

Contents

1. Software Introduction.....	1
1.1 FDM 3D Printer Type.....	1
2. Software Installation.....	1
2.1 Download.....	1
2.2 Installation & Start.....	1
3. Software Operating Instructions.....	3
3.1 Load File.....	4
3.1.1 Generate Relievo.....	4
3.2 Mouse Operations.....	8
3.2.1 Left click.....	8
3.2.2 Left click and hold the left button.....	9
3.2.3 Right click and hold the right button.....	9
3.2.4 Scroll the mouse wheel.....	9
3.3 Change Views.....	9
3.3.1 Move.....	9
3.3.2 Rotate.....	9
3.3.3 Scale.....	9
3.3.4 Set.....	9
3.3.5 Reset.....	9
3.3.6 Show Model Outline.....	9
3.3.7 Show Steep Overhang.....	10
3.4 Edit Models.....	10
3.4.1 Move.....	10
3.4.2 Rotate.....	10
3.4.3 Scale.....	11
3.4.4 Set.....	11
3.4.5 Cutting the Model:.....	12
3.4.6 Others.....	13
3.5 Edit Supports.....	17
3.5.1 Auto Supports.....	17
3.5.2 Clear Supports.....	17
3.5.3 Add.....	17
3.5.4 Delete.....	18
3.5.5 Model Select.....	18
3.5.6 Support option.....	18
3.6 New Project.....	19
3.7 Saving File.....	19
3.8 Printing Process.....	19
3.8.1 Choose Machine Type.....	19
3.8.2 Connect Machine.....	19
3.8.3 Print.....	19
3.9 Printer Operations.....	40

3.9.1 Connect / Disconnect.....	40
3.9.2 Control Panel.....	43
3.9.3 Firmware Upgrade.....	47
3.9.4 On Board Preferences.....	47
3.9.5 Reset Motherboard Completely.....	48
3.9.6 Machine Information.....	48
3.9.7 Driver Installation.....	48
3.9.8 Multi-Machine Control.....	48
3.10 Others.....	49
3.10.1 Preferences.....	49
3.10.2 First run wizard.....	50
3.10.3 Help Contents.....	51
3.10.4 Feedback.....	51
3.10.5 Check for Updates.....	51
3.10.6 Software Information.....	52

USER GUIDER

1. Software Introduction

1.1 FDM 3D Printer Type

There are sixteen type printers:

FlashForge Adventurer 3/FlashForge Adventurer 3 Lite/FlashForge Dreamer/FlashForge Creator Pro/FlashForge Inventor Series/FlashForge Finder/FlashForge Guider/FlashForge Guider II/FlashForge Guider II S Series/FlashForge Inventor II Series/FlashForge Dreamer NX/FlashForge Creator 3/FlashForge Creator Max/FlashForge Creator Pro 2/FlashForge Adventurer 3C/FlashForge Creator Max 2

2. Software Installation

2.1 Download

A. Insert SD card sent with Dreamer into a computer, we have already prepare the latest driver installation package in the SD card.

B. Open below link to download the installation package:

<http://www.ff3dp.com/#!sup/c1df1>

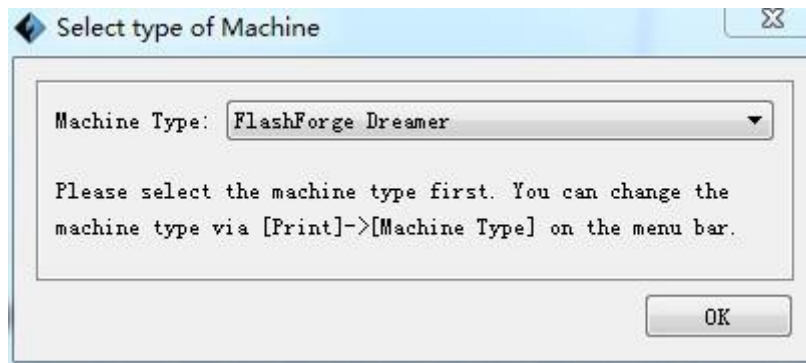
2.2 Installation & Start

A. Uncompress the downloaded RAR, and complete installation according to the instruction. As for windows and mac OS, users only need to decompress the RAR/ZIP file and the finish installation according to the guidelines.

As for the Linux OS, users need to use commands for installation. Place the downloaded .deb file to a folder or on the desktop, open the command window, enter 'sudo dpkg -i +the installation package path' and then press [Enter] in the keyboard. If any dependency package shall be reminded for installing, users need to enter "sudo apt-get -f install" to complete its installation. (Note: Linux 14.0 or higher versions are suggested)

B. Connect your Dreamer desktop 3D printer with USB cable and make sure that the printer is connected to the computer successfully. Start the software with the shortcut on computer desktop or in the Start Menu. When you first start the software, the following dialog box will pop up for your machine type choosing.

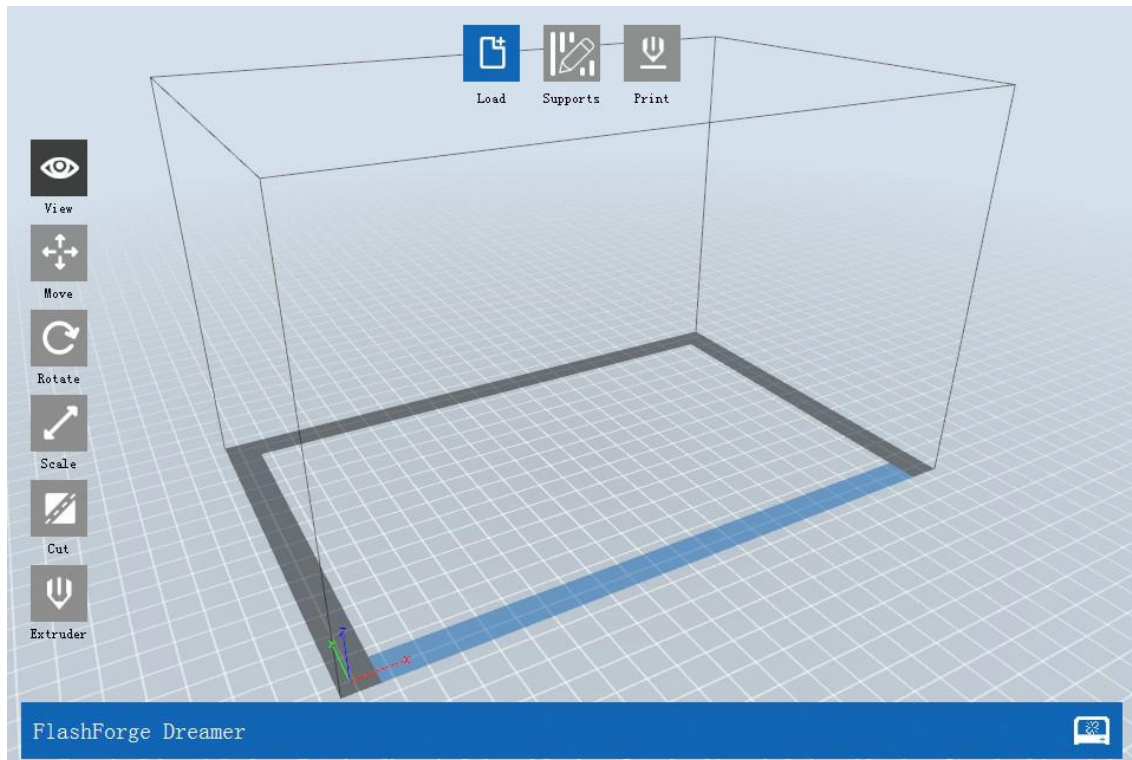
Note: This software supports Windows XP/Win 7 32-bit and 64 bit/ Win 8 and Mac operating system.



When open FlashPrint, Happy3D Pop-Ups will appear automatically. After check 'Do not show this message ',Happy3D Pop-Ups will not appear. Click the button of 'More details ',according to computer language setting, Flashforge official website in Chinese or English will open. (Flashforge official Chinese website only open in Computer with Chinese language setting, while it will open Flashforge official English website in default.)



3. Software Operating Instructions



Load into one or multiple files.



Enter supporting editing mode



View models from one of the six viewing angels. (shortcut:keyboard—V)



Move model around on xy-plane; shift+click to move along z axis (shortcut:keyboard—M)



Turn and rotate your model (shortcut:keyboard—R)



Scale the size of your model (shortcut:keyboard—S)



Select right or left extruder you want to print with (shortcut:keyboard—E)



Print it directly with your Dreamer or export to your SD card.



To cut the model from different directions. (shortcut: keyboard—C)

You can use Flashprint software to control and perform printing tasks.

3.1 Load File

You can load a model file or Gcode file into the software by following six methods:

- A. Click the “Load” button on top of the software interface. Then select the file which is needed for loading.
- B. Select the file which is needed for loading, and then drag the file to the interface of the software.
- C. Click the “File -> Load File” menu. Then select the file which is needed for loading.
- D. Click the “File -> Examples” menu. Then select the file which is needed for loading.
- E. Click the “File -> Recent Files” menu. Then select the file which is needed for loading.
- F. Select the file which is needed for loading, and then drag the file to the icon of the software.

You can load into .stl, .obj, .fpp, .3mf, .png, .jpg, .jpeg, .slc or .bmp model files which are editable in the software. Please refer to Chapter 2.2~2.5 and you can find instructions of that modification, such as operations of mouse, changes of view from different angles and how to edit and save the file. After modification, please refer to Chapter 2.6 if you want to slice the model file, generate Gcode file and Print it.

Important Note: Gcode file can not be edited any more, but you can directly start the printing after loading it into the software. Please refer to the Chapter 2.6.1 and 2.6.2.2 which tells you how to connect to Dreamer and how to print the Gcode file.

3.1.1 Generate Relievo

Relievo is transformed from png, jpg, jpeg and bmp files. The png, jpg, jpeg or bmp file could be transformed into stl file and loaded into the slice software for application.

which includes settings for shape, mode, maximum thickness, base thickness, bottom thickness, width, height, top diameter, Graphic pattern height, equatorial diameter, enable ellipsoid and bottom diameter.

Shape: including plane, tube, sphere, canister, lamp and seal.

Mode: including “darker is higher” and “lighter is higher”.

Maximum thickness: Z value of the model.

Base thickness: The minimum raft thickness and the default value is 0.5mm.

Width: X value of the model.

Depth: Y value of the model.

Bottom thickness: For tube, canister and lamp to set up bottom thickness.

Top diameter: For tube, canister and lamp to set up the top diameter.

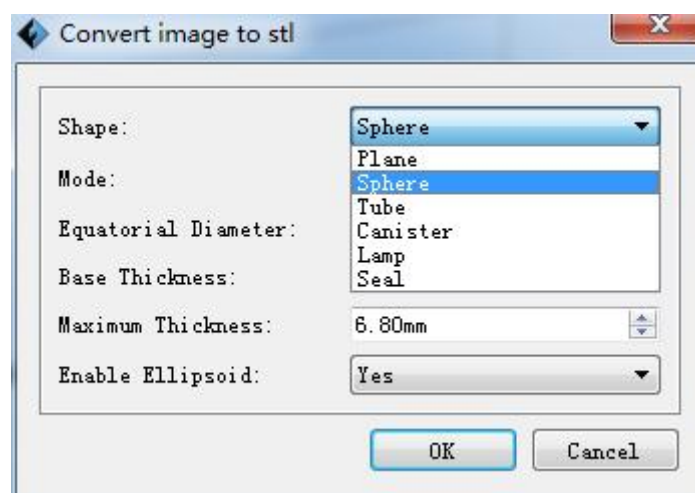
Bottom diameter: For tube, canister and lamp to set up the bottom diameter.

Graphic pattern height: The maximum height of the stamped graphic pattern.

Equatorial diameter: spherical equatorial diameter(inside diameter)size.

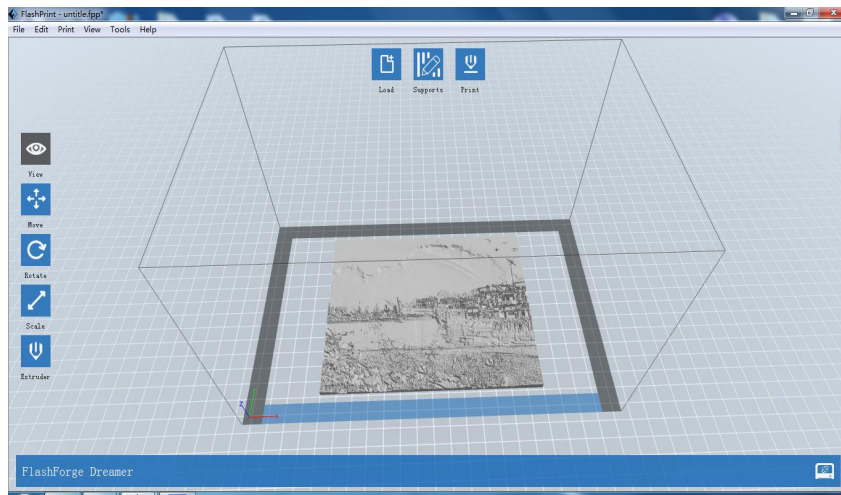
Enable ellipsoid:including two choice:yes,no.Choose ‘ yes ’ ,then generating corresponding ellipsoid according to image loaded(the polar diameter confirmed by the image scale of uploaded image);Choose ‘ no ’ ,then generating sphere according to image loaded.

The effect pictures for different shapes:

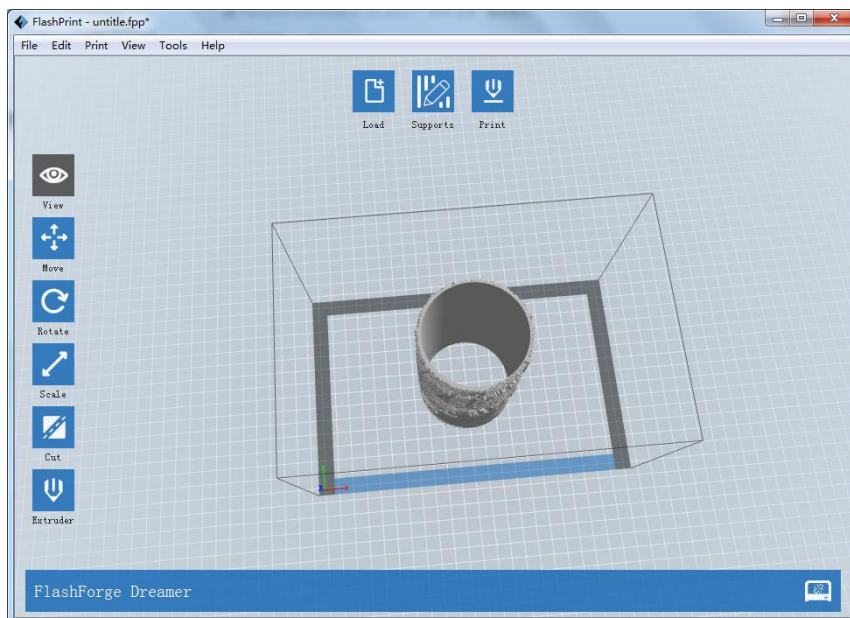


The loaded original png picture:

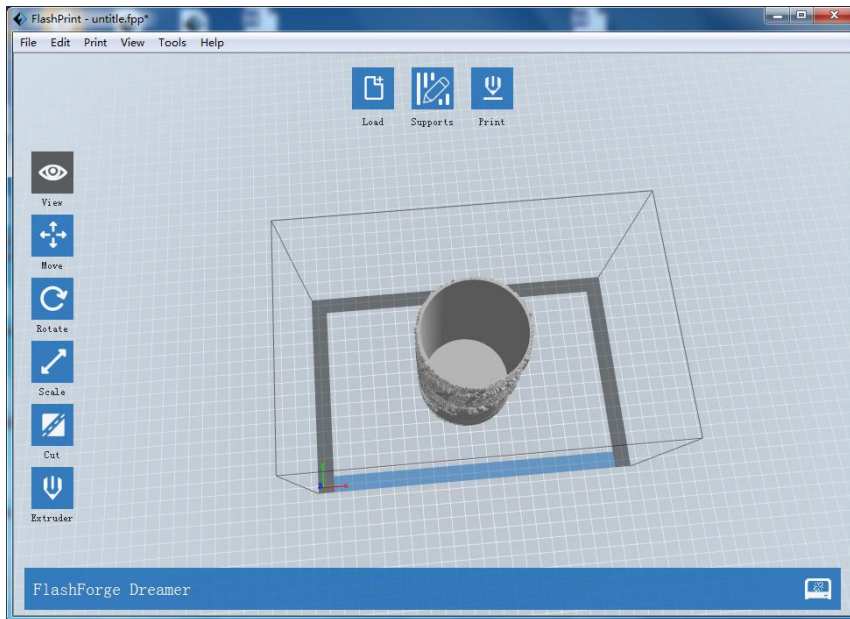




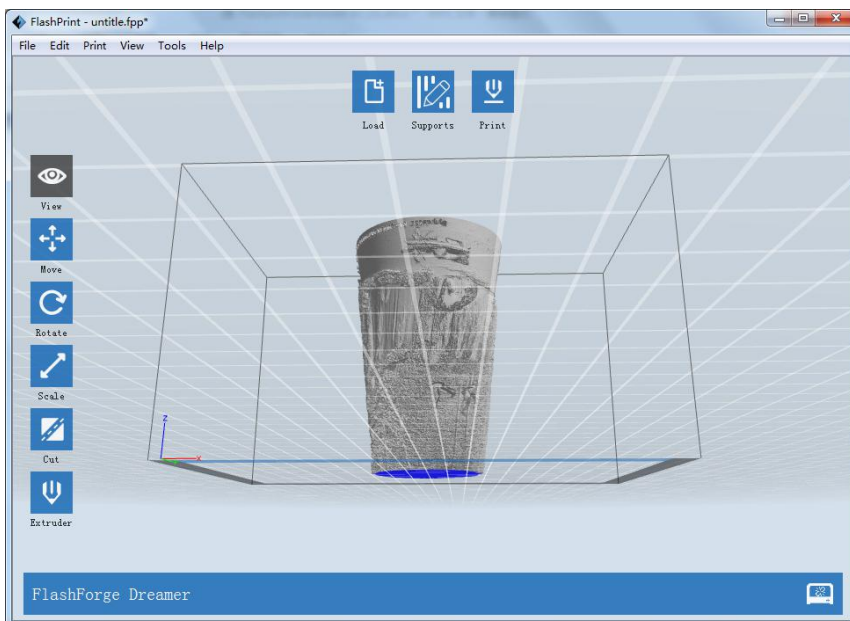
(Plane)



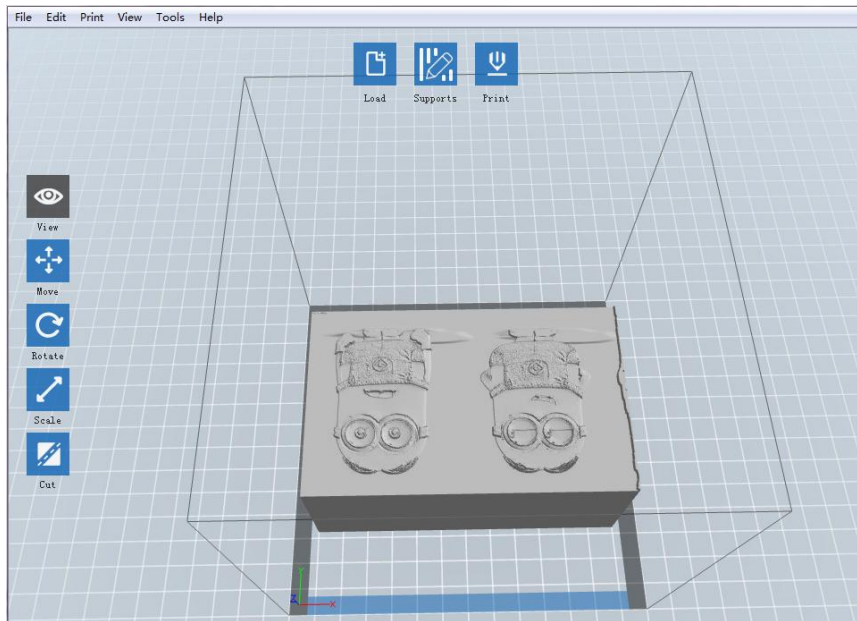
(Tube)



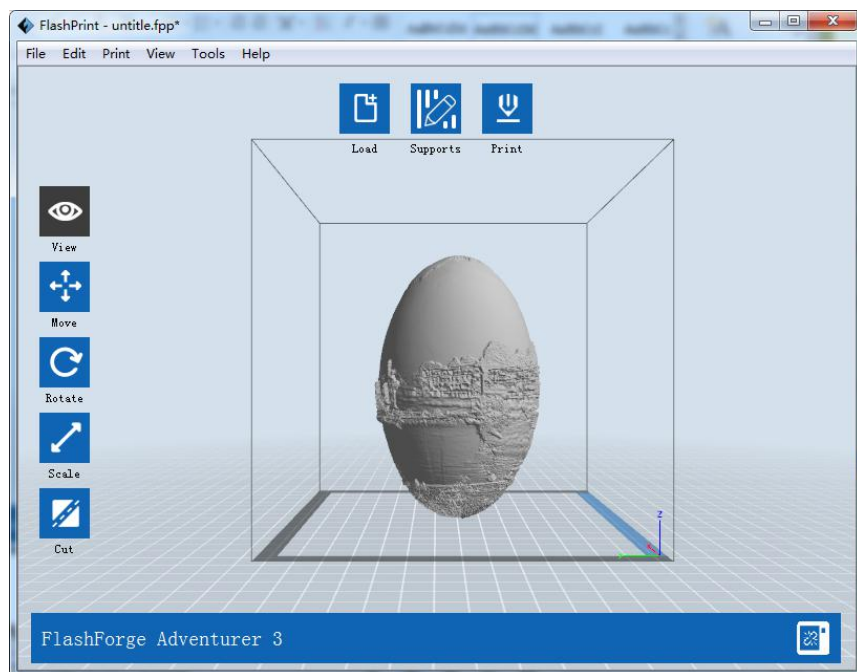
(Canister)



(Lamp)



(Seal)



(sphere)

3.2 Mouse Operations

3.2.1 Left click

- A. Select a model by moving the mouse to it and just left click it.
- B. Select multiple models by holding the **Ctrl** key and left click models you want.
- C. Model looks brighter when selected.
- D. Model can be edited when selected.
- E. Left click blank space to undo the selection of model.

3.2.2 Left click and hold the left button

When changing the viewing angles or editing models, different effects appears by left-clicking and holding the left button. Please refer to Chapter 2.3.1~2.3.2 & 2.4.

3.2.3 Right click and hold the right button

Same results in any operations when right clicking and holding the right button. Please refer to the Chapter 2.3.1 & 2.3.2.

3.2.4 Scroll the mouse wheel

Same effects in any operations when scrolling the mouse wheel. Please refer to the Chapter 2.3.3.

3.3 Change Views

You can choose perspective or orthographic. Change your view of the model by moving, rotating, scaling the view and so on.

3.3.1 Move

You can move the view of printing frame in the software interface by following three ways:

- A. Long-press the left button of mouse and drag.
- B. Press and hold the middle button of mouse, and then drag the mouse.
- C. Hold the **Shift** key, long-press the right button of mouse and drag.

3.3.2 Rotate

You can rotate the view of printing frame in the software interface by following two ways,

- A. Long-press the right button of mouse and drag.
- B. Hold the **Shift** key, long-press the left button of mouse and drag.

3.3.3 Scale

You can scale the view of printing frame in the software interface by scrolling the mouse wheel or use the '+/-' shortcut in any condition.

3.3.4 Set

You can set the view of printing frame in the software interface from six angels by following three ways to observe the model.(**Top/Bottom/Front/Back/Left/Right View**) .

- A. Click the **View** menu, and then select the view you need to observe the model.
- B. Click the **Look** button on the left of the software interface, and select the view you need.
- C. Select and double click the border of build plate to switch views.

3.3.5 Reset

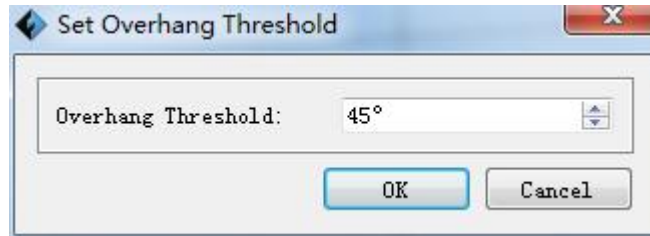
- A. Click the **View** menu, and then select **Home view** to reset the view.
- B. Click the **Look** button on the left of the software interface, and then click **Reset** button to reset it.

3.3.6 Show Model Outline

Click the **View -> Show Model Outline**, it will highlight the outer surfaces of your model file with gray and white color.

3.3.7 Show Steep Overhang

Click the **View -> Show Steep Overhang->Enable/Disable** menu . When the intersection angle between the model surface and horizontal line is within the overhang threshold value, we say the surface has steep overhang and it becomes red in the software. Overhang threshold value could be set as needed. The default value is 45 degree.



3.4 Edit Models

You can edit the models by moving, rotating, scaling the model and so on.

3.4.1 Move

When the model is selected, you can change model location on the build platform by the following two ways:

- A. Click **Move** button on the left of the software interface. Long-press the left button of mouse and drag, you could adjust the location of the model in XY direction. Hold the **shift** key, and long-press the left button of mouse and drag, you can adjust the location of the model in Z direction. You can see the distance and direction of the movement which refers to the relative distance between present and former location.
- B. Click **Move** button on the left of the software interface and then enter the distance value you want to move on X/Y/Z axes positioning. Click **Reset** button to reset distance values.

Note: Generally, we suggest you to click the **Center** and **On Platform** buttons after adjusting location of the model to make sure the model within the printing scope and sticking to HBP. Only click **On Platform** button if you want to print the model in a specified position.

3.4.2 Rotate

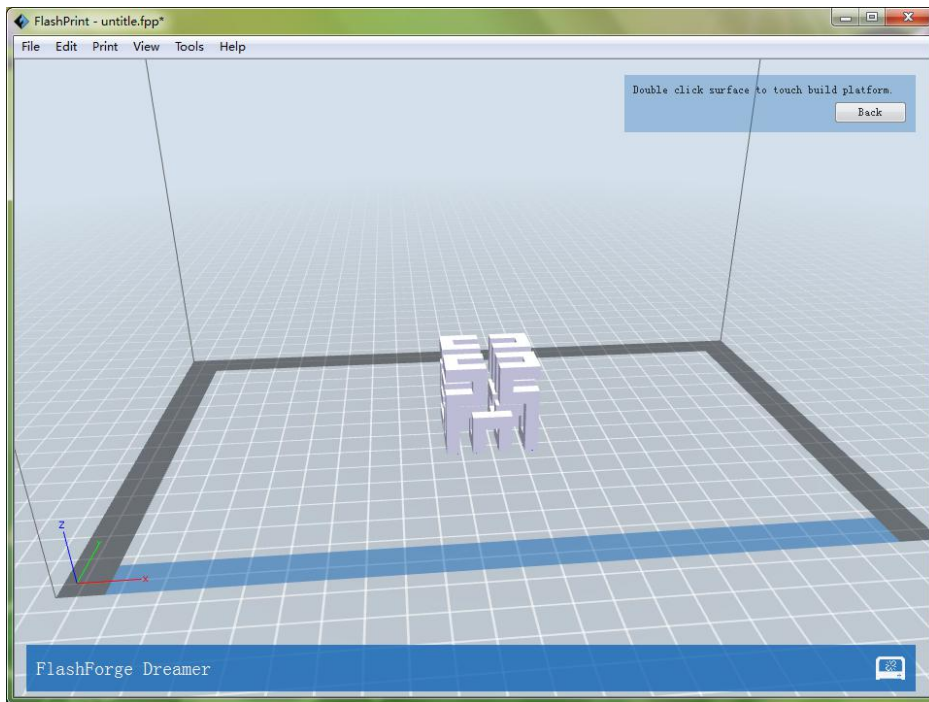
3.4.2.1 When the model is selected, you can change the orientation of your model on the build platform by following two ways:

- A. Click **Rotate** button on the left of the software interface and you can find three mutually perpendicular rings whose color is red, green and blue. Click one ring and rotate on the present axis, you will see the rotation angle and direction in the center of circle. In this way, you could make the model rotate on X/Y/Z axis.
- B. Click the **Rotate** button on the left of the software interface, and then enter into rotating angel values in X/Y/Z axes positioning. Click **Reset** button to reset rotating angel values.

3.4.2.2 Set side flat

After choosing model, operate as below to set side flat

- ◆ check "Rotate->Set side flat" ;
- ◆ Choose the side of model by mouse, double click the left mouse button on mouse floating side, the model side will be automatically set as flat, the selected side will plying-up bottom board.



3.4.3 Scale

When the model is selected, you could change the size of the model on the build platform by the following two ways.

A. Click **Scale** button on the left of the software interface, then press the left button and move the mouse to adjust the size of the model. You will see corresponding values of different axes near the borders.

B. Click **Scale** button on the left of the software interface, and then enter into scale values in X/Y/Z axes positioning. Click **Maximum** button to get largest size possible for building. Click **Reset** button to reset the size of model.

Note: Select **Uniform Scaling** option, it will scale the model in equal proportion when changing value in any positioning of the model. Not select this option, it will only change the value of the corresponding positioning.

3.4.4 Set

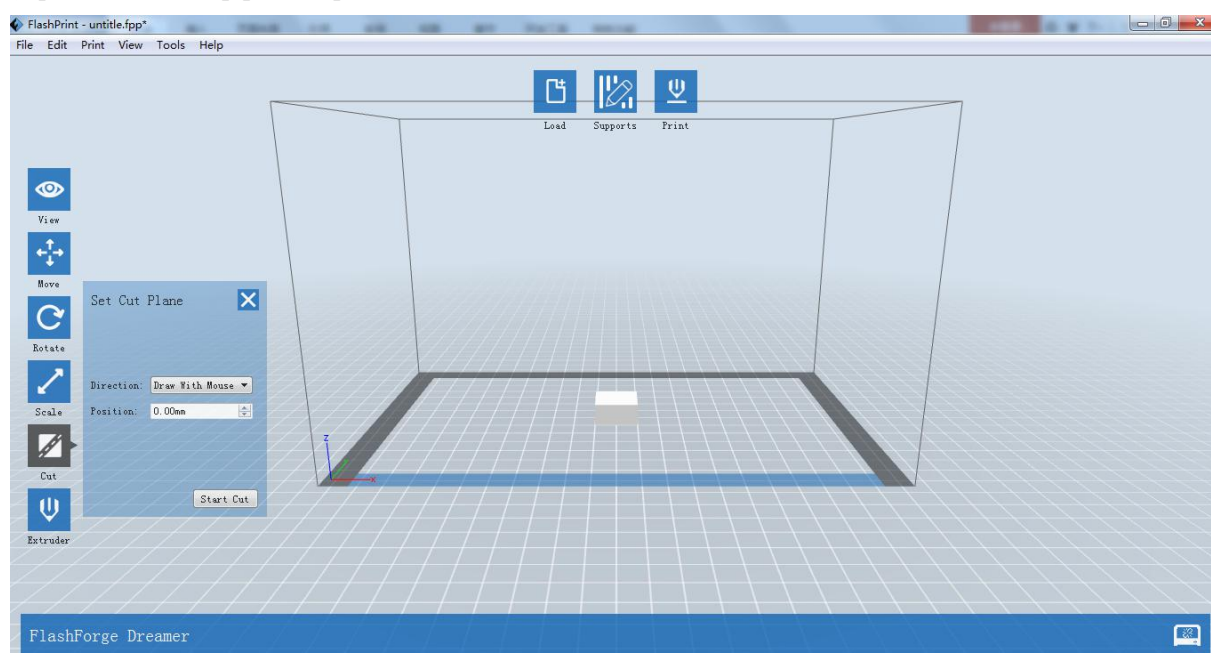
When the model is selected, click **Extruder** button on the left of the software interface and select the extruder you want to print with. The model will turn purple if you choose left extruder or gray if right extruder. If you choose 'FlashForge Adventurer 3/FlashForge Adventurer 3 Lite/FlashForge Finder/FlashForge Guider/FlashForge Guider II/FlashForge Guider II S Series/FlashForge Inventor II Series/FlashForge Dreamer NX/FlashForge Adventurer 3C', the software will be unavailable for nozzle setting and the default nozzle setting shall be "Right Nozzle". If you choose 'Flashforge Creator 3/FlashForge Creator Pro 2/FlashForge Creator Max 2', you can select duplicate or mirror according to your need to print using dual extruder in the meantime (choose 'use dual extruder (duplicate)', then dividing the build plate into two same printing area, generating two same models limited by the vertical plane in the build plate; choose 'use dual extruder (mirror)', then generating symmetrical models limited by the vertical plane in the build plate).

If a model can be printed with dual extruders or the support is needed to complete the printing, you can use dual extruders to print it with different color filaments or even different kinds of filament. However, dual extrusion printing is not available when the model can only be printed with one extruder. You need to know whether the model diagram has single or dual colors when selecting the model. Dual-color model is combined with two single color models.

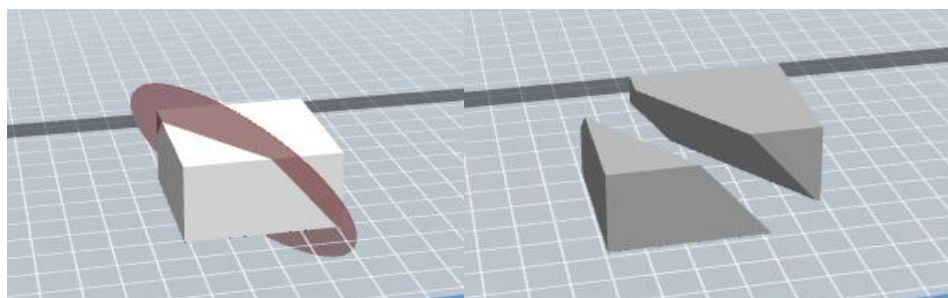
Note: When setting the extruder, mouse operations are same as those of changing views. Please refer to the Chapter 2.3.1~2.3.2.

3.4.5 Cutting the Model:

Left-click to select the target model and then click the **[Cut]** icon on the left to enter the Cut mode(Shown as follows). Users can set up the direction(Free-hand plane, X plane, Y plane, Z plane) ,position and keep parts in place.



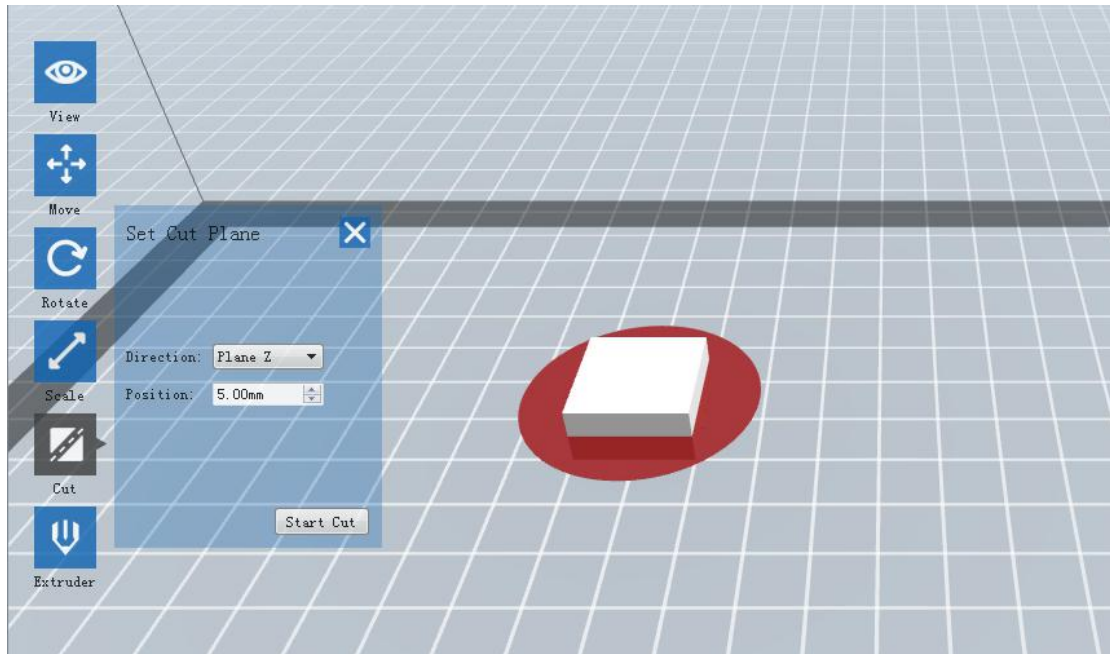
Free-hand plane: Left-click to select the target model and hold down the left mouse button to draw the cutting line (The cutting line will turn into a facet later). To change the position of the facet, users can(1) left-click to select the facet and hold down the left mouse button to drag the facet. (2) change the position values, then click the **[Start Cut]** button or double-click the model .



(Before Cutting)

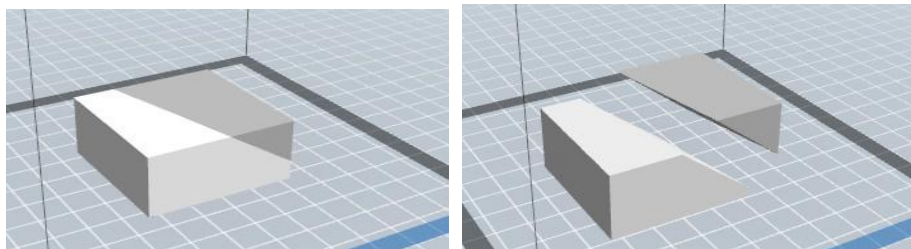
(After Cutting)

X, Y or Z Plane: Select the X, Y or Z plane in the **Direction** drop-down menu . Left-click to select the target model and the software will automatically generate the facet according to the cutting position. To change the position of the facet, users can (1) left-click to select the facet and hold down the left mouse button to drag the facet. (2) change the position values, then click the **[Start Cut]** button or double-click the model .



(Z Plane)

Keep parts in place: After finished cutting, the new model will be in their original position won't be shifted automatically anymore.



(Contrast pictures: Choosing function of keep parts in place or not)

3.4.6 Others

3.4.6.1 Undo

Undo the most recent edit you made to your model file by following two ways:

A. Click **Edit-> Undo** menu.

B. Use the shortcut **Ctrl+Z**.

Note: Selecting this option multiple times will continue to undo edits in reverse order in which they were performed.

3.4.6.2 Redo

Redo the most recent edit you have undone to your model file by following two ways:

A. Click **Edit-> Redo** menu.

B. Use the shortcut **Ctrl+Y**.

Note: Selecting this option multiple times will continue to redo edits in reverse order in which they were removed.

3.4.6.3 Empty Undo-stack

To clean up the recorded operating steps so as to release the memory.

3.4.6.4 Select ALL

By the following two methods, you could select all models in the scene.

A. Click **Edit-> Select ALL** menu.

B. Use the shortcut **Ctrl+A**.

Note: When models are too small to be seen or out of viewing scope, please click **Center** and **On Platform** buttons after selecting all models to bring them to the printing scope.

3.4.6.5 Duplicate

After model is selected, you can duplicate it by the following two methods:

A. Click the **Edit-> Duplicate** menu.

B. Use the shortcut **Ctrl+D**.

3.4.6.6 Delete

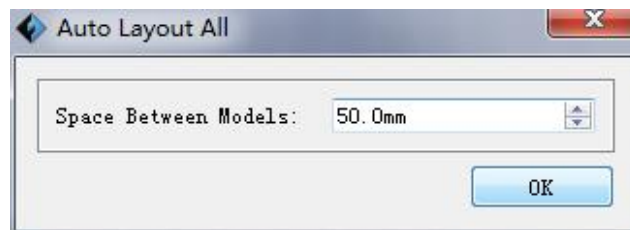
After model is selected, you can delete it by the following two methods:

A. Click the **Edit-> Delete** menu.

B. Use the shortcut **Del**.

3.4.6.7 Automatic put

After loading one model or several models, press "Edit->Automatic placing" and fill in space between models(0-50mm), all models will be put in position according to the automatic put rule.

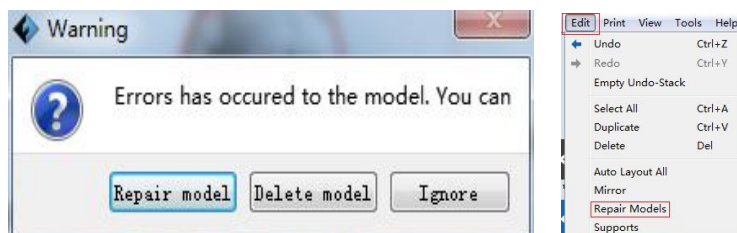


3.4.6.8 Mirror

To make mirror image in X-axis, Y-axis and Z-axis for the chosen model.

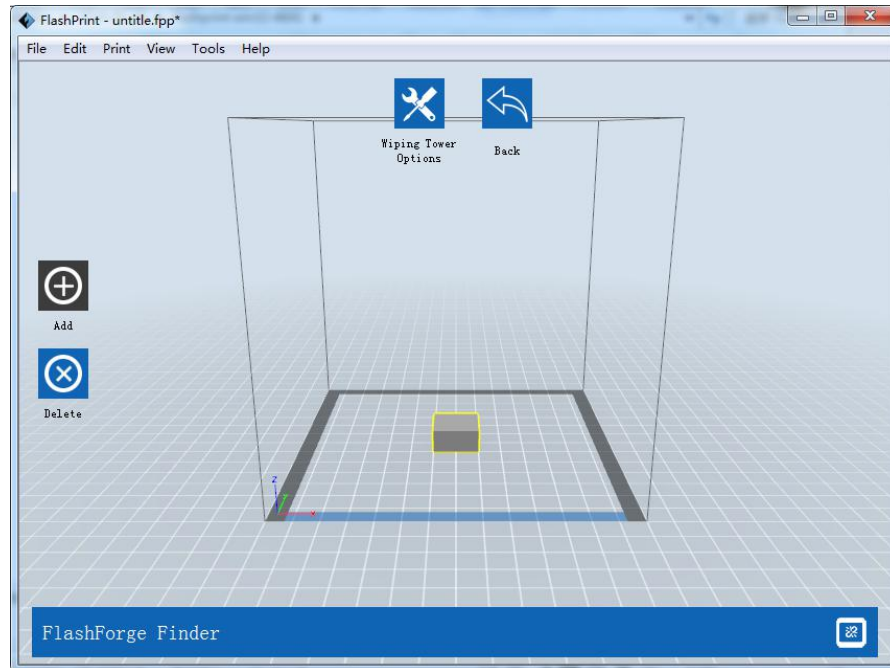
3.4.6.9 Model Repair

The software will detect the model when loading the model. If any problem is detected, a dialog box will pop up for reminding the users. Users can click the "Repair Model" button or select the model, click [edit]--[model repairing] to repair the model.

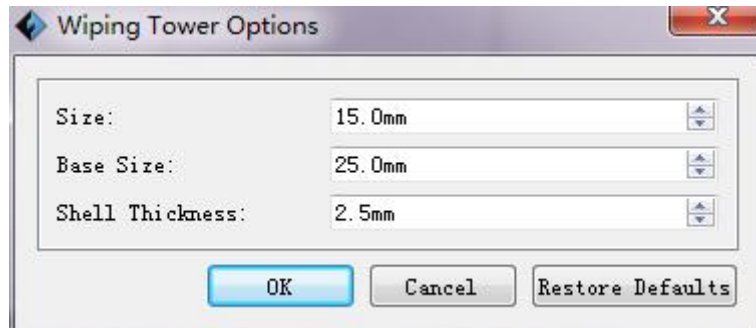


3.4.6.10 Wiping Tower

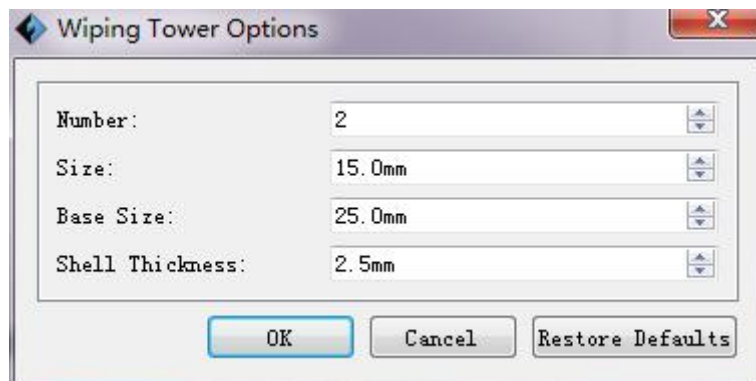
After importing the model, tap 'Edit->Wiping Tower', enter wiping tower editing interface.



single extruder 3D printer ,Support to set wiping tower size , Number , Base Size and shell thickness in wiping tower options, The default wiping tower height is the same as that of the loaded model.If multiple models exist, take the height of the highest model.



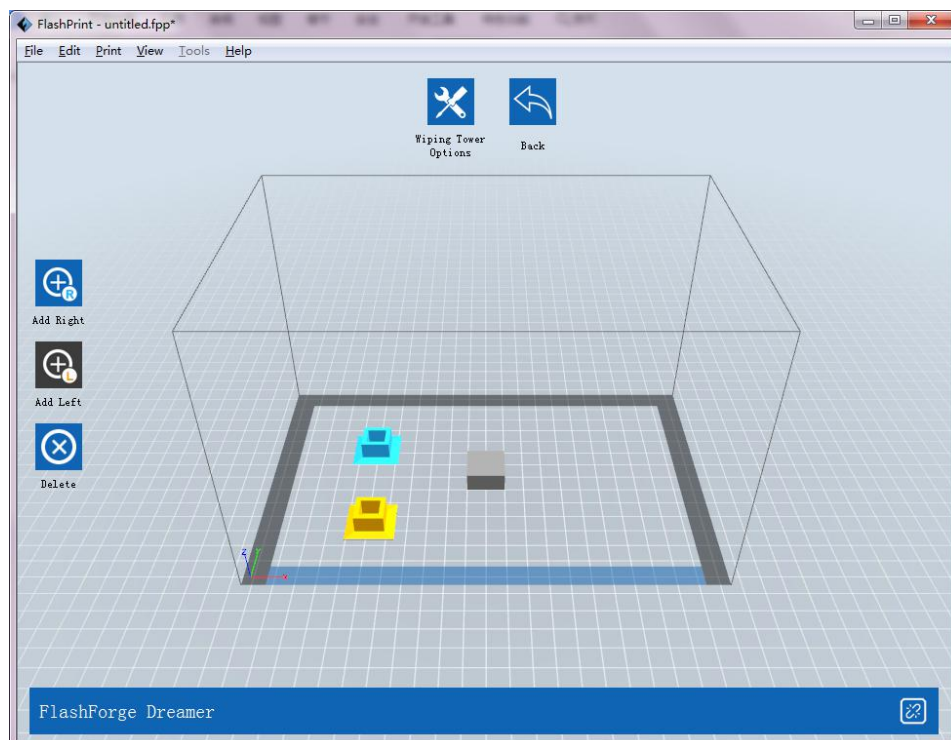
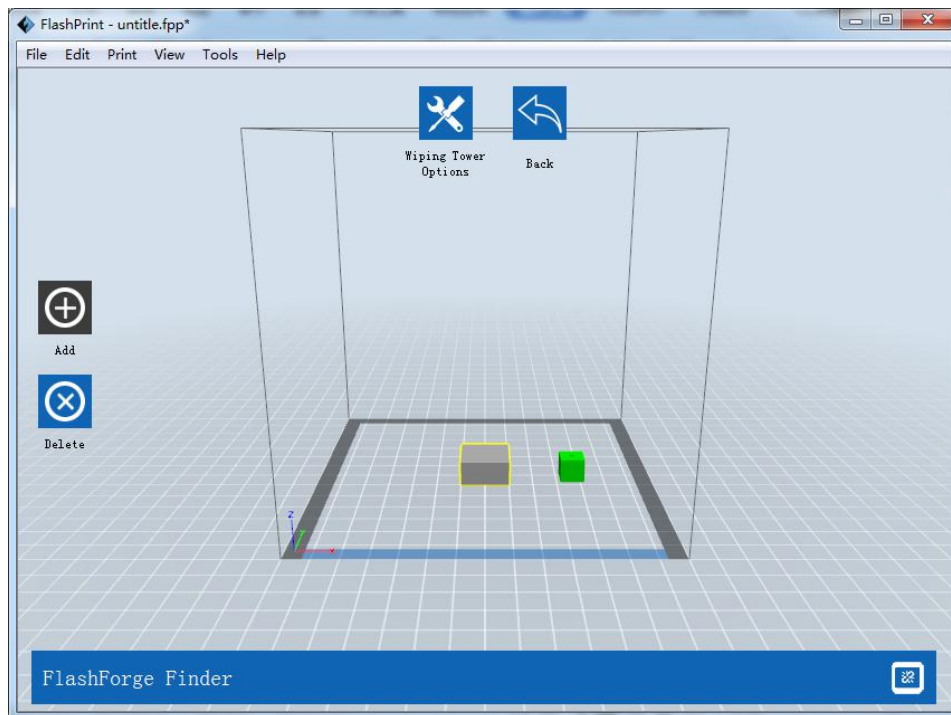
dual extruder 3D printer , Support to set wiping tower size , Base Size and shell thickness in wiping tower options, The default wiping tower height is the same as that of the loaded model.If multiple models exist, take the height of the highest model.



After setting the parameters of wiping tower, click 'add' button, move the mouse to select the


coordinates of wiping tower, and click the left mouse button to add the wiping tower.

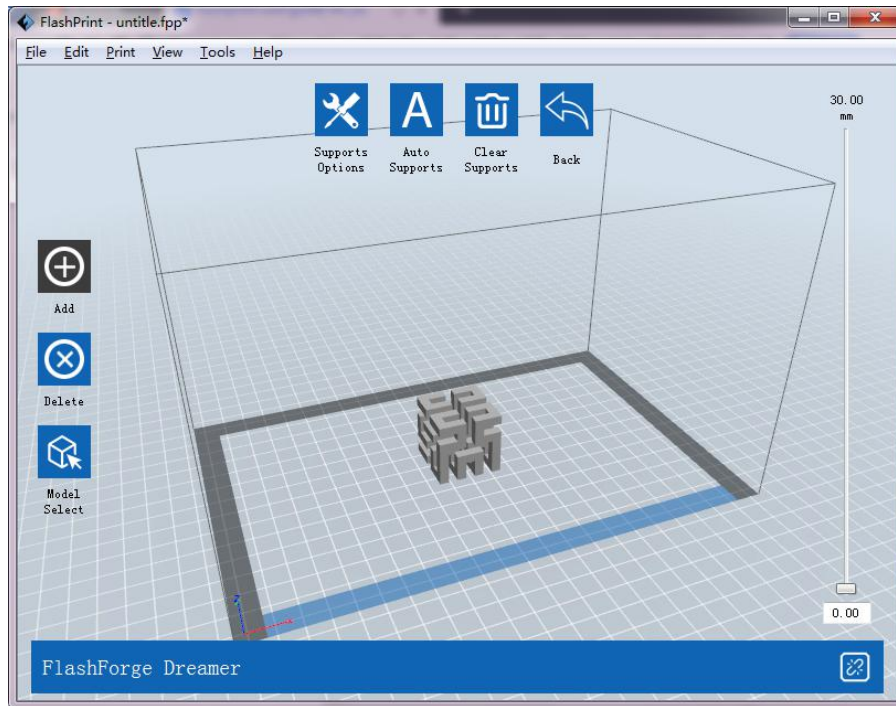
When the number of the wiping tower for a double-extruder 3D printer is 2, it permits to set a single left or right wiping tower or set two together.



click 'delete' when needing to delete the wiping tower , select the wiping tower with the left mouse button, delete successfully.

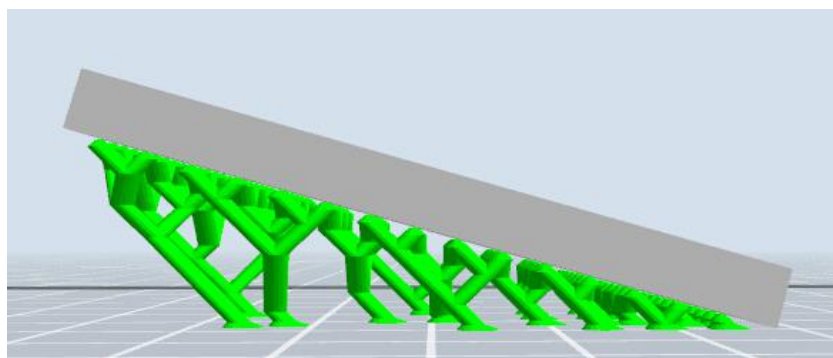
3.5 Edit Supports

After model file is loaded, click **Edit->Supports** in the top menu, or press  button in homepage, you can edit the support structures of your model. Click **Back** on the top-right corner to exit when finishing the edits.



3.5.1 Auto Supports

Select this option to have your model build with support structures. FlashPrint software will automatically generate treelike, Linear, columnar support, and it will generate new support structures to replace original ones included in your model file.



3.5.2 Clear Supports

Select this option to empty all the supports of your model. Click **Edit->Undo** in the menu or **Ctrl+Z** to undo this operation.

3.5.3 Add

Select this option to add supports (shortcut: keyboard—A) . After selecting this option, move the mouse to where you want to add the support, and then left-click the mouse to select the start point

of support structure. Press and hold the left mouse button down, drag the mouse, and then you can preview the support structure you will add (If the support surface does not need to support or the angle of support column is too large, the previewed support structure will be highlighted.) Release the left button of mouse to add this support, the support structure will be generated according to the start and the end position. But if the support column intersects with the model or the previewed support structure is highlighted, support structure will not be generated.

Choose "show model slice outline" on the view menu bar, when the mouse pointer moves to the model, display the corresponding model cross-sectional outline on which the mouse points, in order to better observe and select support position.

3.5.4 Delete

Select this option to delete supports (shortcut:keyboard—D) . After selecting this option, move the mouse to the supports you want to delete, then left-click the mouse to delete it. Press Ctrl button and mouse left to delete support in region selection.

3.5.5 Model Select

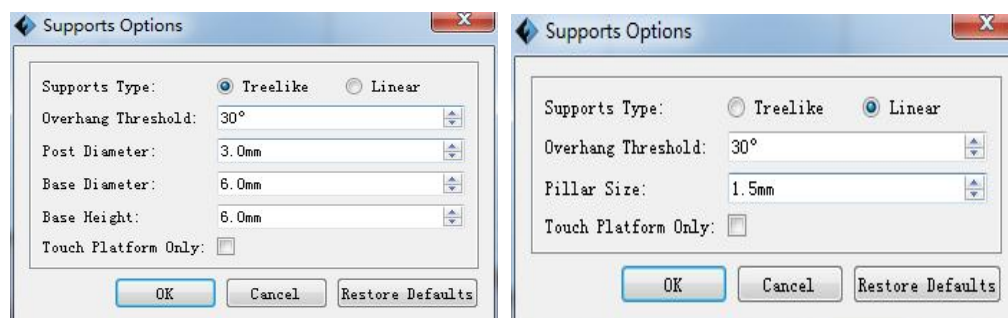
Select this option to Model select supports (shortcut:keyboard—S) .Only add Auto Supports or do Clear Supports to the selected model. Add different support-settings for different models when print file include more than one model.

3.5.6 Support option

3.5.6.1 FDM 3D Printer

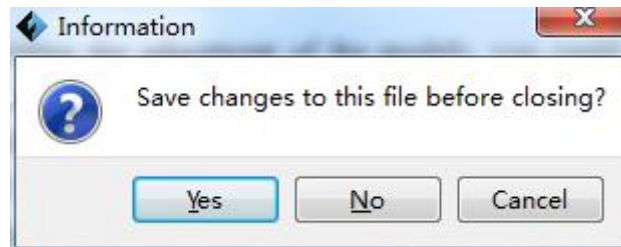
Click [Supports Options] and then a dialog box will pop up for supports type selection and parameter settings. Linear and treelike supports are available for selecting. If you select 'Treelike Supports', you can setup the parameters of overhang thresh, post diameter, base diameter and base height, and then click [OK] to confirm. If you select 'Linear Supports', you can setup overhang thresh and pillar size, then click [OK] to confirm, when choose Linear, output is Linear structure. If the model already has support, when you choose support type, the software will judge the existing support type and decide if it is necessary to delete these supports. It will jump out reminding, you can operate as you Need. (Note:

Overhang thresh: To setup an angle value. If you input 45, then when it comes to a 3D model with an overhang of 45 or more than 45 degrees. The system will generate supports for these positions. The value extends from 30 to 60. Post diameter: The diameter of the treelike supports. The value range is 1-6mm. Base diameter: The diameter of the support raft. The value range is 3-10mm. Base height: The height of the support rafts. The value range is 1-6mm. Pillar size: The side length of the square column. Touch Platform Only: Only create the supports contact with the build plate.)



3.6 New Project

Click **File -> New Project** to create a new project. If your current project hasn't been saved with changes, it will pop up a dialog to prompt you to save them. Click **Yes** button to save those changes, click **No** button to discard them, Click **Cancel** button to close this dialog and it will not create a new project.



3.7 Saving File

You can save changes of your model files by following two ways:

A. Click **File -> Save Project** to save your file only with .fpp format. All the models(including supports) in a .fpp format file are independent and they can be edited again when reloading the file. Besides, extruders' configuration and models' locations remain same as saving the file.

B. Click **File -> Save As** to save your file with .stl, .obj,.3mf or .fpp format. All the models in a .stl or .obj format file are not independent, but combined to a new model. Models' locations remain same as saving the file, while extruders' configuration becomes different when reloading the file. When a model with main part and support need to be saved, it is recommended to save the file as .fpp format. If you save the file as ".Stl" , ".Obj" or ".3mf", The model and the treelike supports in the plate are not independent of each other, while the model and the linear supports are independent of each other, the liner supports won't be saved with the model. The software will not be able to distinguish the original file model from the support and print Quality will be affected.

3.8 Printing Process

3.8.1 Choose Machine Type

Click **Print-> Machine Type** to choose the machine type you have before connecting to the printer, or the FlashPrint will automatically recognize the machine type after connecting, in this case you can not manual switch to other machine type(s). Different machine types have different machine frames which will affect sliced file(s).

If the selected machine type is 'FlashForge Guider II S Series/Creator 3/Adventurer 3', the nozzle size of the printer can be selected in the "Print-Nozzle Size" expanded item.

3.8.2 Connect Machine

Please refer to Chapter 2.9.1.1 for configurations of connecting to Dreamer 3D printer.

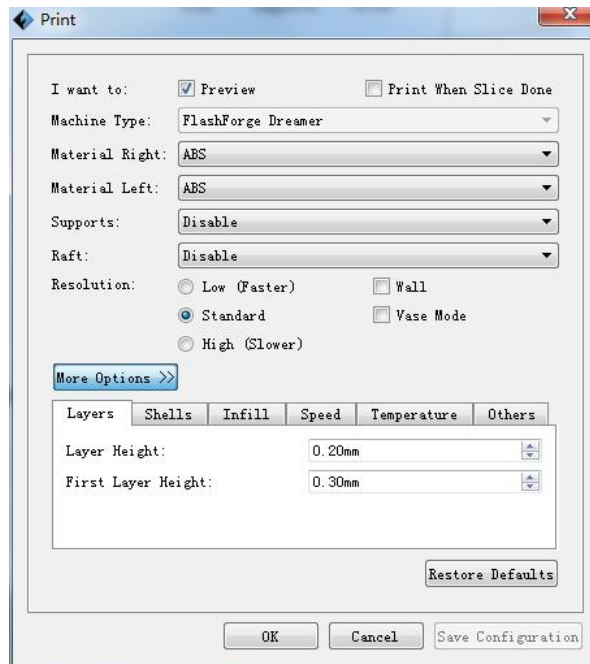
3.8.3 Print

3.8.3.1 FDM 3D printer generate Gcode file

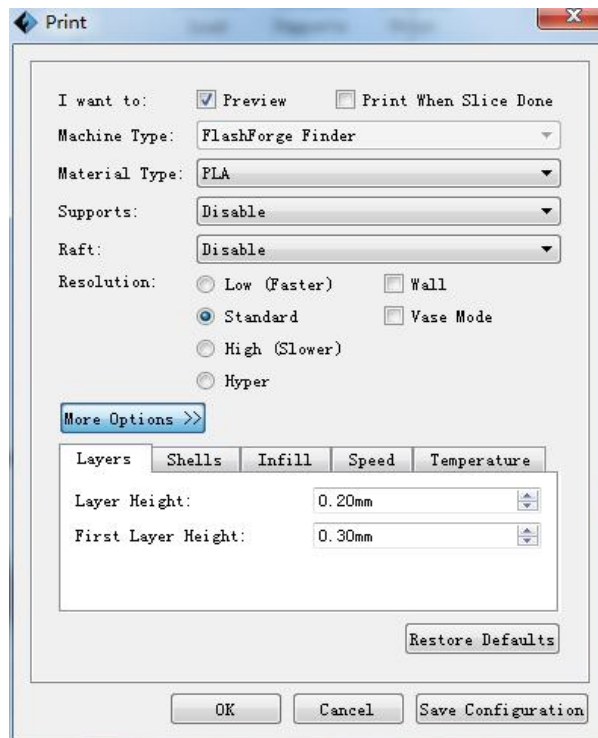
In order to print, you need to slice the model file to generate gcode file. Configuring slicing Parameter and generating slicing files by following steps.

1. Click **Print -> Print** menu or **Print** button on top of the software interface, it will pop up a dialog

box for slicing. If users choose 'Flashforge Dreamer/Flashforge Creator Pro/FlashForge Inventor Series/Flashforge Creator 3/Flashforge Creator Max/FlashForge Creator Pro 2/FlashForge Creator Max 2', the corresponding dialog box of slice parameters shall be "figure 1". If users choose 'FlashForge Adventurer 3/FlashForge Adventurer 3 Lite/FlashForge Finder/FlashForge Guider/FlashForge Guider II/FlashForge Guider II S Series/FlashForge Inventor II Series/FlashForge Dreamer NX/FlashForge Adventurer 3C', the corresponding dialog box of slice parameters shall be 'figure 2'.

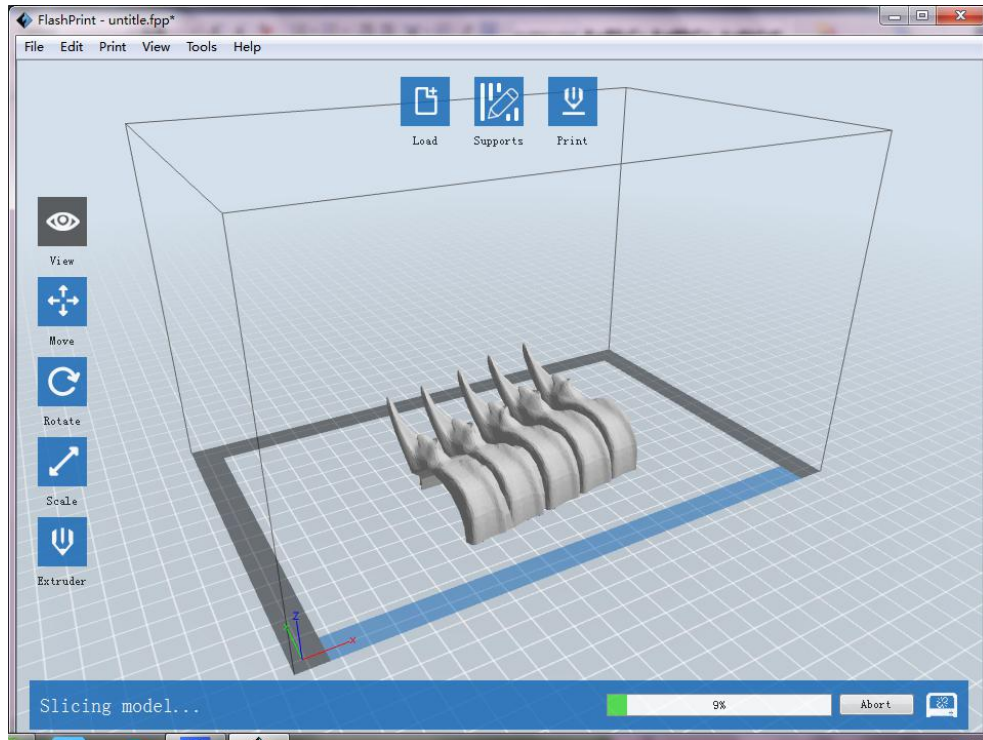


(figure 1)

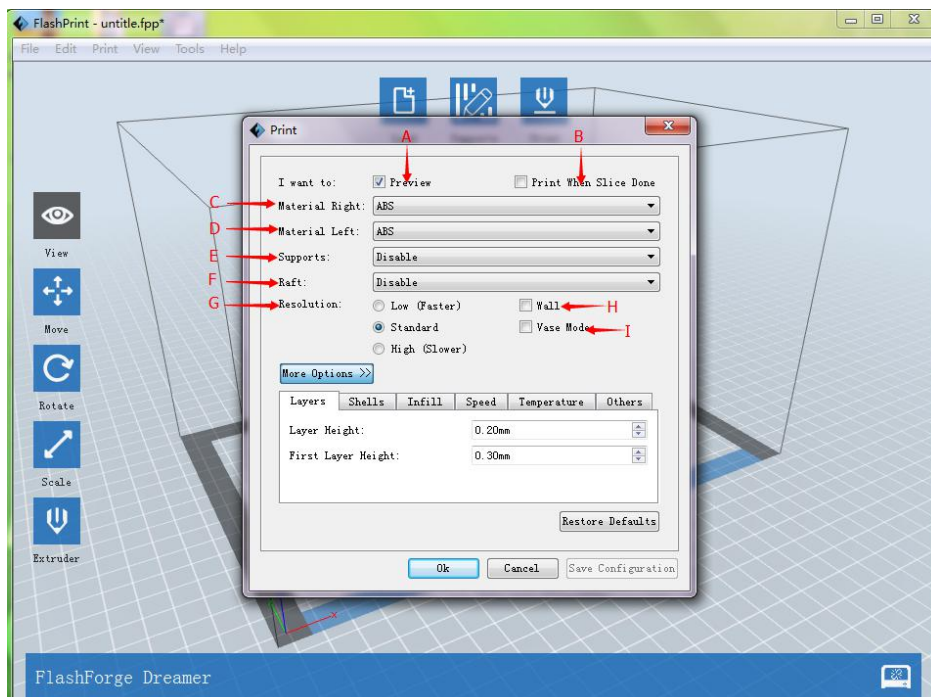


(figure 2)

2. Set slicing parameters and click **Ok** button, and it will pop up a dialog box for saving Gcode file.
3. In the **Save as** dialog box, select the path for saving the Gcode file, and then click **Save** button. You can see the bar at the bottom of software interface start to show the completion progress of slicing, which means it is slicing and uploading Gcode file now. Click **Abort** button on the right side of the process bar to give up slicing.



3.8.3.2 Explanation of standard Slicing settings



A. Floating boxes for slicing proper nouns: Move the cursor to the specific position that a slicing proper noun or a parameter lies and linger for a while, a floating box will appear with the explanation inside.

B. Preview: Select this option to preview the Gcode file when slicing finished.

C. Print When Slice Done: Select this option to directly print the model.

D. Material Right: Choose filament type you want to print with right extruder.

E. Material Left: Choose filament type you want to print with left extruder.

Material Type: Select the appropriate nozzle and material according to the settings of nozzle.

If users choose 'Flashforge Dreamer/Flashforge Creator Pro/FlashForge Inventor Series/Flashforge Creator 3/Flashforge Creator Max/FlashForge Creator Pro 2/FlashForge Creator Max 2', select corresponding material for the 'left/right extruder'. If users choose 'FlashForge Adventurer 3/FlashForge Adventurer 3 Lite/FlashForge Finder/FlashForge Guider/FlashForge Guider II/FlashForge Guider II S Series/FlashForge Inventor II Series/FlashForge Dreamer NX/FlashForge Adventurer 3C', select corresponding material .

F. Supports: Used to hold the overhangs of model. Choose Left or Right Extruder to print supports when overhang is existing, or Disable when no need.

Click the [Supports] button, if the machine type is 'Flashforge Dreamer/Flashforge Creator Pro/FlashForge Inventor Series/Flashforge Creator 3/Flashforge Creator Max/FlashForge Creator Pro 2/FlashForge Creator Max 2', select left/right extruder for supports adding. If the machine type is 'FlashForge Adventurer 3/FlashForge Adventurer 3 Lite/FlashForge Finder/FlashForge Guider/FlashForge Guider II/FlashForge Guider II S Series/FlashForge Inventor II Series/FlashForge Dreamer NX/FlashForge Adventurer 3C', choose 'Able' or 'Disable' for supporting setting.

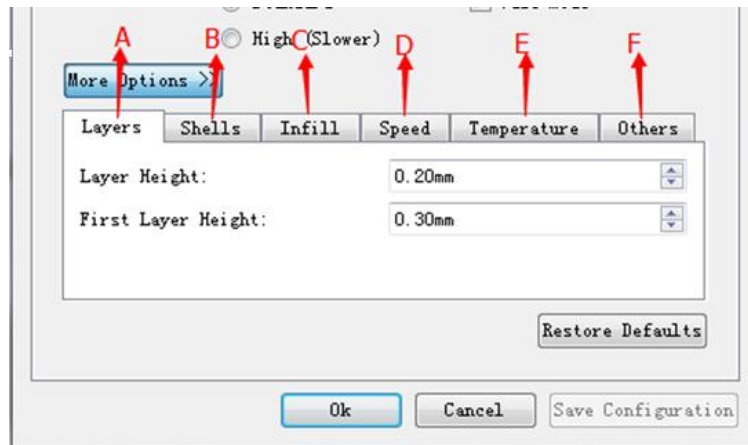
G. Raft: Creates a raft below the model which can help with adhesion and tolerance for non-perfect build plates. Choose Left or Right Extruder to print raft which can make the model attach well to the build plate, or Disable when no need. Click the [Raft] drop-down button, if it is 'Flashforge Dreamer/Flashforge Creator Pro/FlashForge Inventor Series/Flashforge Creator 3/Flashforge Creator Max/FlashForge Creator Pro 2/FlashForge Creator Max 2', choose left/right extruder for raft setting. If it is 'FlashForge Adventurer 3/FlashForge Adventurer 3 Lite/FlashForge Finder/FlashForge Guider/FlashForge Guider II/FlashForge Guider II S Series/FlashForge Inventor II Series/FlashForge Dreamer NX/FlashForge Adventurer 3C', choose 'Able' or 'Disable' for raft setting.

H. Wall: Creates a wall around the model to scrape oozing from the idle extruder. It's suggested to enable in dual extrusion printing.

I. Brim: Expand the outline of model's bottom layers to a Brim which helps anchor the edges of the model to the plate to avoid warping.

J. Vase mode: It can be used to print the single layer model without seams. Under this circumstance, the number of perimeter shells is constant to be 1, the density of infill is constant to be 0, and the number of top solid layers is default to be 0.

K. Resolution: When printing right extruder with ABS or dissolvable filament, you can select High option with better print but lower speed, or Standard option, or Low option with normal print but high speed. Each option has its own configured parameter. While printing right extruder with PLA filament, you have the fourth option, which is Hyper with hyperfine print. If users choose 'Flashforge Finder/Guider', users only need to set up the nozzle temperature according to the material. The nozzle temperature for PLA is recommended to be 220°C.



Click **More Option** button to set the parameters as you want. Different resolutions have different default parameters. Click **Restore Defaults** to restore it.

A. Layers

- a. **Layer Height** : Thickness of each layer. The less thickness of layers, the more time will be used and the better model will be printed .
- b. **First Layer Height:** When printed with thinner layers, comparative thicker bottom layer could improve adhesion and tolerance for non-perfect build plates.

B. Shell : Including outside shell quantity, sealing top layers, sealing bottom layers(When Vase model selected, sealing top layers invalid)

- a. **Outside shell quantity:**Number of paths superimposed for building the shell, more paths make thicker shell.
- b. **Sealing top layers:** Number of solid layers on the upper surface of model.
- c. **Sealing bottom layers:** Number of solid layers on the under surface of model.

C. Infill

- a. **Fill Density** :Determines the interior solidity of the model.
- b. **Fill Pattern:**Determines the infill pattern for the interior of model. Hexagon infill is with the highest strength while line infill is with the shortest print time. Triangle infill takes slightly more time than line infill, but with more cohesiveness between layers,3D Infill is a 3D spiral infill shape,generally used when infilling density is lower,and it is recommended to turn off combined infill function.
- c. **Combine Infill:** This option allows to combine infill and speed up your print by extruding thicker infill paths. If layer height is larger than 0.1mm, “Every N Layers” is suggested, otherwise, “Every N Inner Layers” is suggested.

Layers	Shells	Infill	Speed	Temperature	Others
Fill Density:			15%		
Fill Pattern:			Hexagon		
Combine Infill:			Every 2 Sparse Layers		

D. Speed

a. Print Speed: Used to calculate movement speed during extrusion. The slower the speed is, the more time will be used and the better model will be printed..

b. Travel Speed: Speed at which extruder moves when not extruding filament.

Note: Recommend set the empty speed at 80, printing speed at 60 when use PLA, ABS,HIPS,PA,PC,PETG, Wood and conductive PLA filament printing, use flexible material set empty speed at 70, printing speed at 20,use PA set empty speed at 70, printing speed at 50. Different model have different model parameter, several attempts to find the suitable Parameter.

Different types of printing material and different quality of slice project, the speed value of configuration will be different, users can get the recommended values of different schemes in the default configuration.

E. Temp.

a. Extruder

Set proper printing extruder temperature, according to material. The maximum temperature is 255°C and the hint message will appeared when nozzle temperature is over 245°C.If the machine type is “FlashForge Dreamer/ FlashForge Creator Pro/ FlashForge Inventor Series/Flashforge Creator 3/Flashforge Creator Max/FlashForge Creator Pro 2/FlashForge Creator Max 2” dual extruder 3D printer, Users can set the temperature of dual extruder. As shown below:

Layers	Shells	Infill	Speed	Temperature	Others
Right Extruder:			220°C		
Left Extruder:			220°C		
Platform:			105°C		

If the machine type is “FlashForge Adventurer 3/FlashForge Adventurer 3 Lite/FlashForge Finder/FlashForge Guider/FlashForge Guider II/FlashForge Guider II S Series/FlashForge Inventor II Series/FlashForge Dreamer NX/FlashForge Adventurer 3C” single extruder 3D printer, the setting interface as follow:

Layers	Shells	Infill	Speed	Temperature	Others
Extruder:			220°C		

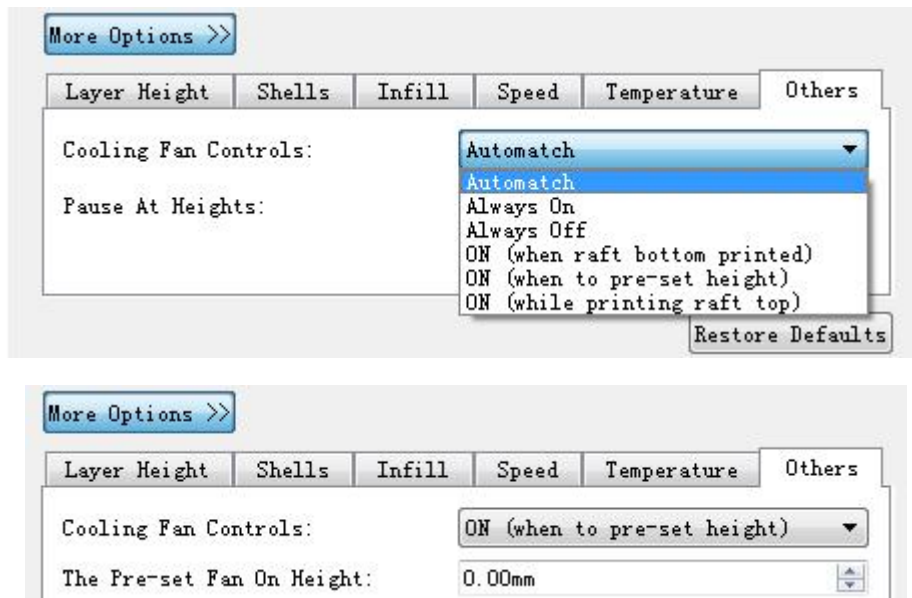
b.Set the temperature for left or right extruder and platform. Set **220** to the **extruder temp** for **ABS**,conductive PLA and flexible material **printing**. Set 50 to the **platform temp** for **PLA**,conductive PLA and flexible material **printing** ,while 105 for **ABS printing**. Set

(left: 240, right: 220) to the **extruder temp** for **dissolvable Filament printing**, Set **105** to the **platform temp**. If users choose 'FlashForge Adventurer 3/FlashForge Adventurer 3 Lite/FlashForge Finder/FlashForge Guider/FlashForge Guider II/FlashForge Guider II S Series/FlashForge Inventor II Series/FlashForge Dreamer NX/FlashForge Adventurer 3C', users only need to set up the nozzle temperature according to the material. The nozzle temperature for PLA is recommended to be 220°C, Conductive PLA and flexible material temperature is recommended to set at 230°C.

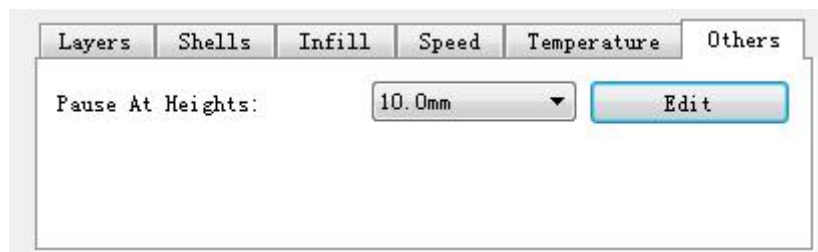
Adjust the temperature according to the condition in order to get a good print.

F. Others

a. Cooling fan controls: Set up the time to turn on the cooling fan. Only used for FlashForge Adventurer 3/FlashForge Adventurer 3 Lite/FlashForge Dreamer/FlashForge Creator Pro/FlashForge Guider II/FlashForge Guider II S Series/FlashForge Inventor Series/Flashforge Creator 3/Flashforge Creator Max/FlashForge Adventurer 3C/FlashForge Creator Pro 2/FlashForge Creator Max 2 at present. You can pre-set the height and make the cooling fan begin to work at the point.



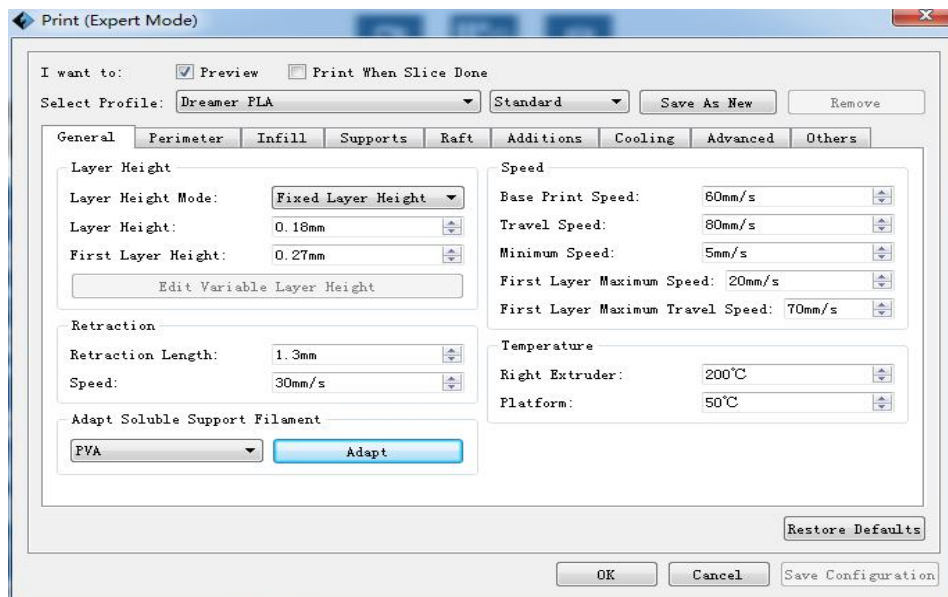
b. Suspend at a Certain Point: Turn on the cooling fan when to specific height. Users can add one or more specific height values (no higher than the model height). It displays as follows.



Steps: Click the [Edit] button, and a box will pop up (See graphic below). Then add the value, click [+] for confirmation, and the value will show in the list of [Delete Value]. If deletion is required, select the specific value on the list and click [-] for confirmation. Click [OK] to finish and exit these steps.



3.8.3.3 Explanation of Expert Slicing settings



A. Preview: Select this option to preview the Gcode file when slicing finished.

B. Print When Slice Done: Select this option to directly print the model.

C. Select Profile:

D. General:

1. Layer Height:

a. Layer Height Mode

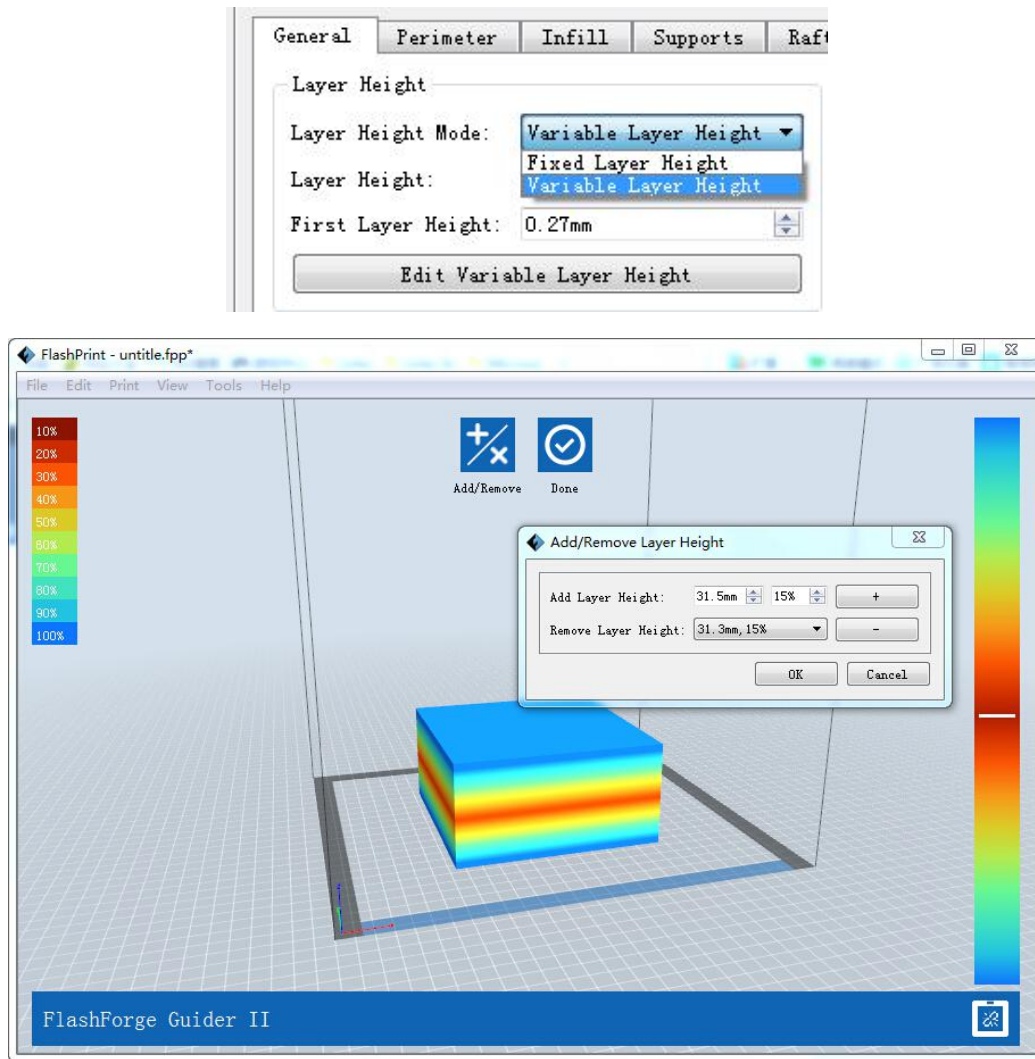
1. Fixed Layer Height

2. Variable Layer Height: click “various layer height” and “Edit variable layer height”, pop-up variable layer height editing dialog box

① **Add/Delete icon:** Click the icon, pop-up variable layer height editing dialog box, which can add or delete the variable layer height in the specified height.

② **The model layer height column:** right click the model layer height column and Add/Delete icon, then pop-up variable layer height editing dialog box. The default added height is the height mouse clicked. The default deleted height is the closest height has been added as various layer height.

③ click “Done” button and enter the expert mode setting dialog box.



- b. **Layer Height:** Thickness of each layer. The less thickness of layers, the more time will be used and the better model will be printed.
- c. **First Layer Height:** When printed with thinner layers, comparative thicker bottom layer could improve adhesion and tolerance for non-perfect build plates.

2. Speed:

- a. **Base Print Speed:** Base speed at which extruder moves during extrusion, which is the base value of subsequent speeds counting.
- b. **Travel Speed:** Speed at which extruder moves when not extruding filament.
- c. **Minimum Speed:** The minimum movement speed during extrusion..
- d. **First Layer Maximum Speed:** To limit the print speed of first layer, which will help with adhesion. When enabling Raft, this limit will not apply.
- e. **First Layer Maximum Travel Speed:** To limit the travel speed of first layer, which will help with adhesion. When enabling Raft, this limit will not apply.

3. Temperature

- a. **Extruder's temperature:** Extruder's temperature during printing. The maximum temperature is 255°C and the hint message will appeared when nozzle temperature is over 245°C.

- b. **Extruder and Platform:** Appropriate temperature of build plate can improve adhesion and help large models avoid warping. If the machine type is “FlashForge Adventurer 3/FlashForge Adventurer 3 Lite/FlashForge Dreamer/FlashForge Creator Pro/FlashForge Guider II/ FlashForge Guider II S Series/FlashForge Inventor Series/FlashForge Dreamer NX/Flashforge Creator 3/Flashforge Creator Max/FlashForge Adventurer 3C/FlashForge Creator Pro 2/FlashForge Creator Max 2”, it displays the Temperatures of “Left/Right Extruder and Platform”. If the machine type is “FlashForge Finder/FlashForge Guider/FlashForge Inventor II Series”, it only displays the temperature of “Extruder”. Set temperature according to the filament type: Set the temperature for left or right extruder and platform. Set **220** to the **extruder temp** for **ABS**, conductive PLA and flexible material **printing**. Set 50 to the **platform temp** for **PLA**, conductive PLA and flexible material **printing**, while 105 for **ABS printing**. Set (left: 240, right: 220) to the **extruder temp** for **dissolvable Filament printing**, Set **105** to the **platform temp**. If users choose ‘FlashForge Finder/FlashForge Guider/FlashForge Inventor II Series’, users only need to set up the nozzle temperature according to the material. The nozzle temperature for PLA is Recommended to be 220°C, Conductive PLA and flexible material temperature is recommended to set at 230°C.

Adjust the temperature according to the condition in order to get a good print.

4. Retraction

- a. **Retraction length:** Length of filament to be pulled back into nozzle before extruder traveling or changing, which help prevent from stringing and oozing.
- b. **Speed:** Speed at which the filament is retracted.

If choosing the machine type of FlashForge Creator Max 2, FlashForge Dreamer, FlashForge Creator Pro, FlashForge Inventor Series, FlashForge Creator 3, FlashForge Creator Pro 2 or FlashForge Creator Max and using dual-extruder to print, you can set retraction length and speed of right/left extruder.

Retraction	
Right Extruder Length:	1.3mm
Right Extruder Speed:	30mm/s
Left Extruder Length:	1.3mm
Left Extruder Speed:	30mm/s

- c. **.Only Retract When Crossing Outline:** Used to reduce retraction, but stringing appears easily on top solid layer. Choose “No” to avoid the issue. Only used for ‘FlashForge Adventurer 3/FlashForge Adventurer 3 Lite/FlashForge Adventurer 3C’ at present.

Retraction

Retraction Length: 5.0mm

Speed: 25mm/s

Only Retract When Crossing Outline: Yes

E. Perimeter:

1. Thickness:

- a. **Shell Count:** Number of paths superimposed for building the shell, more paths make thicker shell.
- b. **Maximum Path Overlap:** The maximum allowed overlap for perimeters inside a thin wall.

2. Start Points:

- a. **Mode:** Determines the rule of selecting layer start points. There are three options for Start Point mode. One is “closest to specific location”, the other is “use random start points”, the other is “Inner recess point”.

Closest to specific location: All layer start points are aligned as close as possible to specified XY location.

Use random start points: Layer start points are randomly distributed all over the model.

Inner recess point: Priority use “inner recess point” as the start point, or use “closest to specific location” as the start point.

- b. **X:** X-coordinate where all layer start points are aligned.
- c. **Y:** Y-coordinate where all layer start points are aligned.
- d. **Permit optimize start points:**

Allow to adjust start points location when needed, which select a location as start point except "closest to specific location". Suggest to select "No" when printing upright reliefs.

3. Speed:

- a. **Exterior Speed:** Percentage of Base Print Speed for printing outer-most perimeter.
- b. **Exterior Maximum Speed:** To limit the print speed of outer-most perimeter.
- c. **Visible Interior Speed:** Percentage of Base Print Speed for printing inner perimeters which is visible when print done.
- d. **Invisible Interior Speed:** Percentage of Base Print Speed for printing inner perimeters which is invisible when print done.

F. Infill:

1. General

- a. **Top Solid Layers:** Number of solid layers on the upper surface of model.
- b. **Bottom Solid Layers:** Number of solid layers on the under surface of model.

- c. **Fill Density:** Determines the interior solidity of the model.
- d. **Fill Pattern:** Determines the infill pattern for the interior of model. Hexagon infill is with the highest strength while line infill is with the shortest print time. Triangle infill takes slightly more time than line infill, but with more cohesiveness between Layers, 3D Infill is a 3D spiral infill shape, generally used when infilling density is lower, and it is recommended to turn off combined infill function.
- e. **Overlap Perimeter:** Percentage of extrusion width that will overlap between infill and perimeters.
- f. **Vase Mode:** When enabled, the interior infill and top solid infill will not be printed.

2. Speed

- a. **Solid Speed:** Percentage of Base Print Speed for printing top and bottom solid infill.
- b. **Sparse Speed:** Percentage of Base Print Speed for printing interior infill.

3. Combine Infill

- a. **Maximum Solid Combine Layers:** This option allows to combine the overlapped infill to speed up printing. After combined, infill will be thicker while the thickness of perimeter keeps unchanged. And the top/bottom combine layer height is not suggested more than 0.2mm.
- b. **Maximum Sparse Combine Layers:** This option allows to combine the overlapped infill to speed up printing. After combined, infill will be thicker while the thickness of perimeter keeps unchanged. And the inner combine layer height is not suggested more than 0.36mm.
- c. **Combine Area Threshold:** When printing area of an layer is smaller than the threshold area, Flashprint will not do the combined infilling in this layer, to increase the printing time of this layer and make sure filament has cooled down before printing next layer.

4. Strength Infill

- a. **Interval Layers :** At regular intervals layer, force several solid layers to make model more solid. 0 means not activated.
- b. **Solid Layers:** The number of solid layers force added.

G.Support:

1.General

- a. **Enable Support:** Used to hold the overhangs of model. Allows users to turn on/off the support option. Support structure can prevent the model from collapsing. If choosing “Yes”, then both treelike and linear supports are available for setting up. If choosing “No”, then both treelike and linear supports are unavailable for setting up.
- b. **Select Extruder:** Allows users to select the corresponding extruder. If the machine type is “FlashForge Dreamer/FlashForge Creator Pro/ FlashForge Inventor Series/Flashforge Creator 3/Flashforge Creator Max/FlashForge Creator Pro

2/FlashForge Creator Max 2”, it’s available for users to select extruder. If selecting “Automatch”, the software will select the extruder automatically depending on the extruder(s) used. If dual extruders are used, then the right extruder will be used for supports printing. If the machine type is “FlashForge Finder/Gudier”, it’s unavailable for users to select extruder.

- c. **Adapt soluble support filament:** Adjust the preinstalled support slicing parameters to fit more for printing soluble material supports.

2.Treelike

- a. **Speed:** Percentage of Base Print Speed for printing treelike support.
- b. **Space to Model(X/Y):** Determines the minimum horizontal space between model and supports, which makes supports easier to be removed.
- c. **Shell Count:** Number of paths superimposed for building treelike support shell. More paths will generate a thicker skin.
- d. **Auto Setup Z Hop :** Auto setup Z Hop parameters. Lift the extruder or lower the platform when the extruder is traveling, to avoid the tree support from being knocked down by the extruder.

3.Linear

- a. **Speed:** Percentage of Base Print Speed for printing linear supports.
- b. **Space to Model(X/Y):** Determines the minimum horizontal space between model and supports, which makes supports easier to be removed.
- c. **Space to Model(Z):** Determines the vertical space between model and supports, which makes supports easier to be removed.
- d. **Space to raft(Z):** Determines the vertical space between raft and support, which makes supports easier to remove.
- e. **Path Space:** Determines the space between adjacent support paths. Smaller space give finer surface supported, but take more time to print and against to supports Removing.
- f. **Support Thickness:** Setting higher thickness to get the more solid support, while lower thickness result in easy-remove support.
- g. **Top Solid Layers:** Solid top of linear supports will improve printing quality of interface between model and supports, but makes supports harder to remove.
- h. **Print Outline:** print outline of line support region or not? print outline to avoid the problem of incomplete region with line support, otherwise it is better to strip support.
- i. **Path Angle:** The angle between support printing path and the first layer infill printing path.
- j. **Path Shape:** Set path shape of linear support, polyline-shaped path is more beneficial for support removal while grid-shaped path support structure is more stable.
- k. **Path Density:** Set support path density, the larger path density makes more stable support, but longer printing time and more filament consumption as well.

H. Raft:

General:

- a. **Enable Raft:** Creates a raft below the model which can help with adhesion and tolerance for non-perfect build plates. Enable to print raft during printing. Raft could help the model stick to the build plate.
- b. **Select Extruder:** is Allows users to select the corresponding extruder. If the machine type goes to "FlashForge Dreamer/FlashForge Creator Pro/ FlashForge Inventor Series/Flashforge Creator 3/Flashforge Creator Max/FlashForge Creator Pro 2/FlashForge Creator Max 2", it's available for users to select extruder, while the machine type goes to "FlashForge Adventurer 3/FlashForge Adventurer 3 Lite/FlashForge Finder/FlashForge Guider/FlashForge Guider II/FlashForge Guider II S Series/FlashForge Inventor II Series/FlashForge Dreamer NX/FlashForge Adventurer 3C", it's unavailable for users to select extruder.
- c. **Margin:** Raft offset from the outline of the model's first layer.
- d. **Space to Model(Z):** Determines the vertical space between model and raft, which makes raft easier to be removed.
- e. **Above Raft Maximum Speed:** Limits the print speed of the first layer above raft to make models stick to raft better.
- f. **Above Raft Extrusion Ratio:** Set the extrusion ratio for the first layer; the higher the extrusion ratio, the better the first layer and more difficult for the raft to be removed.

Bottom layer:

- a. **Speed:** The higher layer height, the wider extrusion, the slower printing speed should be chosen, or extruder will not be able to extrusion in time.
- b. **Layer Height:**
Higher bottom layer height will improve adhesion for non-perfect build plates. When build plate temperature is high, the thicker raft also has better insulation.
- c. **Path Width:** width makes it easier to paste raft to the build plate, but the wider path requires the lower printing speed.
- d. **Fill Density:** The higher filling density, the more stable the raft will paste to build plate, the longer printing time.

Middle Layers:

- a. **Layer Height:** The thickness of middle raft (transition layers).
- b. **Layers:** The layers of middle raft (transition layers).
- c. **Speed:** The lower printing speed, the easier to paste transition layer on bottom layer, the more printing time it will take.

Top layers:

- a. **Layer Height:** The thickness of middle raft (transition layers).
- b. **Cross Angle:** The angle between top solid infill paths.
- c. **Layers:** The layers of top raft, when build plate temperature is higher than 110°C, it is

recommended to print more layers to improve the thermal insulation.

- d. **Speed:** The lower speed, the better surface finish the top raft will have, the longer printing time.

I. Additions:

1.Pre-extrusion

- a. **Enable Pre-extrusion:** Print an extra extrusion path before model printing.
- b. **Margin:** Determines the spacing between pre-extrusion path and the outline of the model's first layer.
- c. **Path Length:** Determines the length of pre-extrusion path.
- d. **Speed:** The printing speed of pre-extrusion.

2.Wall

- a. **Enable Wall:** Creates a wall around the model to scrape oozing from the idle extruder. It's suggested to enable in dual extrusion printing.
- b. **Shell Count:** TNumber of paths superimposed for building wall. More paths will generate a higher strength wall but take more print time.
- c. **Margin:** Determines the spacing between model and wall.
- d. **Speed:** Speed at which the wall is printed.

3.Brim

- a. **Enable brim:** Expand the outline of model's bottom layers to a Brim which helps anchor the edges of the model to the plate to avoid warping.
- b. **Select extruder:** To select left extruder, right or automatch extruder if the machine type is "FlashForge Dreamer/ FlashForge Creator Pro/ FlashForge Inventor Series/Flashforge Creator 3/Flashforge Creator Max/FlashForge Creator Pro 2/FlashForge Creator Max 2"; If the machine type is "FlashForge Adventurer 3/FlashForge Adventurer 3 Lite/FlashForge Finder/FlashForge Guider/FlashForge Guider II/FlashForge Guider II S Series/FlashForge Inventor II Series/FlashForge Dreamer NX/FlashForge Adventurer 3C" there is no need to set extruder, the default extruder is right.
- c. **Margin:** The width of Brim, the wider brim is, the more stable it will be, the longer it will take.
- d. **Brim layers:** The more layer of brim the better anchoring the edges of the model, the harder to remove.
- e. **Speed:** The speed of brim printing.
- f. **Don't generate Brim inside holes:** Brim won't be generated inside holes, to avoid the difficulty of removing.

4.Wiping Tower

- a. **Speed:** Speed at which the Wiping Tower is printed.

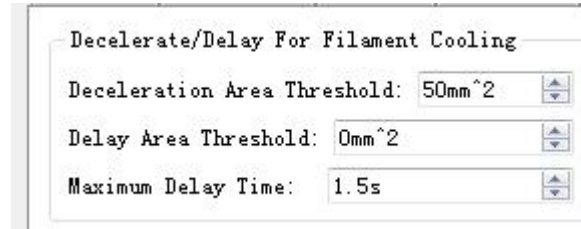
J.Cooling

1.Decelerate/Delay For Filament Cooling

- a. **Deceleration Area Threshold:** When the print area of one layer is less than the area threshold, printer will slow down to ensure enough time for filament cooling.
- b. **Delay Area Threshold:** When the print area of one layer is less than the area

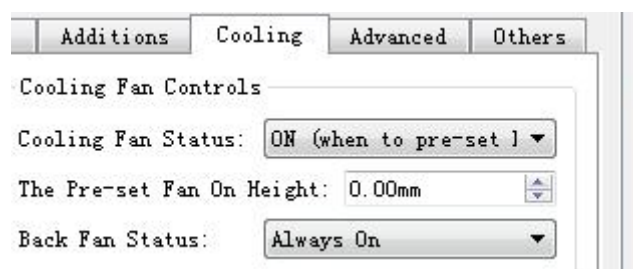
threshold, printer will delay printing to ensure enough time for filament cooling. Printer doesn't delay printing for filament cooling if the Delay Area Threshold is 0.

- c. **Maximum Delay Time:** The maximum delay time for filament cooling.



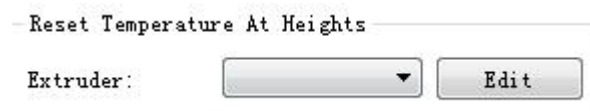
2. Cooling Fan Controls

- a. **Cooling Fan Controls:** Set up the time to turn on the cooling fan. Fan Status: Including “Always On”, “Always Off”, “ON(when raft bottom printed)”, “ON(when to pre-set height)”, “ON(while printing raft top)” which only apply to “FlashForge Adventurer 3/FlashForge Adventurer 3 Lite/FlashForge Creator Pro/FlashForge Dreamer/FlashForge Guider II/FlashForge Guider II S Series/FlashForge Inventor Series/Flashforge Creator 3/Flashforge Creator Max/FlashForge Adventurer 3C/FlashForge Creator Pro 2/FlashForge Creator Max 2” at present.
- b. **The Pre-set Fan On Height:** Turn on the cooling fan when to specific height.
- c. **Back Fan Status:** Set the status of cooling fan on the printer backboard. Including “Always On” and “Always Off”. Only apply to FlashForge Adventurer 3/FlashForge Adventurer 3 Lite/FlashForge Adventurer 3C and FlashForge Guider II S Series at present.



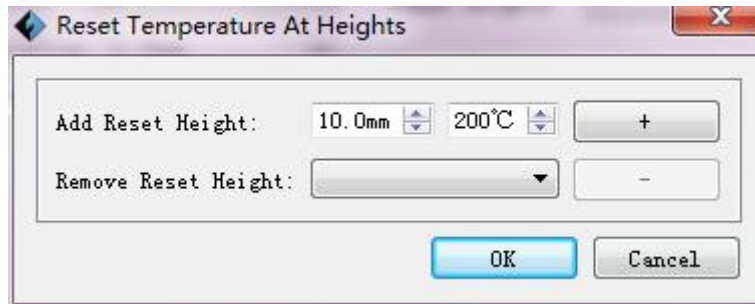
3. Rest temperature at heights

- a. **Extruder:** Reset the temperature of extruder once reach to specific height. Edit the height value which needed to reset temperature, please don't over the model height. (When printing dual model file, please set left and right extruders) As shown below:



Step: click “edit” button, enter into edit page (as shown below), edit reset height and

temperature value, click “+” button to start setting,the added value will showed in “delete reset height” list.



b.Platform: Reset the temperature of platform once reach to specific height. Edit the height value which needed to reset temperature, please don't over the model height.Only used for“FlashForge Dreamer/FlashForge Creator Pro/FlashForge Guider II/FlashForge Guider II S Series/FlashForge Inventor Series/FlashForge Dreamer NX/FlashForge Creator 3/FlashForge Creator Max/FlashForge Creator Pro 2/FlashForge Creator Max 2” at present,As shown below:



Step: click “edit” button,enter into edit page(as shown below), edit reset height and temperature value, click “+” button to start setting,the added value will showed in “delete reset height” list.

K.Advanced

1.Stepper Motor Voltage(Usually keep default)

- a. **X-Axis:** Voltage parameter of X-axis stepper motor. The bigger the value is, the more powerful X-axis stepper motor is, but the more heater will produce.
- b. **Y-Axis:** Voltage parameter of Y-axis stepper motor. The bigger the value is, the more powerful Y-axis stepper motor is, but the more heater will produce.
- c. **Z-Axis:** Voltage parameter of Z-axis stepper motor. The bigger the value is, the more powerful Z-axis stepper motor is, but the more heater will produce.
- d. **A-Axis:** Voltage parameter of A-axis stepper motor. The bigger the value is, the more powerful A-axis stepper motor is, but the more heater will produce.
- e. **B-Axis:** Voltage parameter of B-axis stepper motor. The bigger the value is, the more powerful B-axis stepper motor is, but the more heater will produce.

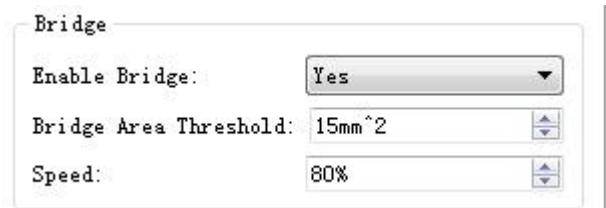
2.Others

- a. **Extrusion Ratio:** Extrusion multiplier for all extrusion movements.
- b. **Path Width:** The width of extrusion path. The default value is 0.4mm which equals the diameter of nozzle
- c. **Path Resolution:** Determines the extrusion path resolution when model is over-detailed. The bigger the value is, the lower path coordinate resolution is. However, the extrusion precision will be higher.

- d. **First Layer Extrusion Ratio:** Extrusion multiplier for the first layer extrusion movement.

3. Bridge

- a. **Enable Bridge:** Optimized effect of tool path and shorten the span of tool path to improve the effect of floating printing.
- b. **Bridge Area Threshold:** When the floating area is less than the specified threshold area, bridge function is not available.
- c. **Speed:** The ratio of the bridge print speed to the reference speed.



Bridge

Enable Bridge: Yes

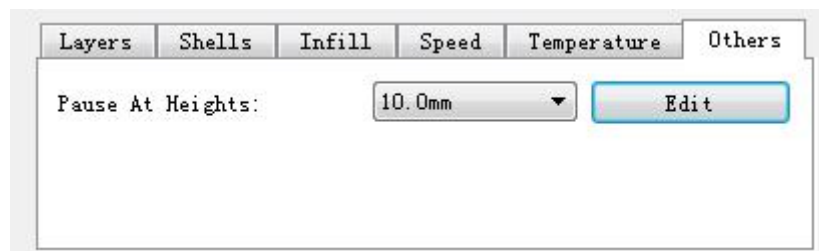
Bridge Area Threshold: 15mm²

Speed: 80%

L. Others

1. Pause At Heights

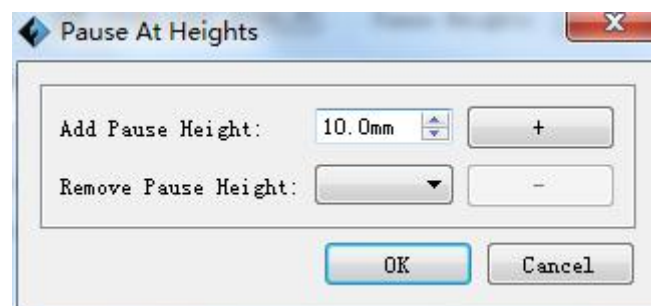
- a. **Pause At Heights:** Pause printing automatically when to specific heights. Users can add one or more specific height values (no higher than the model height). It displays as follows



Layers Shells Infill Speed Temperature Others

Pause At Heights: 10.0mm Edit

Steps: Click the [Edit] button, and a box will pop up (See graphic below). Then add the value, click [+] for confirmation, and the value will show in the list of [Delete Value]. If deletion is required, select the specific value on the list and click [-] for confirmation. Click [OK] to finish and exit these steps.



Pause At Heights

Add Pause Height: 10.0mm +

Remove Pause Height: -

OK Cancel

- b. **Edit Pause At Heights :** Add or remove the Pause Heights.

2. Dimensional Adjustment

- a. **Enable Adjustments:** Adjust extrusion paths to compensate printing error.

- b. **External Compensation:** Adjust the model's outer diameter, set positive to enlarge dimension and set negative to shrink dimension.
- c. **Internal Compensation:** Adjust the model's inner diameter, set positive to enlarge dimension and set negative to shrink dimension.
- d. **Z Compensation:** Adjust the model's print height, set positive to enlarge height and set negative to reduce height.

Dimensional Adjustments	
Enable Adjustments:	No
External Compensation:	0.00mm
Internal Compensation:	0.10mm
Z Compensation:	0.00%

3.Sequential Printing

- a. **Enable Sequential Printing:** Print one model completely and then print the next one. It can reduce the stringing caused by the nozzle moving back and forth between models.
- b. **Right Nozzle Margin X:** Maximum distance from the nozzle center to the extruder border. In order to avoid the collision between the extruder and models, model spacing must be greater than this distance. If raft or brim is enabled, the model spacing needs to be additionally increased by the maximum width of raft or brim.
- c. **Right Nozzle Margin Y:** Maximum distance from the nozzle center to the extruder border. In order to avoid the collision between the extruder and models, model spacing must be greater than this distance. If raft or brim is enabled, the model spacing needs to be additionally increased by the maximum width of raft or brim.
- d. **Left Nozzle Margin X:** Maximum distance from the nozzle center to the extruder border. In order to avoid the collision between the extruder and models, model spacing must be greater than this distance. If raft or brim is enabled, the model spacing needs to be additionally increased by the maximum width of raft or brim.
- e. **Left Nozzle Margin Y:** Maximum distance from the nozzle center to the extruder border. In order to avoid the collision between the extruder and models, model spacing must be greater than this distance. If raft or brim is enabled, the model spacing needs to be additionally increased by the maximum width of raft or brim.
- f. **Gantry Height:** The minimum distance from the bottom of the nozzle to the Y-axis screw. In order to avoid the collision between the gantry and models, only one model is allowed to exceed this height.

Sequential Printing

Enable Sequential Printing:

Right Nozzle Margin X:

Right Nozzle Margin Y:

Left Nozzle Margin X:

Left Nozzle Margin Y:

Gantry Height:

4. Z Hop

- a. **Z Hop Mode:** Lift the extruder or lower the platform when traveling to avoid the extruder from bumping into the model.
- b. **Z Hop Height:** The height at which the extruder is lifted or the platform is lowered when the extruder is traveling.
- c. **Z Hop Minimum distance:** When the traveling distance is less than the specified value, the Z Hop is not performed.

Z Hop

Z Hop Mode:

Z Hop Height:

Z Hop Minimum Distance:

M . Save as new: Allows users to save the model as a new file after parameter modification.

How to do?

After getting all the required parameters modified, click [Save as new], then a dialogue box will pop up. Users need to enter the file name into the box, then click [OK]. Click the drop-down menu of [Select Profile], the new added scheme can be found in the list.

N. Remove

Allows the users to delete the added scheme(s). Select one of the added schemes, click [Remove] and a dialogue box will pop up for confirmation. Click [Yes] to remove it or click [No] to cancel the current operation.

O. Restore Default: Allows users to restore to the default settings.

P. Save Configuration: Allows users to save the present configuration.

R.port/Export Profile: After modifying the required parameters according to the user's needs, save the current slice parameter scheme as a local file for subsequent use.

Action: When the profile needs to be exported, click the 'Export Profile' button to save the current slice parameter scheme as a local "fprofile" format file; Click the 'Import Profile' button when you need to import a profile: Import the .fprofile-formatted slice parameter profile into the profile table, and there is an additional option in the 'Select Profile' list (the new scenario is related to the mode selected when the profile is exported, only added to the 'Select Profile' list for the corresponding model)

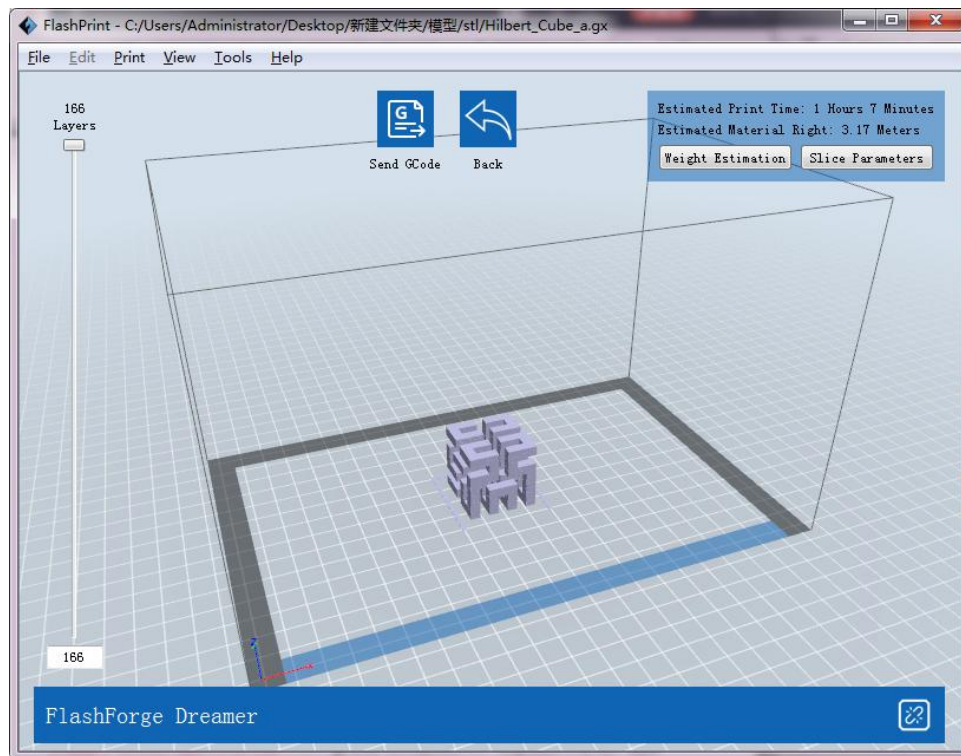
3.8.3.4 FDM 3D Printer Print Gcode File

a. Slice

After finishing slicing, it will load into the generated Gcode file automatically if you select **Preview** option when slicing, or need you manually load into it and then go to preview interface. In the preview interface, there is a vertical scroll bar which shows each layer of the model and you can find **Estimated Material Right** 、 **Slice Parameters** 、 **Estimated Print Time** and **Weight Estimation** it costs on the top right corner. Then click **send Gcode** button on the same location, the machine starts to print while click **Back** button to exit from Preview mode.

b. Weight Estimation

- ◆ **Material length:** The material length required to print the model.
- ◆ **Material Type :** FlashPrint 3.19.0 support material weight estimation. When you select "Custom Material", you can manually change the "Material Density".
- ◆ **Material Density:** The material density, which is corresponding to material type.
- ◆ **Weight Estimation:** Filament weight is estimated from the length and density of the filament.



3.8.3.5 Pause or Terminate Printing

Once printing starts, click the machine icon on the bottom right corner of the software interface and it will show the process status in a frame. **Pause** and **Stop** button displayed at the bottom of the current model's status bar. Click **Pause** button to pause printing and then click **Continue** button to resume it. Click **Stop** button to cancel this printing task and you need to restart this print task if you want to print it again.

Note: Please **DO NOT** click **Pause** button unless necessary, for it will affect printing result.

3.9 Printer Operations

3.9.1 Connect / Disconnect

The computer can be connected to multiple printers at the same time in two ways: 1. Auto connection; 2. Wi-Fi/ Ethernet connection. The machine icon on the bottom right corner of the software interface shows a broken chain pattern means disconnected, while shows a unbroken one means connected, and tap icon to hide/show the status bar of all printer connections.

3.9.1.1 Connect

A. Auto Connection

- a. Find USB ports on the right-side of machine and computer, plug in and connect.
- b. Open Flashprint software and turn on the printer.
- c. Click **Print -> Connect Machine** menu, then select **Automatic** in the **Connection Mode** option and select machine you want to connect in **Select Machine** option. If you can not find your machine, click **Rescan** button to scan your machine and select it. Finally click **Connect** button to connect to the printer. If you still can not find your machine after rescan, which means you haven't installed the driver in the software. You have to install the driver manually, please refer to the Chapter 2.8.3.1. Normally, driver shall be installed automatically along with software.
The "Automatic" connection mode scans and displays all available machines, including Wi-Fi connections and USB cable connections.

B. WIFI connection

- a. Via wireless network
 1. Switch on the printer, and make the WIFI available by clicking the menu **Tools -> Set up -> WIFI -> WIFI ON**.
 2. Turn on the WIFI and then connect the computer to the network whose name is the same as SSID. (It is the printer's default network without password if no settings have been made.)
 3. Open your browser and enter the IP address **10.10.100.254** in the Address Bar. Then enter in **admin** as the account and password . (The above-mentioned IP address /Account /Password are default when the setting haven't been changed)



4. Enter the WIFI setting panel, select “AP+STA mode” for Dreamer, select “AP” or “STA” mode for Finder, then click [OK]. Note: Don’t click [Restart].
(Note: If users only choose STA mode, it has the potential risk that Dreamer will fails WIFI connection forever once STA settings crash.)



5. Click STA Setting, and then click Scan button and choose a commonly used network. Type in password, save it, and then click Reboot to the printer.

闪铸科技
FLASHFORGE

中文 | English | 日本語

System Info
Work Mode
STA Setting
AP Setting
COM Socket
Account
Firmware
Module admin

Network Name (SSID)
Note: case sensitive
猎豹免费WiFi250
Scan

Encryption Method
WPA2PSK

Encryption Algorithm
AES

Password
.....
Show passwords

Obtain an IP address automatically
Enable

IP Address
192.168.191.18

Subnet Mask
255.255.255.0

Gateway Address
192.168.191.1

DNS Server Address
192.168.1.1

Save

6. 5 seconds after the reboot, choose disconnect and try to connect WIFI again. Open FlashPrint, click the menu Print->Connect to select WIFI connection, and then enter the network IP address (same as the IP showed on the touchscreen) in the IP port option. Finally click connect to complete it. After this, you do not need to connect WIFI in the future if no WIFI setting is changed. If you want to change WIFI setting, please try the same way.

b. Via wireless source set by the printer

1. Switch on the printer, and make the WIFI available by clicking the menu **Tools ->Set up->WIFI->WIFI ON**.
2. Turn on the WIFI and then connect the computer to the network whose name is the same as SSID.(It is the printer's default network without password if no settings have been made.)
3. menu **Print -> Connect**, and select Wi-Fi in the **Connection Mode** bar. Type in 10.10.100.254:8899 in the **IP Address, Port** bar, then click **Connect** button to complete it.

Note: One printer could only have one connection. If printer is operating by another process, connections should be switched off firstly and try to connect again. If AP mode is chosen, the computer will be unable to go online by WIFI.

3.9.1.2 Disconnect from the printer

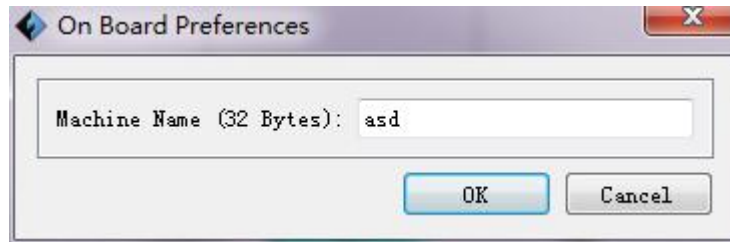
When one or more printers are connected, select any printer and tap the "disconnect"button in the status bar of this printer to disconnect the computer from this printer.Tap the menu bar "Print-> Disconnect All Connections" to disconnect the computer fromall printers.

3.9.1.3 Set As Current

When the computer is connected to multiple printers at the same time,tap the "set as current" button in the printer status bar to Switch the operation interface to the corresponding interface of the printer you chose.

3.9.1.4 Modify Printer Name

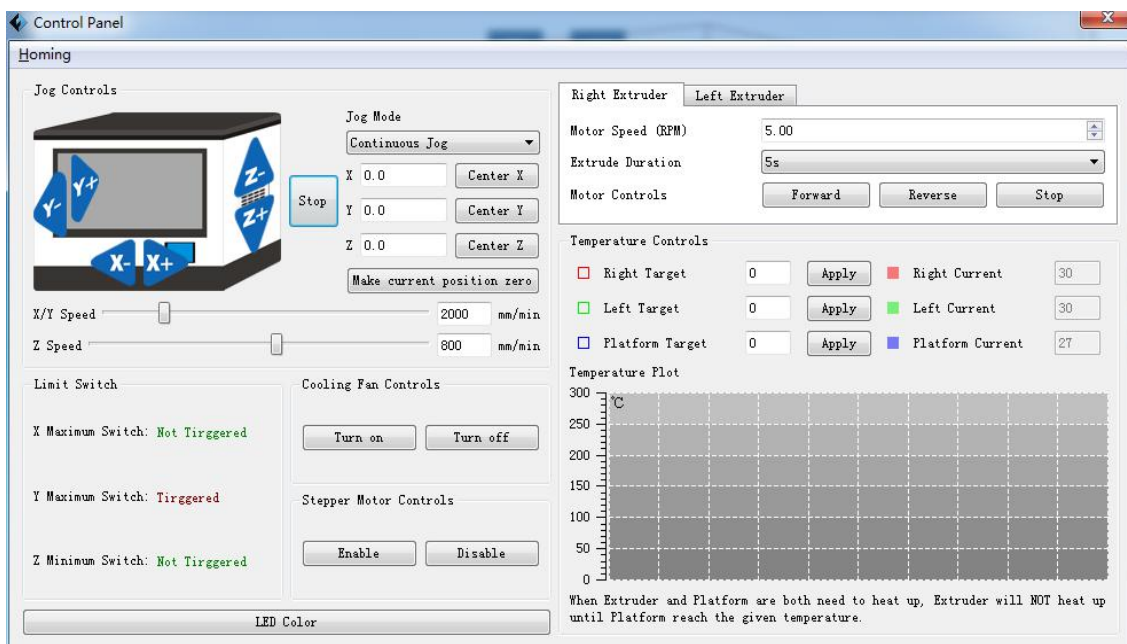
In the status bar of connected printer, tap the button to the right of printer name, the dialog box would pop up for modifying the name. Tap "OK" after modifying the name, and the printer name is successfully modified; Tap cancel if you don't modify the printer name.



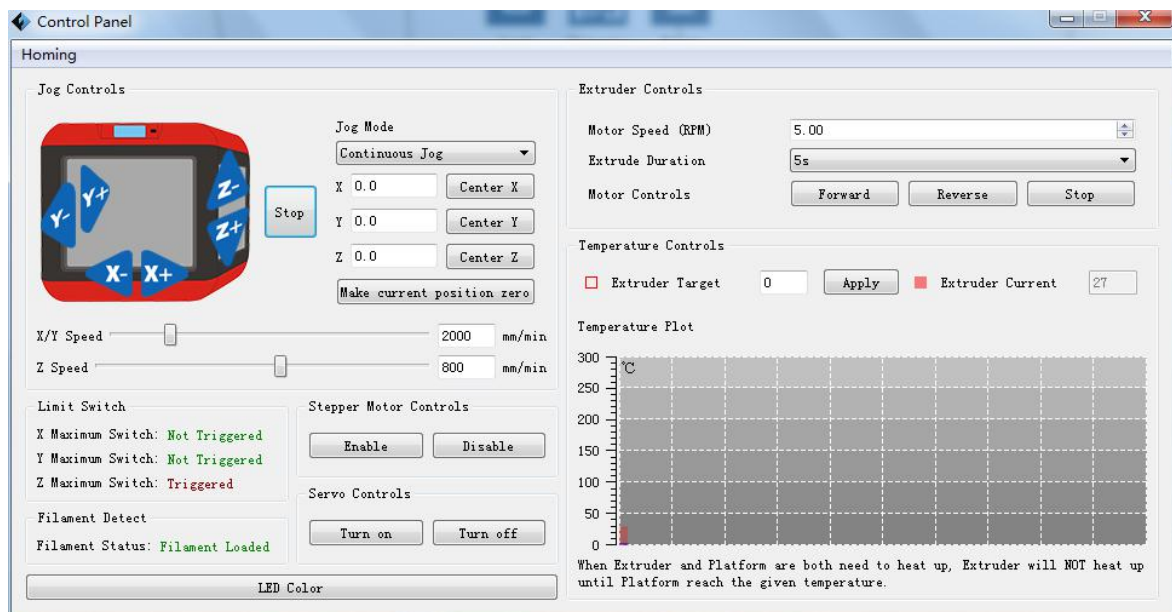
3.9.2 Control Panel

After connection between software and 3D printer is built, click **Tools->Control Panel** menu to open the control pane. Following operations are available in the control panel to control the printer.

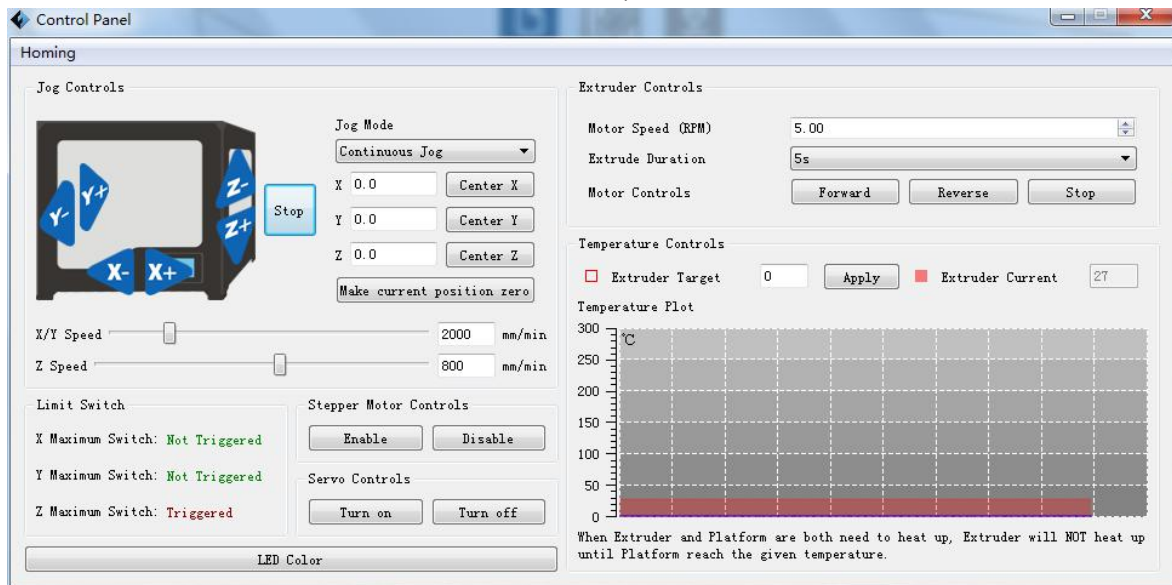
The control panels of different machine types are different. The control panel of 'FlashForge Adventurer 3/FlashForge Adventurer 3 Lite/FlashForge Dreamer/FlashForge Creator Pro/FlashForge Inventor Series/FlashForge Inventor II Series/FlashForge Dreamer NX/Flashforge Creator Max/FlashForge Adventurer 3C/FlashForge Creator Pro 2/FlashForge Creator Max 2' shows in the 'control panel 1', while that of 'Flashforge Finder' shows in the 'control panel 2', while that of 'FlashForge Guider/FlashForge Guider II/FlashForge Guider II S Series' shows in the 'control panel 3'. While that of 'Flashforge Creator3' shows in the 'control panel 4'. (this function is only available to FDM 3D printer)



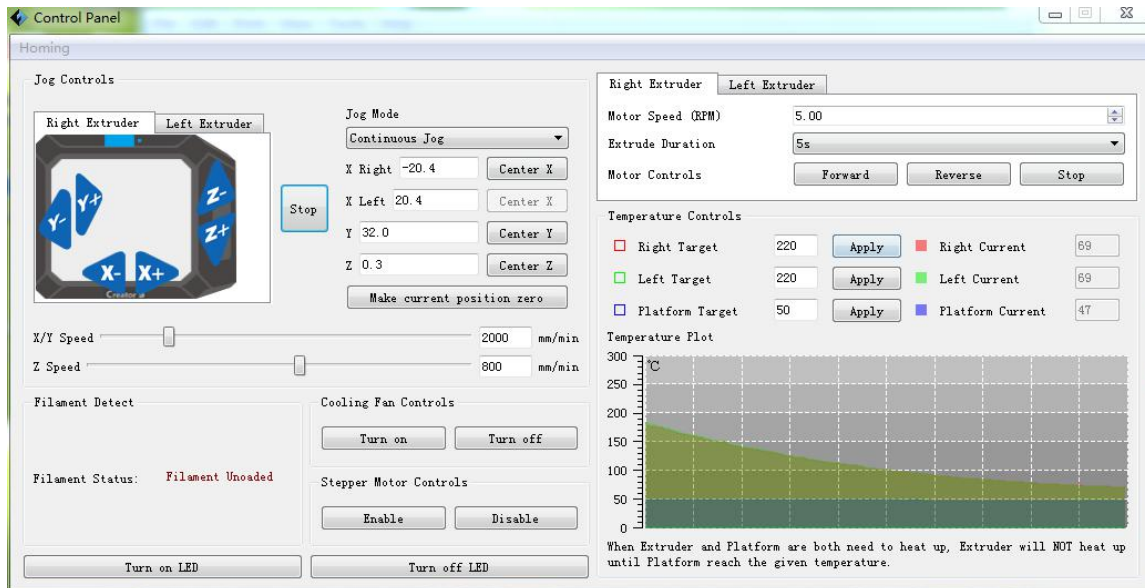
(Control panel 1)



(Control panel 2)



(Control panel 3)



(Control panel 4)

A. Jog Controls

a. Jog Mode:

You could set the distance the extruder or the platform moves by a single jog click time.

b. Direction Buttons:

These six blue-arrow buttons are set to control the movements on X/Y/Z directions. Click the direction buttons of x-axes and y-axes to control the horizontal movements of the extruder. Click the direction buttons of z-axes to control the vertical movements of the platform. Click X- button to make the extruder move to the left; click X+ button to make the extruder move to the right. Click Y- button to make the extruder move forward; click Y+ button to make the extruder move backward. Click Z- button to make the platform move upward; click Z+ button to make the platform move downward. The distance of movement will be showed in the X/Y/Z value bar.

c. Stop Button:

Click this button to terminate current movement.

d. Coordinate frame:

It displays the coordinated positions of extruder and platform.

e. Make current position zero button:

Click this button to set the current position of the extruder and platform as the origin of coordinates.

f. Center X/Y/Z button:

Click these buttons to return the extruder or platform to the origin of coordinates automatically.

g. X/Y/Z Speed sliding bar:

Set the moving speed of the extruder and platform by sliding the speed sliding bar.

B. Limit Switch

In order to protect the 3D printer, there are three limit switches on X/Y/Z axes used to control the movement limited position. they have the following two statuses:

a. Open Status:

If the extruder or platform has not reached to the limit position of X/Y/Z direction, the switch won't be triggered and its status is **Open**.

b. Triggered Status:

If the extruder or the build plate moves to the maximum position, the maximum or the minimum limit switch of X/Y/Z axis will be triggered and shows "Triggered" according different machine types.

C. Fan Controls

Click **Open** button to turn on the cooling the fan, while **Close** button to turn it off.

D. Stepper Motor Control

It is used to control the stepper motor. Click **Open** button to lock the stepper motor and users can not adjust the position of extruder and platform manually, while **Close** button to unlock it and users can do manual adjustment on extruder and platform.

E. LED Color

The printer has a built-in LED, Click **LED Color** button to change its color built in the printer.

F. Extruder Setting

In the interface of extruder setting, you could feed in or withdraw the filament from the left/right extruder. By changing the value of "Motor Speed (RPM)", you could control the speed of feeding in and withdrawing filament. In addition, you could control the time of these actions through changing the value of "Extrude duration". Generally, we suggest you set the extrude duration as **60s**.

Before feeding in or withdrawing the filament, the extruder should reach target temperature to melt the filament before you click **Forward / Reverse** button to start these operations. For ABS filament, target temperature of extruder is 220°C; for PLA filament, target temperature of extruder is 200°C. for dissolvable Filament filament, target temperature of extruder is 240°C. Click **Stop** button to stop feeding in or withdrawing filaments.

G. Temperature Control

Type in the target temperature in the left blank and click **Apply** button to heat the extruder or platform. The printer starts to heat the extruder or the platform, and you will see the actual temperature the extruder or platform has reached on the right and a temperature changing table with 3 different color curve line below.

Note: The printer will heat platform firstly to reach target temperature and then heat extruders when both are needed to heat.

H. Filament Detection

To detect whether the filament is loaded in Finder.

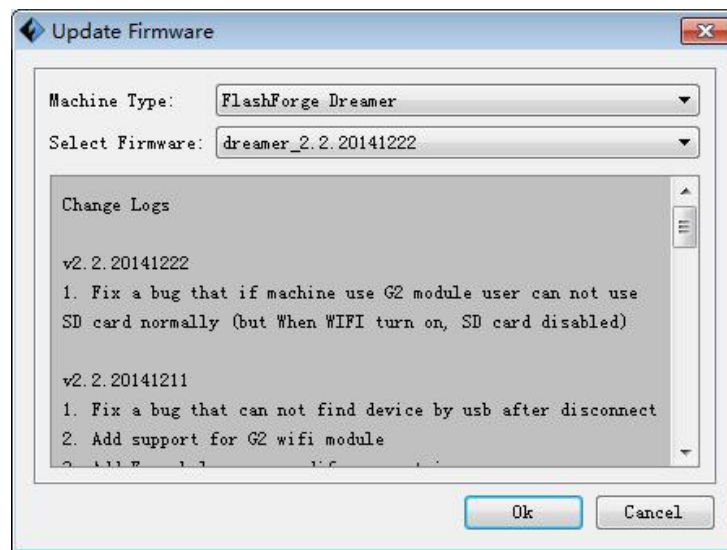
L. Servo Control

To control the servo's on-off.

3.9.3 Firmware Upgrade

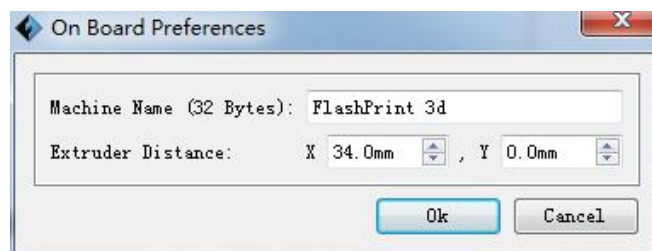
Only Dreamer/Dreamer NX/Inventor/Creator Max/Creator Pro 2/Creator Max 2 support the function of firmware updating. Finder doesn't display 'firmware updating' by default, and 'firmware updating' displays when using the old Finder. The software will check and download updates automatically when users open it, and remind users to upgrade to new version if available. Ways to install firmware as follows, Finder 3D printer must connect computer with USB cable before update firmware (this function is only available to FDM 3D printer):

- A. Disconnect the computer and printer, or Click **Tools -> Update Firmware** menu and select **Disconnect** option,
- B. Select your machine type and corresponding firmware in the popped up dialog and then click **OK** to upgrade. If the machine is idle, it will start to upgrade to new firmware.



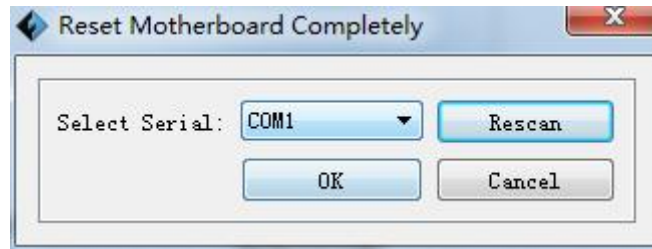
3.9.4 On Board Preferences

When the printer has been connected, click **Tools-> On Board Preferences** menu to review the configuration of mother board including Machine Name, and Extruder count etc. When choosing extruder count as 2, Extruders Distance option is available. Users can modify the X/Y distance between the extruders as need. The X/Y distance refers to the distance of X or Y directions between the two extruders. If it is 'FlashForge Adventurer 3/FlashForge Adventurer 3 Lite/FlashForge Finder/FlashForge Guider/FlashForge Guider II/FlashForge Guider II S Series/FlashForge Inventor II Series/FlashForge Dreamer NX/FlashForge Adventurer 3C', the content only includes the name of the machine. (this function is only available to FDM 3D printer)



3.9.5 Reset Motherboard Completely

At present, only Creator Pro contains this function. Use USB cable to connect the printer, and click ok to reset the motherboard parameter.



3.9.6 Machine Information

When the printer has been connected, click **Tools->Machine Information** menu to review the information about the machine, including MACHINE TYPE, MACHINE NAME, FIRMWARE VERSION, etc.
(this function is only available to FDM 3D printer)

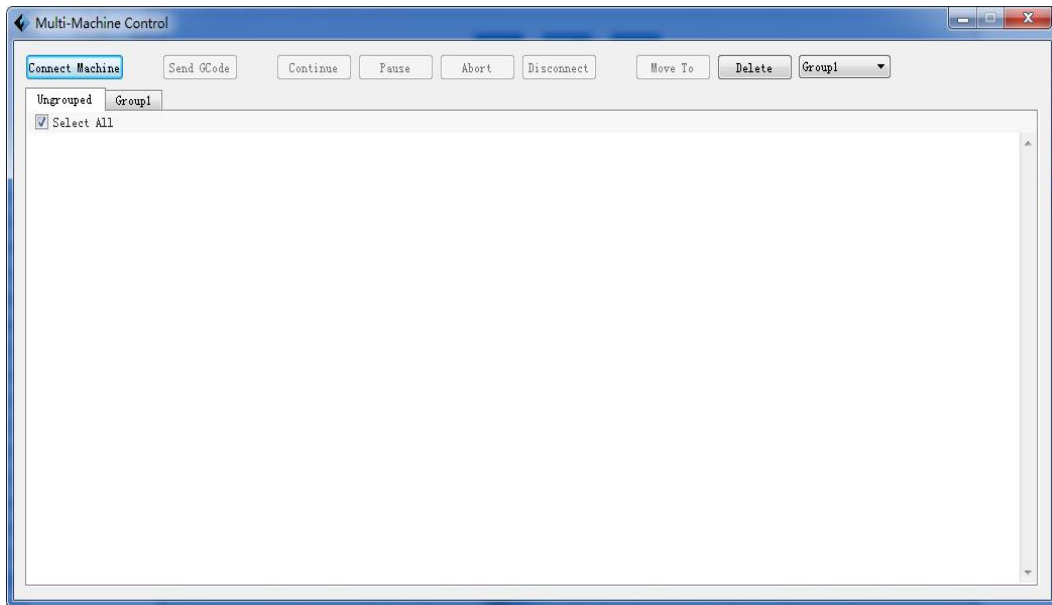
3.9.7 Driver Installation

- A. Open the root directory of the software. (e.g. C:\Program Files\flashforge\FlashPrint)
- B. Open the driver folder which in the root directory and find the driver. There are two driver installation packages, please install one of them according to your computer system. For 64bit edition, please install dpinst_amd64.exe file; while for 32bit edition, please install dpinst_x86.exe file.

(this function is only available to FDM 3D printer)

3.9.8 Multi-Machine Control

Click the <Multi-Machine Control> under the <Tool> to enter the interface:

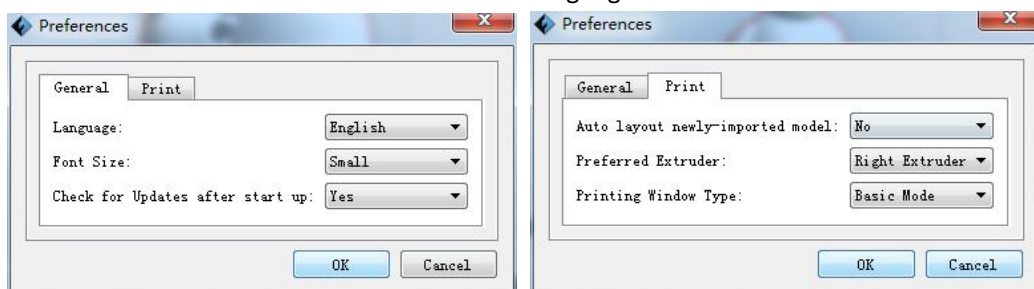


- ◆ Connect the machine: same as the <Print→Connect Machine> function, the connected machine will display in the group.
- ◆ Send the GCode: send gcode to the same types of this machine, at most five machines at one time with the rest queuing for sending gcode; the other types of machine are not allowed to send gcode.
- ◆ Continue: send the <continue printing> instruction to all selected printers in suspend
- ◆ Pause: send the <pause printing> instruction to all selected printers in printing
- ◆ Suspend: send the <suspend printing> instruction to all selected printers in printing
- ◆ Disconnect: disconnect all selected printers
- ◆ Remove: remove all selected printers to the designated group or a new group
- ◆ Delete: delete this group, the printers in this group will be automatically moved to **【ungrouped】**
- ◆ Select All: select all printers in the group

3.10 Others

3.10.1 Preferences

Click **File-> Preferences** menu to select the interface language and etc.



1. General

A. Language

So far this software supports English, Japanese, Simplified Chinese ,Traditional Chinese,French, German,Arabic,Spanish,Czech,Turkish,Portuguese,Polish,Korean,Russian.

B. Check for Update after Startup

Select **Yes** option to command the software auto-check updates after starting up it. The software will remind you to update to new version when it is available after checking.

C.Font Size

Use to setting the font size of the software.Large,Middle and Small front mode to choose. If 'font size' chose 'large', the software font is large font mode.If you choose middle mode, the front of software is middle.otherwise is small font mode. The default font size is small font size.

2. Print

D.Auto-layout newly-imported model

To setup the model's position and posture. If you select [Yes], the model will be on the platform and auto-centered when you load the model. If you select [No], the model will maintain its initial position and posture when you load the model.

E. Expert Mode

Expert Mode grants the users more freedom of parameter edition. There are two modes are available for users, one is “Basic Mode” and the other is “Expert Mode”.

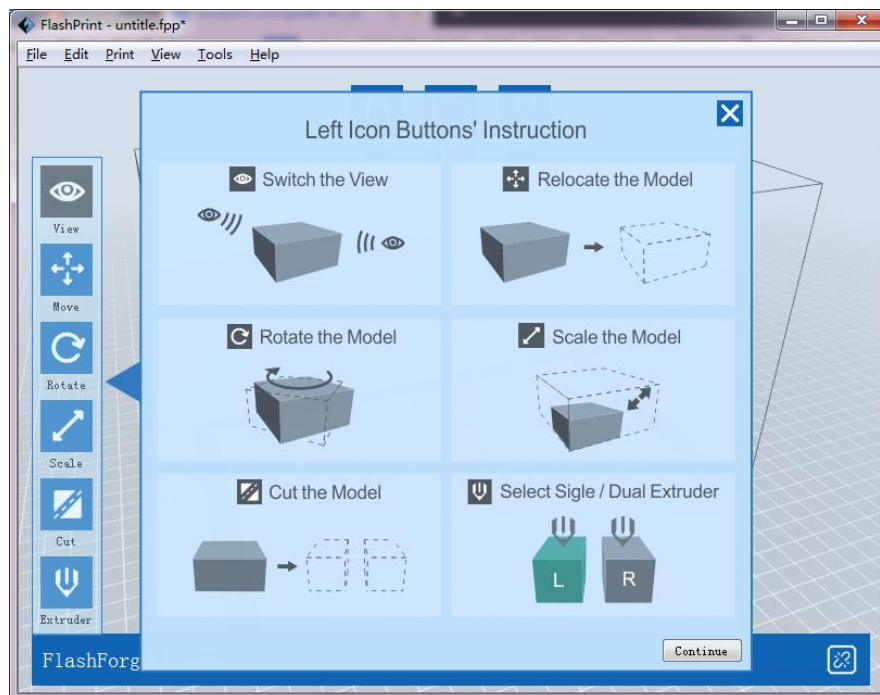
F. Performance extruder

It is the setting to save the choice of performance extruder, which is only for Flashforge Creator 3,Flashforge Creator Max,Flashforge Creator Pro ,FlashForge Creator Pro 2,FlashForge Creator Max 2 and Flashforge Dreamer. According to 3D printer condition and usage habit, user can set the performance extrude. Extruder choosing will not needed before silcing.(**this function is only available to FDM 3D printer**)

3.10.2 First run wizard

An explanatory menu pops up the first time you open Flashprint. Click Continue to learn more about the features, or close the menu if you're already familiar with the features.

You can go to Help -> First time user guide to re-open the menu when you need to learn the features Later.



3.10.3 Help Contents

Click **Help-> Help Contents** menu to be directed to online user manual.

3.10.4 Feedback

Click **Help->Feedback**, users can feedback suggestions and problems.

3.10.5 Check for Updates

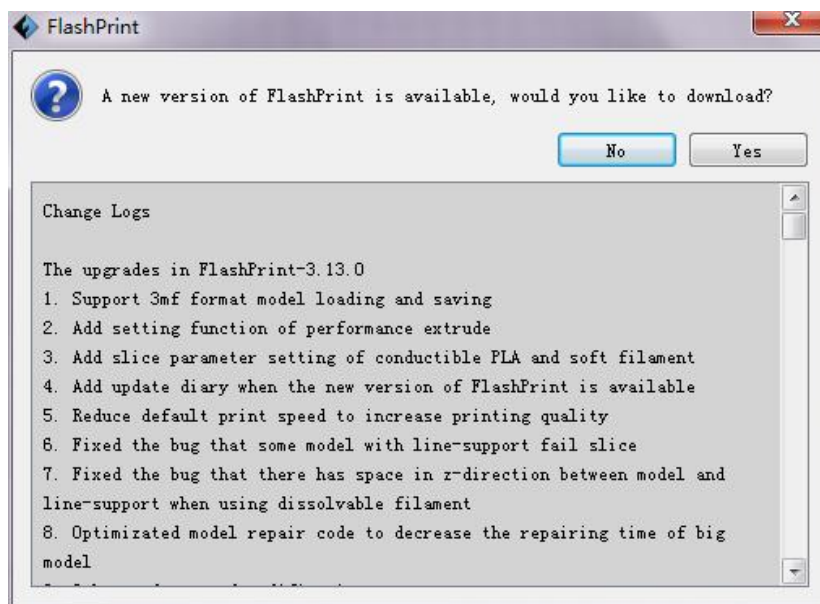
A. Automatic Update

Refer to 2.9.1 to set auto update. Click button of skip this version, you can skip the update.

B. Manual Update

Click **Help-> Check for Updates** menu to check new version of software online. If there is an

updated version of the software to be detected, it will pop up a new version of update dairy, according to their own needs users can download and install the updated version or close the prompt .



1.1~1.2 to install the software.

3.10.6 Software Information

Click **Help-> About FlashPrint** menu to review information about the software, including software version and copyrights, update log etc.

