

WEEDO

# 3D PRINTER

## User's Guide



|  |    |
|--|----|
| 1.SAFETY WARNINGS AND GUIDELINES ..... | 1  |
| 2.CUSTOMER SERVICE .....               | 2  |
| 3.INTRODUCTION .....                   | 2  |
| 4.FEATURES .....                       | 2  |
| 5.FUNCTION INTRODUCTION .....          | 3  |
| 5.1 COMPATIBLE CONSUMABLES .....       | 4  |
| 5.2 IMAGE PREVIEW .....                | 4  |
| 5.3 INTELLIGENT FUNCTION .....         | 4  |
| 6.PACKAGE CONTENTS .....               | 5  |
| 7.PRODUCT OVERVIEW .....               | 6  |
| 8.INTERFACE SYSTEM .....               | 6  |
| 8.1 INTERFACE OPERATION .....          | 6  |
| 8.2 MAIN INTERFACE .....               | 7  |
| 8.3 SELECT PRINT FILE .....            | 7  |
| 8.4 PRINTING INTERFACE .....           | 8  |
| 8.5 SETTING INTERFACE .....            | 11 |
| 8.5.1 LANGUAGE .....                   | 12 |
| 8.5.2 MACHINE INFO .....               | 12 |
| 8.5.3 UPDATE .....                     | 12 |
| 8.5.4 POST .....                       | 13 |
| 8.5.5 RUNOUT SENSOR .....              | 13 |
| 8.5.6 PARAMETERS .....                 | 13 |
| 8.6 MAINTENANCE INTERFACE .....        | 14 |
| 8.6.1 FILAMENT FEED .....              | 15 |
| 8.6.2 FILAMENT RETRACT .....           | 15 |
| 8.6.3 PREHEAT .....                    | 15 |
| 8.6.4 JOG MODE .....                   | 16 |

|                                     |    |
|-------------------------------------|----|
| 8.6.5 LEVEL BED .....               | 16 |
| 8.6.6 Z-OFFSET .....                | 17 |
| 8.6.7 PROXIMITY HEIGHT .....        | 17 |
| 8.7 HELP INTERFACE .....            | 18 |
| 9.WIIBUILDER SLICING SOFTWARE ..... | 18 |
| 9.1 INSTALLATION .....              | 18 |
| 9.2 WIIBUILDER SETUP .....          | 21 |
| 9.3 SPEED TAB .....                 | 25 |
| 9.4 INFILL TAB .....                | 25 |
| 9.5 SUPPORT TAB .....               | 27 |
| 9.6 BUILD PLATE ADHESION TAB .....  | 28 |
| 9.7 RETRACTION TAB .....            | 29 |
| 9.8 MATERIAL TAB .....              | 29 |
| 9.9 TRAVEL TAB .....                | 30 |
| 9.10 MACHINE TAB .....              | 30 |
| 9.11 LINE WIDTH TAB .....           | 31 |
| 9.12 DUAL EXTRUSION TAB .....       | 32 |
| 9.13 WARPING PRECAUTION TAB .....   | 32 |
| 9.14 SEAM TAB .....                 | 33 |
| 9.15 OTHERS TAB .....               | 34 |



Please read this entire manual before using this device, paying extra attention to these safety warnings and guidelines. Please keep this manual in a safe place for future reference.

- Do not reach inside the printer during operation.
- Always allow the printer and extruded filament to cool before reaching inside.
- Take care to avoid touching hot parts, including heat blocks, extruder nozzle, build platform, and extruded filament.
- Do not wear gloves when operating or repairing to avoid entanglement.
- Keep the printer and all accessories out of reach of children.
- Do not force or tear anything when unpacking and during setup. This may cause damage to the printer and/or its accessories.
- Ensure that the printer is turned off and unplugged from its power source before making repairs or performing service.
- Do not install this device on an unstable surface where it could fall and cause either personal injury or damage to the device and/or other equipment.
- Do not subject the product to extreme force, shock, or fluctuations in temperature or humidity.
- This device is intended for indoor use only.
- Do not expose this device to water or moisture of any kind. Do not place drinks or other containers with moisture on or near the device. If moisture does get in or on the device, immediately unplug it from the power outlet and allow it to fully dry before re-applying power.
- Do not touch the device, the power cord, or any other connected cables with wet hands.
- Prior to operation, check the unit and power cord for physical damage. Do not use if physical damage has occurred.



- Before plugging the unit into a power outlet, ensure that the outlet provides the same type and level of power required by the device.
- This device uses a grounded power cord and requires a ground connection for safe operation. Ensure that the power source has a proper ground connection. Do not modify the plug or use a "cheater" plug to bypass the ground connection.
- Unplug this device from the power source when not in use.
- Take care to prevent damage to the power cord. Do not allow it to become crimped, pinched, walked on, or become tangled with other cords. Ensure that the power cord does not present a tripping hazard.
- Never unplug the unit by pulling on the power cord. Always grasp the connector head or adapter body.
- Use only in a well-ventilated area. Do not use in close, confined spaces.
- Take care when using a scraper to remove a model. Never direct the scraper towards your fingers or body.
- The printer is designed to operate with an ambient temperature ranging from +41°F (+5°C) to +104°F (+40°C). Printing outside these limits may result in low quality prints.
- Use eye protection when cleaning or sanding the printed models to avoid getting small particles in your eyes.

## 2.CUSTOMER SERVICE

WEEDO

If you have any problem with your order, please give us an opportunity to make it right. You can contact a WEEDO Customer Service representative through the Live Chat link on our website [www.weedo.ltd](http://www.weedo.ltd) or via email at [support@weedo3d.com](mailto:support@weedo3d.com). You can check the products wiki website for more information: <http://www.weedo3dprinter.com>

## 3.INTRODUCTION

WEEDO

This printer uses the FDM method of printing. It features a metal frame, heated build platform. It supports auto leveling, with a 4.3-inch touch screen, a removable and a heated build plate.

## 4.SPECIFICATIONS

WEEDO

|                             |   |
|-----------------------------|---|
|                             |   |
| Maximum Printing Area       | 11.8x11.8x15.7" (300 x 300 x 400 mm)    |
| Filament Diameter           | 1.75mm                                  |
| Nozzle Diameter             | 0.4mm                                   |
| Layer Thickness             | 0.1~0.4mm                               |
| Maximum nozzle temperature  | 295°C                                   |
| Printing Speed              | 30 ~ 150 mm/s                           |
| Touch Screen                | 4.3"                                    |
| Positioning Accuracy        | XY Axis: 0.011mm Z Axis: 0.0025mm       |
| Supported Filament Types    | PLA, PLA +, ABS,TPU, PC, PETG, Nylon... |
| Supported Software          | WiiBuilder/Cura                         |
| Supported File Formats      | .STL, .g-code, .OBJ                     |
| Print Interface             | SD card,USB                             |
| Supported Operating Systems | Microsoft® Windows®                     |
| Maximum Power               | 360w                                    |
| AC Adapter Input Power      | 100 ~ 240 VAC, 50/60 Hz                 |
| Dimensions                  | 19.1"x 24.4"x 26" (508 x 575 x 655 mm)  |

## 5.1 Compatible consumables



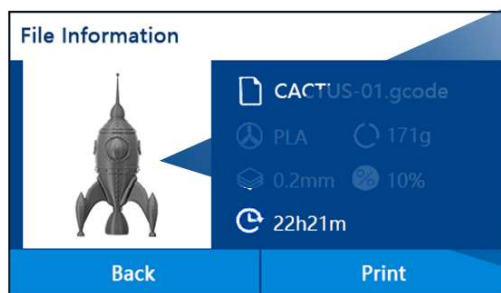
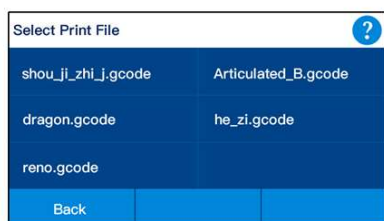
Heat Break

| Type             | Stainless steel | Teflon   | Titanium pipe |
|------------------|-----------------|----------|---------------|
| High temperature | Smoothly        | <240°C   | Smoothly      |
| Low temperature  | Easily blocked  | Smoothly | Smoothly      |

Nozzle

| Type          | Brass nozzle   | Die steel nozzle |
|---------------|----------------|------------------|
| Hardness      | Bottom         | Tall             |
| Ordinary      | Smoothly       | Smoothly         |
| Hard material | Not applicable | Smoothly         |
| Carbon fiber  | Not applicable | Smoothly         |

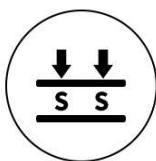
## 5.2 Image preview



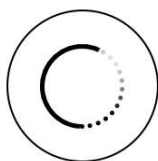
## 5.3 Intelligent function



Post



Automatic leveling



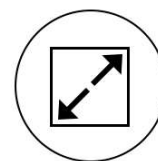
Runout Sensor



Silent drive



Error Diagnosis



Power Loss Recovery



## 6.PACKAGE CONTENTS

WEEDO

Please take an inventory of the package contents to ensure you have all the items listed below. If anything is missing or damaged, please contact WEEDO Customer Service for a replacement.



1x frame base



1x Z/X Gantry Assembly



1x 200g Filament



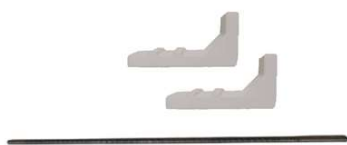
1x USB cable



1x Power Cord



1x Pilers  
1x Glue stick  
1x Metal Scraper  
1x Teflon connector



2x 3D Printed Positioning Block  
1x Lead screw with Plastic sleeve



No.1 Bag:  
1x 4.0mm L Wrench  
4x M5\*25 Screw



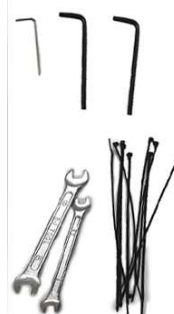
No.2 Bag:  
2x T-shaped  
Metal sheet  
8x M4\*8Screw  
8x T-Nut  
1x 2.5 mm L  
Wrench



No.3 Bag:  
1x Holder Base  
1x Filament Roll  
2x M4\*8 Screw  
2x T-Nut



No.4 Bag:  
1x TF card、  
Reader  
1x Y end stop  
sensor  
1x Y end stop  
cable  
1x thermistor



No.5 Bag:  
10x Cable ties  
1x 2mm L wrench  
1x 1.5mm L wrench  
1x 3mm L wrench  
1x 5.5-7 Wrench  
1x 8-10 Wrechr



- ① TF slot
- ② Data cable socket
- ③ Heating platform

- ④ Spray nozzle assembly
- ⑤ Extruder
- ⑥ Wire rack

- ⑦ Power socket
- ⑧ Power switch
- ⑨ Touch Screen

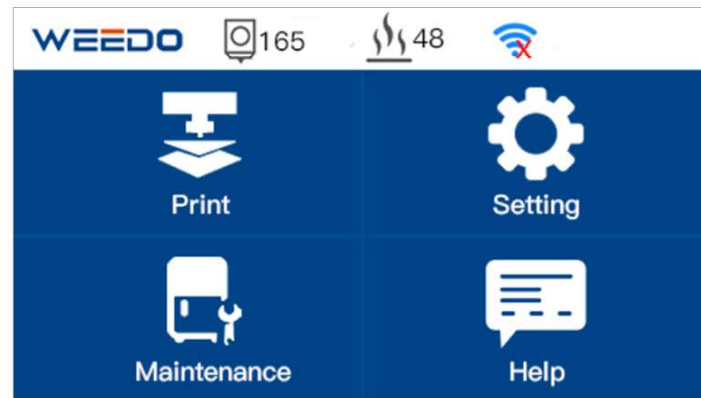
## 8.INTERFACE SYSTEM

### 8.1 INTERFACE OPERATION

Touch the printer display with your Fingertips.

- Enter into the Main interface after Post.
- There is a question mark icon in the upper right corner of each interface, this is the help button, you can touch the button, it will show you the functions of icons in the current interface.
- In the Main interface there are four icons, you can touch different icons to get into sub-interface.
- You can click the Back Button to return to the previous interface.
- You can click the Next Button to go to the next interface.

## 8.2 MAIN INTERFACE

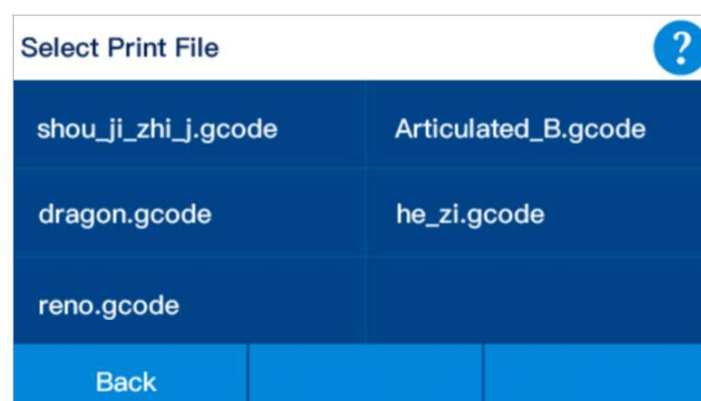


Pic1

- Status Bar: Displays the printer information screen, include: Nozzle Temp, Bed temp.
- Print: Click Print Button and get into Printing File interface, display a list of TF Card , choose the file position you want to print.
- Setting: Click the Setting Button get into sub-interface. There are Language Setting, Machine Info,Runout Sensor, Post, Power Save Mode, Update Parameters Buttons.
- Maintenance: Click the Maintenance get into sub-interface. There are Filament Feed, Filament Retract, Preheat, Jog Mode, Level Bed, Z Offset, Proximity Height, Nozzle Offset Motor Off Buttons.
- Help: Error Diagnosis, WIKI and Contact Us in Help Interface.

## 8.3 SELECT PRINT FILE

- Click TF Card and get into TF Print File Interface, choose the Gcode File in the lists.



Pic2



- Return to the Main Interface by clicking Back icon. Click Next icon to see other files
- Click the file you selected, you will see the file information in the (Pic3), the picture of the file, File name, filament type, layer height, infill, printing time. You can select 'print' .



Pic3

- Click the Back Icon to get into the previous interface(Pic2)

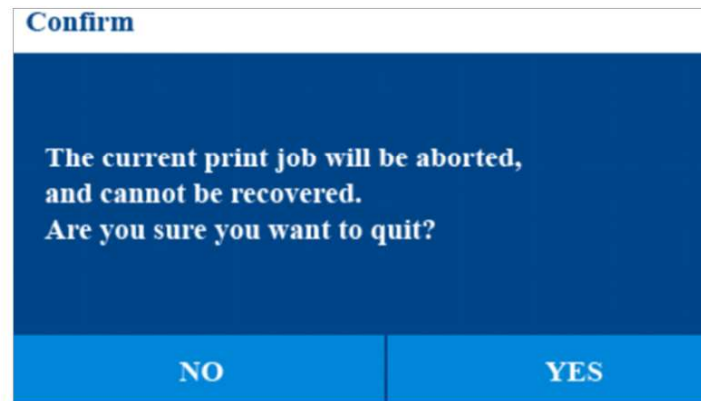
### 8.4 PRINTING INTERFACE



Pic3

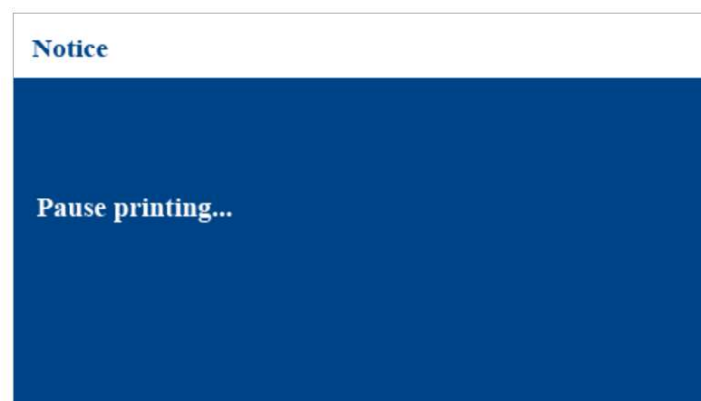
- It shows File Name, Printing Speed, Printing Elapsed Time, Printing Remain Time, Printing Percent. It can Quit, Pause, and Set the Print process. File Name shows the currently printed file name, Speed shows the currently print speed, Nozzle Temp shows the currently printing nozzle temperature, Bed temperature shows the currently bed temperature. Elapsed time shows the time you have printed. Remain time shows the printing time left, remaining time is an estimated time, at the beginning of printing, the estimated time will be less accurate. As the printing progresses, the remaining time after calculation will become more accurate. Print progress bar shows the progress percentage of printing.

- Click Quit and cancel the current print process. Click Quit will pop up a prompt window (Pic4) for operation confirmation to prevent accidental touch to cancel printing.



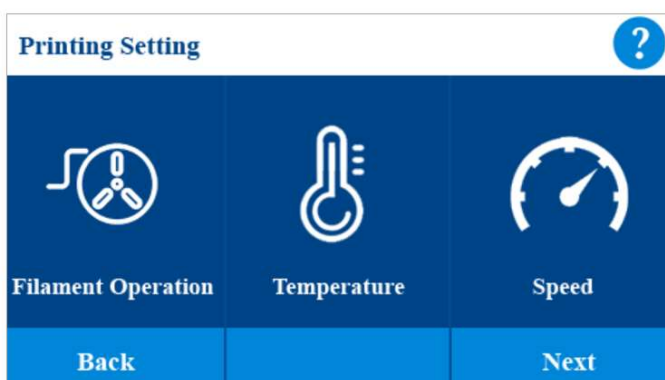
Pic4

- Click Pause and it will pop up a prompt window (Pic5) and then pause the current print process. After pausing the print process, Filament operation can be used.

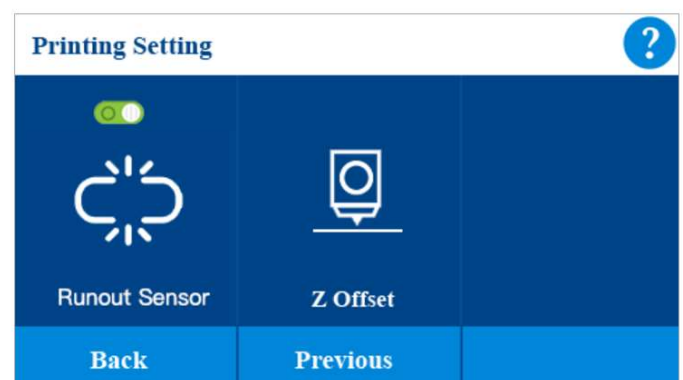


Pic5

- Click the Setting button, printing parameter (Filament Operation, Printing Temperature, Printing Speed, Runout Sensor and Z offset) can be reset. (Pic6 Pic7).

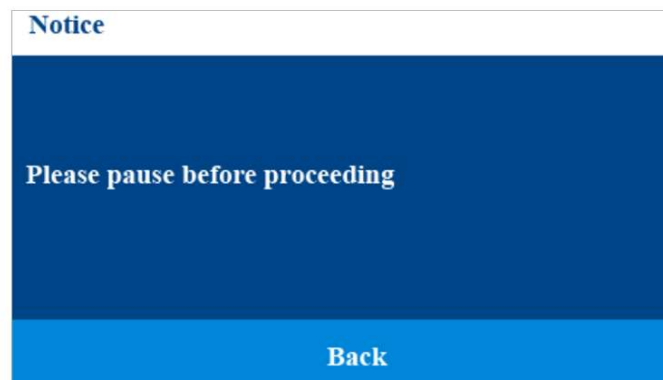


Pic6



Pic7

- Click the Filament Operation button, if printing is in progress, this function cannot be used, and it can only be used after printing has been paused (Pic8).



Pic8

- Pause the printing process and then click the Filament Operation button(Pic9 Pic10). When your filaments are running out and the model still needs a lot of filaments to print, you can pause and replace the supplies. When you print some models, the model you want different colors, you can also pause and then perform the replacement of filaments

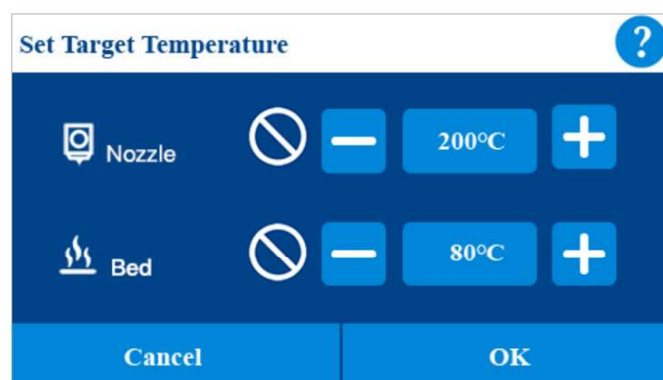


Pic9



Pic10

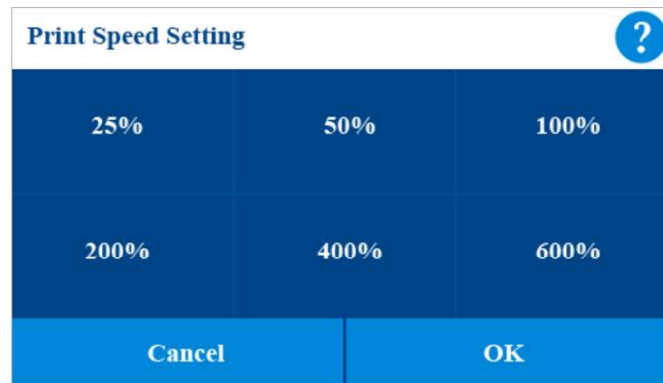
- Click the Temperature button to change the current printing temperature (Pic11). In the printing progress the printing temperature and bed temperature can be reset.



Pic11

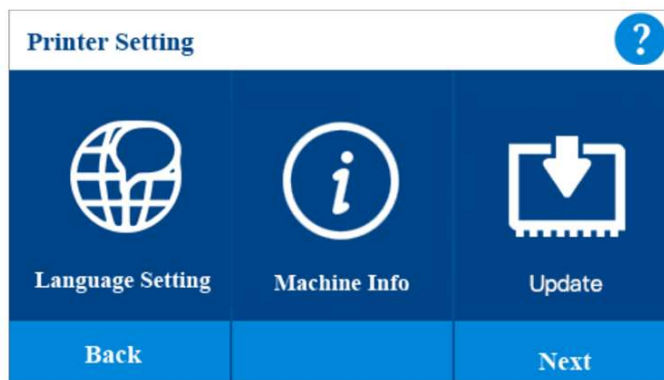


- Click the Speed button to change the current printing speed (Pic12). In the printing progress the printing speed can be reset. Notice: if the speed is set to more than 2 times, the printing effect on the model surface will become worse. If you want to print faster, set the speed in the slicing software.

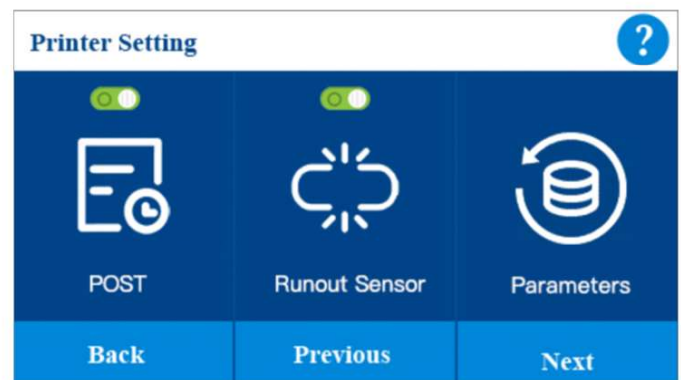


Pic12

## 8.5 SETTING INTERFACE



Pic12



Pic114

- Language Setting: select different language.
- Machine Info: check the machine information, check firmware version, UI version.
- Update: update the latest firmware for the printer..
- Post: POST function ON and OFF.
- Runout Sensor: Runout Sensor function ON and OFF
- Parameters: check parameters, Save firmware parameters to TF card, Load firmware parameters from TF card to printer.

### 8.5.1 LANGUAGE

Select the language you want to show on the interface.



Pic15



Pic16

### 8.5.2 MACHINE INFO



Pic17

- Device Name: Printer type.
- Printed Time: The time print has printed, the first time you get the print the printed time may not show zero, do not worry, all of our machine has tested before leaving the factory.
- Firmware Version: the firmware version of the printer, the firmware version will change after the new firmware is updated.
- UI Version: the UI version of the printer, the UI version will change after the new UI is updated.

### 8.5.3 UPDATE

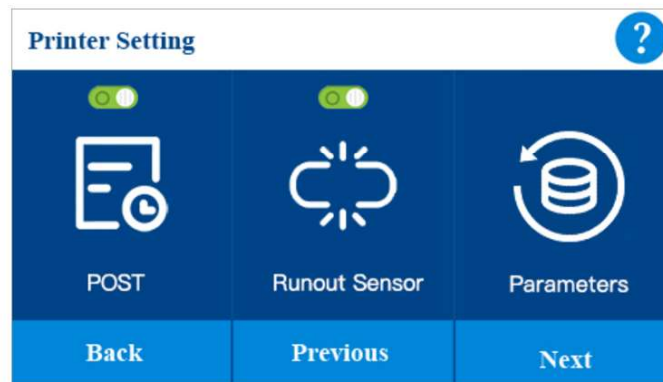
If there is a new version of firmware published form WEEDO website or wiki website, you can download it to TF card. The file type is: **wmf** and name: **flash**.



Click update it will update firmware automatically.

### 8.5.4 POST

POST: printer will self-testing when the printer turns on. It can turn on or turn off.



Pic18

### 8.5.5 RUNOUT SENSOR

Runout Sensor: printer will paused if the filament runout. It can turn on or turn off.



Pic19

### 8.5.6 PARAMETERS

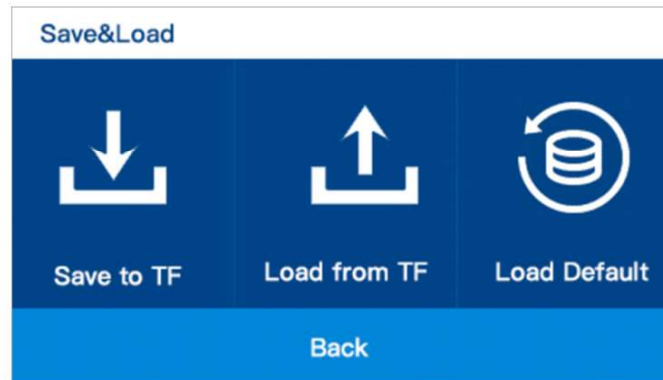
This interface shows the Firmware parameters of the printer, Home Offset, Z-Probe Offset.



Pic20

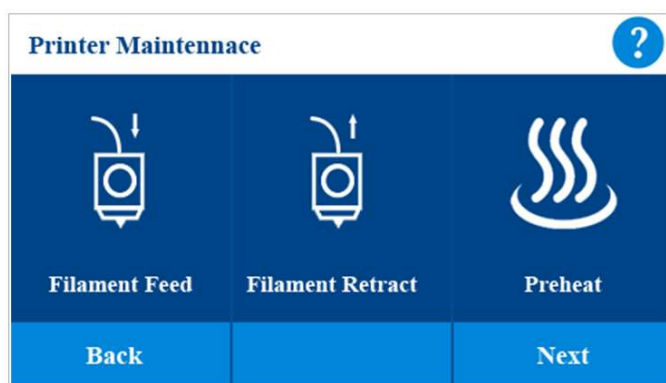


If you connect to the computer and do some parameters testing, please save the parameters to TF card, and Load from TF card for the originally parameters if your changed parameters gets wrong printing.(Pic21)

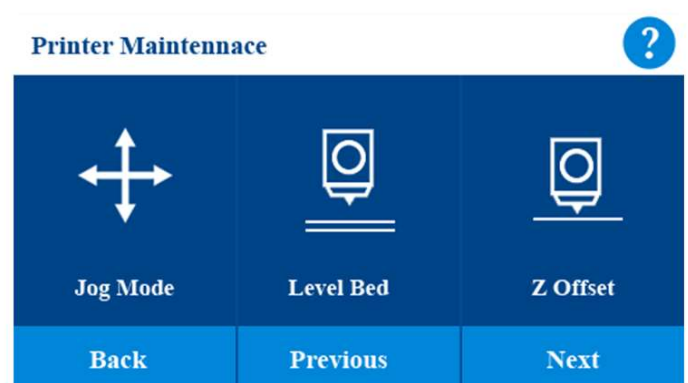


Pic21

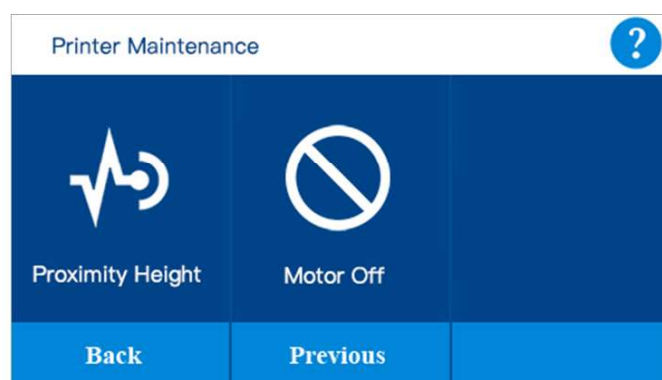
## 8.6 MAINTENANCE INTERFACE



Pic22



Pic23



Pic24

- Filament Feed: Feed the filament to left and right nozzle.
- Filament Retract: Retract the filament of left and right nozzle.
- Preheat: Set the temperature and preheat the nozzle and bed
- Jog Mode: Move X Y Z axis and X Y Z go to home

- Level Bed: Adjust the flatness of the platform
- Z offset: Set the gap between nozzle and platform
- Proximity Height: Set the height of Proximity Switch
- Motor Off: Unlock all the motor
- Click the Back icon to return to the main interface, Click the Previous icon to return to the precious interface, Click the Next icon to enter the next interface.

### 8.6.1 FILAMENT FEED

Click Filament Feed icon, get into filament feed interface. Click the Back icon to return to the previous interface.



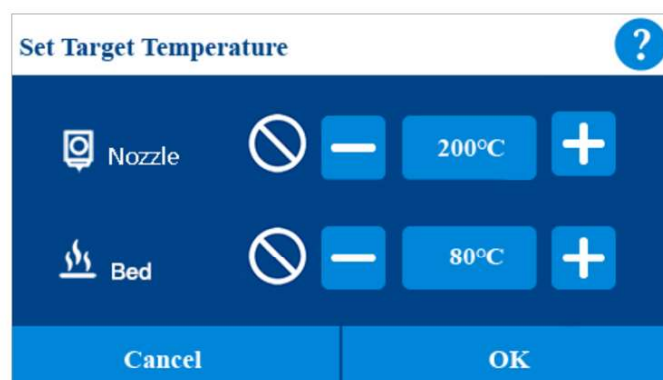
Pic25

### 8.6.2 FILAMENT RETRACT

Click Filament Retract icon, get into filament feed interface (Pic25). Click the Back icon to return to the previous interface.

### 8.6.3 PREHEAT

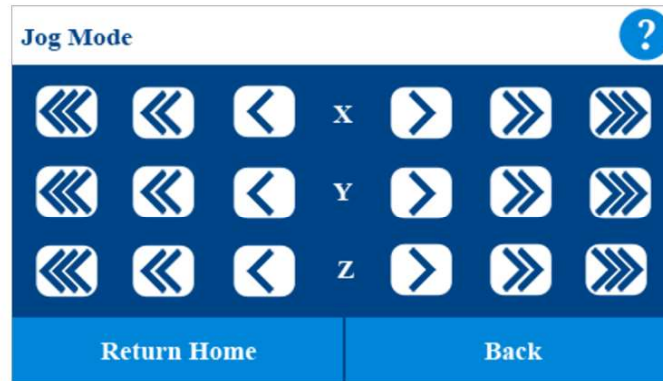
Click the Preheat Icon get into Preheat interface (Pic25). In the interface, Nozzle and Bed can be preheated. Choose the target temperature of nozzle and bed and click OK icon, it begins heating, Click the Cancel icon to return to the previous interface.



Pic26

### 8.6.4 JOG MODE

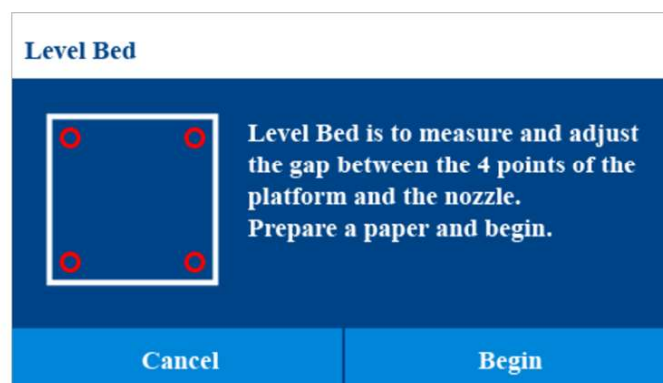
Click the Jog Mode icon get into Jog Mode interface (Pic27). Click Return Home Icon X Y Z will go to the home position. Click X Y Z, it will move in X Y Z direction respectively, there are three step sizes to choose from: 1mm, 1cm and 5cm. Click the Back icon to return to the previous interface.



Pic27

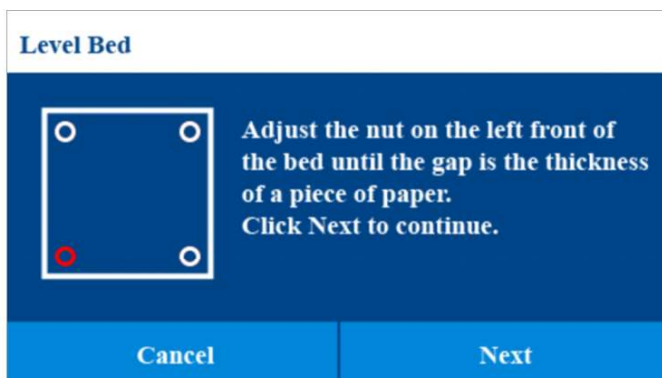
### 8.6.5 LEVEL BED

Click the Level Bed icon get into Level Bed interface (Pic28).

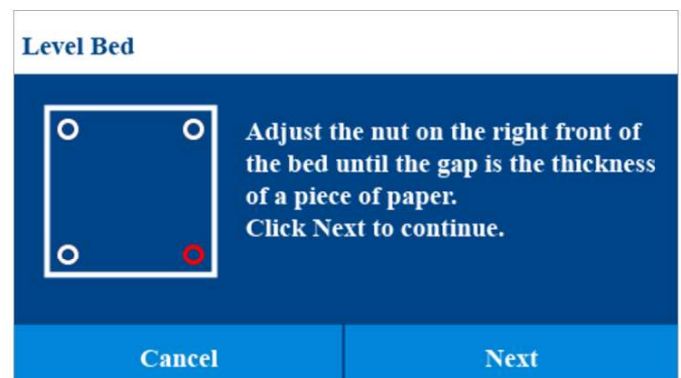


Pic28

Prepare a piece of paper, Click Begin icon to enter the leveling process. It will adjust four points. (Pic29、 Pic30). Follow the interface prompts to complete the adjustment of each point.



Pic29



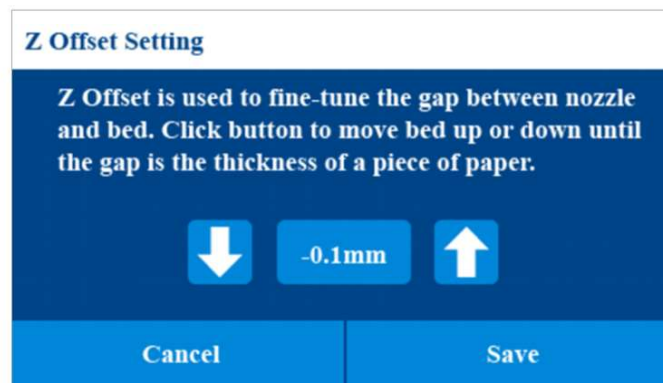
Pic30



### 8.6.6 Z OFFSET

After Level Bed, Please Click the Z Offset icon to adjust the gap between Nozzle and platform.

Note: if the gap is too small, the nozzle will leave deep marks on the platform during printing, which may cause plugging of the print head and damage the nozzle and the platform. If the gap is too big, the nozzle will be suspended during printing, causing printing failure.



Pic31

**Please prepare a business card, after our test, the distance between the nozzle and platform is the best in the thickness of a business card.**

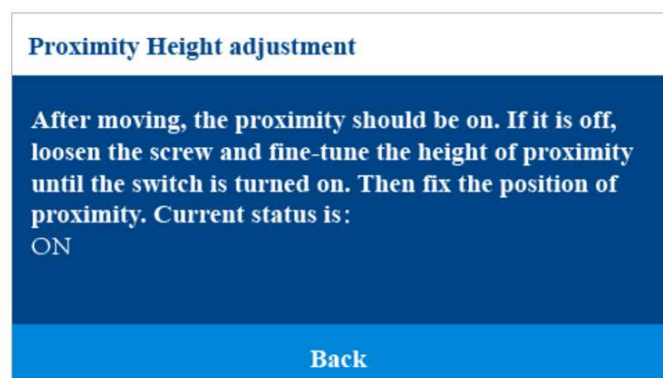
Click the Down icon the nozzle moves downward, and the gap between the nozzle and the platform becomes smaller, Click the UP icon the nozzle moves up, and the gap between the nozzle and the platform becomes larger, after adjusting to the appropriate gap, click Save icon to save (Pic31). The value displayed on the interface will be different for different machines.

### 8.6.7 PROXIMITY HEIGHT

Click Proximity Height Icon to adjust the height of proximity switch.

There is a proximity switch on the left side of the nozzle. When we replace the proximity switch, we need to adjust the height of the proximity switch. This function can help us adjust the height of the proximity switch.

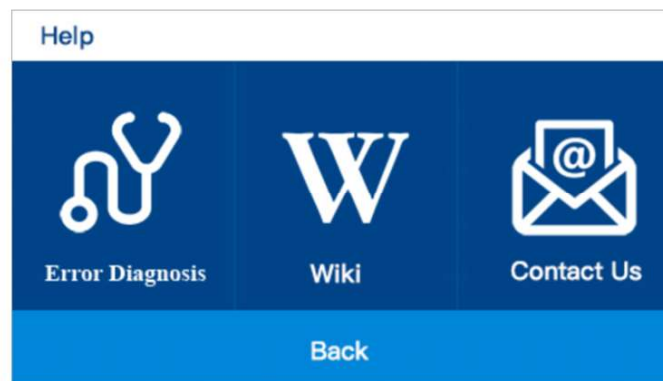
Follow the prompts on the interface to complete the adjustment. (Pic32)



Pic32

## 8.7 HELP INTERFACE

Click Help icon to enter into Help interface.



Pic33

- Error Diagnosis: Diagnose equipment failure step by step.
- Wiki: Check our wiki website for this printer
- Contact Us: contact Us through email or website.

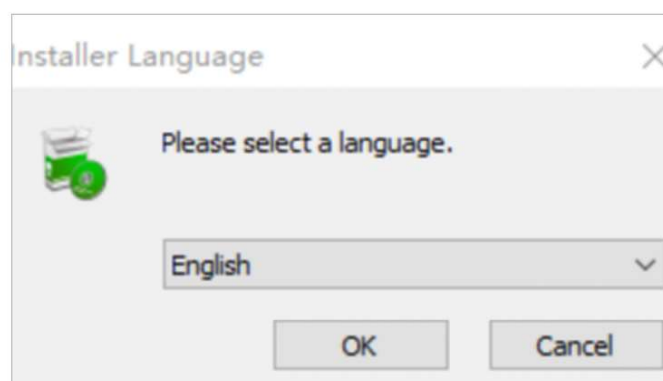
## 9.WIIBUILDER SLICING SOFTWARE

The printer includes the WiiBuilder slicing software on the included SD card. Use the included card reader to display the contents of the SD card on your PC to install the program.

### 9.1 INSTALLATION

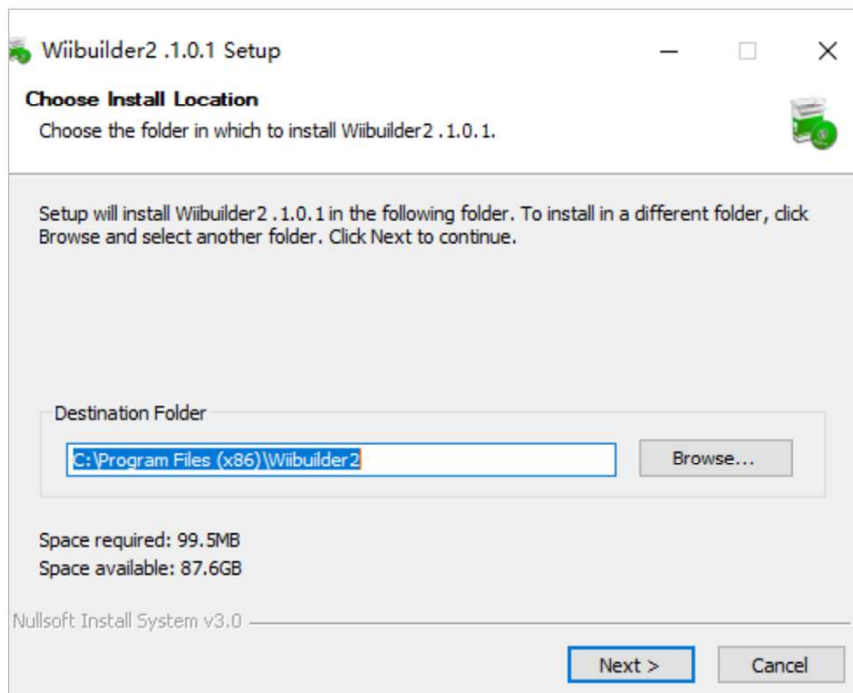
Perform the following steps to install the WiiBuilder slicing software.

1. Locate and run the WiiBuilder.exe software on the SD card to install the WiiBuilder software program. The Installer will prompt you for the language to use in the Install Shield Wizard. Use the pull-down menu to select your preferred language, then click the OK button to continue.



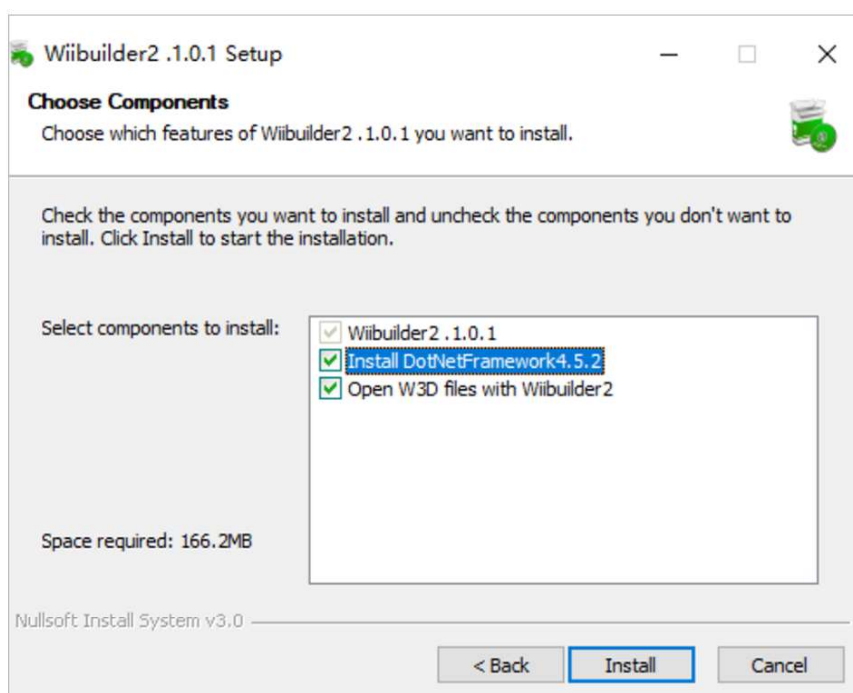
Pic34

2. The Wizard will prompt you to select the install location. If you don't want to use the default install location, use the Browse... button to select a different directory. Click the Next button when you are satisfied with the install location.



Pic35

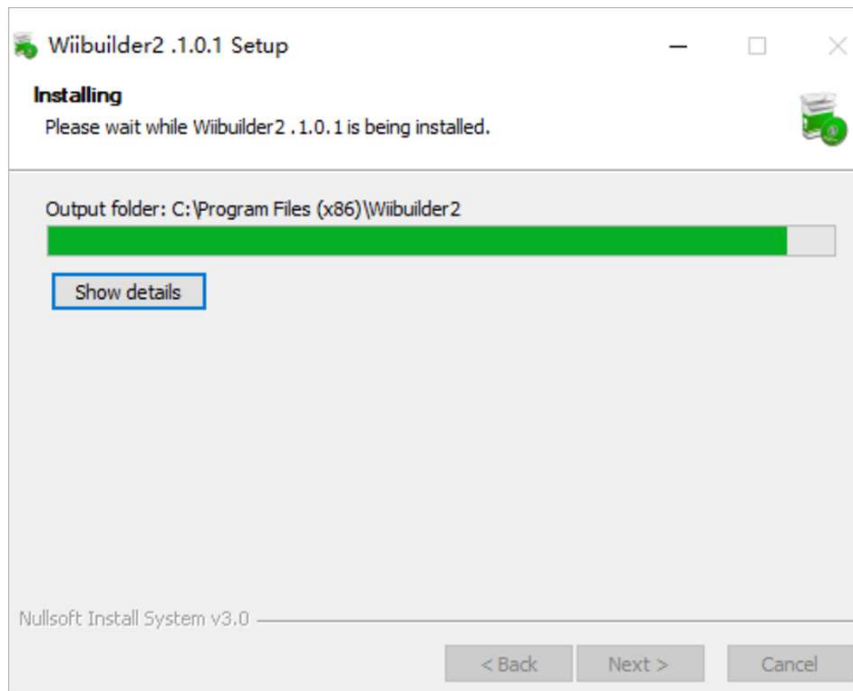
3. Click the Install button to continue.



Pic36

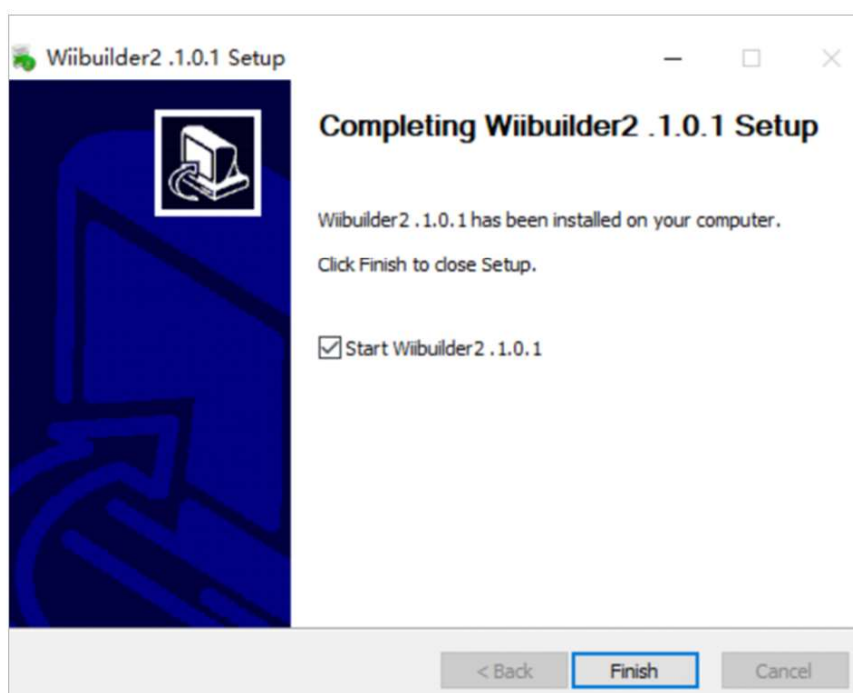


4. Once the installation is complete, click the Next button to continue.



Pic37

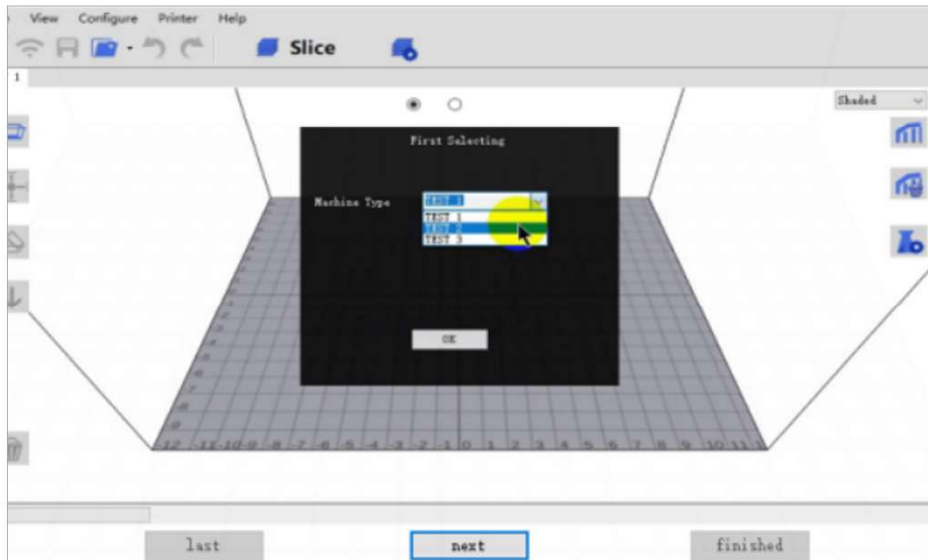
5. Click the Finish button to complete the installation and launch the program.



Pic38

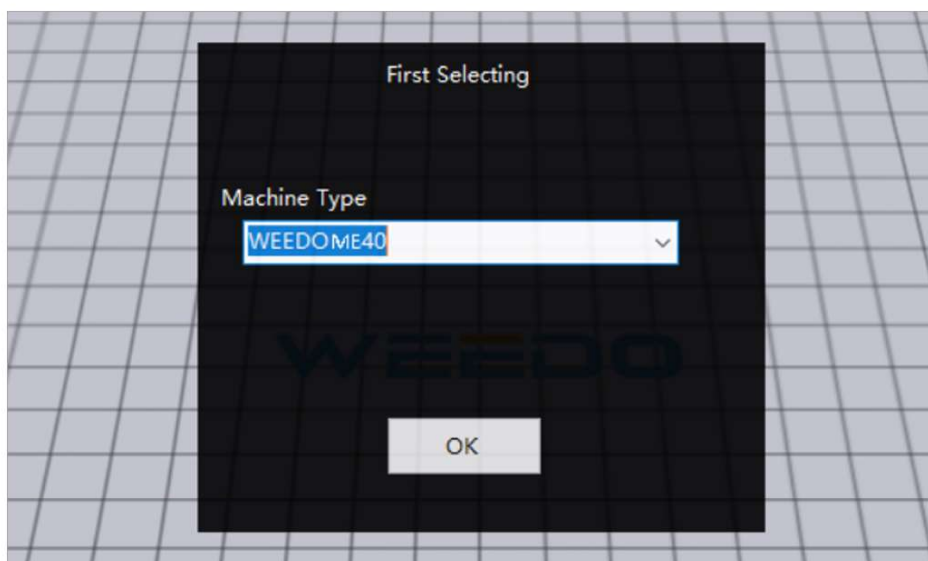
## 9.2 WIIBUILDER SETUP

1. Once the program launches, the Initial Setup Wizard will launch. It will inform you of several program basics, including how to load model files, the locations of the slice buttons, etc. Read each page, clicking the next button to proceed from page to page. Click the finished button on the final page to close the Wizard.



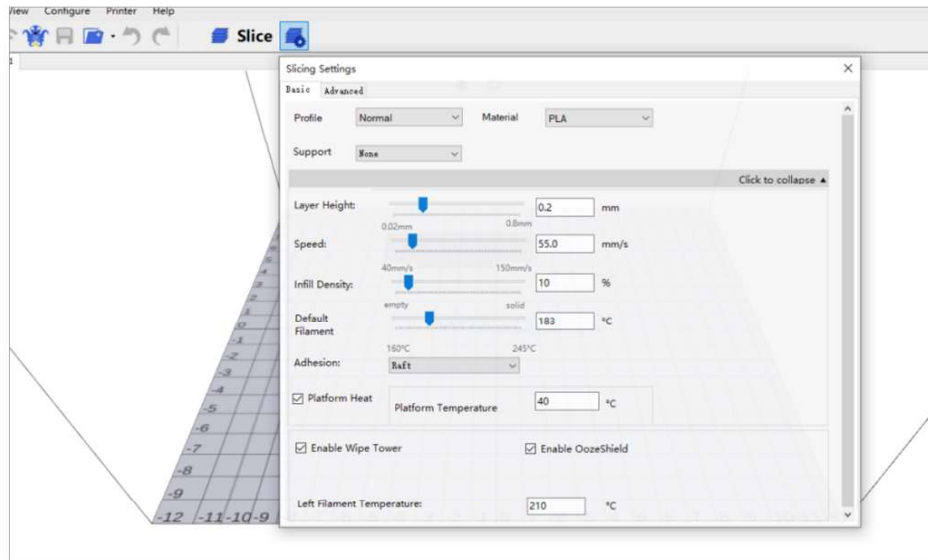
Pic39

2. Once the Wizard closes, you will be prompted to select the printer model and the units of measure you want to use. Select ME40 . Click the OK button to continue.



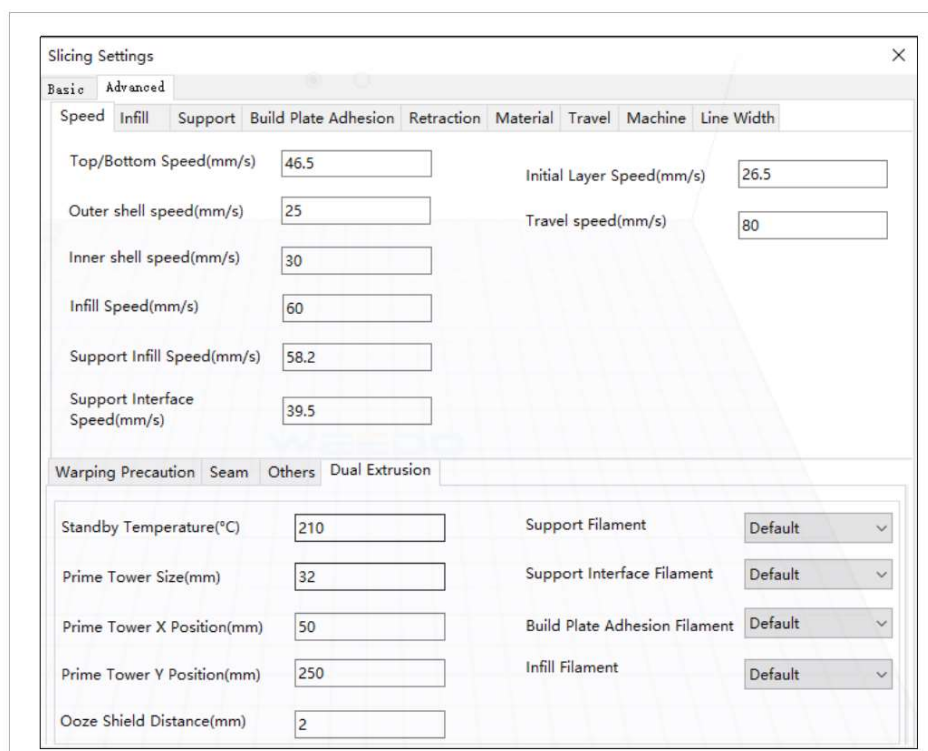
Pic40

- Click the Toolbar button Slicing Settings, open the Slicing Settings dialog. (Pic41)



Pic41

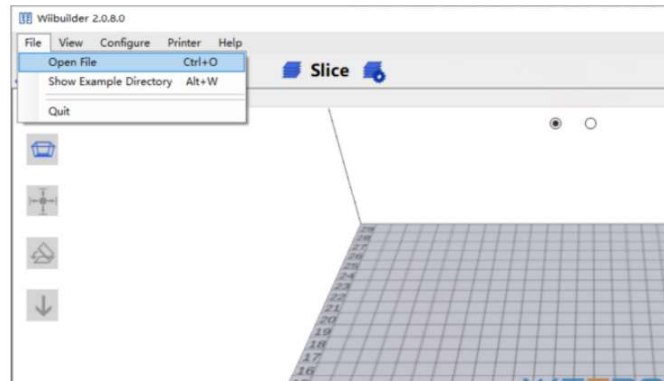
- Click the Advanced tab to display the following dialog. The following sections detail the options on each tab. (Pic42)



Pic42

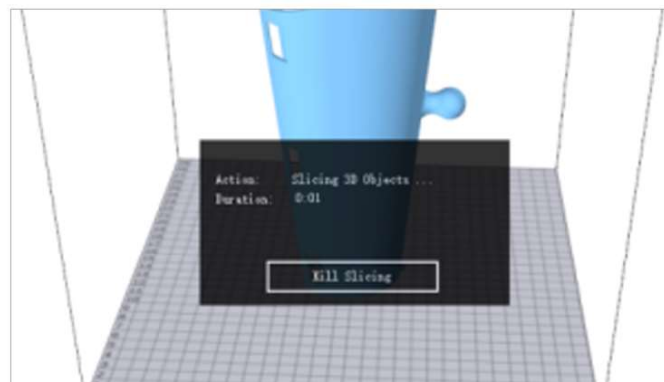


5. Load STL File. Click File- Open File, select your STL file on your computer. You can select example file also. (Pic43)



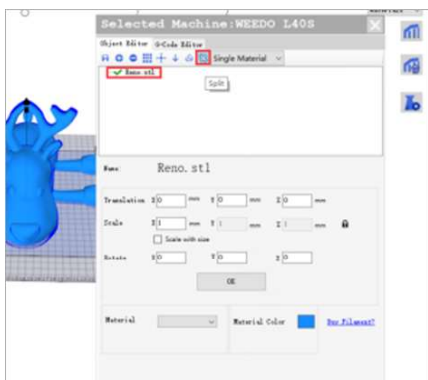
Pic43

6. Slice File: Click Slice, it will slice the file. The bottom left corner of the interface shows the progress of the slice (Pic44), and the bottom right corner of the interface shows the estimated printing time and weight of the printing supplies. Note: you can search this website for free stl model: [www.thingiverse.com](http://www.thingiverse.com)

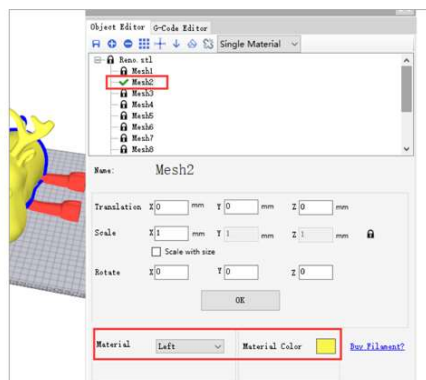


Pic44

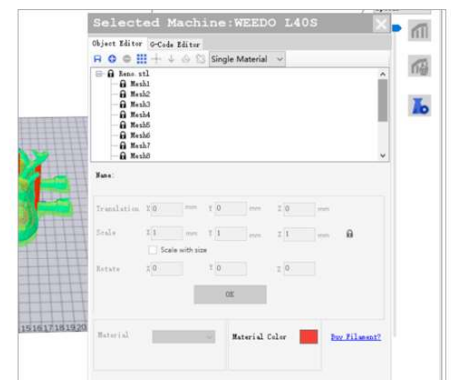
7. How to color different parts of the movable model. Load a stl file-reno, which is movable show in Pic45, Click 'Split' , the model will split into different parts, we can choose different nozzle for different parts to print, show in Pic46. When printing two colors, the upper left corner will print the erase tower for switching colors, show in Pic47.



Pic45



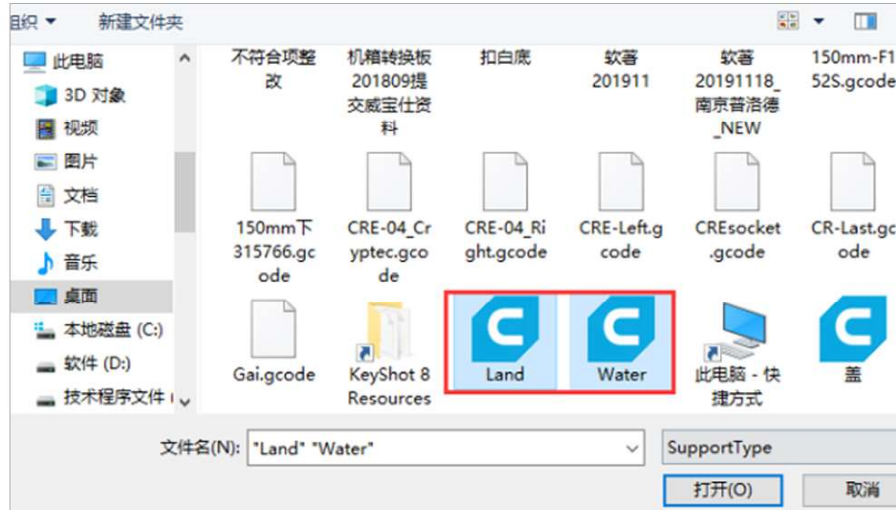
Pic46



Pic47

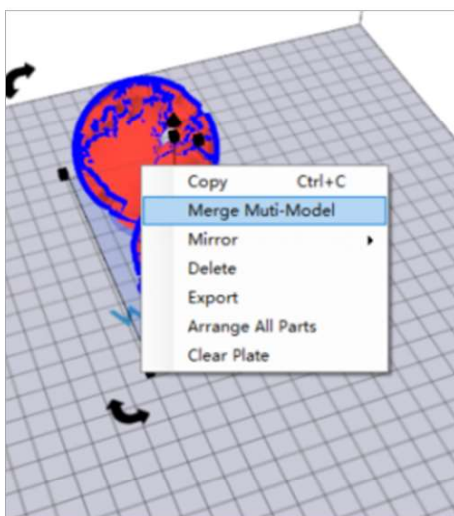
## 8. How to load two models and merge them together.

Load two models simultaneously with the 'Ctrl' key on your keyboard.  
Show in Pic48.

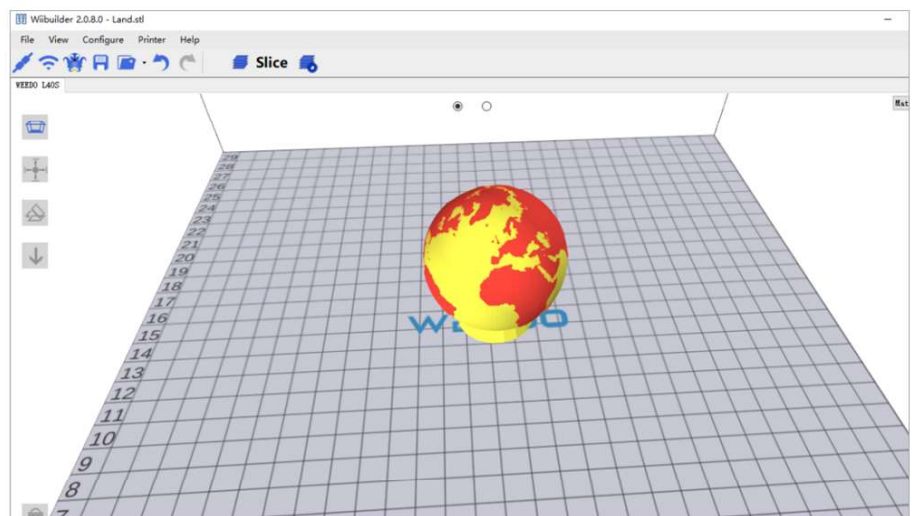


Pic48

After the model is loaded into the software, hold down the ctrl key on the keyboard, click the left mouse button to select both models at the same time, then right-click and select 'Merge Muti-Model', show in Pic49. After merge the model, two model become one model, as show in Pic50. Note: both models must be models that can be merged to perform this step. Set the two-color model according to the setting method of the movable model described above.



Pic49



Pic50

## 9.3 SPEED TAB

| Speed                         | Infill | Support | Build Plate Adhesion | Retraction | Material | Travel                    | Machine | Line Width |
|-------------------------------|--------|---------|----------------------|------------|----------|---------------------------|---------|------------|
| Top/Bottom Speed(mm/s)        | 46.5   |         |                      |            |          | Initial Layer Speed(mm/s) | 26.5    |            |
| Outer shell speed(mm/s)       | 25     |         |                      |            |          | Travel speed(mm/s)        | 104.8   |            |
| Inner shell speed(mm/s)       | 30     |         |                      |            |          |                           |         |            |
| Infill Speed(mm/s)            | 75     |         |                      |            |          |                           |         |            |
| Support Infill Speed(mm/s)    | 58.2   |         |                      |            |          |                           |         |            |
| Support Interface Speed(mm/s) | 39.5   |         |                      |            |          |                           |         |            |

Pic51

The Speed Tab features the following options:

- Top/Bottom Speed (mm/s): Sets the printing speed of the top and bottom surfaces of the model.
- Outer shell speed (mm/s): Sets the printing speed of the external shell surfaces.
- Inner shell speed (mm/s): Sets the printing speed of the internal shell surfaces.
- Infill Speed (mm/s): Sets the printing speed of the infill inside the model.
- Support Infill Speed (mm/s): Sets the printing speed of infill inside the model supports.
- Support Interface Speed (mm/s): Sets the printing speed of the top and bottom surfaces of the model supports.
- Initial Layer Speed (mm/s): Sets the printing speed of the first layer of the model.
- Travel speed (mm/s): Sets the movement speed of the nozzle when not printing.

## 9.4 INFILL TAB

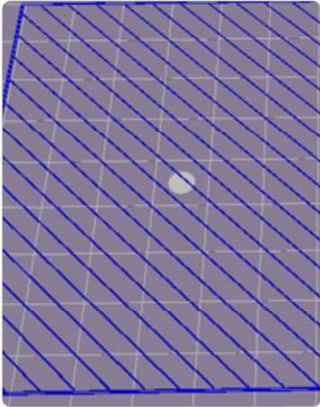
| Speed                     | Infill                   | Support | Build Plate Adhesion | Retraction | Material | Travel                        | Machine                  | Line Width |
|---------------------------|--------------------------|---------|----------------------|------------|----------|-------------------------------|--------------------------|------------|
| Infill Pattern:           | Zig Zag                  |         |                      |            |          | Enable Adaptive Infill        | <input type="checkbox"/> |            |
| Infill Before Walls:      | <input type="checkbox"/> |         |                      |            |          | Adaptive Infill Threshold(mm) | 7                        |            |
| Outer Before Inner Walls: | <input type="checkbox"/> |         |                      |            |          | Adaptive Infill Density(%)    | 60                       |            |

Pic52

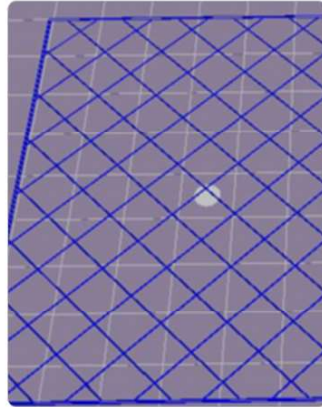


- **Infill Pattern:** Use the drop-down menu to select one of seven different infill patterns, including Lines, Grid, Triangle, Zig Zag, Concentric, Cross, and Octet. The individual patterns are illustrated in the table below.
- **Infill Before Wall:** Check this box to print the model after filling and printing the outline.
- **Outer Before Inner Walls:** Check this box to print the exterior walls before printing the interior walls.
- **Gradual Infill Steps:** For models that need to gradually change the fill rate, this value determines how many layers to change at once.
- **Enable Adaptive Infill:** Check this box to enable the adaptive infill. When a thin wall area appears in the model, the slice engine automatically increases the fill density of this area.
- **Adaptive Infill Threshold:** When the thin wall thickness is less than this value, the slice engine increases the fill density of the area.
- **Adaptive Infill Density:** Infill density used in thin wall area.

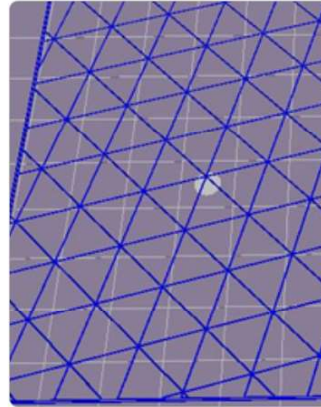
### Infill Patterns:



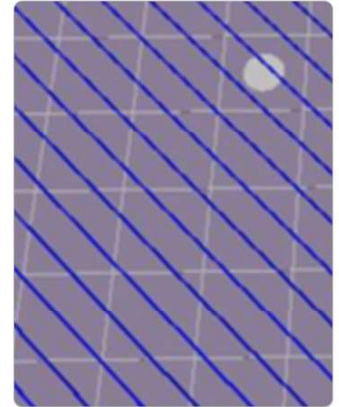
Lines



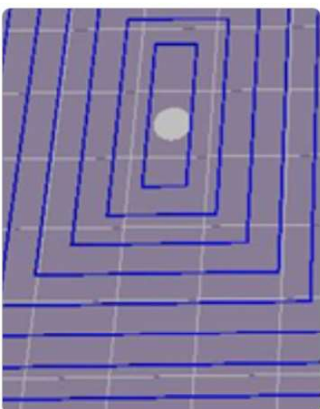
Grid



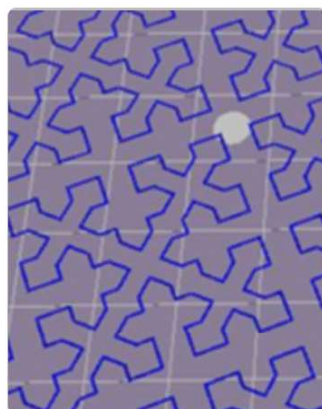
Triangle



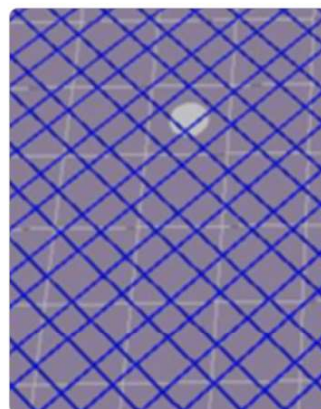
Zig Zag



Concentric



Cross



Octet

## 9.5 SUPPORT TAB

| Speed                          | Infill | Support | Build Plate Adhesion             | Retraction | Material                            | Travel | Machine | Line Width |
|--------------------------------|--------|---------|----------------------------------|------------|-------------------------------------|--------|---------|------------|
| Support Pattern:               |        | Zig Zag | Enable Support Interface         |            | <input checked="" type="checkbox"/> |        |         |            |
| Overhang angle for support(*): |        | 60      | Support Top Thickness(mm)        |            | 0.8                                 |        |         |            |
| Support infill density(%):     |        | 10      | Support Bottom Thickness(mm)     |            | 0.8                                 |        |         |            |
| Support Top Gap(mm):           |        | 0.18    | Support Interface Density(%)     |            | 70                                  |        |         |            |
| Support Bottom Gap(mm):        |        | 0.1     | Support Interface Infill Pattern |            | Lines                               |        |         |            |
| Distance X/Y(mm):              |        | 0.7     | Connect Support                  |            | <input type="checkbox"/>            |        |         |            |

Pic53

- **Support Pattern:** Use the drop-down menu to select one of five support patterns, including Lines, Grid, Triangles, Zig Zag, and Concentric. The pattern designs are the same as those of the infill patterns of the same name.
  - Lines support is easier to remove and is used on models that require more support.
  - Grid support is used on small models, which need fewer supports.
  - Zig Zag support is used for models that are particularly difficult to remove. It is stronger than Lines support and is better than Grid support.
- **Overhang angle:** The overhang angle is the angle between the support and the surface of the model. Larger settings make the supports easier to remove, while smaller settings provide better support. The default angle is 60 degrees.
- **Support infill density (%):** Determines the infill density for supports. The higher the density, the stronger the supports.
- **Support Top Gap (mm):** The distance between the top of the support and the model surface. The smaller the distance, the more effective the support, but is more difficult to remove from the model surface, resulting in residual material on the model surface. The larger the distance, the less effective the support, but is easier to remove from the model surface, resulting in a smoother surface.
- **Support Bottom Gap (mm):** The distance between the bottom of the support and the model surface. The effects of this parameter are the same as the effects of the Support Top Gap (mm) parameter.
- **Distance X/Y (mm):** The distance between the support and the model surface in the horizontal plane. The effects of this parameter are the same as the effects of the Support Top Gap (mm) parameter.
- **Enable Support:** Check this box to use supports.



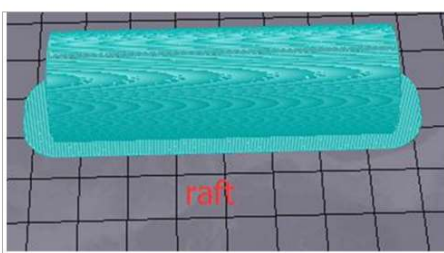
- Support Top: Determines the thickness of the top layer of the supports.
- Support Bottom: Determines the thickness of the bottom layer of the supports.
- Support Interface: Sets the percentage of infill used inside the supports.
- Support Interface Infill Pattern: Use this drop-down menu to choose one of five infill patterns for the supports, including Lines, Grid, Triangles, Zig Zag, and Concentric. The pattern designs are the same as those of the infill patterns of the same name.
- Connect Support: Check this box to connect separate supports into one part.

## 9.6 BUILD PLATE ADHESION TAB

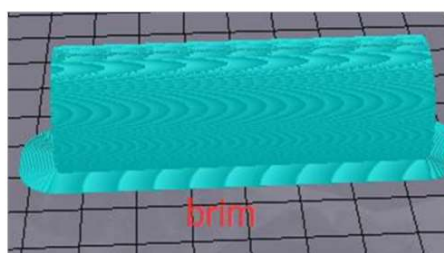
| Speed                   | Infill | Support | Build Plate Adhesion | Retraction       | Material | Travel | Machine | Line Width |
|-------------------------|--------|---------|----------------------|------------------|----------|--------|---------|------------|
| Raft Air Gap(mm)        |        | 0.24    |                      | Brim line amount |          | 20     |         |            |
| Raft Extra Margin(mm)   |        | 5       |                      | Skirt Line Count |          | 1      |         |            |
| Raft Base thickness(mm) |        | 0.3     |                      |                  |          |        |         |            |
| Initial Layer Z Overlap |        | 0.09    |                      |                  |          |        |         |            |

Pic54

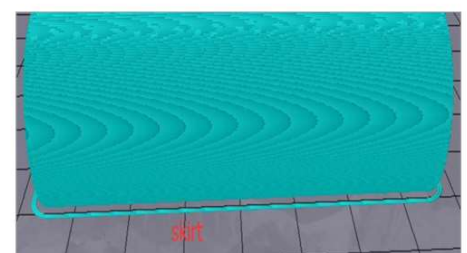
- Raft Air Gap (mm): The distance between the raft and the model. This determines the difficulty of removing the raft from the model.
- Raft Extra Margin (mm): The distance between the edge of the raft and the model surface.
- Raft Base thickness (mm): Determines the thickness of the raft.
- Initial Layer Z Overlap: Determines the amount of overlap between the first and second layers of the model.
- Brim line amount: Sets the number of ring gaskets that are added to the edge of the model in contact with the build platform.
- Skirt Line Count: Sets the number of anti-overflow lines at the end of the model in contact with the build platform.



Pic55

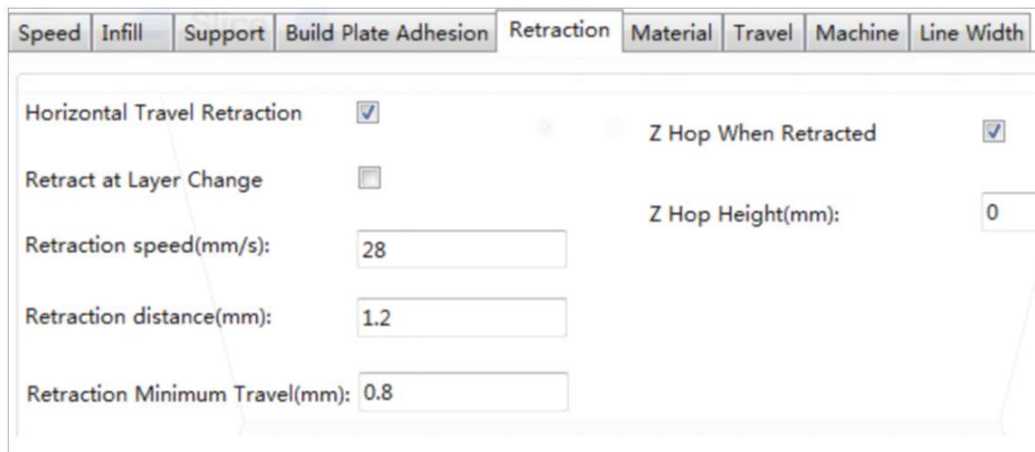


Pic56



Pic57

## 9.7 RETRACTION TAB



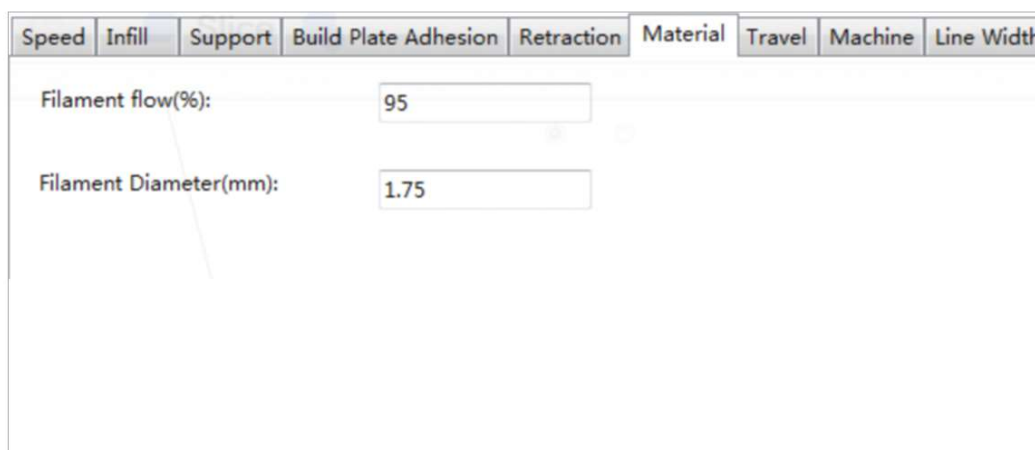
The screenshot shows the 'Retraction' tab selected in the software interface. The tab bar at the top includes: Speed, Infill, Support, Build Plate Adhesion, Retraction (active), Material, Travel, Machine, and Line Width. The main area contains the following settings:

- Horizontal Travel Retraction: ☒
- Retract at Layer Change: ☐
- Retraction speed(mm/s):
- Retraction distance(mm):
- Retraction Minimum Travel(mm):
- Z Hop When Retracted: ☒
- Z Hop Height(mm):

Pic58

- Horizontal Travel Retraction: Check this box to enable filament retraction when the nozzle is not printing and is moving in a horizontal direction.
- Retract at Layer Change: Check this box to retract the filament when switching from layer to layer.
- Retraction speed (mm/s): Sets the speed at which filament is retracted.
- Retraction distance (mm): The distance the filament is retracted within the nozzle.
- Retraction Minimum Travel (mm): Sets the minimum nozzle movement distance before printing and before retracting the filament.
- Z Hoe When Retracted: Check this box to enable lift the nozzle when filament is returned after retraction.
- Z Hop Height (mm): The distance the nozzle is lifted when filament is returned after retraction.

## 9.8 MATERIAL TAB



The screenshot shows the 'Material' tab selected in the software interface. The tab bar at the top includes: Speed, Infill, Support, Build Plate Adhesion, Retraction, Material (active), Travel, Machine, and Line Width. The main area contains the following settings:

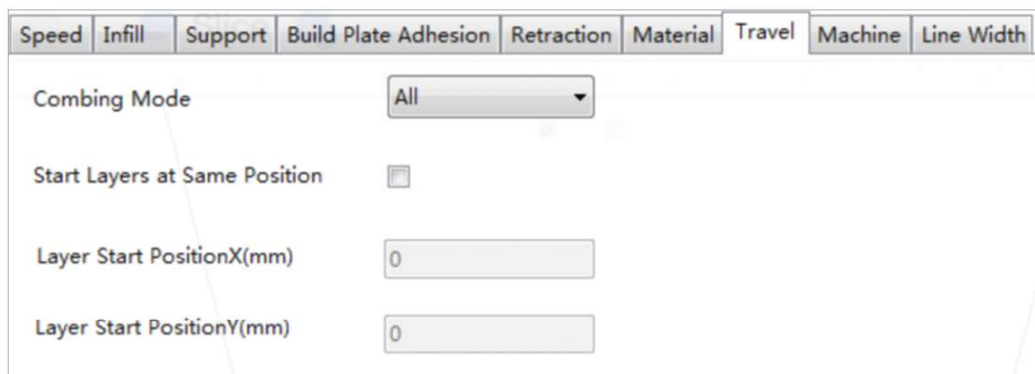
- Filament flow(%):
- Filament Diameter(mm):

Pic59



- Filament flow (%): Sets the flow rate of filament in the melting state. This is set according to the type of filament being used. In general, the flow rate for PLA or PLA Pro is 90 and the flow rate of ABS is 100.
- Filament Diameter (mm): Sets the diameter of the filament being used. This printer only supports 1.75mm diameter filament.

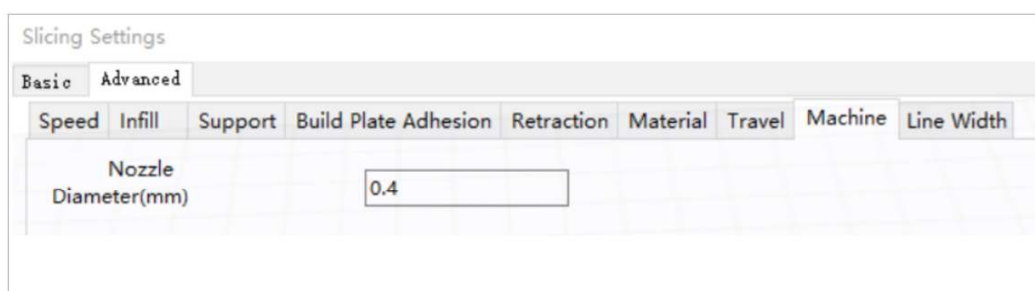
## 9.9 TRAVEL TAB



Pic60

- Combing Mode: This option determines how the nozzle will move when not printing. The Off option has the nozzle move the shortest distance between the previous extrusion location and the new start location. The All option causes the nozzle to move along anything it has already extruded. The No Skin option will avoid the outer layers to move the nozzle to the new start location, which can greatly improve print quality.
- Start Layers at Same Position: This option changes the accuracy of the model in the same plane. It is generally set by default.
- Layer Start Position X (mm): This option allows you to change the X axis coordinates of the position of the model layer.
- Layer Start Position Y (mm): This option allows you to change the Y axis coordinates of the position of the model layer.

## 9.10 MACHINE TAB



Pic61

- **Nozzle Diameter (mm):** Sets the diameter of the nozzle on the right extruder. This printer only has a single extruder, which is designated the right extruder. The nozzle diameter of this printer is 0.4mm.

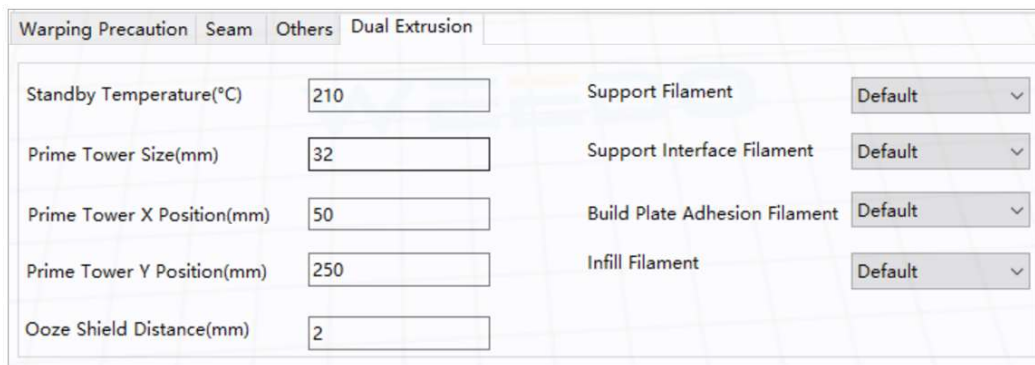
## 9.11 LINE WIDTH TAB

| peed                         | Infill | Support | Build Plate Adhesion | Retraction                 | Material | Travel | Machine | Line Width |
|------------------------------|--------|---------|----------------------|----------------------------|----------|--------|---------|------------|
| Outer Wall Line Width(mm)    |        | 0.4     |                      | Skirt/Brim Line Width(mm)  |          | 0.4    |         |            |
| Inner Wall(s) Line Width(mm) |        | 0.4     |                      | Raft Top Line Width(mm)    |          | 0.4    |         |            |
| Top/Bottom Line Width(mm)    |        | 0.4     |                      | Raft Middle Line Width(mm) |          | 0.7    |         |            |
| Infill Line Width(mm)        |        | 0.5     |                      | Raft Base Line Width(mm)   |          | 0.8    |         |            |
| Support Line Width(mm)       |        | 0.4     |                      | Prime Tower Line Width(mm) |          | 0.4    |         |            |

Pic62

- **Outer Wall Line Width (mm):** This is the width of the outermost wall line. By lowering this value, higher levels of detail can be printed.
- **Inner Wall(s) Line Width (mm):** This is the width of a single wall line for all walls except the outermost wall.
- **Top/Bottom Line Width (mm):** This is the width of the top and bottom lines.
- **Infill Line Width (mm):** This is the width of a single infill line.
- **Support Line Width (mm):** This is the width of a single support structure line.
- **Skirt/Brim Line Width (mm):** This is the width of a single skirt or brim line.
- **Raft Top Line Width (mm):** This is the width of the lines in the top surface of the raft. These lines can be thin so that the top of the raft is smooth.
- **Raft Middle Line Width (mm):** This is the width of the lines in the middle raft layers. Making the second layer extrude more causes the lines to stick to the build plate.
- **Raft Base Line Width (mm):** This is the width of the raft base layer. These should be thick lines to assist with build plate adhesion.
- **Prime Tower Line Width (mm):** This is the extrusion width of the prime tower.

## 9.12 DUAL EXTRUSION TAB

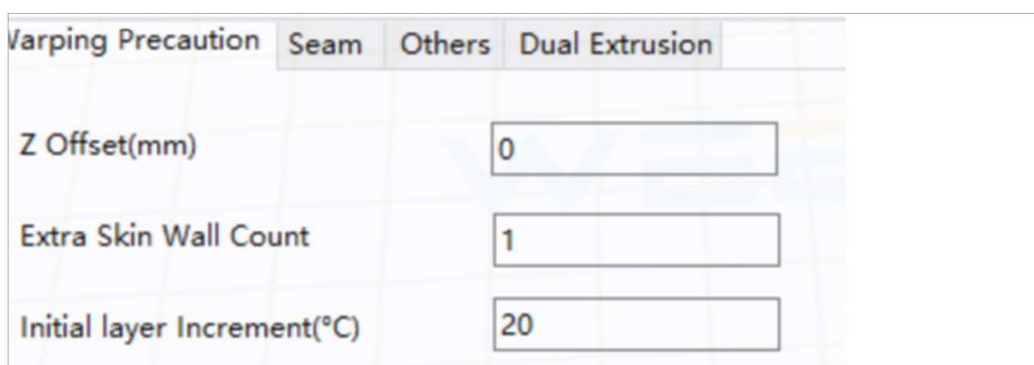


| Dual Extrusion                |         |
|-------------------------------|---------|
| Standby Temperature(°C)       | 210     |
| Prime Tower Size(mm)          | 32      |
| Prime Tower X Position(mm)    | 50      |
| Prime Tower Y Position(mm)    | 250     |
| Ooze Shield Distance(mm)      | 2       |
| Support Filament              | Default |
| Support Interface Filament    | Default |
| Build Plate Adhesion Filament | Default |
| Infill Filament               | Default |

Pic63

- Standby Temperature (°C): Sets the temperature of the second extruder when not actively printing.
- Prime Tower Size (mm): This is the extrusion width of the prime tower. The prime tower is a location that the printer will use to switch nozzles during dual extrusion to prevent a gap in the model by printing a small amount of filament at the prime tower location.
- Ooze Shield Distance (mm): This is the size of the circle of protection layers printed on the edge of the model.
- Support Extruder: Selects which extruder to use for printing support structures.
- Support Interface Extruder: Selects which extruder to use for printing supports.
- Build Plate Adhesion Extruder: Selects which extruder to use for printing the initial layer.
- Infill Extruder: Selects which extruder to use for printing the infill.

## 9.13 WARPING PRECAUTION TA



| Warping Precaution          |    |
|-----------------------------|----|
| Z Offset(mm)                | 0  |
| Extra Skin Wall Count       | 1  |
| Initial layer Increment(°C) | 20 |

Pic64

- Z Offset (mm): When the Z axis bias is set to negative, the nozzle will print closer to the build platform, which helps reduce warping on large models.



- **Extra Skin Wall Count:** This value sets the number of contours on the outer surface of the model.
- **Initial Layer Increment (°C):** This value is used to increase the printing temperature of the first layer, which helps reduce warping on large models.

## 9.14 SEAM TAB

| Warping Precaution     | Seam                     | Others | Dual Extrusion |
|------------------------|--------------------------|--------|----------------|
| Z Seam Type            | Shortest                 |        |                |
| Z Seam X(mm)           | 100                      |        |                |
| Z Seam Y(mm)           | 300                      |        |                |
| Hiding Seam Preference | Hide Seam                |        |                |
| Z Seam Relative        | <input type="checkbox"/> |        |                |

Pic65

Note: The Z Seam is where the printer finishes its motion when printing the skin (outside layer) of a model. This can result in a small blob or zit where the printer changes the Z height. If in alignment, there can be a noticeable line up the side of the print, referred to as a Z Seam, because the filament continues to ooze at the start/stop location. The options on this screen are used to mitigate this effect.

- **Z Seam Type:** Determines where the Z Seam will appear.
  - **Shortest:** This option selects the most time efficient start/stop location.
  - **User Specified:** This option allows you to specify the X and Y start/stop location, which determines where the Z Seam will appear.
  - **Random:** With this option, the printer will randomly choose the start/stop location, which prevents building a column.
  - **Sharpest Corner:** The start/stop location and the Z Seam will appear in the sharpest corner of the model.
- **Z Seam X (mm):** This option is the X location of the Z Seam. This option can only be set when the Z Seam Type is set to User Defined.
- **Z Seam Y (mm):** This option is the Y location of the Z Seam. This option can only be set when the Z Seam Type is set to User Defined.
- **Hiding Seam Preference:** This option is only available when the Z Seam Type is set to Sharpest Corner. It determines whether the Z Seam will be on the inside or outside of the corner.
- **Z Seam Relative:** Checking this box will set the Z Seam in respect to the object's center, whereas leaving the box unchecked will set the Z Seam along the



absolute position on the build plate. This option is only available when the Z Seam Type is set to User Defined.

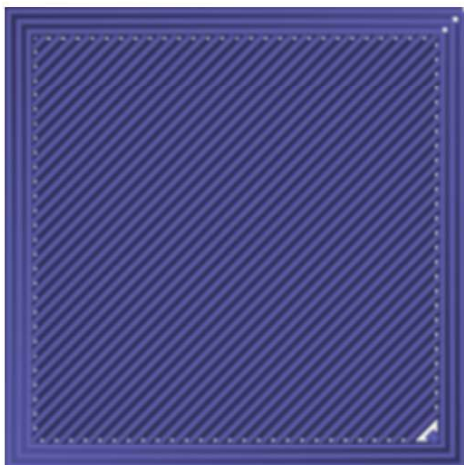
## 9.15 OTHERS TAB

| Warping Precaution        | Seam | Others                              | Dual Extrusion  |
|---------------------------|------|-------------------------------------|-----------------|
| Skin Layers Thickness(mm) |      | 0.8                                 | Wall Line Count |
| Horizontal Expansion(mm)  |      | 0                                   | 2               |
| Enable Print Cooling      |      | <input checked="" type="checkbox"/> |                 |
| Enable Draft Shield       |      | <input type="checkbox"/>            |                 |

Pic66

- Skin Layers Thickness (mm): This option determines the thickness of the top and bottom skin layers.
- Horizontal Expansion (mm): Thermoplastics tend to shrink when cooling. This option allows you to fine tune the part size to offset shrinkage for prints that require tighter tolerances.
- Skin Alternate Rotation: Typically, a 3D printer will print solid layers for the top and bottom layers. When doing this, it changes direction 90 degrees from layer to layer. This setting changes that behavior to add an additional 45 degrees of rotation every two layers.

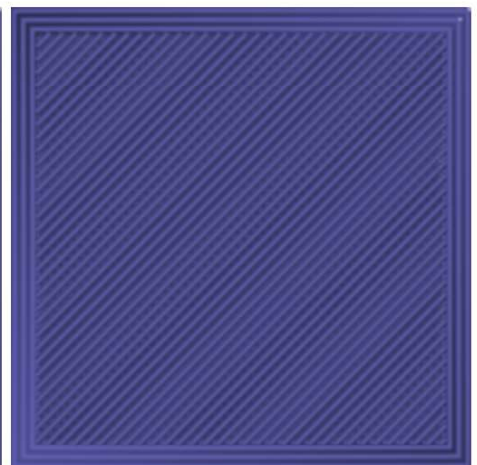
The following images illustrate the normal print direction of the first three layers.



Pic67

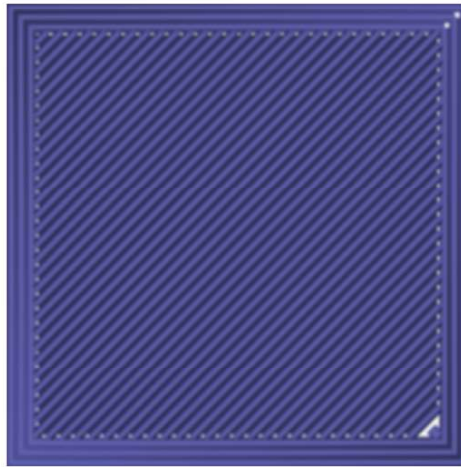


Pic68



Pic69

The following image illustrates the print direction of layer three when the Skin Alternate Rotation option is enabled.



Pic67

- Enable Print Cooling: When enabled, cooling air will be directed at the printed part.
- Enable Draft Shield: When enabled, this printer will print a wall around the model to prevent environmental breezes or drafts from affecting the cooling. This is typically used when Enable Print Cooling is disabled for filament that needs a longer cooling time, such as ABS.
- Wall Line Count: This option determines the number of walls to print.