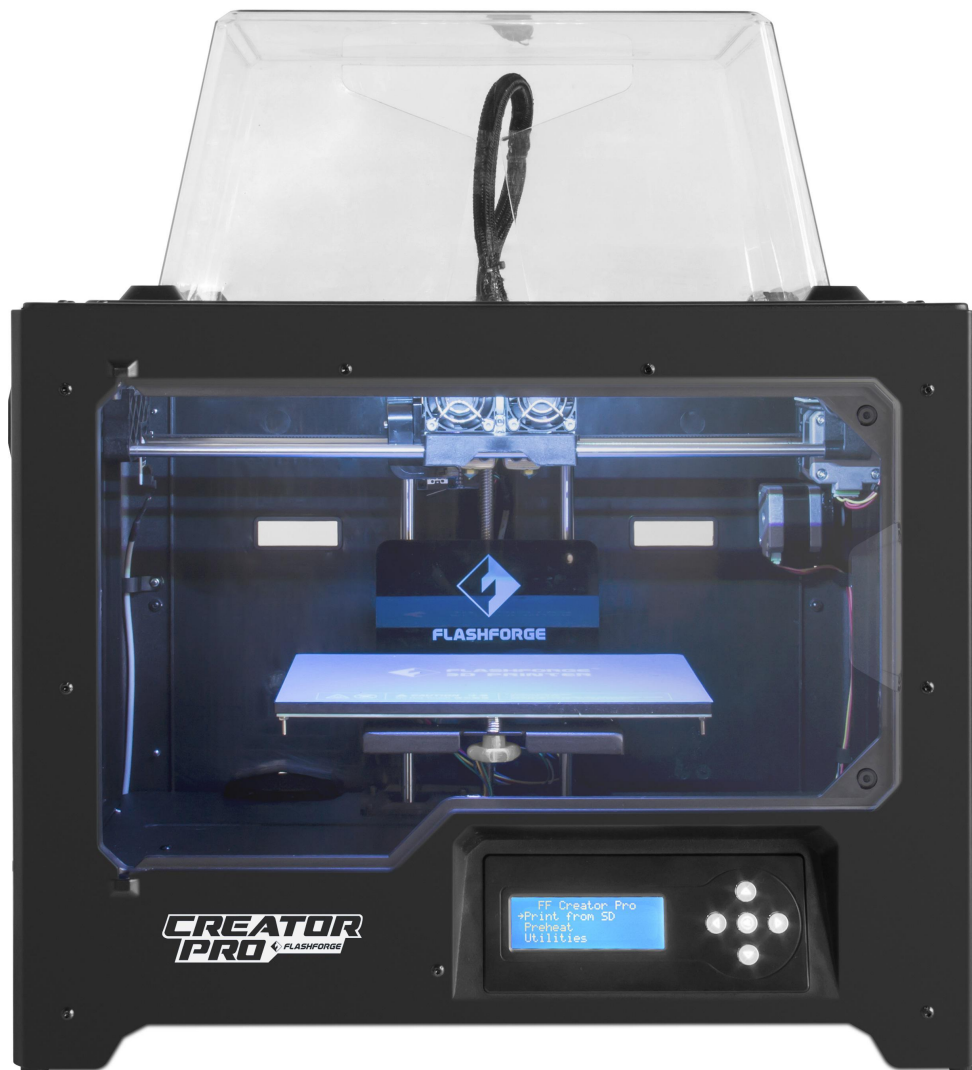


You can find electronic version of this manual on the included SD card



Start Up and Instruction Guide

Seller information

Seller Name: FlashForge 3D Printer

Amazon Storefront: www.amazon.com/shops/A8RK4QCP0IFCY

Support Contact: Mr. Tang

E-mail: **597891915@QQ.COM**

FlashForge Creator Pro Start-up Guide

Content

Precautions:.....	1
What's in the box?.....	1
Un-boxing.....	2
Initial Hardware Installation.....	6
Special Notes to Win 8, Win 8.1 and Win 10 users.....	8
Software Installation.....	9
Recommended GCode Settings for ABS and PLA.....	10
How does the side cooling fan work?.....	11
USB connection.....	13
Loading and unloading filament.....	16
Setting the filament up.....	16
Filament Loading.....	17
Filament Unloading.....	17
Level Build Plate before initial print.....	18
Print from SD card.....	20
Special note to PLA printing.....	20
Initial print over USB.....	20
Dual Extruder print.....	22
Firmware Update.....	25

Precautions:

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

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

What's in the box?

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

In the foam sheet (we call it **Foam A**) on the top of the package, you will find:

- 2 filament guide tubes
- 2 sheets of 3M platform sticker (spare parts) and Leveling card

Below the foam sheet, you will see the Creator Pro protected by a plastic bag. On the top of the Creator Pro, you will find extruder foam sheet (**Foam B**), in it you will find:

- 2 extruder heads
- 2 spool holders
- 4GB SD card (Contains sample models, software and latest operation manual).
- Tool bag (including 2 M3×8 screws, 3 hex wrenches in different sizes, 1 spare leveling knob, 2 spare Teflon tubes)
- Extruder's accessory kit (Contains Turbo fan duct, 2 M3×8 screws, M3×6 screw)
- Power cable
- USB cable

Under the extruder foam sheet, you will find another foam sheet (**Form C**) seated in the top lid. In this foam sheet you will find two ABS spools of filament. (Colors are randomly selected)

Un-boxing

The Creator Pro 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 wrapping around the nozzle. It consists of a ceramic fiber 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 **Foam A**, set it aside. Inside the **Foam A**, you will find 2 filament guide tubes, 2 spare platform stickers, and the leveling card.



Take the Creator Pro out from the box; Put it on your work place.



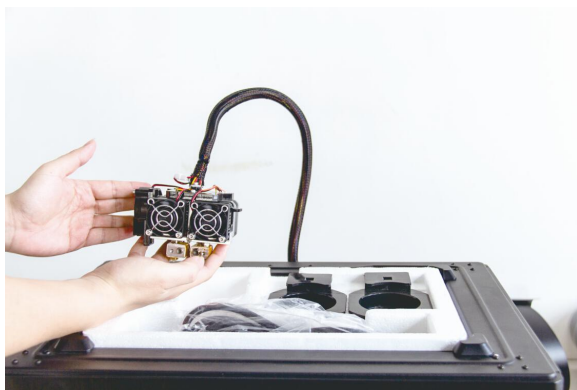
Turn the printer over and remove the protective bag from the bottom.



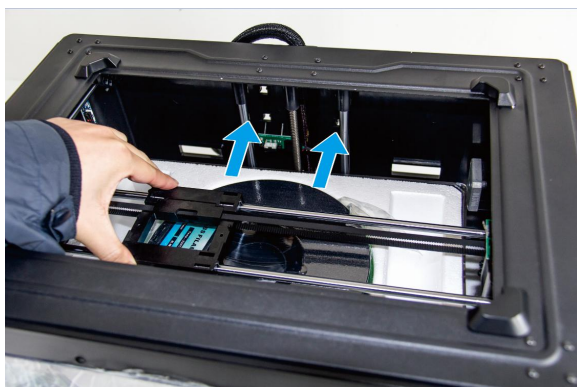
Now you can see **Foam B**. In **Foam B** you will find the extruders, Spool holders, SD card, tool bag, Extruder's accessory kit, Power cable and USB cable.



Take the extruders out and place it on your work surface. Then take **Foam B** out and set it aside.



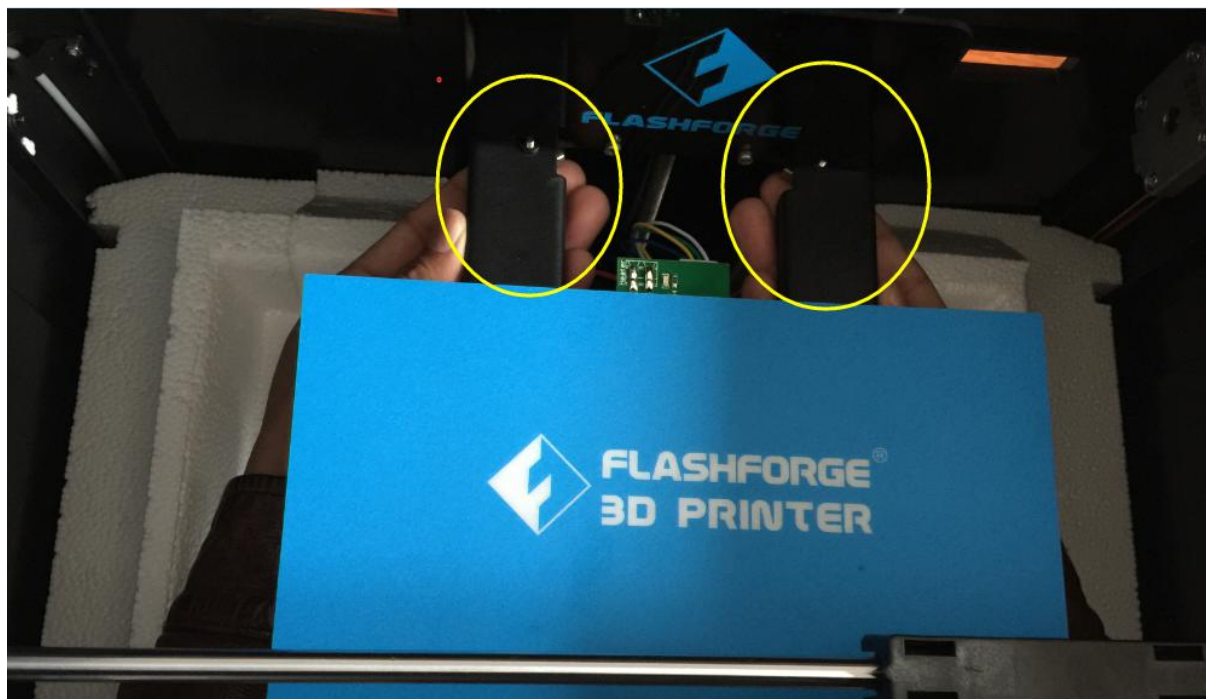
Move X axis rods to the back of the printer manually, remove the tapes on the front door, open the door and take the top lid together with **Foam C** out. Take two spools from **Foam C**, set them aside. Remove the **Foam C** from top lid and set the top lid aside.



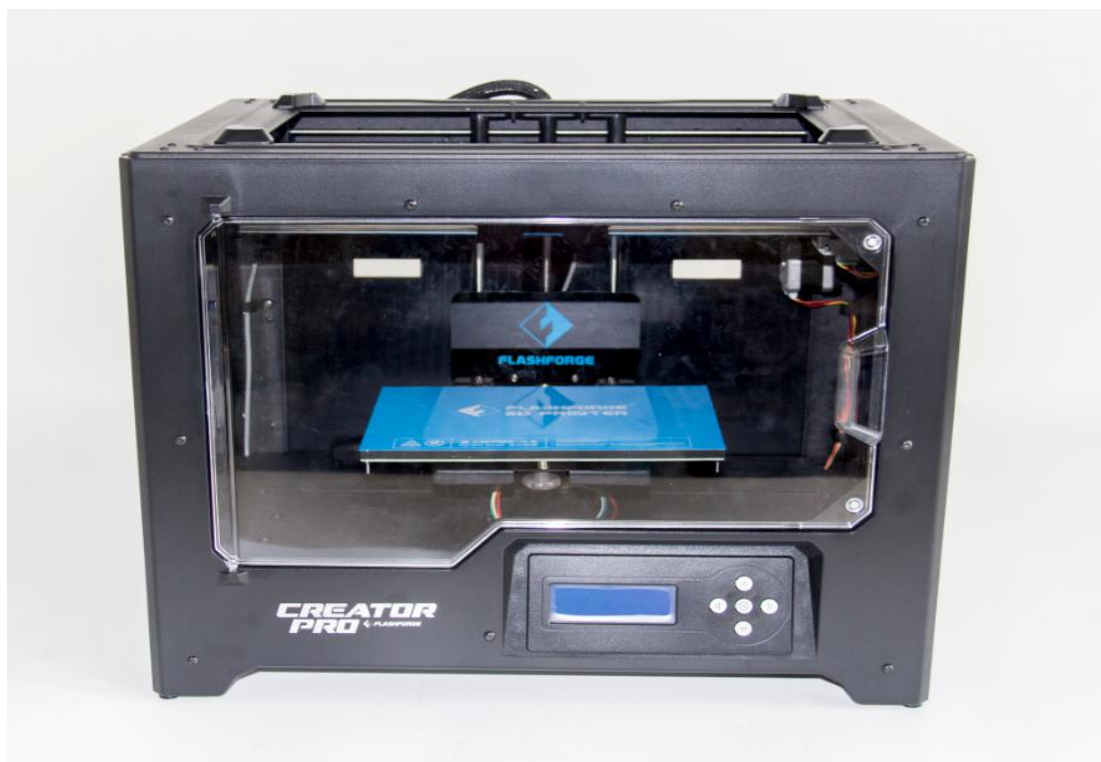
Now the build platform should be visible. It is an aluminum plate covered in a preloaded blue 3M platform sticker. This is the surface that your objects will be printed on. Remember: **Do not remove the platform sticker. It helps a lot extruded filament stick on the platform.**

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

- 1) Turn the Z motor rod clockwise with your hand which is between two Z axis guide rods until there is enough space for your hands to hold the rear of platform support.
- 2) Hold the platform support with one hand on each side. **Important note: user must hold the rear of the support near the Z guide rod bearings; otherwise the platform will get stuck.** Raising it slowly and keeping it level. Stop once the platform is at its highest position (about 2~3 cm lower than the X guide rods).



Remove all other packing materials out from Creator Pro. Now the machine shows you like this.

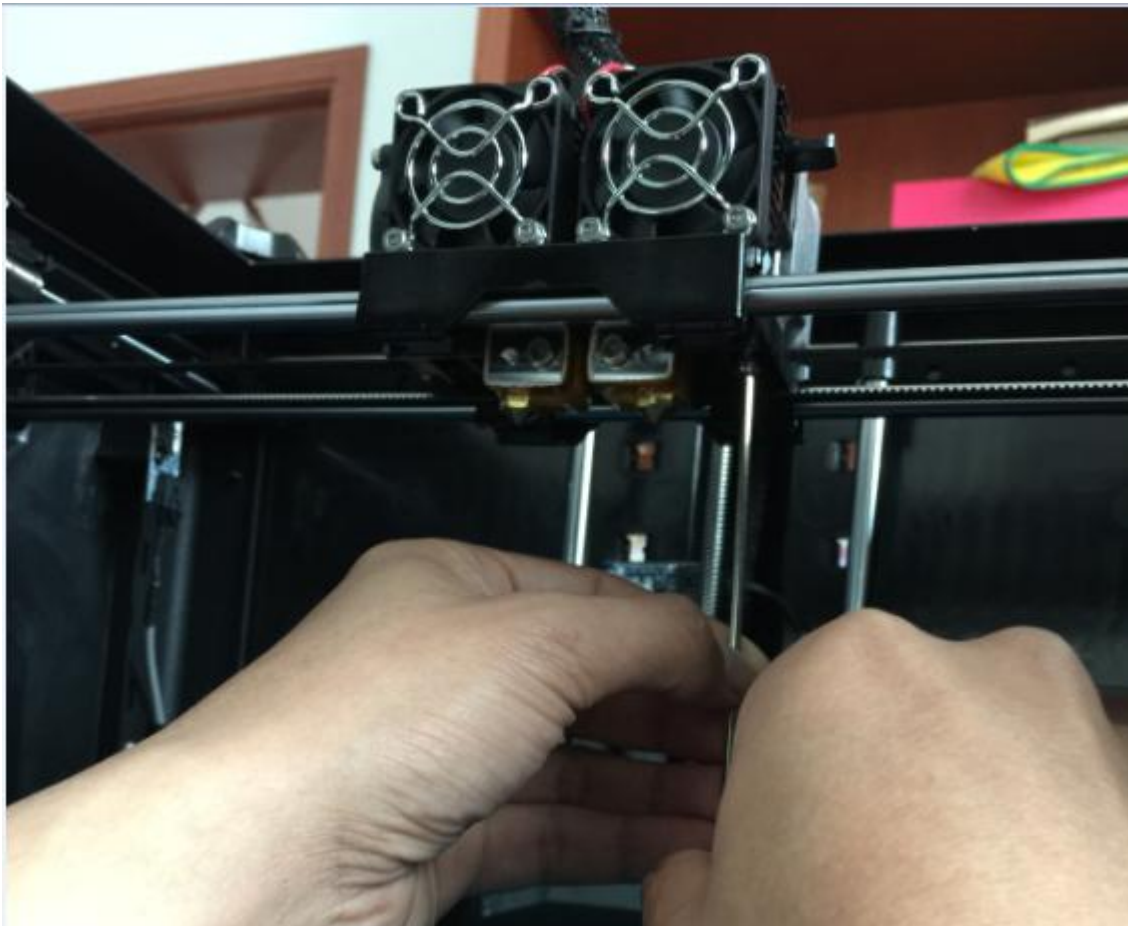


You have now finished un-boxing! The next task is to set up the hardware.

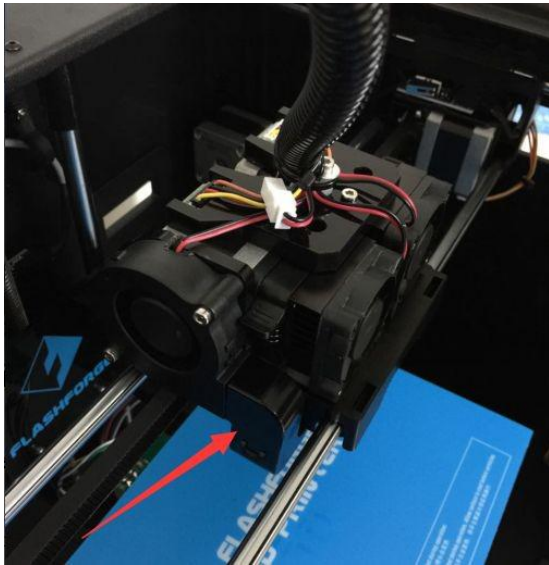
Initial Hardware Installation

Start by installing the extruder. You'll need two screws in M3×8 from Extruder's accessory kit and the appropriate hex wrench from Tool bag found in the **Foam B**.

First, lower the build platform using one of the methods described in the previous section. Holding the extruder by both sides and positions it on the extruder carriage **with the fans facing forward**. Align the screw holes and fasten with the two screws.

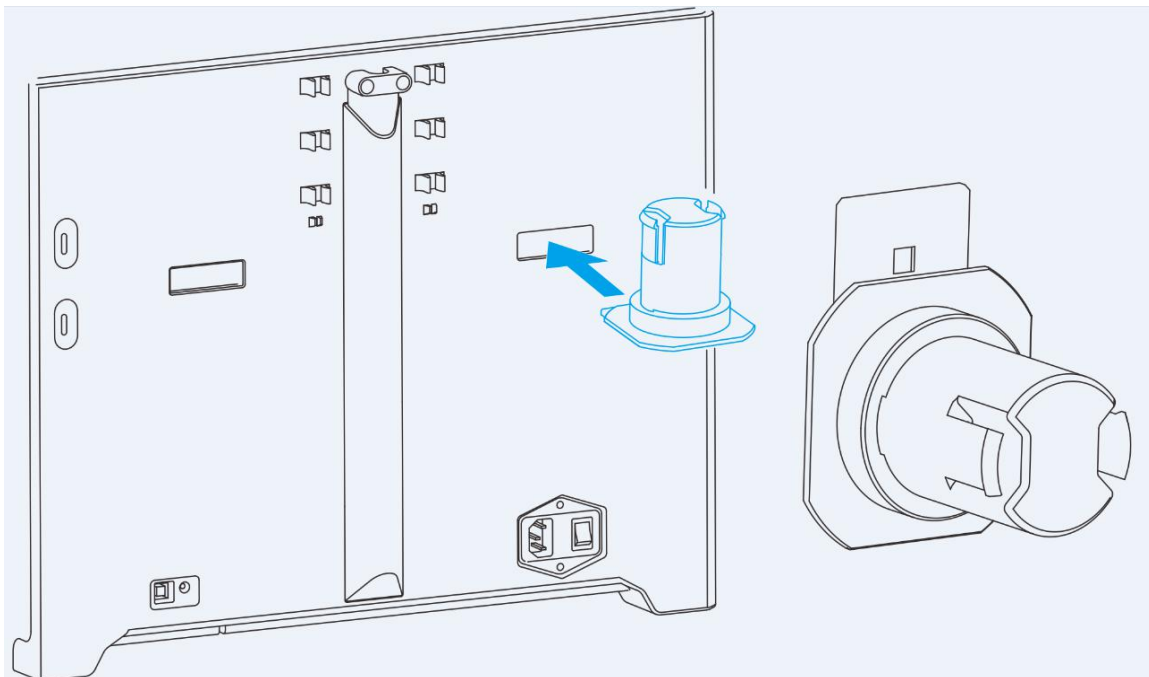


Take the fan duct and M3×6 screw (the shorter one) out from Extruder accessory kit in **Foam B**, install the duct onto the side cooling fan and use the M3×6 screw to fasten it.



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 rectangular opening.



Then install the filament guide tube to the empty spot on the extruder, no need to insert guide tube end into the hole, just leave it outside, and pass the other end through buckle and buckle it.

Next is the installation of top lid. Place two anterior angles in the corresponding front slots. Slightly squeeze the two relief angles and place them to the corresponding rear slots.

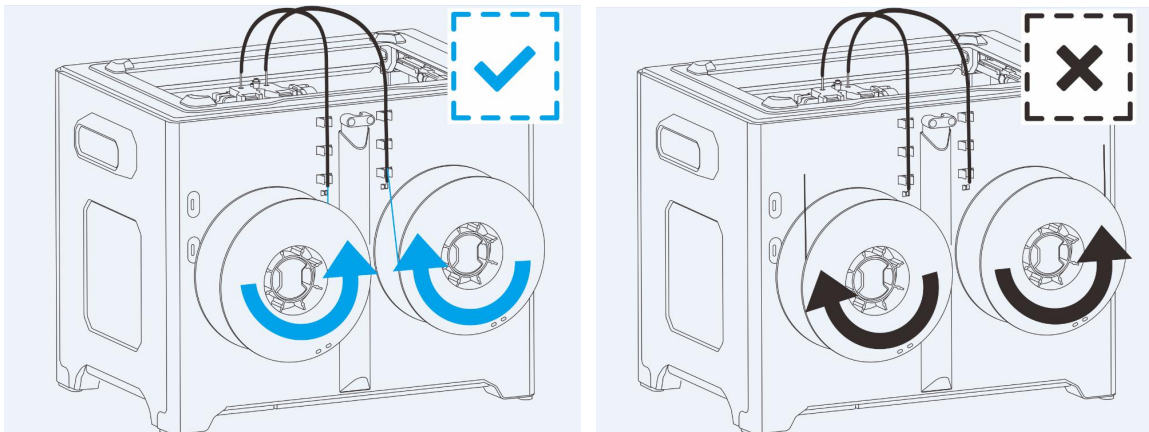


Till now, the hardware installation is almost complete.

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 two spools of filament on the spool holders. Take a roll of ABS filament, slightly squeeze the top of spool holder and mount the spool to the holder. Install the other side in the same way. Make sure the filament will unwind in correct direction. I attach two diagrams to address.



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 Win 8, Win 8.1 and Win 10 users

Win 8, Win 8.1 or Win10 users please make sure that you have disabled the Driver Signature Enforcement setting before installing software installation; **otherwise printer driver can not be installed successfully.**

Below are video links showing you how to accomplish this:

Win 8.0 users: <http://www.youtube.com/watch?v=NM1MN8QZhmk>

Win 8.1 users: <http://m.youtube.com/watch?v=gmw86KplqmU>

Win 10 user: https://www.youtube.com/watch?v=71YAlw7_-kg

Software Installation

ReplicatorG0040r24-Sailfish is the recommended software to use with the dual extruder Creator Pro. Users can find the installer on the provided SD card, which shipped together with the Creator Pro.

Steps for installing your software on Windows operating system:

1. From the SD card shipped with your printer, browse to the “For Windows” folder; Copy this folder to your computer.
2. Locate the “For Windows” folder; double click to install the Python.
3. Decompress the zip file, you will get a folder named “replicatorg-0040r24-Sailfish”
4. Open this folder and right click the icon “ReplicatorG.exe”, choose “send to desktop shortcut”.
5. To run the Replicator G software, double click the shortcut on your desktop or in your start menu.

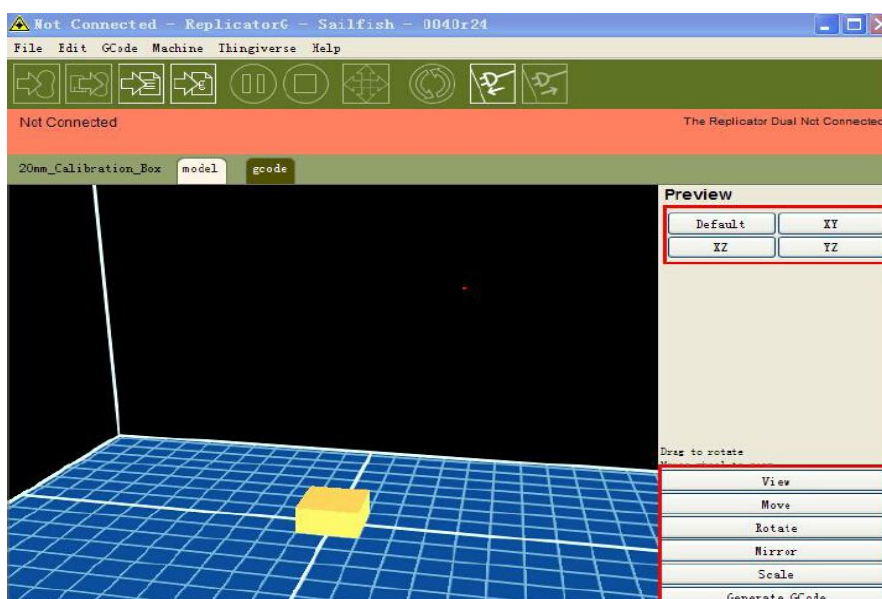
For Windows 32 bits operation system, please double click “**python-2.7.8.msi**” to install the python; For Windows 64 bits operation system, please double click “**python-2.7.8.amd64.msi**” to install the python.

For Mac users, only need to install ReplicatorG, no need for Python.

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.

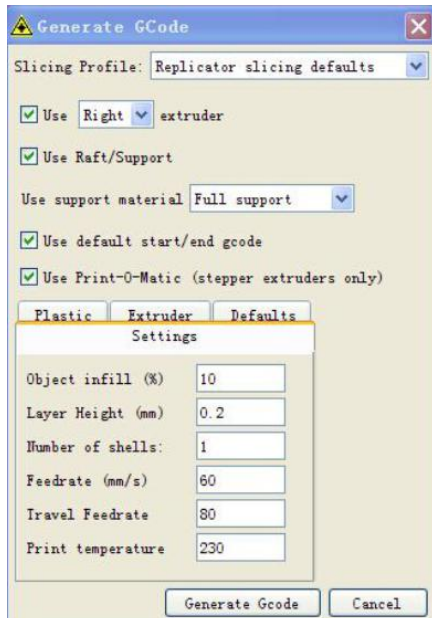
When the object is imported you may find that it is not on the virtual build platform or even on the screen, using the function keys indicated by the red boxes you can change the camera angle and reposition the object onto the center of the build platform. Once you've done this, the next step is to generate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



Recommended GCode Settings for ABS and PLA

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

ABS:

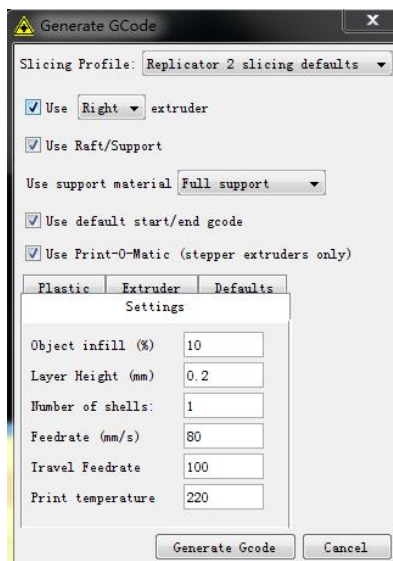


And set HBP at 90



Note: The HBP temperature is related with the temperature of the surroundings and the size of the model. At the room temperature of 20°C, to print a square of 20x20 mm, the HBP temperature is generally 90°C. If the temperature of the surroundings drops (below 20°C), or the size of the model is bigger, then the HBP temperature should be increased accordingly to avoid warping, but not exceed 120°C.

PLA:



And set HBP at 50



Note: The HBP temperature is related with the temperature of the surroundings and the size of the model. At the room temperature of 20°C, to print a square of 20x20 mm, the HBP temperature is generally 50°C. If

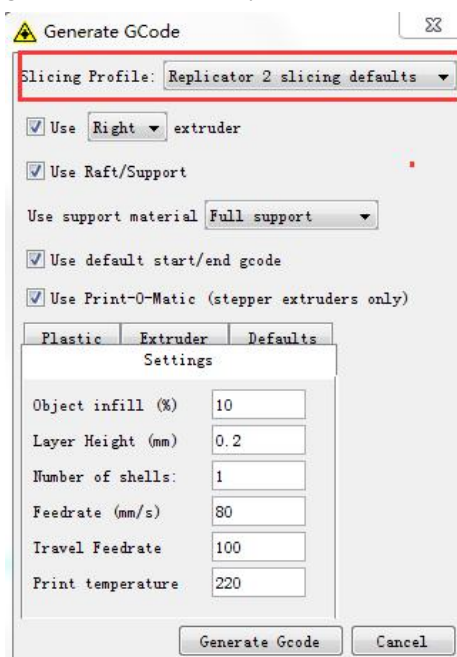
the temperature of the surroundings drops (below 20°C), or the size of the model is bigger, then the HBP temperature should be increased accordingly, but not exceed 70°C.

Click **Generate GCode** on the popped up window, an Acceleration Warning will pop up. Please click ok to continue.

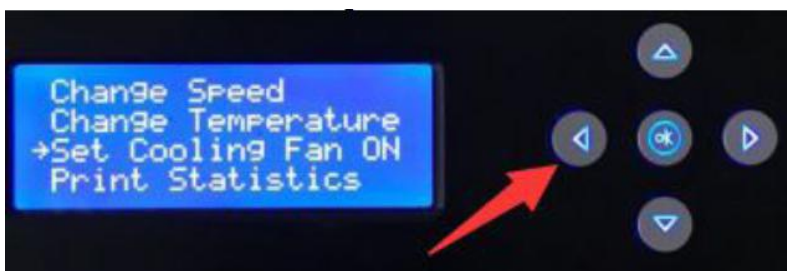


How does the side cooling fan work?

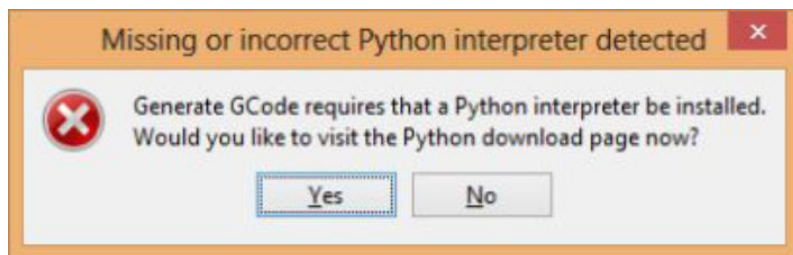
The side cooling fan will start working only when user uses slicing profile “Replicator 2 slicing defaults” to generate Gcode in ReplicatorG.



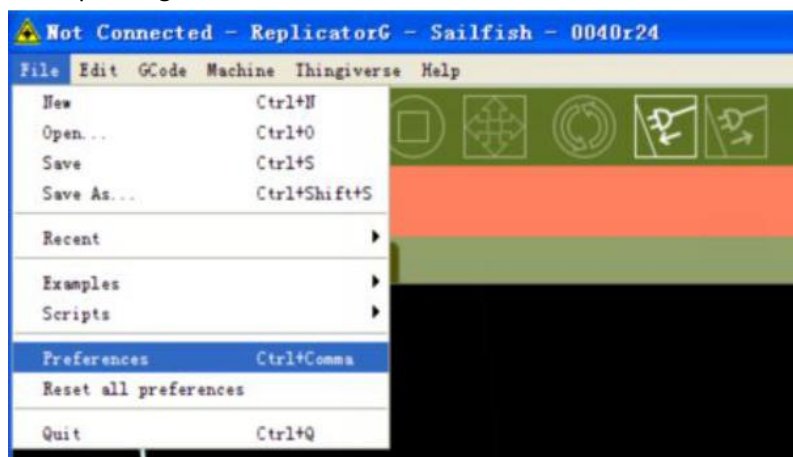
User also can turn on or off the cooling fan **during heating or printing process** manually by pressing the left button on button panel and choose the menu accordingly on LCD screen.



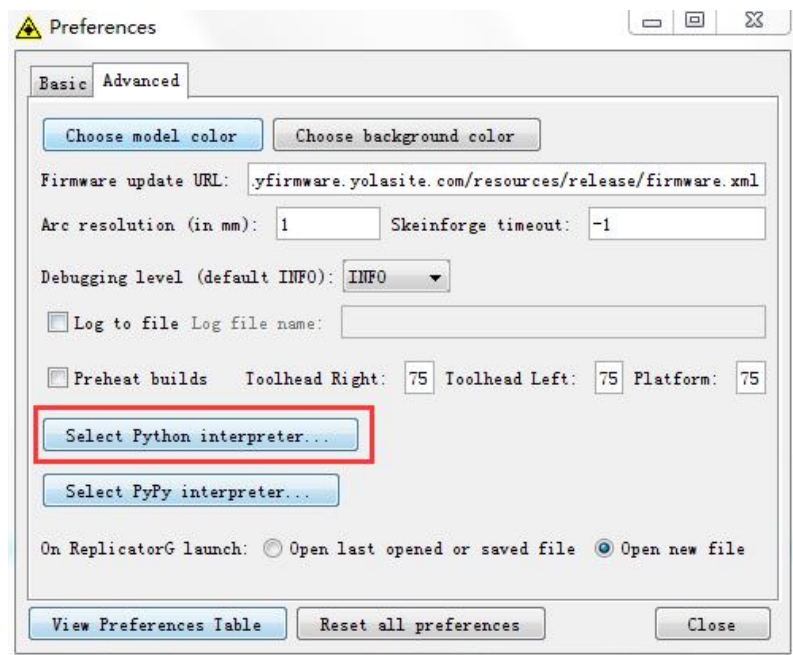
NOTE: For users who did not choose the default installation path in the installation of Python, clicking the Generate Gcode button in Replicator G, will result in dialogue box popping up alerting that the executable Python file cannot be found.

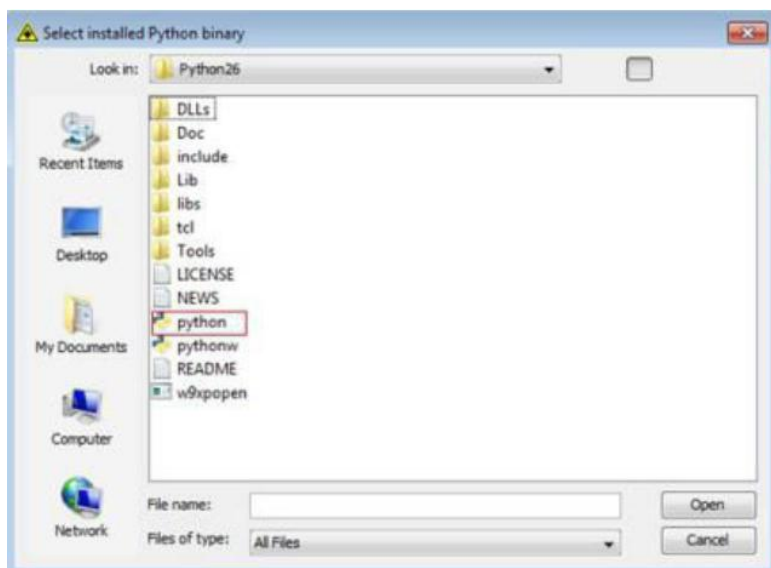


First click the 'No' button to dismiss the dialogue box. To solve this problem we need to configure the corresponding menu.



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.

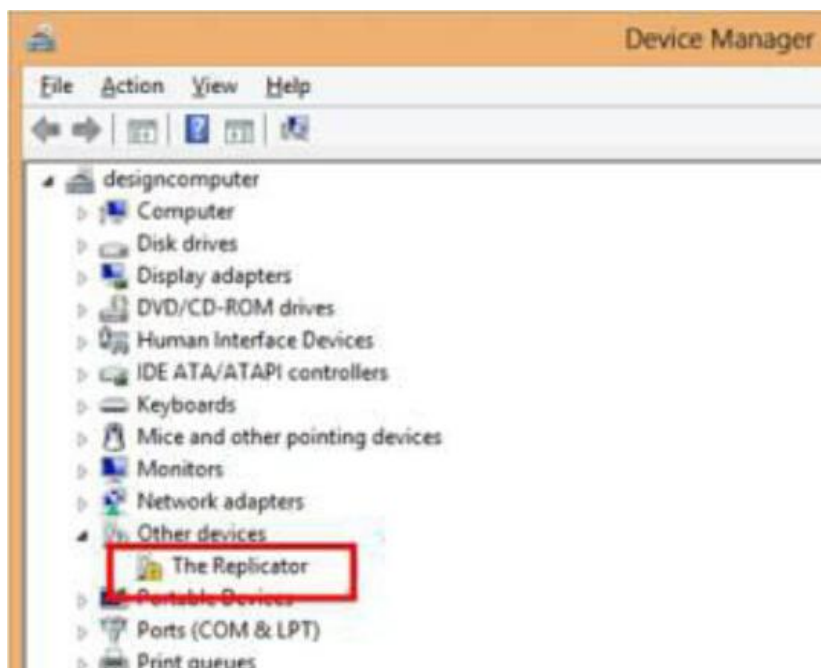
USB connection

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

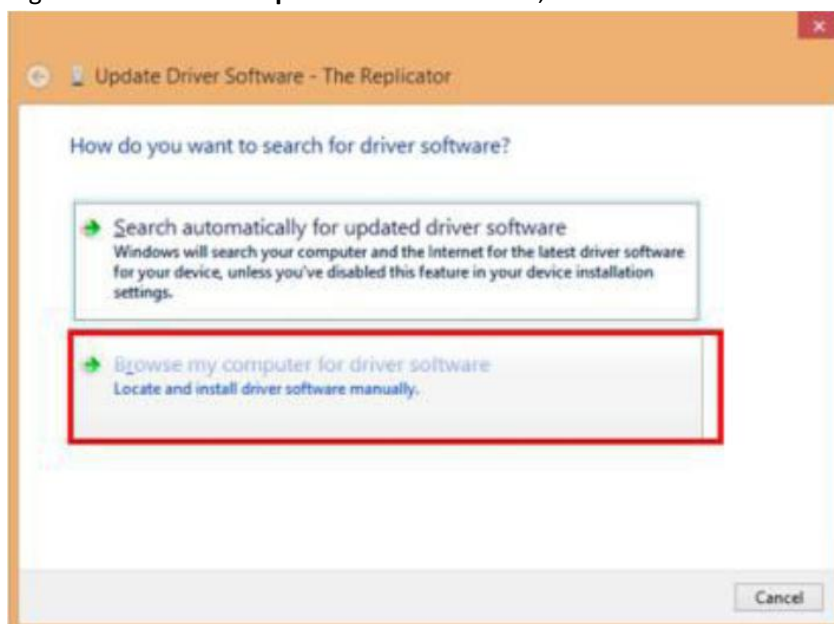


The USB port on the machine is bound by the red box in the above image. After connecting the cable, open the Replicator G software, we are going to connect the computer and printer.

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**'.



Right click and select '**Update Driver Software**', locate the software driver shown in the red box,.



Click '**Browse**'... to find the location of ReplicatorG0040r24-Sailfish 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.

First, select the correct machine type "The Replicator Dual (Sailfish)" for Creator Pro:



Second, rescan the serial ports and select the one that appears on your machine (on our test machine the port was COM7, 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.

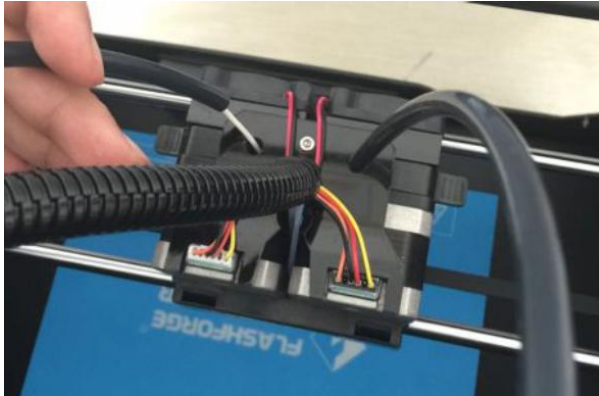


Loading and unloading filament

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, which has stated in previous section '**Initial Hardware Installation**'

Filament Loading

1. Start the machine, the display indicates:
 - Print from SD
 - Preheat
 - Utilities
2. Select **Utilities**; click the **ok** key in the middle of the keypad. The display will show:
 - Monitor Mode
 - Filament Loading
 - Preheat Settings
 - Level Build Plate
3. Select **Filament Loading**; click the **ok** key in the middle of the button board. The display will show:
 - Unload right
 - Load right
 - Unload left
 - Load left
4. Select **Load right**. Click the **ok** key on the button board. The display will indicate: **Heating**: At this time, the temperature of the right nozzle is being heated up. When the temperature of the nozzle reaches target, display will ask you to feed the filament in, just finish the next step following the script instruction.

Filament Unloading

- 1) Turn on the power switch, you will see the LCD panel shows:
 - Print from SD
 - Preheat
 - Utilities
- 2) Select **Preheat**, Click **ok** key in the middle, then you will see:

►Start Preheating

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 Preheating

Right Tool	OFF
Left Tool	ON
Platform	OFF

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

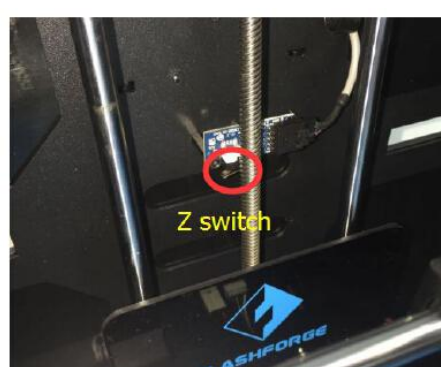
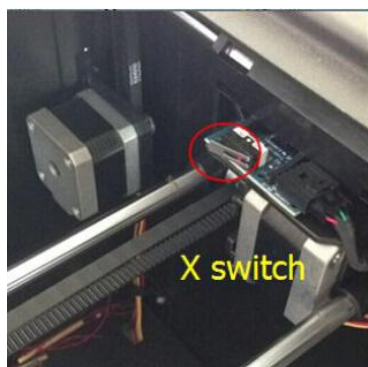
Heating:
R Extruder: 033C
L Extruder: 033/230C
Platform: 024

This means the left extruder is heating up, when it reaches 230°C, first push in the filament a little bit manually until you see filament come out of the nozzle, then press the spring lever down and meanwhile pull filament out quickly. This will ensure you have gotten rid of the filament inside the nozzle successfully.

Level Build Plate before initial print

First, make sure that you've completed below steps:

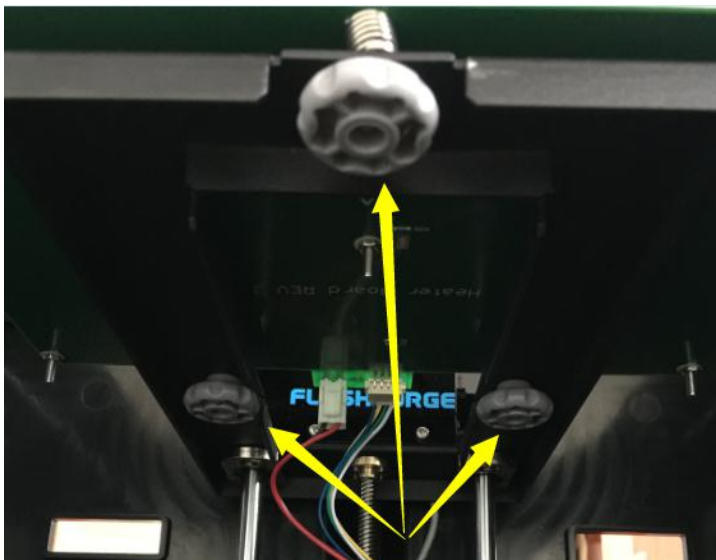
1. Your Dual Extruders should be bolted in place.
2. Your filament guide tubes connected.
3. Your spools of filament mounted on the spool holders.
4. Filament has been loaded for both extruders.
5. **Check X, Y, Z end stop switch. Make sure the little metal tabs are all still there. See below pictures for your location use.**



If everything is ready, Insert the provided SD card into its slot, then flip on the power button on the back of your Creator Pro.



Lower the platform as possible as you can by turning three leveling knobs counterclockwise. In this way, platform can avoid the initial scratch damage from nozzles.



Pay close attention: leveling your platform is very important to print quality! Creator Pro 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 the initial print.

Select LCD menu '**Print from SD → Plateleveling.x3g**'. Leveling instruction scripts will shown on the LCD, please read full scripts by click '**ok**' button for next pages. After instruction, the build plate and Extruder assembly starts moving, once they suspend, you need to adjust the corresponding leveling knob under the right nozzle. As you adjust the knob, make sure the provided leveling card can just slides between the nozzle and the build plate with some resistance. If the card can slide easily, you need rotate the knob clockwise to decrease the distance. On the contrary, you need to rotate the knob counterclockwise to increase the distance.

When you finish leveling the first point, press '**ok**' button to level the next point and the third one. Finally the extruder assembly will move to the middle of the platform and stop there. Make a final check of the

distance between build plate and nozzle, click '**ok**' to finish the leveling process.

You can refer to video instruction: <https://www.youtube.com/watch?v=vXyHNHRY9fg>

If your platform is too low, your prints might not stick to the surface. Users always complain their prints do not stick to platform. Actually this is because the platform is not well leveled, it is too low. If it's too high, the nozzles could tear the sticker on the platform surface.

Print from SD card

Important note: All sample models pre-loaded on SD are generated base on ABS printing, NOT for PLA. So if you want to print them out, please use ABS spool.

Go to LCD display, select menu '**Print from SD**'---select what you want to print.



Note: The model '**traffic cone**' is only for dual color extrusion. If you want to print this model out, make sure you have loaded ABS filament for both extruders.

If you want to print models designed by yourself from SD card, first you need to generate a Gcode for the model, and then click the third button from left, save it as an x3g file. Copy it onto SD card. Now you can print it out from SD.

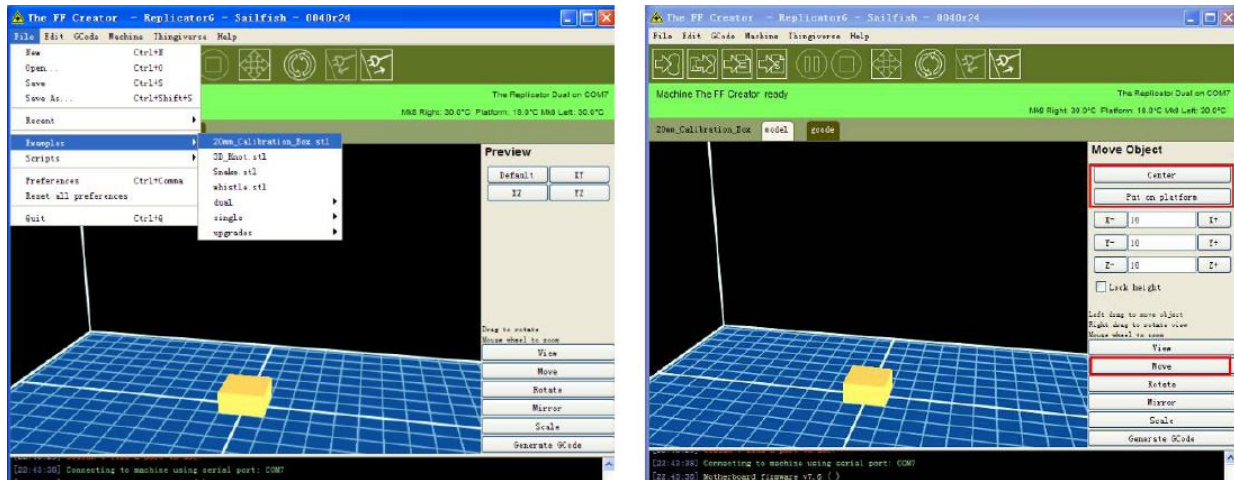


Special note to PLA printing

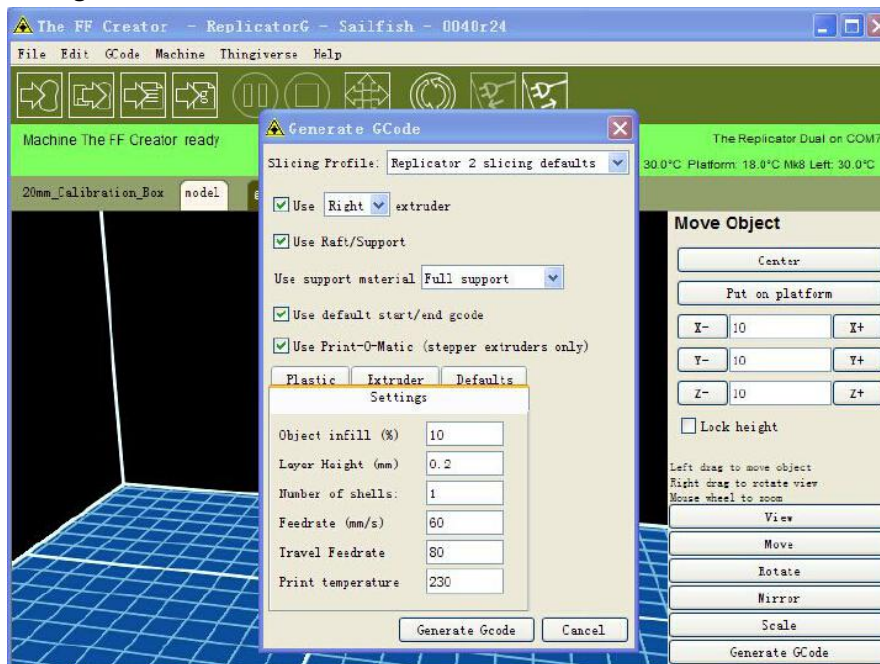
The top lid and the front door are designed exclusively for ABS printing, NOT for PLA. If user wants to print with PLA, please make sure you have removed the top lid and open the front door before starting printing. Otherwise, PLA filament will get soft and wrapped around feeding gear due to high temperature inside the case which causes print failure.

Initial print over USB

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 centre of the build plate.



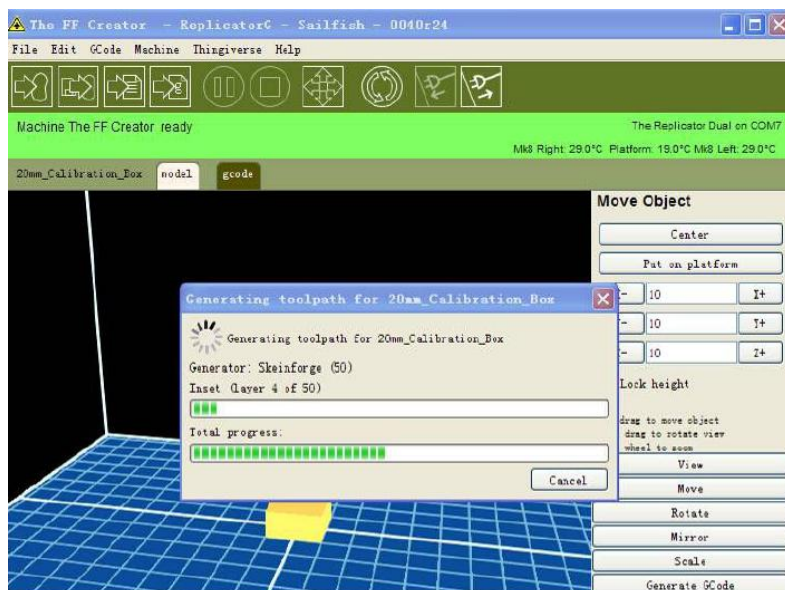
Next click **Generate GCode**, a dialogue box will pop up, after inputting your chosen parameters the Gcode will be generated. You can find recommended Gcode settings at previous section of '**Recommended Gcode settings for ABS and PLA**'



Next click **Generate GCode** on the popped up window, an Acceleration Warning will pop up. Please click **ok** to continue.



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



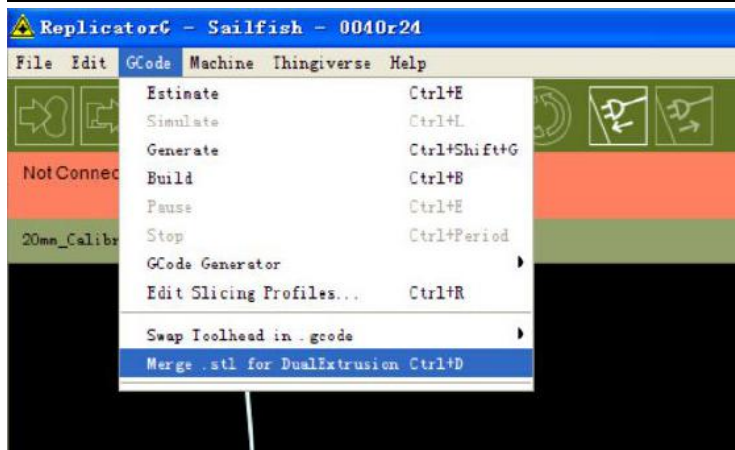
After modifying, click the file button to save the sample and click the left most button on the blue box.



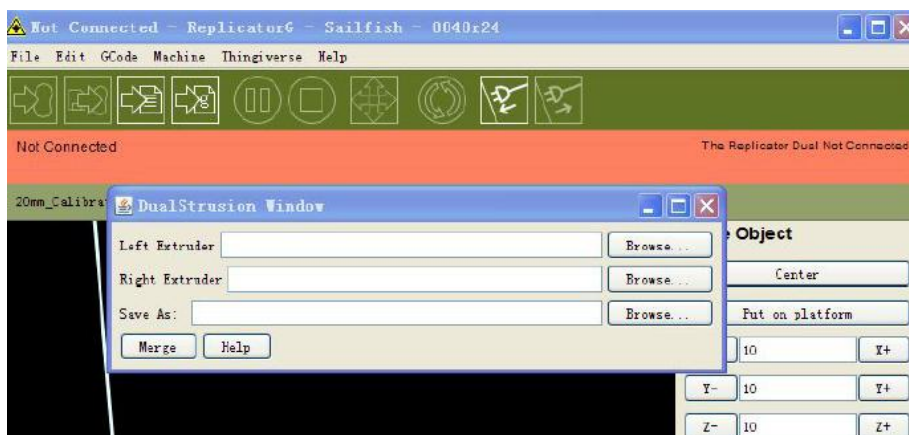
The Gcode has now been sent to the printer and the object will start printing shortly.

Dual Extruder print

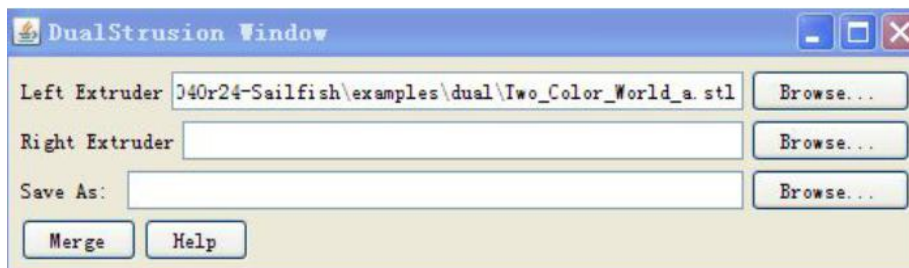
To use the two-color printing that comes with the Replicator G software, open it up and select **Gcode > Merge .stl for Dual Extrusion**.



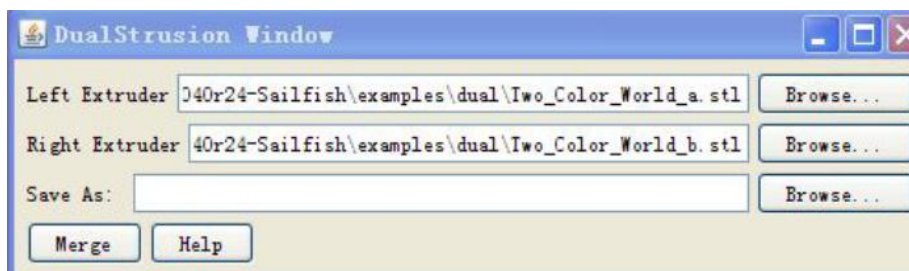
The following dialogue box will then appear:



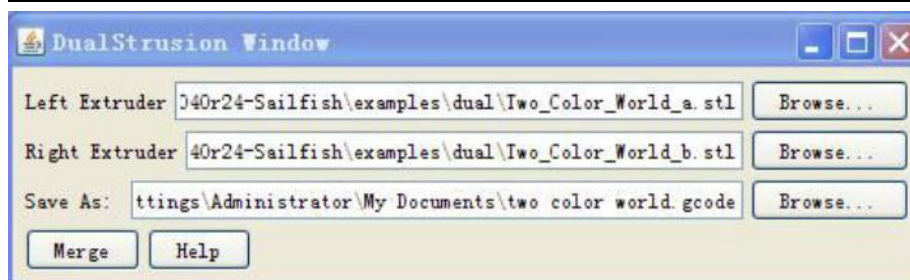
Click browse for left extruder; locate the folder 'examples' found in folder 'replicatorg-0040r24- Sailfish'. Then open the folder 'dual' select **Two_color_world_a.stl** as shown below:



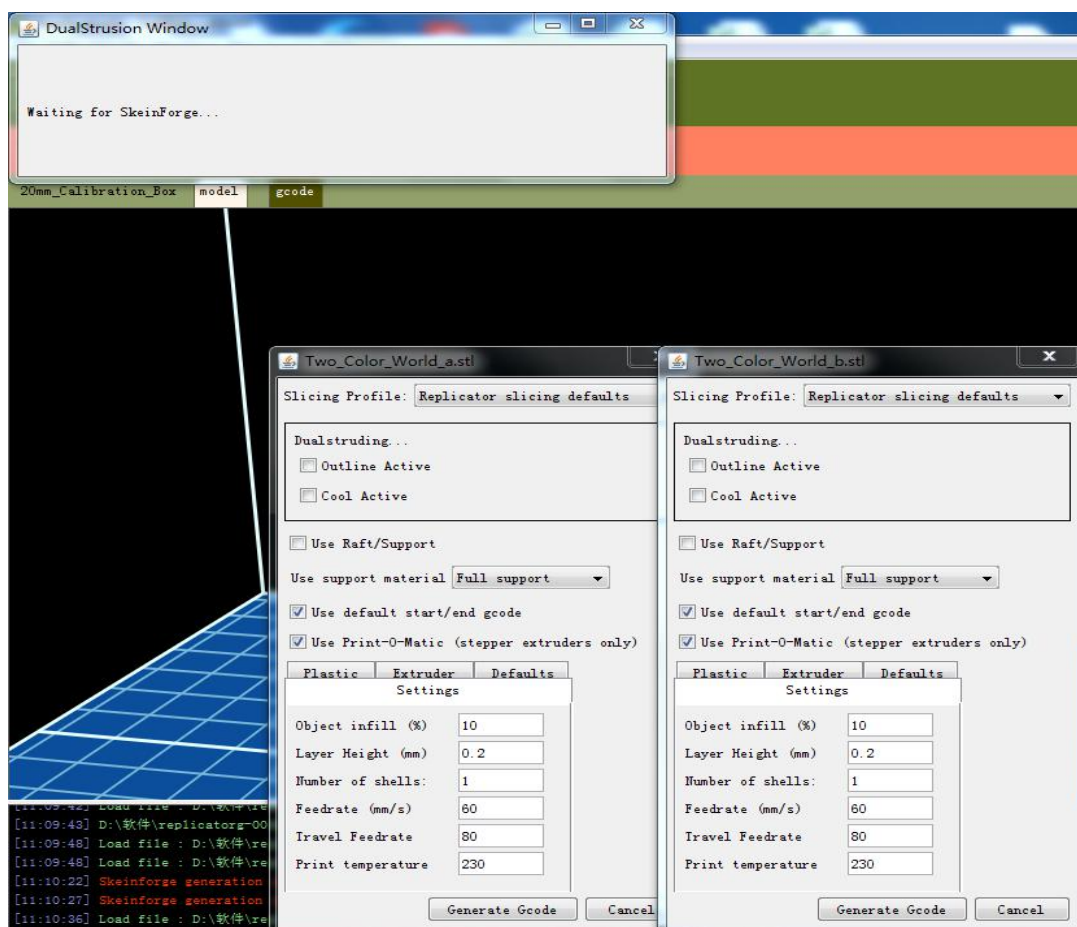
Repeat the process for the right extruder and select **Two_color_world_b.stl** as shown below:



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

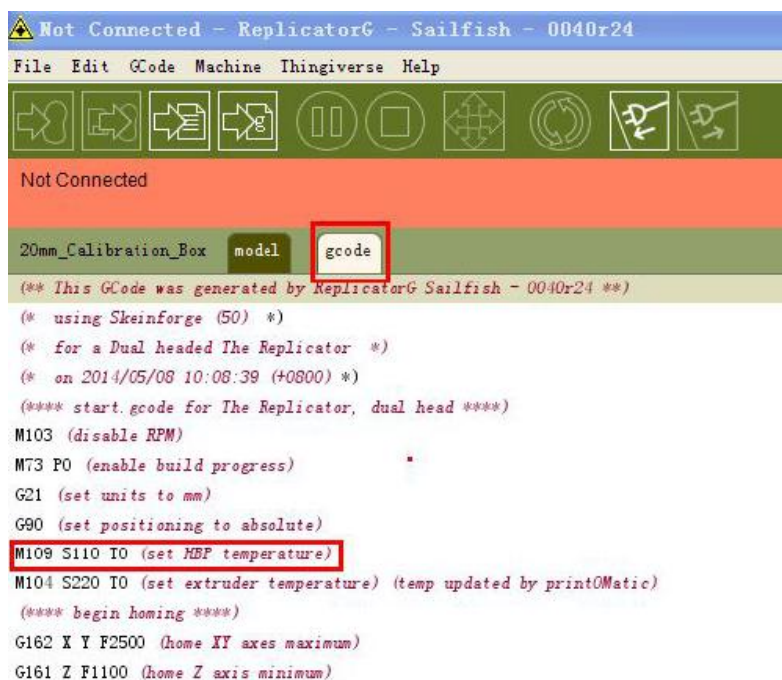


Now click **Merge**, two dialogue boxes will then pop up.



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 according to recommended settings.



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



Firmware Update

Creator Pro has been already installed with the latest and most appropriate Sailfish firmware before it leaves factory. So there is no need to upgrade firmware anymore.

Caution: Please do not upgrade firmware without FlashForge Amazon professional instruction. Any wrong uploading will cause unexpected problems or even damage the printer.

Thank you for shopping with FlashForge Amazon!

-----End-----