
PowerSpec Ultra 3D Printer Start-up Guide

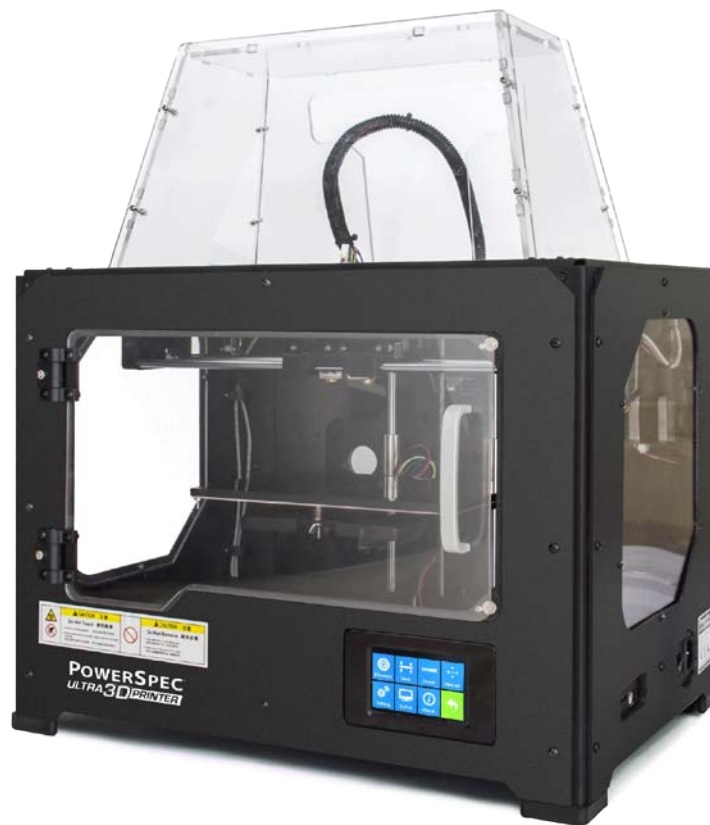


Table of Contents

1 What's Included in the Box?.....	Page 3
2 Un-boxing.....	Page 4
3 Initial Hardware Installation.....	Page 6
4 Software Instruction.....	Page 8
5 Filament.....	Page 24
6 Initial Print.....	Page 27

PRECAUTIONS:

Please ensure this page is read carefully prior to setting up or operating the 3D printer.

-  The 3D printer is very sensitive to static electricity, so please ensure sure you connect to a properly grounded surge protected power outlet.
-  Before repairing or making any alterations to the 3D printer, it is essential that the machine is turned off and the power cord is unplugged.
-  The 3D printer operates at very high temperature so please allow: the nozzle, the extruded plastic, and the heating plate to cool before handling them.
-  Some plastic filaments may give off a slight odor when heated and because of this; the machine should always be operated in a well-ventilated area.
-  Do not wear gloves when operating or repairing the 3D printer, as entanglement may occur and result in injury.
-  Do not leave the machine unattended while in operation.

1 What's Included in the Box?

Along with your 3D printer, the box also contains the following:

In the two long boxes on top:

- Power cord x 1pc
- USB type A to type B cable x 1pc
- Sensor Line x 1pc
- Filament guide tube x 2 pcs
- Filament spool holder x 2 pcs

In the 3D printer Acrylic Accessory box:

- Acrylic sheet x 6pcs
- Handle x 1pcs
- Accessory bag x 1pc

An accessory box on the top of the 3D printer:

- Dual extruder heads x 1pc
- 4G SD card x 1 pc
- Bolt, hex wrench, and screws kit x 1 bag.

2 Un-packing

The 3D printer was carefully packaged at the manufacturing facility. Please follow the unpacking steps as noted below.

CAUTION

-  Handle the package and its contents with extreme care and do not use any unnecessary force.
-  Do not remove the thin yellow film from the heating plate. It is heat resistant tape that improves the adhesion of the extruded plastic to the plate.
-  Do not remove the wrapping around the nozzle. It consists of a ceramic fiber fabric and heat resistant tape that helps to keep the nozzle at a consistent temperature.

First, put the box on a clean, flat surface. Remove the top long box and inside you will find: **1 power cord, 1 USB type A to type B cable, 1 sensor line , 2 filament holders, and 2 filament guide tubes.**



You will find another box containing the relevant assembly tools under the long box. You should assemble the Acrylic cover yourself using these tools, but please view the assembly video contained in the SD card.



Please remove both of the foam supports holding the printer in place and [make certain that you can grasp the frame firmly, and then lift the printer out of the carton.](#)



Now you can see the top of the printer along with more boxes inside. The large box with the black wire is the **accessory box**. This contains the **dual extruder heads, SD card, and other important components**. Don't remove the accessory box or its contents yet. [Note: Do not lift the box by the black cable! Doing so could cause damage to the component.](#)

Please remove the protective wrap and you can now open the accessory box and remove the accessory sleeve. You will find the **dual extruder heads** in the protective packaging along with the black cable, carefully remove it and place it on your work surface. Remove the cardboard packing material and take the **accessory box** from the printer, set it aside for later.

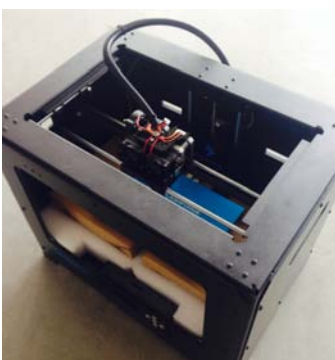


The build platform should now be visible. It is an aluminum plate covered in a thin polyamide film. This is the surface that your objects will be printed on. [Note: Remember to not remove this film.](#)

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

1. Turn the screw which is behind the rotating platform.
 2. Grasp the printing platform with one hand on each side raising it slowly while keeping it level.
- Stop once the platform is just a little below the bronze nozzle.

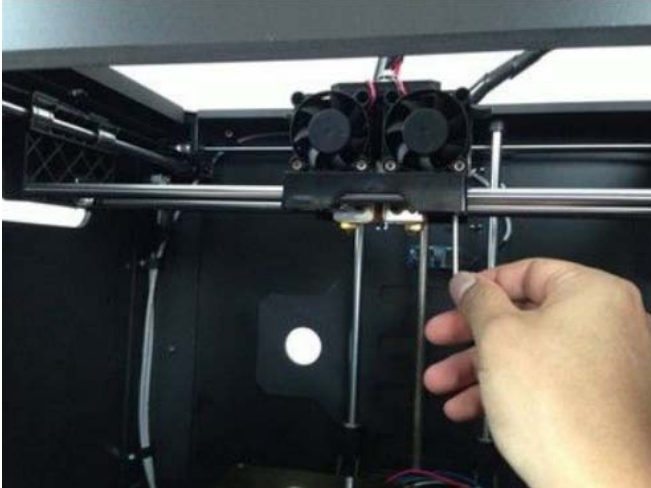
Now you can see two empty boxes, you can easily take them out after removing the remaining packaging.



3 Initial Hardware Installation

You will need the two shortest silver screws from the kit found in the accessory box, and the appropriate hex wrench.

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



Next is the installation of the spool holders install one on each side.

The installation of the spool holders is very simple – just insert it into the circular opening and tighten the nut behind it. Then install the **filament guide tube** onto the empty spot on the extruder. Place one end of the guide tube into the hole. The hardware installation is now almost complete.



Next, with the power switch in the **OFF** position, confirm that the power cord is plugged into the power outlet next to the power switch.



Finally install the filament on the [spool holders](#)



Congratulations! You have completed the initial hardware installation! You are now ready to start printing. Please proceed to the next step: Software Installation.

4 Software Installation

1.1 Preparation

A. Insert the included SD card with 3D printer software into your computer. We have already prepared the latest driver installation package in the SD card. Copy the RAR program to your PC.

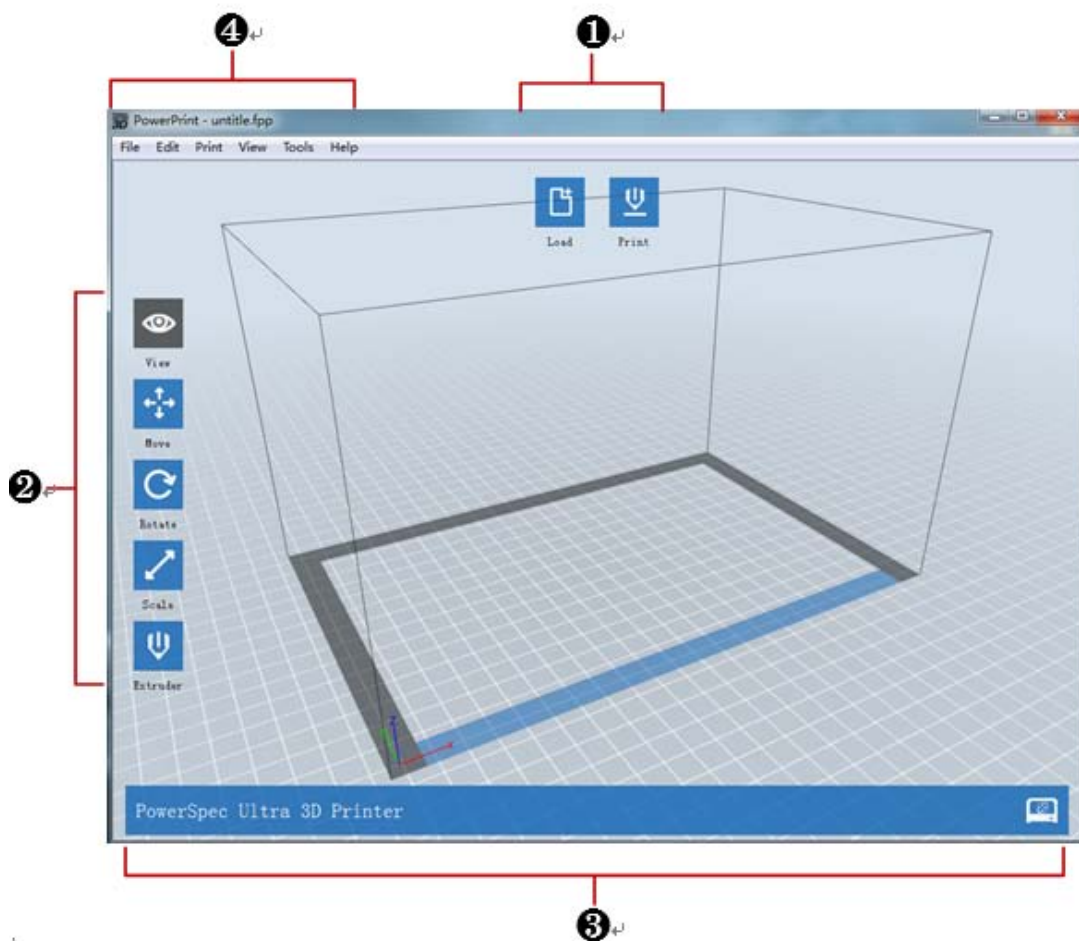
1.2 Installation & Startup

A. Uncompress the copied RAR file, and complete the installation according to the provided instructions.

B. Connect your 3D printer to your computer using the included USB cable, ensure that the printer is connected to your computer successfully. Start the software by selecting the shortcut on your computer's desktop or in the Start Menu.

Note: This software supports Windows XP/Win 7/Win8 32-bit and 64 bit, and Mac operating system.

2. Software Operating Instructions



Load one or multiple files.



View Powerprint home screen from one of six viewing angles.



Move model around on the 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 3D printer or export to your SD card.

You can use Powerprint software to control 3D printer and perform printing tasks.

2.1 Load File

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

- A. Click the “Load” button at the top of the software interface and then select the file to be loaded.
- B. Select the file to be loaded, and then drag the file to the software interface.
- C. Click the “File -> Load File” menu. Then select the file to be loaded.
- D. Click the “File -> Examples” menu. Then select the file to be loaded.
- E. Click the “File -> Recent Files” menu. Then select the file to be loaded.
- F. Select the file to be loaded then drag the file to the software icon.

You can load .stl, .obj, or .fpp model files which are editable using this software. Please refer to Chapter 2.2~2.5 and you can find instructions regarding modifications, 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, or Print it.

Important Note: Gcode files cannot be edited, 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 3D printer and how to print the Gcode file.

2.2 Mouse Operations

2.2.1 Left click

- A. Select a model by moving the cursor to it and then left click.
- B. Select multiple models by holding the **Ctrl** key and then left click each model you want.
- C. Models look brighter when selected.
- D. Models can be edited when selected.
- E. Left click any blank space to undo the selection of model.

2.2.2 Left click and hold the left button

When changing viewing angles or editing models, different effects will appear 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 affects as with any operation when right clicking and holding the right button. Please refer to the Chapter 2.3.1 & 2.3.2.

2.2.4 Scrolling using the mouse wheel

Same affects as with any operations when scrolling using 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 using the following three methods:

- A. Press and hold the left button of the mouse and then drag the cursor.
- B. Press and hold the middle button of mouse and then drag the cursor.
- C. Hold the **Shift** key while pressing the right button of the mouse and then drag the cursor.

2.3.2 Rotate

You can rotate the view of printing frame in the software interface using the following two methods:

- A. Hold the right button of the mouse and then drag the cursor.
- B. Hold the **Shift** key while pressing the left button of the mouse and then drag the cursor.

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 to observe the model from six angels by the following two methods (**Top/Bottom/Front/Back/Left/Right View**):

- A. Click the **View** menu, and then select the view to use to observe the model.
- B. Click the **Look** button on the left of the software interface, and then select the view you require.

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, where the surface has a steep overhang and it is highlighted red in the software. Overhang threshold value can 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 the model location on the build platform using the following two methods:

- A. Click the **Move** button on the left of the software interface. Holding the left button of the mouse and then drag the cursor to adjust the location of the model in the XY direction. Hold the **shift** key and while pressing the left button of the mouse, drag the cursor to adjust the location of the model in the Z direction. You can see the distance and direction of the movement which refers to the relative distance between present and former locations.
- B. Click the **Move** button on the left of the software interface and then enter the distance value you want to move on the 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 on the build platform using the following two methods:

A. Click the **Rotate** button on the left of the software interface and you will see three mutually perpendicular rings: red, green and blue. Click one of the rings and you can rotate it on the present axis, you will see the rotation angle and direction in the center of circle. Using this method you can rotate your model on its X/Y/Z axis.

B. Click the **Rotate** button on the left of the software interface, and then enter the rotating angel values in the X/Y/Z axes positioning. Click the **Reset** button to reset the rotating angel values.

2.4.3 Scale

When the model is selected, you can change the size of the model on the build platform by using the following two methods:

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

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

Note: Select the **Uniform Scaling** option and it will scale the model in equal proportions when changing values in any positioning of the model. Not selecting this option will only change the value of the corresponding positioning.

2.4.4 Set

When the model has been selected, click the **Extruder** button on the left of the software interface and then select the extruder you want to print with. The model will display purple if you choose the left extruder or gray if the right extruder.

If a model can be printed with dual extruders, or support is needed to complete the printing, you can use the dual extruders to print 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. A dual-color model is combined with two single color models.

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

2.4.5 Other Functions

2.4.5.1 Undo

To **Undo** the most recent edit you made to your model file use the following two methods:

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.

2.4.5.2 Redo

To **Redo** the most recent edit you have undone to your model file use the following two methods:

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.

2.4.5.3 Select ALL

Using the following two methods, you can 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.

2.4.5.4 Duplicate

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

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

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

2.4.5.5 Delete

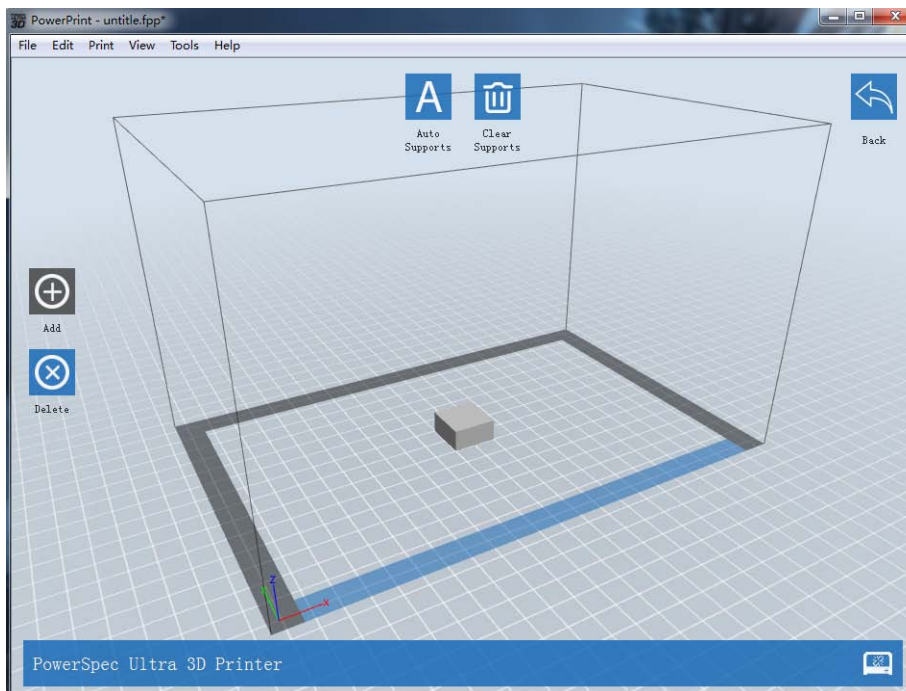
After a model is selected, you can delete it by using the following two methods:

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

B. Use the shortcut **Del**.

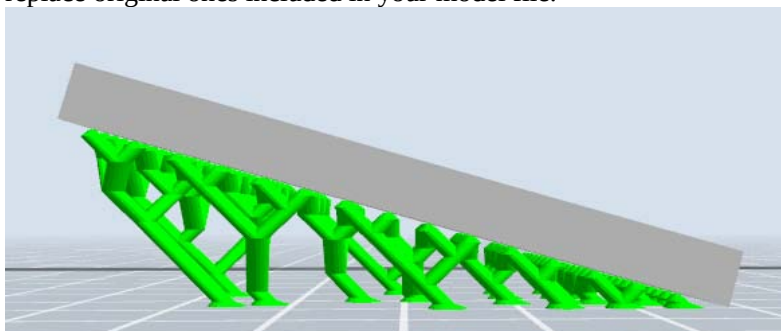
2.5 Edit Supports

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



2.5.1 Auto Supports

Select this option to have your model built with support structures. Powerprint software will automatically generate supports for any overhanging sections of your model, 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

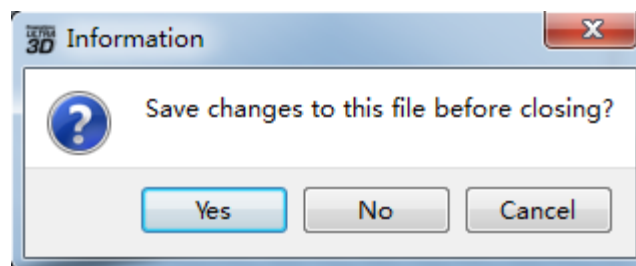
Select this option to add supports. After selecting this option, move your mouse to where you want to add the support, and then left-click the mouse to select the start point of support structure. Press and hold the left mouse button down, drag the cursor, and then you can preview the support structure 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, the 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.

2.6 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 using the following two methods:

- A.** Click **File -> Save Project** to save your file use only .fpp format. All models (including supports) are in the .fpp format file, are independent, and can be edited again when reloading the file. Additionally, an extruder configuration and model locations remain same when saving the file.
- B.** Click **File -> Save As** to save your file with .stl, .obj, or .fpp format. All the models (including supports) in a .stl or .obj format file are not independent, but combined to a new model. All model locations remain the same as saving the file, while an extruder's configuration becomes different when reloading the file.

2.8 Printing Process

2.8.1 Choose Machine Type

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

2.8.2 Connect Machine

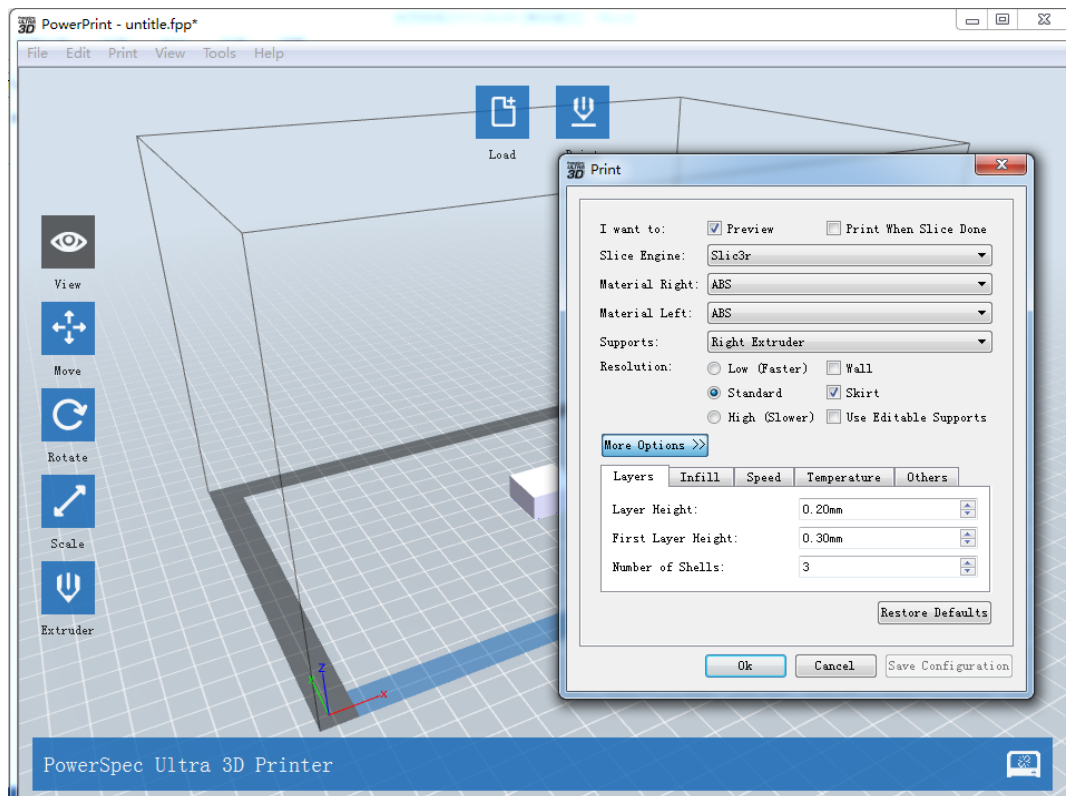
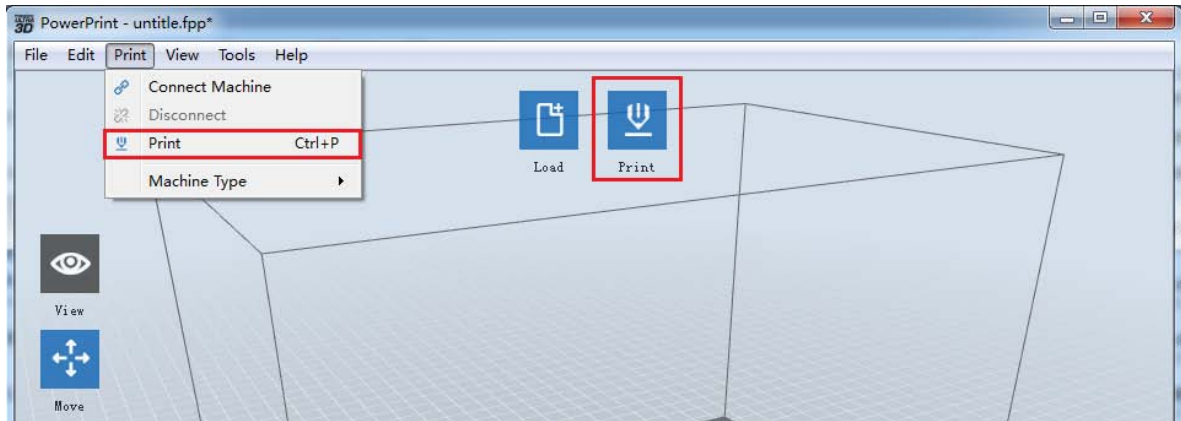
Please refer to Chapter 2.9.1.1 for configurations of connecting to 3D printer.

2.8.3 Print

2.8.3.1 Generate Gcode File

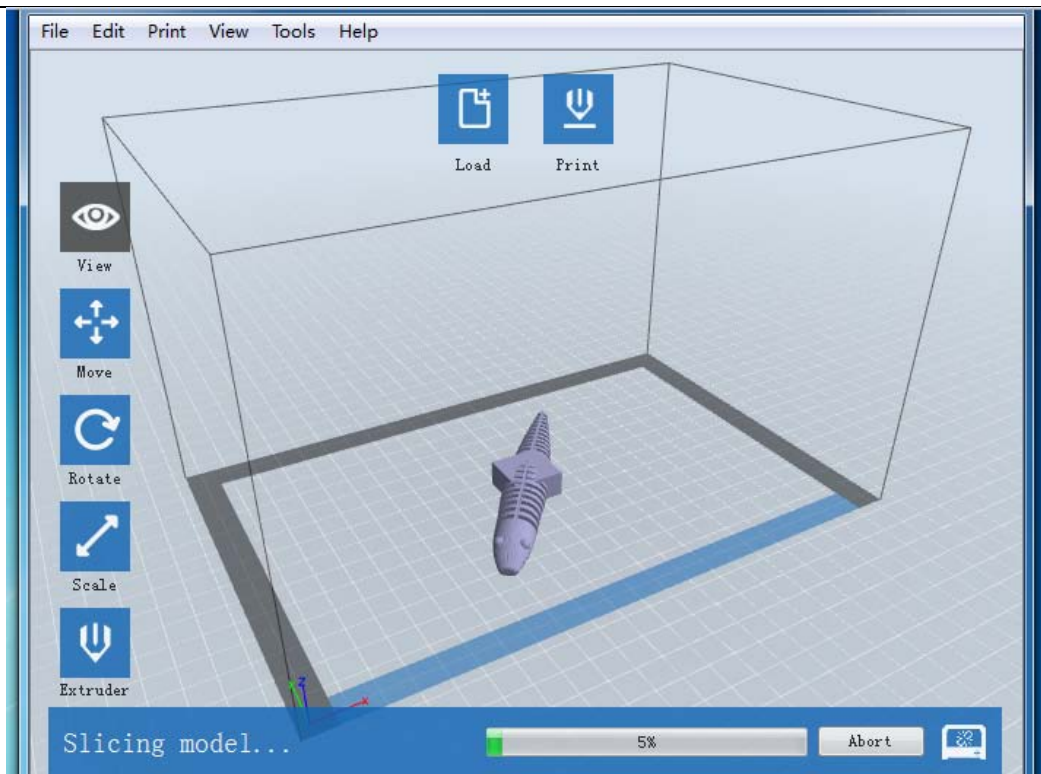
In order to print, you need to slice the model file to generate a gcode file by using the following steps:

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

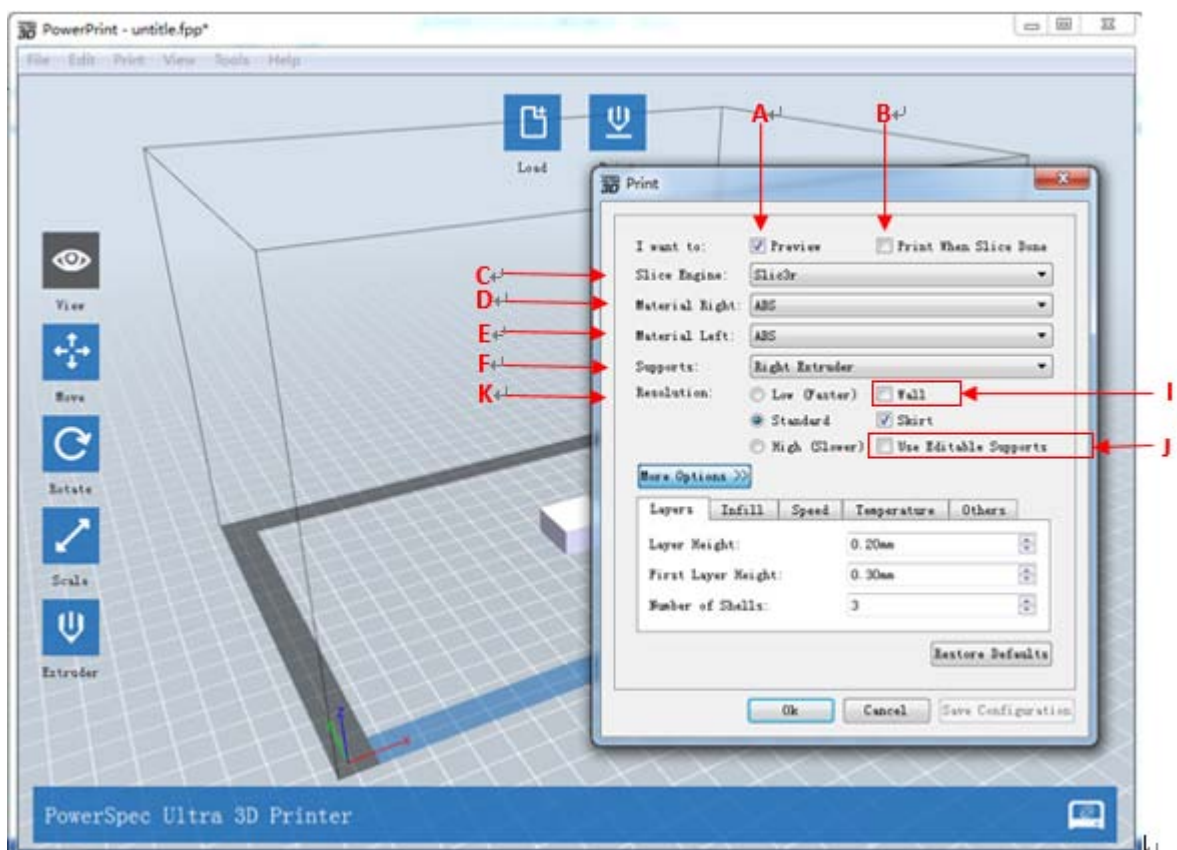


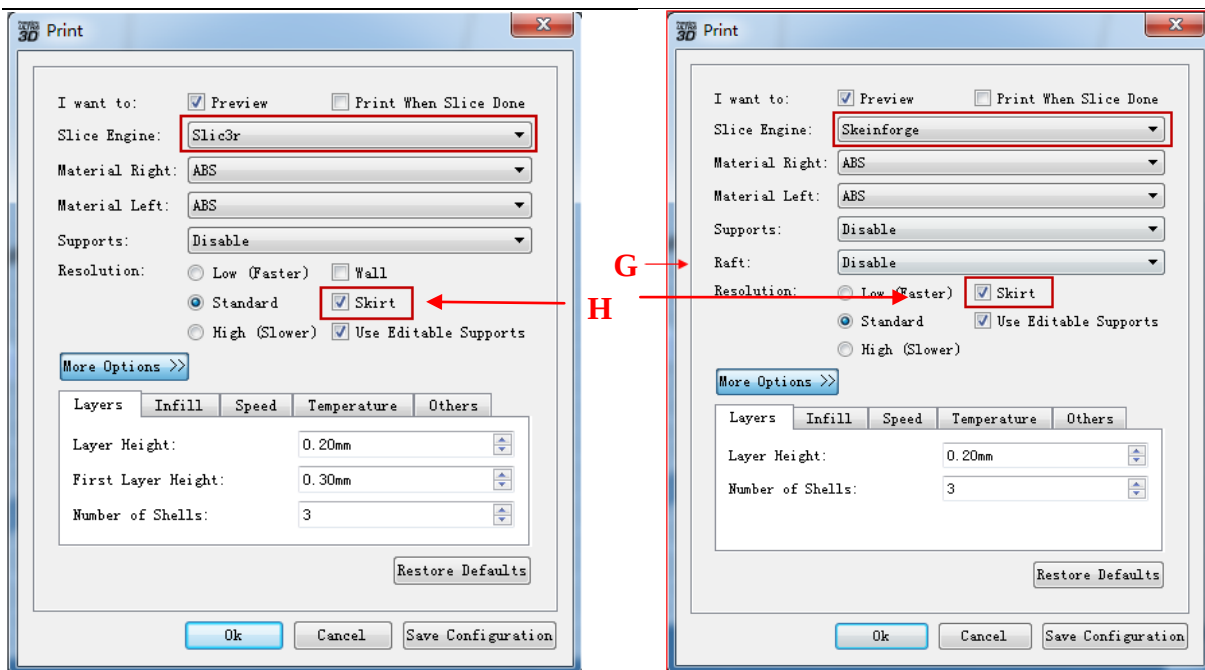
2. Set slicing parameters and click the **Ok** button, and a pop-up dialog box will open for saving the Gcode file.

3. In the **Save as** dialog box, select the path for saving the Gcode file, and then click **Save** button. You can see the progress bar at the bottom of software interface start that shows the completion progress of slicing and uploading Gcode file now. Click the **Abort** button on the right side of the process bar to cancel the slicing.



2.8.3.2 Explanation of Slicing settings





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

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

C. Slice Engine: Three slice engines are provided: Slic3r, Skeinforge and ffslicer. Different slice engines have different configurations. It is recommend to select Slic3r, as it has better performance.

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

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

F. Supports: Choose Left or Right Extruder to print supports when overhang is existing, or disable when not needed.

G. Raft: Choose Left or Right Extruder to print rafts which can helps the model to adhere better to the build plate, or Disable when not needed. (Note: Slic3r has no this function, while Skeinforge has this function with the default extruder used to print the raft. You can choose left or right extruder to print the raft when using ffslicer.)

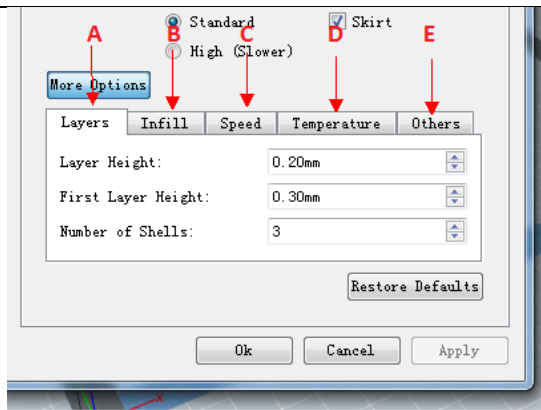
H. Skirt: Select this option to help extruder extrude filament more smoothly. (Note: ffslicer does not have this function.)

I. Wall: Select this option to scrape extruded filament from a non-printing extruder during dual extrusion print. (Note: Skeinforge does not have this function. This function can't be used with Skirt at the same time.)

J. Use Editable Supports: Select this option to disable the built-in supports function in the slice engine and use the support structures generated by Powerprint software. ffslicer has no built-in support functions, so this function is always enabled.

K. Resolution: When printing using the 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 with the right extruder with PLA filament, you have the fourth option, which is Hyper with hyperfine print.

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** is the height of each layer. The lower the value of layer height is, the finer the surface of the model will be .
- b. **First Layer Height** exists when users choose Slic3r as the slice engine, and it affects the adhesion degree between model and platform. The max value is 0.4mm. Normally we recommend you to use the default value.
- c. **Number of Shells** is the quantity of shells of each layer. The max value is 10.

B. Infill

- a. **Fill Density** means fill rate.
- b. **Fill Pattern** is the pattern of filling shape which effects printing duration.

C. Speed

- a. **Print Speed** is the moving speed of the extruder. Generally, the lower the speed is, the better the final print will be. For **ABS printing**, **60** is recommended; for **PLA printing**, **80** is recommended.
- b. **Support Print Speed** is needed to set when choosing Slic3r as the slice engine which can control the moving speed of the extruder when printing the supports.
- c. **Travel Speed** is to control the moving speed of the extruder under non-printing status during work. For **ABS printing**, **80** is recommended; for **PLA printing**, **100** is recommended.

Note: Modify parameters settings to get better prints as different models need different parameters.

- D. Temp.** Set the temperature for left or right extruder and platform. Set **220** to the **extruder temp** for **ABS and PLA printing**. Set **110** to the **platform temp** for **ABS printing** while **50** for **PLA printing**. **Adjust the temperature according to the condition in order to get a good print.**

E. Other Functions

- a. **Turn Fan On** to turn on the back fans to cool the chamber if printing PLA .
- b. **Overhang Threshold** helps to measure whether a support structure is needed or not. If the intersection angle between model surface and horizontal line is within the overhang threshold value, then this surface will need a support.

2.8.2.3 Print Gcode File

After finishing slicing, it will load into the generated Gcode file automatically if you select the **Preview** option when slicing, or you can manually load into it and then go to the **Preview** interface. In the preview interface, there is a vertical scroll bar which shows each layer of the model and you can find the **Estimated Material Right** and **Estimated Print Time** it will take located in the top right corner. Then you can click the **Print** button in the same location, the machine will start to print. Click the **Back** button to exit from the **Preview** mode.

2.8.2.4 Pause or Terminate Printing

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

Note: Please **DO NOT** click the **Pause** button unless necessary because it will affect the overall printing result.

2.9 Printer Operations

2.9.1 Connect / Disconnect

You can connect the 3D printer to Powerprint software via a USB cable or WIFI (if equipped). The machine icon in the bottom right corner of the software interface displays a broken chain pattern means disconnected, while displaying an unbroken one means connected.

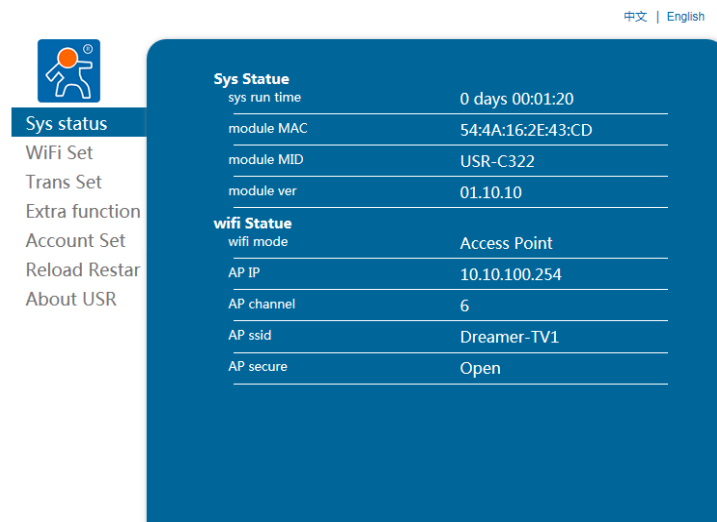
2.9.1.1 Connecting

A. USB cable connection

- Find the USB ports on the right-side of machine and on your computer and connect them.
- Open Powerprint software and turn on the printer.
- Click the **Print -> Connect Machine** menu, then select USB in the **Connection Mode** options and then select the machine you want to connect to in the **Select Machine** option. If you cannot find your machine, click the **Rescan** button to scan your machine and then select it. Finally click the **Connect** button to connect to the printer. If you still cannot find your machine after rescanning you haven't installed the driver software. You will need to install the driver manually, please refer to the Chapter 2.8.3.1. Normally, the driver will be installed automatically along with the software install.

B. WIFI connection

- Via a wireless network
 - Switch on the printer, and verify the WIFI is available by clicking **Tools ->Set up->WIFI->WIFI ON**.
 - Open the wireless network connection in your computer, and choose PS_3D_Ultra_Printer Wireless network. It is the default network of the printer, and no password is needed if you haven't changed it in the settings.
 - Open your browser and enter the IP address **10.10.100.254** in the Address Bar. Then enter **admin** as the account and password . (The above IP address /Account /Password are default if setting haven't been changed)



- Click **WiFi Set** and you can check and set the WIFI Operating Mode, as shown in the picture below:

AP Mode Setup: In the setup section of AP Parameters, you should change the SSID, password, IP and subnet mask. To setup WIFI mode as AP mode, click **Save** and reboot and AP mode should be set successfully.

STA Mode Setup: In the setup section of STA Parameters, you can change the SSID, Internet Password, and DHCP Mode. To setup WIFI mode as STA mode, click **Save** and reboot and STA mode should be successfully.

(Note: If, under the STA Mode, there are incorrect parameter settings or an area beyond the valid range, the WIFI will fail to connect, correct by switching to the AP Mode by using Reset on WIFI interface.)

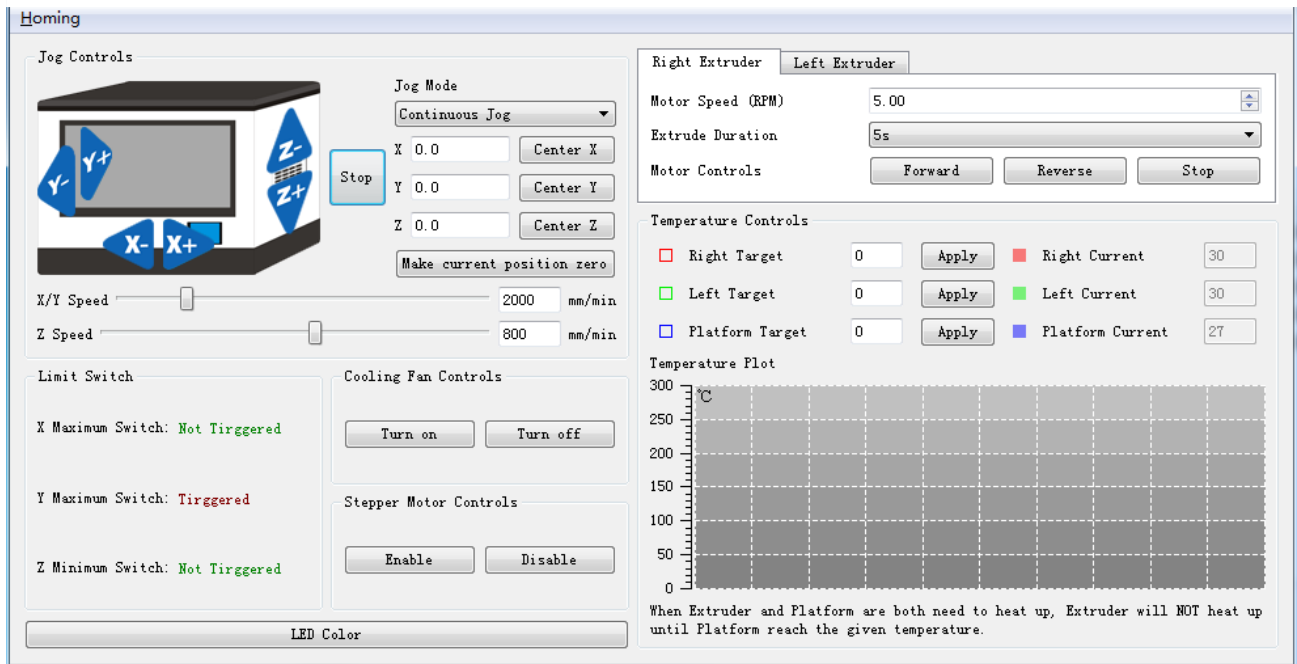
1. Reboot the printer WIFI to re-establish a connection, then connect the printer to the Internet in AP mode on the computer, make sure the Printer and the computer access the same WIFI, then open the PowerPrint and click on Menu—Print—Connect successfully. Choose WIFI as the link mode, input the IP address of the user's Internet (The IP address shown on the screen of the 3D printer) to the IP port below, and then click **Connect**.
2. The printer has connected with PowerPrint. The status bar in the lower right corner should show the temperatures of the nozzle and printing platform.
3. A printing selection dialog will be displayed after clicking **Printing** icon. You should ensure that you choose "ABS/PLA" in the material options. You can click the **More** icon then select **Options** to implement advanced options. Check on Printing after Layering and click **OK**.
4. G-code files could be saved in any location. After saving, the model starts to be cut into slices and will be uploaded to the Printer. Then the printer will turn on pre-heat mode and start to print when the preheat finishes.

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 established, click **Tools->Control Panel** menu to open the control window. The following operations are available in the control panel to control the printer.



A. Jog Controls

a. Jog Mode:

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

c. Stop Button:

Click this button to terminate the current movement.

d. Coordinate frame:

Displays the coordinated positions of the extruder and platform.

e. Make the 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 buttons:

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 the X/Y/Z axes used to control the movement limit positions. The following are the two statuses:

a. Open Status:

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

b. Triggered Status:

If the position of the extruder or platform has reached the maximum point of the X/Y/Z direction, the limit switch will be triggered and its status will be displayed as **Triggered**.

C. Fan Controls

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

D. Stepper Motor Control

To control the stepper motor click the **Open** button to lock the stepper motor to prevent adjusting the position of the extruder and platform manually and the **Close** button will unlock it so manual adjustment of the extruder and platform can be made.

E. LED Color

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

F. Extruder Setting

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

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

G. Temperature Control

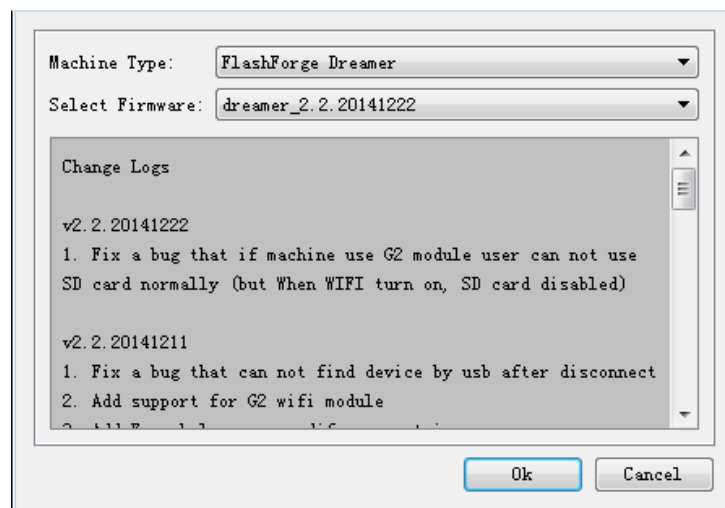
Type in the target temperature in the left blank and click the **Apply** button to heat the extruder or platform. The printer starts to heat the extruder or the platform, and you will see the actual temperature the extruder or platform has reached on the right in a temperature changing table with 3 different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

Note: The printer will heat the platform first to reach the target temperature and then heat extruders when both are needed to be heated.

2.9.3 Firmware Upgrade

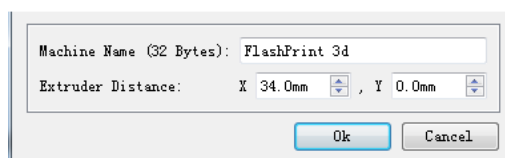
The software will check and download updates automatically when opened, and remind you to upgrade to new version if available. To install firmware:

- A.** Follow the software tips, Click **Tools -> Update Firmware** menu and select **connect** option,
 - B.** Select your machine type and corresponding firmware in the pop 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 needed. The X/Y distance refers to the distance of X or Y directions between the two extruders.



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.

2.9.6 Driver Installation

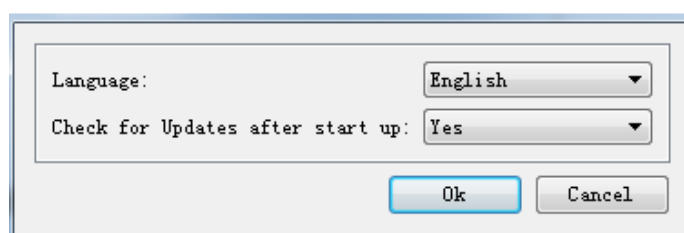
A. Open the root directory of the software

B. Open the driver folder located 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 and for 32bit edition, please install dpinst_x86.exe file.

2.10 Other Functions

2.10.1 Preferences

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



A. Language

Software supports English, Japanese, Simplified Chinese, and Traditional Chinese.

B. Check for Update after Startup

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

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 a new version is available, it will remind you to download and install the new one. Please refer to the Chapter 1.1~1.2 to install the software.

2.10.4 Software Information

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

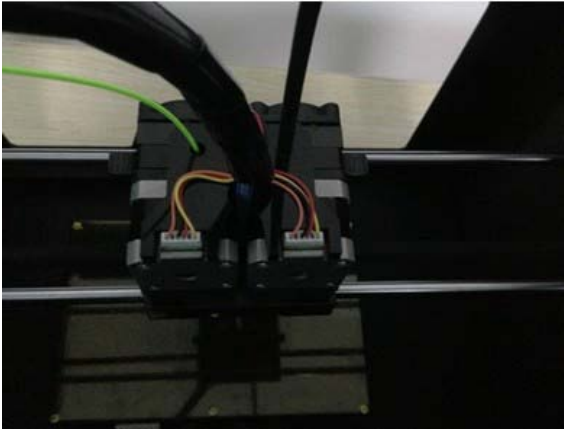
5 Filament

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

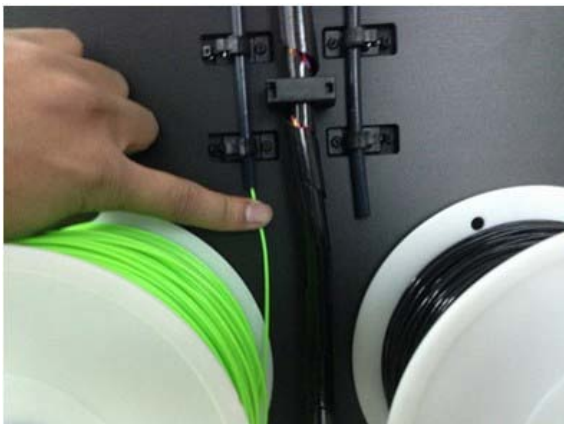
After inserting the filament into the feeding hole, do not force it further until the extruder temperature reaches 200°C or more. Once the machine reaches this point, you will feel the filament being pulled into the extruder head.

5.1 Filament Install

First, remove the filament guide tube from the extruder head.

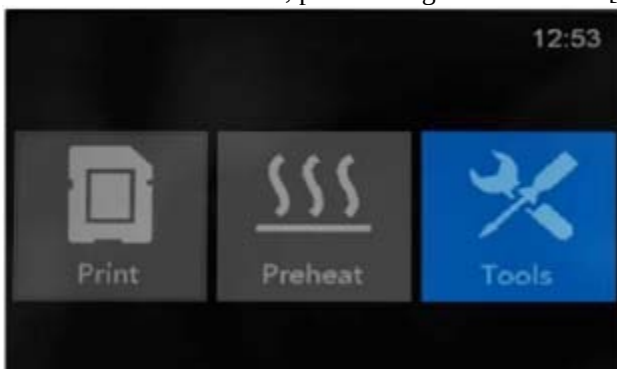


When you have removed the guide tube you can remove the filament that is inside the guide tube. To avoid any blockages during printing, please ensure that the two threads are loaded from the middle. There are two wire trays, one runs clockwise and the other one runs counter - clockwise, as shown below:

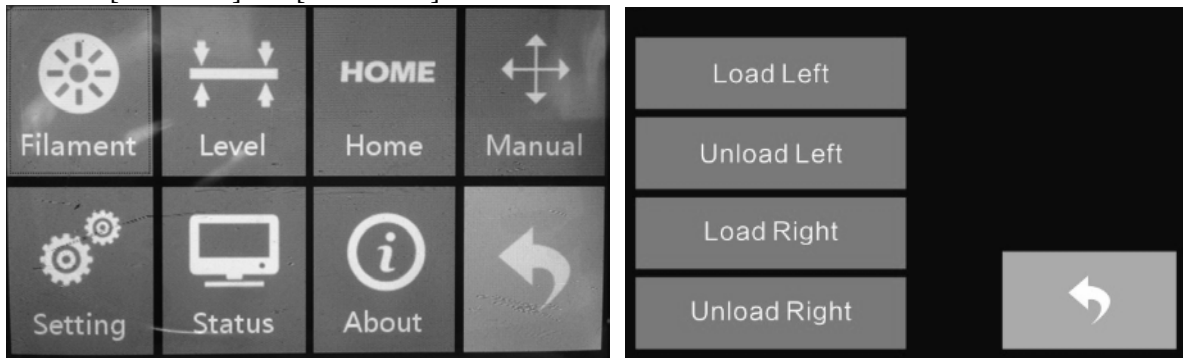


5.2 Load Filament

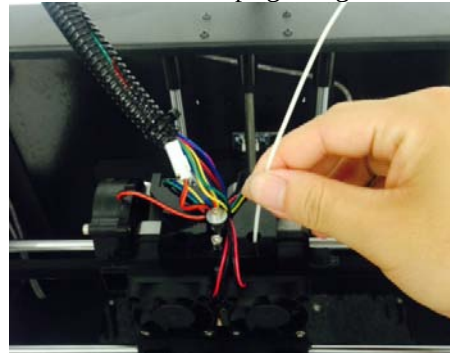
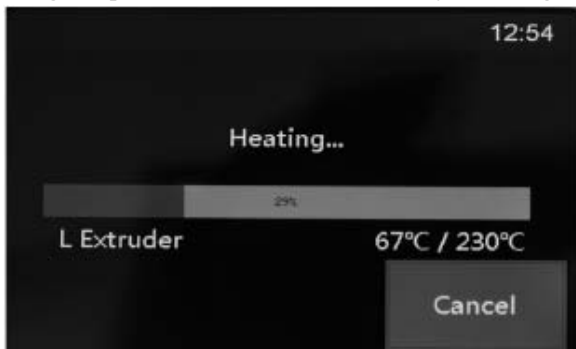
1. From the main menu, press the right icon labeled [Tools].



2. Select [Filament] and [Load Left].



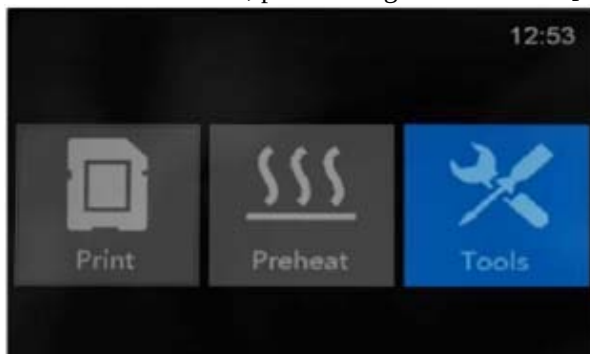
3. Wait for the extruder to heat up to the operating temperature. The extruder will alert you once it is at the operating temperature. Load the filament by inserting it into the extruder at an upright angle.



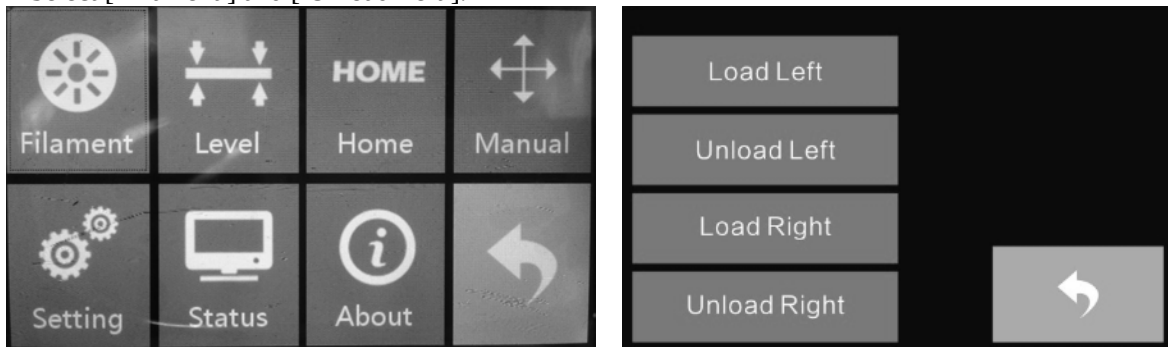
4. Filament will start to extrude out of the nozzle. Continue loading to ensure that the filament is extruding in a straight line.

5.3 Unload Filament

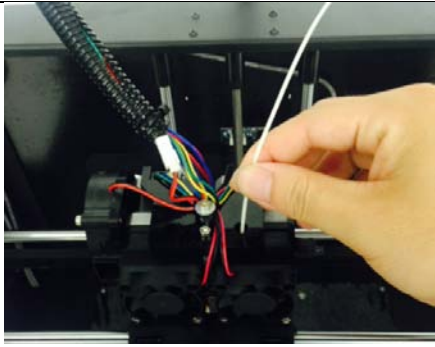
1. From the main menu, press the right icon labeled [Tools].



2. Select [Filament] and [Unload Left].



3. Wait for the extruder to heat up to the operating temperature. The extruder will alert you once it is at the operating temperature. Unload the filament by gently guiding it out of the extruder.



7 Initial Print

CAUTION

 The printer build plate has been leveled perfectly before leaving factory. However, it may move during shipping so please verify the build plate is level before starting a print. See instructions below.

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

1. Your dual extruder heads should be bolted in place.
2. Your filament guide tubes are connected.
3. Your filament is mounted on the spool holders.

Once all these have been completed, you can plug in your power supply. If everything is ready, then turn on the power switch on the back of your Printer.

Every printer will be leveled before being shipped, but the build platform could move during delivery, so it is important to level the platform before you begin to print.

The 3D printer utilizes a three-point leveling system for its build plate. At the bottom of the build plate, there is one spring-loaded screw in the front and two in the back. Tightening the screws will create a bigger gap between the build plate and the nozzle while loosening them will create lesser gap.

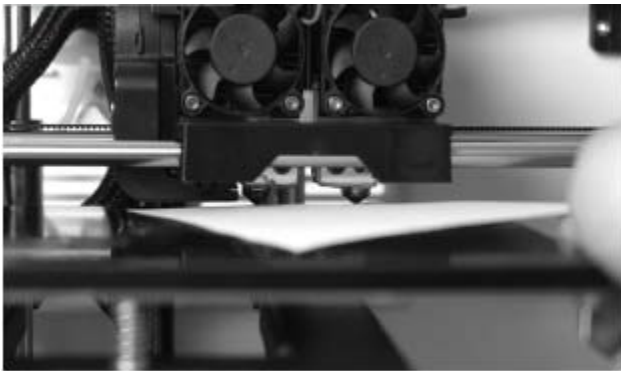
From the main menu, select [Tools] and [Level]. The extruder and the build plate will begin to move to the starting position.



Cut a piece of paper and put it between the plate and nozzle. Once the extruder and build plate stops moving, slide the piece of paper continuously back and forth between the nozzle and the build plate. And simultaneously adjust the screw just enough so that the paper causes a slight friction.



Press [NEXT] and wait for the extruder to move to the second position. Slide the paper back and forth again, and adjust the screws to create the same amount of friction as in the previous step.



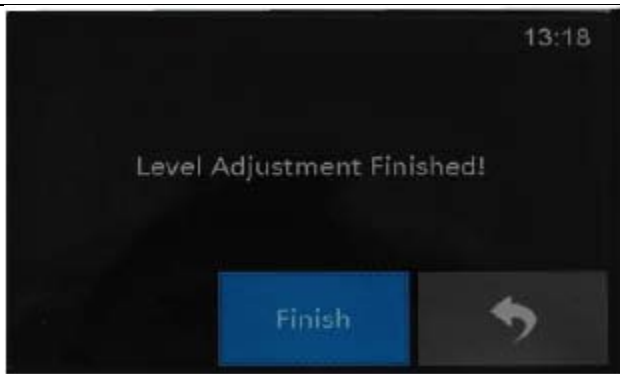
Press [NEXT] again and repeat the same leveling technique.



Press [NEXT]. The nozzle will move to the center of the build plate. Slide the paper through to make sure there is a slight friction. Slowly adjust all the screws by the same amount if there is no friction or too much friction.



Press [FINISH] once you have finished this.



Single-Extrusion Print

1. Open Powerprint by double-clicking on the icon.
2. Click on [Load] and choose a .stl file from hard drive.
3. The object will then be shown on the screen.
4. Click on the object and then click on [Extruder], select [Use left extruder]. Left extruder will be used for illustration purposes.
5. Now the 3D model is ready to be created. Proceed to part C of this chapter (page 47) to select your connectivity method for printing.

Dual-Extrusion Print

1. Open Powerprint by double-clicking on the icon.
2. Press [Load] and choose a .stl file from hard drive.
3. The object will then be shown on the screen.
4. Press [Load] and choose another (or the same) file from your hard drive.
5. Click on the object and then click [Extruder], select [Use left extruder].
NOTE: if the object is displayed as green it indicates that it will be printed with the left extruder.
6. Now the 3D model is ready to be created. Proceed to part C to select your connectivity method for printing.

Methods of printing

Printing from USB

1. Connect the 3D printer to your computer using the included USB 2.0 cable.
2. Turn on the 3D printer. Make sure the build plate is leveled and filament is loaded to the left extruder. Please refer to page 22 for leveling the build plate and pages 19 and 21 for loading and unloading filament.
3. Select [Print] from menu bar, then select [Connect].
4. Click on [Rescan], then [Connect].
5. Now the printer is connected with Powerprint. A status box at the lower right corner will show the temperature of both extruders and the platform.
6. Press [Print] icon, and a printing options screen will appear. Make sure that "ABS" is selected under Material Left. Advanced settings can be set under [More Options] icon. Check the box "Print When Slice Done" and click [OK].
7. Save the file at any location, and the object will start slicing
8. After the object is done slicing, it will automatically upload the gcode to the 3D printer.
9. After the gcode is done uploading, the printer will go into a preheat phase . The 3D printer will begin printing once it has finished preheating.

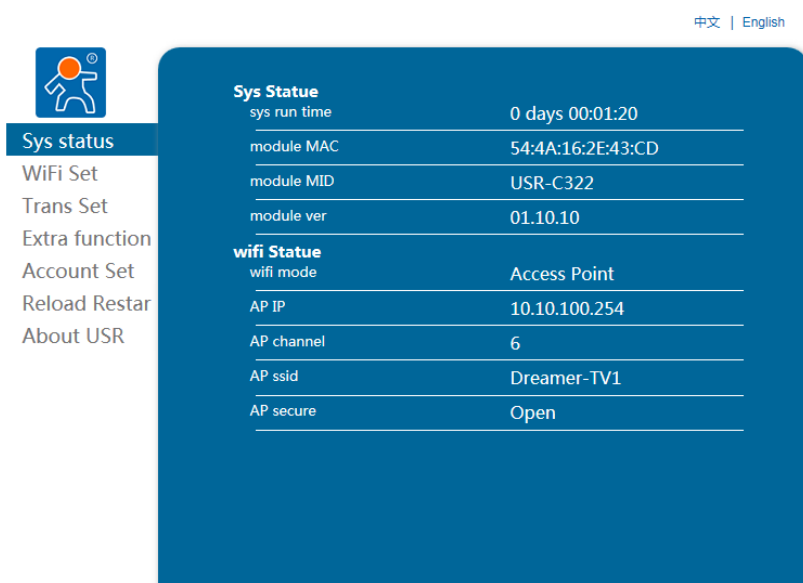
Printing from SD Card

1. Press [Print], and a printing options screen will show up.
2. Make sure that the "Material Left" is selected as "ABS." Advanced settings can be set under [More Options] icon.
3. Click [OK], and save the gcode file in the SD card.

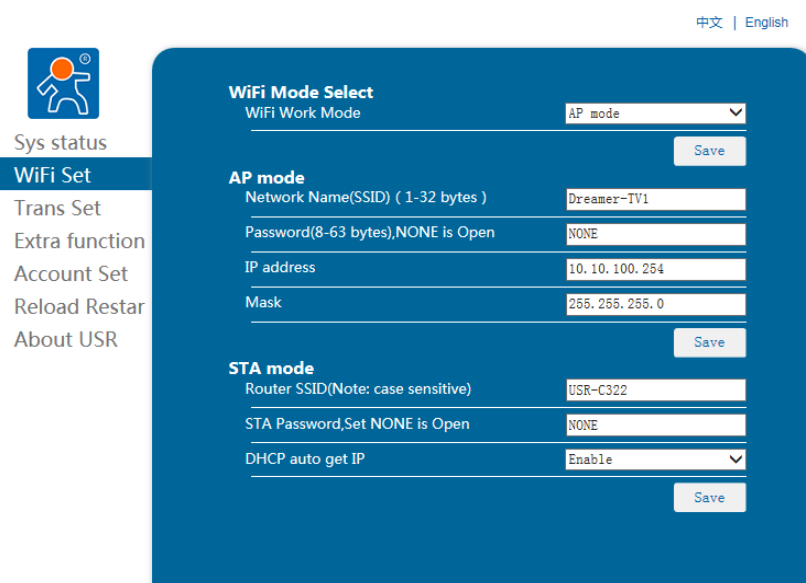
4. Powerprint will begin slicing the 3D model.
5. After the object is done slicing, take the SD card from the computer. Insert it into the SD card slot on the 3D printer.
6. Turn on the 3D printer. Make sure the build plate is leveled and that filament is loaded in the left extruder.
7. Press [Print] and then press the middle SD Card icon.
8. A list of file(s) will show up, press the file that you would like to print, then press [Yes].
9. The printer will now enter the preheating phase and will start printing once it finishes preheating

Printing from Wi-Fi

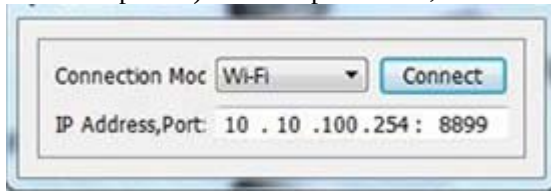
1. Turn on 3D printer. Make sure the build plate is leveled and that filament is loaded in the left extruder.
2. Turn on Wi-Fi on the 3D printer. To do this, press [Tools], [Setting], [WIFI], and [WIFI ON].
A connection called “PS_3D_Ultra_Printer” can be found on the list of available networks. Connect to this network.
3. After connecting the printer with the computer, you should access to “10. 10. 100. 254” through IE browser, then login in the website with the account and password (Account: admin Password: admin). You will find the following wireless settings panel below.



4. Click on the WIFI Set, you can check and set the WIFI Operating Mode, as shown in the picture below:
AP Mode Setting: In the setup section of AP Parameters, you should change the SSID, password, IP and subnet mask. Then set up WIFI mode as AP mode, click on Save and reboot and will be set successfully.



5. **Referring to the example above for reference:** Reboot the WIFI SSID of Dreamer to establish connection, then connect the Dreamer-TV1 to the Internet under the AP mode of the computer, make sure that Dreamer and the computer access the same WIFI, then open the FlashPrint and click on Menu—Print—Connect successfully. Choose WIFI as the link mode, input the IP address of user Internet (The IP address will be shown on the screen of the 3D printer) to the IP port below, and then click **ON** to connect.



7. The 3D printer should be connected with Powerprint. A status box at the lower right corner will show the temperature of both extruders and the Platform.

8. In Powerprint, click [Print]. A printing options screen will appear. Make sure that “ABS” is selected under “Material Left.” Advanced settings can be set by clicking on [More Options].

9. Check the box of “Print When Slice Done,” and select [OK].

10. Save the file at any location, and the object will start slicing.

11. After the object is done slicing, it will automatically upload the gcode to the 3D printer.

12. After the gcode is done uploading, the printer will go into a preheat phase. The 3D printer will start printing once preheating is finished.

NOTE: Printing from SD card is unavailable when WIFI is ON.