FANSYS

WIIBUILDER | WIFI | Eco-friendly Filament | One-Click Filament In | Auto Leveling | Magnetic Sticker

TINA2S

WIFI 3D PRINTER

User's Manual

CONTENTS

1. Important Information

It is important that you read this manual completely before you start using the printer. Please pay specific attention to the guidelines and safety warnings in this section. This manual should be stored in a safe place for future reference.

1.1. Safety Warnings

- As the nozzle temperature goes as high as 230°C, don't touch the motors or nozzle while the 3D printer is printing or has just completed printing.

- Don't expose the printer to water or any other type of liquid. Don't place any container with liquid on or near the printer. If the printer does get wet, unplug it from the power outlet immediately and allow it to dry fully before applying power again.

- Don't touch the printer, the power cord, or any other cables connected to it with wet hands.

- Before operating the printer, check the power cord and unit for physical damage. Don't use it if physical damage is detected.

- Ensure that the power outlet provides the same type and level of power required by the printer before plugging it into a power outlet.

- When the printer is not in use unplug it from the power source.

- Avoid damaging the power cord. Don't permit it to become pinched, crimped, become tangled with other cords, or walked on. Make sure that the power cord doesn't present a tripping hazard.

- Don't unplug the printer by pulling on the power cord but grasp the adapter body or connector head.

Ensure that the printer is unplugged from the power source and turned off before performing a service or making repairs.

1.2. Filament

When using this 3D printer, only use filaments provided by the supplier. Filaments available on the retail market have uneven quality and different specifications, which may result in clogging and cause damage to the motor or nozzle. If the 3D printer fails as the result of using third-party consumables, the warranty will be void.

In cases where the filaments won't be used for a long time after unpacking, they should be sealed to preserve them. PLA absorbs moisture when exposed to air for extended periods, and this will affect the quality of the products that are printed.

1.3. Environmental Requirements

This 3D printer should only be used indoors within an ideal ambient temperature range of between 15°C and 25°C. At an ambient temperature of lower than 10°C, the printing model will have difficulty adhering to the platform. The printer will not start at an ambient temperature lower than 0°C. At ambient temperatures of higher than 30°C, the print quality will be significantly degraded and the nozzle could become clogged.
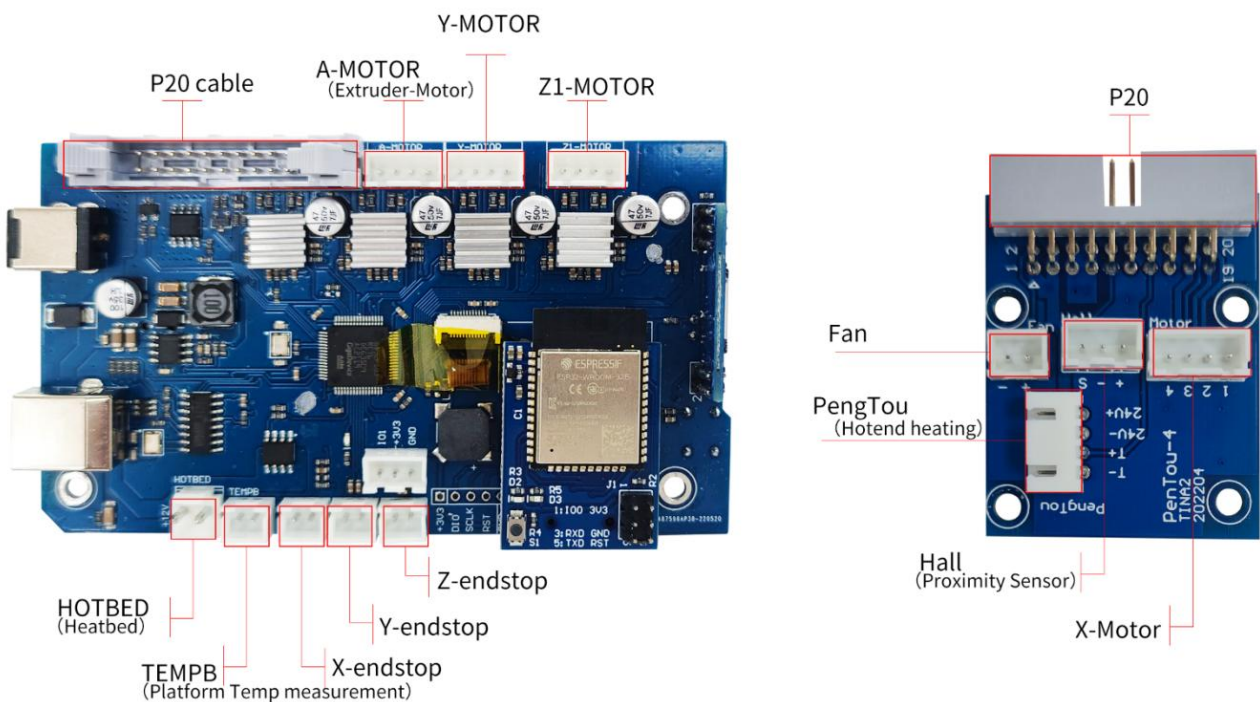
2. INTRODUCTION

2.1. Product Specifications

Model	TINA2S (WiFi model)
Build Volume	3.9'' x 4.0'' x 3.9'' (100 x 105 x 100mm)
Printing Technology	FDM
Layer Thickness	0.1-0.4mm
Nozzle Diameter	0.4mm
Leveling Technology	3 Points Auto Bed Leveling
Platform Material	Flexible Spring Steel
Motherboard	Silent Motherboard with TMC2208
Maximum Heated Bed Temperature	60°C
Maximum Nozzle Temperature	245°C
Printing Precision	±0.1mm
Maximum Printing Speed	200mm/s
Power Supply	AC 100-240V, DC 12V5A, 60W
Product Weight	6.6lbs (~3Kg)
Supported Filament Types	PLA / PLA+ / TPU
Filament Diameter	1.75mm
Supported Filament Spool Weight	100g-250g with built-in filament holder OR up to 1Kg when using a filament rack (not included)
Supported Model File Types	STL/OBJ/GCODE
Supported Slicing Software	Wiibuilder (Windows/MacOS) Cura-SANSYS (Windows/MacOS) OctoPrint (Pi OS/Linux)
Input Method	TF Card / WiFi / USB / App
App	PoloPrint Cloud (Android/IOS)

2.2. Product Overview
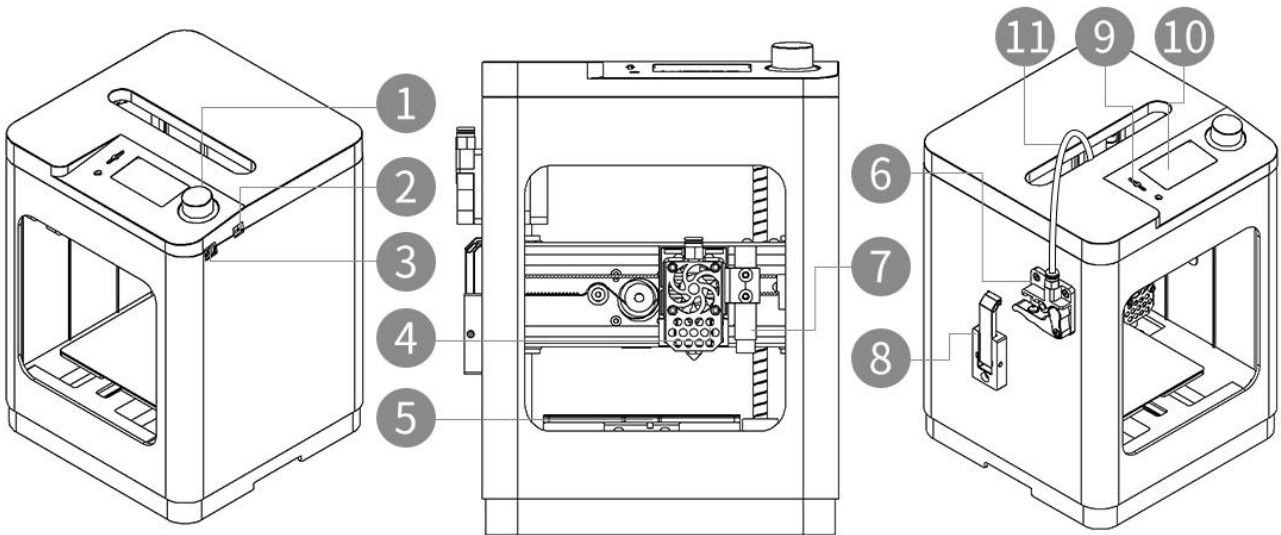
1. Control Knob	5. Heated Bed Platform	10. Screen Display
2. Power Socket	6. Extruder	11. Filament Tube
3. USB Socket	7. Proximity Sensor	
4. Hot End (Nozzle, Heated Aluminum Block)	8. Filament Holder	
	9. TF Card Slot (MicroSD)	

2.3. Motherboard and Adapter Board

2.4. Internal Structure

· Z endstop

· LED

· Extruder motor

· X motor

· X endstop

· Hotend

· Proximity Sensor

· Platform

· Heatbed

· Y endstop

· Coupling

· Y motor

· Z motor

Hotend:

Tube Fitting

Heat Dissipation
Aluminum Block

Heat Break Screw

Heat Break

Buckle

Heating pipe

Temperature
Measuring Cable
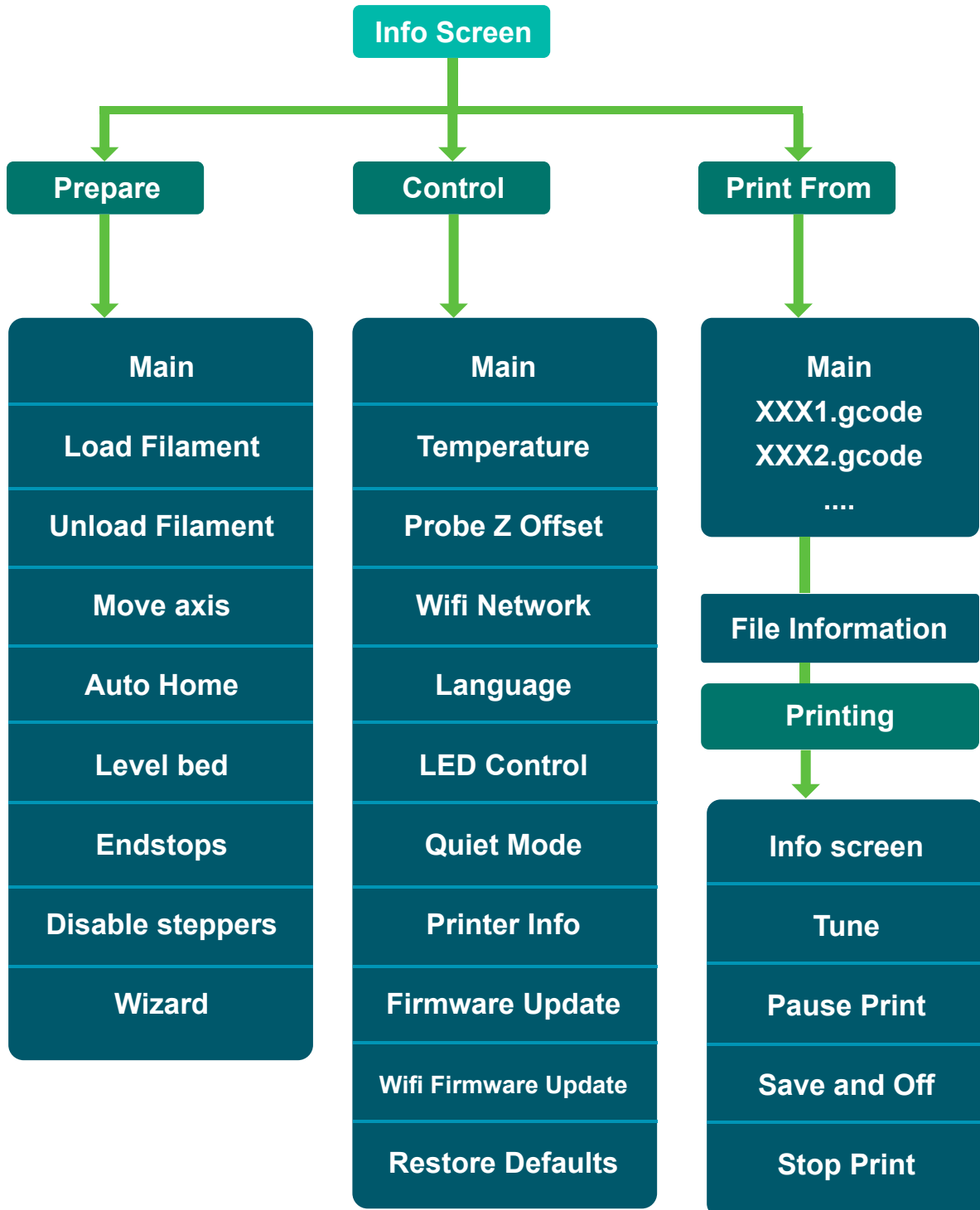
Heated Aluminum Block

Nozzle
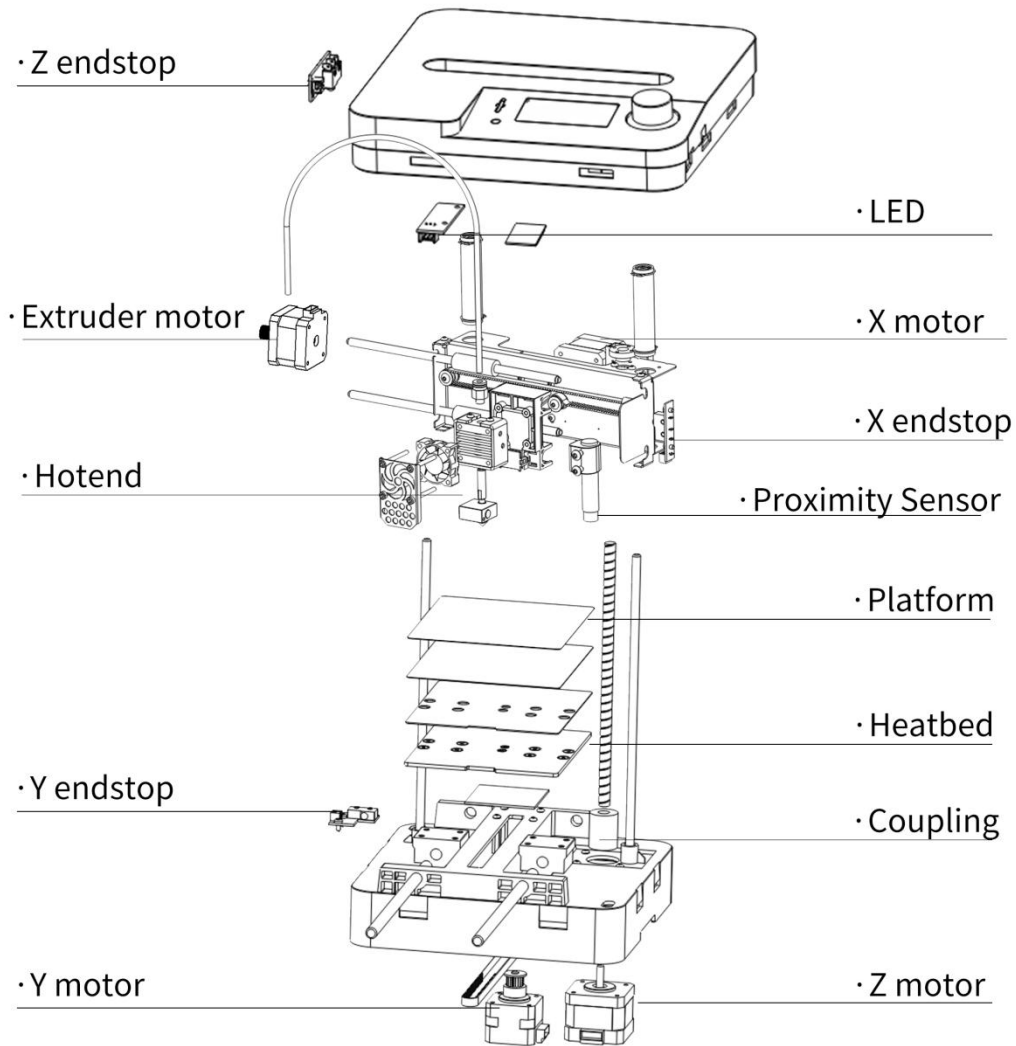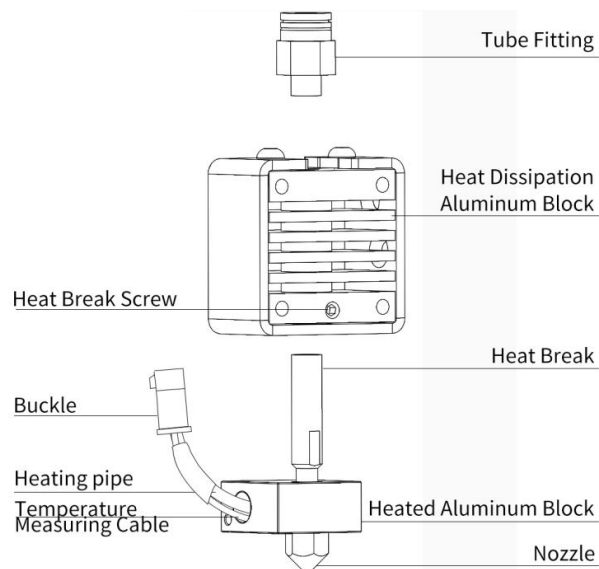
NOTE:
Never touch or put your hands near the hotend while the device is turned on and do
not touch it for several hours after it has been turned off.

3. SYSTEM MENU

In the operation panel area on the top of the 3D printer, the left side is the MicroSD card slot. The print data file is stored in the MicroSD card. In the menu, the MicroSD card is referred to as "TF", which stands for "TransFlash", the technology that MicroSD cards are based on.

In the middle is an LCD screen, and on the right is a knob button. Press the knob button to open the system menu. Below is a graph that illustrates the menu structure shown in the display.

Info Screen

Prepare	Control	Print From
Main	**Main**	**Main** **XXX1.gcode** **XXX2.gcode** **....**
Load Filament	**Temperature**	
Unload Filament	**Probe Z Offset**	**File Information**
Move axis	**Wifi Network**	**Printing**
Auto Home	**Language**	**Info screen**
Level bed	**LED Control**	**Tune**
Endstops	**Quiet Mode**	**Pause Print**
Disable steppers	**Printer Info**	**Save and Off**
Wizard	**Firmware Update**	**Stop Print**
	Wifi Firmware Update	
	Restore Defaults	

3.1. Basic Menu Control

■ Turn the control knob clockwise to move the highlight on the menus down or to decrease a value.

■ Turn the control knob counterclockwise to move the highlight on the menus up or to increase a value.

■ Press (click) the control knob to enter the sub-menu, select a highlighted option, or accept an edited value.

3.2. Info Screen

The Info Screen will be displayed once the 3D printer has been started. Use this screen to monitor the status of the 3D printer.

The info screen displays the main component statuses. This includes:

■ Nozzle temperature (preset and actual temperature),
■ Heated bed temperature (preset and actual temperature)
■ Network connection status
■ XYZ position information
■ Printing time remaining
■ TF card status (MicroSD)

The information bar is at the bottom of the screen, and it displays the network status after powering on. It will display 'OFF' when the printer is disconnected from the Internet. If the printer is connected to a WiFi network, the IP address of the printer will be displayed.

Press the control knob to access the main menu.

■ **Info Screen:** Selecting this option will display the info screen.

■ **Prepare**: Selecting this option will display the prepare menu.

■ **Control**: Selecting this option will display the control menu.

■ **Print From MicroSD**: Selecting this option will display the list of compatible files on the SD card. The printer supports the following file formats: STL, OBJ, and GCODE.

3.3. Prepare Menu

```
Info screen        ⬆
Prepare            →
Control            →
Print from TF
```
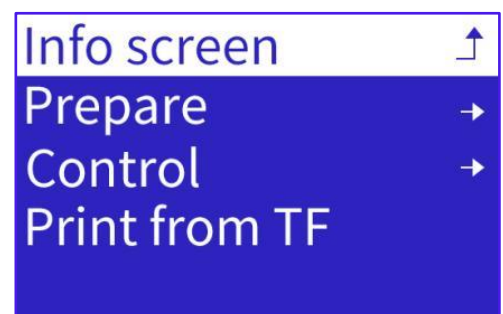
Select and click the control knob to access the prepare menu. This menu consists of two screens. Turning the control knob will scroll the screen.

```
Main               ⬆
Load Filament
Unload Filament
Move axis          →
Auto home
```

```
Auto home
Level bed
Endstop            →
Disable steppers
Wizard
```

3.3.1. Load Filament

This option automatically heats the nozzle and loads the filament.

Select and click the control knob to access the Load Filament interface.

```
Main               ⬆
Load Filament
Unload Filament
Move axis          →
Auto home
```

```
Load Filament
Heating nozzle
Please wait...

Nozzle:  E1    185/210
```

The nozzle will first be heated to 210°C, after which the extruder will feed the filament into the nozzle. The control knob can be pressed to exit the interface during heating. The process will continue in the background.

3.3.2. Unload Filament

This option automatically heats the nozzle and unloads the filament.

Select and click the control knob to access the Unload Filament interface.

The nozzle will be heated to 210°C, then the extruder will retract the filament from the nozzle.

During heating or extrusion, you can press the knob button to exit the current operation.

```
Main               ⬆
Load Filament
Unload Filament
Move axis          →
Auto home
```

```
Unload Filament
Heating nozzle
Please wait...

Nozzle:  E1    185/210
```

3.3.3.Move Axis

The Move axis option is used to manually operate each stepper motor individually. This is used to troubleshoot or debug the 3D printer. Press the control knob to access the sub-menu.

Main	⤴
Load Filament	
Unload Filament	
Move axis	→
Auto home	

Prepare	⤴
Move X	→
Move Y	→
Move Z	→
Extruder	→
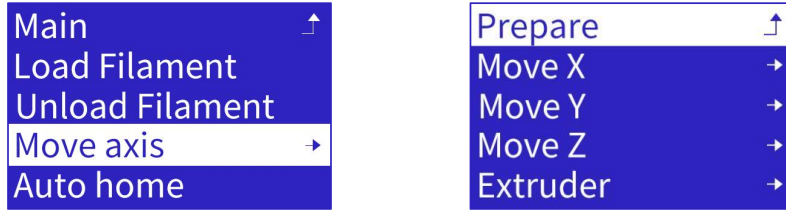
The Move X, Y, and Z options are used to control the movement of the individual motors. The distances of the X, Y, and Z axis are 10mm, 1mm, and 0.1mm respectively. The distances of the extruder motors are 10mm, 1mm, and 0.1mm respectively.

The control knob can be used to control the bidirectional operation of the axis motor in the debugging interface of each axis.

X-Axis Motor Adjustment

Prepare	⤴
Move X	→
Move Y	→
Move Z	→
Extruder	→

Move X	
Move axis	⤴
Move 10mm	→
Move 1mm	→
Move 0.1mm	→

| Move X: | +100.0 |

Y-Axis Motor Adjustment

Prepare	⤴
Move X	→
Move Y	→
Move Z	→
Extruder	→

Move Y	
Move axis	⤴
Move 10mm	→
Move 1mm	→
Move 0.1mm	→

| Move Y: | +100.0 |

Z-Axis Motor Adjustment

Prepare	⤴
Move X	→
Move Y	→
Move Z	→
Extruder	→

Move Z	
Move axis	⤴
Move 10mm	→
Move 1mm	→
Move 0.1mm	→

| Move Z: | +100.0 |

Extruder Adjustment

Prepare	⤴
Move X	→
Move Y	→
Move Z	→
Extruder	→

Extruder	
Move axis	⤴
Move 10mm	→
Move 1mm	→
Move 0.1mm	→

| Extruder: | +100.0 |

NOTICE:

1. The movement range of each motor is between 0mm and 100mm. First, perform the 'Auto home' operation before jogging to ensure that the 3D printer obtains the correct coordinates of the nozzle. If the 'Auto home' operation is not performed, the 3D printer will set the current position as the origin position. This will lead to a smaller movement range for the jog operation.

2. The movement of each motor is controlled by an endstop on each axis. When the endstop is reached, the movement of the motor will stop immediately. Note that if the endstop is not wired properly or is faulty, this may lead to a false-trigger or no-trigger signal, causing the motor to not move or stop.

3. There is a thermal protection function on the Extruder Motor, and it will only move when the temperature of the nozzle is more than 170°C.

3.3.4. Auto home

When the 'Auto home' option is selected, the X, Y, and Z axes will move independently, and the 3D printer nozzle and platform will return to the origin automatically. Movement will stop when each individual endstop is reached.

This function can be used for troubleshooting. If the motor or endstop fails, this function will display a fault prompt.

3.3.5. Level Bed

Click "Level bed", the system will use the proximity sensor to do an Auto Bed Leveling operation.

At the beginning of each print, system will always do the auto level bed operation, so that the nozzle and the platform maintain a proper distance during printing.

3.3.6.Endstop

This option displays the status of the X, Y, and Z endstops and the proximity sensor. When you move the nozzle or the platform to reach the endstops, you can monitor the state of each endstop or the proximity sensor.

Auto home		Endstops
Level bed		X: Off
Endstop →		Y: Off
Disable steppers		Z: Off
Wizard		Z-Probe：Off
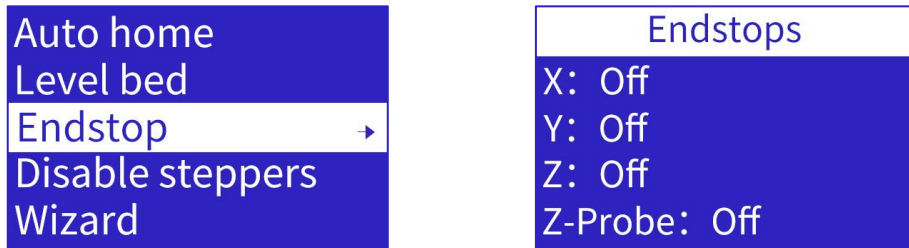
This function can be used for fault diagnosis of the endstops and proximity sensor. If an endstop or proximity sensor fails, the state will not change when it is reached.

3.3.7.Disable Steppers

This option will release the stepper motor locks. It is handy if the nozzle or platform needs to be moved manually.

```
Auto home
Level bed
Endstop              →
Disable steppers
Wizard
```

3.3.8.Wizard

This option displays the steps of printing from the TF card (MicroSD Card). This function is most often used by novice printers.

First, select the language. Selecting Next on 'Step 1: Load Filament' will heat the nozzle to the specified temperature, after which the 'Step 2: Insert TF card' and 'Step 3: Choose File' options will be displayed. Select 'Print from TF', choose the file, and then 'Print'.

Auto home	Language	中文
Level bed	English	Italiana
Endstop →	Franoais	日本
Disable steppers	Deutsohe	Portugal
Wizard	Espanol	Nederlands

STEP1: LOAD FILAMENT	Load Filament	PRINT PAUSED
Next	Heating nozzle	Wait for
Quit	Please wait...	filament purge
	Nozzle: E1 185/210	Nozzle：E1 209/210

```
STEP2:INSERT TF CARD      Main                    ↱      STEP3:CHOICE FILE
Next                      Flexi-Rex-improve...            Print from TF
Quit                      FRUIT_SL.GCO                    Quit
                          228F974B.GCO
                          DFE92F8B.GCO
```

```
Print                            Density：NONE
Quit                      ↱      Filament：NONE
LayerHeight：0.2mm               Usage：NONE
Density：NONE                    Temperature：NONE
Filament：NONE                   PrintTime：01h53m
```

3.4. Control Menu

```
Info screen          ↱
Prepare              →
Control              →
Print from TF
```

Select and click the control knob to access the control menu. This menu has three screens, and the screen can be scrolled by turning the knob.

```
Main              ↱      LED Control          →      Quiet Mode              →
Temperature       →      Quiet Mode           →      Printer Info            →
Prode Z Offset    →      Printer Info         →      Firmware Update         →
Wifi Network             Firmware Update      →      Wifi firmware Update    →
Language          →      Wifi firmware Update →      Restore Defaults
```

3.4.1. Temperature

This option manually sets the temperature of the heated bed or the nozzle.

Select and click the 'Temperature' option to access the option. Select the nozzle or bed to be heated and click the button to confirm.

```
Main              ↱      Control              ↱
Temperature       →      Nozzle:              0
Prode Z Offset    →      Bed:                 0
Wifi Network
Language          →
```

Turn the control knob to reach the required temperature and press the control knob to confirm the setting. As the 3D printer starts to heat up, the real-time temperature of the nozzle or the bed will be displayed on the home page.

If other operations are to be performed during this time, click the control knob to return to the information screen. The heating process will continue in the background.

To stop the 3D printer preheating, enter the preheating menu again and turn the control knob until the temperature shows zero, and click to confirm.

Nozzle: 50	Bed: 30
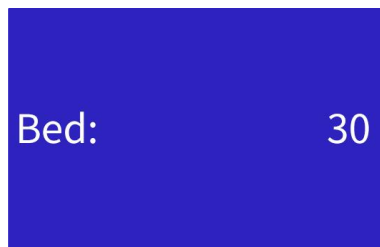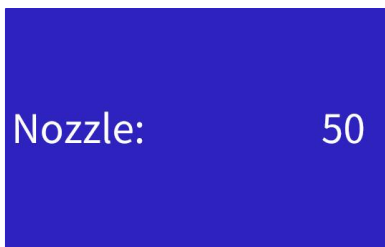
3.4.2.Prode Z Offset

The gap between the platform and the nozzle has to be set after the bed has been leveled. If the gap is too big, the printed model won't adhere to the platform well, and this may lead to the model separating from the platform.

If the gap is too small, it will cause the nozzle to clog, or the print head will scratch the platform.

The Z offset should always be recalibrated after the proximity switch or the nozzle has been replaced.

Select and click the 'Prode Z Offset' option to enter the page.

Main ⤒ Temperature → Prode Z Offset → Wifi Network Language →	Adjust Z Offset(mm): 00.30

Turn the control knob to adjust the height of the printing position down and up. Press the control knob to confirm the setting.

3.4.3.WIFI Network

Main ⤒ Temperature → Prode Z Offset → Wifi Network → Language →

Press the control knob to open the submenu.

Control ⤒ Wifi Info → Wifi QR Code Wifi Reset Wifi OTA Update →
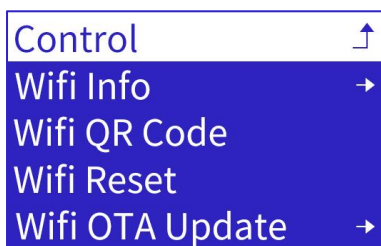
The 3D printer's network function has two connection modes, remote and local.

Remote mode: Use the PoloPrint Cloud (see the Quick Start guide that came with your package) App to control the 3D printer remotely or to download the print model. The remote mode will enable you to control the 3D printer remotely from any location. To enable remote mode, connect the 3D printer to the WiFi network and register it with the App.

Local mode: Cura or Wiibuilder on a computer can connect to the 3D printer on the same local area network to send print files or control the printer remotely. The 3D printer should be connected to the local WiFi network to use local mode.

Note: A 3D printer can only be registered on one app account.

3.4.3.1.WIFI Info

Select and click the 'WiFi Info' option to open the WiFi info screen. It will show the WiFi firmware version number, serial number, IP address, AP name, and connection status to the cloud server. The 3D printer will only connect to the cloud server after it has been registered on the App.

3.4.3.2.WIFI QR Code

Select and click the 'WiFi QR Code' option to show the QR code generated by the printer. The QR code is used to scan for the App and cloud access.
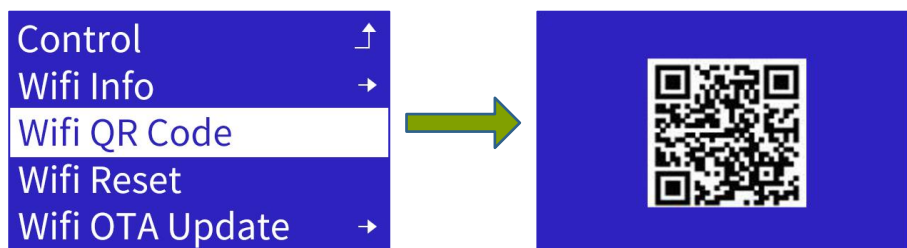
The picture above is only an example. Please scan the QR code on the printer's screen.

3.4.3.3.WIFI Reset

The 3D printer will automatically connect to the WiFi network that it was previously connected to whenever it is turned on. If the 3D printer has to be connected to a new WiFi network, the reset WiFi setting function must be performed to clear the saved WiFi network information.

14
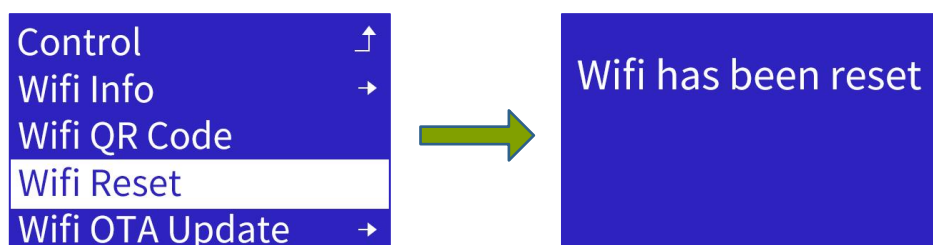
3.4.4. Language

Select and click the 'Language' option to display the list of languages available: English, Chinese, French, German, Italian, Japanese, Portugal, Dutch, and Turkish.

Turn the control knob to select the language you want to use and press it to save.

```
Main              ⬑          Control            ⬑
Temperature       →          English
Prode Z Offset    →          Chinese
Wifi Network                 French
Language          →          German

Spanish                      Italian
Italian                      Japanese
Japanese                     Portugal
Portugal                     Dutch
Dutch                        Turkish
```
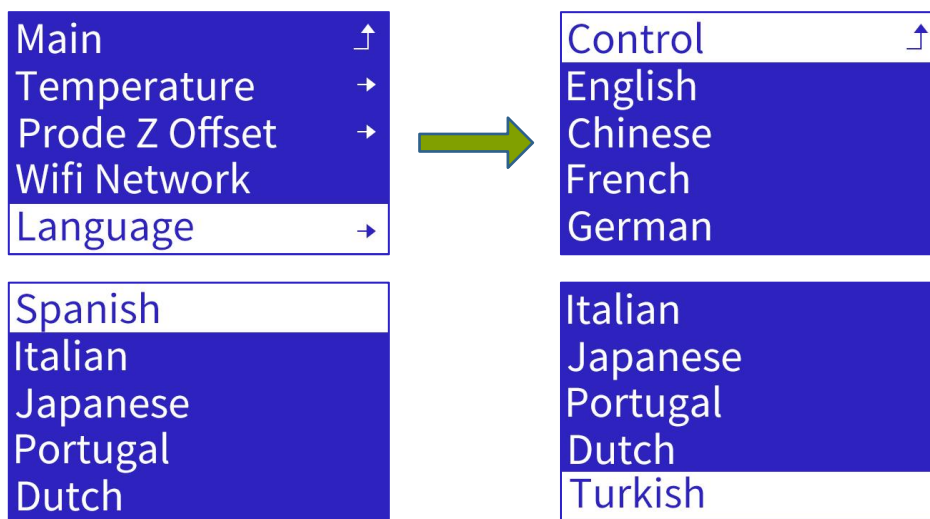
3.4.5. LED Control

Select and click the 'LED Control' option to open the LED menu.

Turn the control knob to select the option you want (On or Off) and press it to save.

```
LED Control        →         LED Control
Quiet Mode         →         On
Printer Info       →         Off
Firmware Update    →
Wifi firmware Update→
```
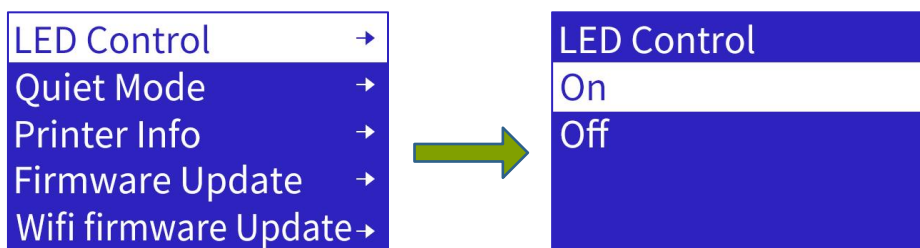
3.4.6. Quiet Mode

Select and click the 'Quiet Mode' option to open the quiet mode switch menu. With the Quiet Mode On, it will reduce the print speed significantly to limit the noise generated.

Turn the control knob to select the option you want (On or Off) and press it to save.

```
LED Control        →         Quiet Mode
Quiet Mode         →         On
Printer Info       →         Off
Firmware Update    →
Wifi firmware Update→
```
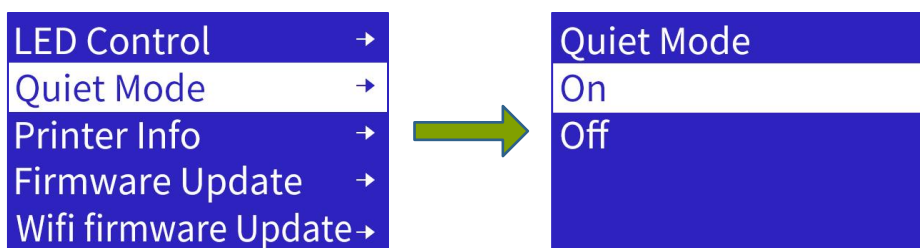
3.4.7.Printer Info

Select and click the 'Printer Info' option to view the firmware version of the 3D printer.

LED Control	→		Printer Info
Quiet Mode	→		Machine：TINA2S
Printer Info	→	→	Version：1.1.5
Firmware Update	→		
Wifi firmware Update	→		
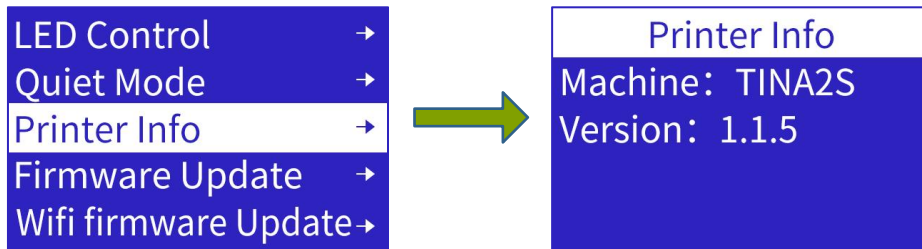
3.4.8.Firmware Update

1. Download or copy the file containing the motherboard firmware to the root directory of the TF card. The firmware file's extension is 'wfm'. If the file is compressed, it should be uncompressed first. Rename the firmware file to 'flash.wfm'.

2. Insert the TF card into the 3D printer and select and click the 'Firmware Update' option.

flash.wfm
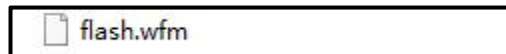
3. The 3D printer will check the firmware file's version, and then update the motherboard firmware if the version in the file is newer.

LED Control	→		Firmware Update
Quiet Mode	→		flash.wfm not found
Printer Info	→	→	
Firmware Update	→		
Wifi firmware Update	→		
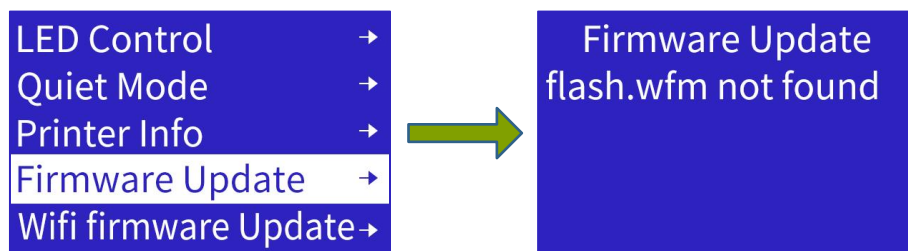
3.4.9.WIFI Firmware Update

1. Download or copy the file containing the WiFi firmware file to the root directory of the TF card. The firmware file's extension is 'efm'. If the file is compressed, it should be uncompressed first. Rename the firmware file to 'WiFi.efm'.

wifi.efm
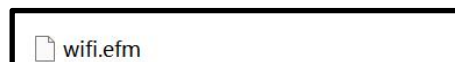
2. Insert the TF card into the 3D printer and select and click the 'WiFi firmware Update' option.

3. The 3D printer will check the version of the firmware file, and then update the WiFi firmware if the version in the file is newer.

LED Control	→		Open sdcard ok
Quiet Mode	→		wifi.efm found
Printer Info	→	→	Upgrade begin...
Firmware Update	→		
Wifi firmware Update	→		
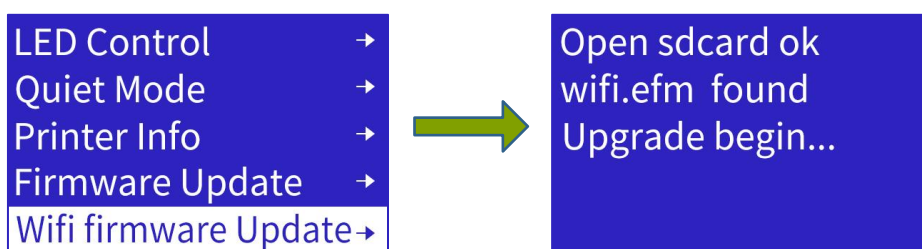
3.4.10.Restore Defaults

Select and click the 'Restore Defaults' option to restore all parameters to the factory default values.

```
Quiet Mode           →
Printer Info         →
Firmware Update      →
Wifi firmware Update →
Restore Defaults
```

3.5. Print from TF (MicroSD)

```
Info screen          ↱
Prepare              →
Control              →
Print from TF
```

Select and click the 'Print from TF' option to show the files in the TF card. The extension of the file is gco or gcode.

```
Main                 ↱
Flexi-Rex-improve...
FRUIT_SL.GCO
228F974B.GCO
DFE92F8B.GCO
```
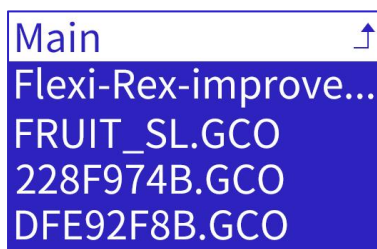
Navigate to a file by turning the control knob and pressing it to select. The next screen will show the printing information of the gcode file. Select the 'Print' option.

The system will check the machine model parameter in the gcode file to prevent damage to the printer. If the machine model parameter does not match the printer or is missing, the gcode file will not print. Use the Wiibuilder or Cura program to select the correct model for printing.

Note: The file name length can't exceed 20 characters, or the system won't recognize it or display random or garbled characters.

```
Print
Quit                 ↱
LayerHeight: 0.2mm
Density: NONE
Filament: NONE
```
→
```
Density: NONE
Filament: NONE
Usage: NONE
Temperature: NONE
PrintTime: 01h53m
```
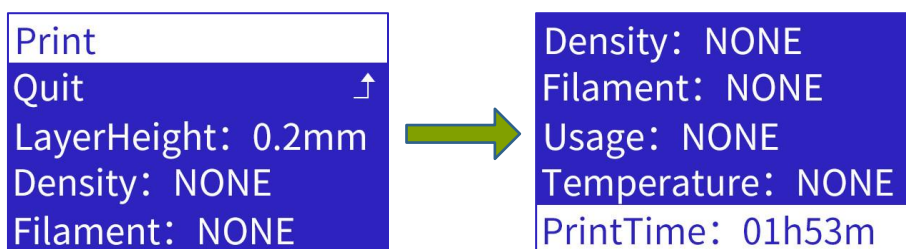
While the machine is printing, you can press on control knob to open the print setting menu (as shown in the next page).

3.5.1.Tune

Select and click the 'Tune' option to open the tune menu. This menu is used to adjust the print speed, nozzle temperature, heated bed temperature, extruder speed, or Z offset during the printing process.
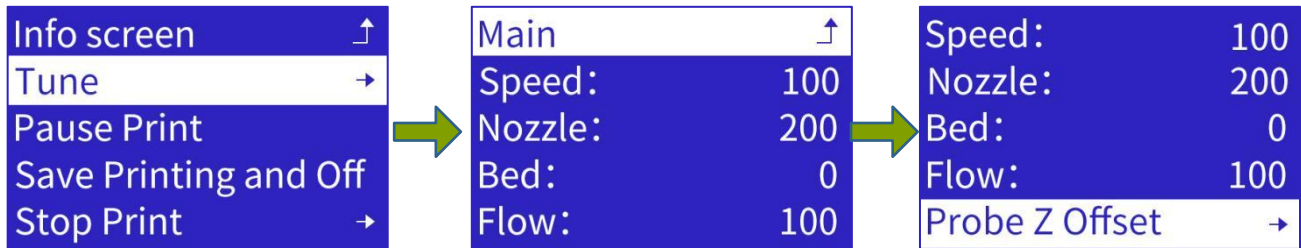
1. Select and click the 'Speed' option, turn the control knob until the desired speed is displayed, and press the knob to confirm.
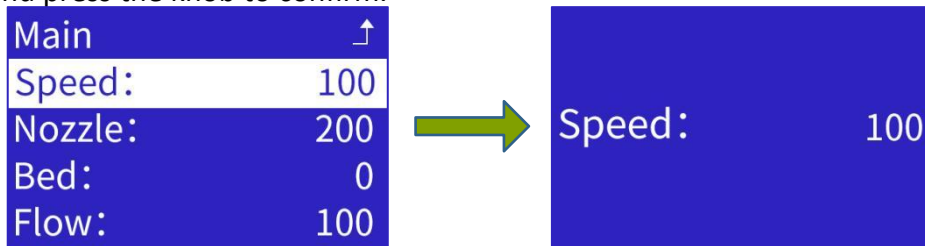
2. Select and click the 'Nozzle' option, turn the control knob until the desired temperature of the nozzle is displayed, and press the knob to confirm.
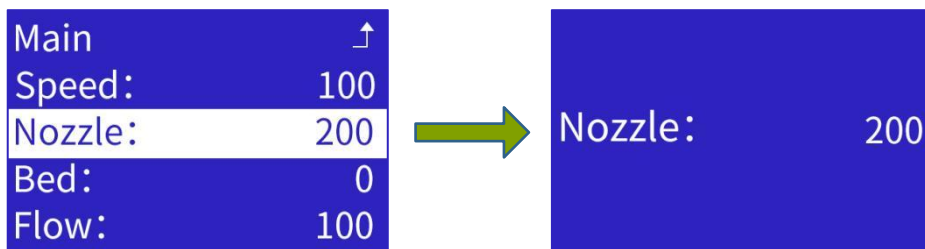
3. Select and click the 'Bed' option, turn the control knob until the desired temperature of the heatbed is displayed, and press the knob to confirm.
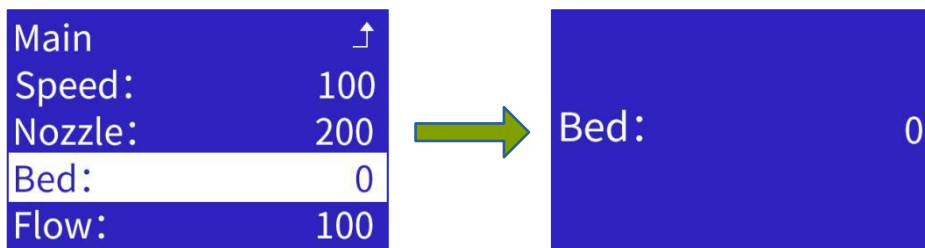
4. Select and click the 'Probe Z Offset' option, turn the control knob until the desired gap between the nozzle and the platform is displayed, and press the knob to confirm.

NOTE: This function can be used to adjust the height of the Z axis in real time to change the gap between the platform and the nozzle during the printing process. When printing starts and it is noticed that the gap between the platform and the nozzle is too small, and the nozzle doesn't produce silk, the z offset should be increased by 0.2mm.

If it is noticed that the gap between the platform and the nozzle is too big and the filament doesn't adhere to the platform well, the z offset can be reduced gradually.
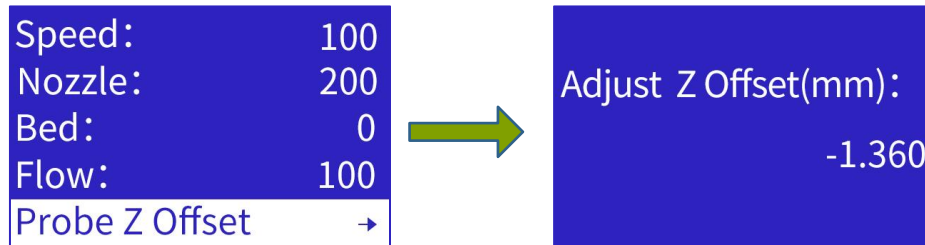
3.5.2.Pause Print

Selecting and clicking the 'Pause Print' option will pause the 3D Printer and the nozzle will move to the pause position. The filament can be replaced in pause mode or printing can be resumed by selecting that option.
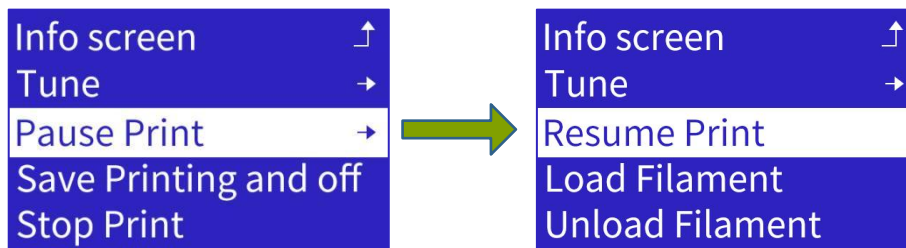
3.5.3. Save Printing and Off

Selecting and clicking the 'Save Printing and off' option will result in the 3D printer saving the current printing progress and moving the nozzle to the pause position. The 3D printer can then be powered off when the 3D printer stops moving. When the 3D printer is powered on again it will prompt whether the printing should be resumed or stopped.

```
Info screen          ⤒          Save Printing and Off          Power Loss Recovery
Tune                 →          Wait for printing             Resume  Print
Pause Print          →          stop, then turn off           Stop  Print
Save Printing and off           the power
Stop Print
```
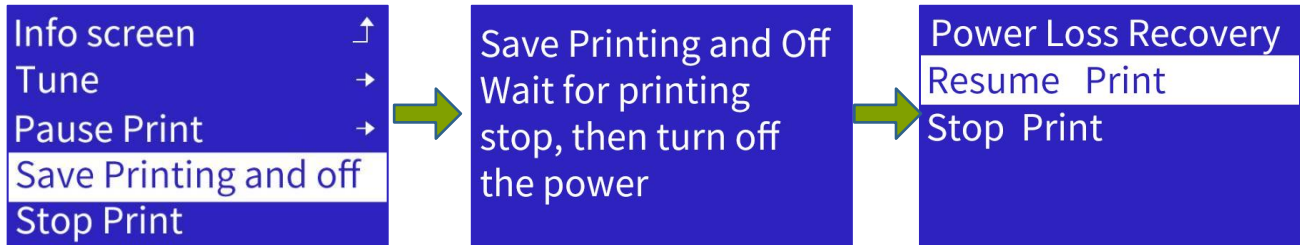
NOTE:

When either the 'Pause Print' or 'Save Printing and off' option has been selected, the 3D printer will move to a home position. There will be slight deviations for each homing operation due to the mechanical nature of the endstop. This will result in a seam or slight deviation in the position when printing is continued. It is therefore recommended to print a model without pausing whenever possible.

3.5.4. Stop Print

Selecting and clicking the 'Stop Print' option will cancel the current printing job and the operating panel will return to the Info screen.

```
Info screen          ⤒          200° TINA2    40°
Tune                 →          150°          20°
Pause Print          →          X  100  Y    ?    Z    100
Save Printing and off           FR 100%      00h00m  →
Stop Print                      192.168.10.11
```
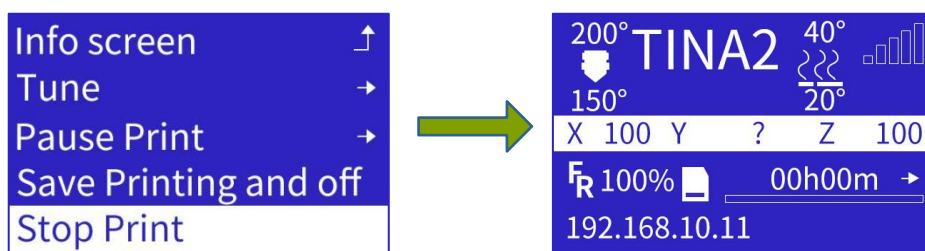
NOTE: Always remove the model from the platform after printing has stopped

4. COMMON MENU OPERATIONS

4.1. Load Filament

Info screen ⤒	Main ⤒	Load Filament
Prepare →	**Load Filament**	Heating nozzle
Control →	Unload Filament	Please wait...
Print from TF	Move axis →	
	Auto home	Nozzle: E1 185/210

1. From the main menu, select the prepare menu.

2. Turn the control knob to select the 'Load Filament' option and click. The printer will first heat the nozzle and then feed the filament

.

4.2. Unload Filament

Info screen ⤒	Main ⤒	Unload Filament
Prepare →	Load Filament	Heating nozzle
Control →	**Unload Filament**	Please wait...
Print from TF	Move axis →	
	Auto home	Nozzle: E1 185/210

1. From the main menu, select the prepare menu.

2. Turn the control knob to select the 'Unload Filament' option and click. The printer will first heat the nozzle and then retract the filament.

4.3. Print from TF Card (MicroSD)

0° 🖨 TINA2 0° ✖📶	Info screen ⤒
30° 30°	Prepare →
X 100 Y ? Z 100	Control →
F R 100% 📄 ___ 00h00m →	**Print from TF**
OFF	

Main ⤒	Print	0° 🖨 TINA2 0° ✖📶
dragon.gcode	Quit ⤒	30° 32°
FRUIT_SL.gcode	LayerHeight: 0.2mm	X 53 Y ? Z 0.35
228F974B.gcode	Density: NONE	F R 100% 📄 ___ 01h51m →
DFE92F8B.gcode	Filament: NONE	dragon.gcode
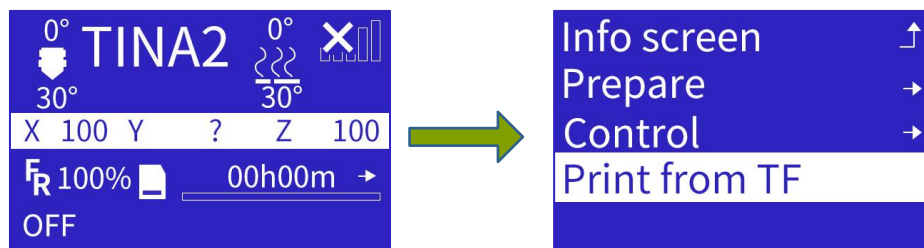
1. From the main menu, turn the control knob to select the 'Print From TF' option and click to open the TF card file list menu..

2. Turn the control knob to select the file to print and press it to start printing

4.4. Common setting during printing

4.4.1.Pause Print

While the printer is printing, pressing the control knob to open the printing setting menu and selecting the 'Pause Print' option will pause the printing. The nozzle will move to the pause position, allowing you to replace the filament, or resume printing.

```
  0°  TINA2   0°  🗙📶
 30°          32°
 X  53  Y    ?    Z  0.35
 FR 100% 💾 ___ 01h51m →
 dragon.gcode
```
→
```
Info screen          ⬏
Tune                 →
Resume Print
Load Filament
Unload Filament
```
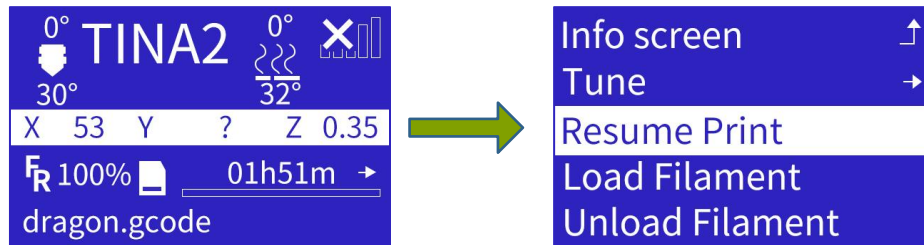
4.4.2.Modify Temperature

While the printer is printing, press the control knob to open the printing setting menu. Select and click the tune menu option and then select the 'Nozzle' option. Turn the control knob to decrease or increase the temperature and press the control knob to accept the setting.

```
  0°  TINA2   0°  🗙📶
 30°          32°
 X  53  Y    ?    Z  0.35
 FR 100% 💾 ___ 01h51m →
 dragon.gcode
```
→
```
Info screen          ⬏
Tune                 →
Pause Print
Save Printing and Off
Stop Print           →
```

→
```
Main                 ⬏
Speed:           100
Nozzle:          200
Bed:               0
Flow:            100
```
→
```
Nozzle:          200
```
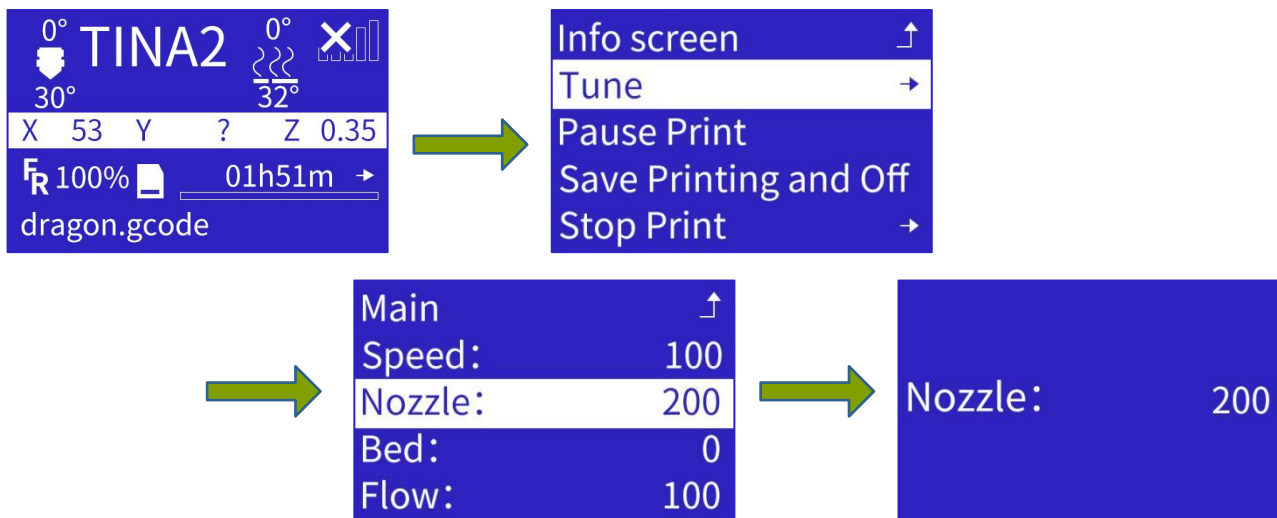
4.4.3.Modify Z Offset

While the printer is printing, press the control knob to open the printing setting menu. Select and click the tune menu option and then select the 'Probe Z Offset' option. Turn the control knob to fine-tune the gap between the nozzle and the platform and press the control knob to accept the setting.

```
  0°  TINA2   0°  🗙📶
 30°          32°
 X  53  Y    ?    Z  0.35
 FR 100% 💾 ___ 01h51m →
 dragon.gcode
```
→
```
Info screen          ⬏
Tune                 →
Pause Print
Save Printing and Off
Stop Print           →
```

```
Speed:          100
Nozzle:         200
Bed:              0
Flow:           100
Probe Z Offset    →
```

```
Adjust Z Offset(mm):
                -1.360
```
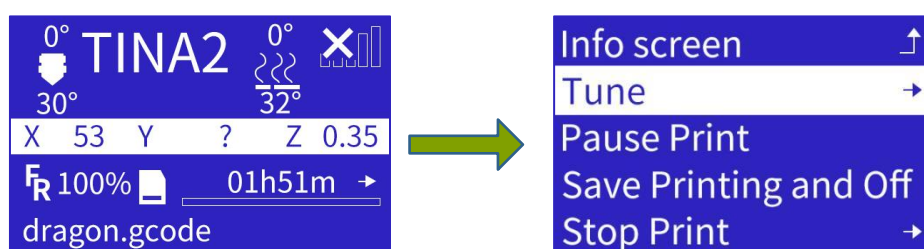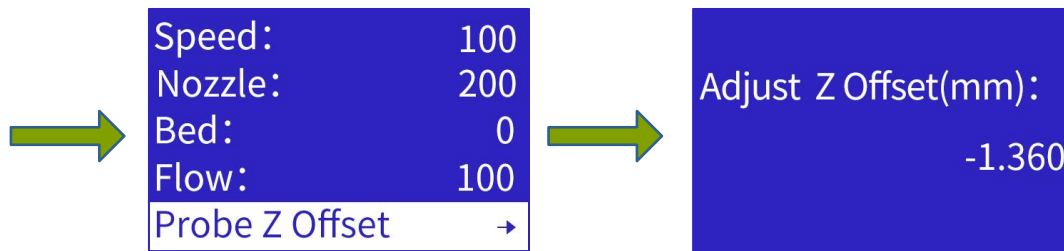
NOTE: This option adjusts the Z axis height in real time to modify the gap between the platform and the nozzle while printing.

If you see that the gap between the nozzle and the platform is too small at the start of printing, and the nozzle doesn't produce silk, increase the z offset by 0.2mm.

If you see that the gap between the nozzle and the platform is too big and the filament doesn't adhere to the platform properly, gradually reduce the z offset.

4.5. Power On Resume Print

The 3d printer will save the printing progress every 10 seconds during printing. If power loss occurs unexpectedly, the printer will ask whether it should resume printing when power is applied again.

NOTE: The printing progress is written into the '.bin' file of the TF card and the previous progress is deleted. When power is lost and reapplied, the printer will check if there is a '.bin' file on the TF card. If there is, it will prompt 'Continue printing and Stop printing'.

If continue printing is selected, the printer will read the saved progress and resume printing. If Stop printing is selected, the printer will clear the '.bin' file.

As the print progress is only saved every 10 seconds, there is a possibility that the power-off replay may fail.

5. WIFI NETWORK

The 3D printer can be connected to the network using one of the two methods described below.

Please contact technical support if you encounter any problems.

5.1. File connect network

1. Use a computer connected to a TF card reader to read the TF card. Open the 'WiFi.gcode' file and modify the string after the W300 command to the WiFi network name Also modify the string after the W301 command to the password of the WiFi network. Save the file and exit.

NOTE: there must be a space after w300 and w301 (see example below)

2. Insert the TF card into the 3D printer and select to print the 'WiFi.gcode' file. The info screen should show an IP address after a few seconds, indicating that the network configuration has been successful.

5.2. EspBluFi connect network

1. Download and install EspBluFi.
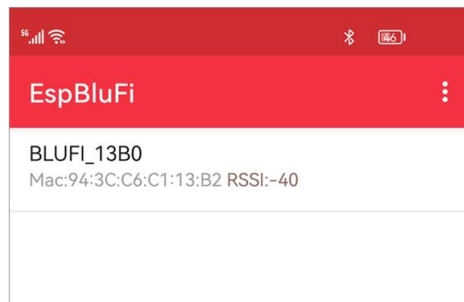
Note: Bluetooth must be turned on.

2. Open 'WiFi Info' and make a note of the SN's last four digits.

.
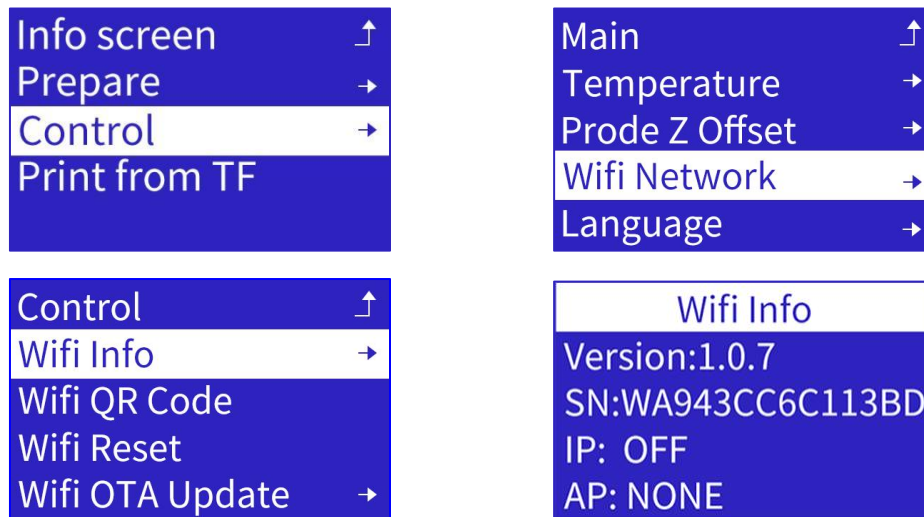
3. Select the printer in the EspBulFi program. Click on 'Connect' and then 'Configuration'.
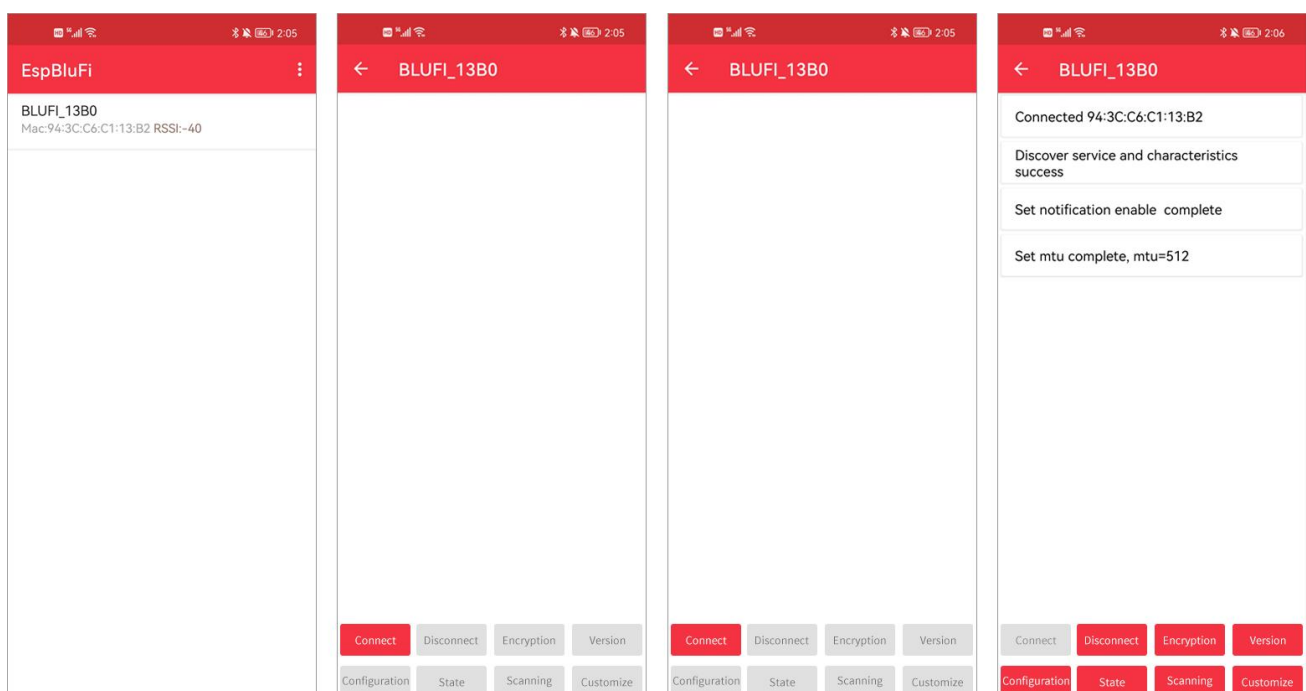
4. Click on the "   " Icon, select the 2.4G network, and then enter the password.
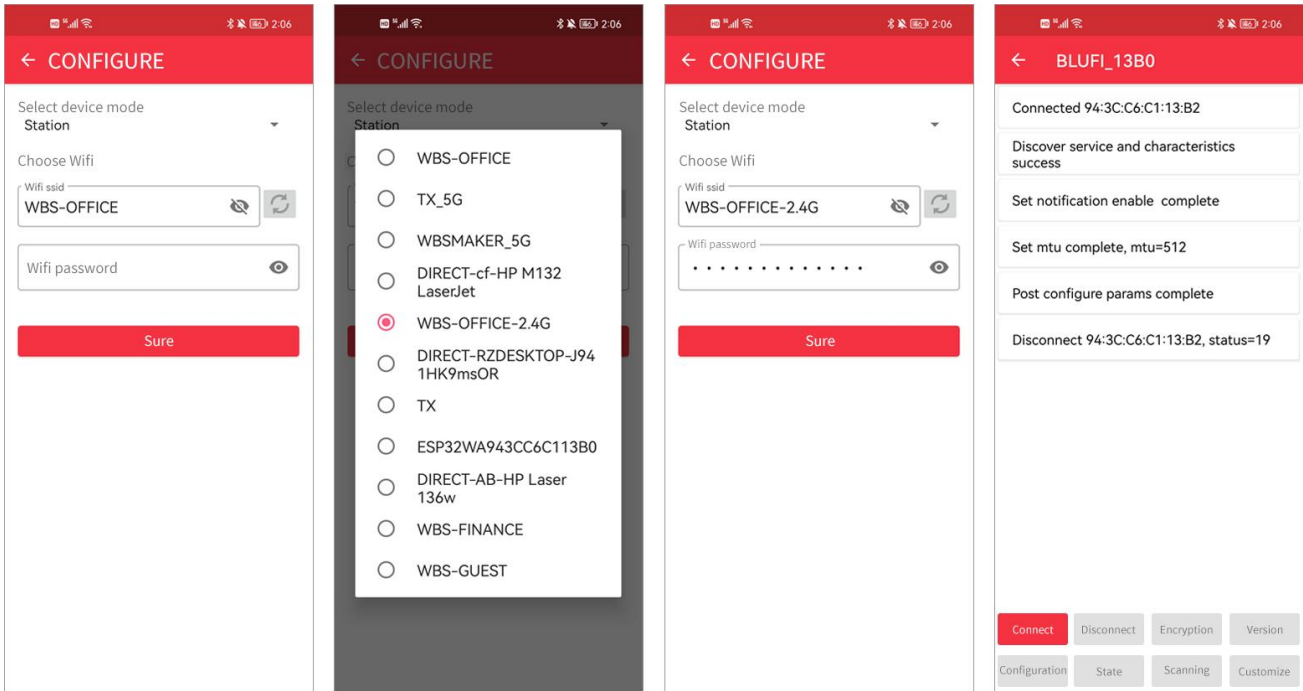
5. If the 3D printer displays the IP address after it has been restarted, the network configuration has been successful.

6. WIIBUILDER SLICING SOFTWARE

The 3D printer includes the Wiibuilder slicing software on the TF card included in the package. Use a computer connected to a card reader to display the contents of the TF card.

Configuration requirements to run Wiibuilder:
- **CPU: Pentium 1GHz**
- **Memory: minimum 1Gb**
- **Hard disk: more than 1Gb disk space**
- **Display resolution: minimum 640x480 pixels**
- **Graphics card: support for 3D acceleration & OpenGL2.0 and above**
- **Color: minimum 256 colors**
- **Operating system: Windows7 and above / MACOS8.0 and above**
- **Other runtime libraries: .Net Framework 4.5.2 and above**

6.1. Installation

Locate and run the Wiibuilder.exe file on the SD card and run it to install the Wiibuilder software program. Click the Install, Next, and Finish buttons at the appropriate times.
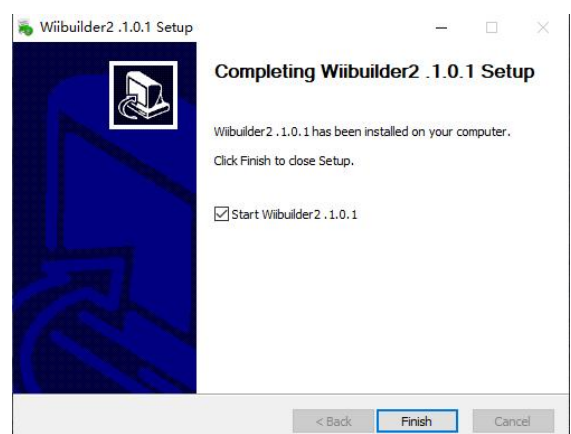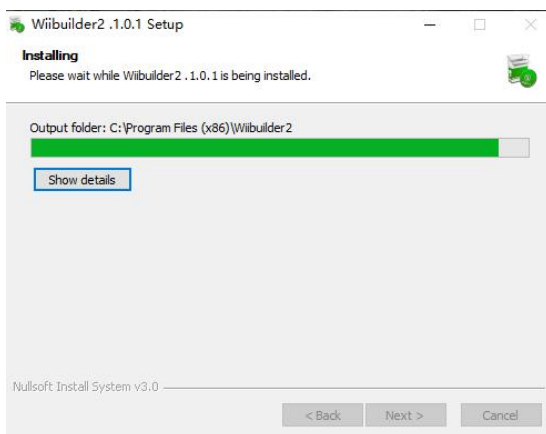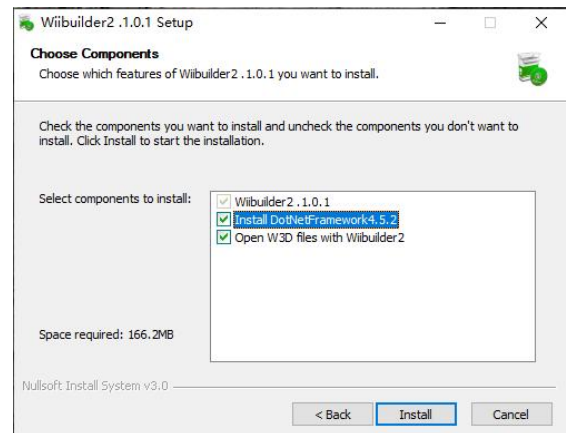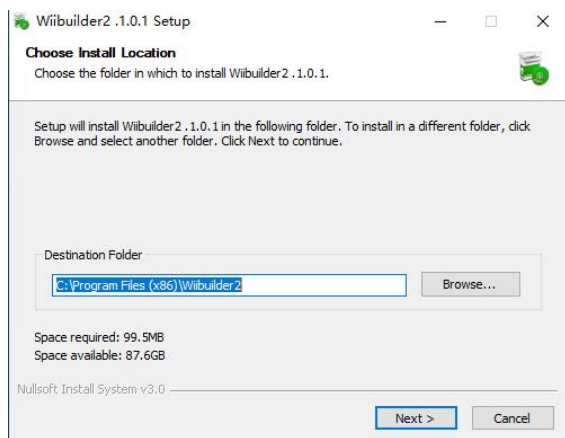
6.2. Interface Introduction
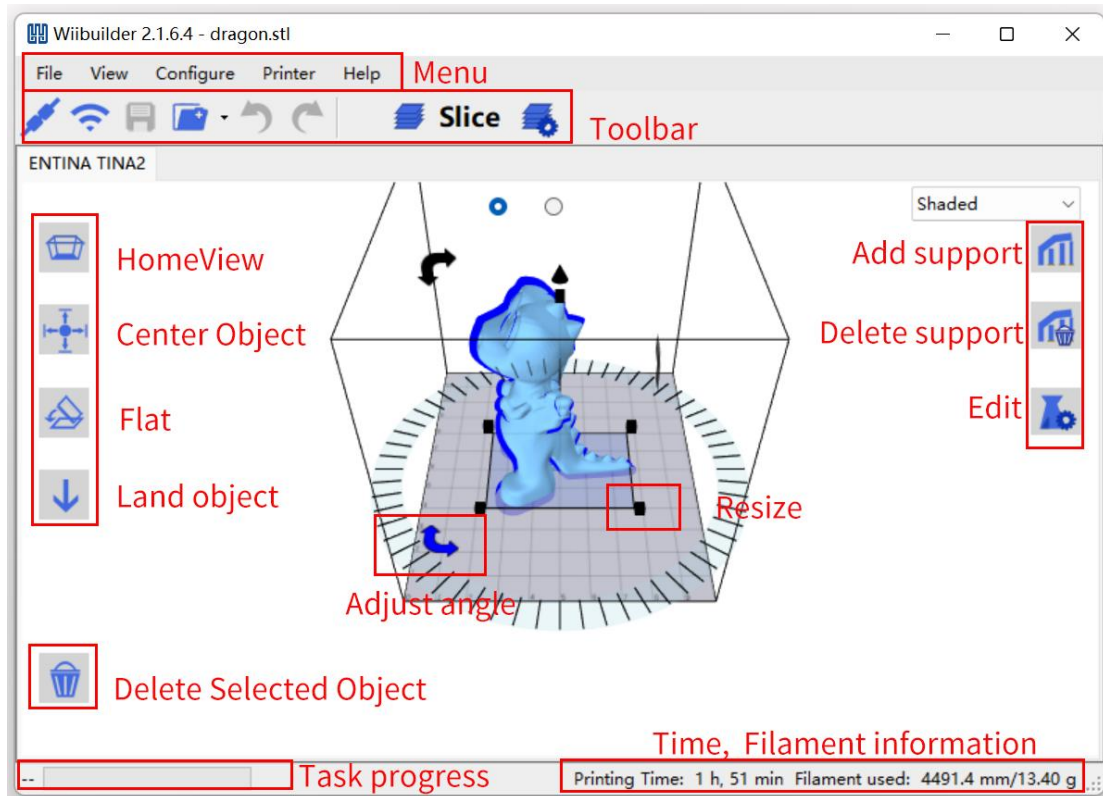
- **Menu:** Opens various Wiibuilder options.

- **Toolbar:** USB connection, WiFi connection, Save gcode File, Load Model, Undo, Redo, Execute Slice, and Slice Settings.

- **HomeView:** Restores the default viewing angle.

- **Center Object:** Automatically adjust the X and Y coordinates of the model and place it in the middle of the platform.

- **Flat:** Automatically adjusts the inclination of the model and flattens the inclined model on the platform.

- **Land object:** Automatically adjusts the Z coordinate of the model to align the bottom of the model with the platform.

- **Delete Selected Object:** Delete the model that is currently selected.

- **Add Support:** Adds a manual support to the model.

- **Delete Support:** Deletes the manual support that has been added.

- **Edit:** Open and edit the model's gcode file.

Right clicking on the model will open the context menu:

- ■ **Copy:** Copies the model.

- ■ **Mirror:** Mirrors the model along the X, Y, and Z axis directions.

- ■ **Delete:** Deletes the current model

- ■ **Export:** Exports the current model to an stl file.

- ■ **Arrange All Parts:** Adjusts the spacing and position of multiple models on the platform automatically.

- ■ **Clear Plate:** Deletes all models on the platform.
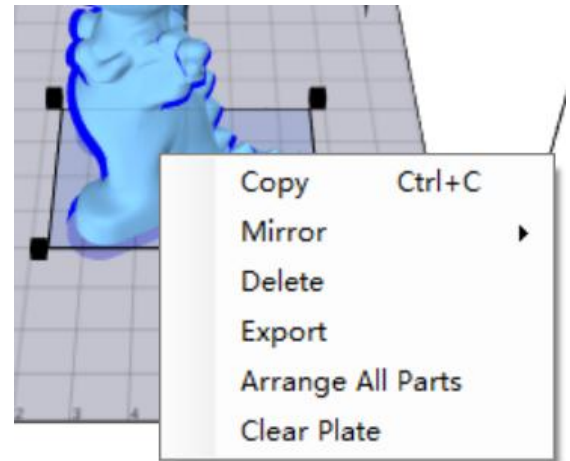
Slicing Settings:
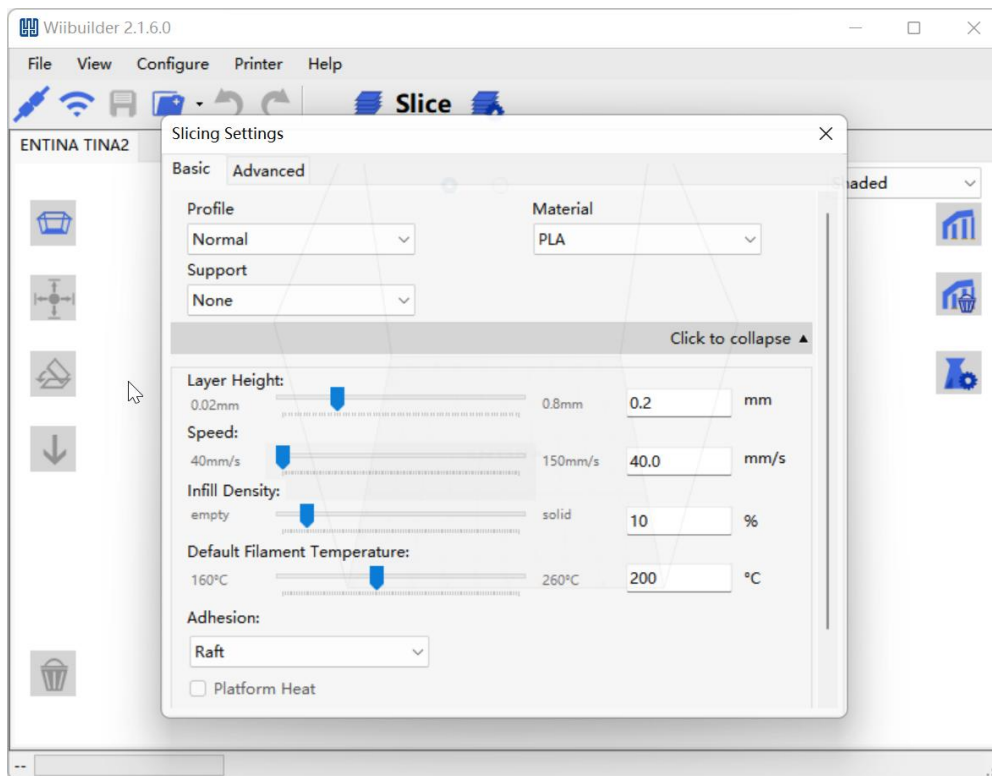
6.3. Add Model

1. Open Wiibuilder and select the 'File' menu to load the model, or just drag the model into it.
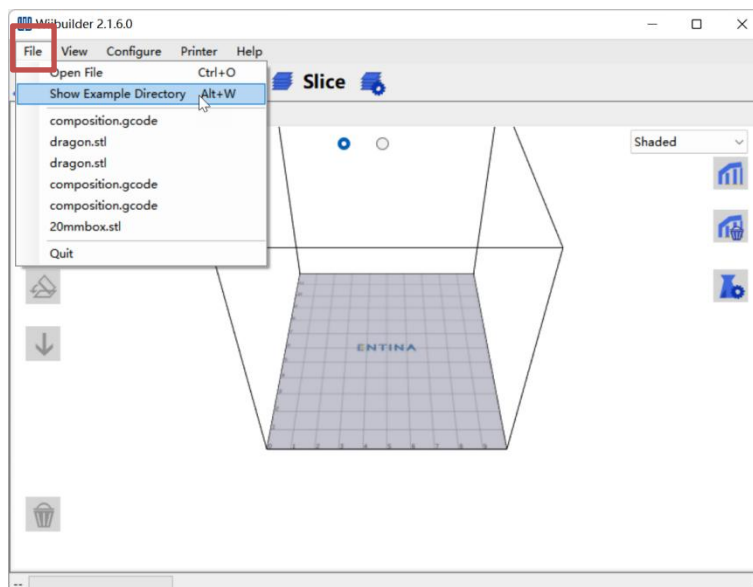
2. There are some models stored in the 'Show Example Directory' which can be used for test printing.
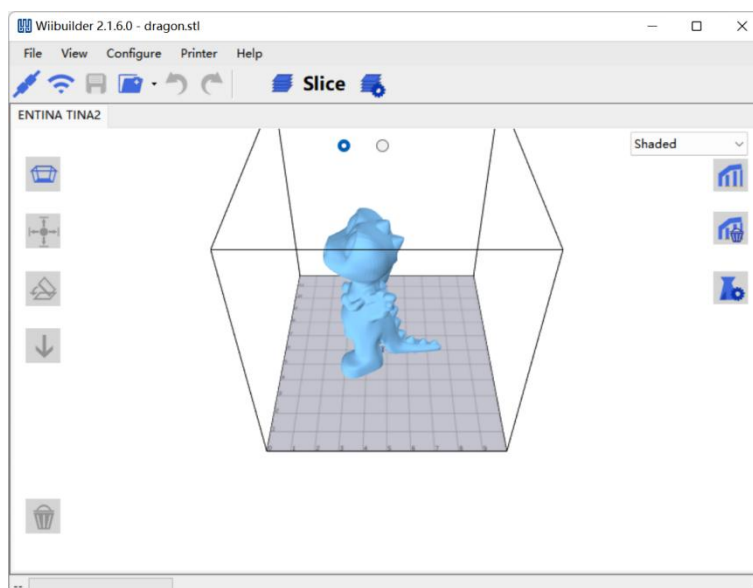
6.4. Adjusting The Model

1. Click on the model to adjust its position, angle, and size.
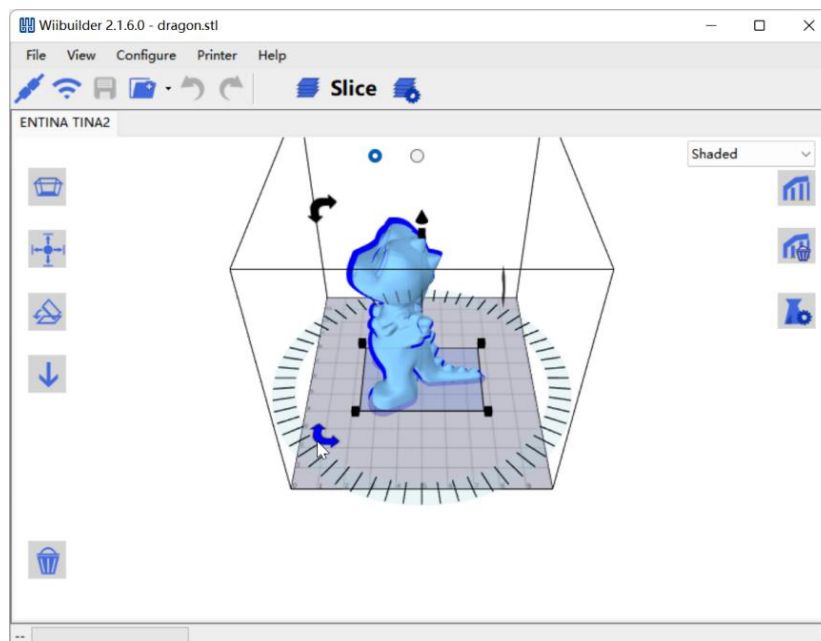
2. The model has to be correctly placed and centered on the platform. To do this, use the 'Center Object', 'Flat', and 'Land object' options.
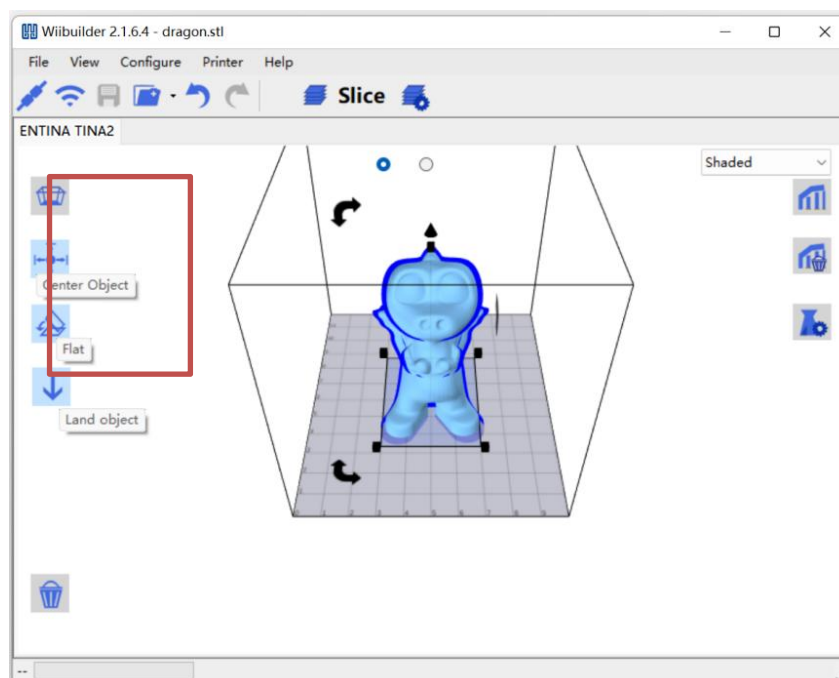
6.5. Basic Parameter Settings

Novice users can use the default parameters as they are without having to adjust them.
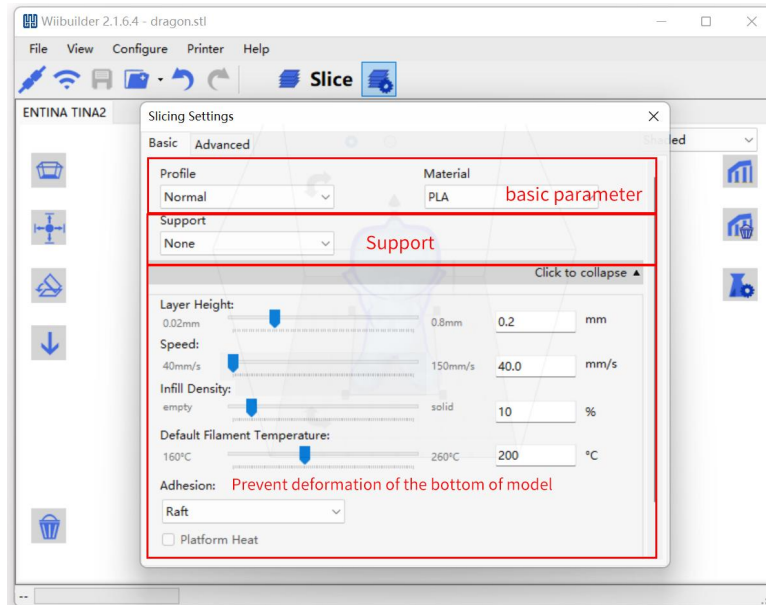
After the parameters have been set, click the 'Slice' option to convert the file.
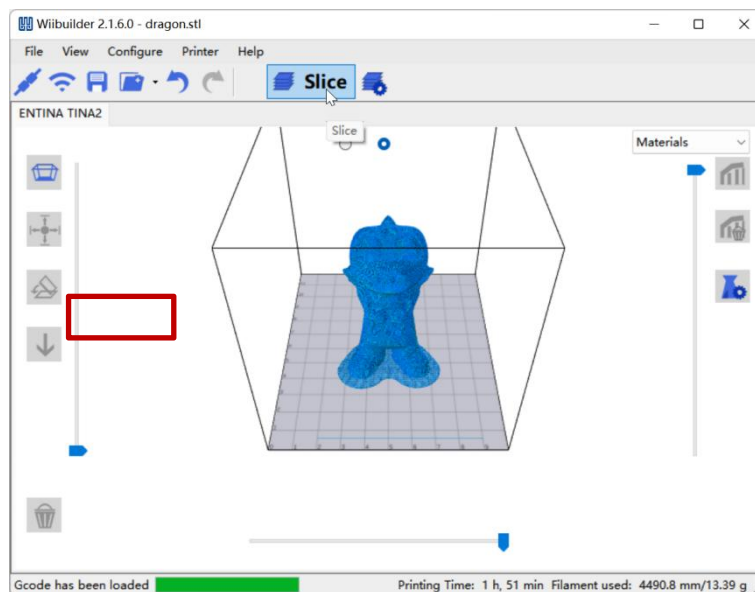
6.6. Send Files by The Network

NOTE: The 3D printer must be connected to the network with its IP address displayed on the screen.

For more details, refer to: '5. WiFi network'.

The IP address must be showing ➡

1. The computer should connect to the same local network as the 3D printer.
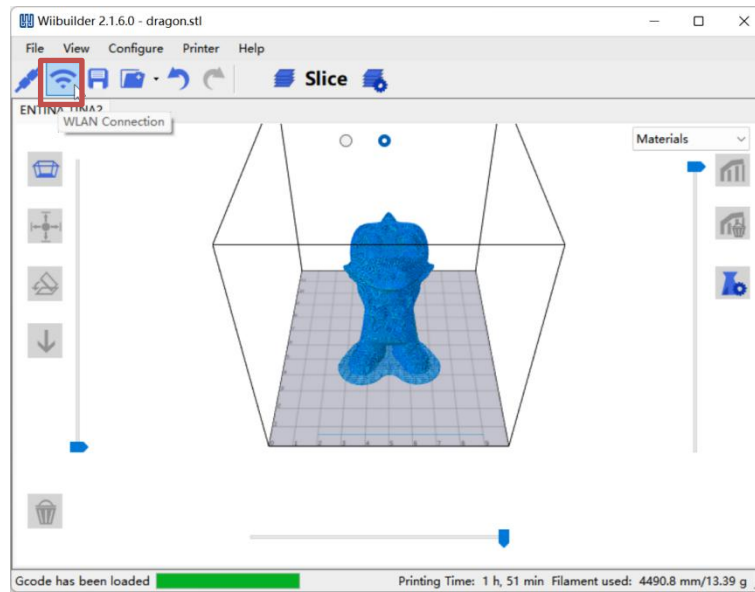
2. When clicking 'WLAN Connection', the software will search for 3D printers in the local network automatically and show them in a list. If no 3D printer is found, you can try to enter the IP address of the 3D printer manually to connect.
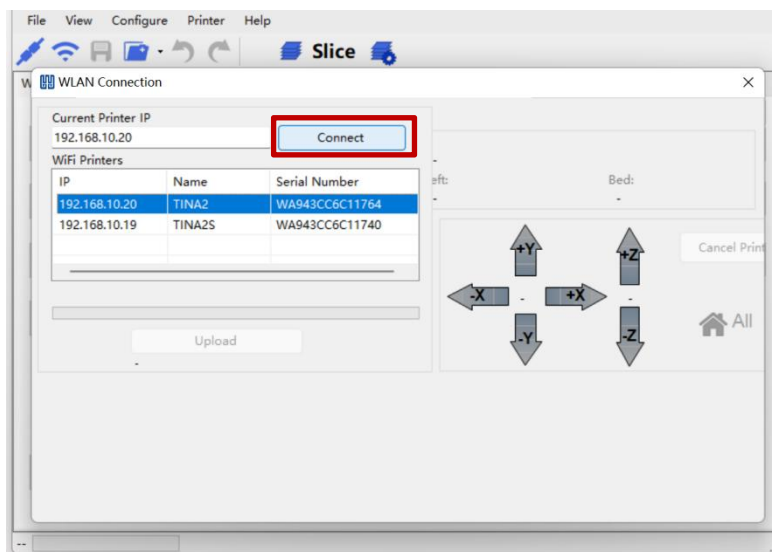
NOTE: If you are unable to connect to the 3D printer, check to ensure that a firewall on the computer doesn't prevent the software from networking. Also check that the 3D printer and the computer are on the same local area network.

3. Send the sliced model to the 3D printer via the network. The printer will start printing after receiving the file.
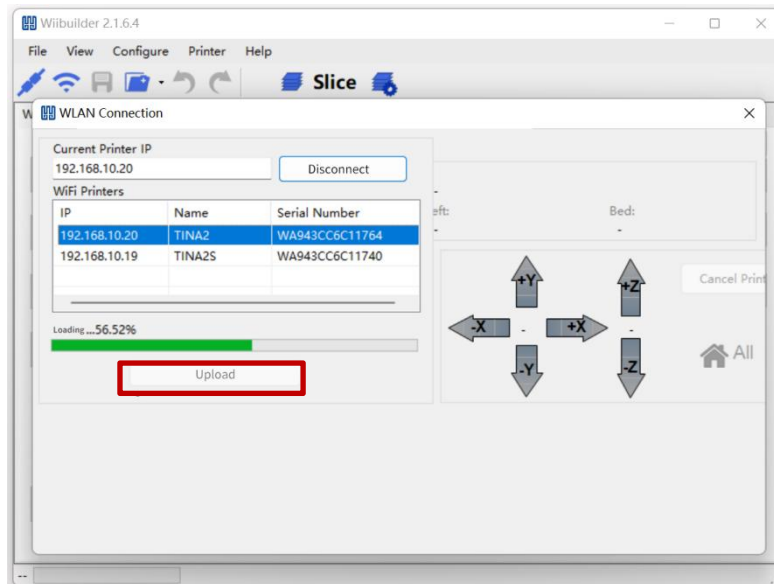
6.7. Print Files from the TF Card

Save the sliced model to the TF card, insert it into the 3D printer, and print.
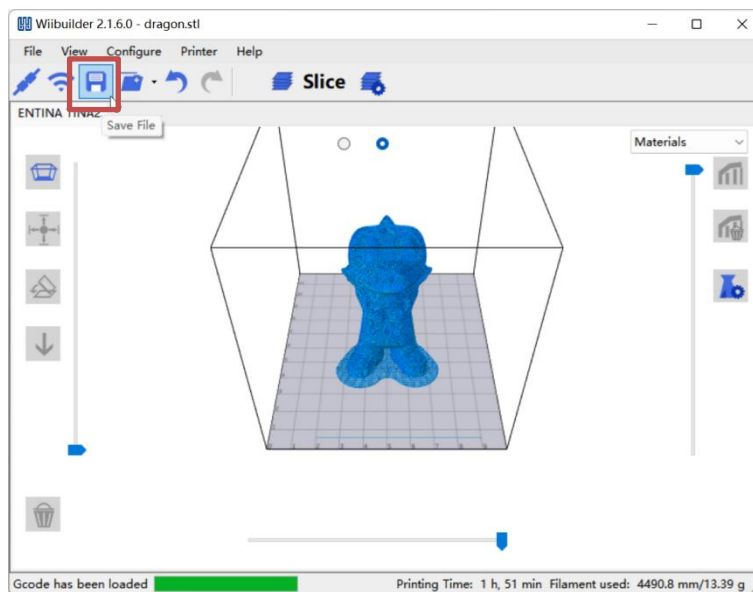
6.8. Print Files via USB

1. Connect the 3D printer to a computer with a USB cable. Note: The CH340 driver should be installed before this option will work. Click on the 'Connect Printer' option.
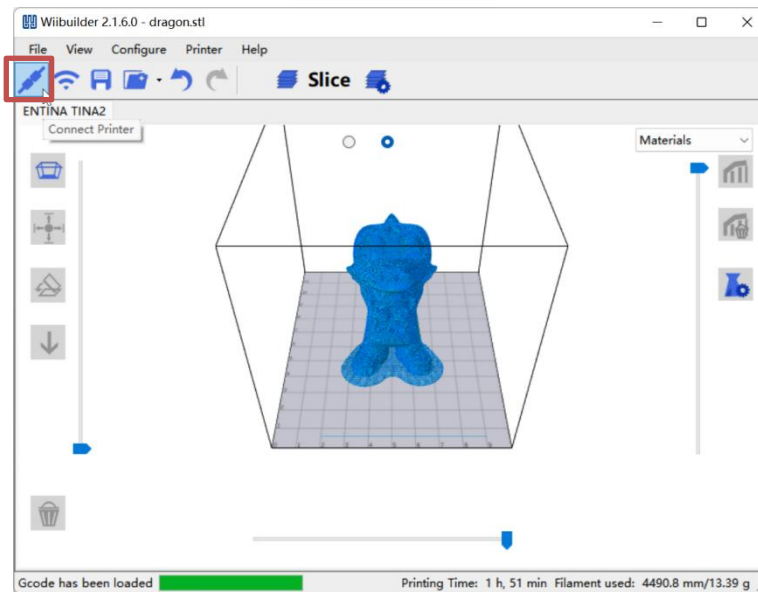
2. Click on 'Connect' and then on 'Upload And Print' to start printing the model.
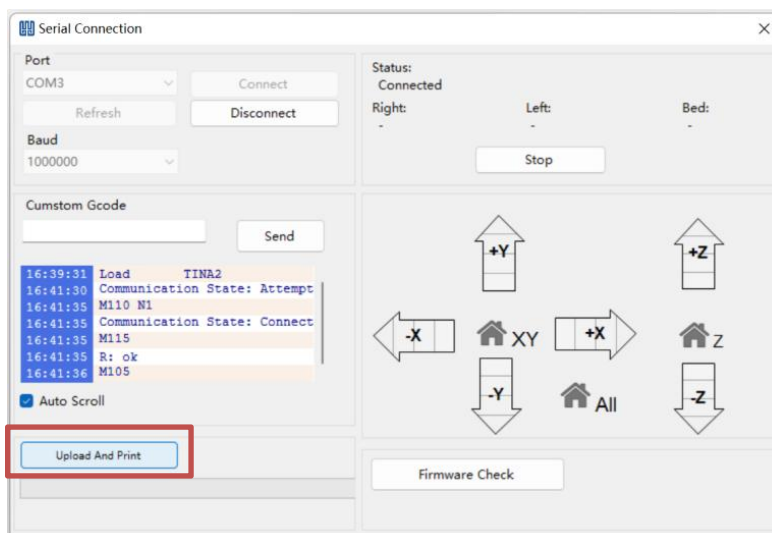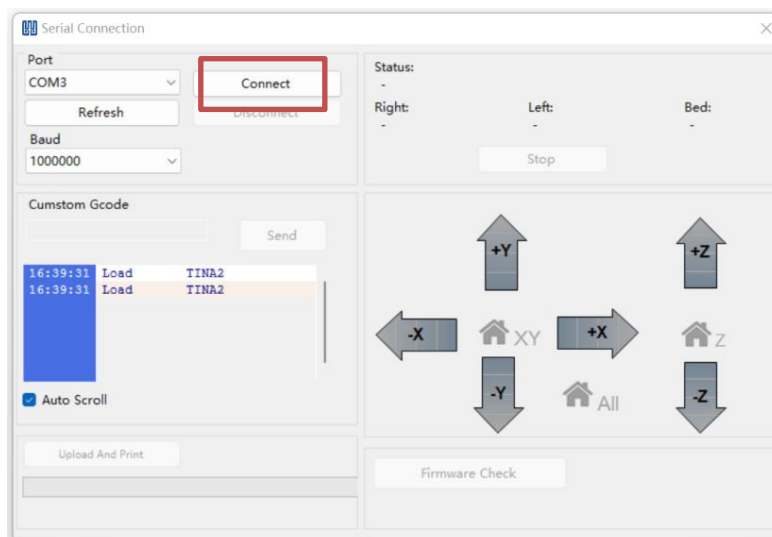
6.9. Advanced Parameter Settings

6.9.1. Speed Tab

The **Speed Tab** contains the following options:

- **Top/Bottom Speed (mm/s):** Set the printing speed for surfaces at the top and bottom of the model.
- **Outer shell speed (mm/s):** Set the printing speed for the surfaces of the external shell.
- **Inner shell speed (mm/s):** Set the printing speed for the surfaces of the internal shell.
- **Infill Speed (mm/s):** Set the printing speed for the infill in the interior of the model.
- **Support Infill Speed (mm/s):** Set the printing speed for the infill in the interior of the model's supports.
- **Support Interface Speed (mm/s):** Set the printing speed for the model supports' top and bottom surfaces.
- **Initial Layer Speed (mm/s):** Set the printing speed of the model's first layer.
- **Travel speed (mm/s):** Set the movement speed of the nozzle when the printer is not printing.

6.9.2.Infill Tab

- ■ **Infill Pattern:** Use the drop-down menu to select one of the 7 different infill patterns available. These include Lines, Grid, Triangles, Zig Zag, Concentric, Cross, and Octet. The details of the individual patterns are shown in the table below.

- ■ **Infill Before Walls:** This box should be checked to print the model after filling and the outline has been printed.

- ■ **Outer Before Inner Walls:** This box should be checked to print the exterior walls before the interior walls have been printed.

Infill Patterns	
Lines	**Grid**
Triangle	**Zig Zag**

Concentric	**Cross**
Octet	
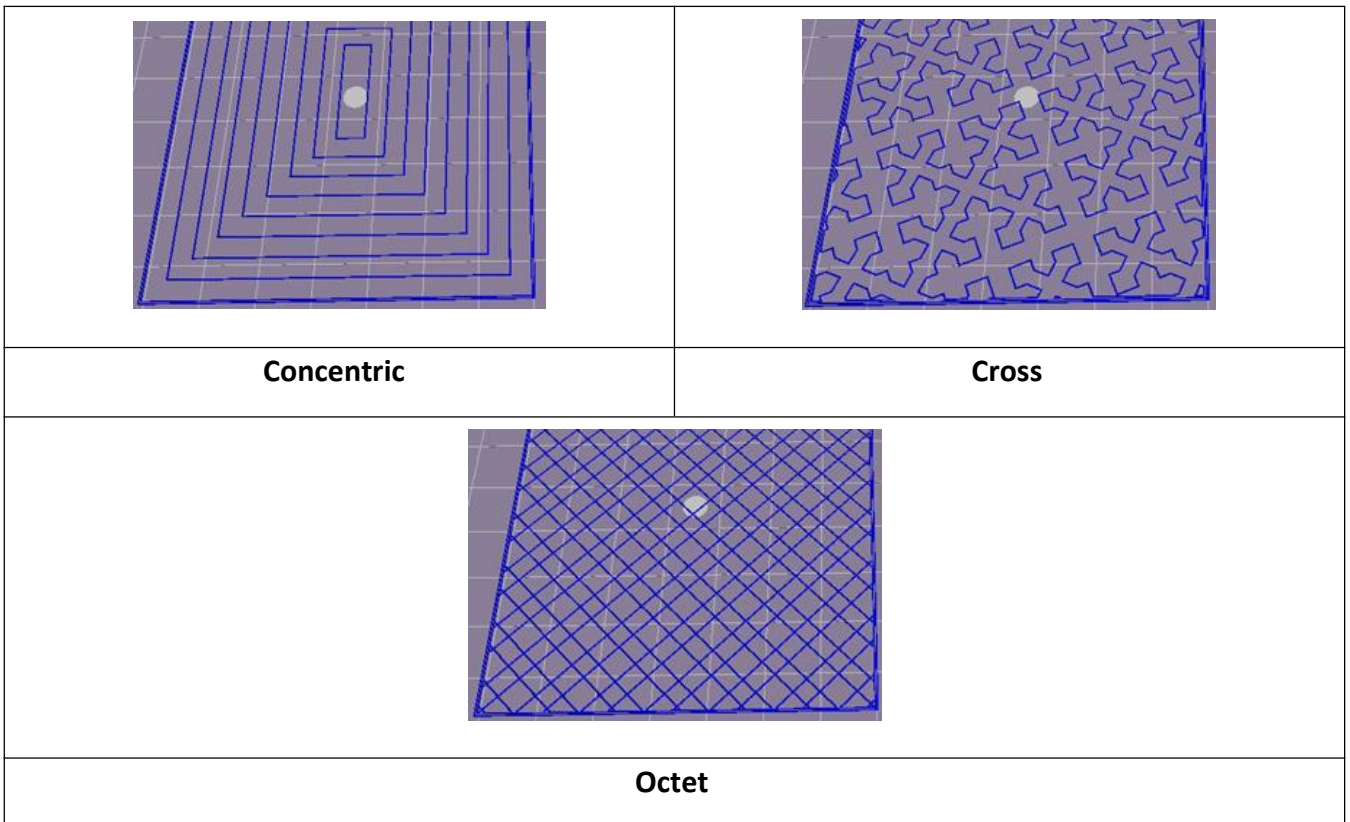
6.9.3. Support Tab
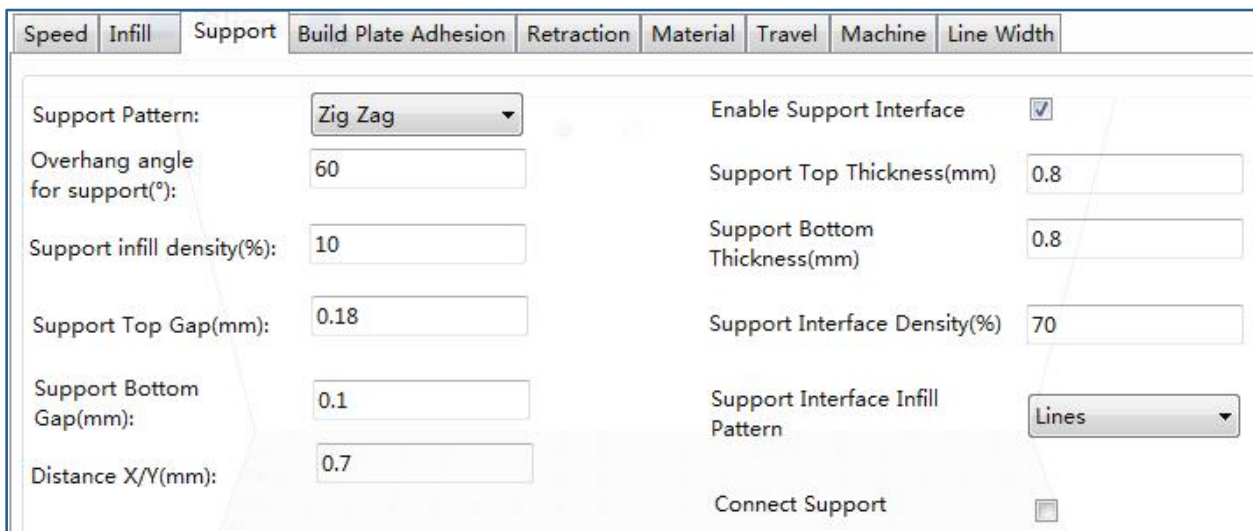
- **Support Pattern:** Use the drop-down menu to select one of the 5 support patterns available. These include Lines, Grid, Triangles, Zig Zag, and Concentric. The pattern designs are the same as those of the infill patterns using the same name.

    - **Lines** support is easier to remove and is normally used on models that require more support.

    - **Grid** support is normally used on small models, which need fewer supports.

    - **Zig Zag** support is normally used for models that are especially difficult to remove. It is stronger than Lines support and better than Grid support.

- **Overhang angle:** The overhang angle is the angle between the surface of the model and the support. Smaller settings provide better support, while bigger settings make the supports easier to remove. The default angle is 60°.

38

- **Support infill density (%):** Set the infill density for supports. The higher the density setting, the stronger the supports will be.

- **Support Top Gap (mm):** This is the distance between the model surface and the top of the support. Although a bigger distance will make the support less effective, it will be easier to remove from the model surface, leading to a smoother surface. Although a smaller distance will make the support more effective, it will be more difficult to remove from the model surface, leading to residual material being left on the model surface.

- **Support Bottom Gap (mm):** This is the distance between the model surface and the bottom of the support. This parameter's effects will be the same as those of the **Support Top Gap (mm)** setting.

- **Distance X/Y (mm):** This is the distance between the model surface in the horizontal plane and the support. This parameter's effects will be the same as those of the **Support Top Gap (mm)** parameter.

- **Enable Support:** This box should be checked to use supports.

- **Support Top:** This setting determines the thickness of the supports' top layer.

- **Support Bottom:** This setting determines the thickness of the supports' bottom layer.

- **Support Interface:** Set the percentage of infill used inside the supports.

- **Support Interface Infill Pattern:** Use the drop-down menu to select one of the 5 support patterns available. These include Lines, Grid, Triangles, Zig Zag, and Concentric. The pattern designs are the same as those of the infill patterns using the same name.

- **Connect Support:** This box should be checked to connect separate supports into one part.

6.9.4. 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		

- **Raft Air Gap (mm):** This is the distance between the model and the raft. This will influence how difficult it is to remove the raft from the model.

- RAFT

- **Raft Extra Margin (mm):** This is the distance between the model surface and the edge of the raft.

- **Raft Base thickness (mm):** This setting determines the raft's thickness.

- **Initial Layer Z Overlap:** This setting determines the amount of overlap between the model's first and second layers.

- **Brim line amount:** Set the number of ring gaskets that should be added to the model's edge in contact with the build platform.

- BRIM
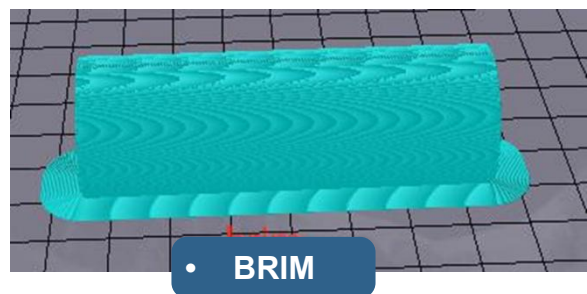
- **Skirt Line Count:** Sets the number of anti-overflow lines at the end of the model in contact with the build platform.

- SKIRT

6.9.5. Retraction Tab

| Speed | Infill | Support | Build Plate Adhesion | Retraction | Material | Travel | Machine | Line Width |

Horizontal Travel Retraction	☑		Z Hop When Retracted	☑
Retract at Layer Change	☐		Z Hop Height(mm):	0
Retraction speed(mm/s):	28			
Retraction distance(mm):	1.2			
Retraction Minimum Travel(mm):	0.8			

- **Horizontal Travel Retraction:** This box should be checked to enable the filament to be retracted when the nozzle isn't printing and is moving horizontally.

- **Retract at Layer Change:** This box should be checked to retract the filament when switching layers.

- **Retraction speed (mm/s):** Set the filament retraction speed.

- **Retraction distance (mm):** Set how far the filament is retracted within the nozzle.

- **Retraction Minimum Travel (mm):** Set the minimum distance of nozzle movement before printing and before filament retraction.

- **Z Hop When Retracted:** This box should be checked to enable the nozzle to lift when the filament is returned after being retracted.

- **Z Hop Height (mm):** Set how far the nozzle should be lifted when the filament is returned after being retracted.

6.9.6. Material Tab

- **Filament flow (%):** Set the filament flow rate in the melting state. This is set based on the type of filament being used. The flow rate for PLA or PLA Pro is generally 90 and for ABS it is 100.

- **Filament Diameter (mm):** Set the diameter of the filament being used. This printer only supports filament with a diameter of 1.75mm.

6.9.7. Travel Tab

Combing Mode: This option determines how the nozzle will move when it is not printing.

- The '**Off**' option results in the nozzle moving the shortest distance between the previous extrusion location and the new start location.
- The '**All**' option results in the nozzle moving along anything it has already extruded.
- The '**No Skin**' option results in the nozzle avoiding the outer layers to move to the new start location. This improves print quality substantially.

6.9.8.Machine Tab

- **Right 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.

6.9.9.Line Width Tab

- **Outer Wall Line Width (mm):** Set the width of the outer wall line. Higher levels of detail can be printed by reducing this value.

- **Inner Wall(s) Line Width (mm):** Set the width of all single walls except the outer wall.

- **Top/Bottom Line Width (mm):** Set the width of the bottom and top lines.

- **Infill Line Width (mm):** Set the width of a single infill line.

- **Support Line Width (mm):** Set the width of a single support structure line.

- **Skirt/Brim Line Width (mm):** Set the width of a single brim or skirt line.

- **Raft Top Line Width (mm):** Set the width of the lines on the top surface of the raft. The top of the raft will be smooth if these lines are set very thin.

- **Raft Middle Line Width (mm):** Set the width of the lines on the middle raft layers. The lines will stick to the build plate if the second layer extrudes more.

- **Raft Base Line Width (mm):** Set the width of the raft base layer. To assist with build plate adhesion these lines should be thick.

- **Prime Tower Line Width (mm):** Set the extrusion width of the main tower.

6.9.10.Seam Tab

NOTE: The Z Seam is where the 3D printer completes its motion when printing the outside layer (skin) of a model. This can lead to a small blob where the Z height is changed. If in alignment, there will be a noticeable line on the side of the print, known as a Z Seam, due to the filament continuing to ooze at the stop/start location. These options are used to reduce this effect.
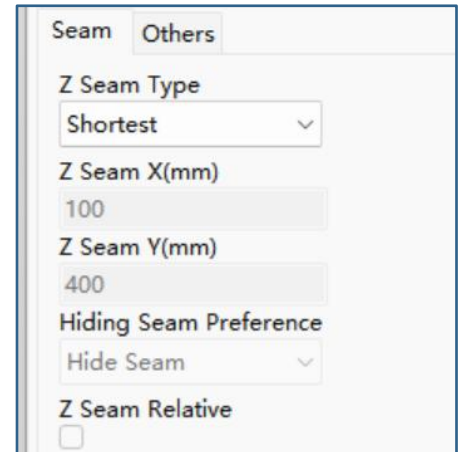
- **Z Seam Type:** Determines where the Z Seam will appear.

  - **Shortest:** Set the most time-efficient start/stop location.

  - **User Specified:** Set the X and Y start/stop location to specify where the Z Seam will appear.

  - **Random:** Select this option to let the printer select the start/stop location randomly, thereby preventing a column from building.

  - **Sharpest Corner:** Select this option to let the stop/start/ location and the Z Seam appear on the model's sharpest corner.

- **Z Seam X (mm):** Set the X location of the Z Seam. This option is only available when the Z Seam Type is set to **User Defined**.

- **Z Seam Y (mm):** Set the Y location of the Z Seam. This option is only available when the Z Seam Type is set to **User Defined**.

- **Hiding Seam Preference: Set** whether the Z Seam should be on the outside or inside of the corner. This option is only available when the Z Seam Type is set to **Sharpest Corner**.

- **Z Seam Relative:** If this box is checked it will set the Z Seam relative to the object's center, whereas leaving it 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**.

6.9.11.Others Tab
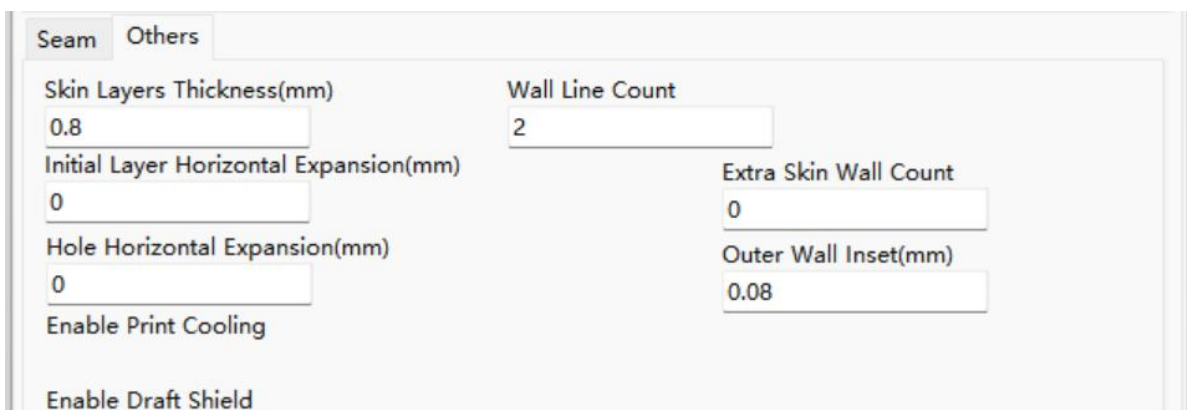
- **Skin Layers Thickness (mm):** Set the thickness of the bottom and top skin layers.

- **Horizontal Expansion (mm):** Thermoplastics generally shrink when cooling. This option allows you to adjust the part size to offset shrinkage and is used for prints that require tighter tolerances.

- **Skin Alternate Rotation:** A 3D printer typically prints solid layers for the bottom and top layers, and it does so by changing the direction by 90° from layer to layer. This setting will add 45° degrees of rotation every two layers.

These images illustrate the normal print direction of the first three layers.

This image illustrates the print direction of the third layer when the **Skin Alternate Rotation** option has been enabled.

- **Enable Print Cooling:** Cooling air will be directed at the printed part when this option is enabled.

- **Enable Draft Shield:** When enabled, the printer will print a wall around the model to prevent environmental drafts or breezes from affecting the cooling. This is normally used when **Enable Print Cooling** has been disabled for filaments that need a longer cooling time, like ABS.

- **Wall Line Count:** Set the number of walls to print.

7. PRINT BY APP

1. Download Poloprint Cloud on the Google Play Store or Apple App Store
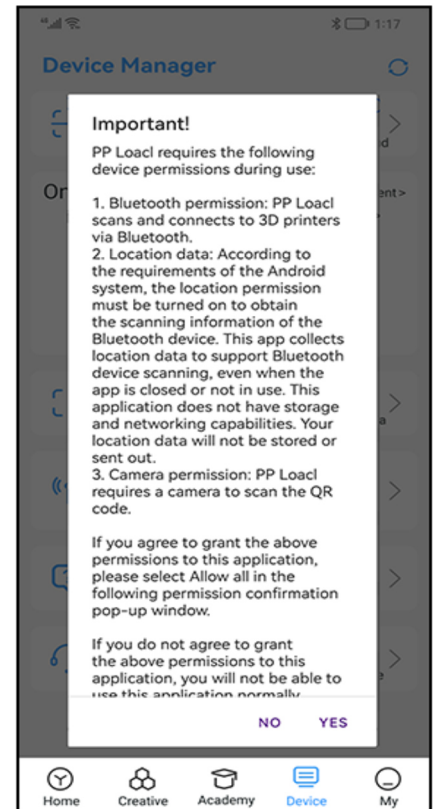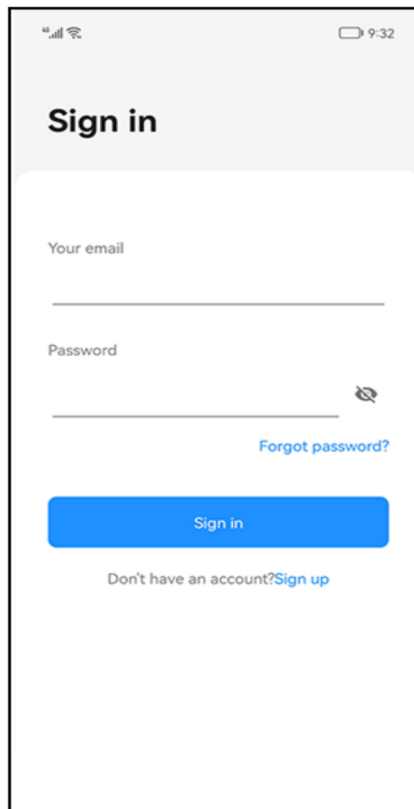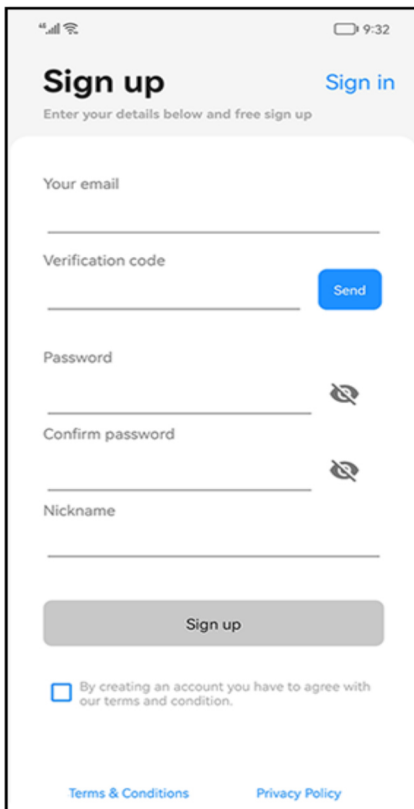
2. Open the up the app. Then sign up and

3. Click "Device" and then agree to allow all permissions

NOTE:

1. **If this is your first time using this app, please read the usage guide in the app.**

2. **If you get a pop-up message on your phone about the app, please agree to "Allow All The Time"**
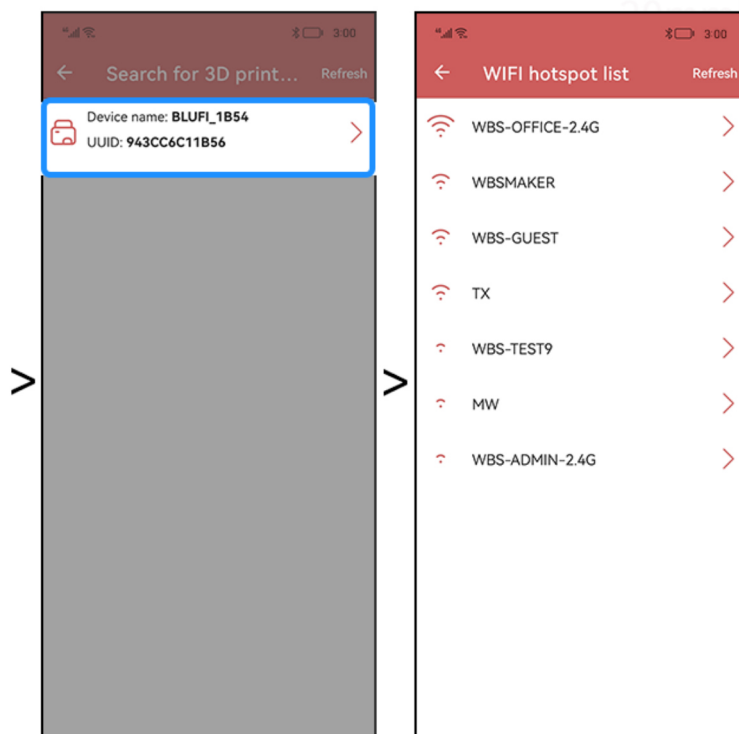
Poloprint Cloud - WiFi Configuration
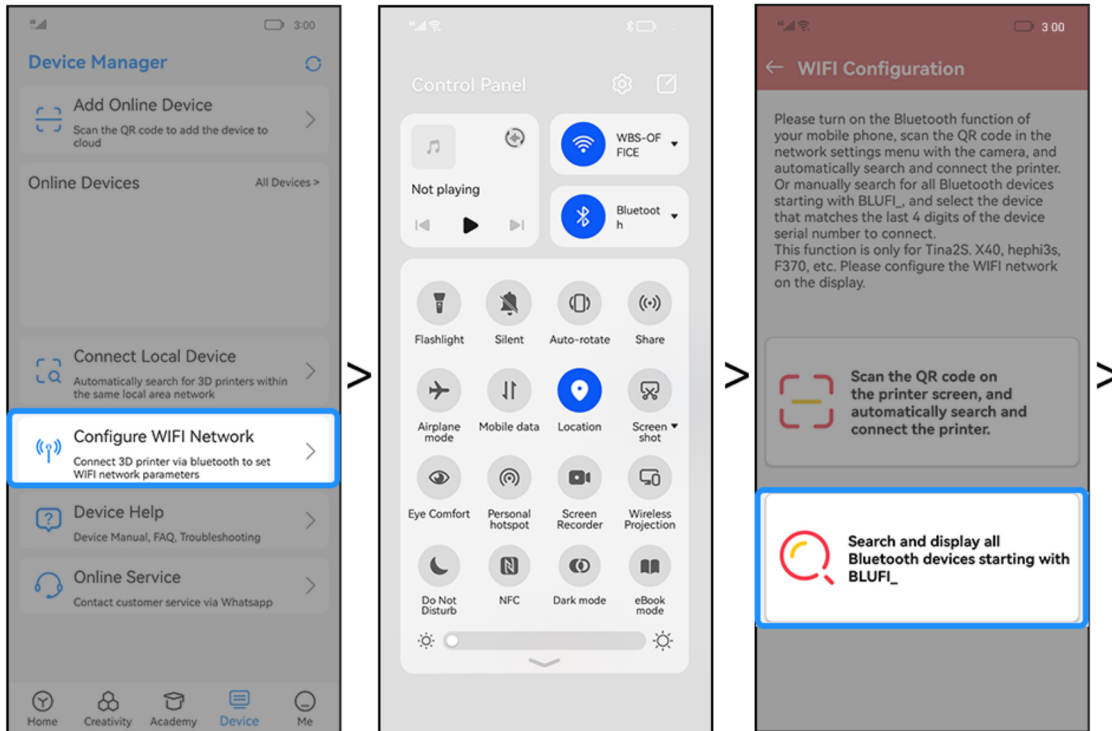
1. Choose " WIFI Configuration".

2.Turn on "Bluetooth" and "Location".

3. Scan the code or search bluetooth.

4. Click "bluetooth".

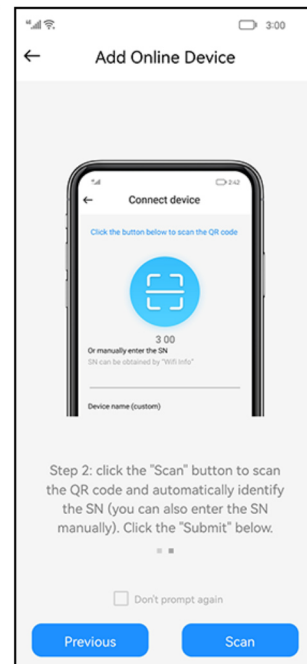5. Select 2.4G WIFI, enter the password.

Poloprint Cloud - Wirelessly Connecting the 3D Printer
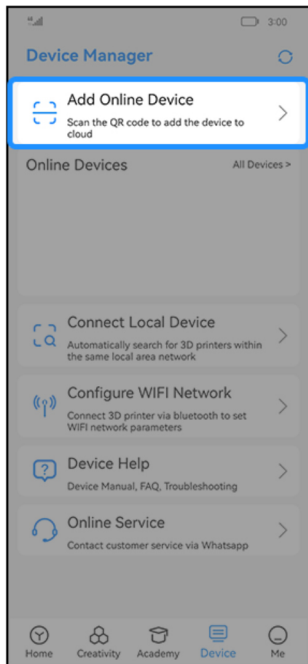
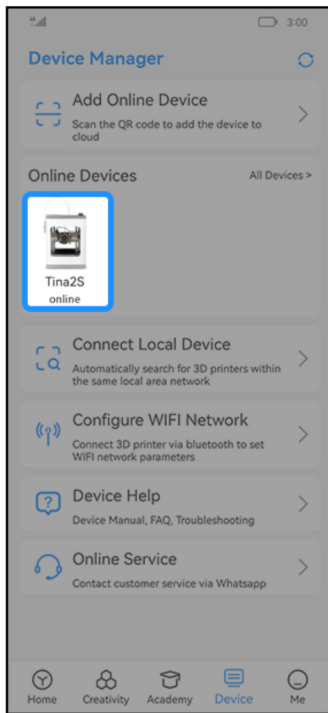
1. Click "Add online device".
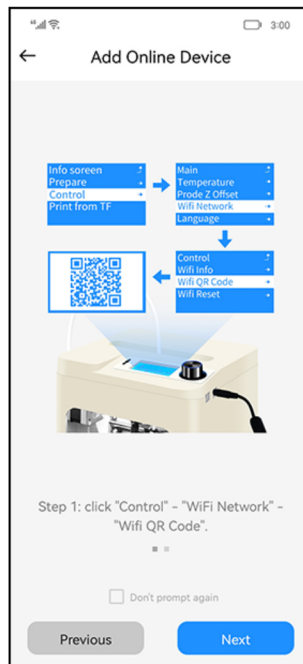
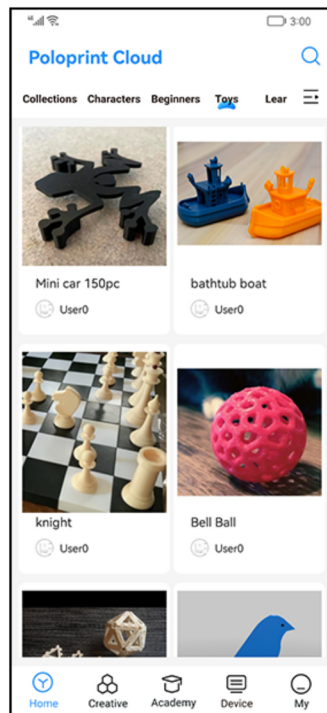2. Operate according to the tutorial.
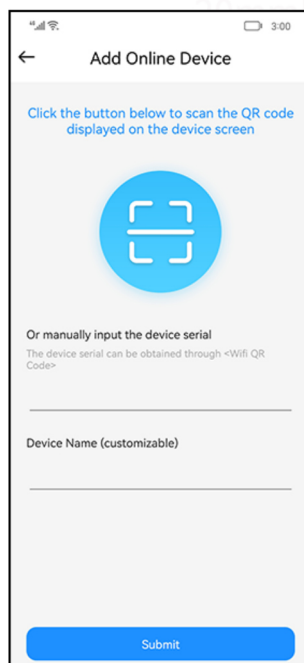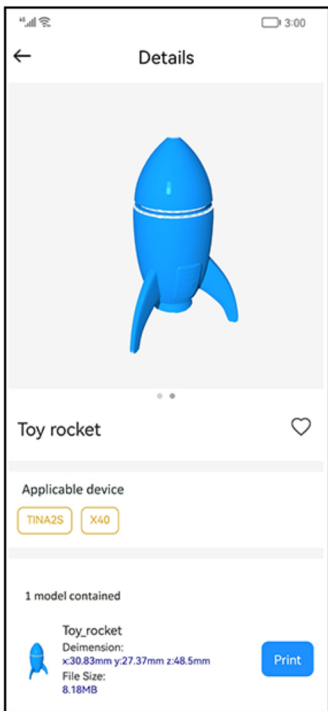

3. Scan and submit.
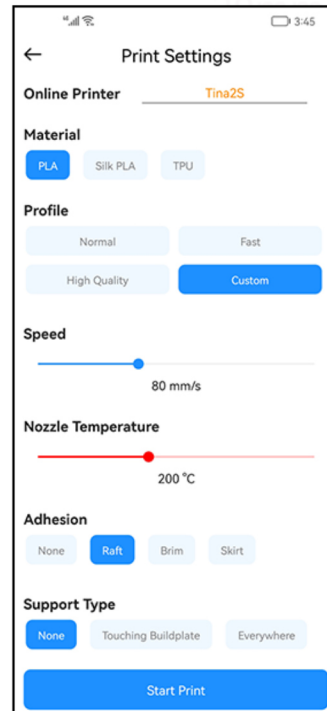
Poloprint Cloud - Printing a Model

1. After the device is "online".
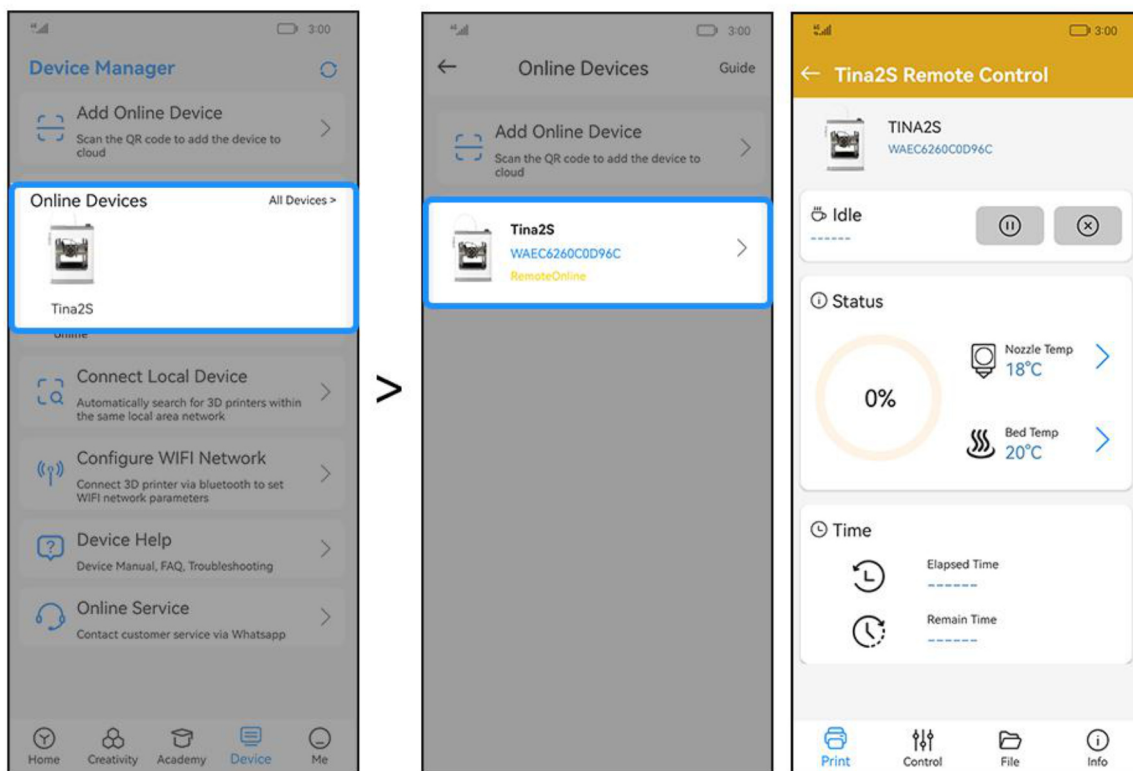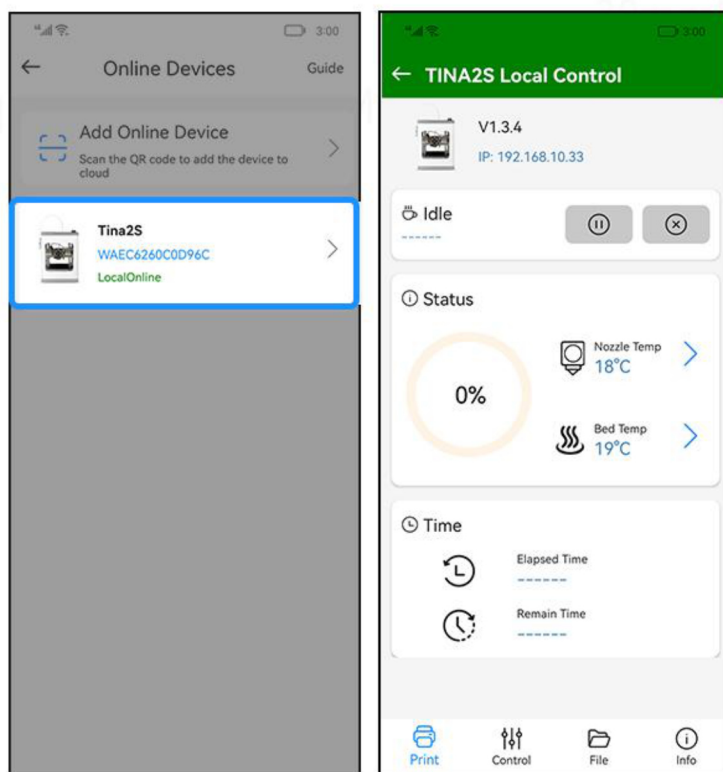

2. Choose a model.

3. Click "Print".

4. Set slice parameters, click "Start Print".

Poloprint Cloud - Controlling the Printer

When you using the app outside of the local network that your 3D Printer is on, only "Remote Control" can be used.
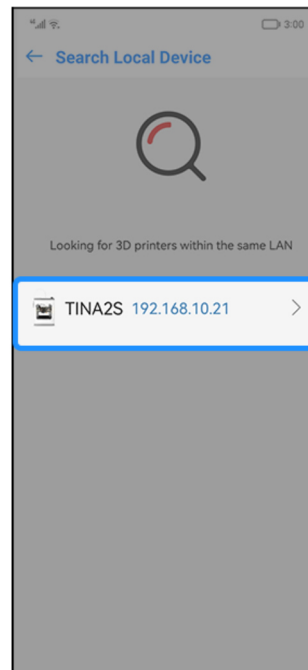

NOTE:

File transfers are faster when using Local Control
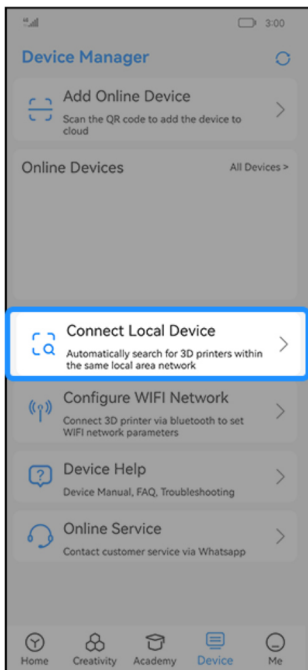
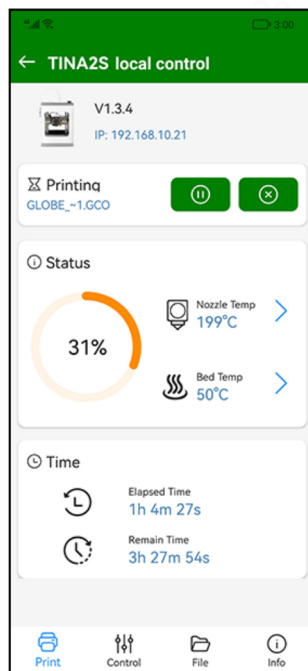**Local Control also enables more control options**

When you are using the app on the same local network as your 3D Printer, only "Local Control" can is used.

Poloprint Cloud - Local Device

1. If the user doesn't want to access the cloud server, the local device can be used.

>

2. Click the IP.

NOTE: The app must be under the same network as the device.

8. COMMON FAULTS

If you encounter a technical problem, please review the following troubleshooting pages before consulting customer support.

8.1. Filament Feeding Failure

If it is found that the extruder cannot load filament normally, and the extruder gear is still rotating, the extruder is blocked. Please clean the nozzle.

8.2. Printing Failure

If the 3D printer fails to work properly, perform the following operations for troubleshooting.

8.2.1. Perform the action: Print the file

(1) Symptom: The nozzle temperature is displayed as: **-14°C**, or a failure is displayed as shown below:

- **Cause:** The temperature measurement cable of the nozzle is faulty, or the cable has come loose.

- **Solution:** Replace the temperature measurement cable or plug it in.

(2) Symptom: A failure prompt as shown in the figure appears:
- **Cause**: The nozzle heating is faulty.
- **Solution:** Replace the sprinkler assembly.

(3) Fault: The printer restarts after 5 seconds, or display the following:
- **Cause:** The files on the TF card is faulty.
- **Solution:** Format the card.

(4) Fault: As soon as the printer starts, it will display whether to "continue printing".
- **Cause:** The files on the TF card is faulty.
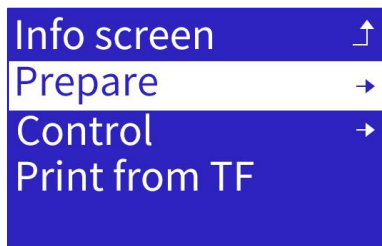- **Solution:** Format the TF card

8.2.2. Turn on the "Endstop" function

Info screen ⬆	Auto home	Endstops
Prepare →	Level bed	X： Off
Control →	Endstop →	Y： Off
Print from TF	Disable steppers	Z： Off
	Wizard	Z-Probe： Off
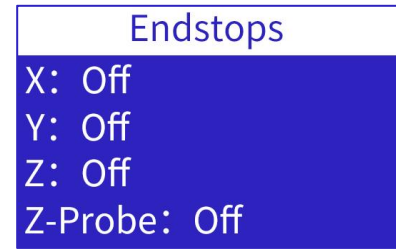
Symptom: Touch the X, Y, and Z endstops by hand, and check that their statuses change. Hold a metal object close to the bottom of the Proximity Sensor, and check that its status changes. Also check whether the light of the Proximity Sensor is on.

- **Cause**: If the state doesn't change, the X, Y, or Z endstop, or the proximity sensor may be broken.

- **Solution**: Replace the relevant endstop or the proximity sensor.

8.2.3. Turn on the "Auto home" function

Info screen ⬆	Main ⬆
Prepare →	Load Filament
Control →	Unload Filament
Print from TF	Move axis →
	Auto home

(1) Symptom: The nozzle moves to the left and then stops.

- **Cause**: The X endstop is broken or its wire has fallen off. Use the 'endstop' function to check.
- **Solution**: Attach or replace the cable on the X endstop or replace the endstop.

(2) Symptom: The nozzle moves to the right but keeps hitting the X endstop.

- **Cause**: The X endstop is broken or its wire has fallen off. Use the 'endstop' function to check.
- **Solution**: Attach or replace the cable on the X endstop or replace the endstop.

(3) Symptom: The nozzle doesn't move and the display shows:

X Homing failed
PRINTER HALTED
Please Reset

- **Cause:** The X motor cable has fallen off or is damaged.
- **Solution**: Attach or replace the X motor cable.

(4) Symptom: The platform moves backward and hits the Y endstop and the display shows:

- **Cause**: The Y endstop is broken or its wire has fallen off. Use the 'endstop' function to check.

Y Homing failed
PRINTER HALTED
Please Reset

- **Solution**: Attach or replace the cable on the Y endstop or replace the endstop.

(5) Symptom: The platform moves forward and then stops.

- **Cause**: The Y endstop wire is loose or faulty, or the Y endstop is broken. Use the 'Travel Switch Information' option to check.
- **Solution**: Attach or replace the cable on the Y endstop or replace the endstop.

(6) Symptom: The platform does not move and displays:

- **Cause:** The Y motor cable has fallen off or is damaged.
- **Solution**: Attach or replace the Y motor cable.

```
Y Homing failed
PRINTER HALTED
Please Reset
```

(7) Symptom: The nozzle does not move upwards.

- **Cause**: The Z endstop is broken or its wire has fallen off. Use the 'endstop' function to check.
- **Solution**: Attach or replace the cable on the Z endstop or replace the endstop.

(8) Symptom: The nozzle keeps moving upwards, then hits the Z endstop, and displays:

- **Cause**: The Z endstop is broken or its wire has fallen off. Use the 'endstop' function to check.
- **Solution**: Attach or replace the cable on the Z endstop or replace the endstop.

```
Z Homing failed
PRINTER HALTED
Please Reset
```

(9) Symptom: The nozzle does not move and displays:

- **Cause:** The Z motor cable has fallen off or is damaged.
- **Solution**: Attach or replace the Z motor cable.

```
Z Homing failed
PRINTER HALTED
Please Reset
```

8.2.4. Perform the action: turn on the " Auto Level " function

```
Info screen      ⮌
Prepare          →
Control          →
Print from TF
```

```
Auto home
Level bed
Endstop          →
Disable steppers
Wizard
```

(1) Symptom: The nozzle doesn't go down (the light of the Proximity Sensor will be off).

- **Cause:** The proximity sensor cable has fallen off or is damaged.
- **Solution**: Attach or replace the proximity sensor cable.

(2) Symptom: The nozzle keeps going down and hits the platform. Although the indicator light of the proximity sensor is on, there is no output signal.

- **Cause**: The proximity sensor is broken or its wire has fallen off. Use the 'endstop' function to check.

- **Solution**: Attach or replace the cable on the proximity sensor or replace the proximity sensor.

8.3. Common Issues

Why doesn't the model stick to the platform or is difficult to remove?
If the gap between the nozzles and the platform is too small, the model and the platform will adhere to each other tightly, making them difficult to separate.

If the gap between the nozzle and the platform is too big, the model won't stick to the platform.

Use the Z offset function to adjust the gap between the nozzles and the platform.

Why does the nozzle hit the platform?
If the nozzle hits the platform during automatic leveling, it may be due to the high relative position of the Proximity Sensor to the nozzle.

If the extruder assembly has been replaced before, loosen the screws securing the extruder and push the extruder to the highest position before locking it.

Use the height adjustment function of the Proximity Sensor to reduce the height of the Proximity Sensor.

If the problem persists, the Proximity Sensor may be faulty.

Why does the nozzle hit the endstop?
There may be a problem with the relevant endstop or motor.

Check that the endstop and motor wires are secure.

Use the troubleshooting section to identify the cause of the failure.

Why does the Auto Bed Leveling fail?
Check whether the printing platform has been installed and is positioned correctly.

Use the height adjustment function of the Proximity Sensor to reduce the Sensor's height.

If the problem persists, the Proximity Sensor may faulty.

Use the troubleshooting section to identify the cause of the failure.

Why can't the nozzle be heated?
Use the troubleshooting section to identify the cause of the failure.

8.4. Maintenance

Required daily maintenance includes cleaning the nozzle, replacing the magnetic sticker, and maintaining the optical axis and the screw rod.

1.1.1. Power

- If the printer isn't used for a long time, turn off the power.
- Keep the area of operation clean and dry.
- Prevent dust accumulation on the power supply's input and output ports.
- The power supply should not be exposed to rain or direct sunlight.
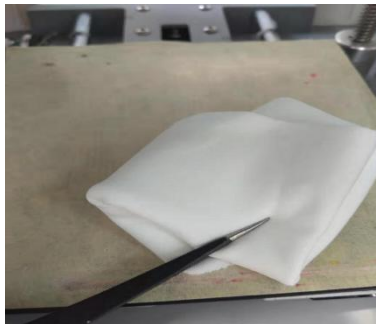
1.1.2. Filament

Remove the remaining filament from the printer once printing has been completed and keep them sealed.

1.1.3. Clean Nozzle

Dust may accumulate around the nozzle during the printing process. This can build up over time and lead to the nozzle clogging. It is therefore necessary to check whether the printing nozzle is blocked before printing and clean it if required.

Maintenance method:

1. The nozzle is generally cleaned with a tweezer, and a wiping cloth will remove impurities.

2. **Clearing print head blockage** *(see next page for pictures)***:** Pull out the end of the guide tube connected to the nozzle by pressing the Tube Fitting to pull out the filament tube. Heat the nozzle to 200° and use the 1.5mm L wrench provided in the accessory box to inset into the nozzle until the clogged filaments are melted and extruded.
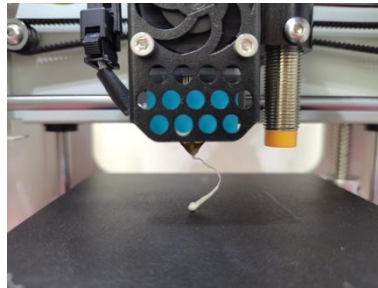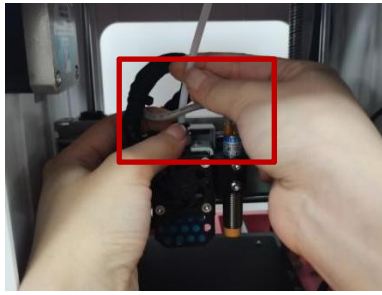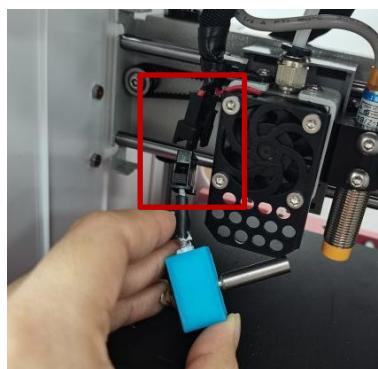
8.4.4.Replace Hotend

Improper selection of filament or improper operation will block the nozzle. If required, the Hotend may have to be replaced.

Disassembly:

1. Put the printer in the shutdown state and use a 2.0mm L-wrench to loosen the screw that fixes the nozzle to the inside of the nozzle fan.

2. Open the buckle and remove the Hotend from the bottom.

3. A new Hotend can be installed following the steps above in reverse. The platform needs to be re-calibrated after the Hotend has been replaced.

8.4.5.Bearings

During printer operations, both X and Y movement uses precision guide rails and the Z axis screw to ensure precise and smooth linear motion. Applying lubricating silicone grease regularly will reduce friction and reduce the wear of mechanical moving parts. This maintenance is recommended after every 1 000 hours of use.

Maintenance method: Apply lubricating silicone grease evenly on the screw rod and the optical axis. Start the printer to execute a full stroke of each axis movement several times to distribute the lubricating silicone grease evenly on each axis' surface.