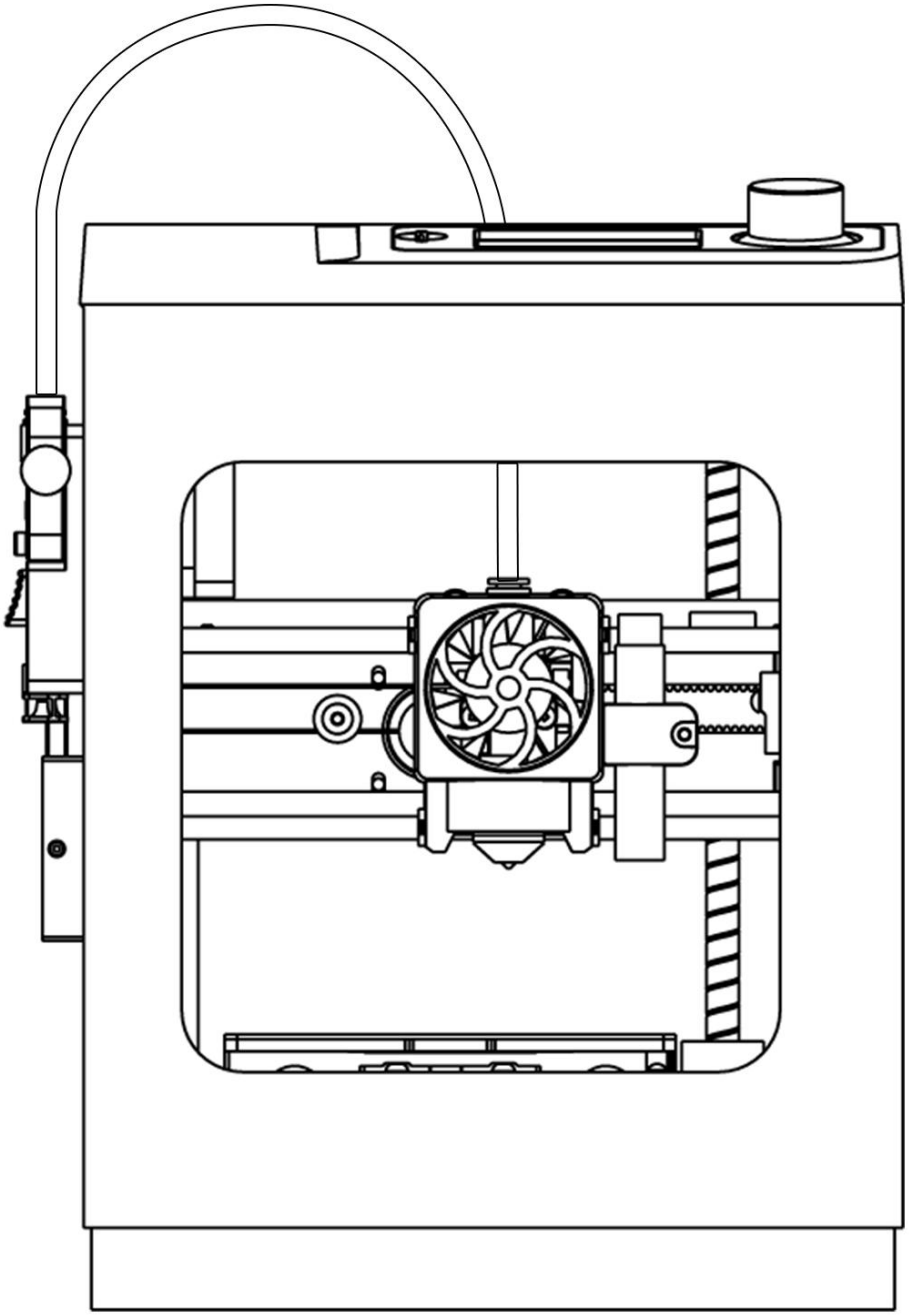
EN

3D
PRINTER

USER'S MANUAL



CONTENTS

1.NOTES

PLEASE READ THIS MANUAL BEFORE USING THE DEVICE, PAYING CLOSE ATTENTION TO THE SAFETY WARNINGS AND GUIDELINES. KEEP THIS MANUAL IN A SAFE PLACE FOR FUTURE REFERENCE.

THIS MANUAL IS ONLY APPLICABLE TO TINA2 PLUS V3.

1.1.Safety Warnings

- Do not touch the nozzle or stepper motors when the 3D printer is operating or just finished printing, as the nozzle temperature can reach up to 230° C.
- Do not expose this device to water or moisture. Keep liquids away from the device. If moisture gets inside, immediately unplug it and let it fully dry before using again.
- Do not handle the device, power cord, or any cables with wet hands.
- Before use, check the unit and power cord for any damage. Do not use if damage is found.
- Ensure the power outlet provides the proper voltage and current required by the device.
- Unplug the device when not in use.
- Protect the power cord from being crimped, pinched, walked on, or tangled. Ensure the cord does not pose a tripping hazard.
- Always unplug by grasping the plug head or adapter body, never by pulling the cord.
- Turn off and unplug the 3D printer before making repairs or performing service.

Warning:

This device complies with Part 15 of the FCC Rules. Operation is subject to the following two conditions: (1) this device may not cause harmful interference, and (2) this device must accept any interference received, including interference that may cause undesired operation.

Changes or modifications not expressly approved by the party responsible for compliance could void the user's authority to operate the equipment.

NOTE: This equipment has been tested and found to comply with the limits for a Class B digital device, pursuant to Part 15 of the FCC Rules. These limits are designed to provide reasonable protection against harmful interference in a residential installation. This equipment generates, uses and can radiate radio frequency energy and, if not installed and used in accordance with the instructions, may cause harmful interference to radio communications. However. there is no guarantee that interference will not occur in a particular installation.

If this equipment does cause harmful interference to radio or television reception which can be determined by turning the equipment off and on, the user is encouraged to try to correct the interference by one or more of the following measures:

--Reorient or relocate the receiving antenna.

-- Increase the separation between the equipment and receiver

--Connect the equipment into an outlet on a circuit different from that to which the receiver is connected.

--Consult the dealer or an experienced radio/TV technician for help

RF Exposure Statement

To maintain compliance with FCC's RF Exposure guidelines, this equipment should be

installed and operated with minimum distance of 20cm the radiator your body. This device

and its antenna(s) must not be co-located or operation in conjunction with any other antenna nr transmitter.

1.2.Filament

To ensure optimal performance, only use filament provided by the manufacturer. Third-party filaments may have inconsistent specifications and quality, potentially clogging or damaging the nozzle and motor. Using unauthorized consumables voids the warranty.Store unused filament in a sealed bag to prevent moisture absorption, which can degrade print quality.

1.3.Environmental Requirements

This 3D printer is for indoor use only, with an ideal ambient temperature of 15° C - 25° C.

Below 10° C: Printed models may not adhere properly to the platform.

Below 0° C: The device will not start.

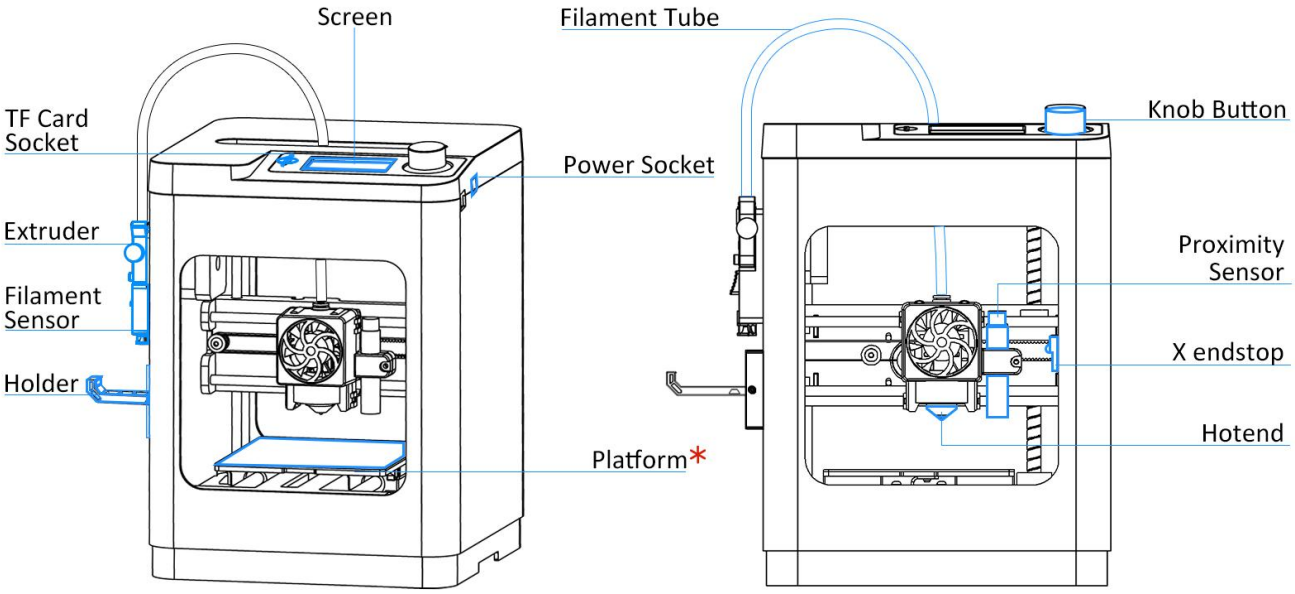
Above 30° C: Print quality will significantly decline and the nozzle may clog.

2.INTRODUCTION

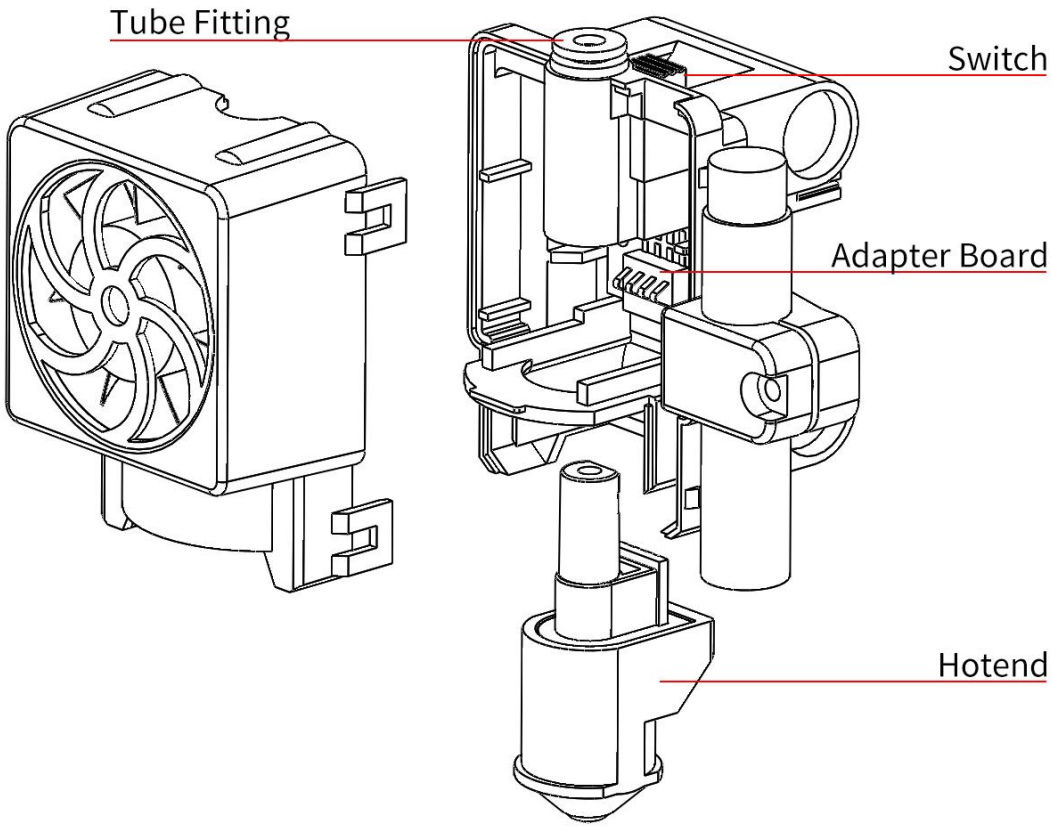
2.1.Specification

Model	TINA2 PLUS V3
Build Volume	100 x 105 x 100mm /3.9’’x 4.0’’x 3.9’’
Nozzle Diameter	0.4mm
Layer Thickness	0.1-0.4mm
Platform Material	PEI build surface
Leveling Tech	9-Point Auto Bed Leveling
Max Nozzle Temp	245°C
Motherboard	R72R
Max Heated Temp	80°C
Max Print Speed	150mm/s
Print Precision	±0.1mm
Product Weight	3Kg/6.6lbs
Output	12V-8A,96W
Filament Diameter	1.75mm
Supported Filament	PLA / PLA+ / TPU / PETG
Filament Capacity	100-250g (built-in holder) 1kg (with extra rack)
Slicing Software	Wiibuilder (Win / Mac) Kiri (Chrome OS / iPad) OctoPrint (Pi / Linux)
Input File Format	STL / OBJ / AMF
Print Format	G-code
Input Method	TF Card / Wi-Fi / APP
APP	Poloprint Cloud (Android/IOS)

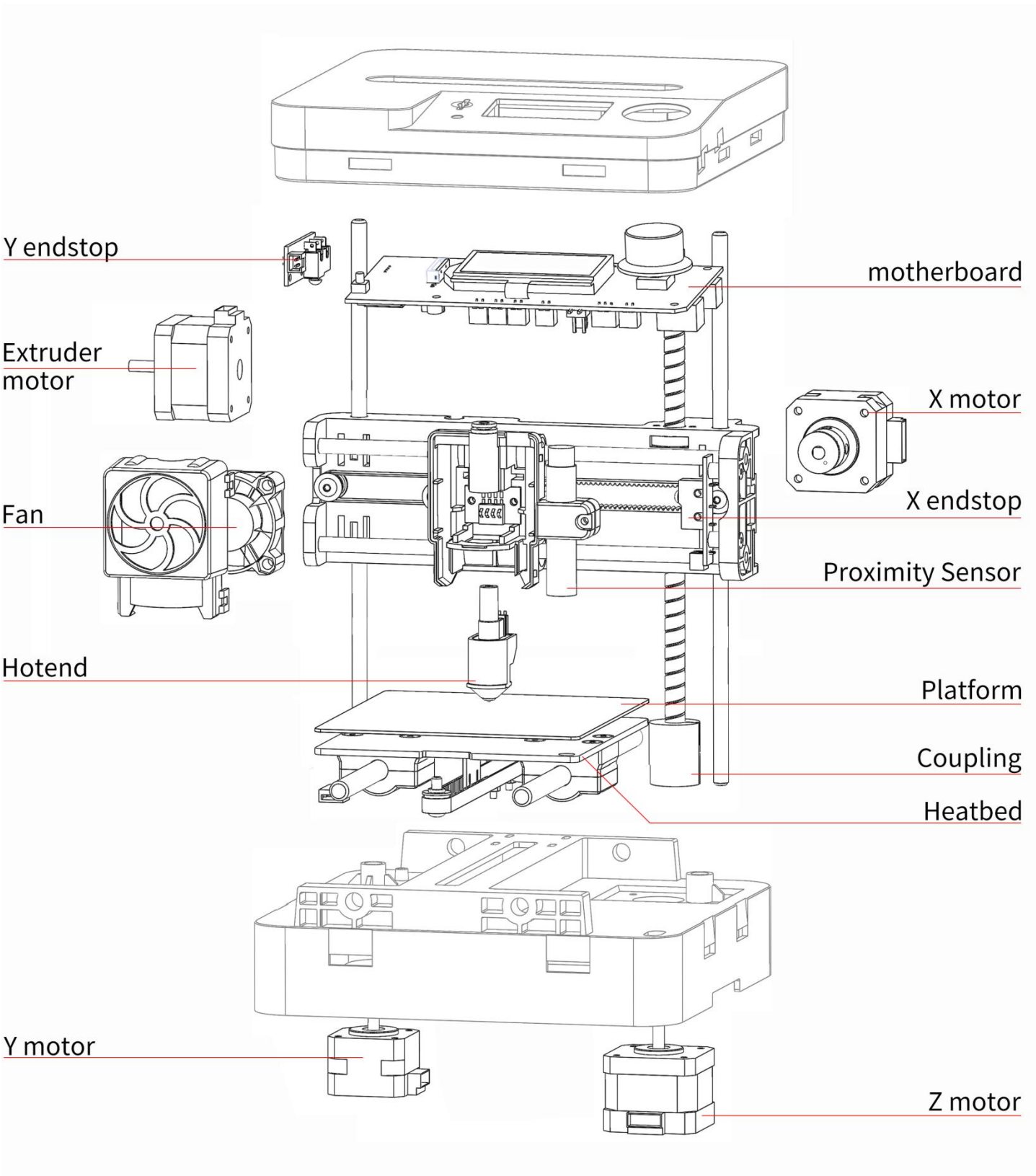
2.2.Product Overview



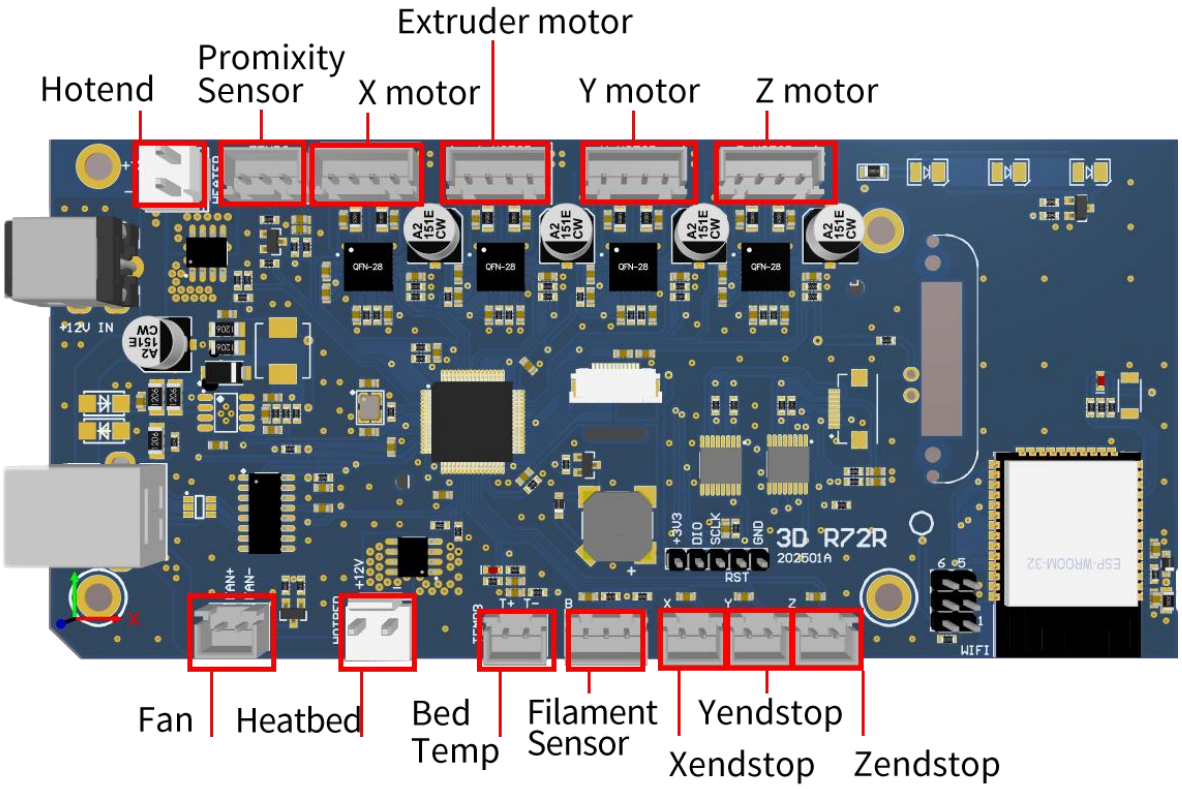
2.3.Hotend Structure



2.4.Internal Structure



2.5.Motherboard

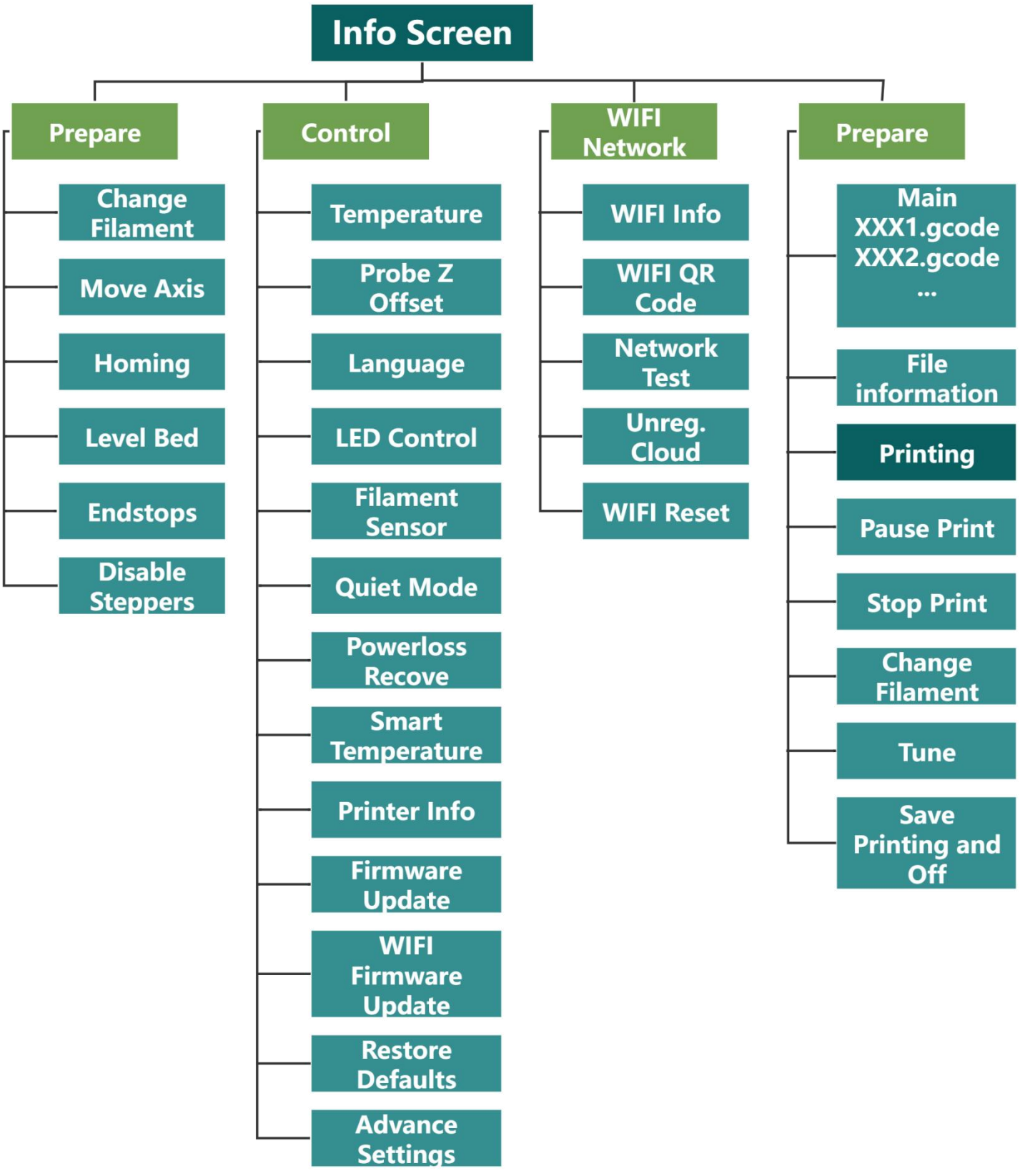


3.SYSTEM MENU

The top control panel features:

TF card slot (left): Stores print files

Knob button (right): Access system menu



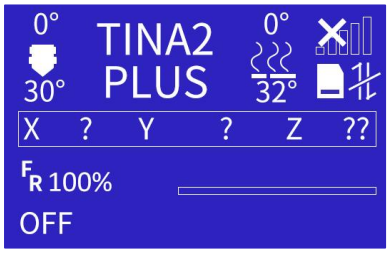
3.1.Info Screen

Knob controls:

●Clockwise turn: Move down in menus, decrease value

●Counterclockwise turn: Move up in menus, increase value

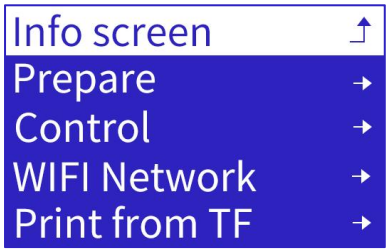
●Press: Enter submenu, select option, confirm value



Info Screen displays the status of the main components, including: Nozzle temperature (preset temperature and actual temperature), heated bed temperature (preset temperature and actual temperature), network connection status, XYZ position information, remaining printing time, TF card status and cloud platform connection status.

The bottom bar of the screen is the information bar, which displays the network status after power on. When the device is not connected to the Internet, it will display "OFF". After the device is connected to the WIFI network, it will display the IP address of the device.

Press the knob to open the main menu.



Press knob for Main Menu:

●Info Screen: Return to Info Screen

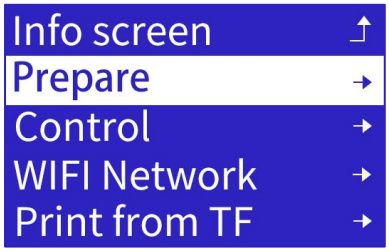
●Prepare: Open Prepare menu

●Control: Open Control menu

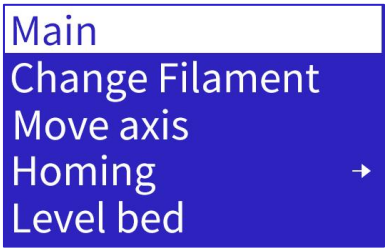
●WIFI Network: Displays the WiFi information.

●Print From TF: Show TF card files

3.2.Prepare Menu

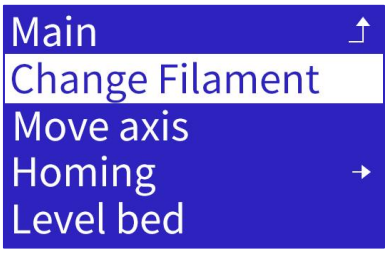


Press knob to open two-screen Prepare menu. Turn knob to scroll.



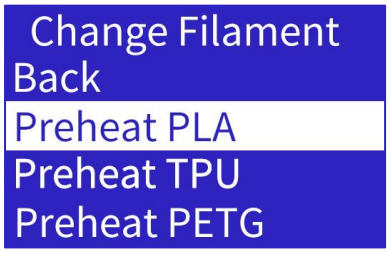
3.2.1.Change Filament

Use to properly unload and load filament. Do not pull filament out directly to avoid clogs.



1. Change Filament:

Select filament type to heat nozzle.

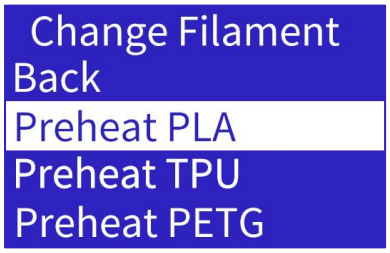


At 240°C, filament auto-unloads.

Beep indicates unload complete, insert new filament as prompted.

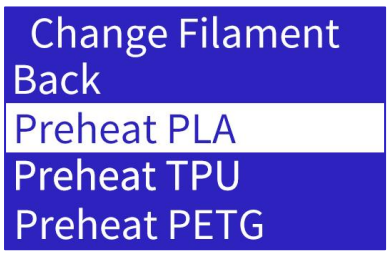
2. Load Filament:

Auto-heats nozzle and loads filament. Press knob during heating/extruding to cancel.



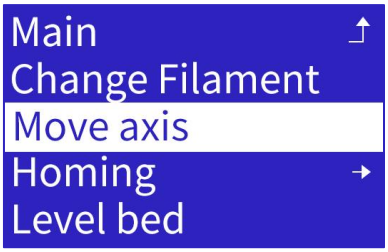
3. Unload Filament:

Auto-heats nozzle and unloads filament. Press knob during heating/retracting to cancel.



3.2.2.Move Axis

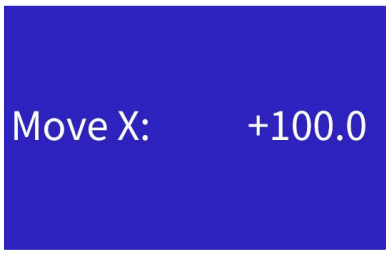
Manually control stepper motors for troubleshooting.

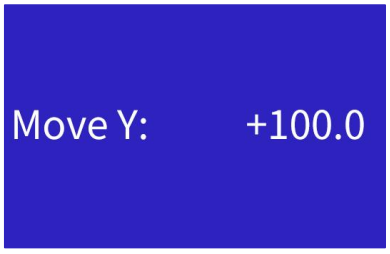


X/Y/Z Axis: 10mm, 1mm, 0.1mm increments

Extruder: 10mm, 1mm, 0.1mm increments

Knob controls bidirectional movement





CAUTION:

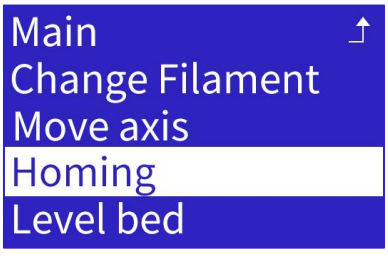
RANGE IS 0-100MM PER AXIS. PERFORM "AUTO HOME" FIRST FOR CORRECT NOZZLE COORDINATES. WITHOUT HOMING, CURRENT POSITION BECOMES ORIGIN, LIMITING JOG RANGE.

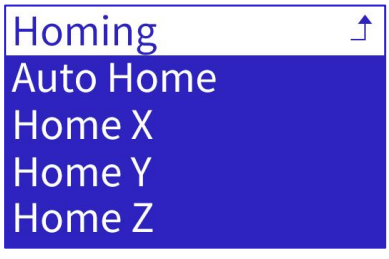
AXIS MOVEMENT LIMITED BY ENDSTOPS. BROKEN OR MISWIRED ENDSTOPS MAY CAUSE MOTOR TO NOT STOP OR MOVE.

EXTRUDER HAS THERMAL PROTECTION, ONLY MOVES WHEN NOZZLE IS >170° C.

3.2.3.Homing

Moves nozzle and platform to origin (X=0, Y=0, Z=0). Can home XYZ together or separately. Helps troubleshoot motor or endstop issues in a specific direction.

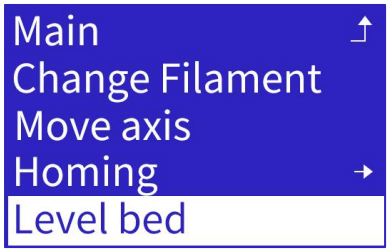




3.2.4.Level Bed

Auto-levels platform using proximity sensor. Done at each print start to ensure proper nozzle-platform distance.

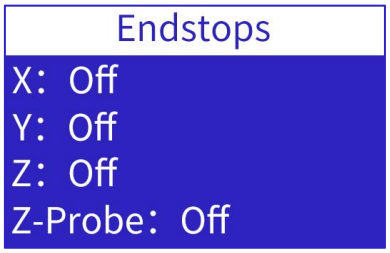
Can diagnose proximity sensor issues - error displays if sensor fails.



3.2.5.Endstops

Shows X/Y/Z endstop and proximity sensor status. Moving nozzle or platform to trigger endstops changes each one's state.

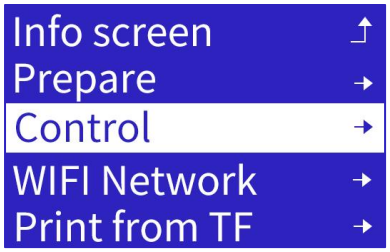
Helps diagnose faulty endstops or sensor - no state change indicates failure.



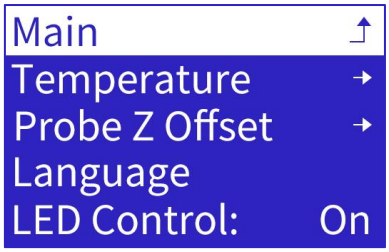
3.2.6.Disable Steppers

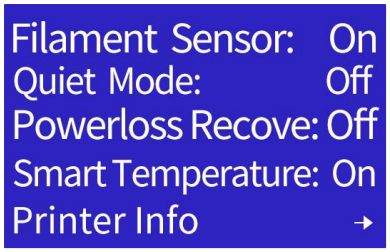
Releases stepper motor locking, allowing manual nozzle and platform movement.

3.3.Control Menu



Press knob to open two-screen Control menu. Turn knob to scroll.





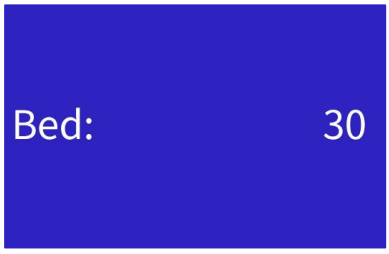
3.3.1.Temperature

Manually set nozzle and bed temperature:

Select Nozzle or bed, turn knob to desired temp, press to confirm.

Info Screen shows real-time temp during heating. Heating continues if exiting to other tasks.

To stop heating, re-enter menu and decrease temp to 0.



Preheat PLA/TPU/PETG: Background heats nozzle to 210°C/220°C/230°C respectively.

3.3.2.Probe Z Offset

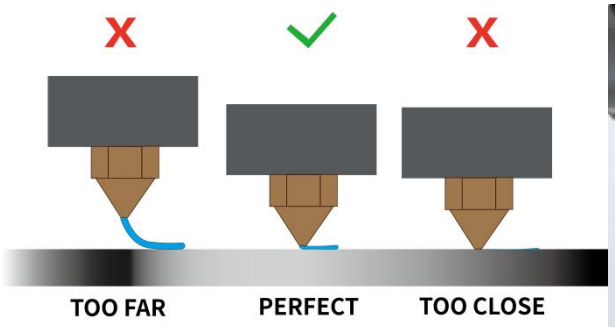
After bed leveling, sets nozzle-platform gap. Press knob to save.

Too small: Nozzle may scratch platform or clog.

Too large: Poor model adhesion, lifting.

Recalibrate after replacing nozzle or sensor.

Turn knob to precisely adjust nozzle height for smooth, flat first layer.



3.3.3.Language

Select from: English, Chinese, French, German, Spanish, Italian, Japanese, Portugal, Dutch, Turkish and Russian.

Turn knob to select, press to save and exit.

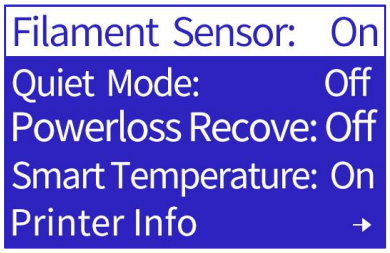
3.3.4.LED Control

Click "LED Control" to open the LED switch menu.

Press the knob to confirm and save.

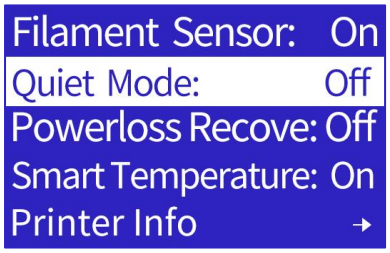
3.3.5.Filament Sensor: On

Press the button to toggle the filament break detection function ON or OFF. This feature is enabled by default. When no filament is loaded and you attempt to start printing or during printing, a reminder will automatically appear on the screen.



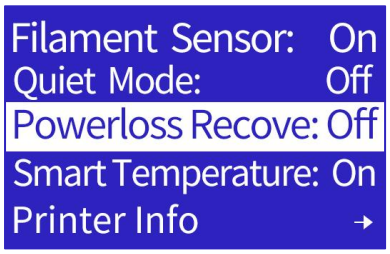
3.3.6.Quiet Mode: Off

Press the button to switch OFF/ON .The mute function is turned off by default. If you choose to enable it, the printer will significantly reduce its printing speed in order to lower the operating noise.



3.3.7.Powerloss Recove: Off

Press the button to switch OFF/ON .



If enabled, saves print progress every 10s. After unexpected power loss, prompts to resume on restart.

Disabled by default. Enable if desired.

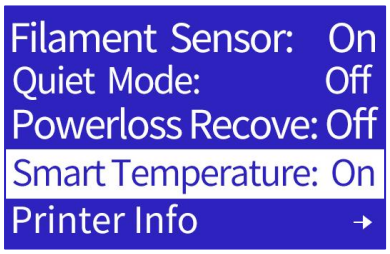
IMPORTANT:

PROGRESS SAVED TO ".BIN" FILE ON TF CARD EVERY 10S, OVERWRITING PREVIOUS DATA. ON RESTART AFTER SUDDEN POWER LOSS, CHECKS FOR ".BIN" FILE AND PROMPTS "CONTINUE PRINTING, STOP PRINTING". CONTINUING READS SAVED PROGRESS, STOPPING DELETES ".BIN" FILE.

SINCE SAVING ONLY EVERY 10S, RECOVERY MAY FAIL DUE TO TIME INTERVAL.

3.3.8.Smart Temperature

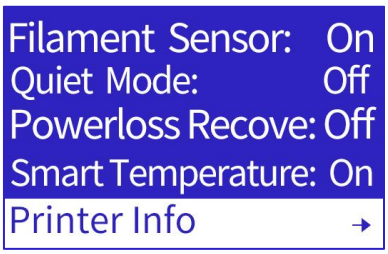
Press the button to switch OFF/ON .

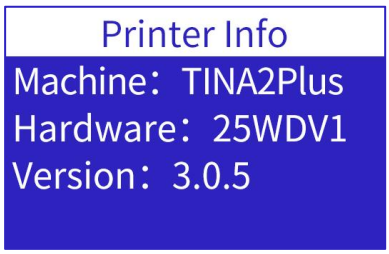


The smart temperature feature is turned on by default. The mainboard can monitor the ambient temperature and increase the temperature of the nozzle and platform according to the ambient temperature.

3.3.9.Printer Info

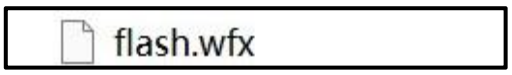
Shows current firmware version.





3.3.10.Firmware Update

1.Copy ".wfx" firmware file to TF card root. Decompress first if needed. Rename to "flash.wfx".



2.Insert TF card and select Firmware Update.

3.Printer checks version and updates.

3.3.11.WIFI Firmware Update

1.Copy ".efx" WiFi firmware file to TF card root. Decompress first if needed. Rename to "wifi.efx".

2.Insert TF card and select WiFi Firmware Update.

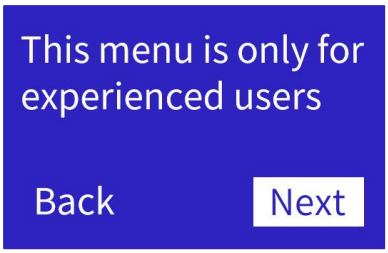
3.Printer checks version and updates.

3.3.12.Restore Defaults

Resets all parameters to factory defaults.

3.3.13.Advance Settings

Modify advanced parameters. Beginners use defaults.



LCD Contrast: Adjust screen clarity

Retract: Auto-retract, amount, speed, etc.

Max Speed (mm/s): Same as in slicer

Acceleration: Increase print speed

Jerk: Increase acceleration ramp time and corner/jitter speed

Probe Offsets: Set X/Y/Z offsets

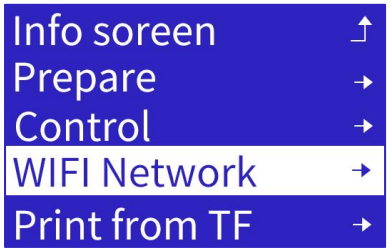
Filament: Set extruder compensation

Store Settings: Save changes

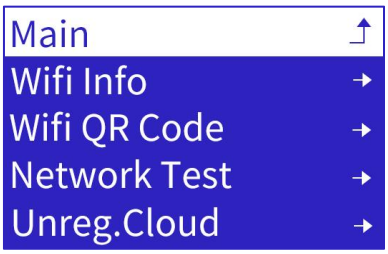
Load Settings: Apply saved changes

Initialize EEPROM: Reset to defaults

3.4.WIFI Network



Press knob to open two-screen Control menu. Turn knob to scroll.



The network function of TINA2S has two connection modes, local mode, and remote mode.

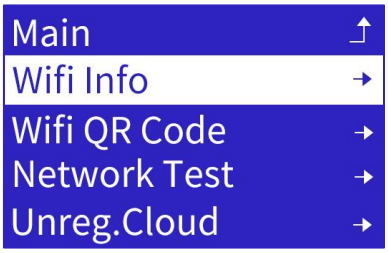
Local mode: In the same local area network, Wiibuilder or Cura on the computer can connect to the 3D printer, and perform remote control or send print files. To use local mode, the 3D printer should be connected to the local WIFI network.

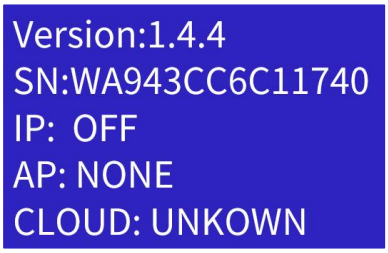
Remote mode: Use our APP, PoloPrint Cloud, remotely control the 3D printer or download the print model. With the remote mode, you can remotely control the 3D printer from any location. To use remote mode, the 3D printer should be connected to the WIFI network and registered with the APP. Notice, a 3D printer can only be registered by one APP account.

3.4.1.WIFI Info

Click "WIFI Info" to open the WIFI info screen. It will show the WIFI firmware version, the serial, IP address, AP name, and the connection status of the cloud server.

Before the network is configured, "OFF" and "UNKNOWN" will be displayed.

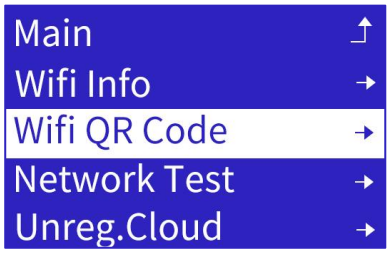




Only after configuring the network and registering in the APP will the 3D printer connect to the cloud server.

3.4.2.WIFI QR Code

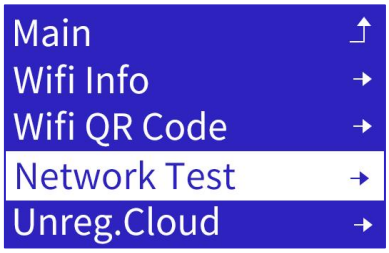
Click "WIFI QR Code" to show the QR code generated by the device serial. Used for App scanning networking and cloud access.



3.4.3.Network Test

Press "Network Test" to start testing the current network connection and response time.

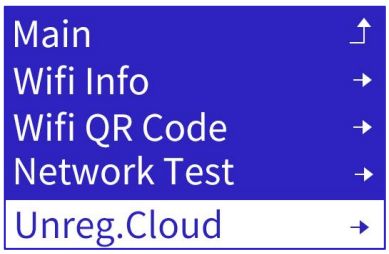
If the printer encounters network or app-related issues, use this function to check whether the network is properly connected or if the connection speed is too slow.

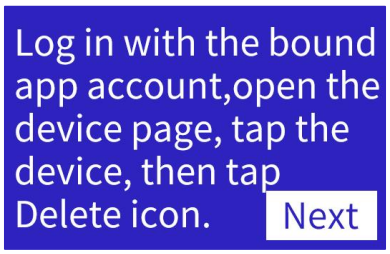


3.4.4.Unreg. Cloud

Press "Unreg. Cloud" to view the instructions for deregistering from the cloud platform.

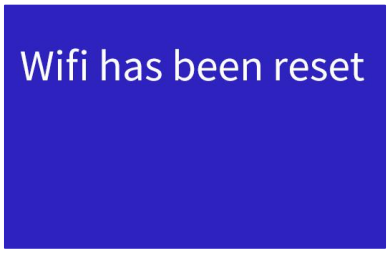
Please follow the tutorial to complete the operation in the app.



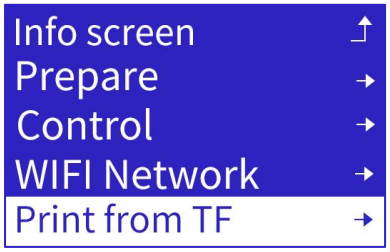


3.4.5.WIFI Reset

Every time TINA2S is turned on, it will automatically connect to the WIFI network that was successfully connected to the last time. If you need to connect TINA2 to a new WIFI network, you should execute the reset WIFI setting function to clear the saved WIFI network information.

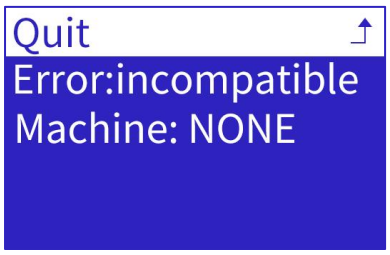


3.5.Print from TF



Shows ".gco" or ".gcode" print files on TF card.

If the correct printer is not selected when using slicer software like Wiibuilder or custom Cura, the generated G-code file will be incompatible with the printer, resulting in an 'incompatible' error message. Make sure to select TINA2PLUS.

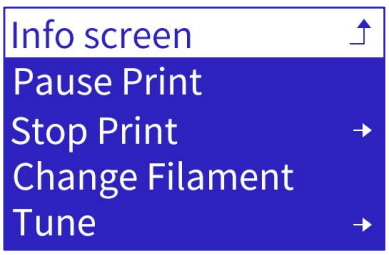


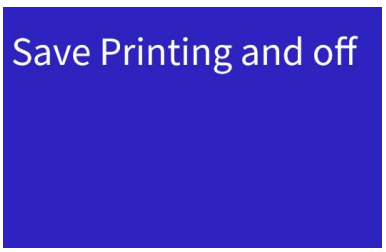
Select file and press knob for print info. "Print" starts printing.

To prevent damage, checks machine model in file. Blocks print if model mismatched or missing. Use Wiibuilder or custom Cura version and select correct model when slicing.

NOTE: FILE NAMES ⩽20 CHARACTERS TO AVOID DISPLAY ISSUES.

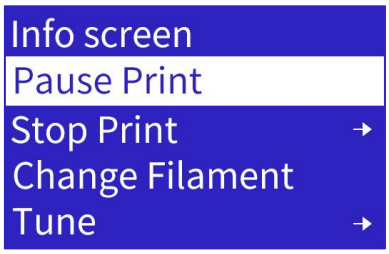
During print, press knob for print settings menu.

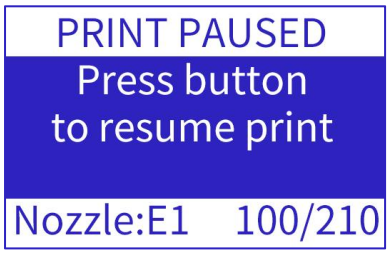


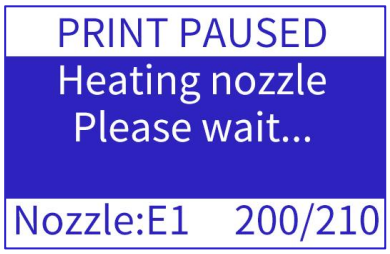


3.5.1. Pause Print

Pauses print and moves nozzle to pause position. Can resume from pause.

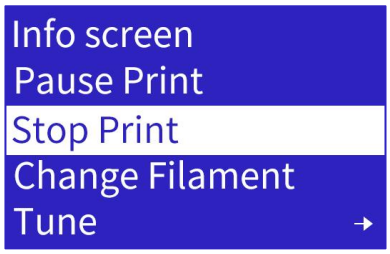






3.5.2.Stop Print

Cancels current print and returns to Info Screen.



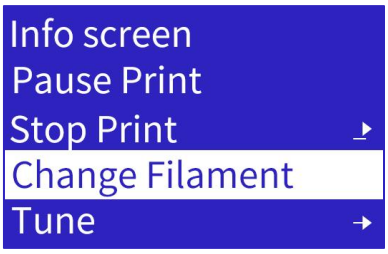
IMPORTANT:

REMOVE PRINTED MODEL FROM PLATFORM AFTER STOPPING.

3.5.3.Change Filament

Same steps as Change Filament in Prepare menu:

Heat to set temp. Auto-unload at temp.



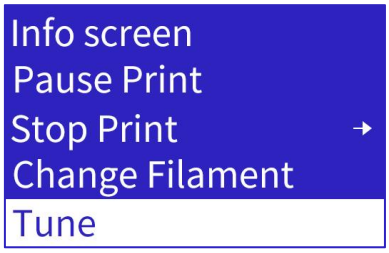
Beep, insert new filament, press knob.Auto-load at temp.

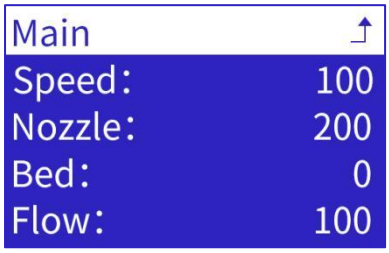
3.5.4.Tune

To change:

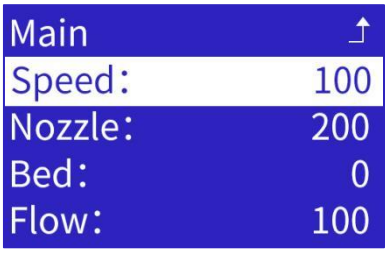
Select setting, turn knob to desired value.

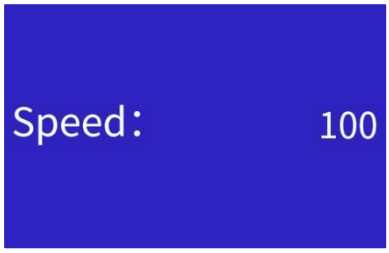
Press knob to confirm.





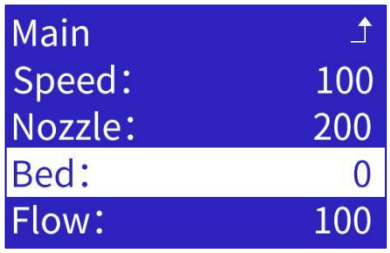
1.Speed

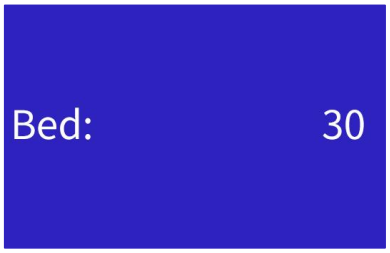




2.Nozzle temp

3.Nozzle temp





4.Advance K

Advance K (Linear Advance) is a parameter used to optimize extruder pressure control.

It dynamically adjusts the extrusion flow during acceleration and deceleration phases to compensate for material pressure changes inside the nozzle.

This improves surface detail and print quality while reducing issues such as stringing and blobs.

In simple terms, it helps the extrusion keep up with speed changes, especially during rapid accelerations and decelerations.

5.Probe Z Offset

IMPORTANT:

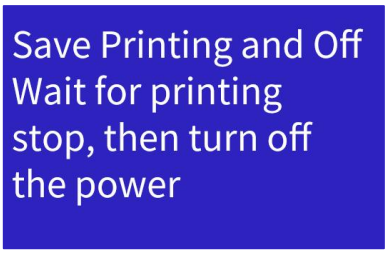
Z OFFSET MICRO-ADJUSTS NOZZLE HEIGHT IN REAL-TIME TO CHANGE GAP:

INCREASE BY 0.2MM IF GAP TOO SMALL (NO EXTRUSION)

GRADUALLY DECREASE IF GAP TOO LARGE (POOR ADHESION)

3.5.5.Save Printing and Off

Saves progress, moves to pause position, allows safe power off. Prompts to resume on next power on.



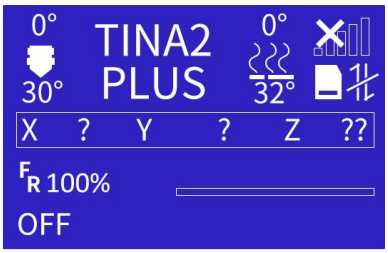
IMPORTANT:

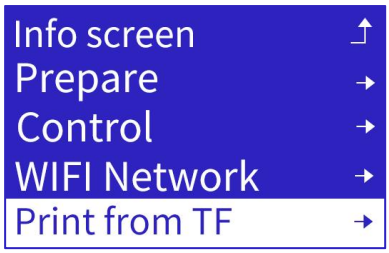
"PAUSE PRINT" AND "SAVE PRINT" TRIGGER HOMING. MECHANICAL ENDSTOP TOLERANCES CAUSE SLIGHT DEVIATION ON EACH HOME, POTENTIAL SEAM/SHIFT AT RESUME POINT.

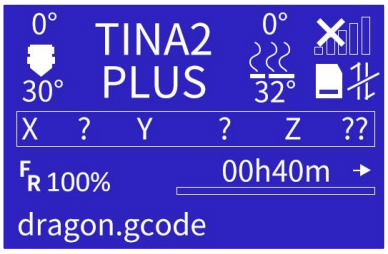
PRINT MODELS IN ONE SESSION WHEN POSSIBLE TO AVOID THIS.

4.COMMON MENU OPERATIONS

4.1.Print from TF Card







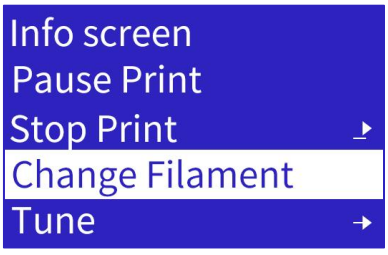
1.Open Main Menu, select "Print From TF"

2.Turn knob to select file, press knob to start print.

4.2.Print Multi-Color Models / Change Filament During Print

Same steps as Change Filament in Prepare menu:

Heat to set temp.Auto-unload at temp.



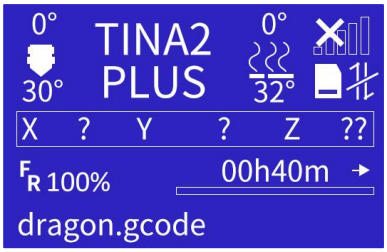
Beep, insert new filament, press knob.Auto-load at temp.

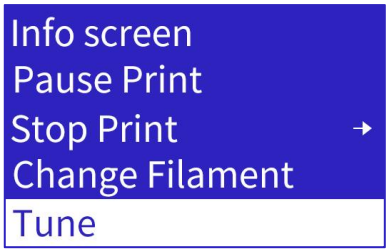
4.3.Adjust Temperature During Print

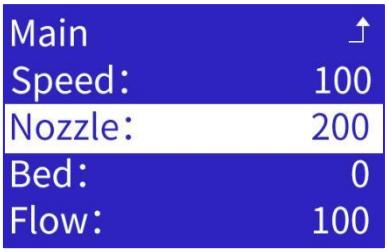
During print, press knob for print settings menu, select Tune > Nozzle to adjust temp:

Turn knob to increase/decrease temp.

Press knob to return.



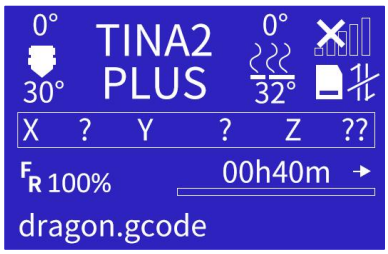


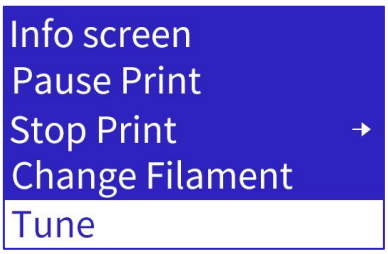


4.4.Adjust Z Offset During Print

During print, press knob for print settings menu, select Tune > Probe Z Offset to fine-tune nozzle-platform gap:

Turn knob to micro-adjust gap.





ALLOWS REAL-TIME Z-AXIS ADJUSTMENT TO CHANGE GAP DURING PRINT:

INCREASE BY 0.2MM IF GAP TOO SMALL (NO EXTRUSION)

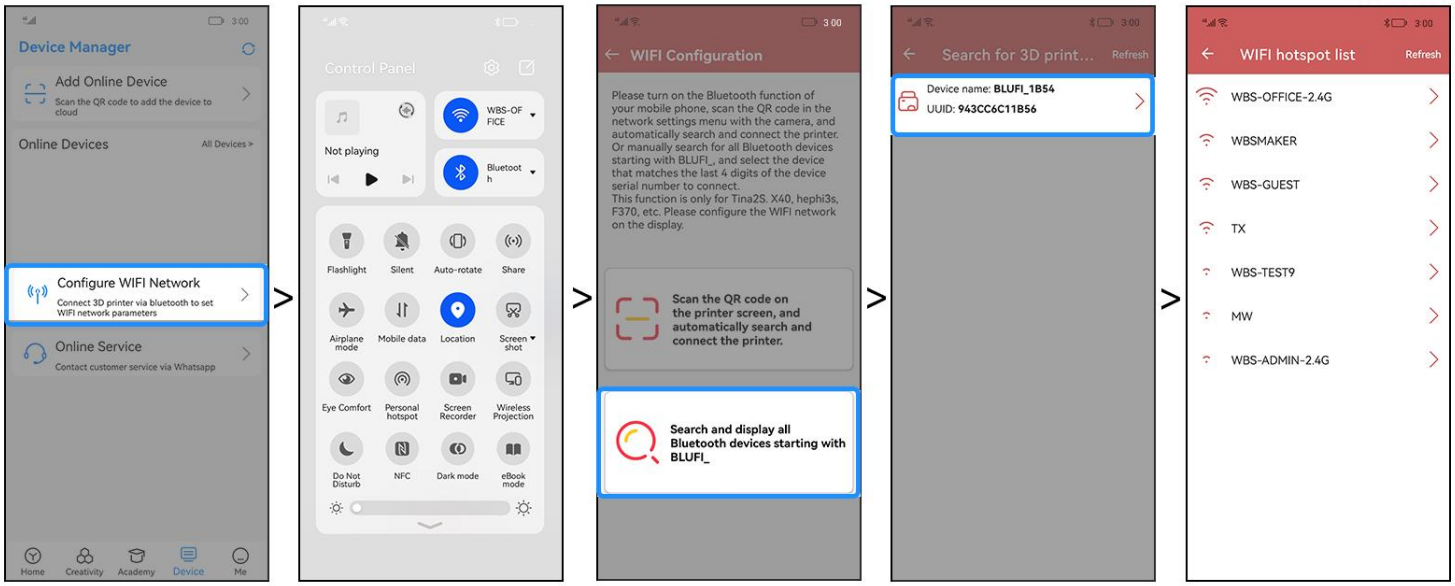
GRADUALLY DECREASE IF GAP TOO LARGE (POOR ADHESION)

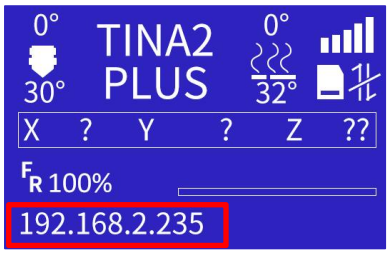
5.PRINT BY APP

1.Download and install the APP : PoloPrint Cloud

2.Register or log in to the APP.

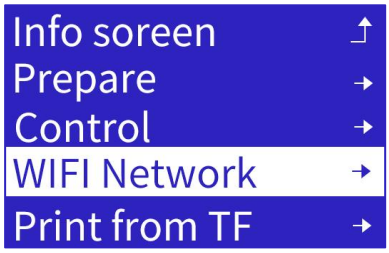
3.Configure WiFi for 3D printer.

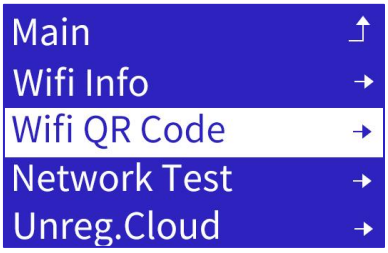




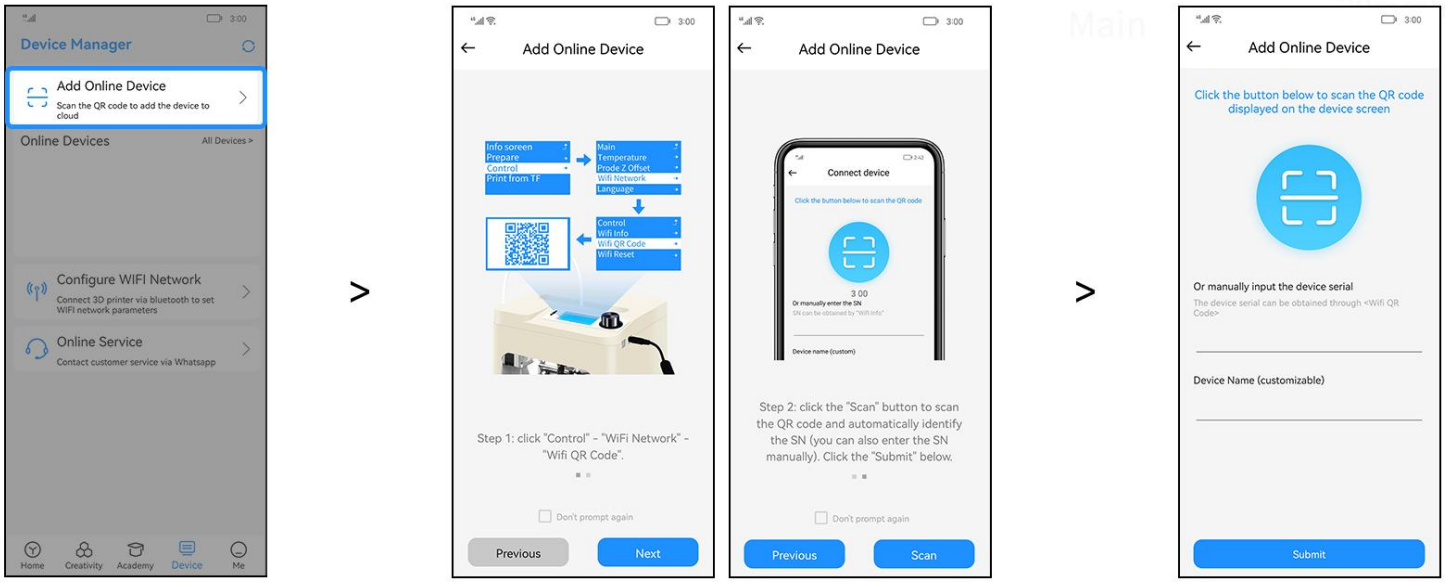
4.Get the “ Wifi QR Code ”.

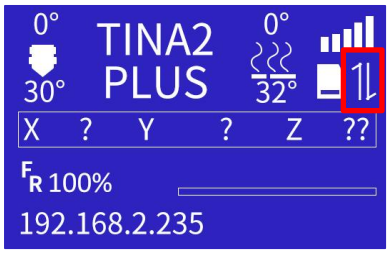
NOTE: THE 3D PRINTER HAS BEEN CONNECTED TO THE NETWORK, AND THE IP ADDRESS IS DISPLAYED.



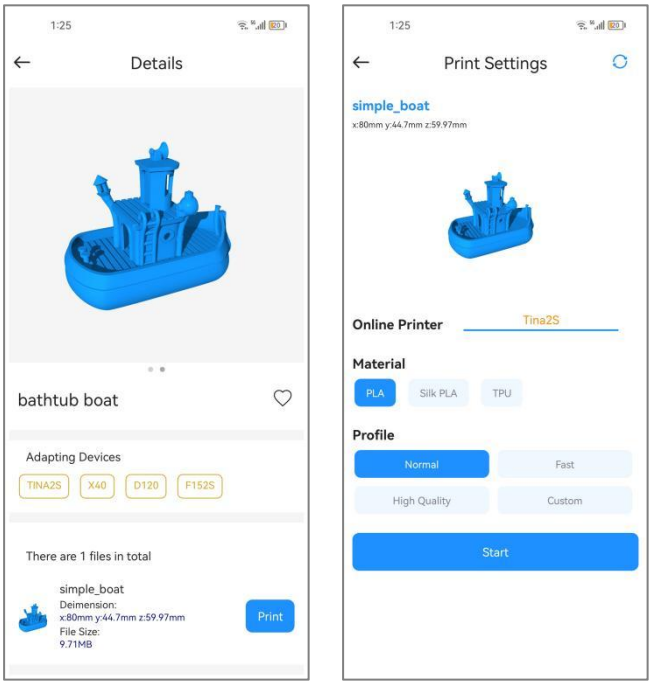


5.Click "Add Online Device ”, scan the QR code displayed on the screen, and submit the recognized SSID.





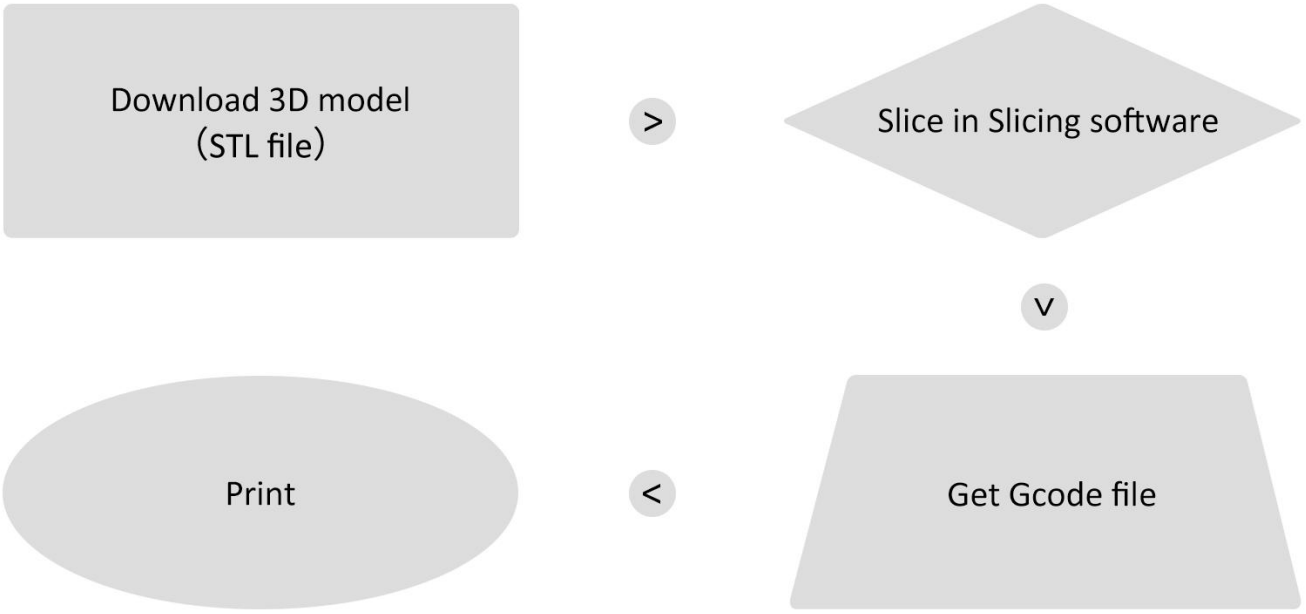
6.Select a model and click "Print".



6.WIIBUILDER SLICING SOFTWARE

3D models usually STL files, not directly recognized by 3D printers. Models processed in slicing software to generate Gcode files for printing.

3D printing process:



Slicing software installed on computer. Common options: Wiibuilder, Cura. Wiibuilder developed in-house, user-friendly, recommended.

Wiibuilder included on printer's TF card. Use card reader to view TF contents on PC for installation.

System requirements for Wiibuilder:

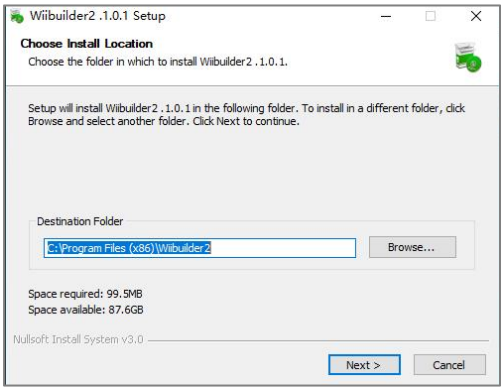
- CPU: Pentium 1GHz
- RAM: 1GB minimum
- Disk: 1GB+ free space
- Display: 640x480 minimum
- Graphics: 3D acceleration, OpenGL 2.0+
- Color: 256 colors minimum
- OS: Windows 7+ / MacOS 10.13.6+
- Other: .Net Framework 4.5.2+

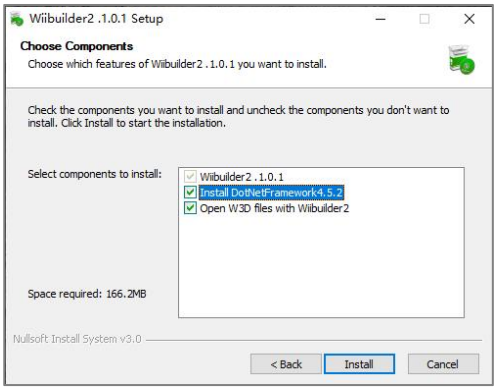
6.1.Installation

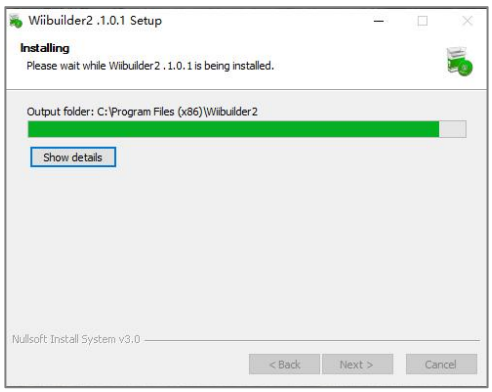
6.1.1.Installation on Windows

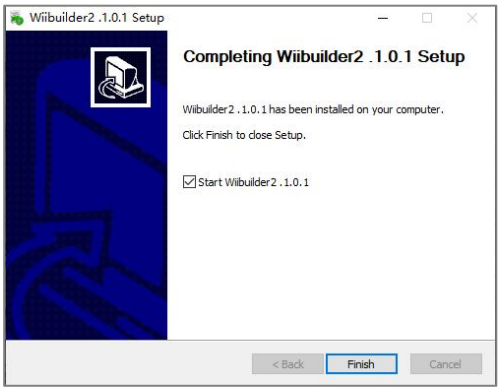
Steps to install Wiibuilder:

Locate and run Wiibuilder.exe on TF card. Click Install > Next > Finish to continue.









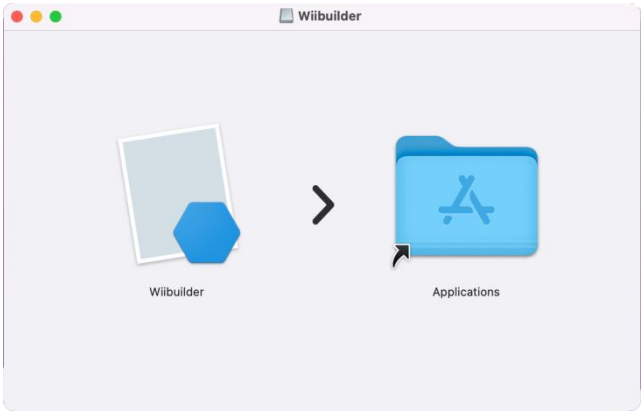
6.1.2.Installation on MacOS

OS: MacOS 10.13.6+

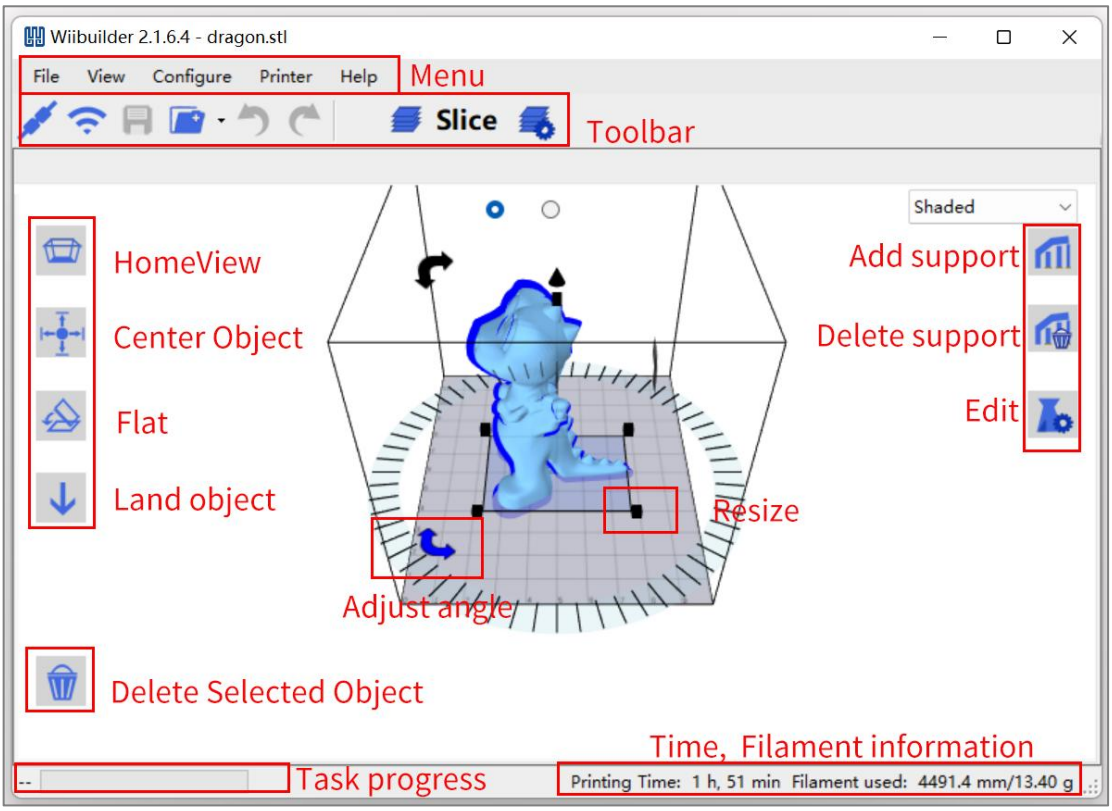
Decompress installer. Double-click package, drag Wiibuilder to Applications.

Wiibuilder-2.3.0.0

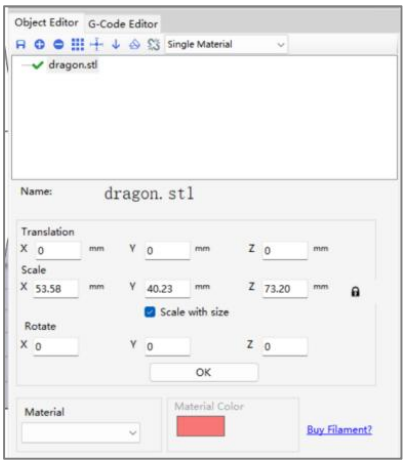
Wiibuilder-2.3.0.0



6.2.Interface Introduction

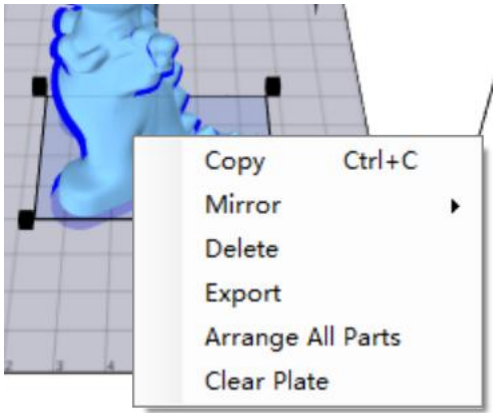


- Menu Bar: Access Wiibuilder menu
- Toolbar: USB/WiFi connect, save Gcode, load model, undo/redo, slice, settings
- Home View: Restore default view angle
- Center Object: Auto-adjust model XY to center on platform
- Flat: Auto-flatten tilted model on platform
- Land Object: Auto-adjust model Z to align bottom with platform
- Delete Selected: Remove current model
- Add/Delete Support: Manually add/remove model supports
- Edit: Open model and Gcode editor

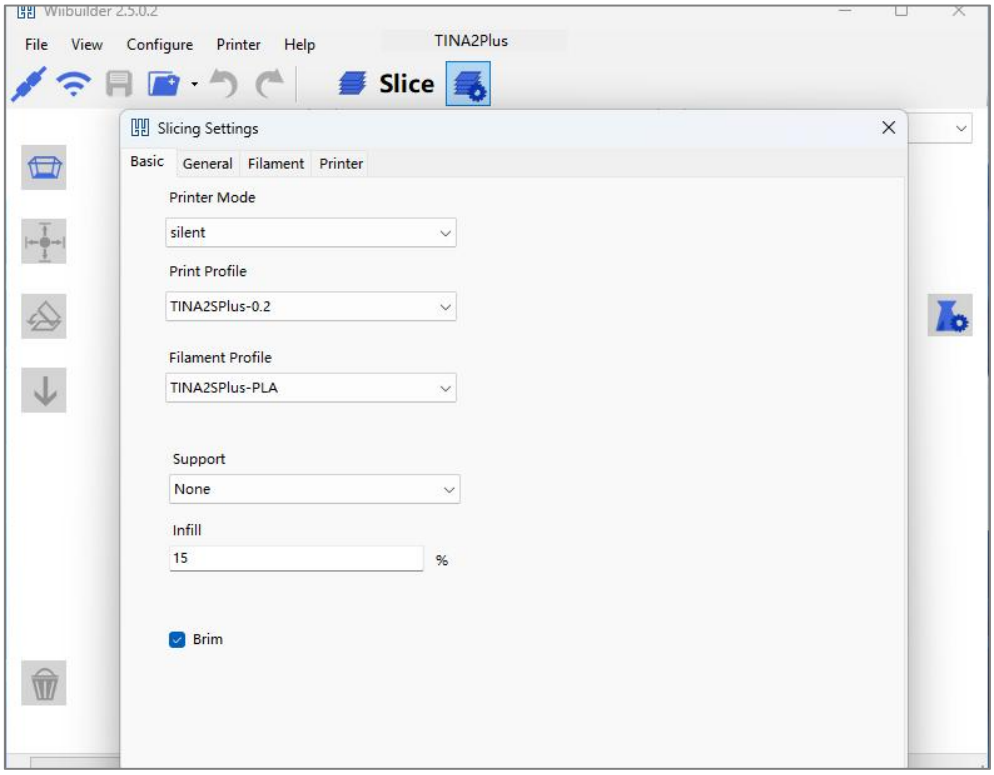


Right-click model for context menu:

- Copy model
- Mirror on X/Y/Z axis
- Delete model
- Export model to STL
- Arrange All Parts: Auto-adjust multi-model position and spacing
- Clear Plate: Remove all models

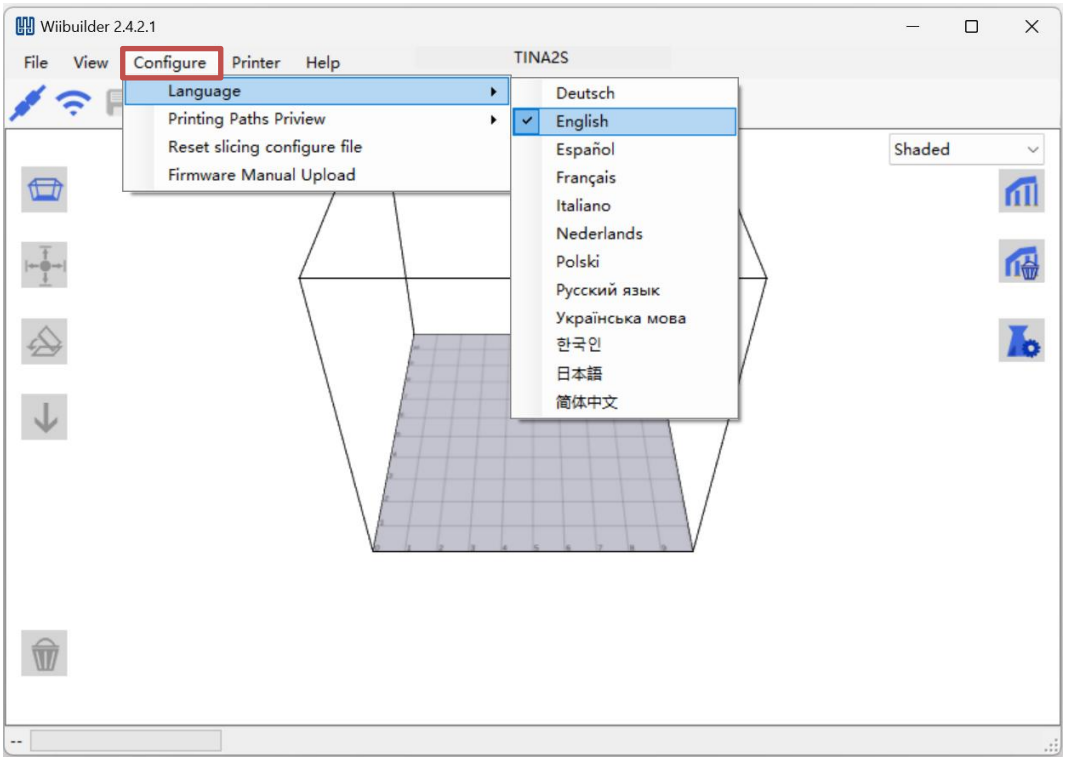


Slicing Settings:



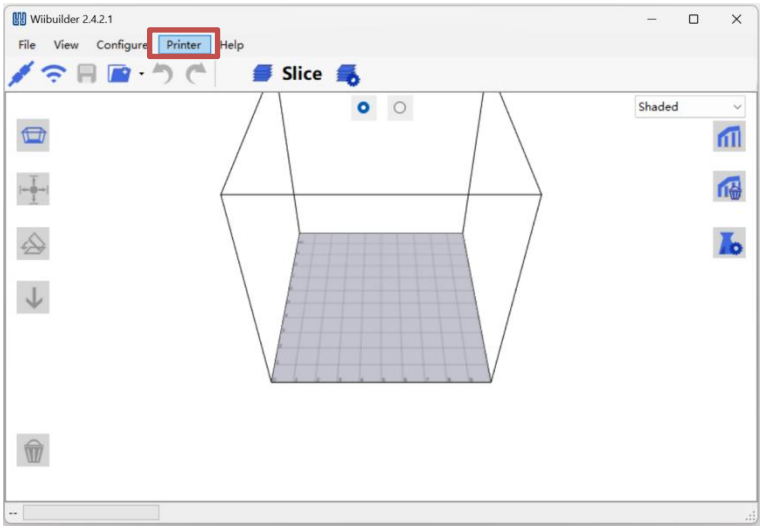
6.3.Select Language

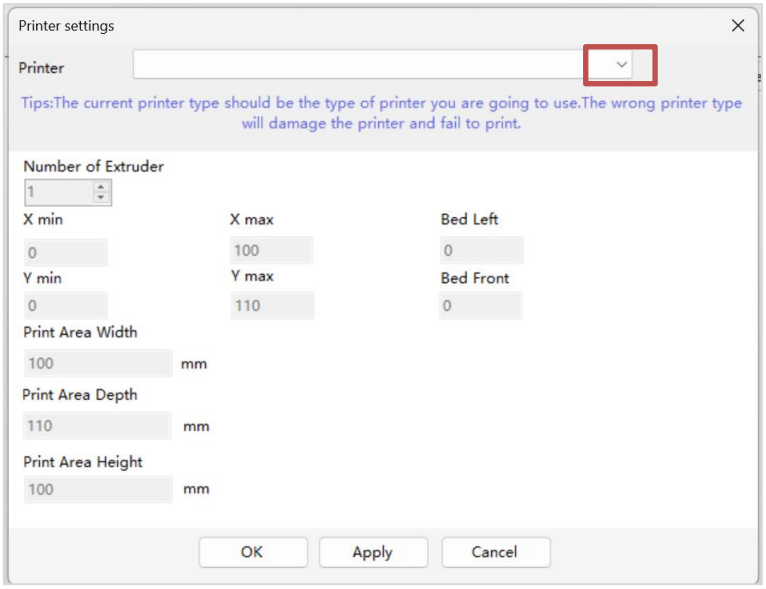
Open Wiibuilder, click Configure > Language to select.



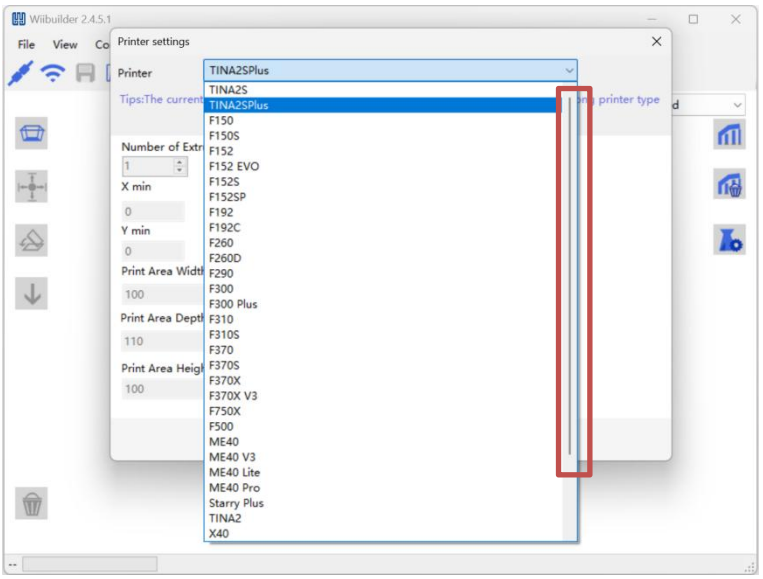
6.4.Select Printer

Click Printer to select model.



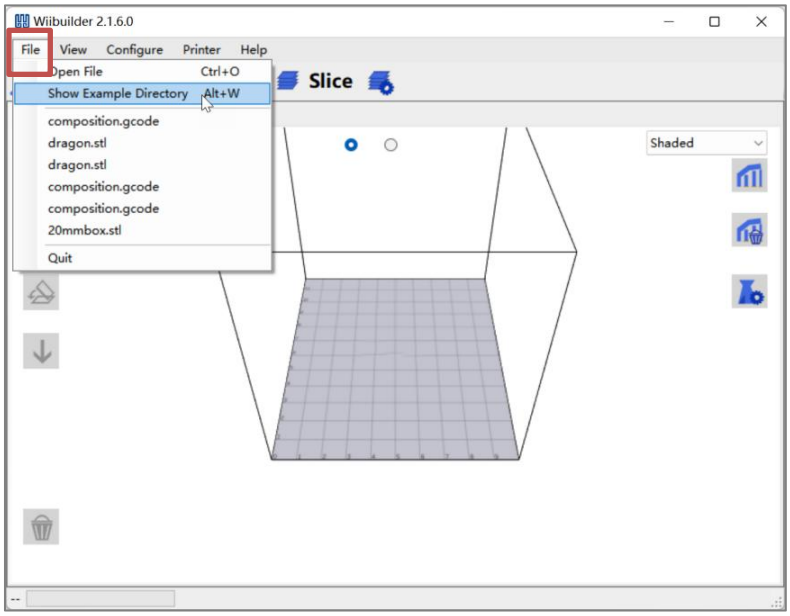


Scroll to find "TINA2Plus".Note: Not "TINA2"or"TINA2S", will fail if wrong.

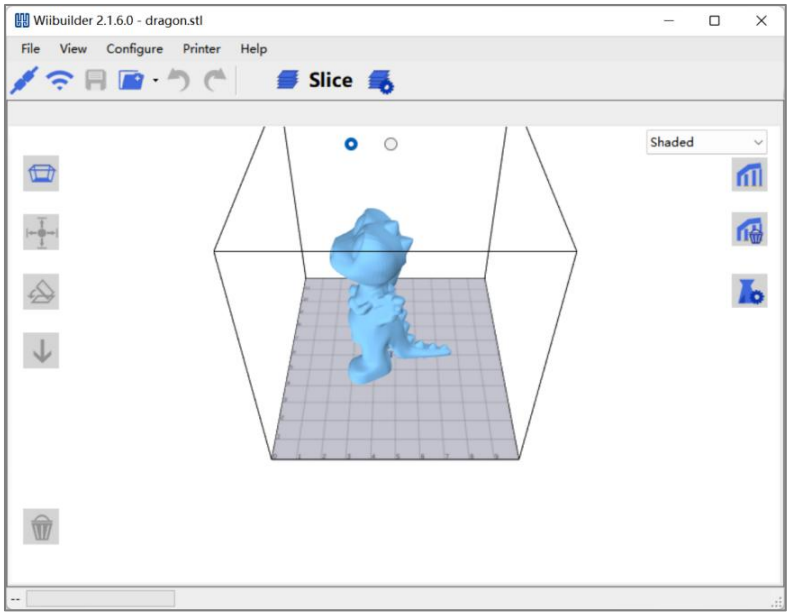


6.5.Add Model

Open Wiibuilder, click File to load model or drag model in.

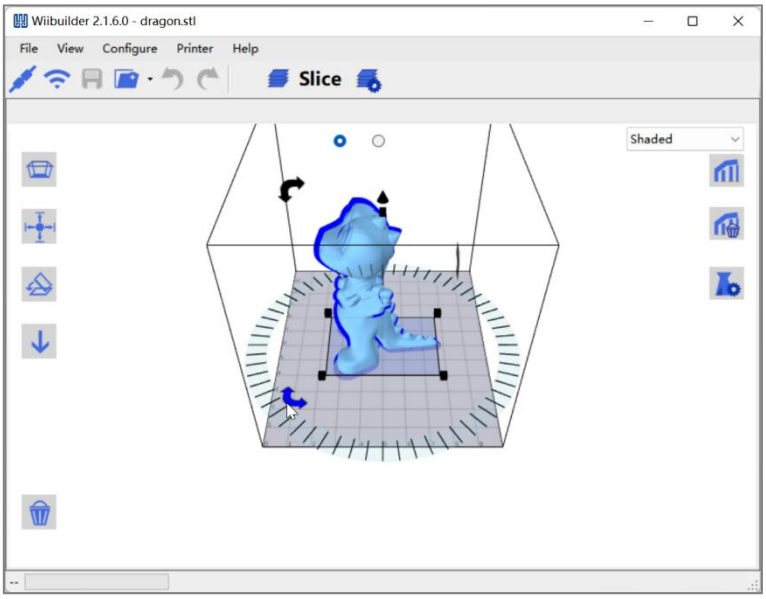


Five sample models in "Show Example Directory" for test printing.

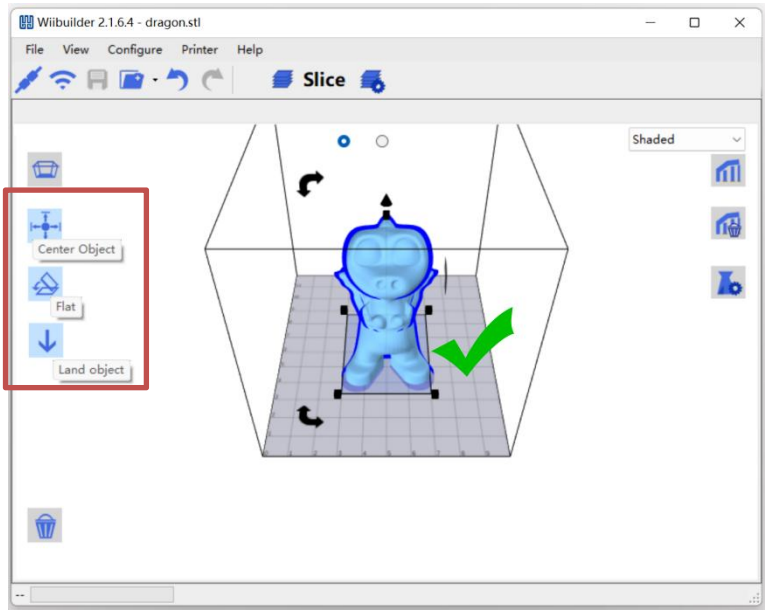


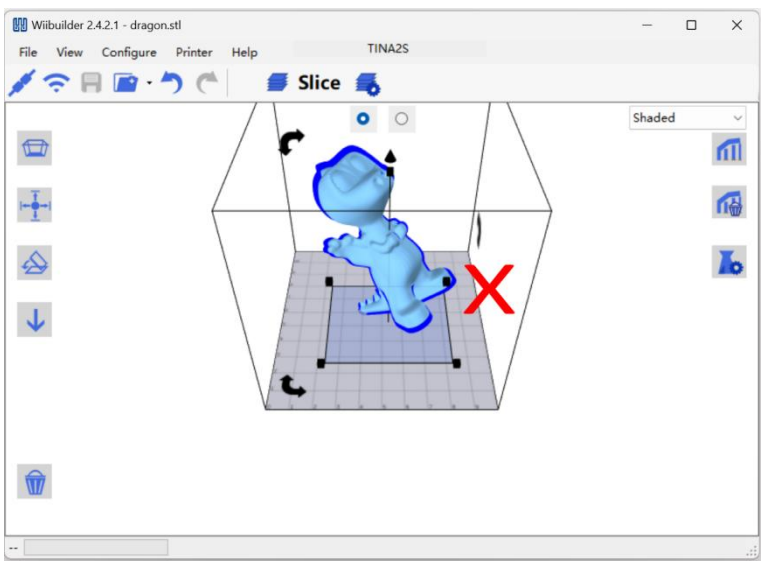
6.6.Adjust Model

Click model to adjust size, angle, position.



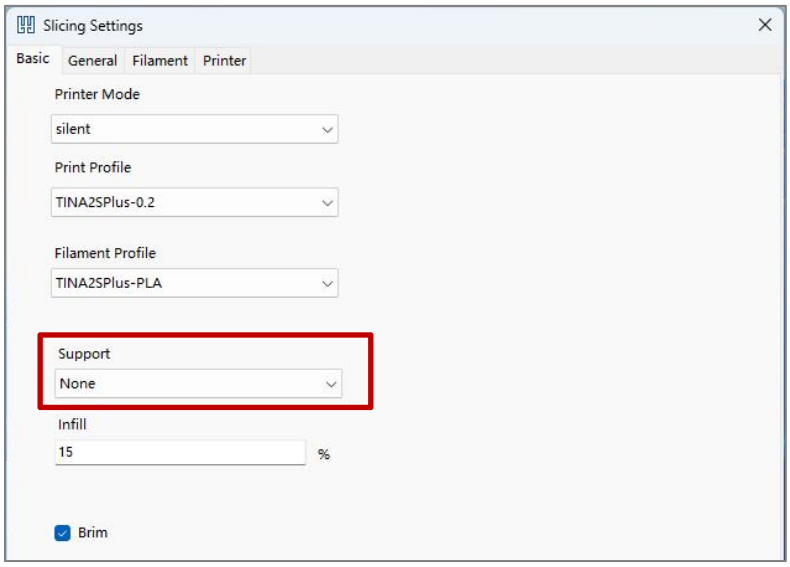
Model should be centered and properly placed on platform. Click Center Object, Flat, Land Object. If floating, print fails.





6.7.Basic Parameter Settings

Beginners can use default parameters and simply add support based on the model's situation.



1. Printer Mode:

   High Speed: Select this mode when the printing speed exceeds 80mm/s.

   Silent: Select this mode when the printing speed is below 80mm/s.

2. Print Profile:

Print Profile refers to printing precision. The finer the precision, the slower the extruder speed; the coarser the precision, the faster the extruder speed. Generally, a 0.2 mm setting is used. If the filament is thick and the flow speed is too slow, it may affect the print quality.

3. Filament Profile:

Choose either PLA or TPU filament.

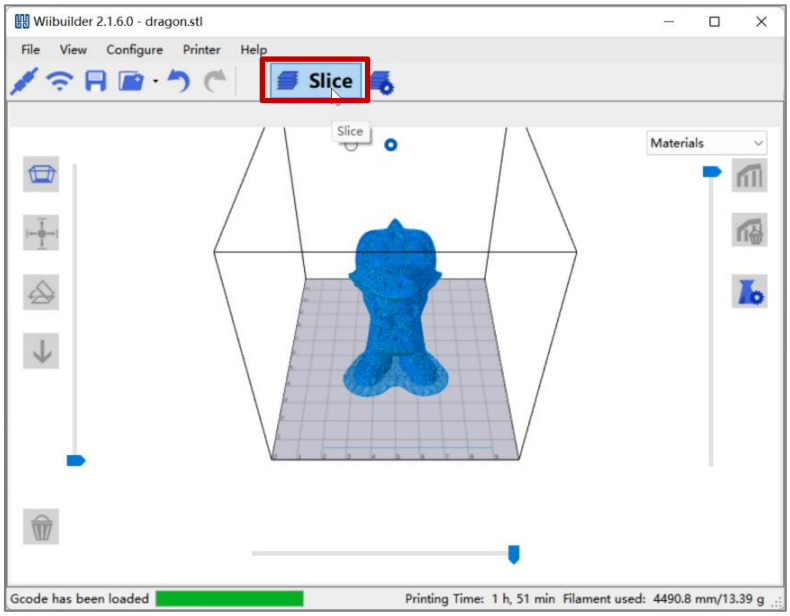
4. Support:

Some models do not require support;If the angle exceeds 60° , Touching buildplate support is required.Hollow models require everywhere support.

5. Infill:

Infill refers to the internal density of the model. The less fill density, the faster the print. However, too little fill density can potentially lead to print failures.

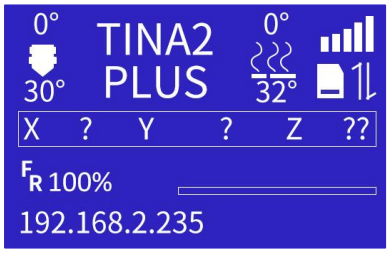
After setting parameters, click Slice to convert file.



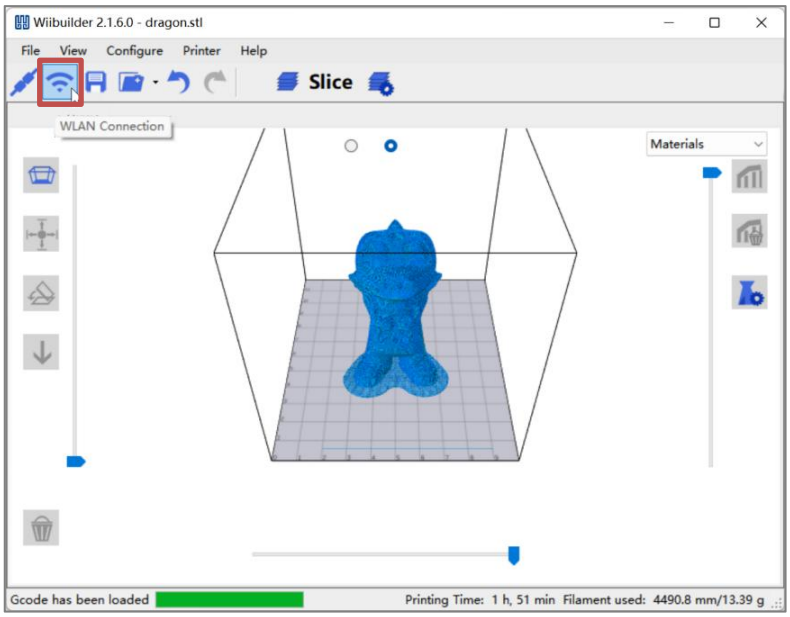
6.8.Send Files via Network

NOTE: THE 3D PRINTER HAS BEEN CONNECTED TO THE NETWORK, AND THE IP ADDRESS IS DISPLAYED. FOR OPERATIONS, PLEASE REFER TO: "PRINT BY APP".

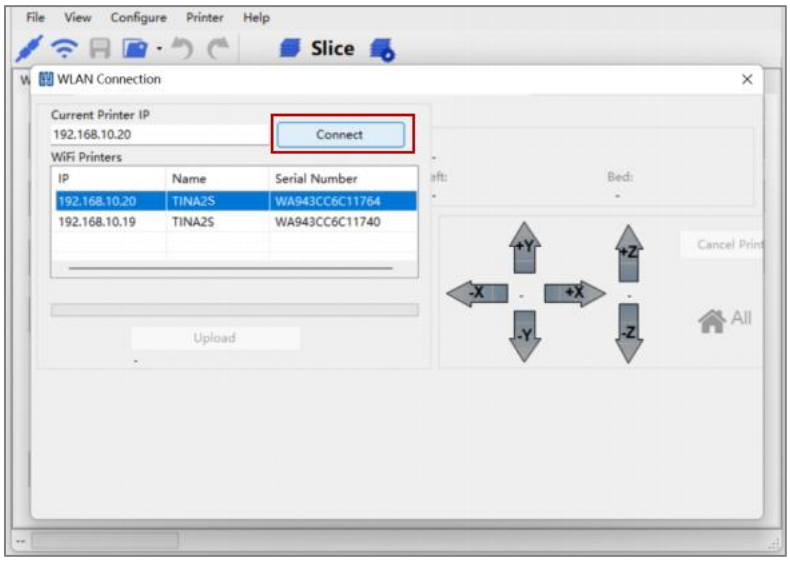
SELECT THE COMPUTER TO USE THE SAME NETWORK AS THE DEVICE. USING DIFFERENT NETWORKS CAN CAUSE TRANSFER FAILURES.



The computer should connect to the same local network with the 3D printer.

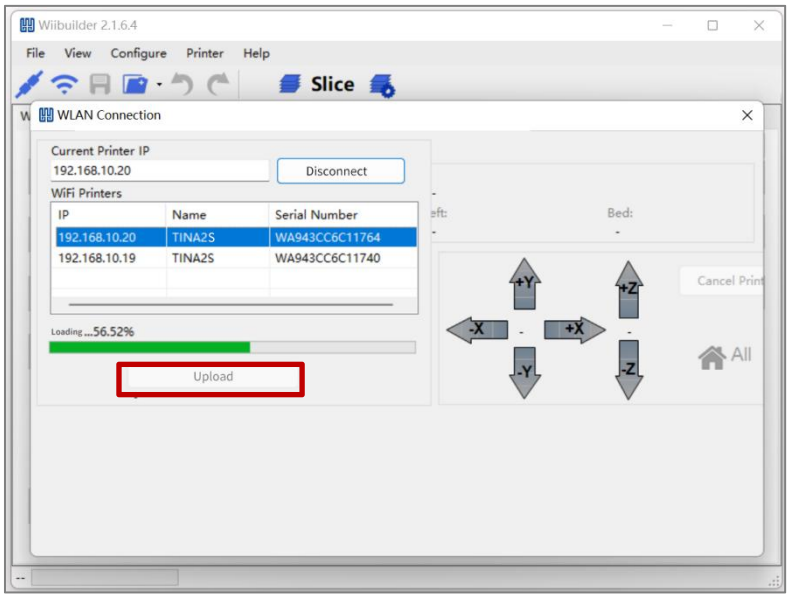


Click "WLAN Connection", the software will automatically search for the 3D printers in the local network and show them in the below list. If no 3D printer be found, you can also try to manually enter the IP address of the 3D printer to connect.



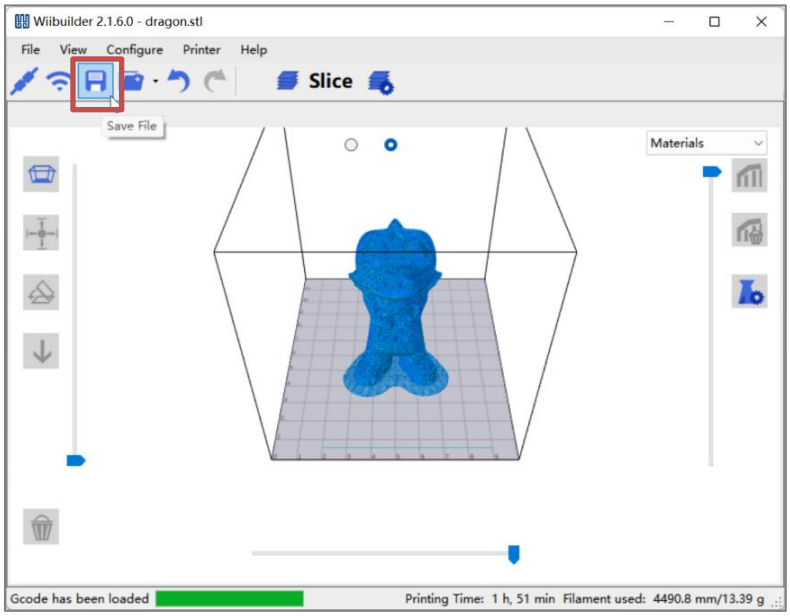
NOTE: IF YOU STILL CANNOT CONNECT TO THE 3D PRINTER, PLEASE CHECK WHETHER THE FIREWALL OF THE COMPUTER PROHIBITS THE SOFTWARE FROM NETWORKING, AND WHETHER THE COMPUTER AND THE 3D PRINTER ARE IN THE SAME LOCAL AREA NETWORK.

Send the sliced model to the 3D printer, and the 3D printer starts printing after receiving it.

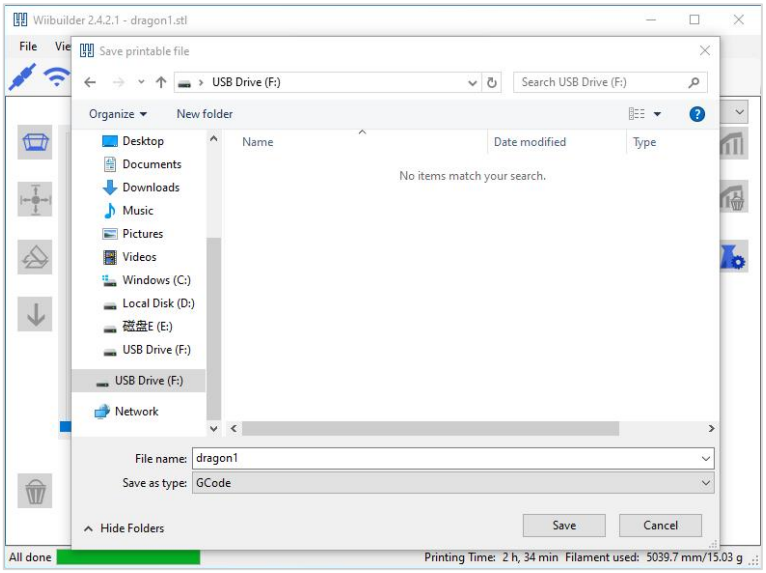


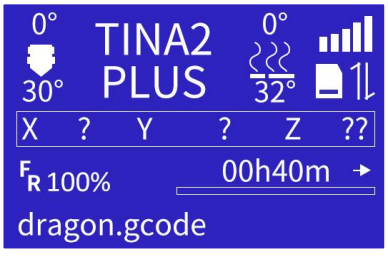
6.9.Send Files via TF Card

Save sliced model to TF card, insert card into printer.



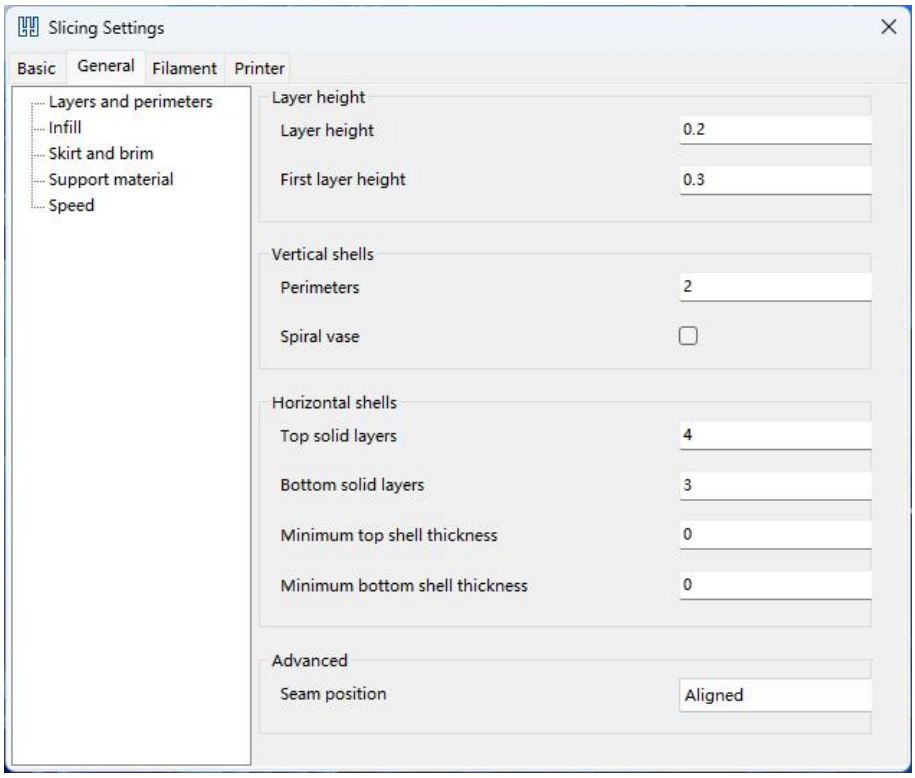
Note: Save directly to TF root, not in folder.



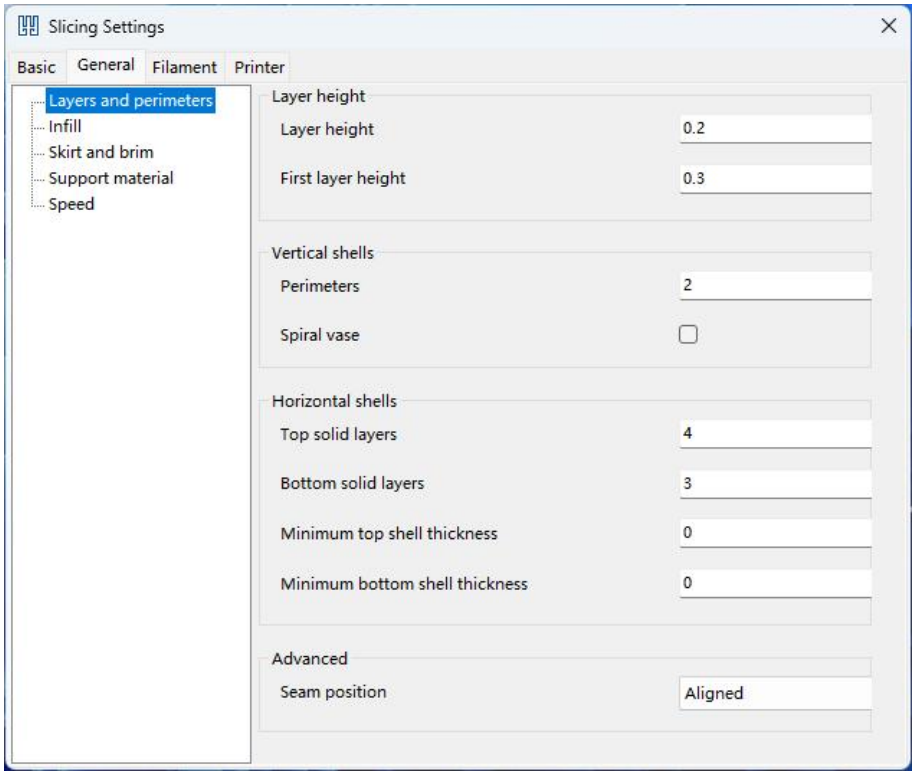


6.10.Advanced Parameter Settings

6.10.1.General



1. Layers and Perimeters:



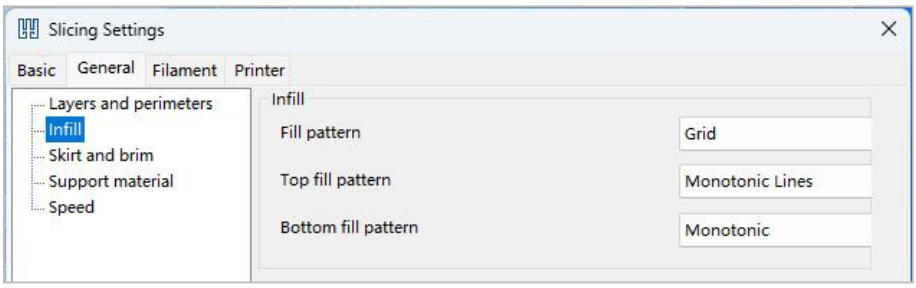
- LAYER HEIGHT: Determines the thickness of each printed layer. Lower values result in finer detail and smoother surfaces but increase printing time.

- PERIMETERS/WALL LINE COUNT: Sets the number of outer layers or walls of the print. Increasing this number enhances the model's strength and durability.

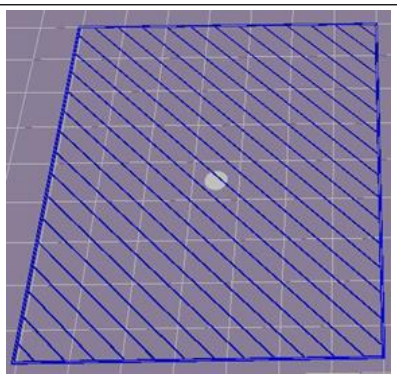
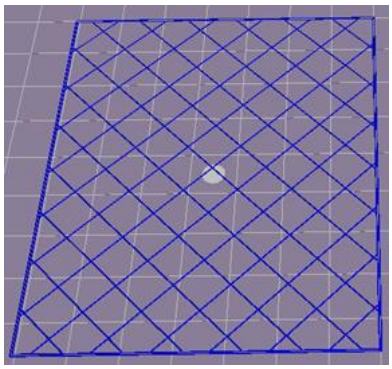
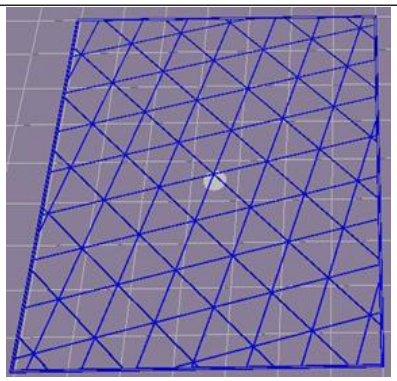
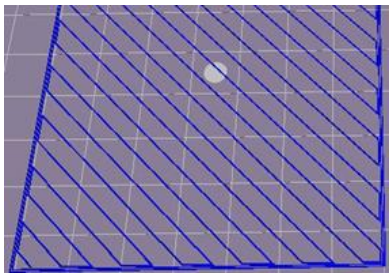
- TOP LAYERS: The number of solid layers printed on top of the infill. More top layers provide a smoother surface on the top of the print.

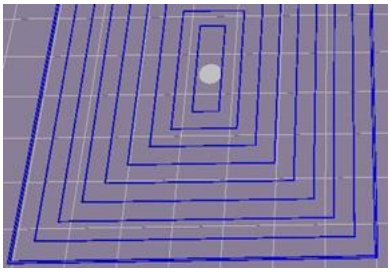
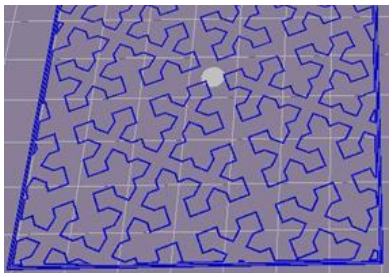
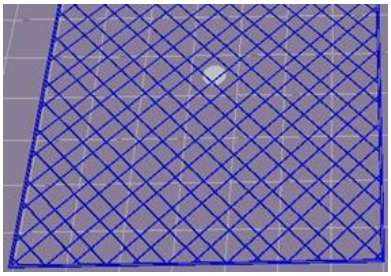
- BOTTOM LAYERS: The number of solid layers printed underneath the infill. More bottom layers ensure better adhesion to the print bed and a solid base for the print.

2. Infill:

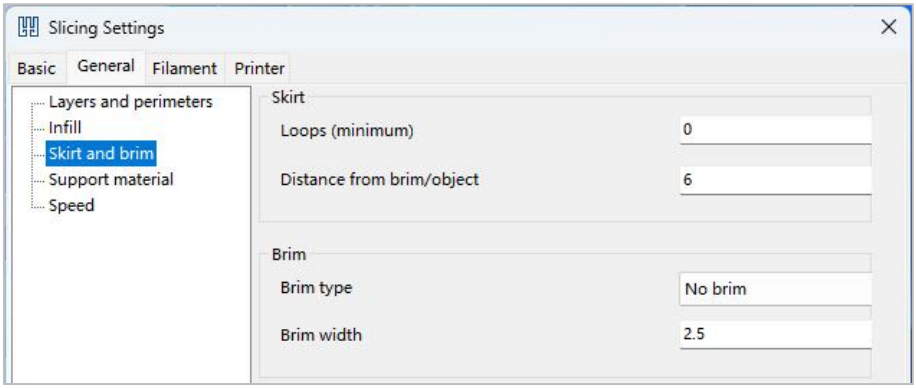


- INFILL: Select from Lines, Grid, Triangles, Zig Zag, Concentric, Cross, Octet. Patterns shown below.

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

	
Concentric	**Cross**
	
Octet	

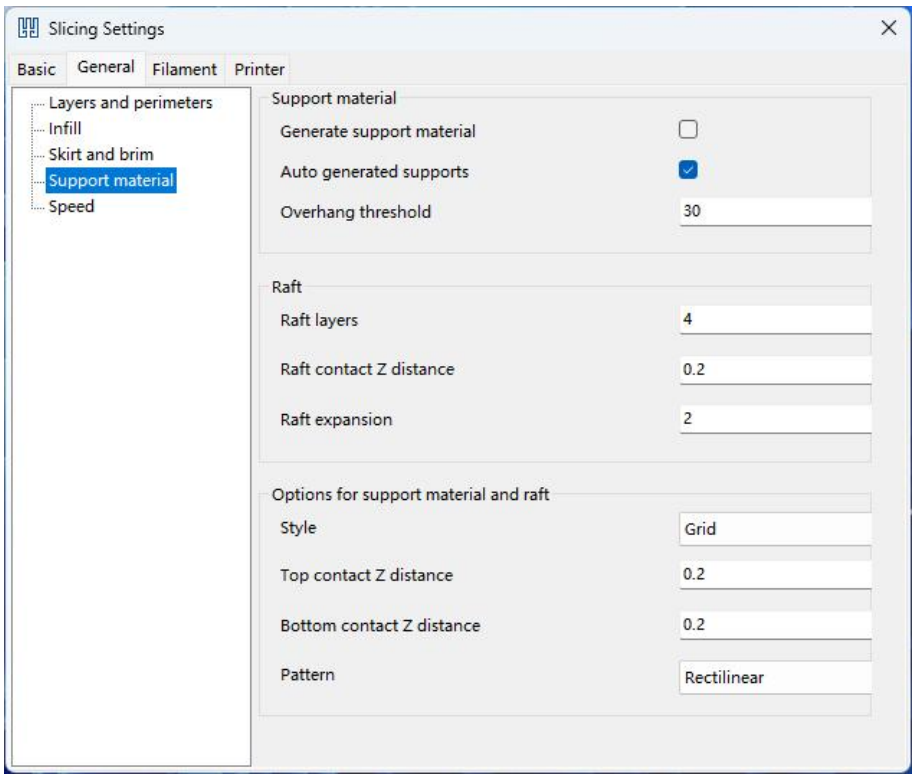
3. Skirt and brim:



- SKIRT: A set of lines printed around the model but not in contact with it. It helps to prime the nozzle and checks the print bed adhesion before starting the actual print.

- BRIM: Adds a few rows of extra material around the base of the print to help with adhesion, particularly useful for prints with small contact areas or that tend to warp.

4. Support material:



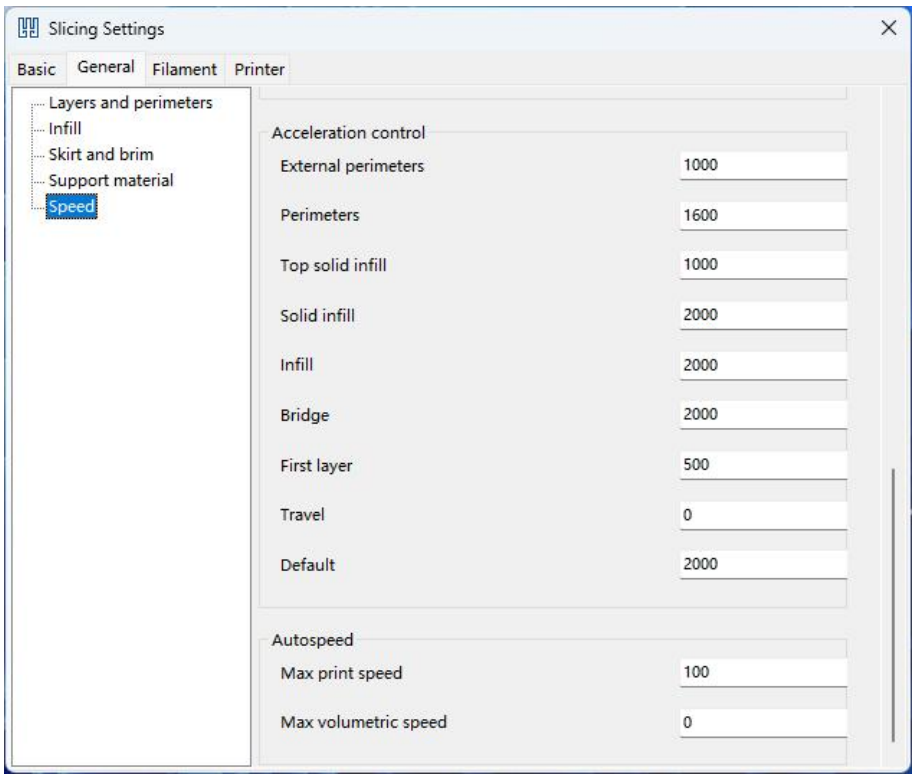
- OVERHANG THRESHOLD (SUSPENSION THRESHOLD):This setting defines the angle at which overhangs start to require support. For instance, if you set it to 45 degrees, any part of the model that extends beyond this angle will have support structures generated underneath it. This helps manage overhangs and prevents sagging or collapsing during printing.

- RAFT LAYERS: This setting specifies the number of layers of the raft that will be printed. Typically, a raft includes several types of layers:

- RAFT CONTACT Z DISTANCE: The Raft Contact Z Distance is an important setting that controls the vertical spacing between the raft and the model. This distance helps ensure that the raft serves its intended purpose—providing a stable and level base for the print—while also making it easier to remove the raft once printing is complete.

- RAFT EXPANSION :refers to a setting that controls the horizontal expansion of the raft beyond the dimensions of the model. This feature helps to improve bed adhesion and stability, especially for models with small bases or challenging geometries.

5. Speed:



- PERIMETERS SPEED (MM/S): Definition: This setting controls the speed at which the printer moves while printing the perimeter layers of the model. Perimeters are the outer walls of the print.Typical Value: Often set between 30 to 50 mm/s. Slower speeds can improve the quality of the outer walls, resulting in smoother surfaces, but will increase print time.

- SMALL PERIMETERS SPEED (MM/S): Definition: This speed is specifically for printing small perimeters or narrow features of the model. These are typically areas where the width of the perimeter is narrower than usual.Typical Value: Generally lower than the standard perimeters speed to ensure higher precision and better quality for fine details. Values might range from 15 to 30 mm/s.

- EXTERNAL PERIMETERS SPEED (MM/S): Definition: This speed applies to the outermost perimeters of the model, which are the external walls. Printing external perimeters slower can enhance the quality of the visible surfaces.Typical Value: Usually set lower than the general perimeters speed to achieve a smoother finish on the outer edges. Typical values might be between 20 to 40 mm/s.

- INFILL SPEED (MM/S): DEFINITION: This setting controls the speed at which the printer moves while filling the internal structure of the model. Infill patterns create the internal support structure of the print.Typical Value: Typically set between 40 to 80 mm/s. Faster infill speeds can reduce print time but might impact the quality and strength of the internal structure.

- SOLID INFILL SPEED (MM/S): Definition: Solid infill speed applies to the denser areas of infill where the printer creates a solid fill without gaps. This is usually for areas like the base or top layers where more material is used.Typical Value: Often slower than the general infill speed to ensure good adhesion and even filling. Common values are between 30 to 50 mm/s.

- TOP SOLID INFILL SPEED (MM/S): Definition: This speed is used specifically for the solid infill layers on the top of the model. These layers are crucial for a smooth and solid top surface.Typical Value: Usually set slower to improve the surface finish on the top of the print. Values might range from 20 to 40 mm/s.

- SUPPORT MARERIAL SPEED (MM/S): Definition: Controls the speed at which the printer moves while printing the support structures. Supports are used to prevent overhangs and provide stability for complex geometries.Typical Value: Often set between 20 to 40 mm/s. Slower speeds can improve the quality and reliability of the support structures.

- SUPPORT MARERIAL INTERFACE SPEED (MM/S): Definition: This speed applies to the interface layers of the support structures, which are printed between the support and the model. The interface layers provide a smoother surface for the underside of the model.Typical Value: Typically set slower than the general support material speed to ensure a good quality interface. Values might range from 20 to 30 mm/s.

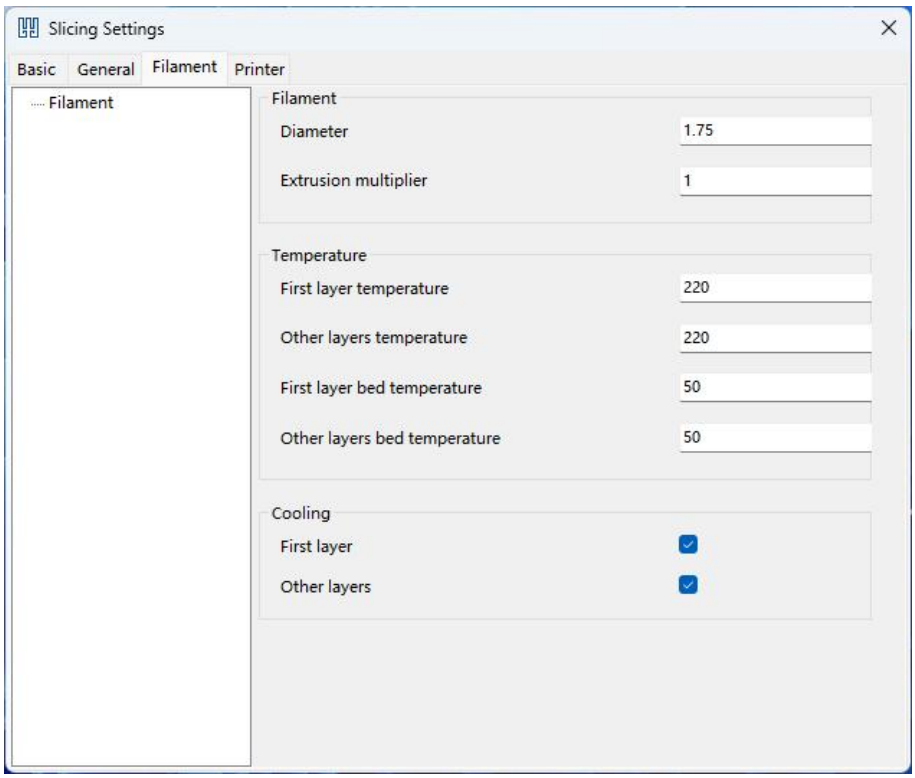
- RAFT MATERIAL SPEED (MM/S): DEFINITION: This setting controls the speed at which the printer extrudes the material for the raft. The raft is a base layer printed underneath the model to improve adhesion and stability.Typical Value: Generally set between 20 to 40 mm/s. Printing the raft slower can help improve the adhesion to the print bed and the overall stability of the raft, leading to a more reliable foundation for the model.

- RAFT MATERIAL INTERFACE SPEED (MM/S): Definition: This speed is specifically for the interface layers of the raft material, which are the layers between the raft and the model. These layers are designed to provide a smoother surface for the model to adhere to.Typical Value: Usually set slower than the general raft material speed, often between 15 to 30 mm/s. Slower speeds for the interface layers can help achieve a smoother surface, enhancing the quality of the model’s base.

- BRIDGES SPEED (MM/S): Definition: Controls the speed at which the printer extrudes material when printing bridges, which are horizontal or near-horizontal spans of material between supports. Bridges require careful handling to avoid sagging or collapsing.Typical Value: Often set between 20 to 40 mm/s. Slower speeds for bridges can improve print quality and reduce issues like sagging or stringing. However, you might need to balance this with the overall print time.

- GAP FILL SPEED (MM/S): Definition: This setting applies to the speed at which the printer fills in small gaps or incomplete areas in the model, often used in the final pass to cover holes or gaps in the print.Typical Value: Typically set between 30 to 50 mm/s. Gap fill speed is often higher than other detailed settings because these areas are generally less critical for fine detail but can still benefit from a balanced speed to ensure complete filling.

6.10.2.Filament



The diameter of the filament is fixed at 1.75 mm.The nozzle printing temperature and platform temperature can be modified.

PLA: Print Temp:180~220°C, Bed Temp:0~50°C

Silk PLA: Print Temp:210~230°C, Bed Temp:30~50°C

Gradient PLA: Print Temp:200~230°C, Bed Temp:40~50°C

Wood PLA: Print Temp:200~220°C, Bed Temp:40~50°C

PLA+: Print Temp:190~230°C, Bed Temp:30~50°C

PLA Pro: Print Temp:210~230°C, Bed Temp:30~50°C

TPU: Print Temp:190~235°C, Bed Temp:0~50°C

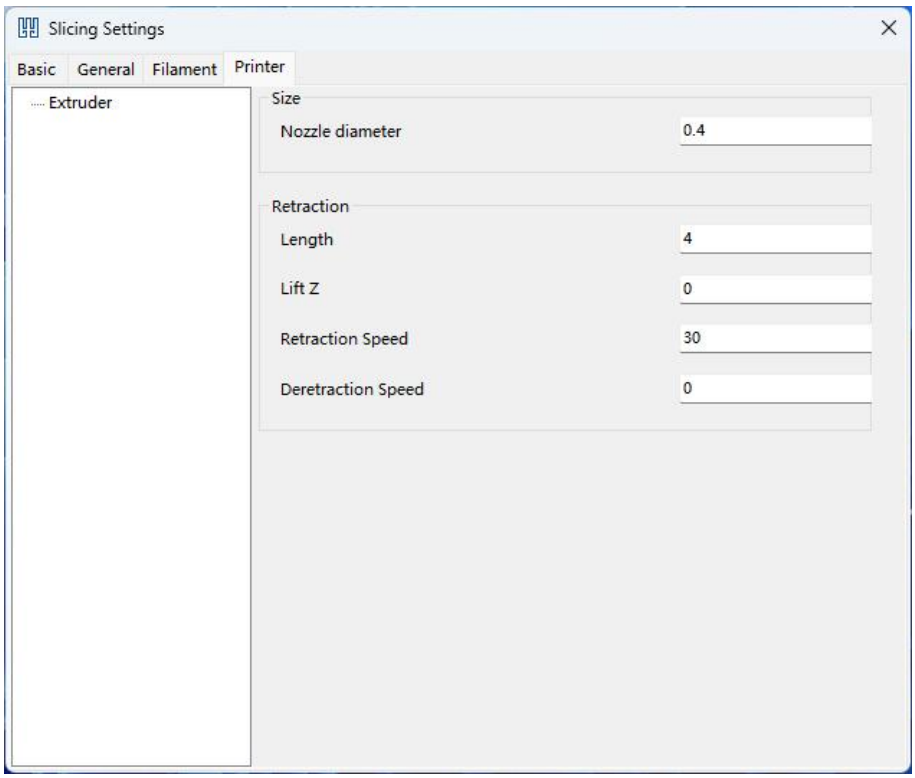
Applicable ambient temperature: 15~25°C

Winter: The temperature is lower than 15°C, which will cause the filament to become hard and brittle, not to come out, and will not stick. In order to ensure smooth silk output, it is necessary to increase the nozzle temperature by 10°C, set the platform temperature to 60°C, adjust Z-offset and apply glue. (Tina2 Basic adjusts the Z-offset, increases the first layer by 15~20°C,and increase the indoor temperature)

Summer: If the temperature exceeds 25°C, it will cause problems such as stringing and clogging.

Please lower the nozzle temperature appropriately. Summer will be more humid, and Silk PLA is easily affected by moisture. Please use it as soon as possible or store it in a sealed container.

6.10.3.Infill Tab



The nozzle diameter is fixed at 0.4mm.

- RETRACTION LENGTH:

Definition: The distance that the filament is pulled back into the nozzle during a retraction. This helps prevent oozing and stringing by reducing the pressure in the nozzle when moving between print areas.

Typical Values: Usually ranges from 0.5 to 6 mm, depending on the printer and filament type.

Shorter Lengths: Suitable for flexible or soft filaments to avoid jamming and clogging.

Longer Lengths: Often used for more rigid filaments like PLA or ABS to effectively reduce stringing.

Impact: Proper retraction length prevents excess filament from oozing out of the nozzle during travel moves, which improves print quality and reduces defects.

- RETRACTION LIFT Z:

Definition: The vertical distance that the nozzle lifts while retracting the filament. This is intended to avoid dragging the nozzle across the print and potentially damaging it.

Typical Values: Usually set between 0.5 to 2 mm.

Smaller Values: Minimizes lifting, which can be suitable for prints with close spacing or delicate details.

Larger Values: Helps in avoiding nozzle contact with the print, especially when moving across tall features or intricate details.

Impact: Proper lift Z helps reduce the risk of nozzle scraping or damaging the print, which is particularly useful when traveling over delicate or detailed parts of the model.

- RETRACTION RETRACTION SPEED:

Definition: The speed at which the filament is retracted and pushed back into the nozzle. This setting affects how quickly the printer can respond to changes in extrusion.

Typical Values: Ranges from 20 to 100 mm/s.

Slower Speeds: Provide more controlled retraction and can reduce the risk of clogging, especially for flexible filaments.

Faster Speeds: Reduce print time by quickly retracting and re-engaging the filament, but may increase the risk of filament grinding or jamming if set too high.

Impact: Optimal retraction speed prevents stringing and reduces the chances of filament issues, contributing to cleaner prints with fewer defects.

- RETRACTION DERETRACTION SPEED:

Definition: The speed at which the filament is reinserted into the nozzle after retraction (also known as “deretraction” or “un-retraction”). This affects how quickly the nozzle starts extruding again after a move.

Typical Values: Usually similar to or slightly different from retraction speed, often between 20 to 100 mm/s.

Matching Speed: When set the same as the retraction speed, it ensures consistent filament flow and helps avoid gaps or inconsistencies in extrusion.

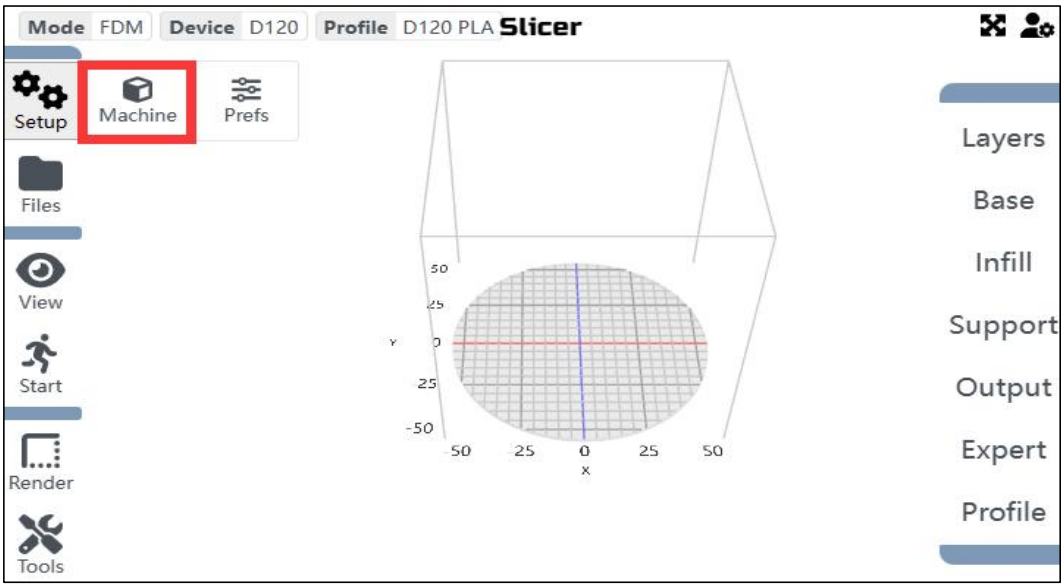
Different Speed: Adjusting it can help fine-tune how quickly the filament resumes extruding, which can be useful for specific materials or print quality improvements.

Impact: Proper deretraction speed ensures smooth transition back to extrusion, reducing the risk of defects such as gaps or blobs in the print.

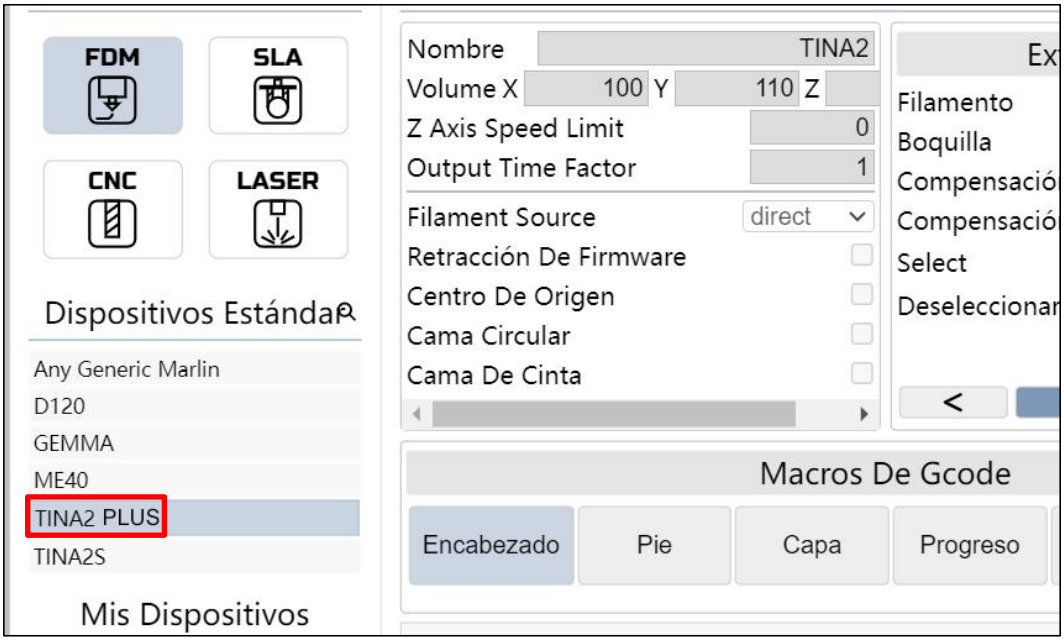
7.KIRI

Go to https://slice.wiibuilder.net/kiri/ for Chrome OS or iPad.When using KIRI slicing, the default plus setting is silent mode, which means the printing speed is less than 80mm/s.

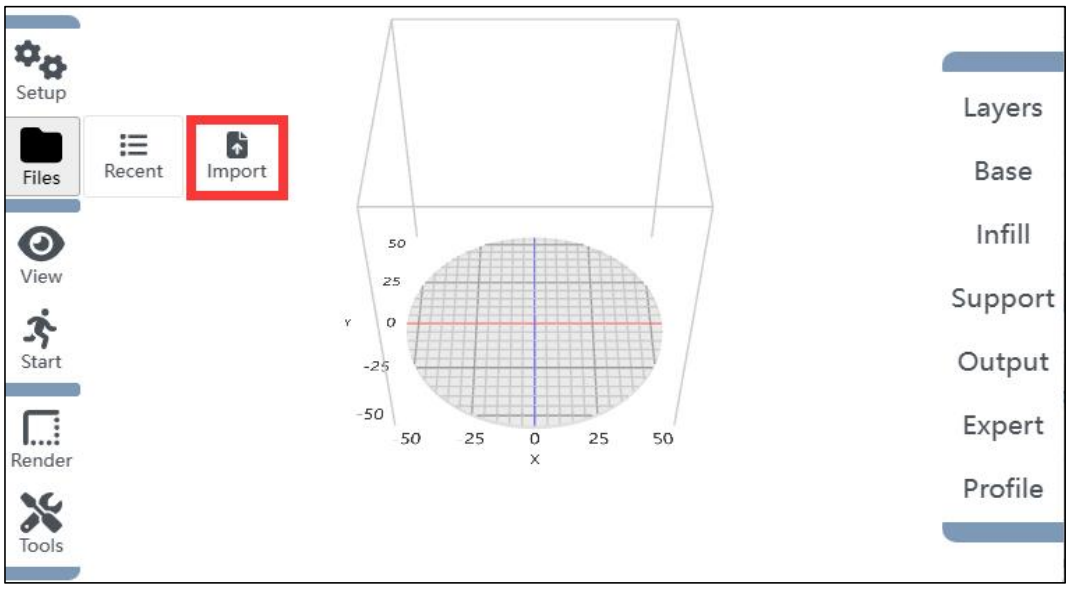
Setup > Machine to select printer model



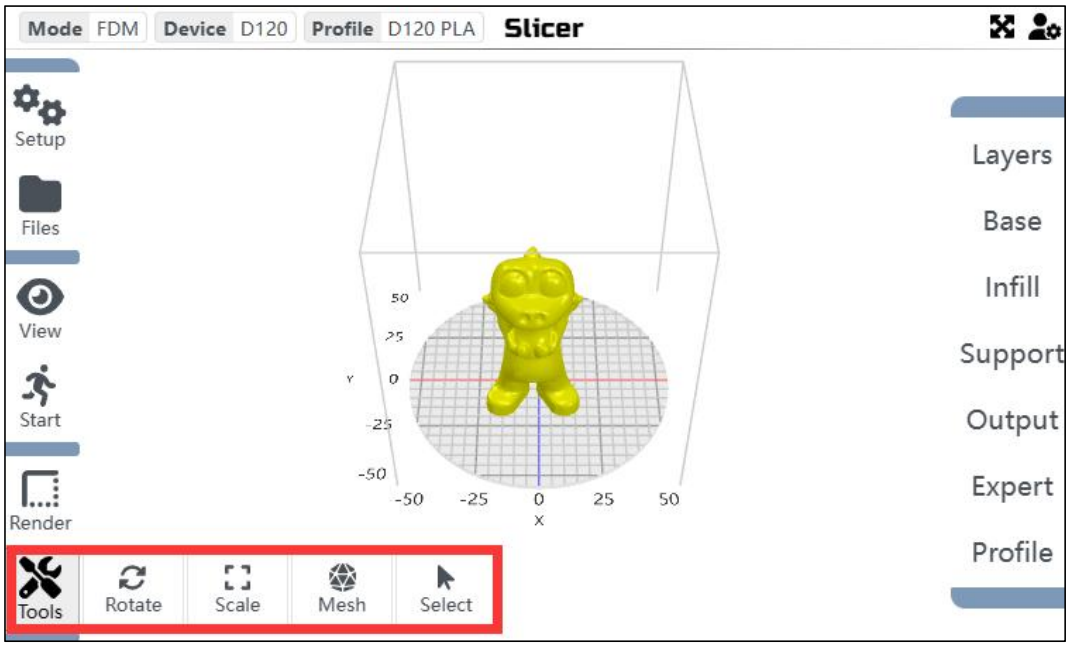
Find "TINA2Plus".Note: Not "TINA2"or"TINA2S", will fail if wrong.



Files > Import to load model



Tools to adjust model size and position.



Slice > Export to TF card for printing.