

Contents

1. Software Installation.....	1
1.1 FDM 3D Printer Type.....	1
1.2 DLP 3D Printer.....	1
1.1 Download.....	1
1.2 Installation & Start.....	1
2. Software Operating Instructions.....	2
2.1 Load File.....	3
2.1.1 Generate Relievo.....	3
2.2 Mouse Operations.....	7
2.2.1 Left click.....	7
2.2.2 Left click and hold the left button.....	7
2.2.3 Right click and hold the right button.....	7
2.2.4 Scroll the mouse wheel.....	7
2.3 Change Views.....	7
2.3.1 Move.....	7
2.3.2 Rotate.....	8
2.3.3 Scale.....	8
2.3.4 Set.....	8
2.3.5 Reset.....	8
2.3.6 Show Model Outline.....	8
2.3.7 Show Steep Overhang.....	8
2.4 Edit Models.....	8
2.4.1 Move.....	8
2.4.2 Rotate.....	9
2.4.3 Scale.....	9
2.4.4 Set.....	9
2.4.5 Cutting the Model:.....	10
2.4.6 Others.....	11
2.5 Edit Supports.....	13
2.5.1 Auto Supports.....	13
2.5.2 Clear Supports.....	13
2.5.3 Add.....	14
2.5.4 Delete.....	14
2.5.5 Support option.....	14
2.6 New Project.....	15
2.7 Saving File.....	15
2.8 Printing Process.....	16
2.8.1 Choose Machine Type.....	16
2.8.2 Connect Machine.....	16
2.8.3 Print.....	16
2.9 Printer Operations.....	30
2.9.1 Connect / Disconnect.....	30

2.9.2 Control Panel.....	33
2.9.3 Firmware Upgrade.....	36
2.9.4 On Board Preferences.....	36
2.9.5 Machine Information.....	37
2.9.6 Driver Installation.....	37
2.10 Others.....	37
2.10.1 Preferences.....	37
2.10.2 Help Contents.....	38
2.10.3 Check for Updates.....	38
2.10.4 First time user guide.....	38
2.10.5 Software Information.....	39

USER GUIDER

1. Software Installation

1.1 FDM 3D Printer Type

There are five type printers:

FlashForge Dreamer/FlashForge Creator Pro/FlashForge Inventor/FlashForge Finder/FlashForge Guider

1.2 DLP 3D Printer

FlashForge Hunter

1.1 Download

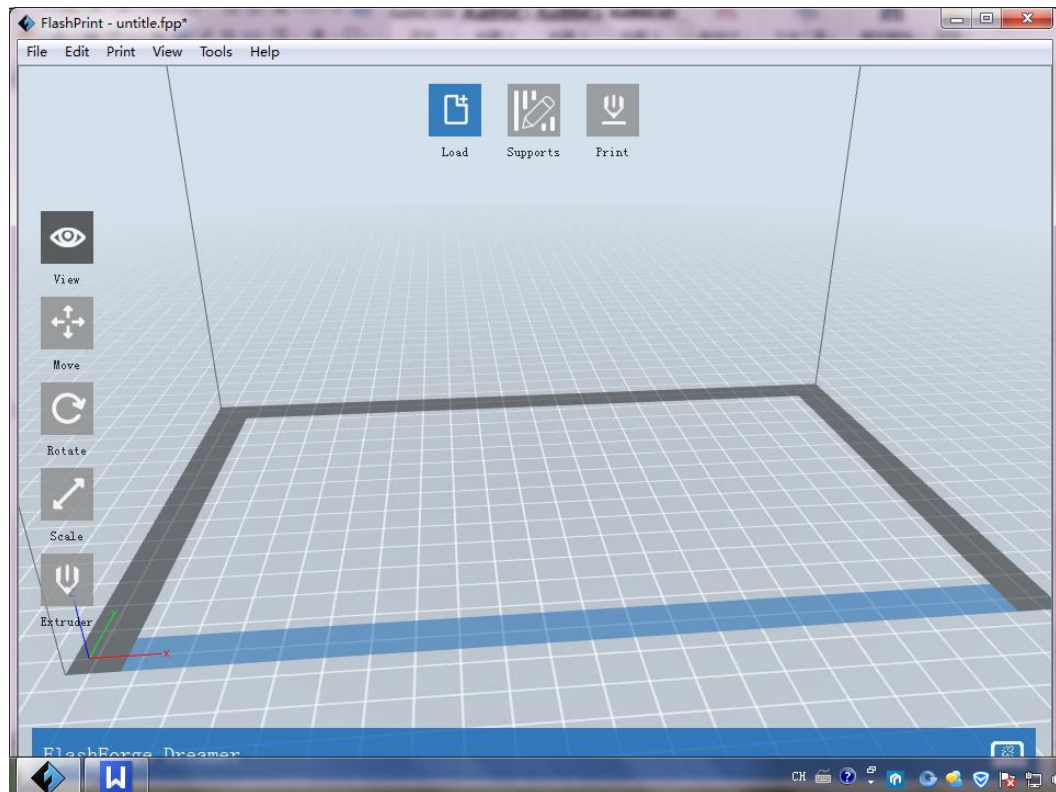
- A. Insert SD card sent with Dreamer into a computer, we have already prepare the latest driver installation package in the SD card.
- B. Open below link to download the installation package:
<http://www.ff3dp.com/#!sup/c1df1>

1.2 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.

2. Software Operating Instructions



Load into one or multiple files.



Enter supporting editing mode



View FlashPrint home screen from one of six viewing angles.



Move model around on xy-plane; shift+click to move along z axis



Turn and rotate your model



Scale the size of your model



Select right or left extruder you want to print with



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



To cut the model from different directions.

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

2.1 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 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.

2.1.1 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 and bottom diameter.

Shape: including plane, tube, 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.

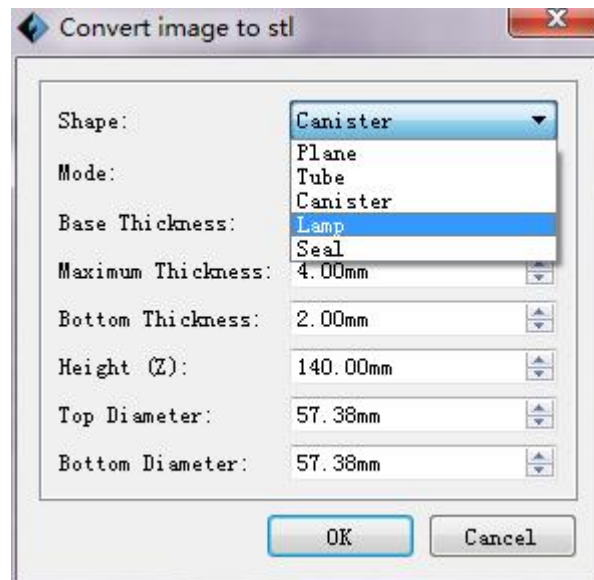
Bottom thickness: For tube, canister and lamp to set up bottom thickness.

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.

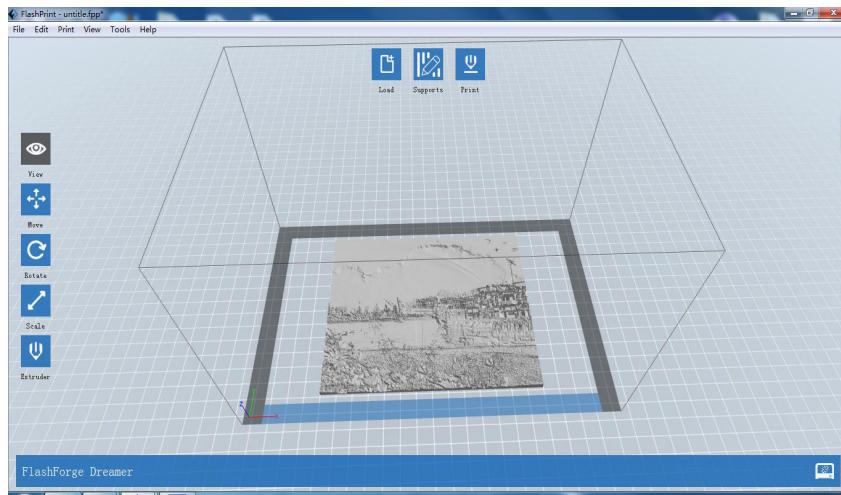
Graphic pattern height: The maximum height of the stamped graphic pattern.

The effect pictures for different shapes:

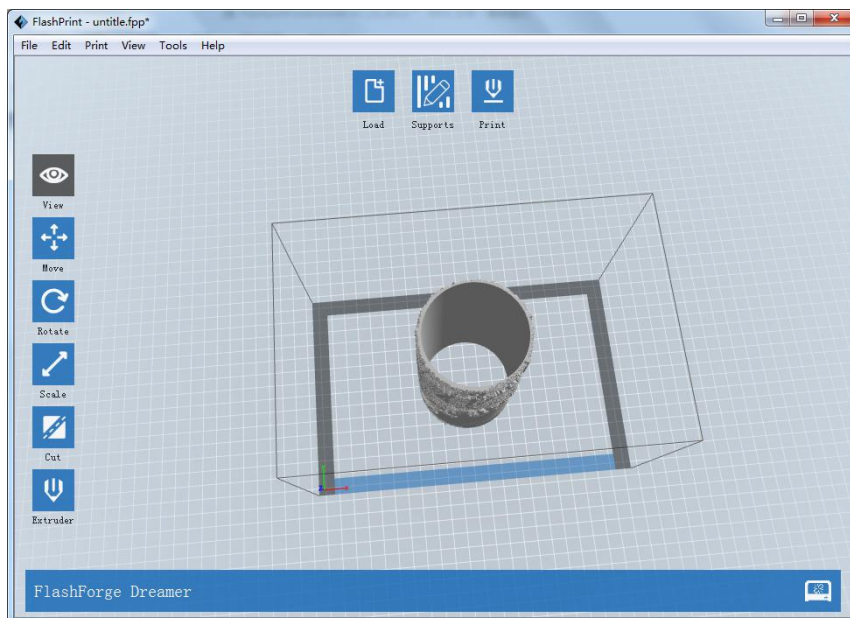


The loaded original png picture:

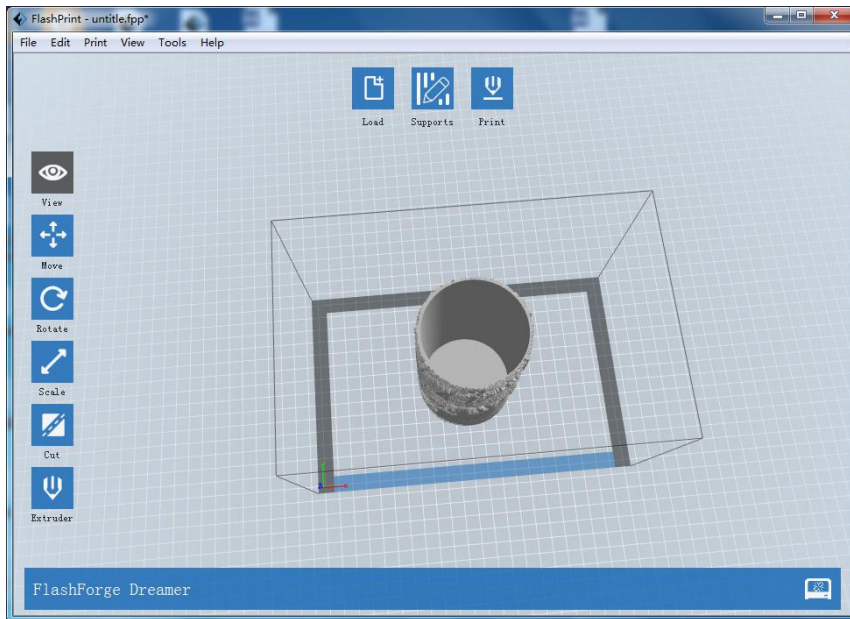




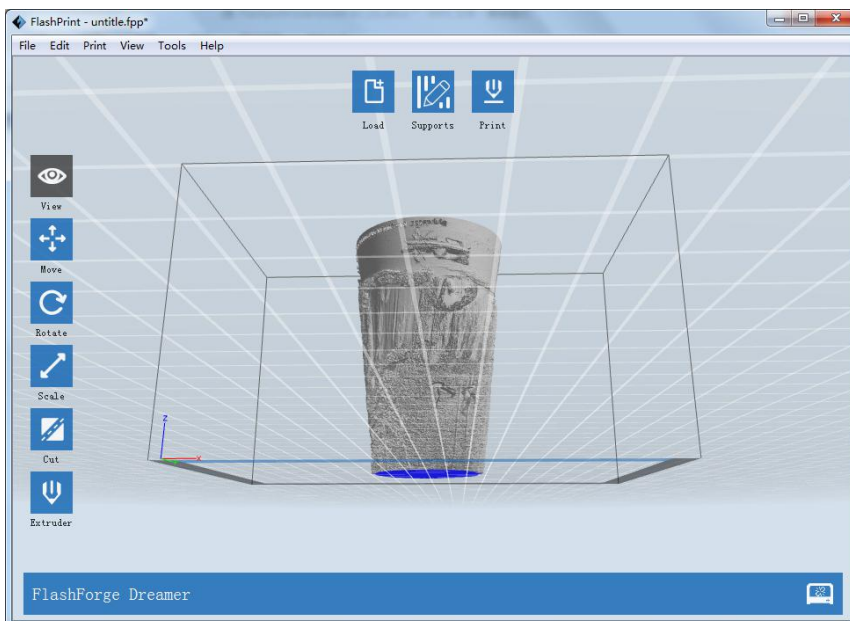
(Plane)



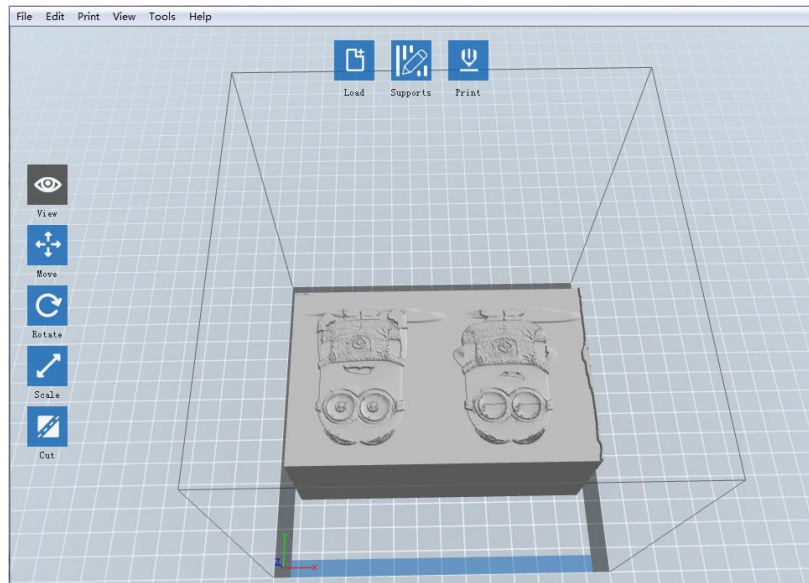
(Tube)



(Canister)



(Lamp)



(Seal)

2.2 Mouse Operations

2.2.1 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.

2.2.2 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.

2.2.3 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.

2.2.4 Scroll the mouse wheel

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

2.3 Change Views

Change your view of the model by moving, rotating, scaling the view and so on.

2.3.1 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.

2.3.2 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.

2.3.3 Scale

You can scale the view of printing frame in the software interface by scrolling the mouse wheel in any condition.

2.3.4 Set

You can set the view of printing frame in the software interface from six angels by following two 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.

2.3.5 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.

2.3.6 Show Model Outline

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

2.3.7 Show Steep Overhang

Click the **View -> Show Steep Overhang** 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.

2.4 Edit Models

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

2.4.1 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.

2.4.2 Rotate

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.

2.4.3 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.

2.4.4 Set

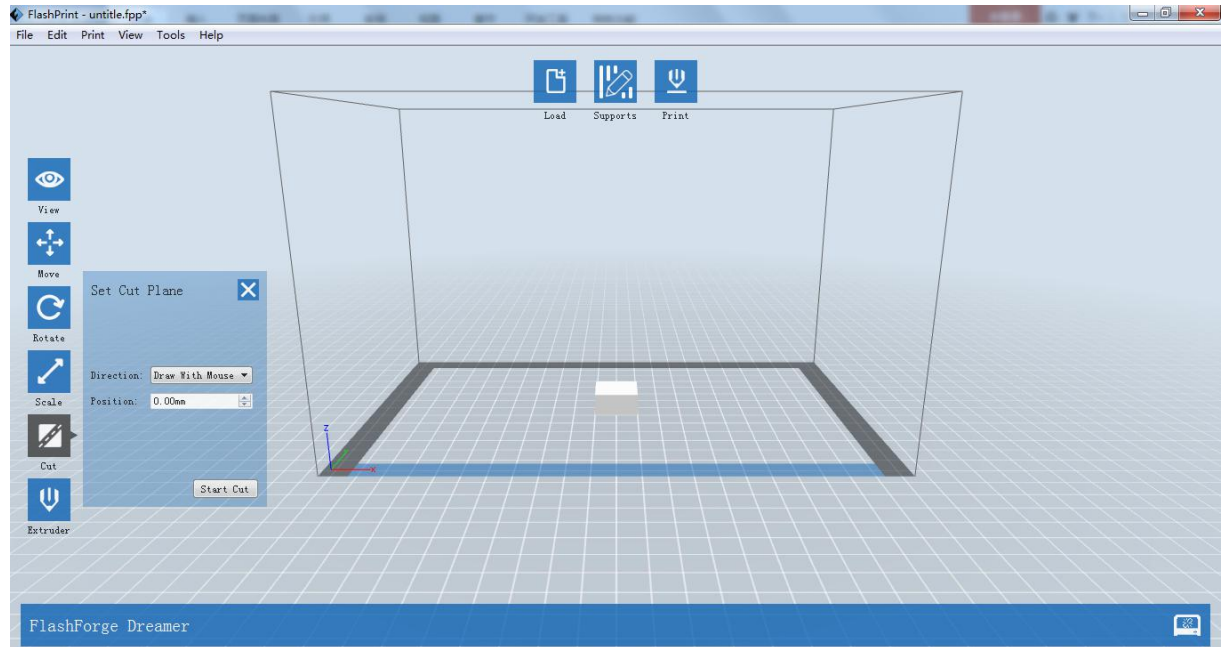
When the model is selected, click **Extruder** button on the left of the software interface and select the extruder you want to print with. The model will turn purple if you choose left extruder or gray if right extruder. If you choose 'Flashforge Finder/Guider', the software will be unavailable for nozzle setting and the default nozzle setting shall be "Right Nozzle".

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.

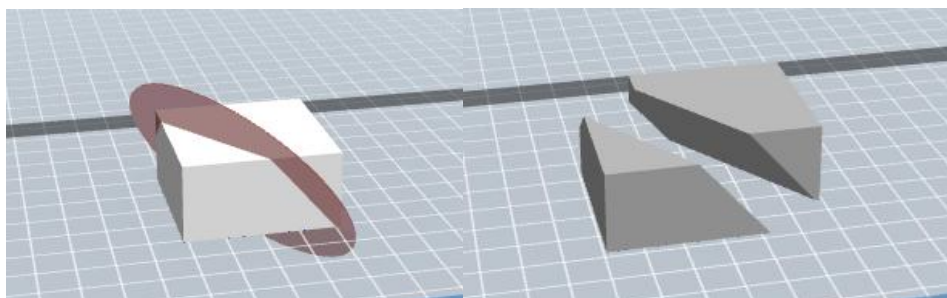
Note: When setting the extruder, mouse operations are same as those of changing views. Please refer to the Chapter 2.3.1~2.3.2.

2.4.5 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) and position.



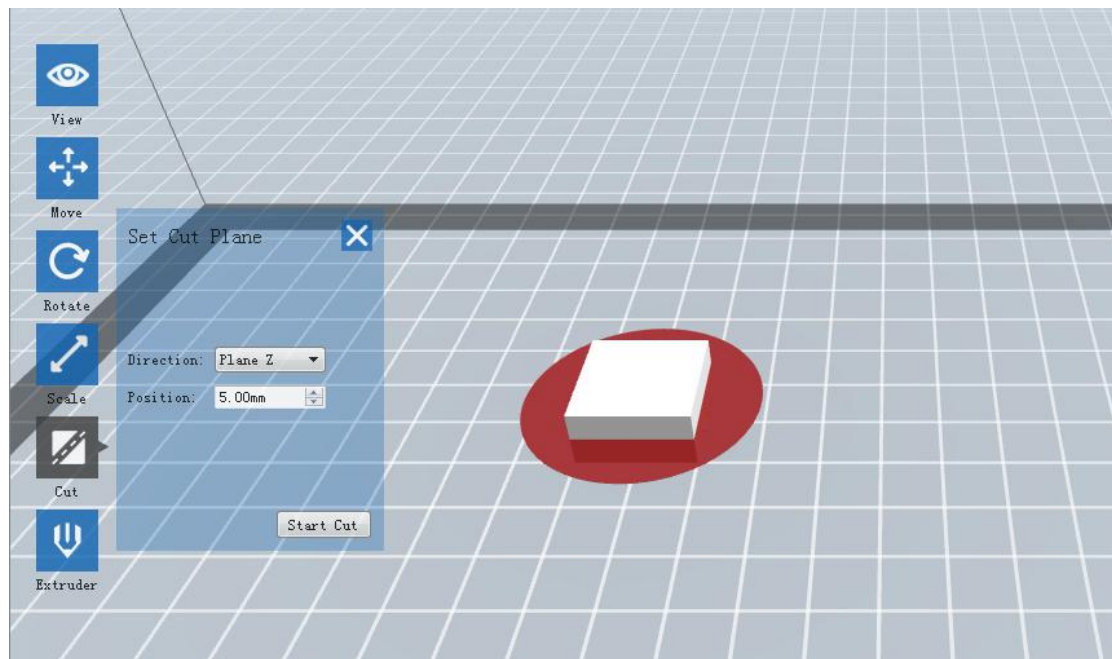
Free-hand plane: Left-click to select the target model and hold down the left mouse button to draw the cutting line (The cutting line will turn into a facet later). To change the position of the facet, users can(1) left-click to select the facet and hold down the left mouse button to drag the facet.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

2.4.6 Others

2.4.6.1 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.

2.4.6.2 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.

2.4.6.3 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.

2.4.6.4 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**.

2.4.6.5 Delete

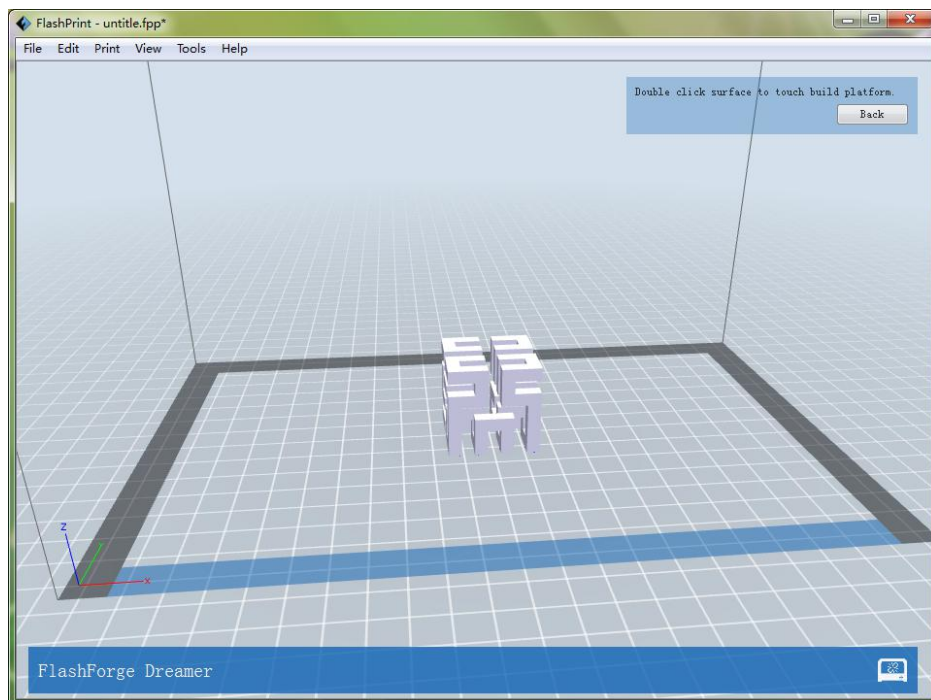
After model is selected, you can delete it by the following two methods:

- A. Click the **Edit-> Delete** menu.
- B. Use the shortcut **Del**.

2.4.6.6 Set side flat

After choosing model, operate as below to set side flat

- ◆ Press "Edit->Set side flat" enter set side flat mode(see picture: 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.



2.4.6.7 Automatic put

After loading one model or several models, press "Edit->Automatic placing", all models will be put in position according to the automatic put rule.

2.4.6.8 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.

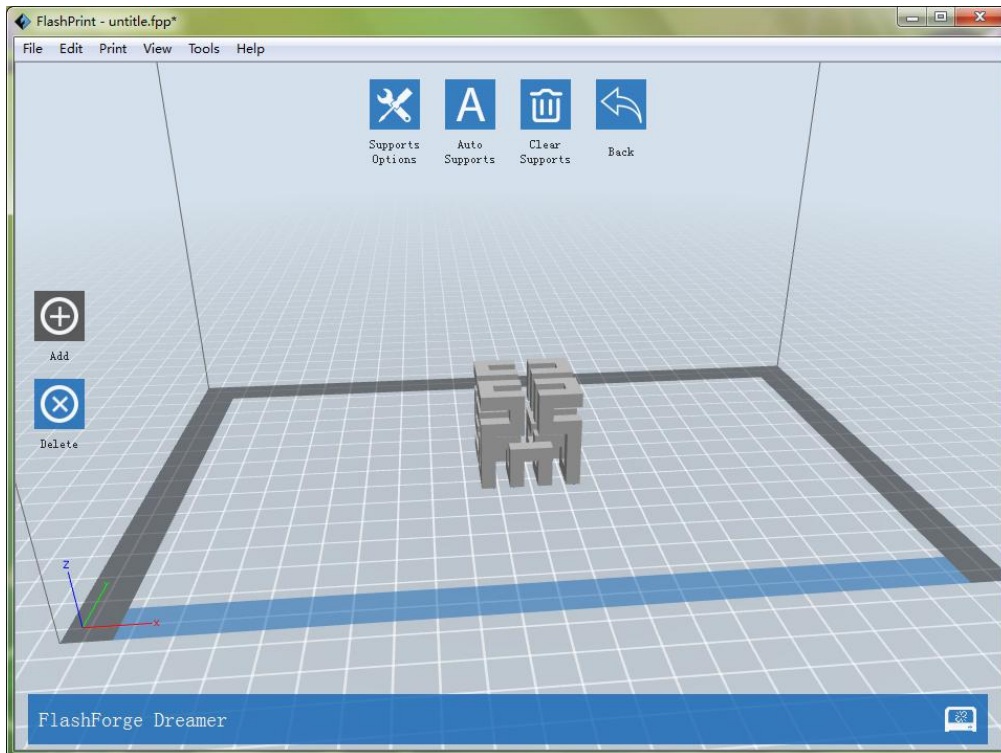


2.4.6.9 Empty Undo-stack

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

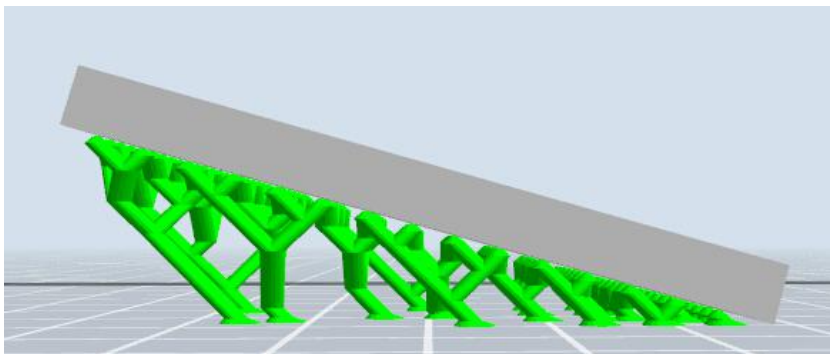
2.5 Edit Supports

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.



2.5.1 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.



2.5.2 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.

2.5.3 Add

Select this option to add supports. 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.

2.5.4 Delete

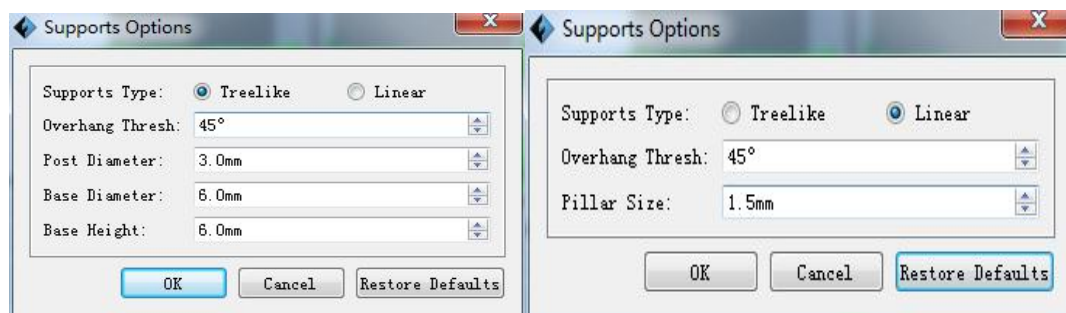
Select this option to delete supports. After selecting this option, move the mouse to the supports you want to delete, then left-click the mouse to delete it.

2.5.5 Support option

2.5.5.1 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.)

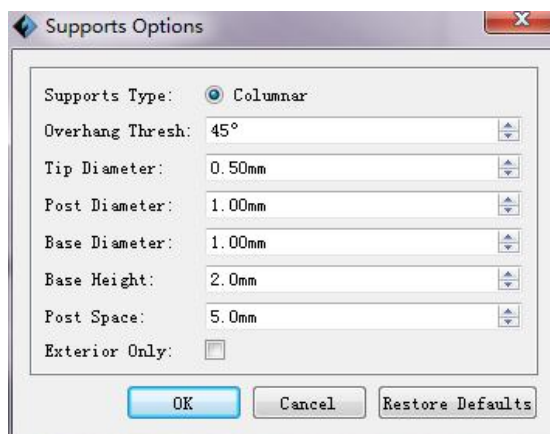


3.5.5.2 DLP 3D Printer

Click [Supports Options] and then a dialog box will pop up "Columnar Supports" and you can setup the parameters of overhang thresh, point diameter, post diameter, base diameter, base

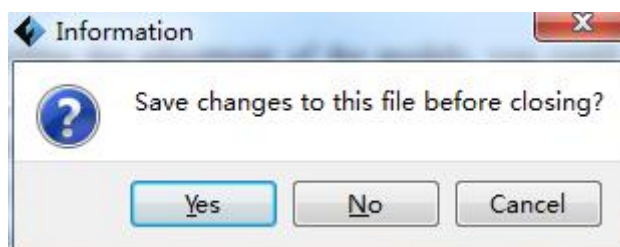
height, post spacing, and external supports then click [OK] to confirm. (Note: Overhang thresh: To setup an angle value. If you input 45, then when it comes to a 3D

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. Point diameter: The diameter of the treelike supports. The value range is 0.25-2mm, and the value could not grater than post diameter value. Base height : the height of the support raft. The value range is 0-5mm. Base diameter: The diameter of the support raft. The value range is 0.25-3 mm. Post spacing: The spacing among posts. The value range is 1-10mm. External supports:Generate supports which can contact with build plate only.)



2.6 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.



2.7 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.

2.8 Printing Process

2.8.1 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).



2.8.2 Connect Machine

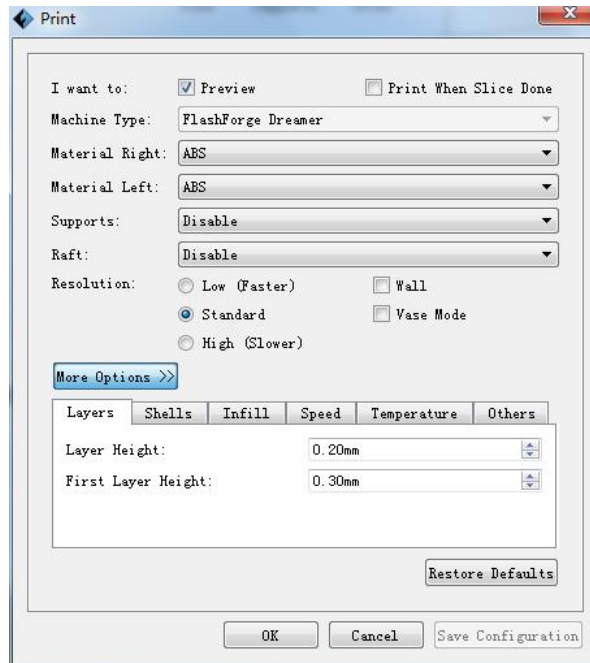
Please refer to Chapter 2.9.1.1 for configurations of connecting to Dreamer 3D printer.

2.8.3 Print

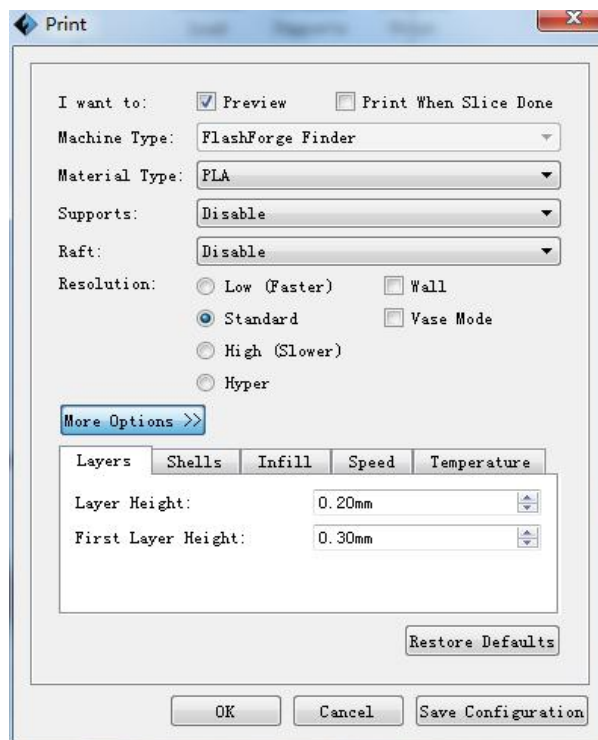
2.8.3.1 FDM 3D printer generate Gcode file

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', the corresponding dialog box of slice parameters shall be "figure 1", Optical materials are PLA, ABS, HIPS, conductive PLA and flexible material. If users choose 'Flashforge Finder/Guider', the corresponding dialog box of slice parameters shall be 'figure 2', Optical materials are PLA, conductive PLA and flexible material.

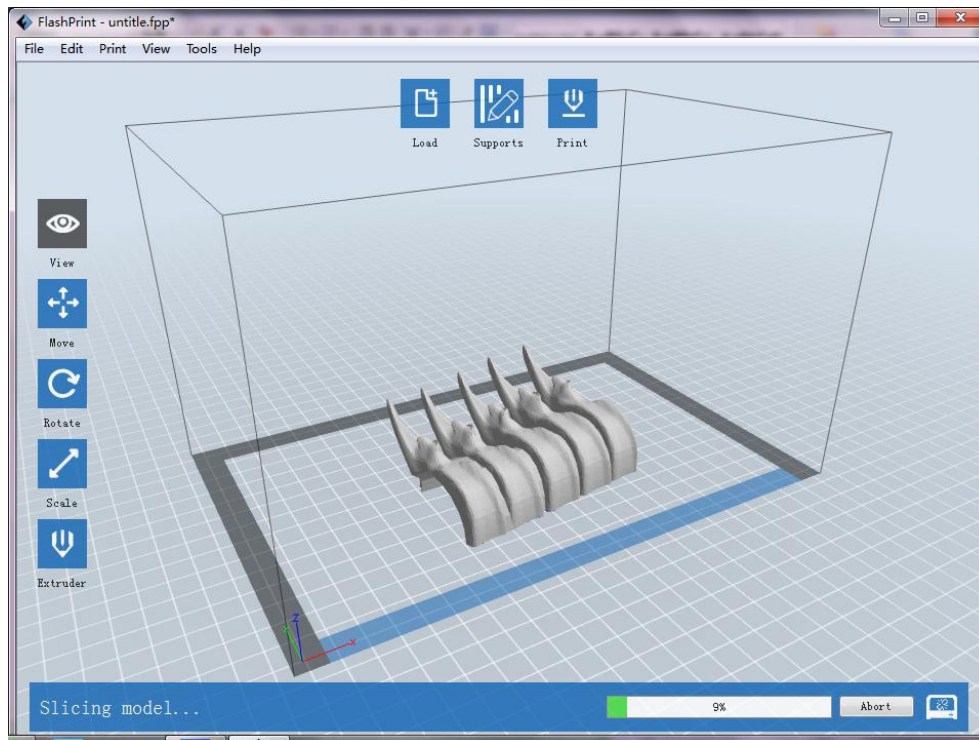


(figure 1)

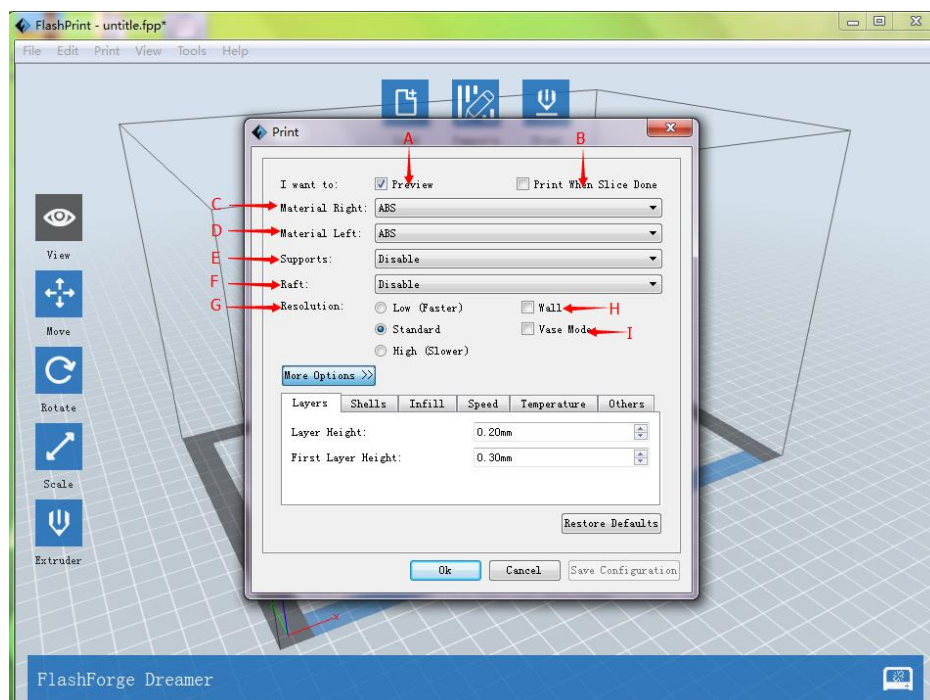


(figure 2)

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.



2.8.3.2 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', select corresponding material for the 'left/right extruder'.

The material types include PLA, ABS, Soluble materials, conductive PLA and flexible material. If users choose 'Flashforge Finder/Guider', select corresponding material and the default material type is PLA, conductive PLA and flexible 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', select left/right extruder for supports adding. If the machine type is 'Flashforge Finder/Guider', choose 'Able' or 'Disable' for raft 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.

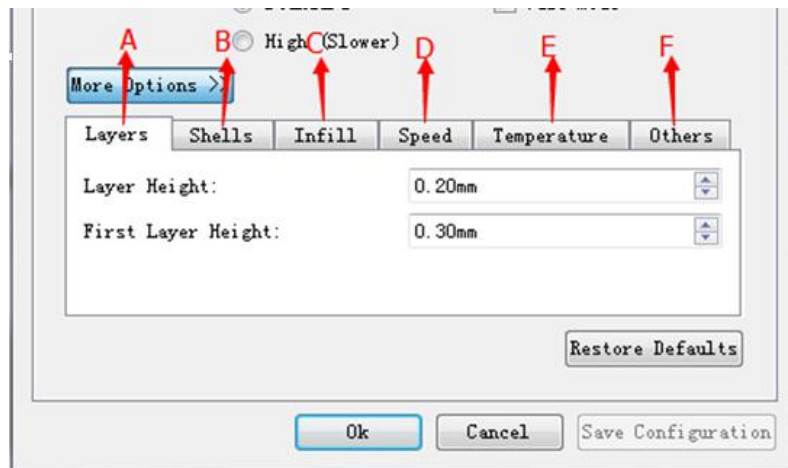
Click the [Raft] drop-down button, if it is 'Flashforge Dreamer/Flashforge Creator Pro/FlashForge Inventor', choose left/right extruder for raft setting. If it is 'Flashforge Finder/Guider', choose 'Able' or 'Disable' for raft setting.

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: When enabled, the interior infill and top solid infill will not be printed.

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.
- 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.



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.

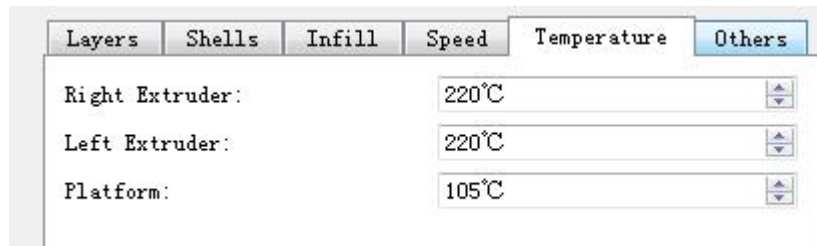
Note: Recommend set the empty speed at 80, printing speed at 60 when use PLA, ABS and conductive PLA filament printing, use flexible material set empty speed at 70, printing speed at 20. Different model have different model parameter, several attempts to find the suitable Parameter.

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. If the machine type is “FlashForge Dreamer/ FlashForge Creator Pro/ FlashForge Inventor” dual extruder 3D printer, Users can set the temperature of dual extruder. As shown below:



If the machine type is “FlashForge Finder/FlashForge Guider” single extruder 3D printer, the setting interface as follow:

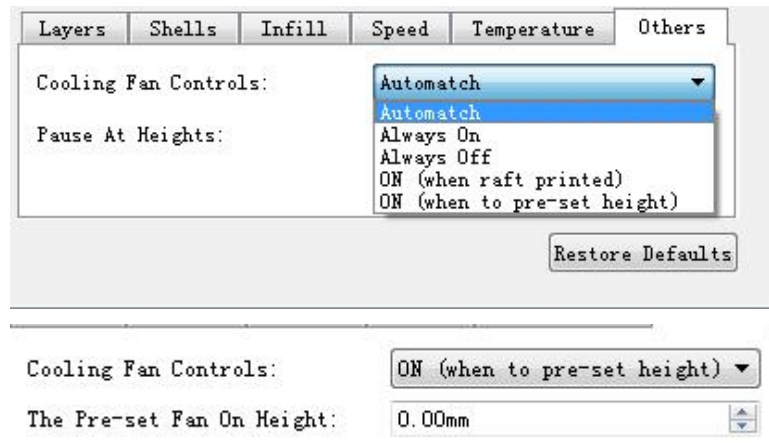


b. Set the temperature for left or right extruder and platform. Set **220** to the **extruder temp** for **ABS**,conductive PLA and flexible material **printing**. Set 50 to the **platform temp** for **PLA**,conductive PLA and flexible material **printing** ,while 105 for **ABS printing**. Set (left: 240, right: 220) to the **extruder temp** for **dissolvable Filament printing**,Set **105** to the **platform temp** .If users choose ‘Flashforge 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, Conductive PLA and flexible material temperature is recommended to set at 230°C.

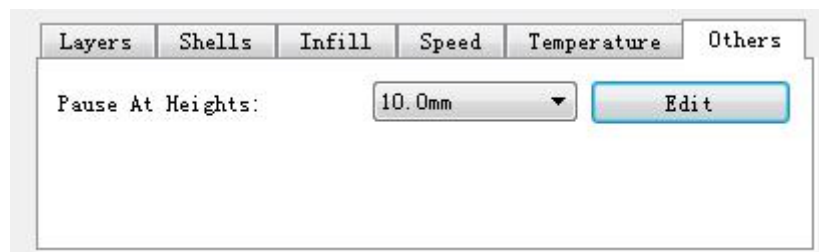
Adjust the temperature according to the condition in order to get a good print.

F. Others

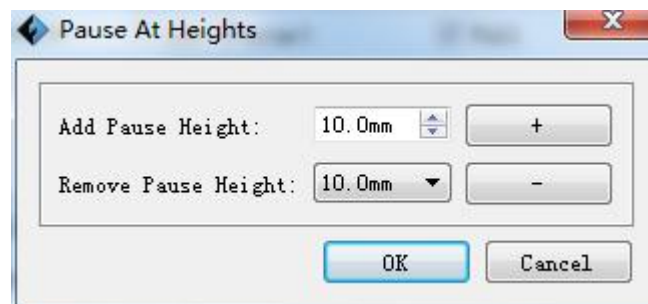
a. Cooling fan controls:Set up the time to turn on the cooling fan. Only used for Dreamer/FlashForge Creator Pro/FlashForge Inventor at present. You can pre-set the height and make the cooling fan begin to work at the point.



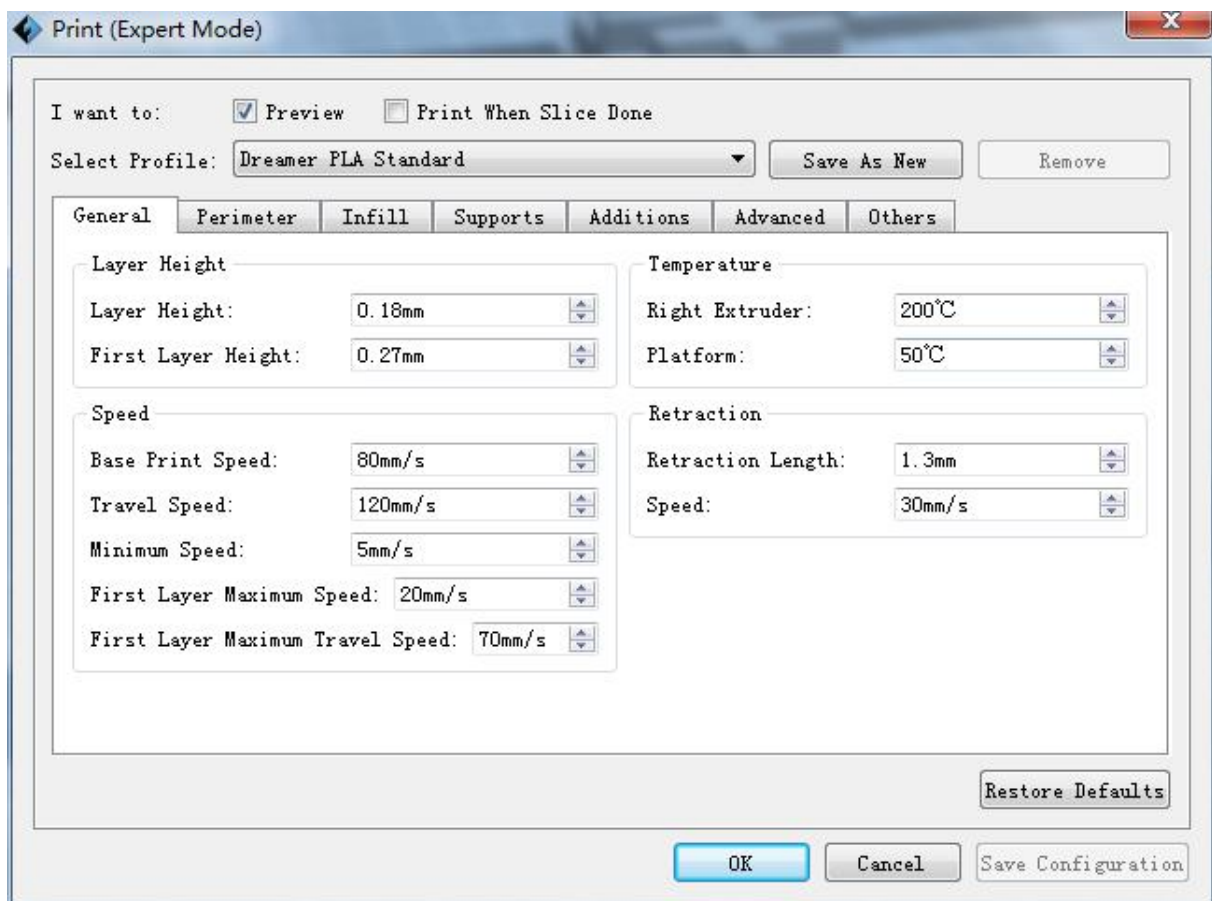
b.Suspend at a Certain Point: Turn on the cooling fan when to specific height.Users can add one or more specific height values(no higher than the model height). It displays as follows.



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.



2.8.3.3 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:** 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.

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.
- b. **Extruder and Platform:** Appropriate temperature of build plate can improve adhesion and help large models avoid warping. If the machine type is

“Flashforge Dreamer/Flashforge Creator Pro/ FlashForge Inventor”, it displays the temperatures of

“Left/Right Extruder and Platform”. If the machine type is “FlashForge Finder/Gudier”, 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/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, 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.

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 two options for Start Point mode. One is “closest to specific location”, the other is “use random start points”.

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.

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

3. Speed:

- a. **Exterior Speed:**Percentage of Base Print Speed for printing outer-most perimeter.
- b. **Visible Interior Speed:**Percentage of Base Print Speed for printing inner perimeters which is visible when print done.
- c. **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.
- e. **Start Angle:** Determines the infill path direction of first layer, subsequent layers will turn 90 degree base previous layer. Hexagon infill is unaffected by this parameter.
- f. **Overlap Perimeter:** Percentage of extrusion width that will overlap between infill and perimeters.
- g. **Vase Mode:** When enabled, the interior infill and top solid infill will not 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 Combine Layers:** This option allows to combine infill and speed up your print by extruding thicker infill paths (The thickness of perimeter keeps unchanged).
- b. **Combine Mode:** If layer height is thicker than 0.1mm Sparse Only is suggested else Sparse And Solid is suggested.

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”, 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.

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.

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. **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.

H. Additions:

1.Raft:

- 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”, it’s available for users to select extruder, while the machine type goes to “FlashForge Finder/Gudier”, 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.

2.Pre-extrusion

- a. **Enable Pre-extrusion:** Print an extra extrusion path before model printing.
- b. **Space to Model:** 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.

3.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. **Select Extruder:** Allows users to select the corresponding extruder. If the machine type goes to “FlashForge Dreamer/FlashForge Creator Pro/ FlashForge Inventor”, it’s available for users to select extruder, while the machine type goes to “FlashForge Finder/Gudier”, it’s unavailable for users to select extruder.
- c. **Shell Count:** TNumber of paths superimposed for building wall. More paths will generate a higher strength wall but take more print time.

- d. **Space to Model:** Determines the spacing between model and wall.
- e. **Speed:** Speed at which the wall is printed.

4.Wall

- 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 or right extruder if the machine type is "FlashForge Dreamer/ FlashForge Creator Pro/ FlashForge Inventor"; If the machine type is "FlashForge Finder/ Guider" there is no need to set extruder, the default extruder is right.
- c. **Enrichment:** The width of Brim, the wider brim is, the more stable it will be, the longer it will take.
- d. **Layer of brim:** The more layer of brim the better anchoring the edges of the model, the harder to remove.

I.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.

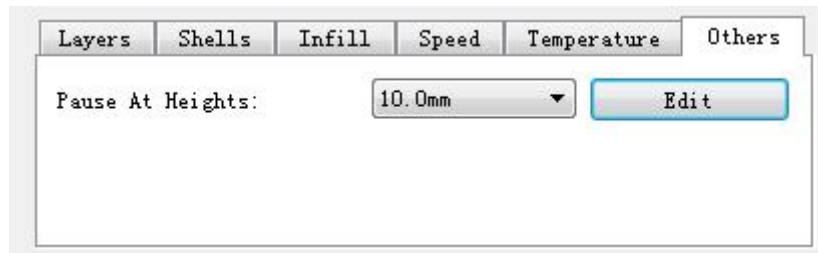
J.Others

1. Cooling Fan Controls

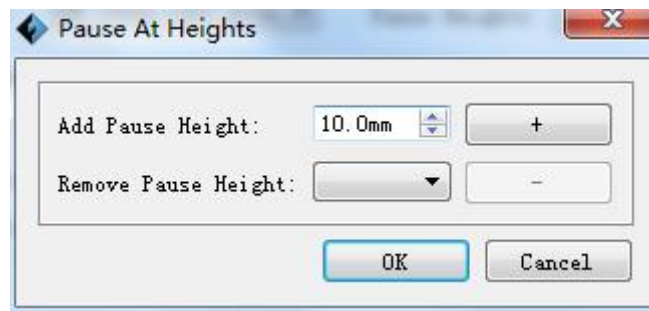
- a. **Cooling Fan Controls:** Set up the time to turn on the cooling fan. Fan Status: Including "Always On", "Always Off", "ON(when raft printed)", "ON(when to pre-set height)", which only apply to "Dreamer/FlashForge Creator Pro/FlashForge Inventor" at present.
- b. **The Pre-set Fan On Height:** Turn on the cooling fan when to specific height.

2. Pause At Heights

- a. **Pause 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



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.



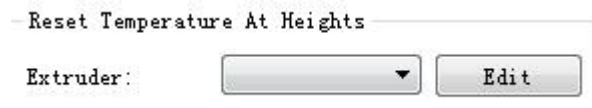
- b. **Edit Pause At Heights :** Add or remove the Pause Heights.

3. 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.

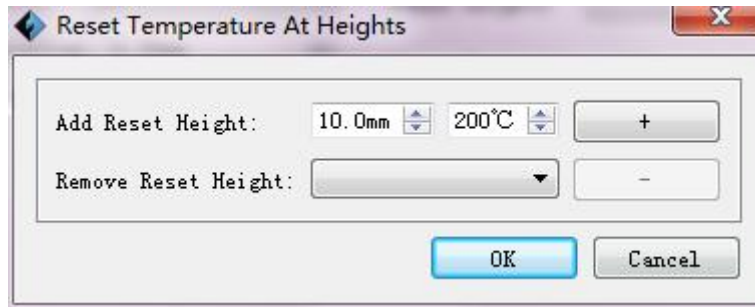
4. Rest extruder temperature once reach to specific heights

- a .Exturder: 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:



b.Edit: add or remove pause height

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. 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.

L. Remove

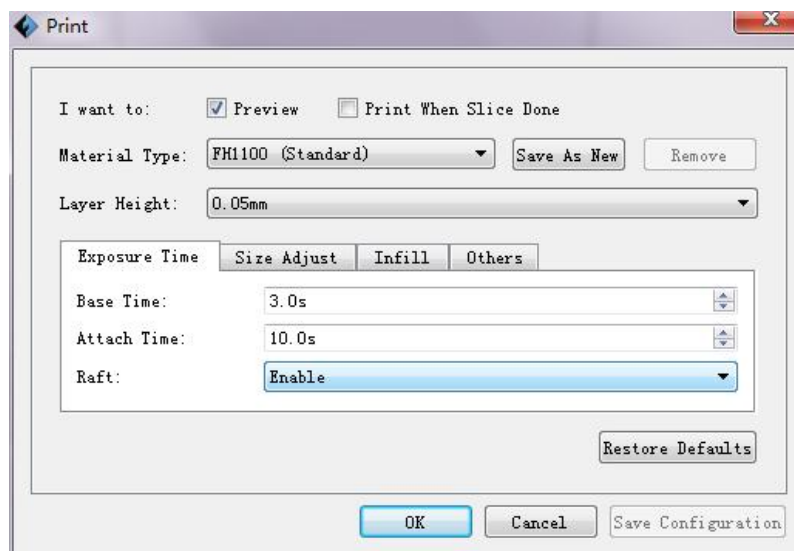
Allows the users to delete the added scheme(s). Select one of the added schemes, click [Remove] and a dialogue box will pop up for confirmation. Click [Yes] to remove it or click [No] to cancel the current operation.

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

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

2.8.3.2 DLP 3D printer Hunter 3D printer

Before printing, the model has to be sliced and generate Svgx printing file. Please refer to 2.8.1 to learn how to generate Svgx files. Slicing parameter as shown below:

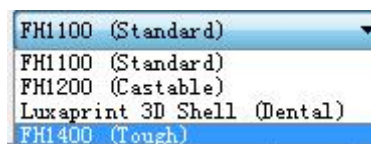


1. Slicing parameter configuration

a. Print Preview: to select whether to enter into preview page.

b. Start to print after slice done: to select whether to start printing immediately after slice done.

c. Material type: there are four materials in the option, the default material is FH1100. The parameters have been ready for different materials.



Layer height: the printing thickness of each layer. The default value is 0.025mm.

prototyping time:

Basic prototyping time: the prototyping time of except layer, different photosensitive resin need different time, the default time is 3.0s, the range is 1.0s-60s.

Layer prototyping time: the prototyping time: of model layer, different photosensitive resin need different time, the range is 1.0s-60s.

Platform: the platform has been printed before print model, which help model to stick to the built plate. size calibration:

X calibration: when the printing size is different form the model size, users can set X calibration value to make up the errors form X axis.; default value 100%, range 50%-150%.

Y calibration: when the printing size is different form the model size, users can set Y calibration value to make up the errors form Y axis.; default value 100%, range 50%-150%.

Note: if there is big error between the printing size and model size, please contact the factory custom service.

Infill:

Infill density: set the degree of infill density, range is 0-100%.

Shell thickness: the thickness of model shells, range is 1.0mm-10.0mm.

Infill thickness: the thickness of internal infill, range is 0.5mm-5.0mm.

Others:

Light intensity: adjust the light intensity of ray machine, different photosensitive resin would be different.

2.8.3.3 FDM 3D Printer Print Gcode File

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** and **Estimated Print Time** it costs on the top right corner. Then click **Print** button on the same location, the machine starts to print while click **Back** button to exit from Preview mode.

2.8.3.4 DLP 3D Printer Print Svgx File

The operation of Svgx files is same with Gcode files, please refer to 2.8.3.3

2.8.3.5 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. 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.

2.9 Printer Operations

2.9.1 Connect / Disconnect

You can connect Dreamer to Flashprint software via USB cable or WIFI. 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.

2.9.1.1 Connect

A. USB cable connection

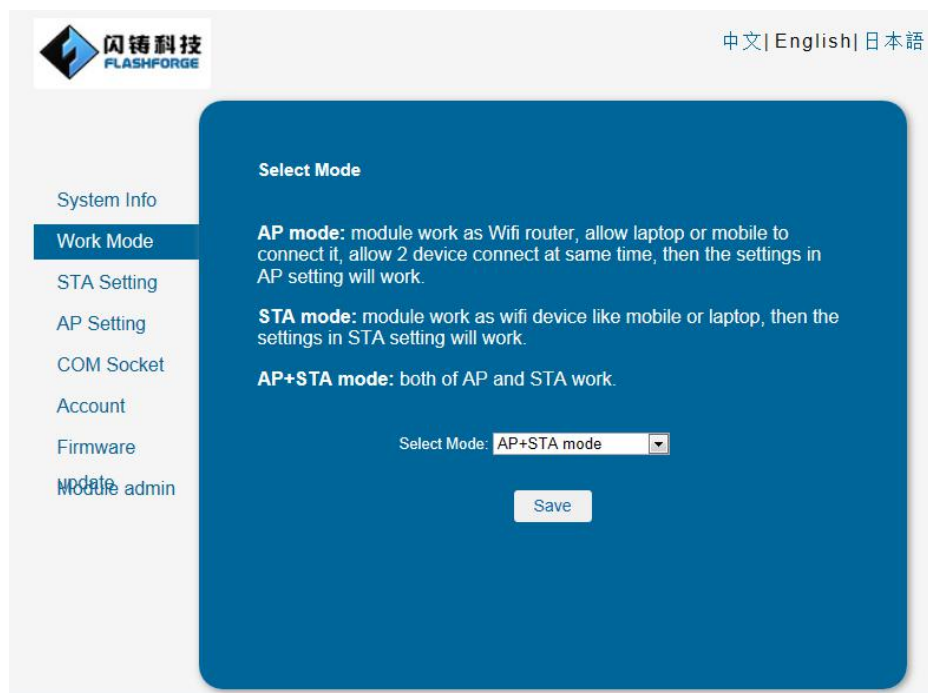
- a. Find USB ports on the right-side of machine and computer, plug in and connect.
- b. Open Flashprint software and turn on the printer.
- c. Click **Print -> Connect Machine** menu, then select USB in the **Connection Mode** option and select machine you want to connect in **Select Machine** option. 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.

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.



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.

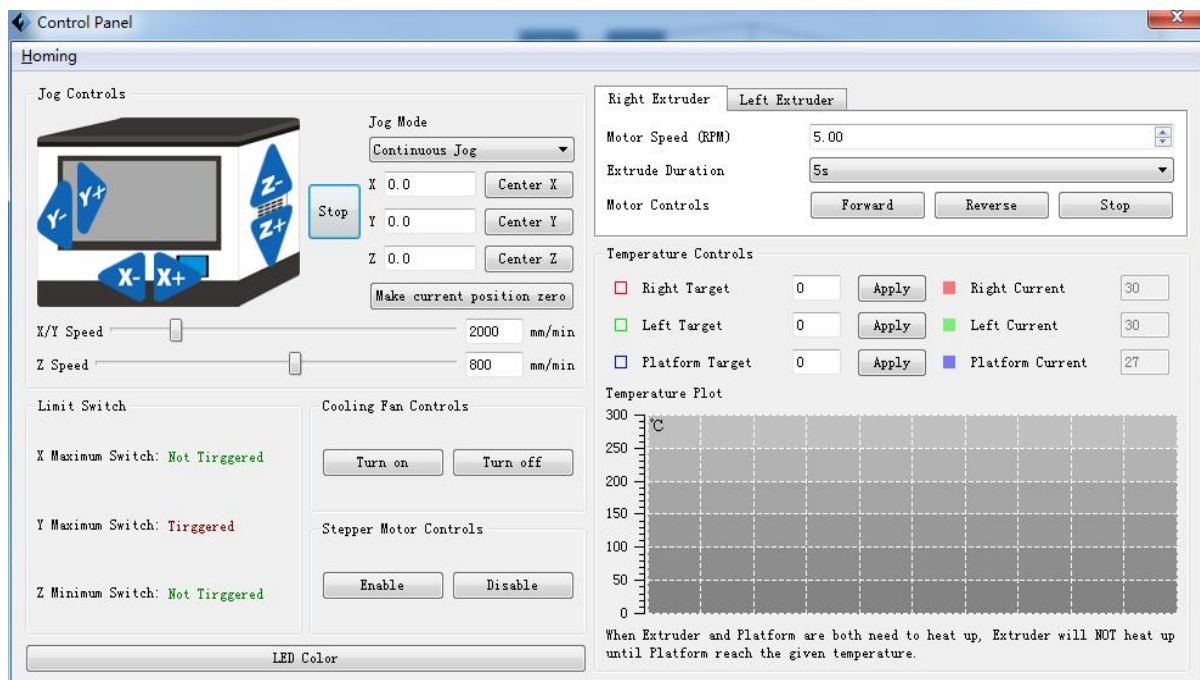
2.9.1.2 Disconnect from the printer

Click **Print -> Disconnect** menu on top of the software interface, the computer will disconnect from the 3D printer.

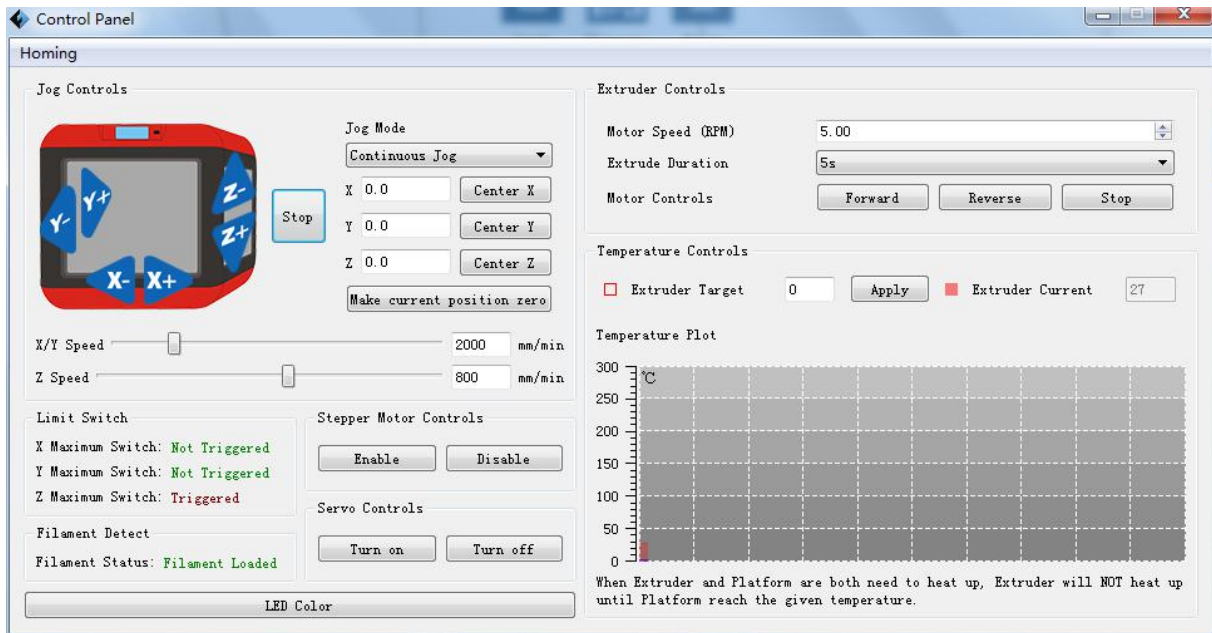
2.9.2 Control Panel

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.

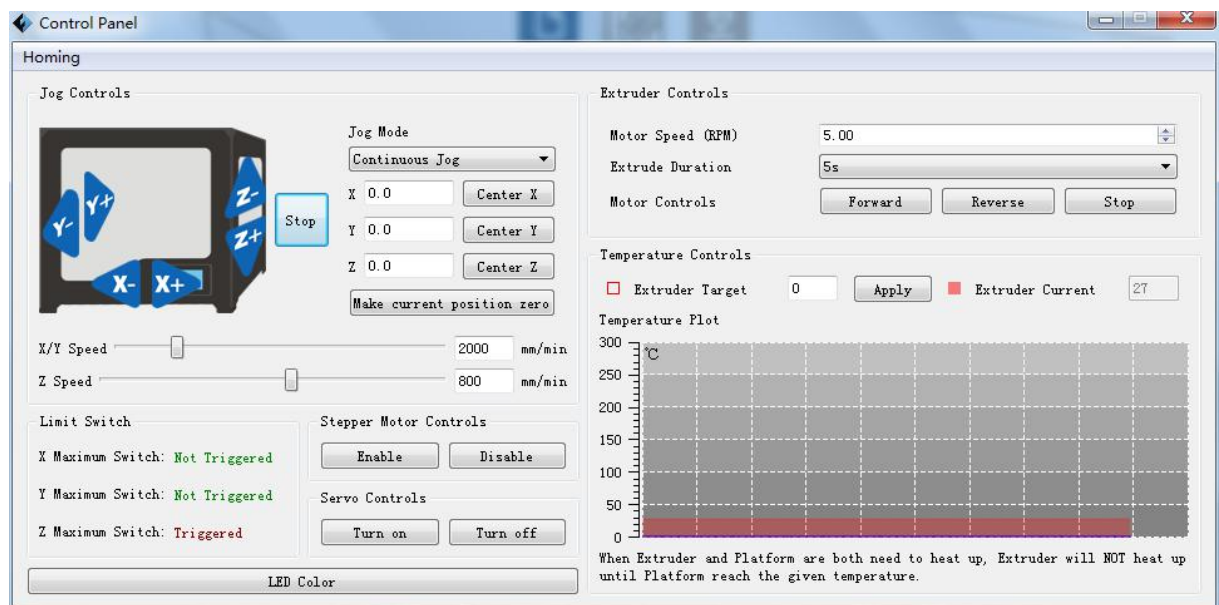
The control panels of different machine types are different. The control panel of 'Flashforge Dreamer/Flashforge Creator Pro/Flashforge Inventor' shows in the 'control panel 1', while that of 'Flashforge Finder' shows in the 'control panel 2',while that of 'Flashforge Guider' shows in the 'control panel 3'. (this function is only available to FDM 3D printer)



(Control panel 1)



(Control panel 2)



(Control panel 3)

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.

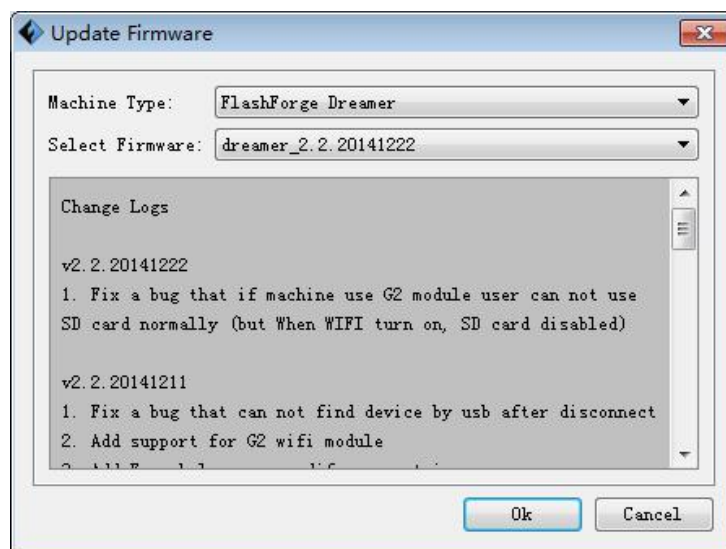
L. Servo Control

To control the servo's on-off.

2.9.3 Firmware Upgrade

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

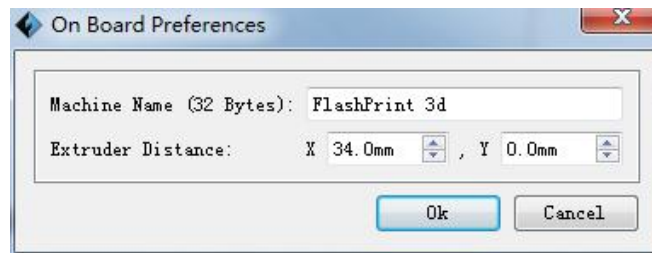
- A. Disconnect the computer and printer, or Click **Tools -> Update Firmware** menu and select **Disconnect** option,
- B. Select your machine type and corresponding firmware in the popped up dialog and then click **Ok** to upgrade. If the machine is idle, it will start to upgrade to new firmware.



2.9.4 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 Finder/Guider', the content only includes the name of the machine. (this function is only

available to FDM 3D printer)



2.9.5 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)

2.9.6 Driver Installation

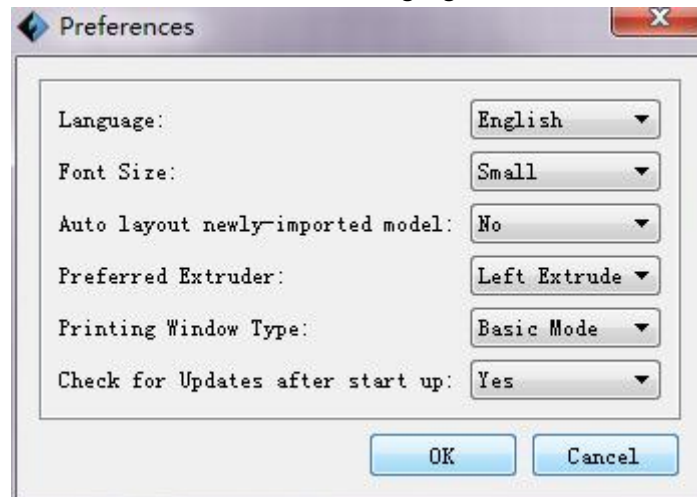
- A. Open the root directory of the software. (e.g. C:\Program Files\flashforge\FlashPrint)
- B. Open the driver folder which in the root directory and find the driver. There are two driver installation packages, please install one of them according to your computer system. For 64bit edition, please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

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

2.10 Others

2.10.1 Preferences

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



A. Language

So far this software supports English, Japanese, Simplified Chinese ,Traditional Chinese,French, 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. If 'font size' chose 'large', the software font is large font mode, otherwise is small font mode. The default font size is small font size.

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 Pro 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)

2.10.2 Help Contents

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

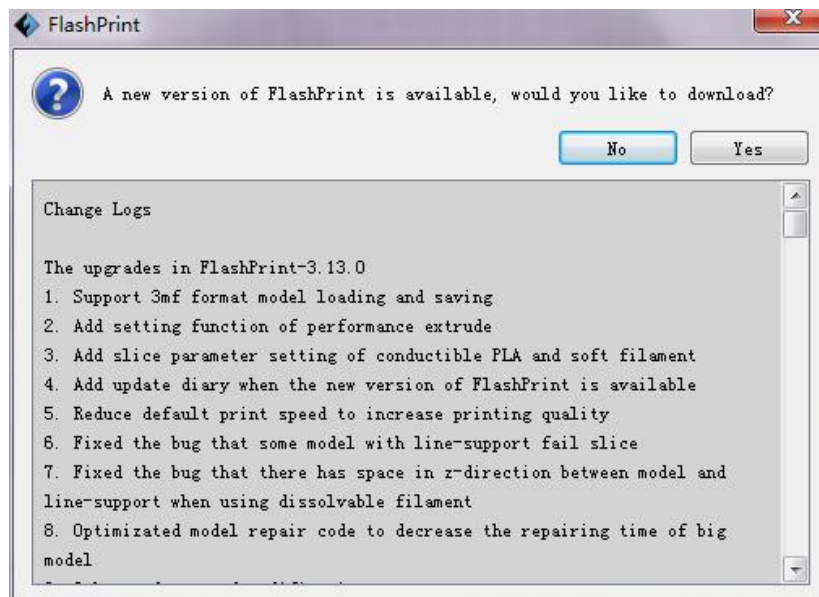
2.10.3 Check for Updates

A. Automatic Update

Refer to 2.9.1 to set auto 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 .

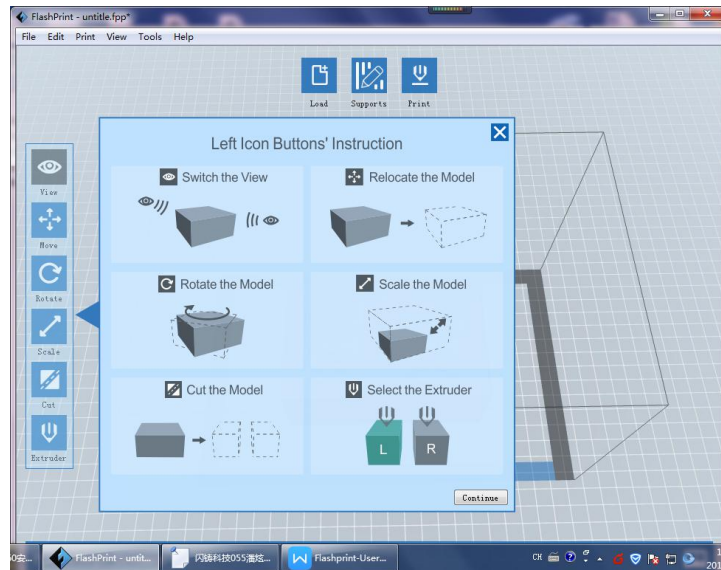


1.1~1.2 to install the software.

2.10.4 First time user guide

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.



2.10.5 Software Information

Click **Tools-> About FlashPrint** menu to review information about the software, including software version and copyrights.

