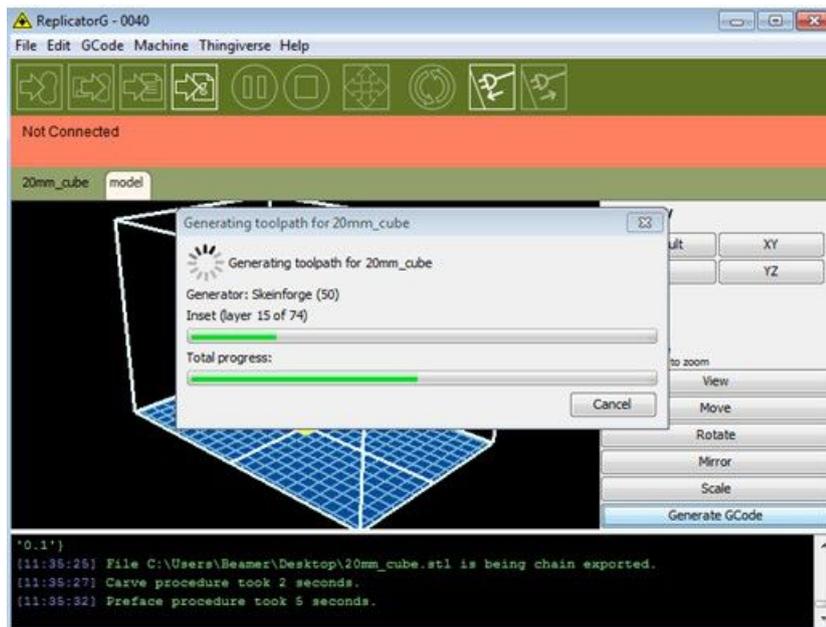


A. Slicing profile: select **Replicator** slicing defaults for ABS printing, and select **Replicator**

2 slicing defaults for PLA printing.

- B.** This tells the printer which extruder to use for a dual-extruder head printer, either the **left** head or the **right** head can be selected.
- C.** If your sample will have any hanging surfaces, it is recommended to have support. **None** means 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 **0.20mm**.
- 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. For ABS, set it to **220**. For PLA, set it to **200**.

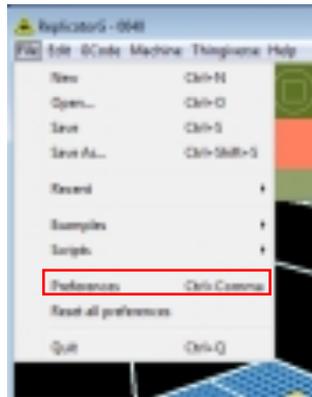
Click **Generate Gcode** and a progress bar will appear.



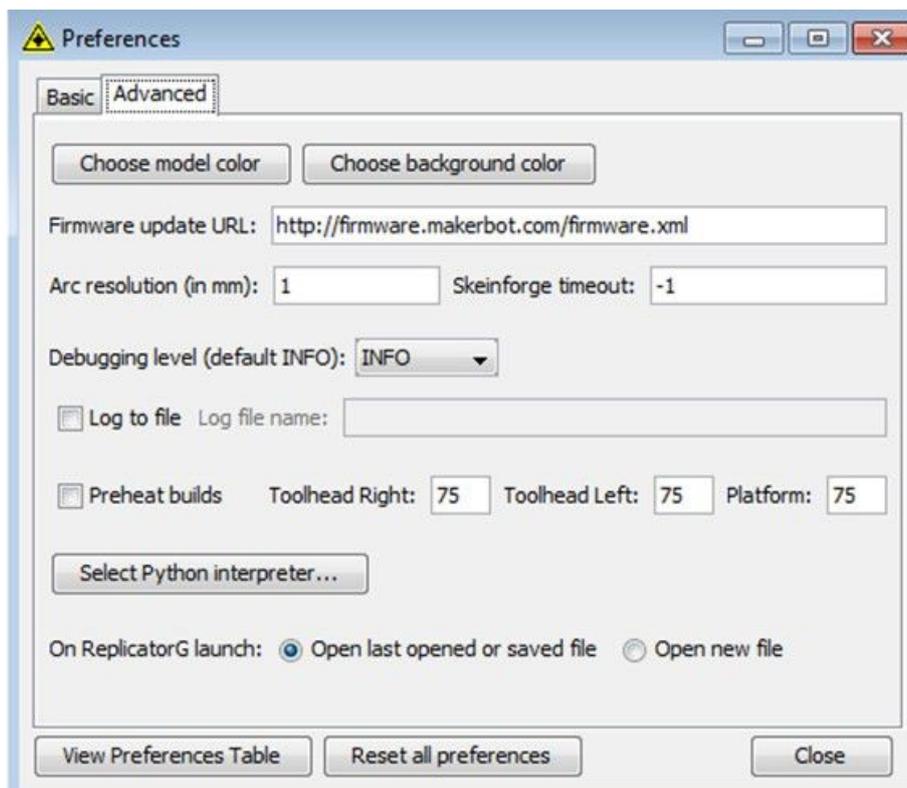
NOTE: For users who did not choose the default installation path in the installation of Python, clicking the **Generate Gcode** button in Replicator G, will result in dialogue box popping up alerting that the executable Python file cannot be found.

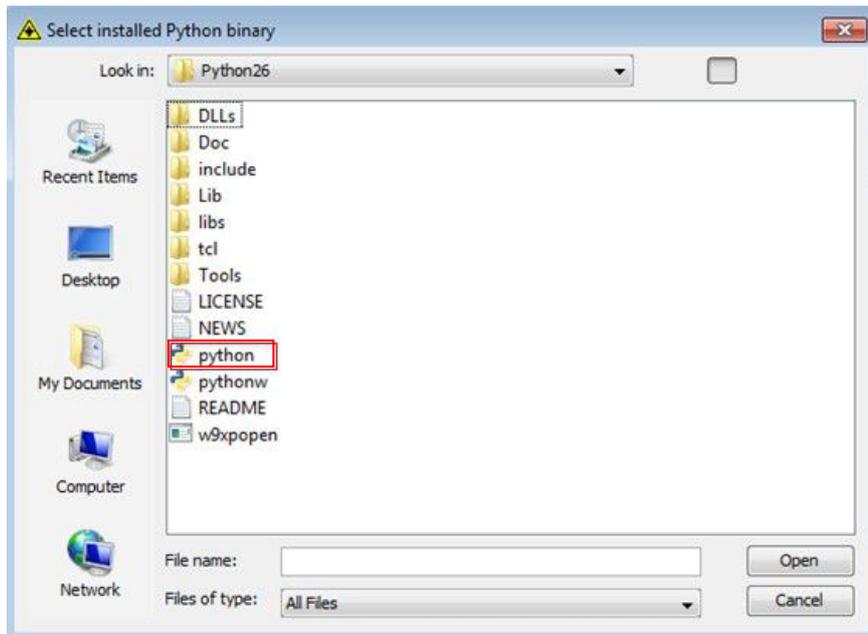


First click the ‘No’ button to dismiss the dialogue box. To solve this problem we need to configure the corresponding menu.



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





Click ‘Close’ on the Preferences menu and you are done!

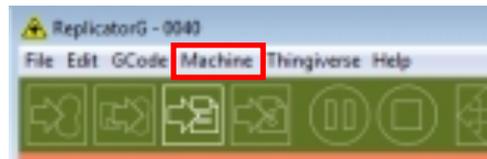
The machine will now work as normal when generating Gcode. Now we will start a preliminary test on the machines connections and we will heat the platform and extruder ready for printing.

USB connection & setting the extruder and platform temperature

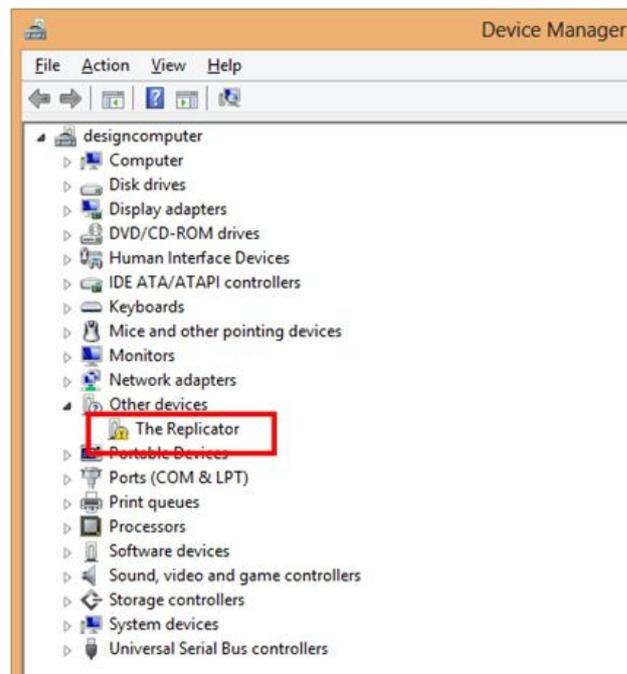
First, connect the machine and computer with the provided **USB A to B** cable.



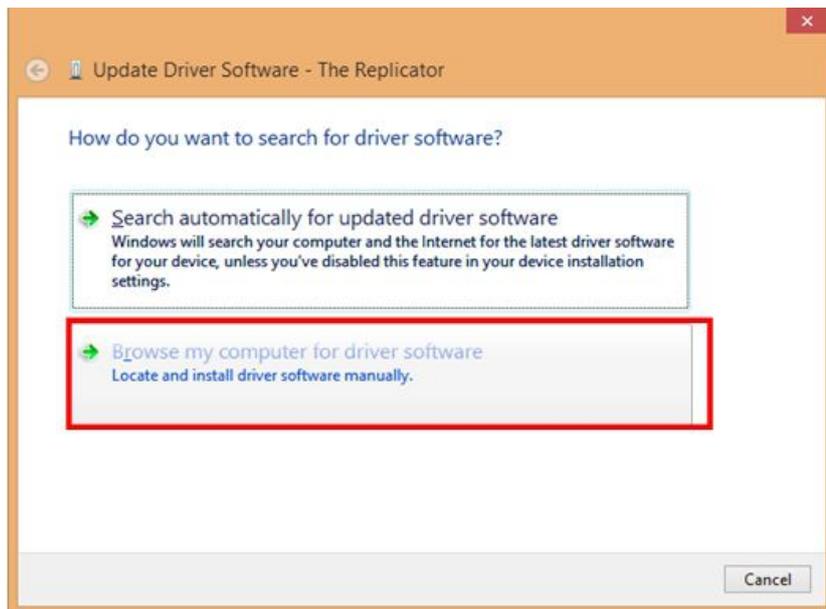
The USB port on the machine is bound by the red box in the above image. After connecting the cable, open the Replicator G software, we are going to connect the computer and printer.



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, click 'My computer' then right click properties, the basic system parameters appear. Then select Device Manager.



Locate the software driver shown in the red box, right click and select **Update Driver Software**.

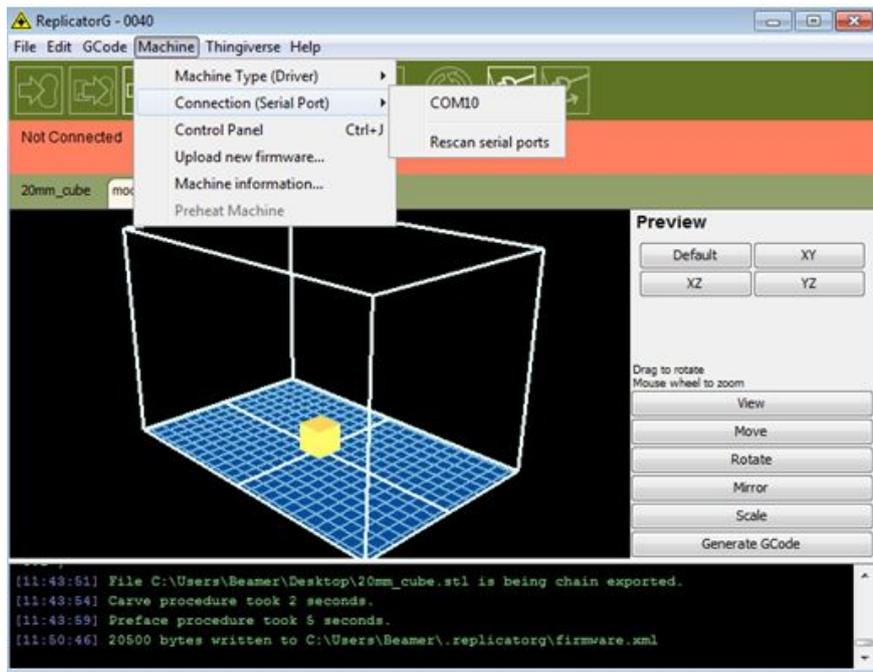


Click **Browse...** to find the location of ReplicatorG0040 on your system.



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

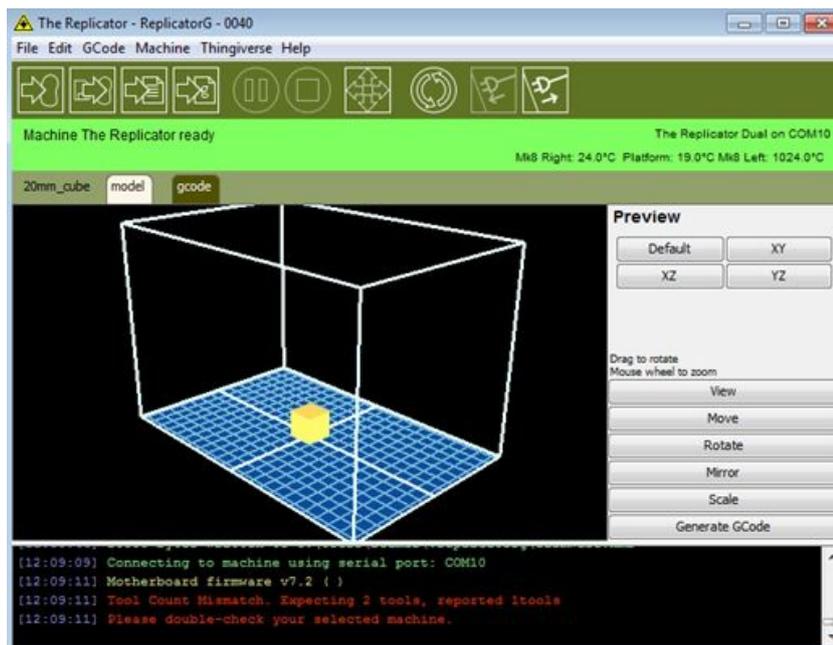
The next step is to connect the printer. Rescan the serial ports and select the one that appears on your machine (on our test machine the port was COM10, but different computers vary).



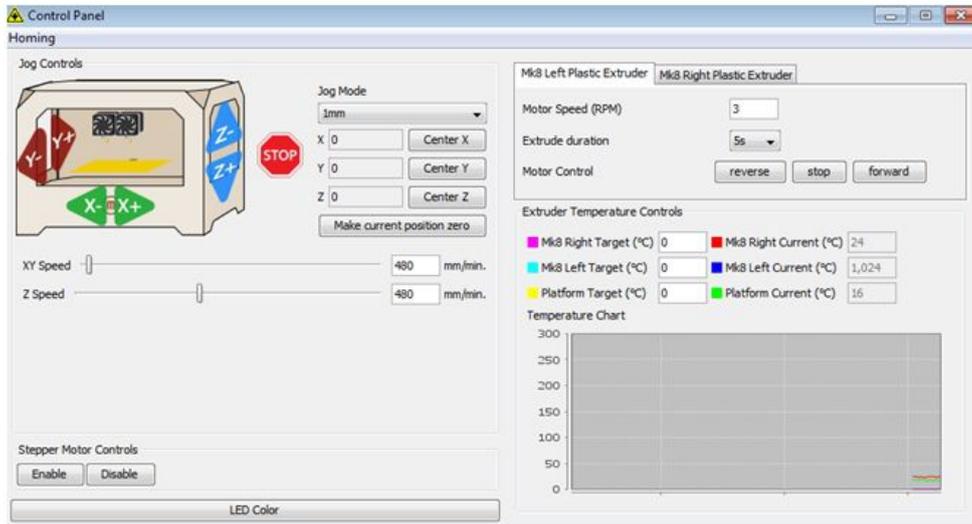
Now we can connect the machine, select the function key in the red box below:



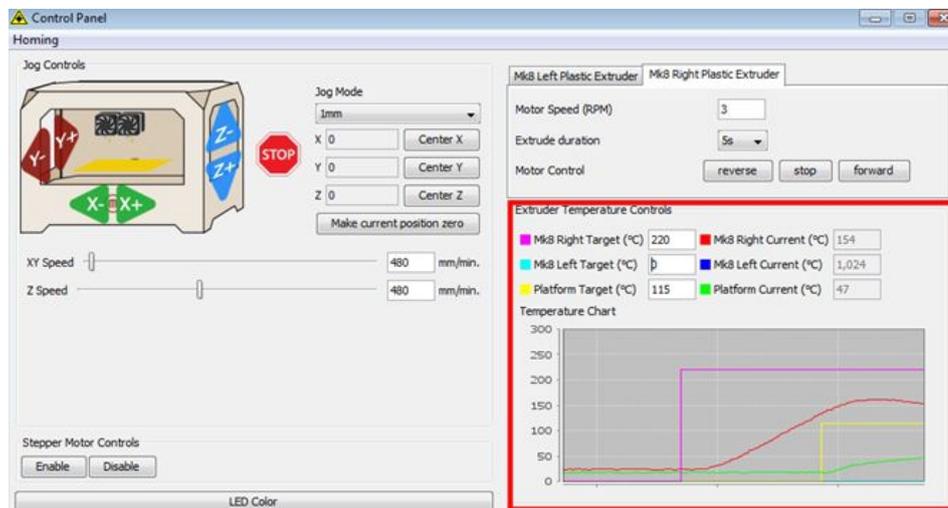
If the red area turns green it means the printer is connected with the computer. Now we need to heat the extruder and build platform.



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



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

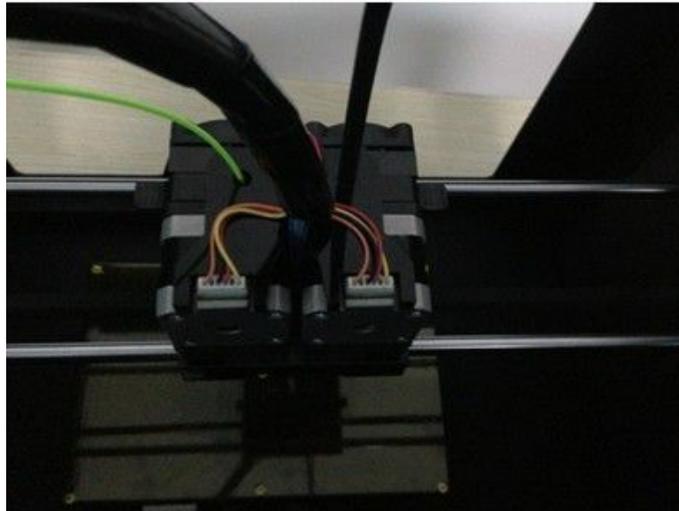


Feeding and withdrawing filament

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

Setting the filament up

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.

Feeding using the LCD screen

1. Start the machine, the display indicates:

- ▶Build from SD
- Preheat
- Utilities

- Using the directional arrows to the right of the screen click the page down key to scroll to the next page. The display will show:

- Preheat
- Utilities
- Info and Settings

- Select **Utilities**; click the **ok** 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

- Select **Change filament**; click the **ok** key in the middle of the button board. The display will show:

- Load Right
- Unload Right
- Load Left
- Unload Left

- Select **Load right**. Click the **ok** key on the button board. 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 220 degrees, click the **ok** key on the keypad, the nozzle should start extruding material, if not, keep clicking the **ok** key until it does.

Withdrawing using the LCD screen

Notes regarding withdrawing filament!!!

A. If you just finished your printing or feeding the filament and want to withdraw the filament, at this moment, the extruder is still over 200 degrees, first push the filament in a little, and then directly pull it out.

B. If you want to change another color filament, please firstly do as following shows:

- Turn on the power switch, you will see the LCD panel shows:

- Build from SD
- Preheat
- Utilities

- Using the directional arrows to the right of the screen click the page down key to scroll to the next page, and select **Preheat**, Click **ok** key in the middle, then you will see:

- Start Preheat
- Right Tool OFF
- Left Tool OFF
- Platform OFF

3) Click **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) Click **page up** key back to **Start Preheat**, and click **ok** key, you will see:

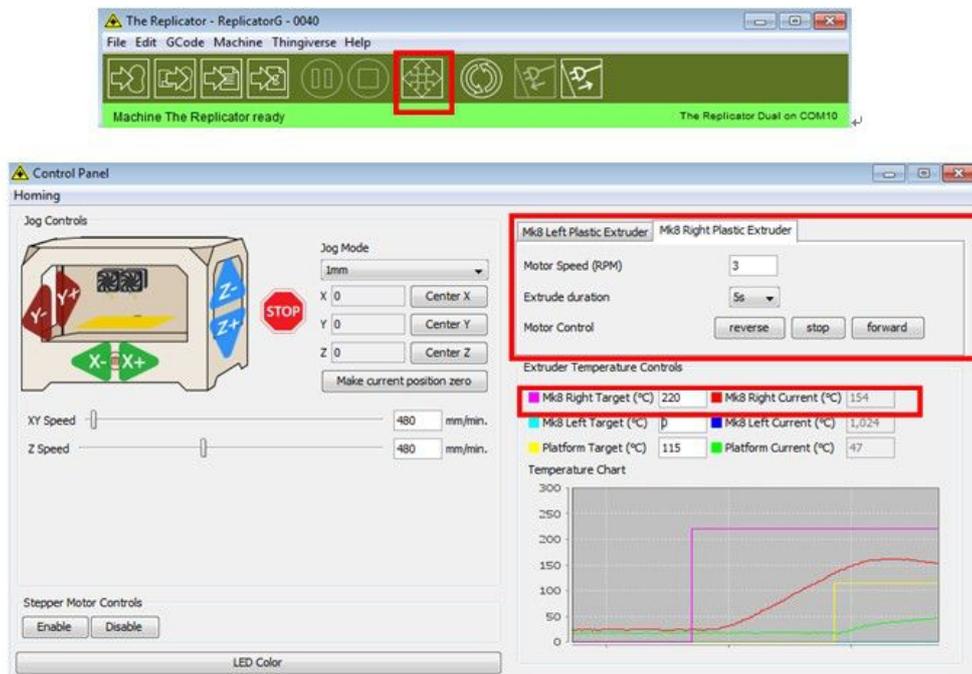
Heating: ■ ■
R Extruder: 033/230C
L Extruder: 033/230C
Platform: 024

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 gotten rid of the filament inside the nozzle successfully.

Feeding using the 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 degrees. A red line will be plotted in the temperature map. When the actual (Current) temperature reaches 220 degrees you can pull the filament out.

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 a [previous section](#).

Level Build Plate before initial print

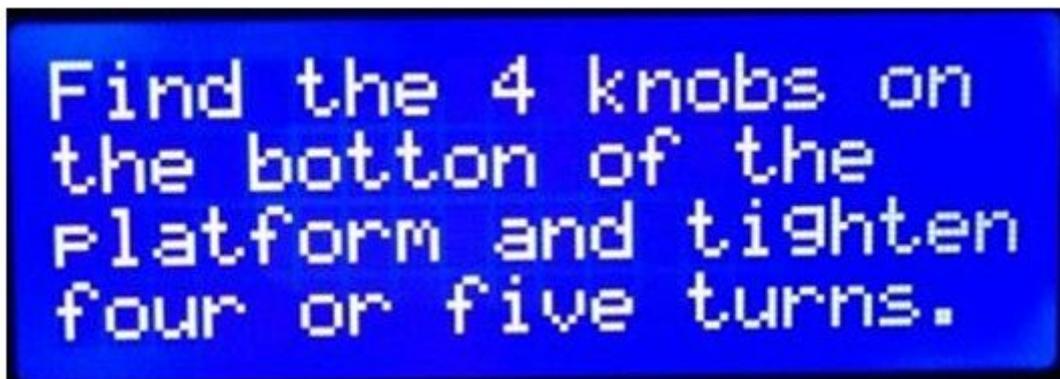
First, make sure that you’ve completed all the steps in the [Un-boxing and Hardware Setup section](#):

1. your DualExtruder should be bolted in place,
2. your filament guide tubes connected,
3. and your spools of filament mounted on the spool holders.
4. Once all these have been completed, you can plug in your power supply.

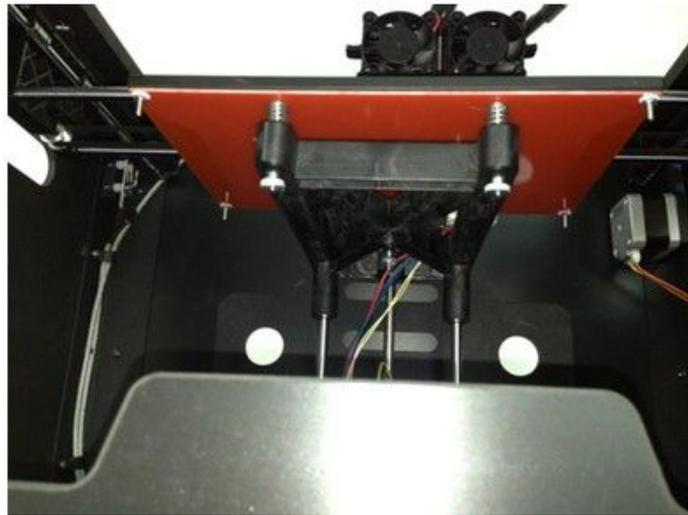
If everything is ready, then flip on the power button on the back of your Creator X.

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



Actually Creator X has only three knobs on the bottom of the platform, just inside the plastic arms that hold it up. Creator X has been leveled perfectly before leaving factory. However, it will get unlevelled during long distance shipping. Please do level the build plate again before starting a print.



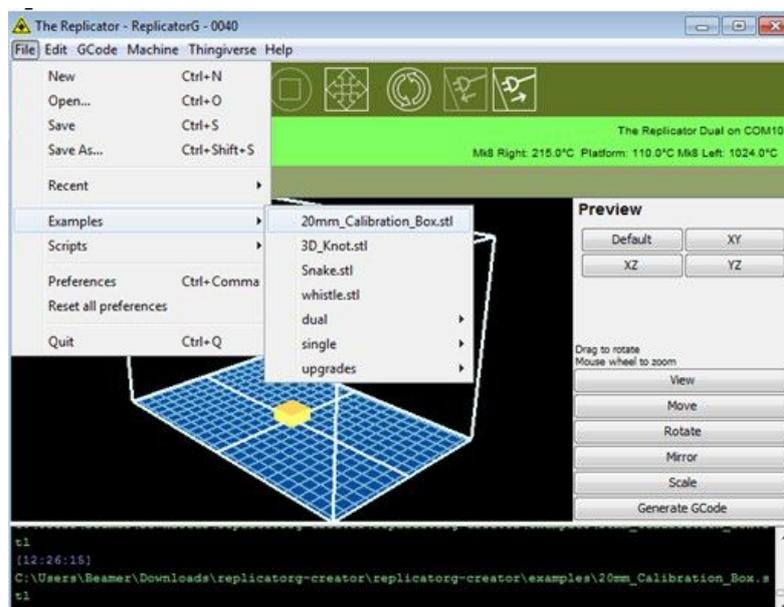
At each stage of the script, your Creator X will ask you to test the platform height by sliding a piece of paper between the platform and nozzle.

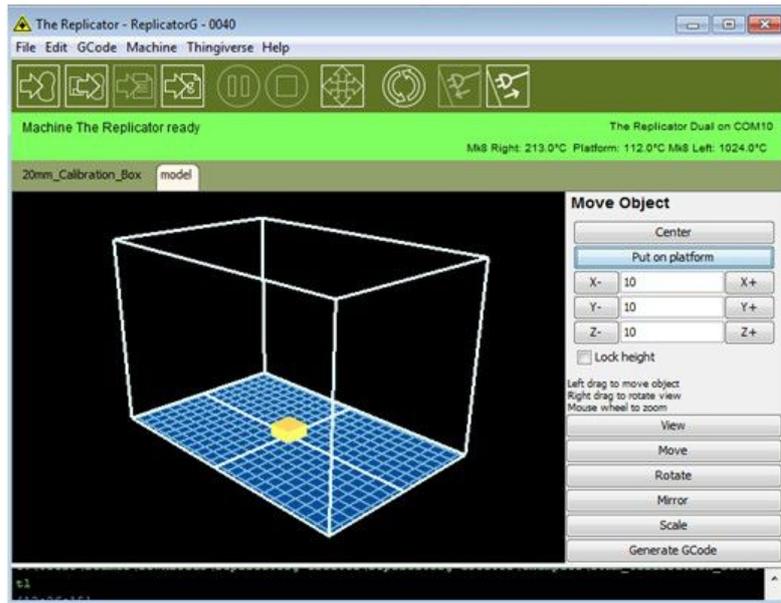
Use a sticky note, sticky side up, to test the platform height one last time. If the note can back and forth freely underneath the nozzle, your platform is too low; if it can slide around a little bit with some resistance, you're ready to go.

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](#).

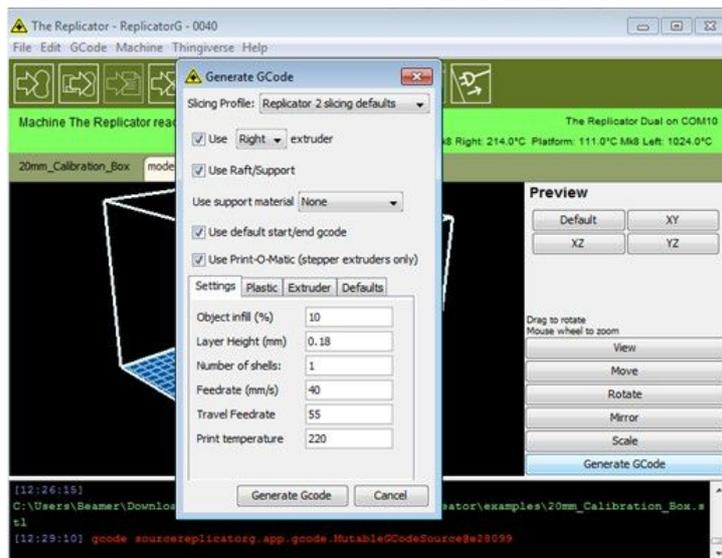
Initial print

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 > Centre** and **Put on platform** so that the sample will be printed on the centre of the build plate.

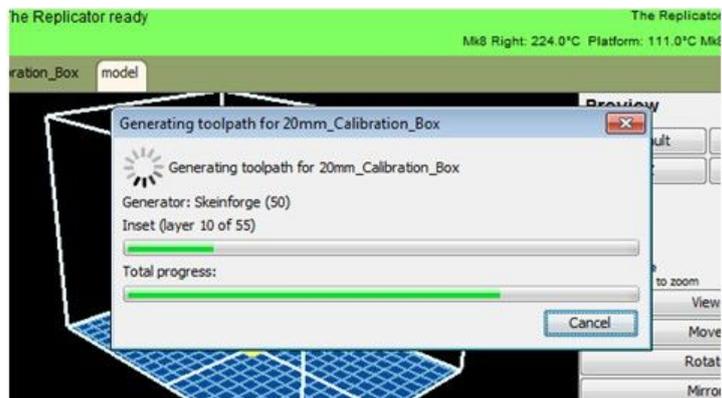




Next click **Generate GCode**, a dialogue box will pop up, after inputting your chosen parameters the Gcode will be generated.

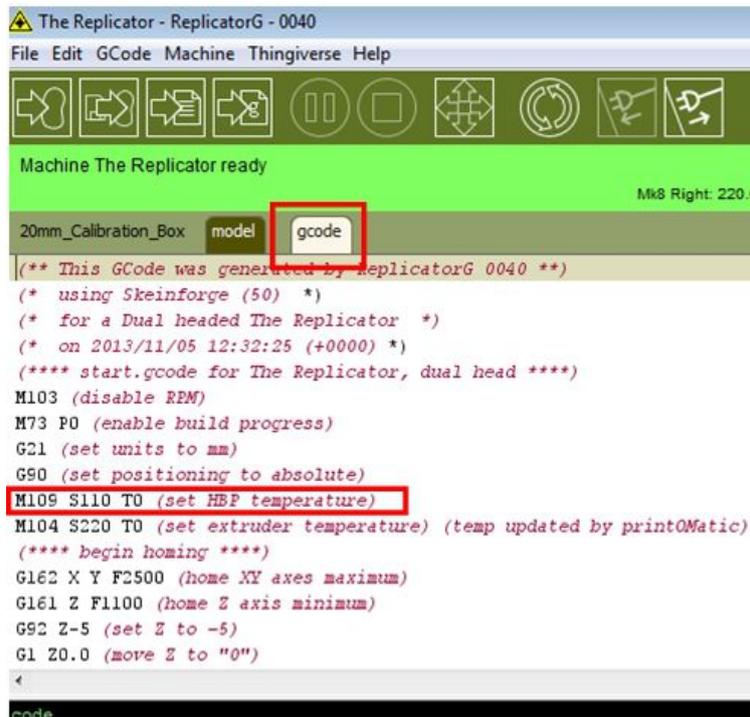


A loading bar will then appear showing the progress of the Gcode generation.

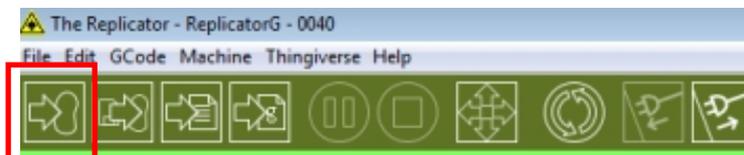


After the Gcode is generated, the temperature of the bottom plate needs to be changed, see below

for the required modification. First select Gcode in the red box shown below and then change **M109 S110 T0** to **M109 S115 T0**.



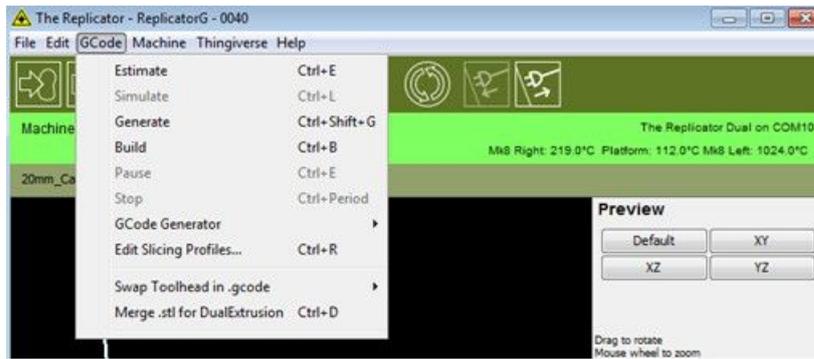
After modifying, click the file button to save the sample and click the left most button on the green bar near the top of the screen.



The Gcode has now been sent to the printer and the object will start printing shortly.

Dual Extruder print

To use the two-color printing that comes with the Replicator G software, open it up and select **Gcode > Merge .stl for Dual Extrusion**.



The following dialogue box will then appear:



Click browse for left extruder; locate the folder 'examples' found in its installation path. (In my case, it is <C:\Documents and Settings\Administrator\My Documents\ReplicatorG\replicatorg-0040>, different computers vary.) Then open the folder 'examples' select **Two_color_world_a.stl** as shown below:

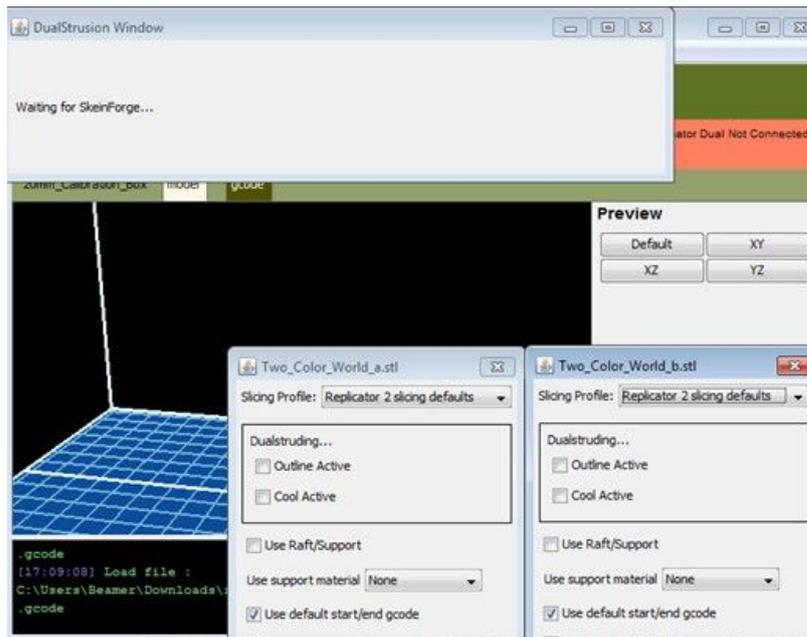


Repeat the process for the right extruder and select **Two_color_world_b.stl** as shown below:



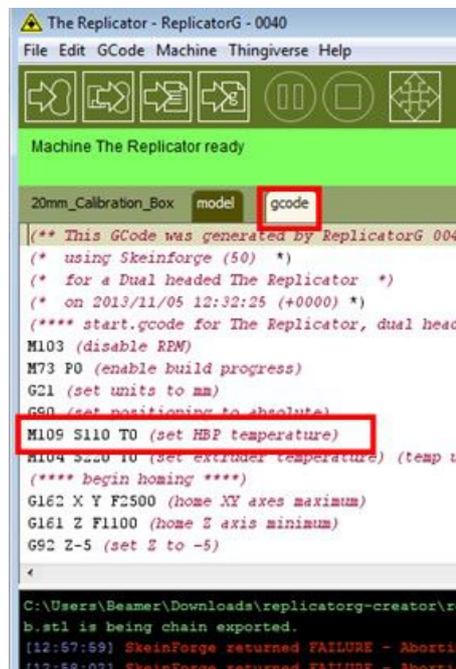
Now you need to save it to the desktop with the suffix set to ".gcode."

Now 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 and then click generate the Gcode for both boxes.

After the Gcode is generated, click the Gcode in the red box and modify the temperature of the bottom plate like before.



After making the modifications to the Gcode save the file and click the build button.

