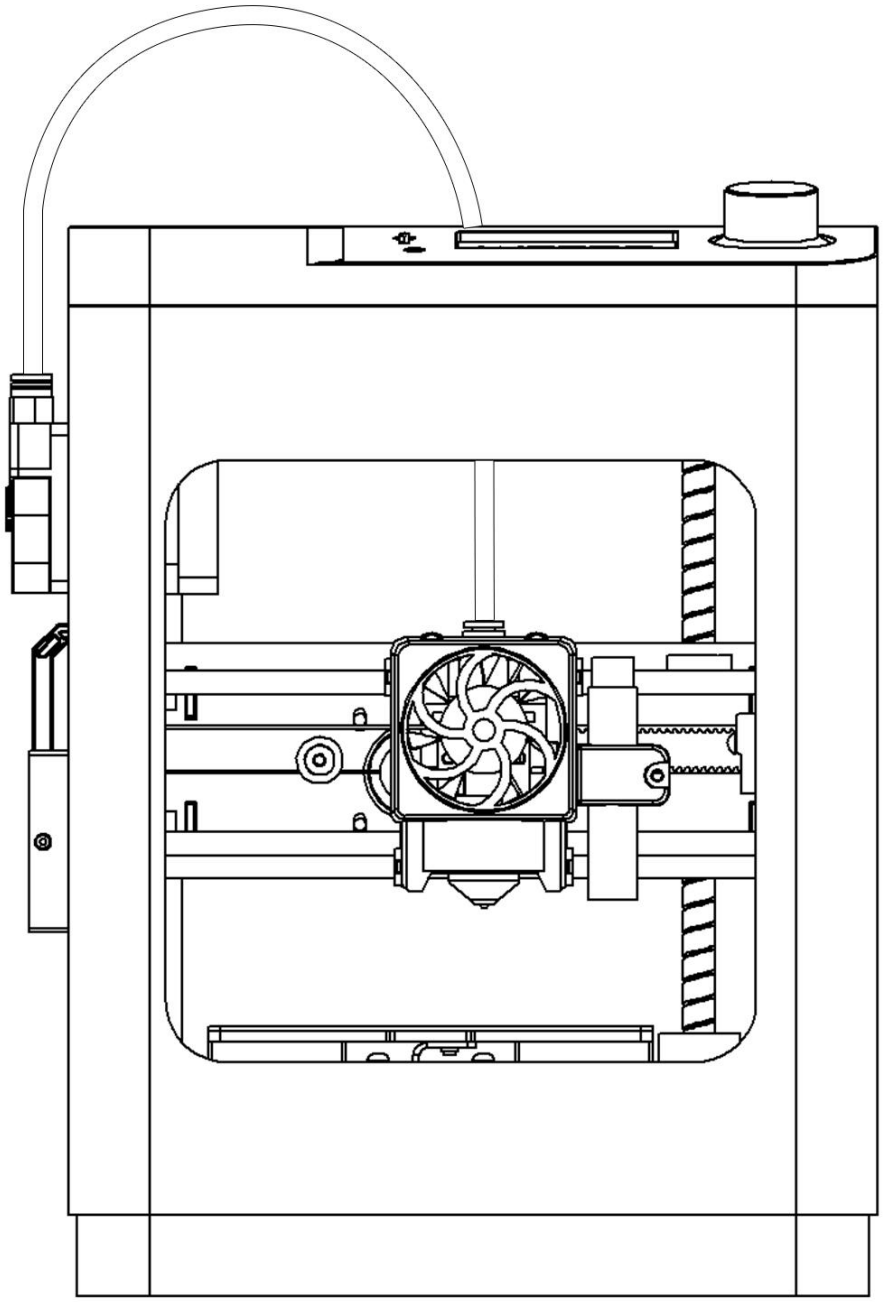
EN

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.

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.

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.

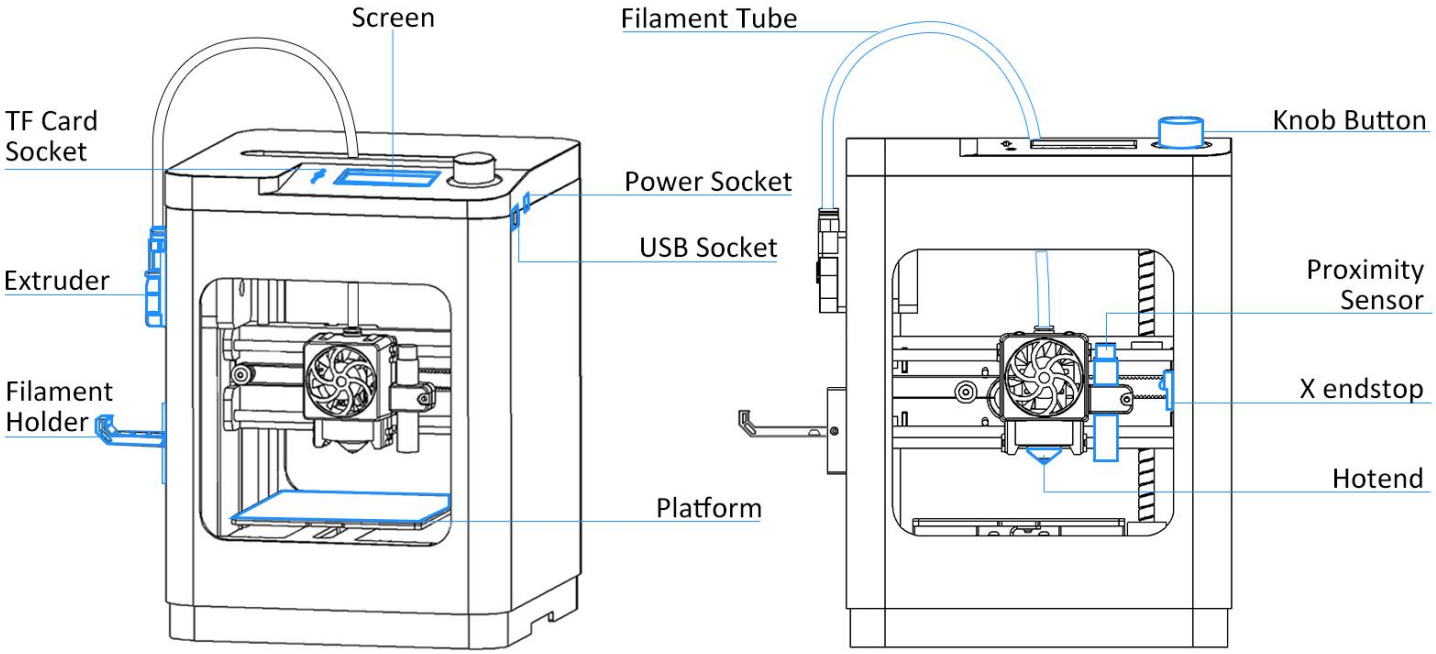
Above 30° C: Print quality will significantly decline and the nozzle may clog.

2. INTRODUCTION

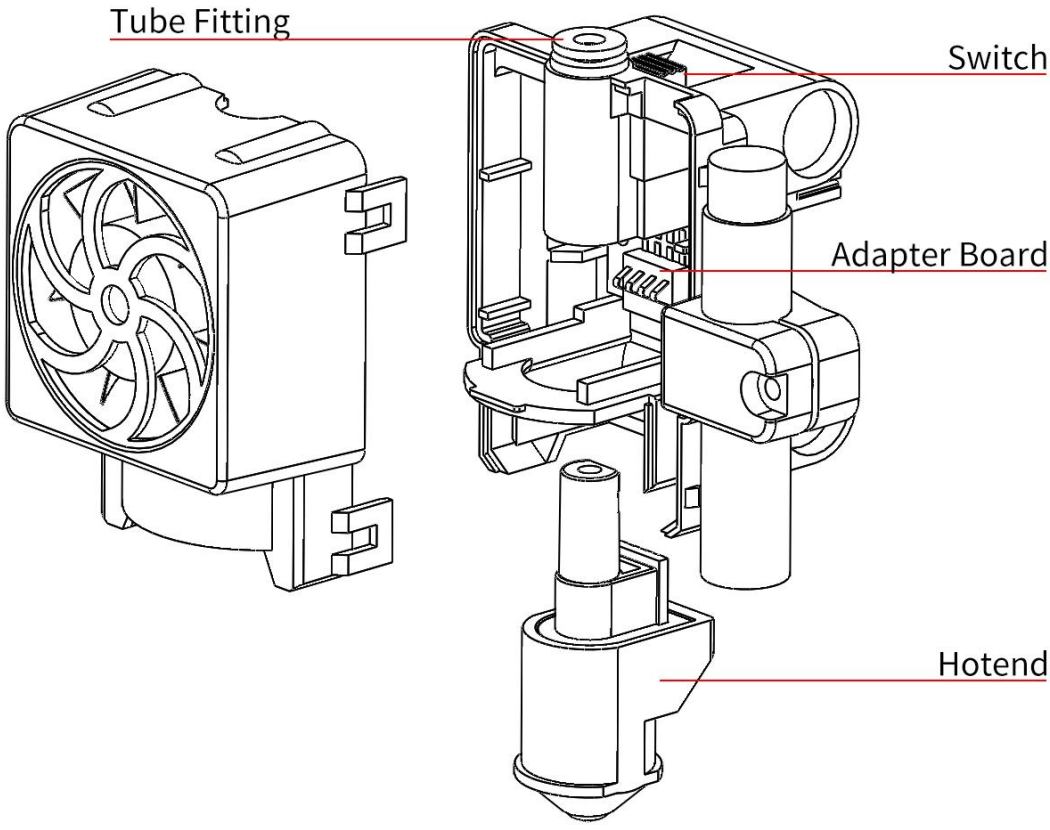
2.1. Specification

Model	TINA2 PLUS
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	Flexible Spring Steel
Leveling Tech	9-Point Auto Bed Leveling
Max Nozzle Temp	245°C
Motherboard	R72Q
Max Heated Temp	80°C
Max Print Speed	120mm/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) Cura (Win/Mac) Kiri (Chrome OS) OctoPrint (Pi/Linux)
Input File Format	STL/OBJ/AMF
Print Format	Gcode
Input Method	TF Card / WIFI / USB / APP
APP	PoloPrint Cloud (Android/IOS)

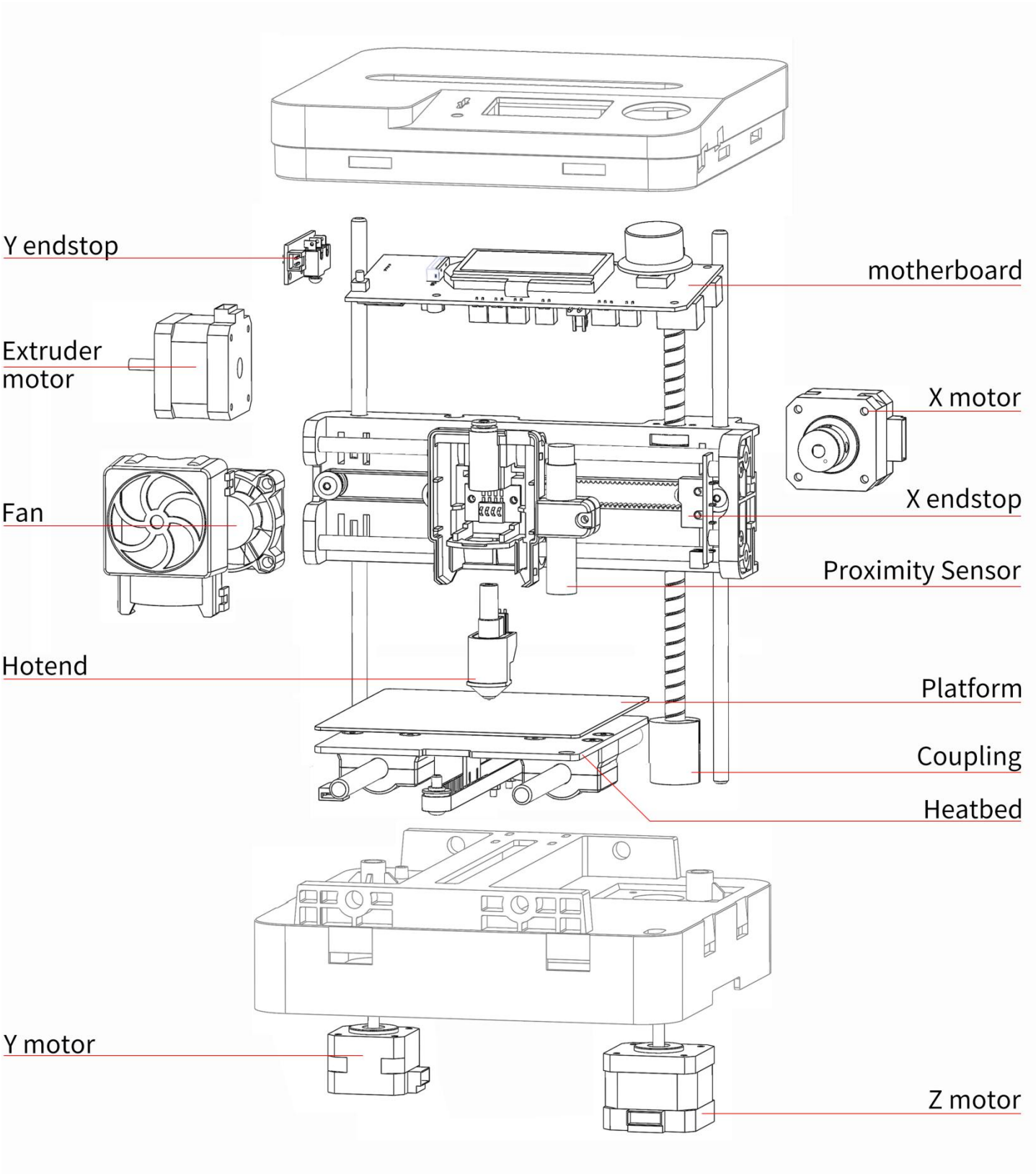
2.2. Product Overview



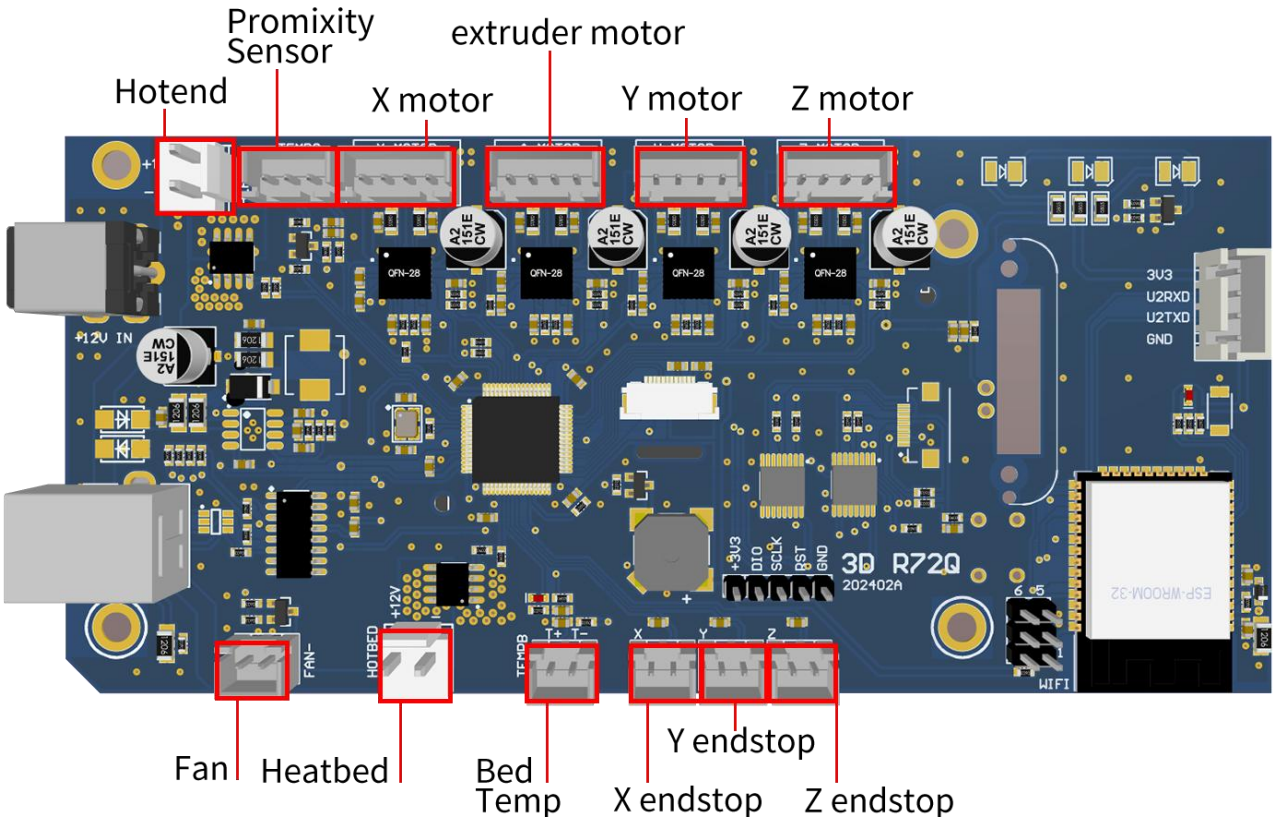
2.3. Hotend Structure



2.4. Internal Structure



2.5. Motherboard

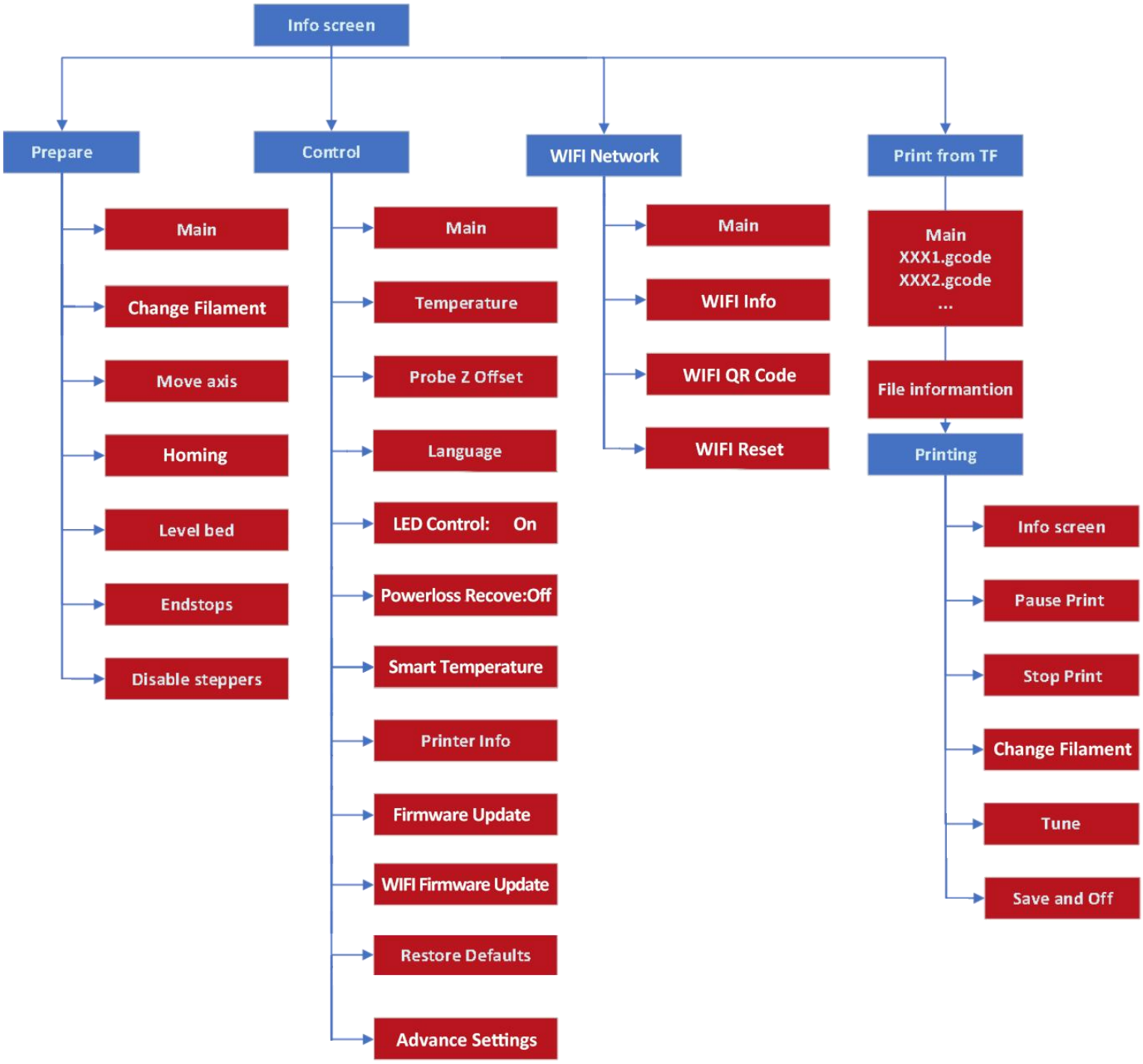


3. SYSTEM MENU

The top control panel features:

TF card slot (left): Stores print files

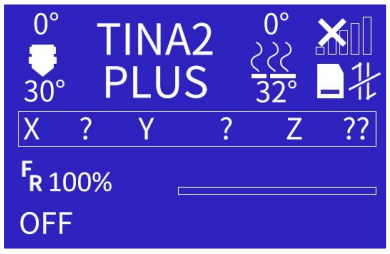
Knob button (right): Access system menu



3.1. Info Screen

Knob controls:

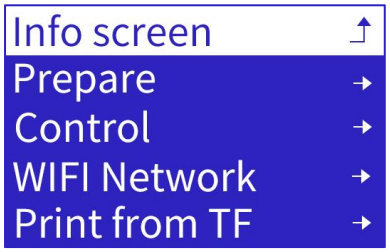
- Clockwise turn: Move down in menus, decrease value
- Counterclockwise turn: Move up in menus, increase value
- Press: Enter submenu, select option, confirm value



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.

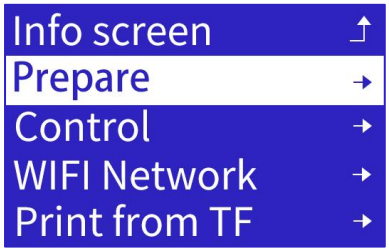
Press the knob to open the main menu.



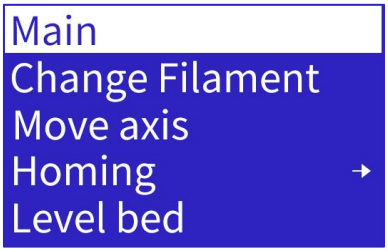
Press knob for Main Menu:

- 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

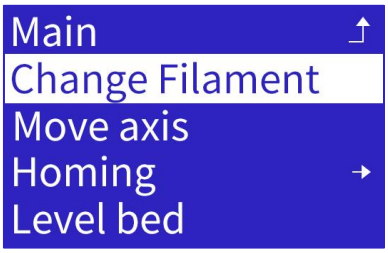


Press knob to open two-screen Prepare menu. Turn knob to scroll.



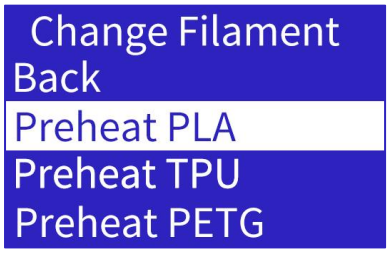
3.2.1. Change Filament

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



1. Change Filament:

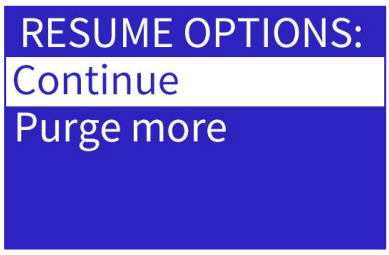
Select filament type to heat nozzle.



At 210°C, filament auto-unloads.

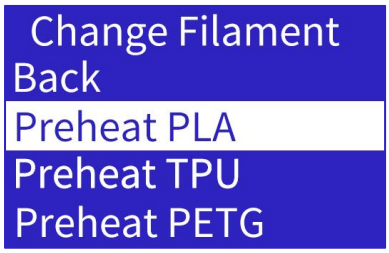
Beep indicates unload complete, insert new filament as prompted and press knob.

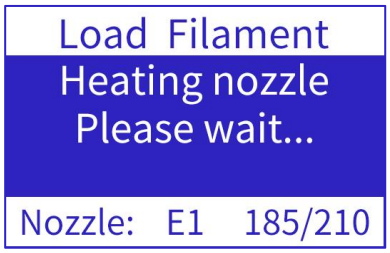
At 210°C again, filament auto-loads through nozzle.



2. Load Filament:

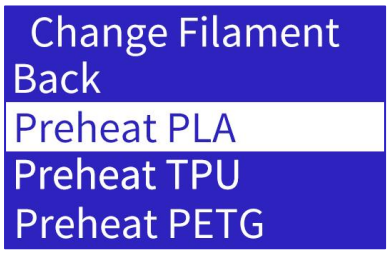
Auto-heats nozzle and loads filament. Press knob during heating/extruding to cancel.





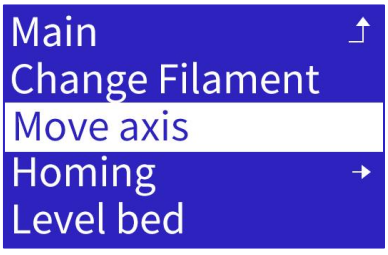
3. Unload Filament:

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:

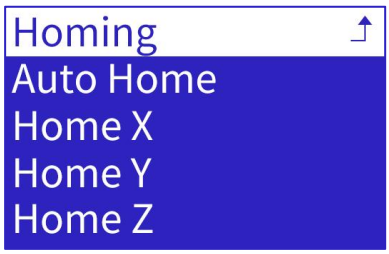
RANGE IS 0-100MM PER AXIS. PERFORM "AUTO HOME" FIRST FOR CORRECT NOZZLE COORDINATES. WITHOUT HOMING, CURRENT POSITION BECOMES ORIGIN, LIMITING JOG RANGE.

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

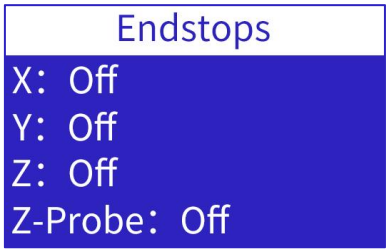
Auto-levels platform using proximity sensor. Done at each print start to ensure proper nozzle-platform distance.

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.

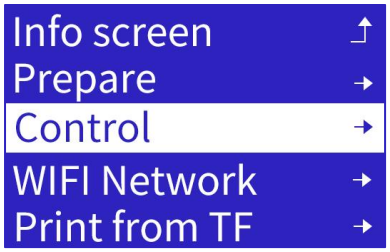
Helps diagnose faulty endstops or sensor - no state change indicates failure.



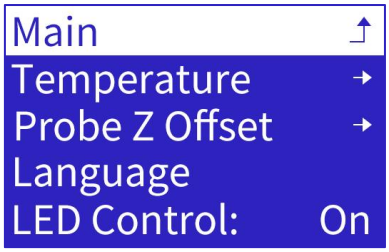
3.2.6.Disable Steppers

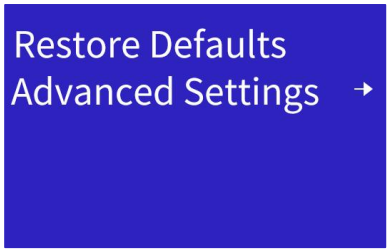
Releases stepper motor locking, allowing manual nozzle and platform movement.

3.3. Control Menu



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





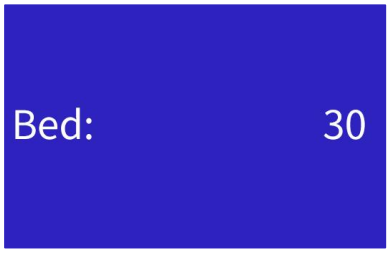
3.3.1. Temperature

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

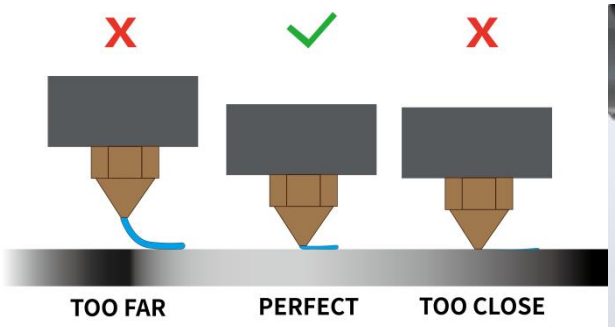
After bed leveling, sets nozzle-platform gap. Press knob to save.

Too small: Nozzle may scratch platform or clog.

Too large: Poor model adhesion, lifting.

Recalibrate after replacing nozzle or sensor.

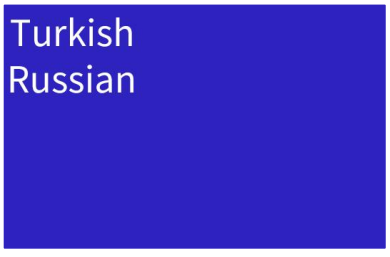
Turn knob to precisely adjust nozzle height for smooth, flat first layer.



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.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.

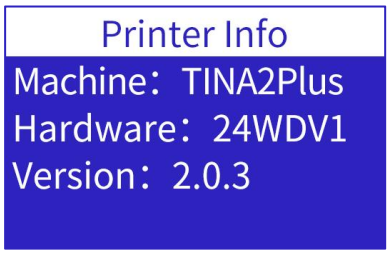
3.3.6.Smart Temperature

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.

3.3.7.Printer Info

Shows current firmware version.



3.3.8.Firmware Update

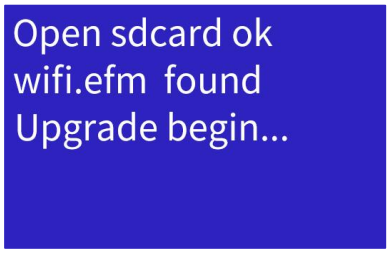
1. Copy ".wfm" firmware file to TF card root. Decompress first if needed. Rename to "flash.wfm".

2. Insert TF card and select Firmware Update.
3. Printer checks version and updates.

3.3.9.WIFI Firmware Update

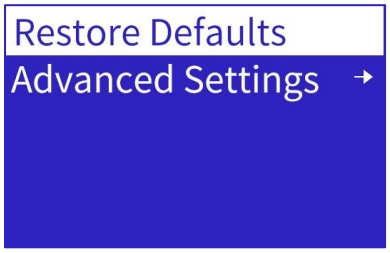
1. Copy ".efm" WiFi firmware file to TF card root. Decompress first if needed. Rename to "wifi.efm".

2. Insert TF card and select WiFi Firmware Update.
3. Printer checks version and updates.



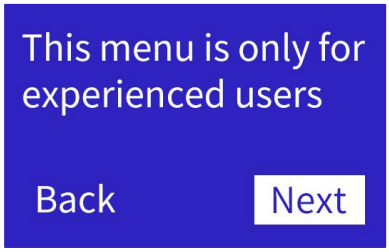
3.3.10.Restore Defaults

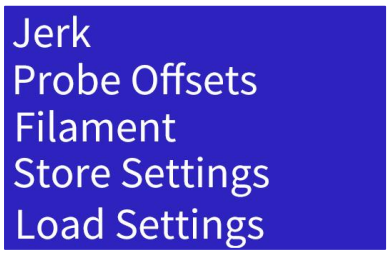
Resets all parameters to factory defaults.



3.3.11.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

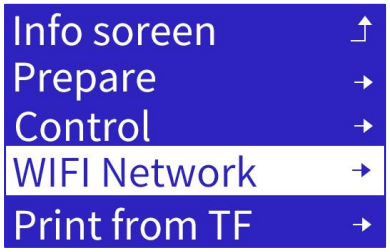
Filament: Set extruder compensation

Store Settings: Save changes

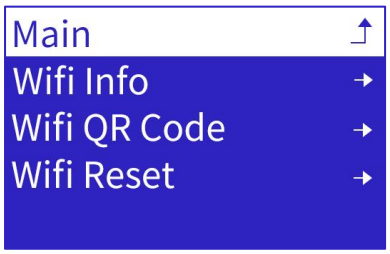
Load Settings: Apply saved changes

Initialize EEPROM: Reset to defaults

3.4. WIFI Network



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



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

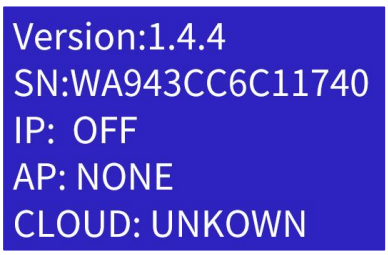
Local mode: In the same local area network, Wiibuilder or Cura on the computer can connect to the 3D printer, and perform remote control or send print files. To use local mode, the 3D printer should be connected to the local WIFI network.

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.



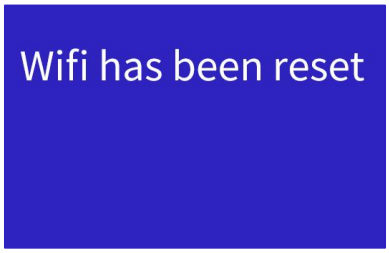
Only after configuring the network and registering in the APP will the 3D printer connect to the cloud server.

3.4.2.WIFI QR Code

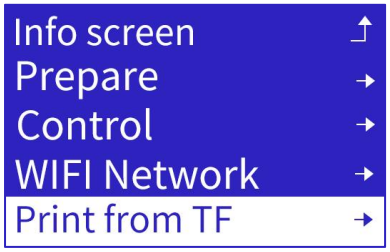
Click "WIFI QR Code" to show the QR code generated by the device serial. Used for App scanning networking and cloud access.

3.4.3.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

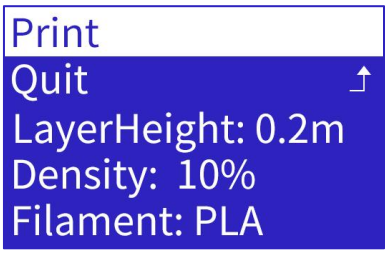


Shows ".gco" or ".gcode" print files on TF card.

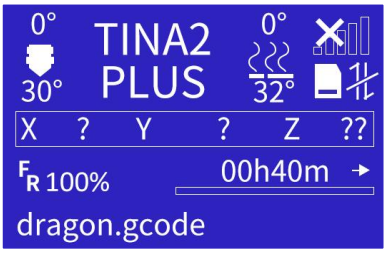
Select file and press knob for print info. "Print" starts printing.

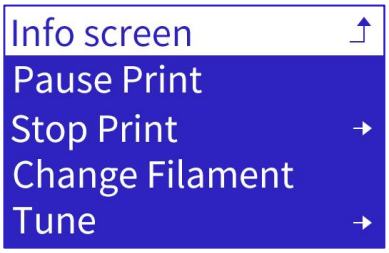
To prevent damage, checks machine model in file. Blocks print if model mismatched or missing. Use Wiibuilder or custom Cura version and select correct model when slicing.

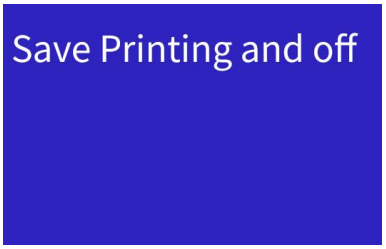
NOTE: FILE NAMES ⩽20 CHARACTERS TO AVOID DISPLAY ISSUES.



During print, press knob for print settings menu.

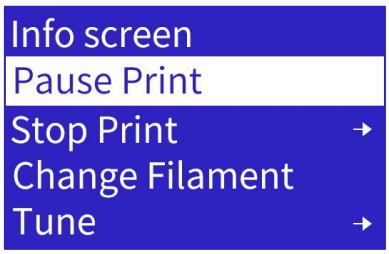


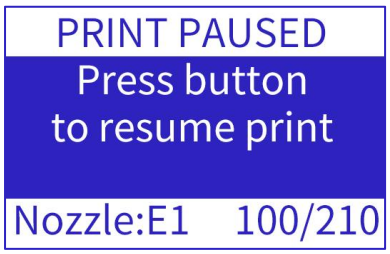


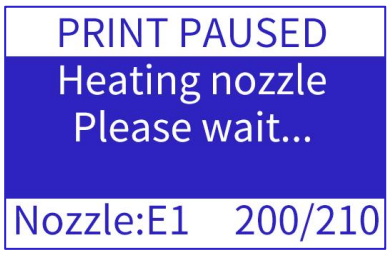


3.5.1. Pause Print

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

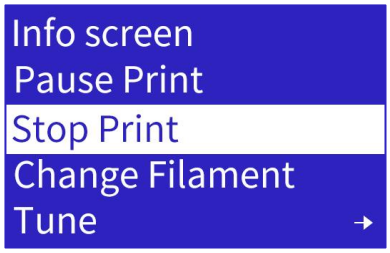


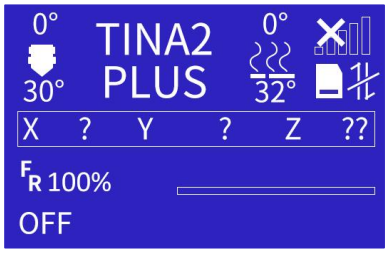




3.5.2.Stop Print

Cancels current print and returns to Info Screen.





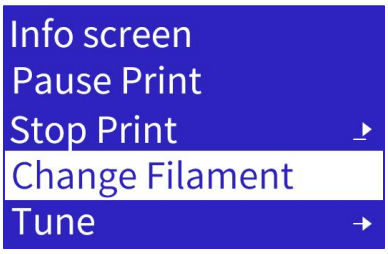
IMPORTANT:

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.



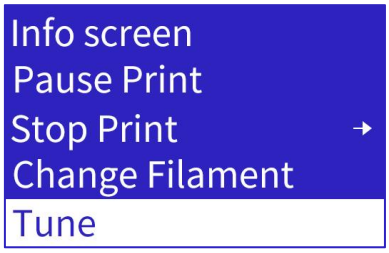
Beep, insert new filament, press knob.Auto-load at temp.

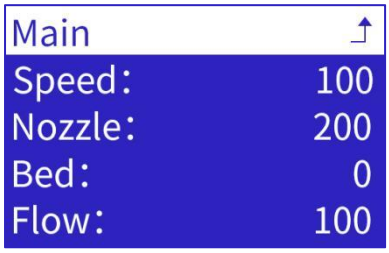
3.5.4.Tune

To change:

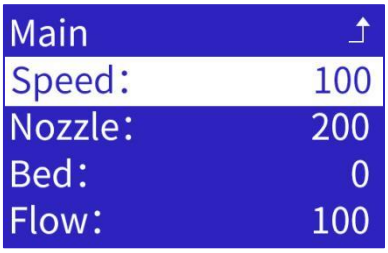
Select setting, turn knob to desired value.

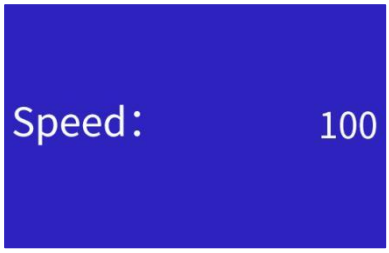
Press knob to confirm.





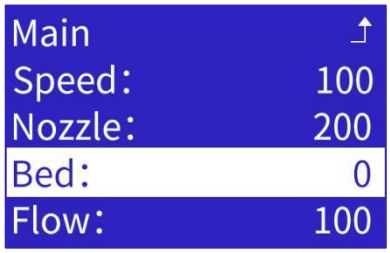
1.Speed

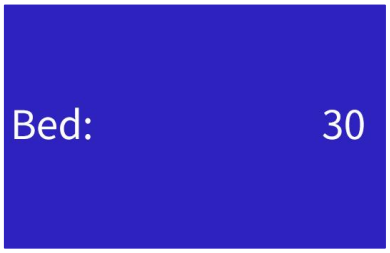




2.Nozzle temp

3.Nozzle temp





4.Probe Z Offset

IMPORTANT:

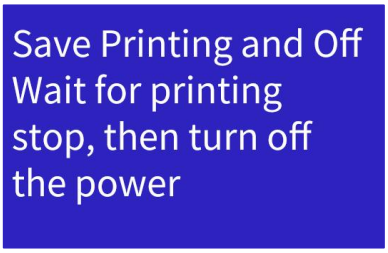
Z OFFSET MICRO-ADJUSTS NOZZLE HEIGHT IN REAL-TIME TO CHANGE GAP:

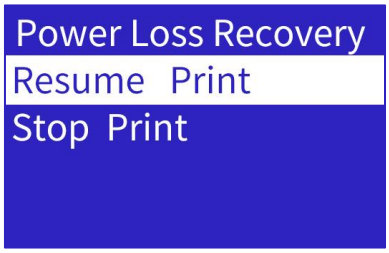
INCREASE BY 0.2MM IF GAP TOO SMALL (NO EXTRUSION)

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.





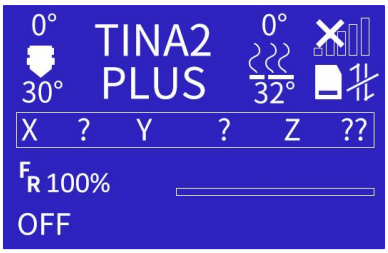
IMPORTANT:

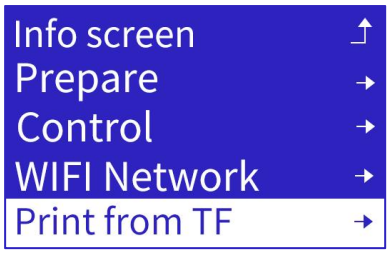
"PAUSE PRINT" AND "SAVE PRINT" TRIGGER HOMING. MECHANICAL ENDSTOP TOLERANCES CAUSE SLIGHT DEVIATION ON EACH HOME, POTENTIAL SEAM/SHIFT AT RESUME POINT.

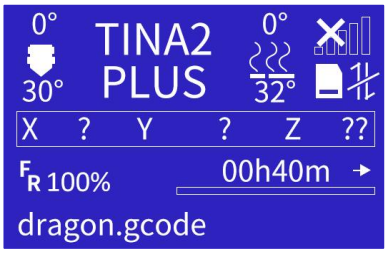
PRINT MODELS IN ONE SESSION WHEN POSSIBLE TO AVOID THIS.

4. COMMON MENU OPERATIONS

4.1. Print from TF Card





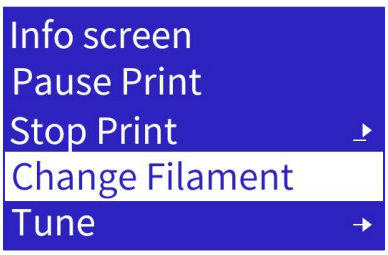


1. Open Main Menu, select "Print From TF"
2. Turn knob to select file, press knob to start print.

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.



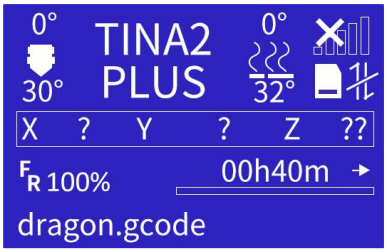
Beep, insert new filament, press knob. Auto-load at temp.

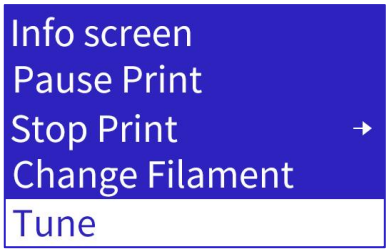
4.3. Adjust Temperature During Print

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

Turn knob to increase/decrease temp.

Press knob to return.

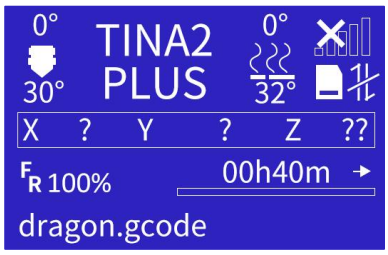


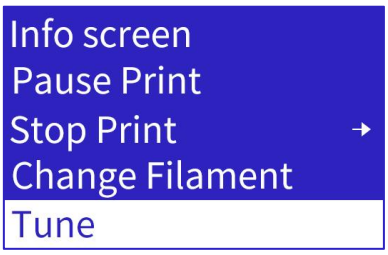


4.4. Adjust Z Offset During Print

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)

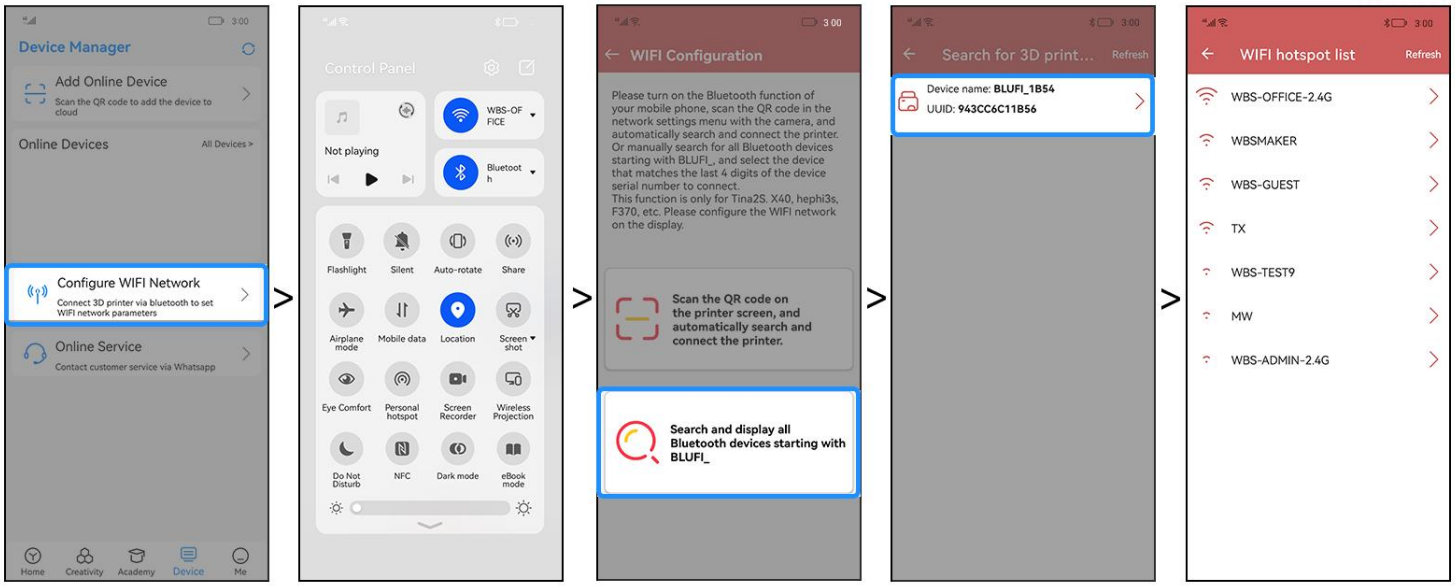
GRADUALLY DECREASE IF GAP TOO LARGE (POOR ADHESION)

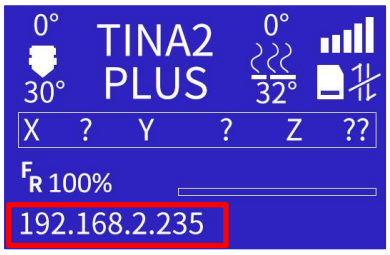
5. PRINT BY APP

1. Download and install the APP : PoloPrint Cloud

2. Register or log in to the APP.

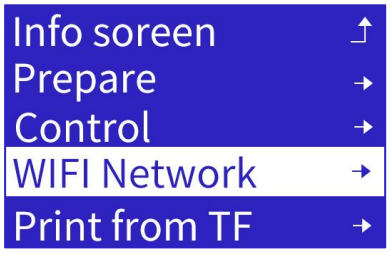
3. Configure WiFi for 3D printer.



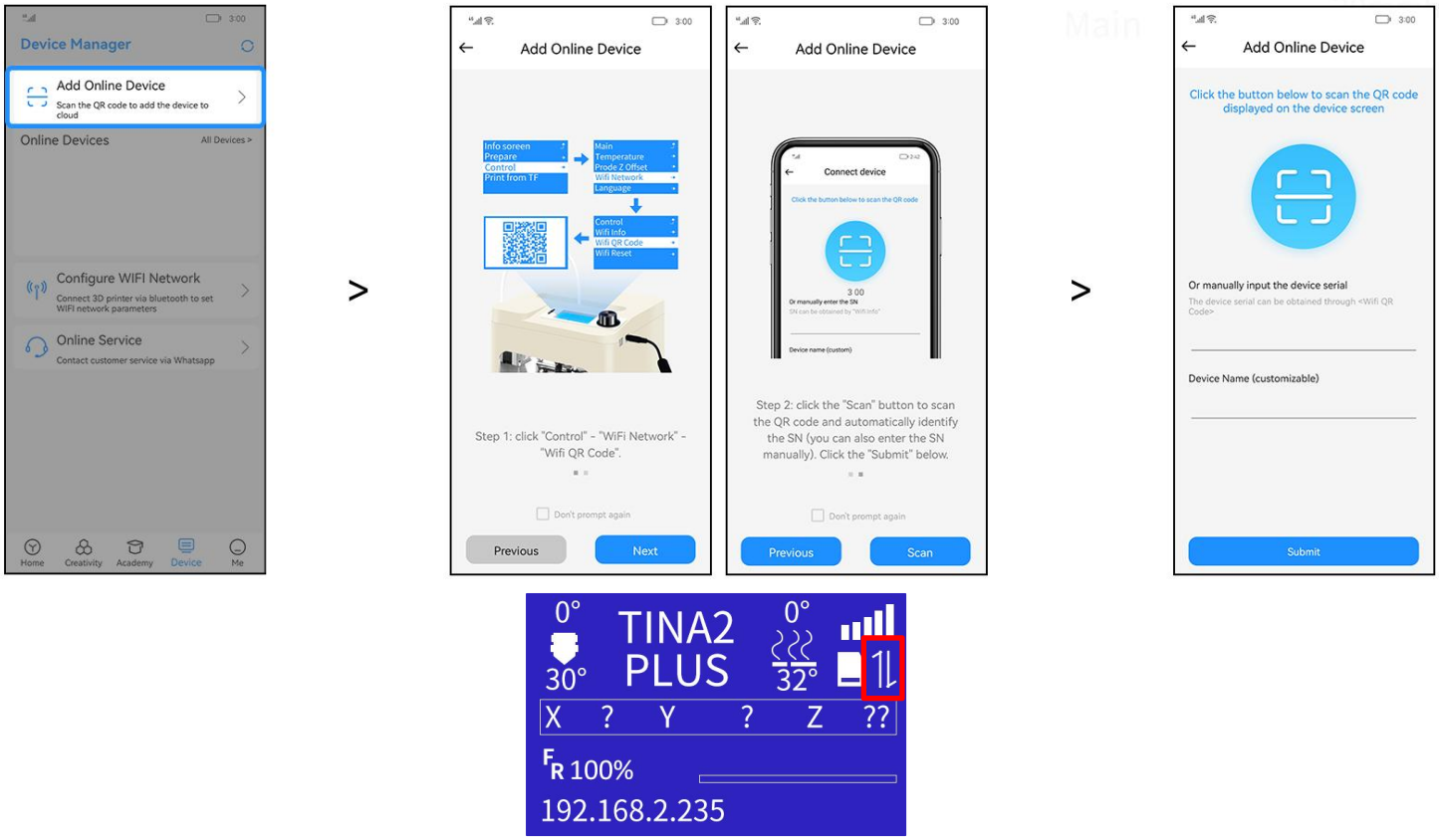


4. Get the “ Wifi QR Code ”.

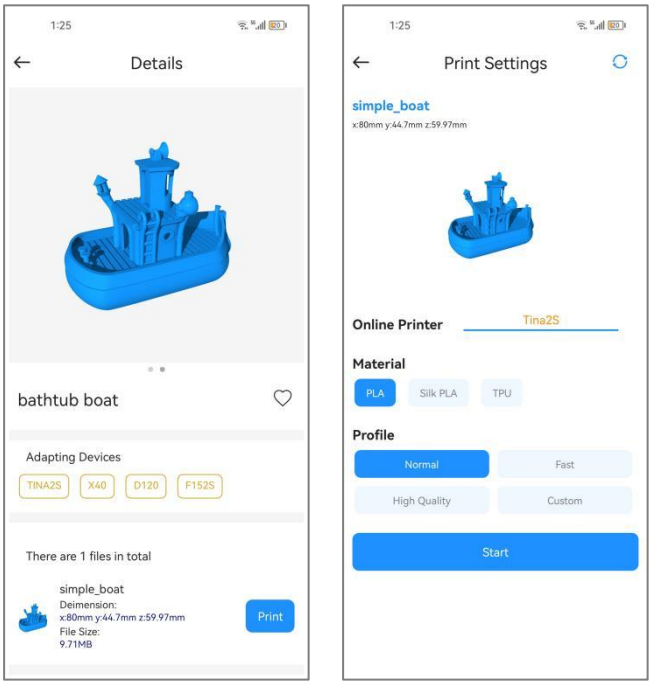
NOTE: THE 3D PRINTER HAS BEEN CONNECTED TO THE NETWORK, AND THE IP ADDRESS IS DISPLAYED.



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



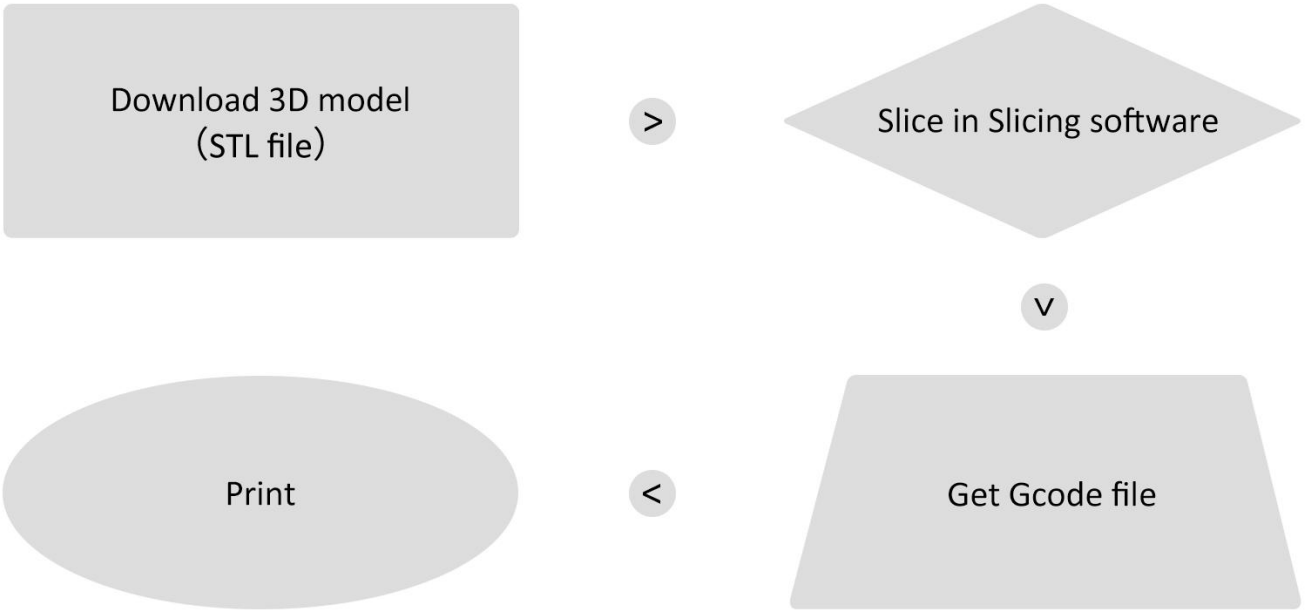
6. Select a model and click "Print".



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:

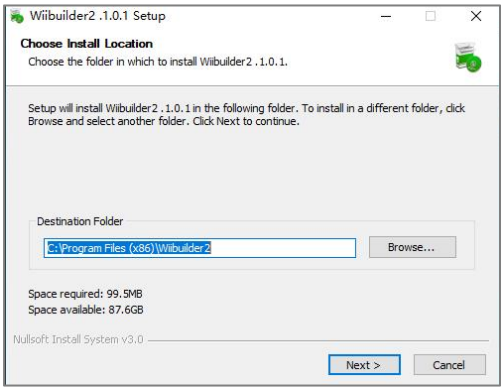
- CPU: Pentium 1GHz
- RAM: 1GB minimum
- Disk: 1GB+ free space
- Display: 640x480 minimum
- Graphics: 3D acceleration, OpenGL 2.0+
- Color: 256 colors minimum
- OS: Windows 7+ / MacOS 10.13.6+
- Other: .Net Framework 4.5.2+

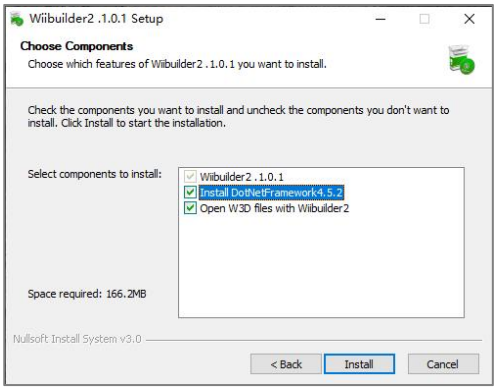
6.1. Installation

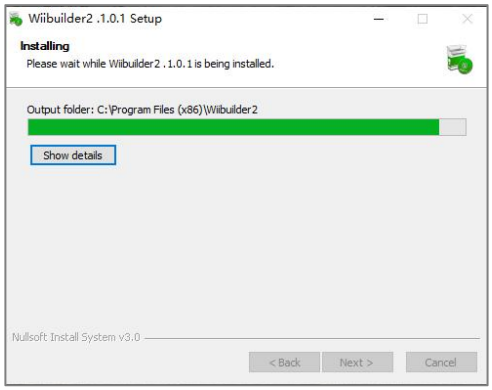
6.1.1. Installation on Windows

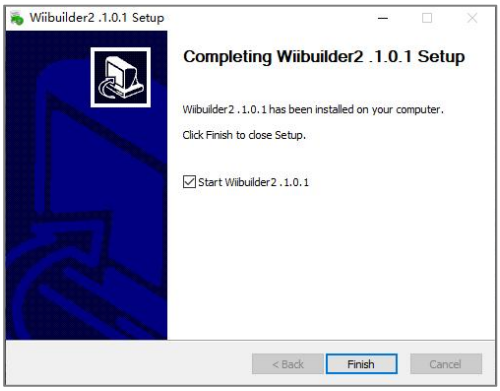
Steps to install Wiibuilder:

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









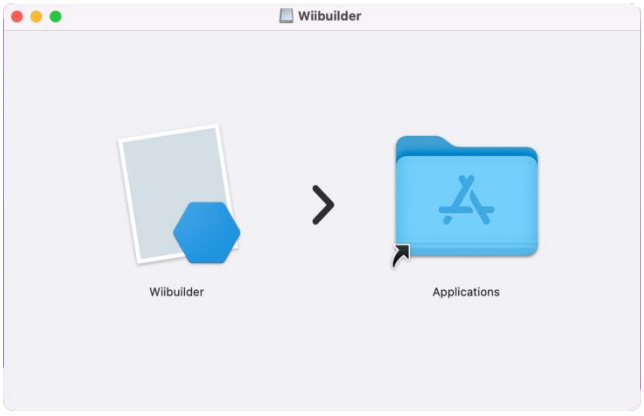
6.1.2. Installation on MacOS

OS: MacOS 10.13.6+

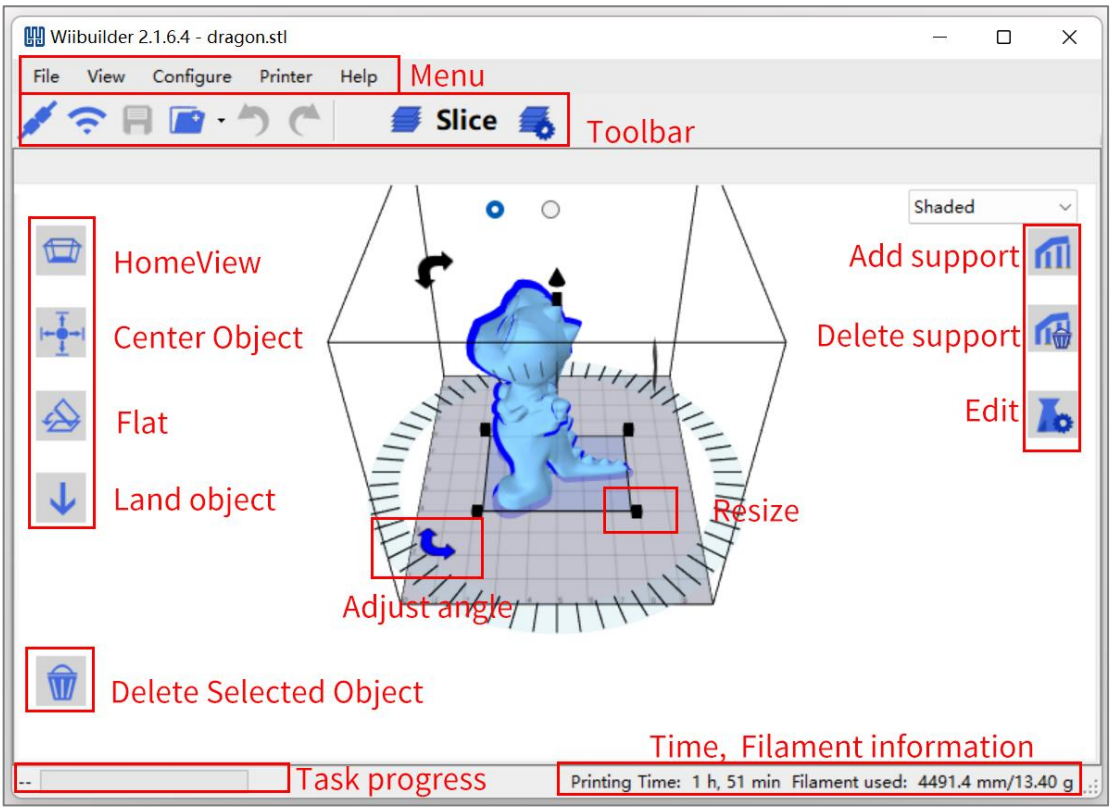
Decompress installer. Double-click package, drag Wiibuilder to Applications.

Wiibuilder-2.3.0.0

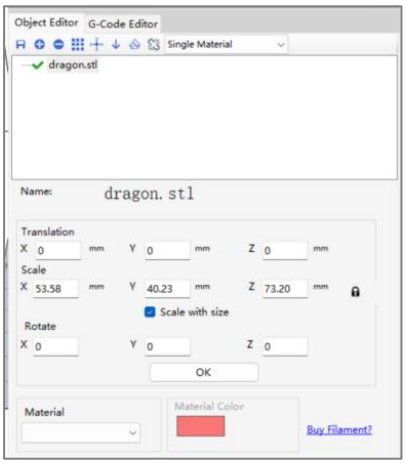
Wiibuilder-2.3.0.0



6.2. Interface Introduction

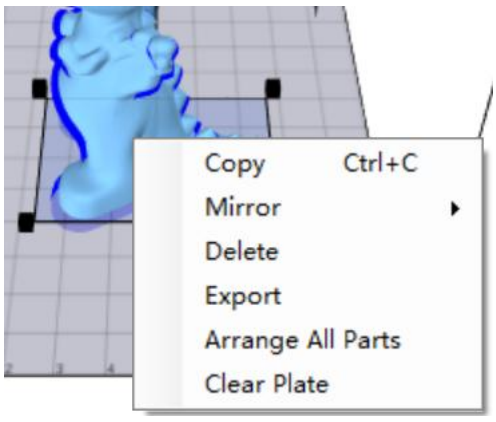


- Menu Bar: Access Wiibuilder menu
- Toolbar: USB/WiFi connect, save Gcode, load model, undo/redo, slice, settings
- Home View: Restore default view angle
- Center Object: Auto-adjust model XY to center on platform
- Flat: Auto-flatten tilted model on platform
- Land Object: Auto-adjust model Z to align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



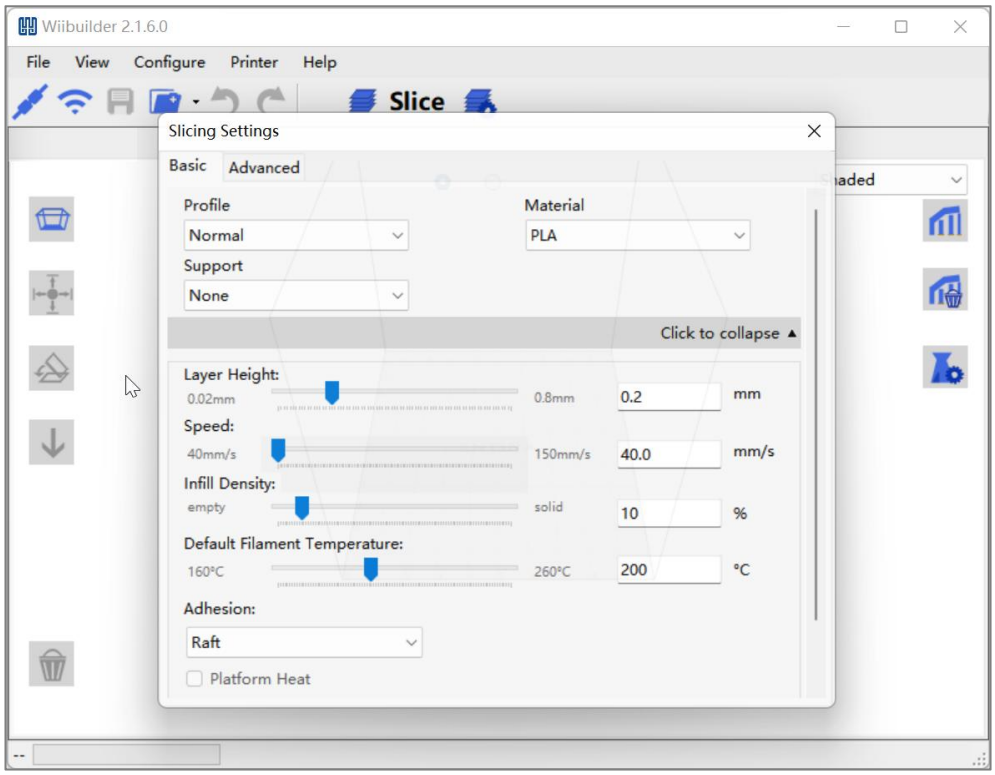
Right-click model for context menu:

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



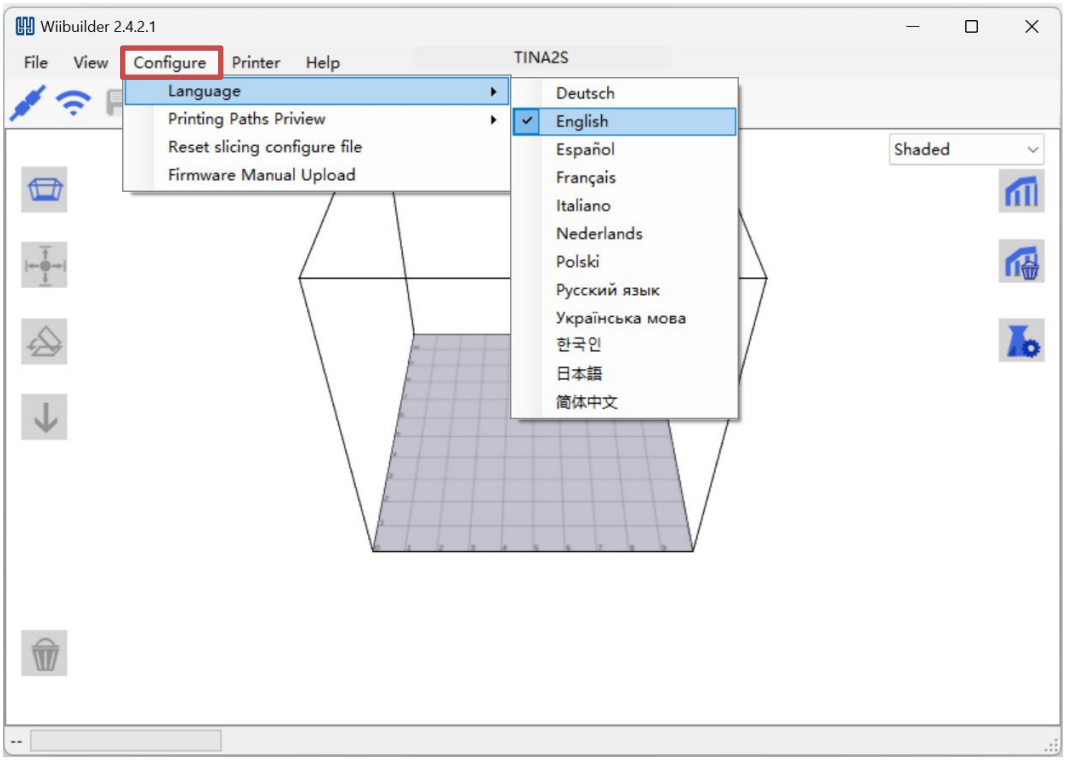
- Clear Plate: Remove all models

Slicing Settings:



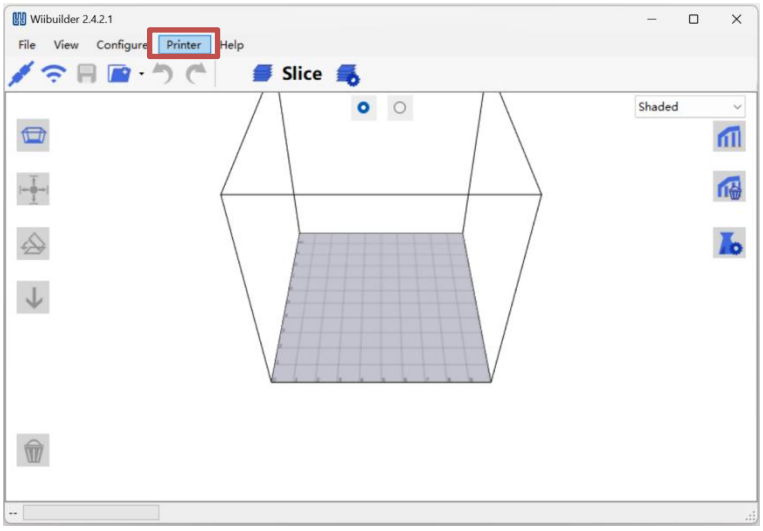
6.3. Select Language

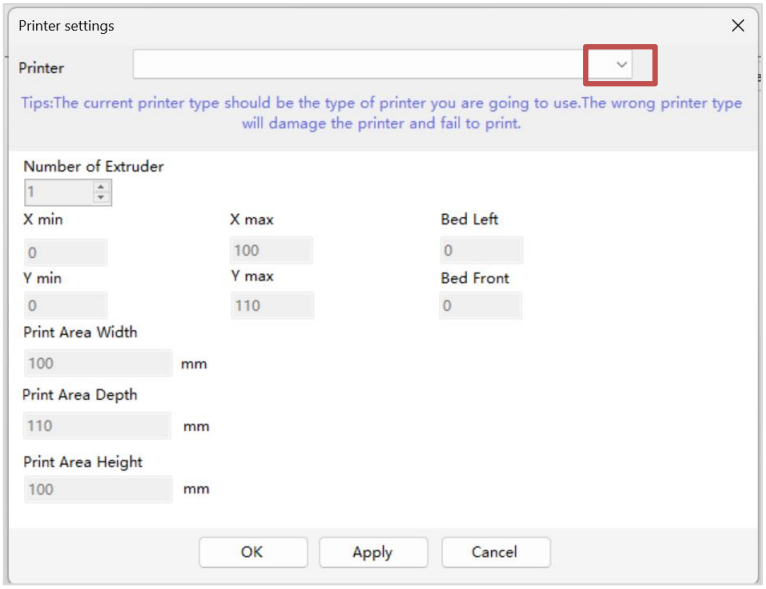
Open Wiibuilder, click Configure > Language to select.



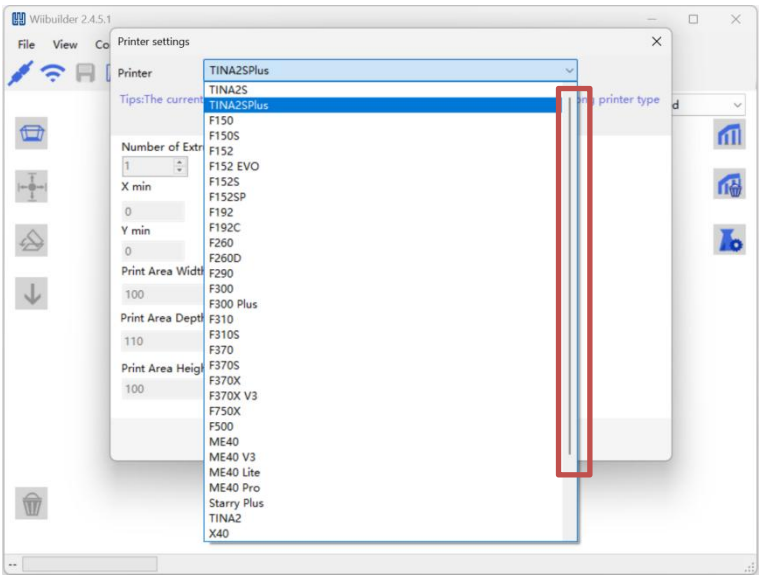
6.4. Select Printer

Click Printer to select model.



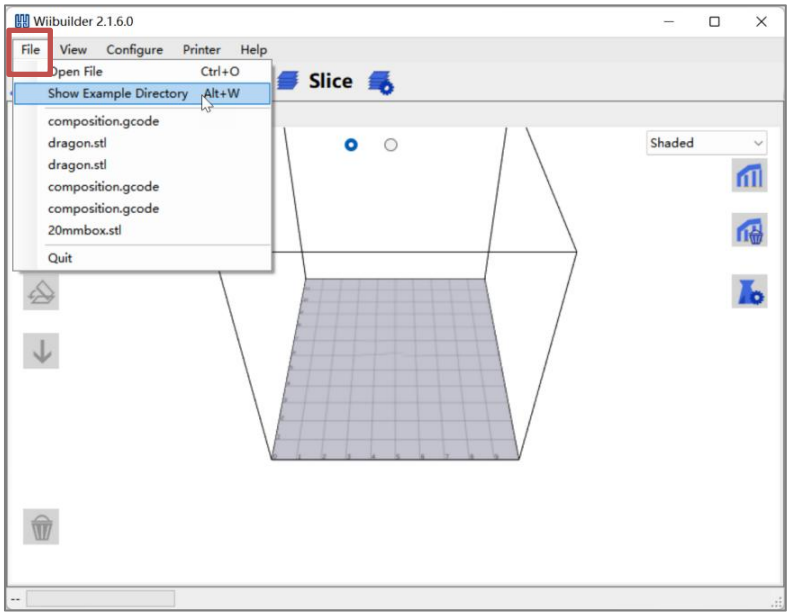


Scroll to find "TINA2Plus".Note: Not "TINA2"or”TINA2S”, will fail if wrong.

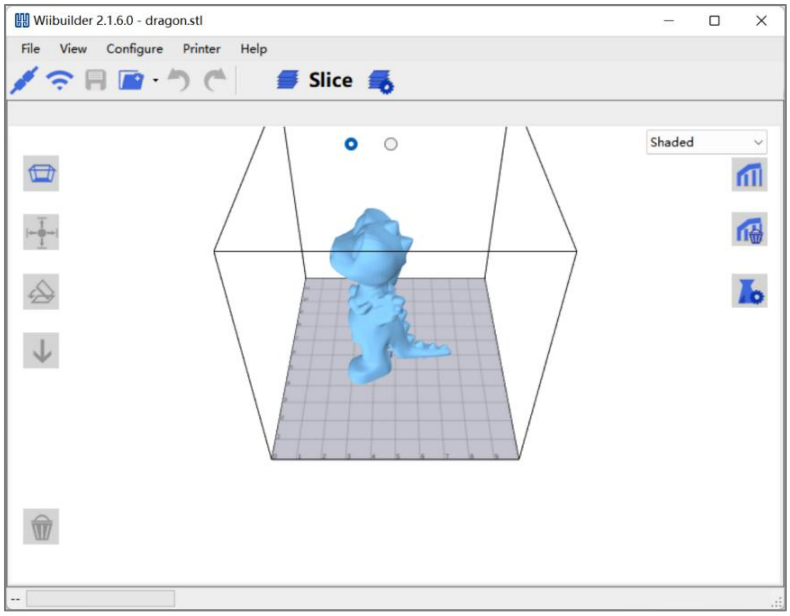


6.5. Add Model

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

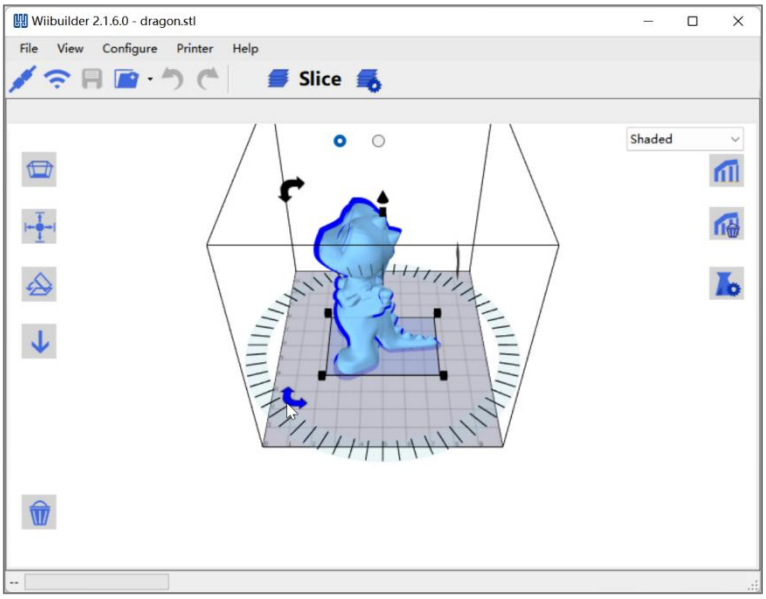


Five sample models in "Show Example Directory" for test printing.

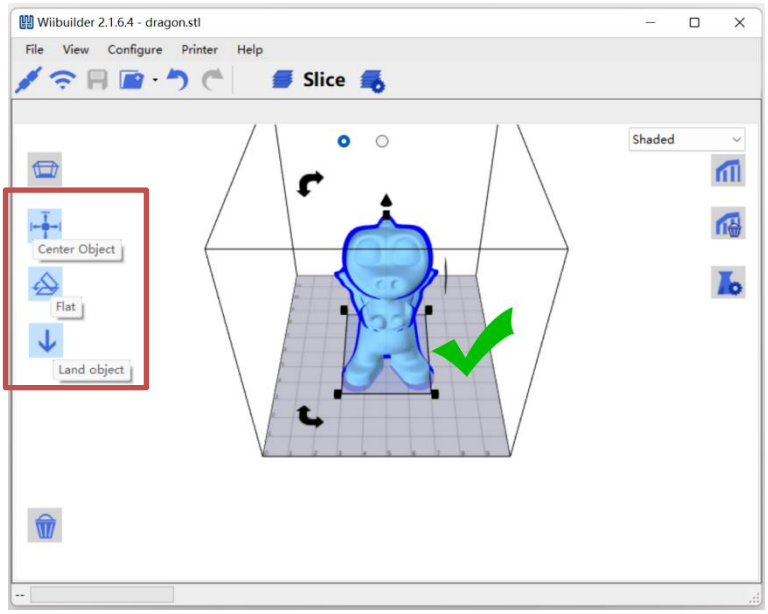


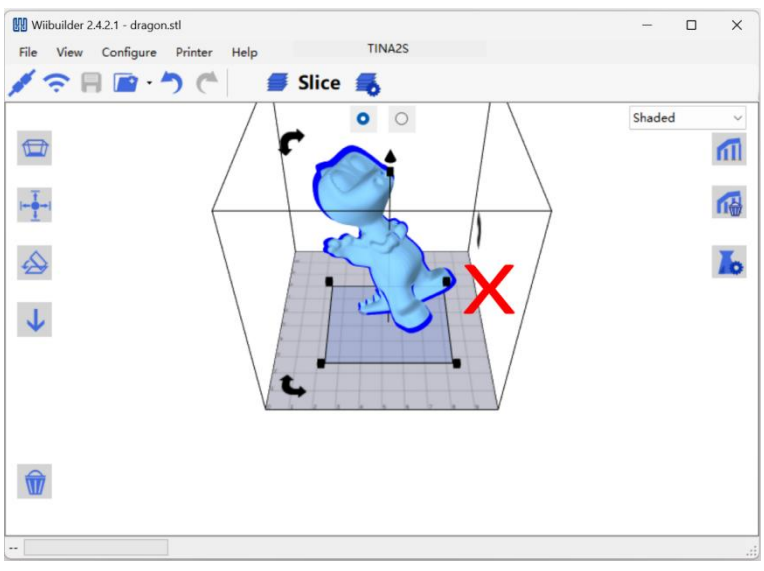
6.6. Adjust Model

Click model to adjust size, angle, position.



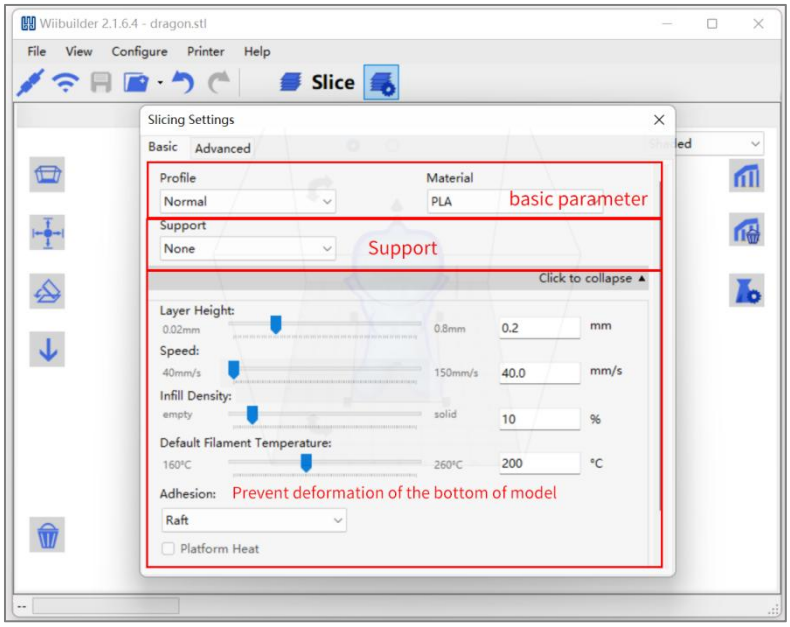
Model should be centered and properly placed on platform. Click Center Object, Flat, Land Object. If floating, print fails.



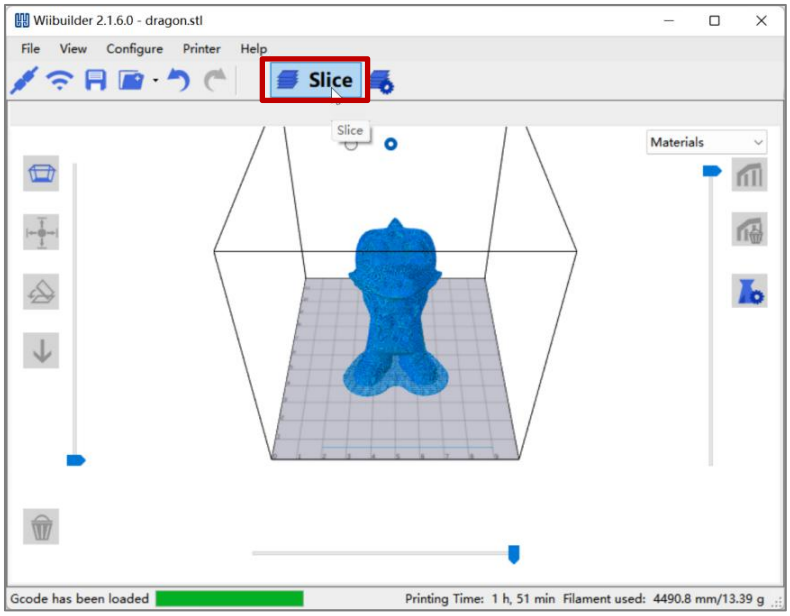


6.7. Basic Parameter Settings

Beginners use defaults without adjustment.



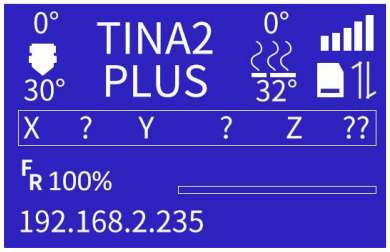
After setting parameters, click Slice to convert file.



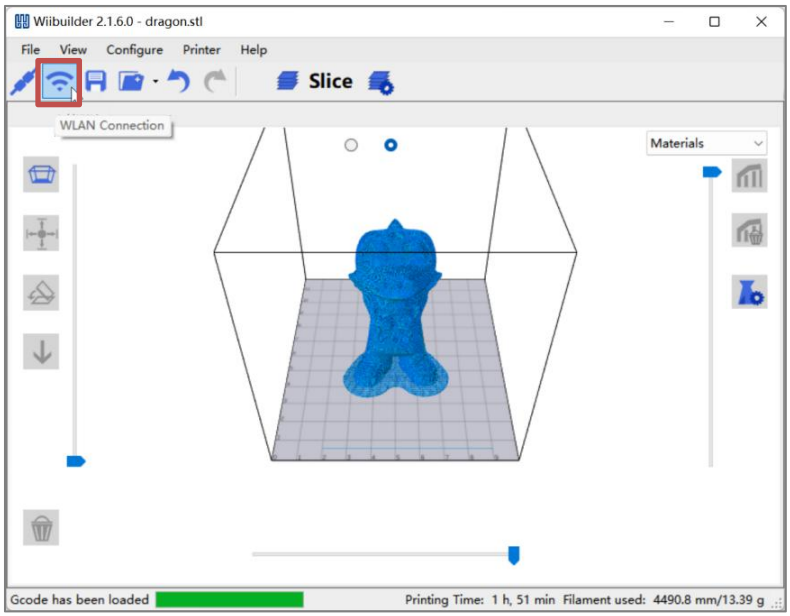
6.8. Send Files via Network

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

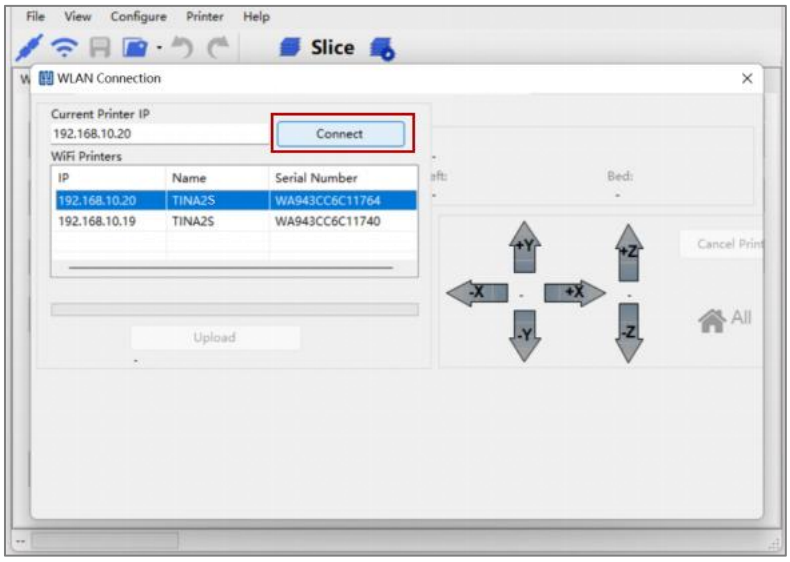
SELECT THE COMPUTER TO USE THE SAME NETWORK AS THE DEVICE. USING DIFFERENT NETWORKS CAN CAUSE TRANSFER FAILURES.



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

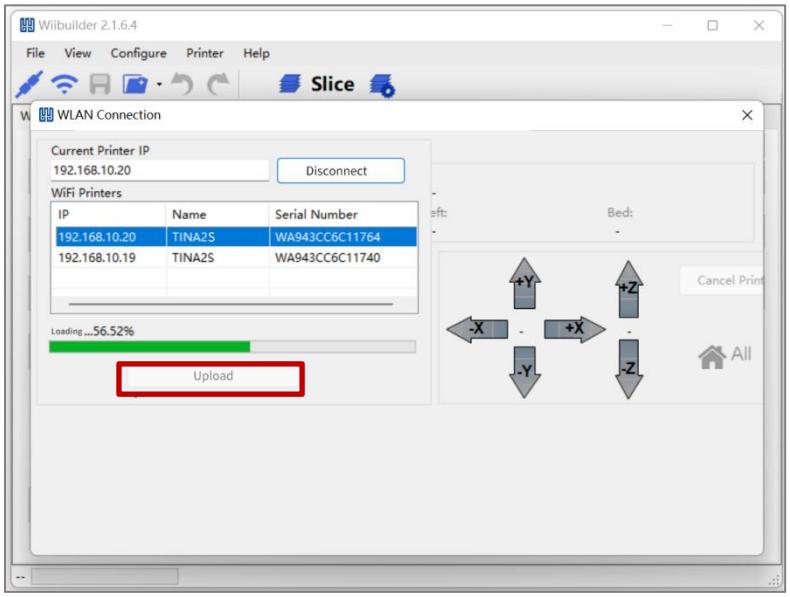


Click "WLAN Connection", the software will automatically search for the 3D printers in the local network and show them in the below list. If no 3D printer be found, you can also try to manually enter the IP address of the 3D printer to connect.



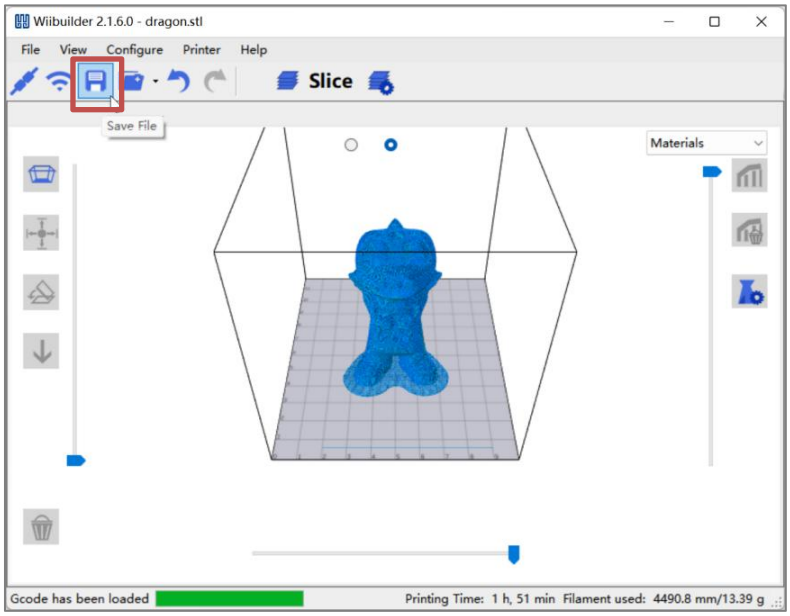
NOTE: IF YOU STILL CANNOT CONNECT TO THE 3D PRINTER, PLEASE CHECK WHETHER THE FIREWALL OF THE COMPUTER PROHIBITS THE SOFTWARE FROM NETWORKING, AND WHETHER THE COMPUTER AND THE 3D PRINTER ARE IN THE SAME LOCAL AREA NETWORK.

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

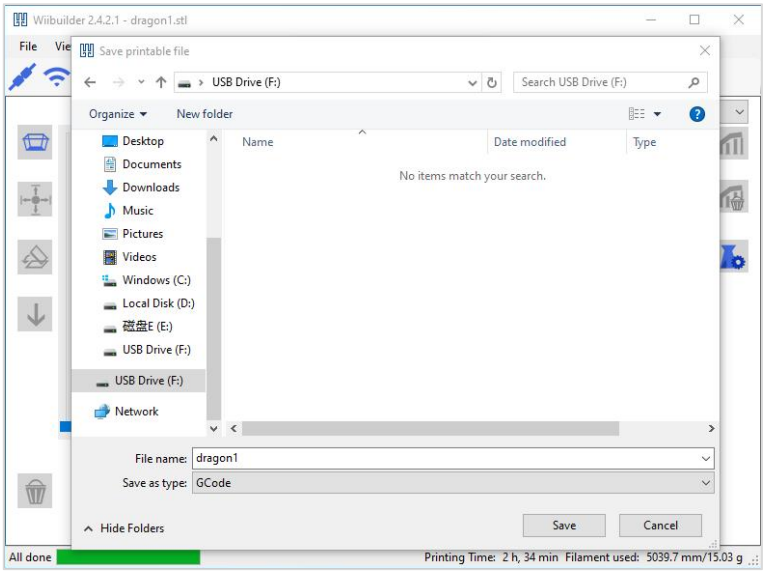


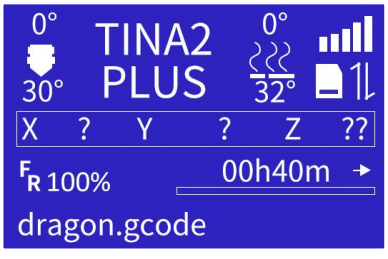
6.9. Send Files via TF Card

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



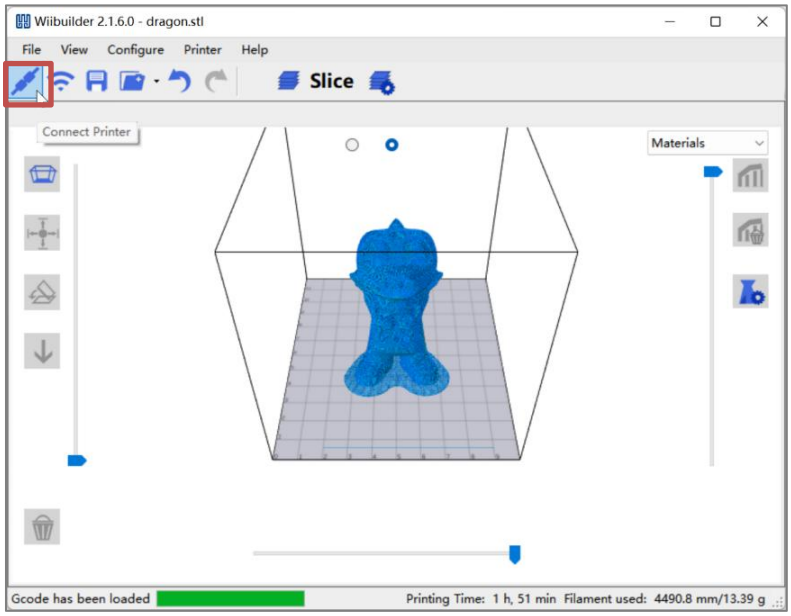
Note: Save directly to TF root, not in folder.



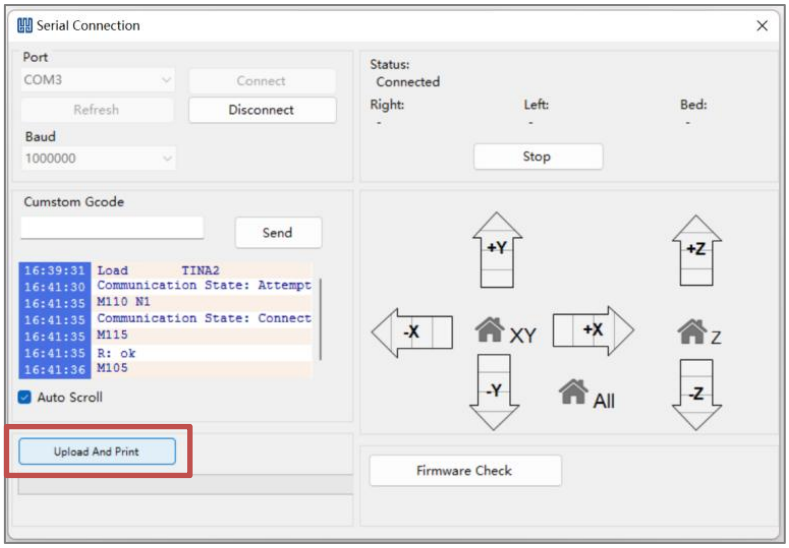


6.10.Send Files via USB

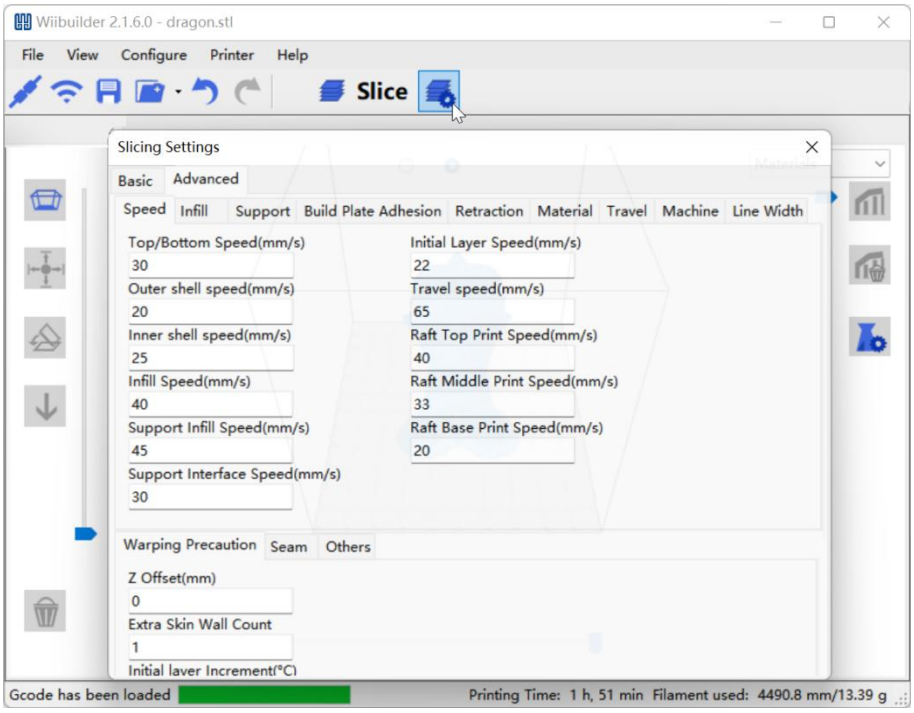
Connect printer to PC with USB. Install CH340 driver first. Click Connect Printer. PC can't sleep.



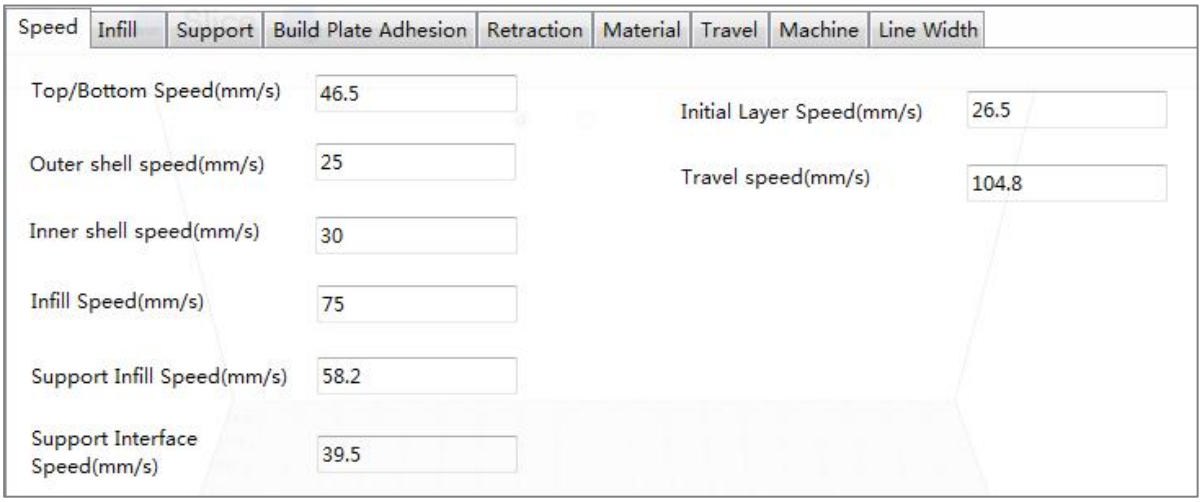
Click Connect. Click Upload And Print to start. Baud rate 115200.



6.11.Advanced Parameter Settings



6.11.1.Speed Tab

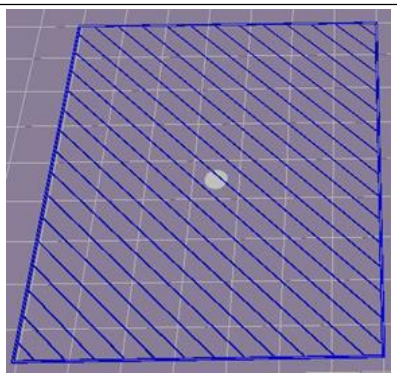
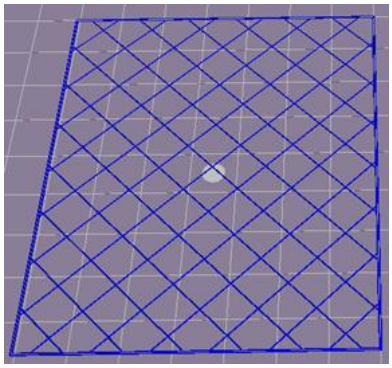
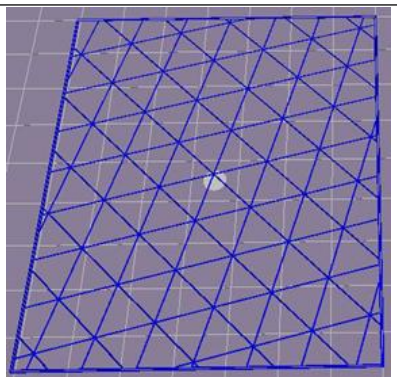
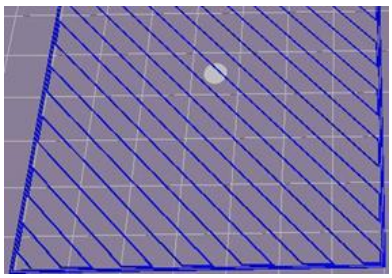


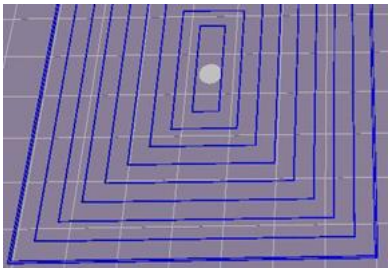
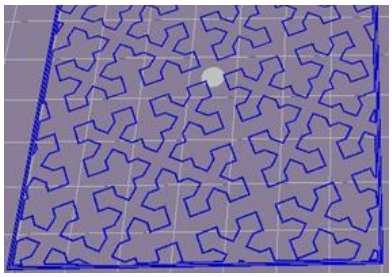
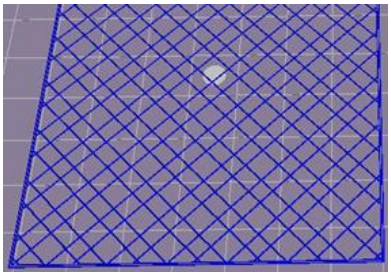
Options:

- Top/Bottom Speed (mm/s): Top and bottom surface print speed
- Outer/Inner Shell Speed (mm/s): External and internal shell surface print speed
- Infill Speed (mm/s): Infill print speed
- Support Infill/Interface Speed (mm/s): Support infill and top/bottom surface print speed
- Initial Layer Speed (mm/s): First layer print speed
- Travel Speed (mm/s): Non-print movement speed

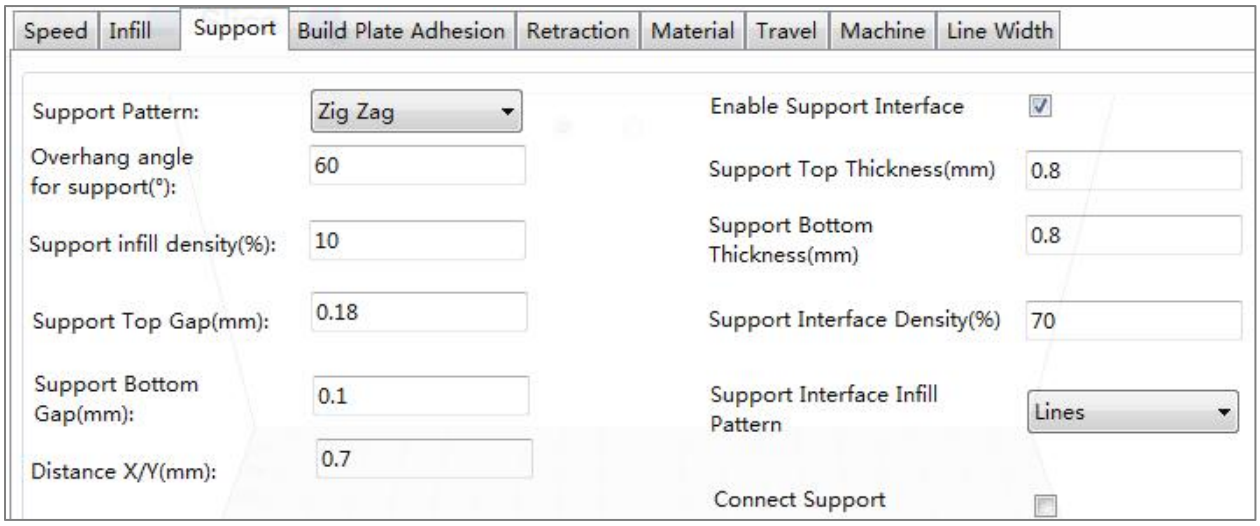
6.11.2.Infill Tab

- Infill Pattern: Select from Lines, Grid, Triangles, Zig Zag, Concentric, Cross, Octet. Patterns shown below.
- Infill Before Wall: Print infill then walls.
- Outer Before Inner Walls: Print outer then inner walls.
- Infill Patterns:

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

	
Concentric	**Cross**
	
Octet	

6.11.3.Support Tab

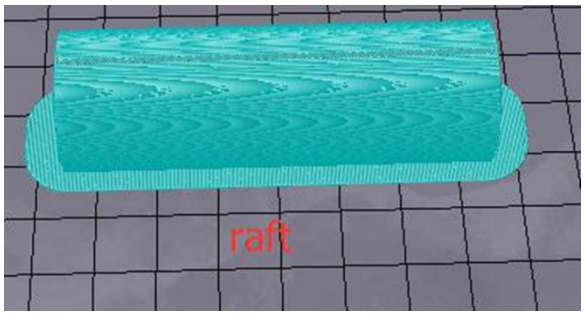


- Support Pattern: Select from Lines, Grid, Triangles, Zig Zag, Concentric. Same designs as infill patterns.
    - **Lines easier to remove, for models needing more support**
    - **Grid for small models needing less support**
    - **Zig Zag stronger than Lines, better than Grid, for difficult to remove supports**
- Overhang Angle: Angle between support and model surface. Larger = easier removal, smaller = better support. Default 60°.
- Support Infill Density (%): Higher density = stronger support.

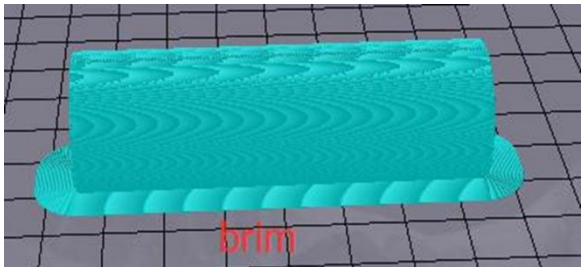
- Support Top/Bottom Gap (mm): Distance from support top/bottom to model. Smaller = more effective but harder removal leaving.
- residue, larger = less effective but easier removal for smoother surface.
- Distance X/Y (mm): Horizontal distance from support to model, same effects as Top/Bottom Gap.
- Enable Support: Use supports.
- Support Top/Bottom: Top and bottom support layer thickness.
- Support Interface: Infill percentage inside supports.
- Support Interface Infill Pattern: Select from Lines, Grid, Triangles, Zig Zag, Concentric. Same designs as infill patterns.
- Connect Support: Join separate supports into one.

6.11.4.Build Plate Adhesion Tab

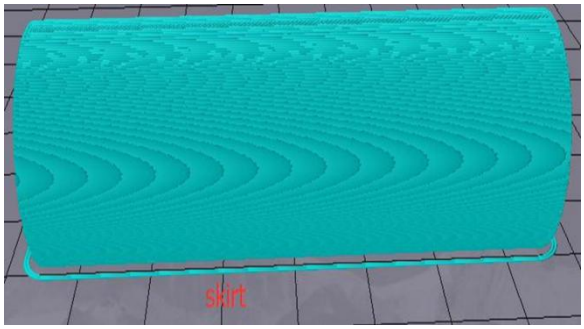
- Raft Air Gap (mm): Distance between raft and model, determines removal difficulty.



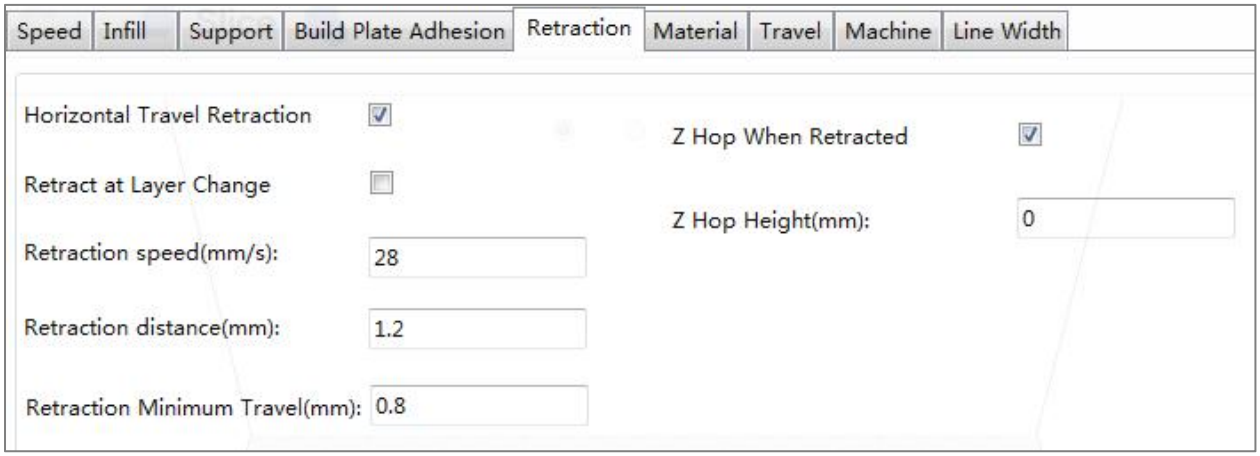
- Raft Extra Margin (mm): Distance from raft edge to model surface.
- Raft Base Thickness (mm): Raft thickness.
- Initial Layer Z Overlap: Overlap between model's first and second layers.
- Brim Line Amount: Number of rings added to model edge contacting platform.



- Skirt Line Count: Number of anti-overflow lines at model end contacting platform.

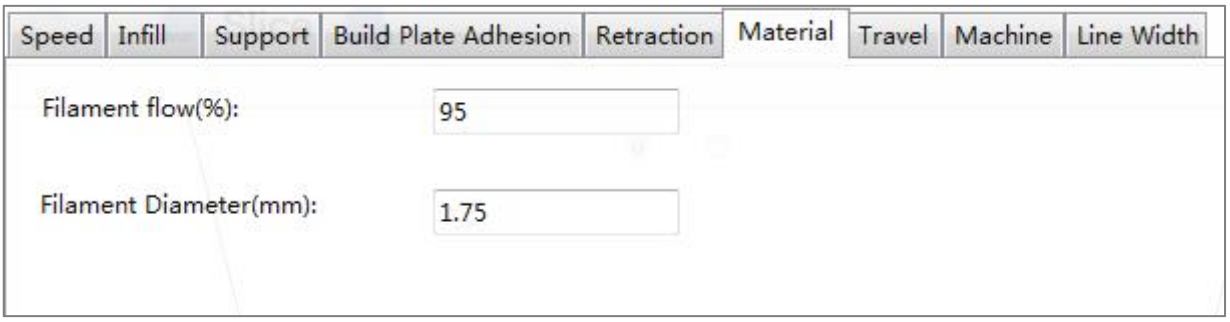


6.11.5.Retraction Tab



- Horizontal Travel Retraction: Enable filament retract during non-print horizontal moves
- Retract at Layer Change: Retract filament between layers
- Retraction Speed (mm/s): Filament retract speed
- Retraction Distance (mm): Filament retract distance inside nozzle
- Retraction Minimum Travel (mm): Minimum pre-print nozzle move distance before retracting
- Z Hop When Retracted: Enable nozzle lift after retract
- Z Hop Height (mm): Nozzle lift distance after retract

6.11.6.Material Tab



- Filament Flow (%): Molten filament flow rate based on material. Generally 90 for PLA/PLA Pro, 100 for ABS.
- Filament Diameter (mm): Diameter of filament used. Printer only supports 1.75mm.

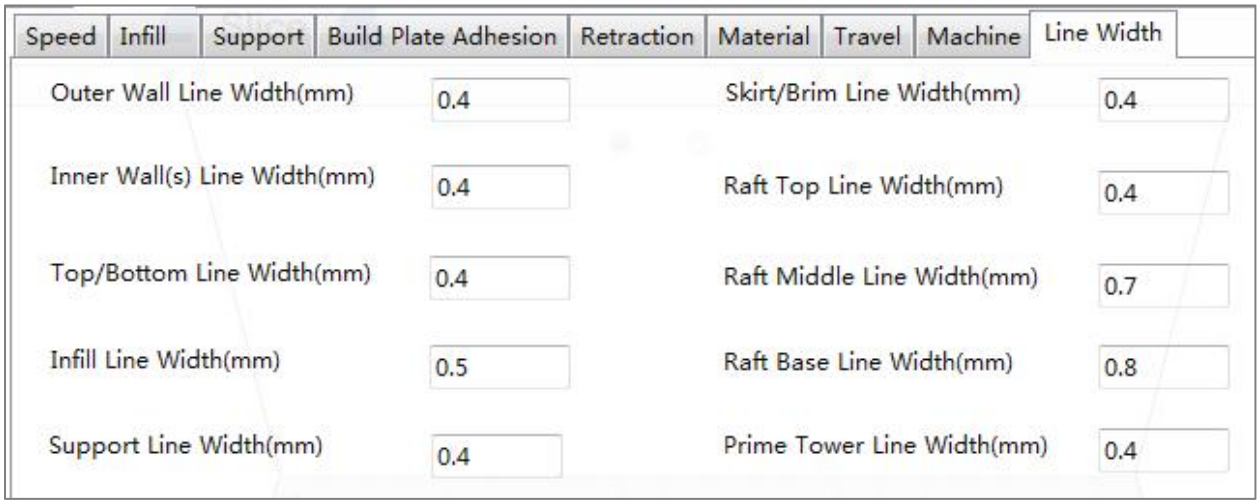
6.11.7.Travel Tab

- Combing Mode: Nozzle movement when not printing.
  - Off: Shortest move from previous extrusion to new start
  - All: Move along already extruded paths
  - No Skin: Avoid outer layers moving to new start, improves quality

6.11.8.Machine Tab

- Right Nozzle Diameter (mm): Diameter of right extruder nozzle. Printer has single right extruder with 0.4mm nozzle.

6.11.9.Line Width Tab



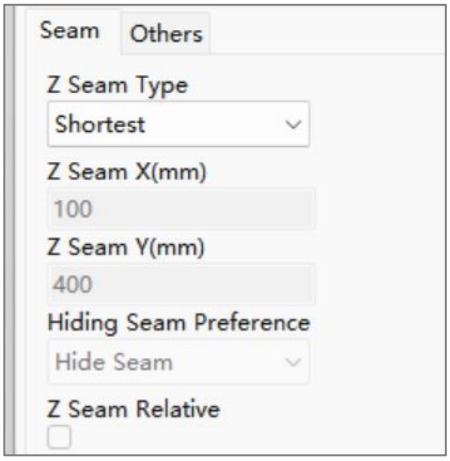
- Outer Wall Line Width (mm): Outermost wall line width. Lower for finer detail.
- Inner Wall(s) Line Width (mm): Single wall line width for all but outermost
- Top/Bottom Line Width (mm): Top and bottom line width
- Infill Line Width (mm): Single infill line width
- Support Line Width (mm): Single support line width
- Skirt/Brim Line Width (mm): Single skirt or brim line width
- Raft Top Line Width (mm): Line width in raft top surface. Thin for smooth top.
- Raft Middle Line Width (mm): Line width in middle raft layers. Thicker second layer for build plate adhesion.
- Raft Base Line Width (mm): Raft base layer line width. Thick for build plate adhesion.
- Prime Tower Line Width (mm): Prime tower extrusion width

6.11.10.Seam Tab

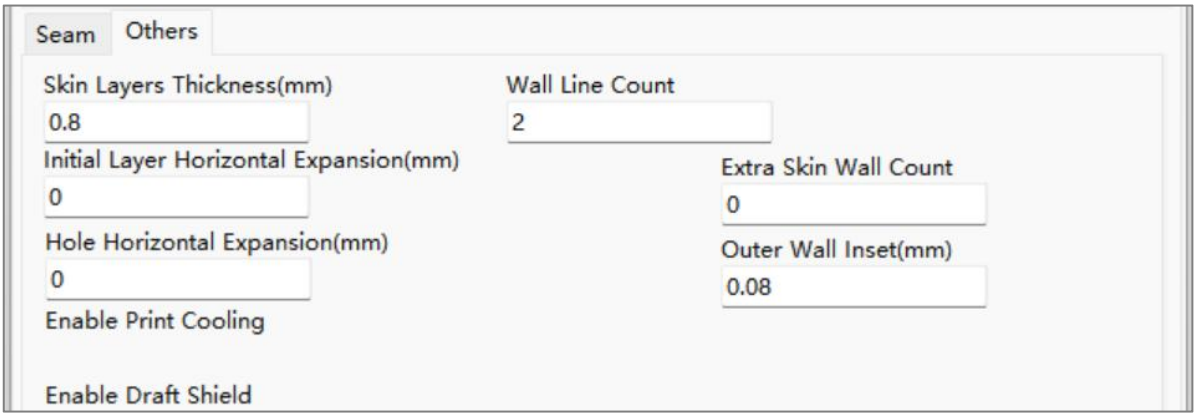
Note: Z Seam is where printer finishes outer layer, may cause blob/zit when changing Z height. Aligned seams make noticeable line (Z Seam) from oozing at start/stop. Options minimize this.

- Z Seam Type: Determines Z Seam location
  - Shortest: Most time-efficient start/stop
  - User Specified: Set X/Y start/stop
  - Random: Random start/stop prevents column buildup
  - Sharpest Corner: Start/stop at sharpest model corner
- Z Seam X/Y (mm): X/Y location of Z Seam. Only for User Specified type.
- Hiding Seam Preference: For Sharpest Corner type, puts seam inside or outside corner

- Z Seam Relative: Relative to object center or absolute on build plate. Only for User Specified type.



6.11.11.Others Tab



- Skin Layers Thickness (mm): Top and bottom skin layer thickness
- Horizontal Expansion (mm): Fine-tune part size to offset cooling shrinkage for tighter tolerances
- Skin Alternate Rotation: Top/bottom layers change 90 ° each layer. This adds 45 ° rotation every 2 layers.

  Normal top/bottom layer directions:

  Layer 3 with Skin Alternate Rotation enabled:

- Enable Print Cooling: Direct cooling air at printed part
- Enable Draft Shield: Print wall around model to block external airflow. Used when Print Cooling off for longer cooling materials like ABS.
- Wall Line Count: Number of walls to print

7. CURA

7.1. Installation

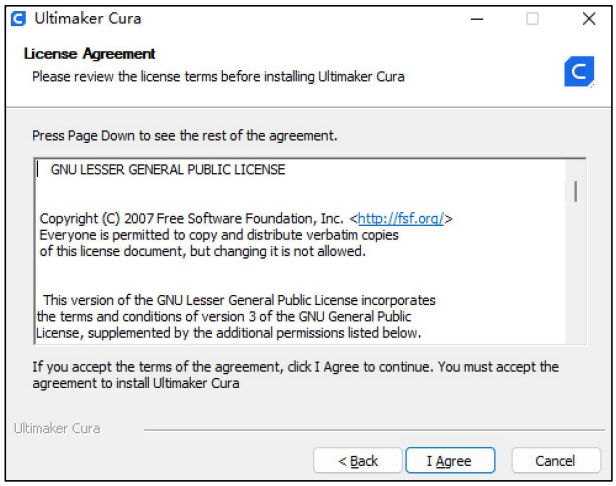
Another good slicer for Tina2S. Models sliced in Cura can't preview File Info screen. Custom Cura installer on microSD for Windows and Mac.

OS: Windows 10+ / MacOS 11.7+

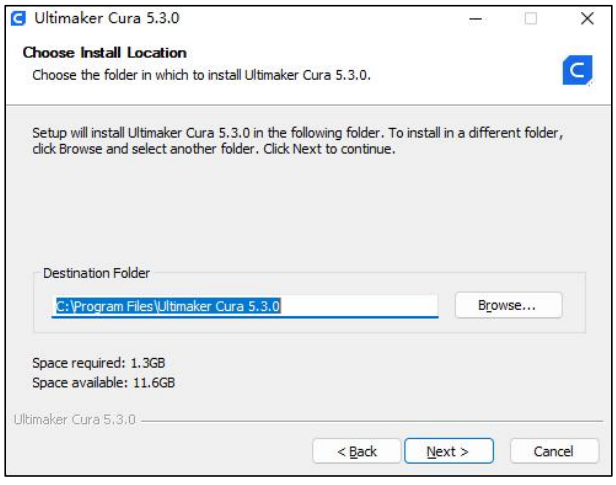
7.1.1. Installation on Windows

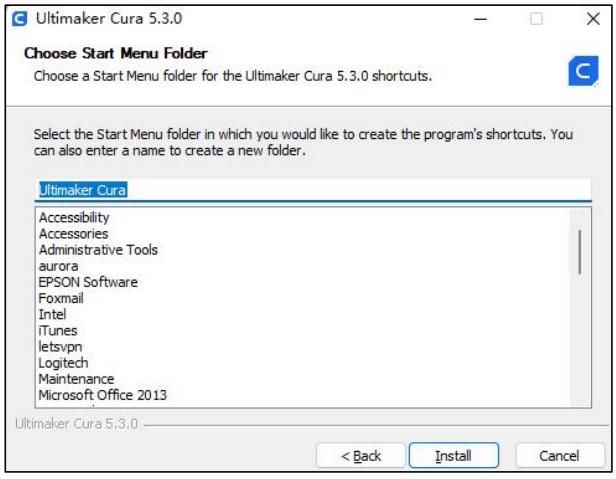
Double-click exe on microSD, then:

Welcome dialog > Next > I Agree

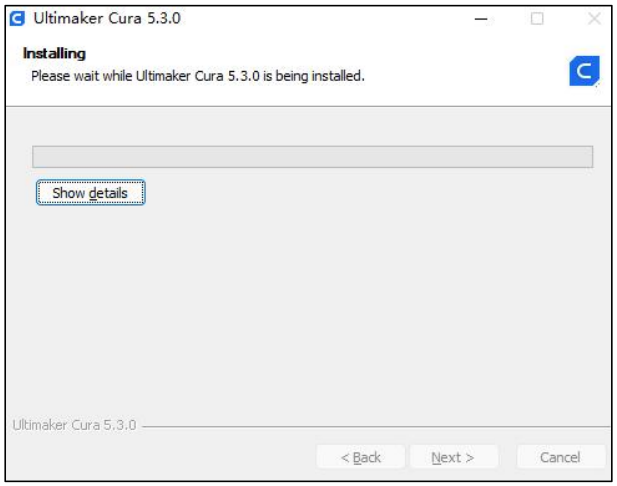


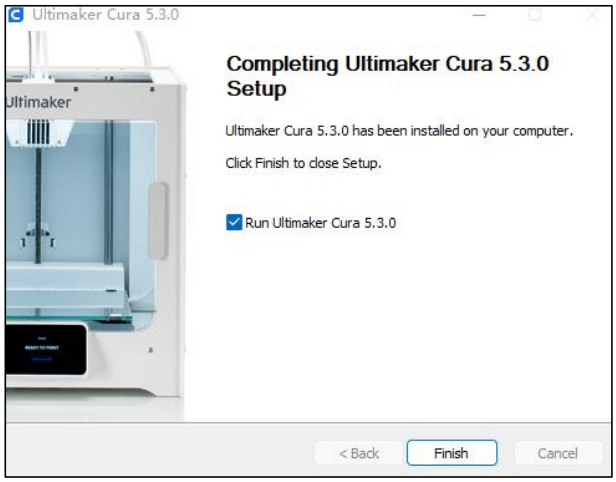
Choose install directory > Next



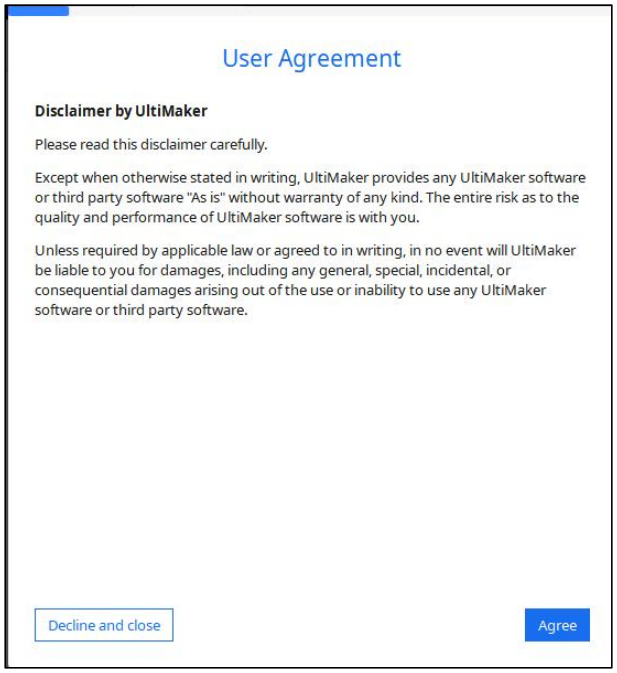


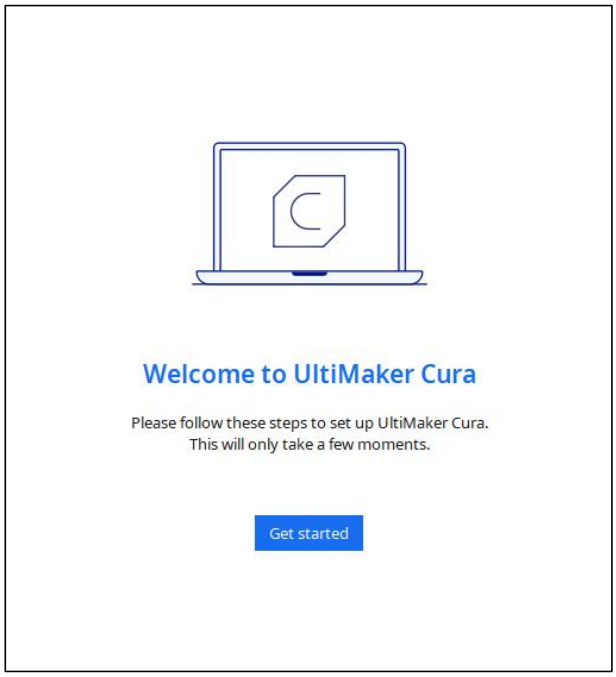
Select file associations > Install. Approve driver install if prompted. Check Run Ultimaker Cura > Finish.



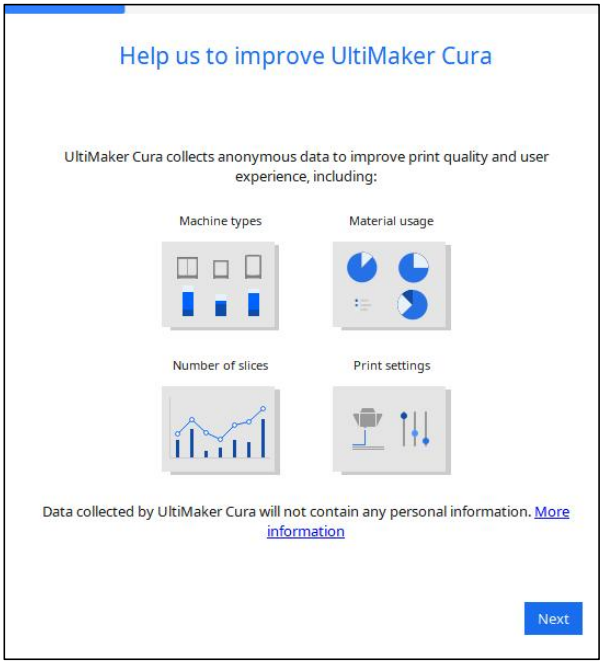


Welcome dialog > Get Started. User Agreement > Agree. If reinstalling, skip to step 6.

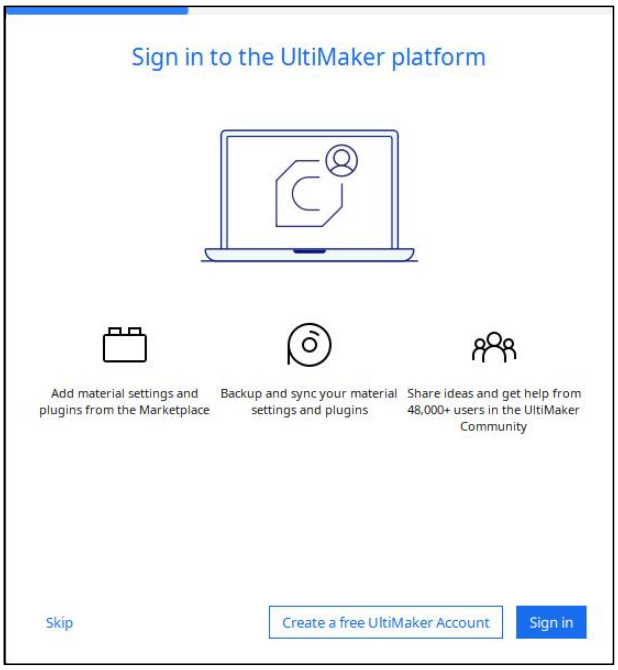




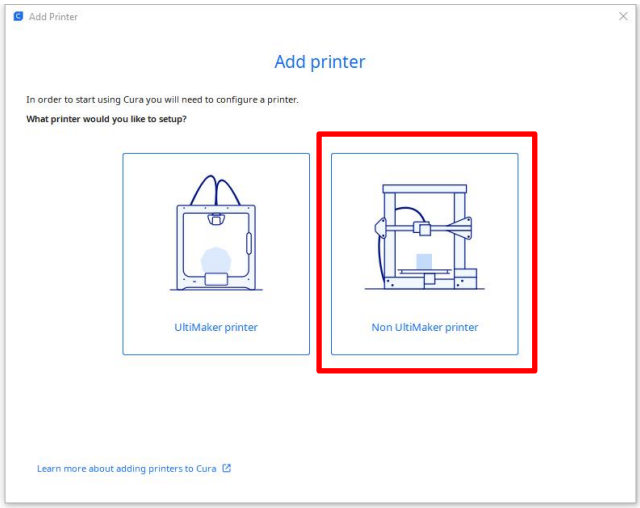
Next on next two dialogs

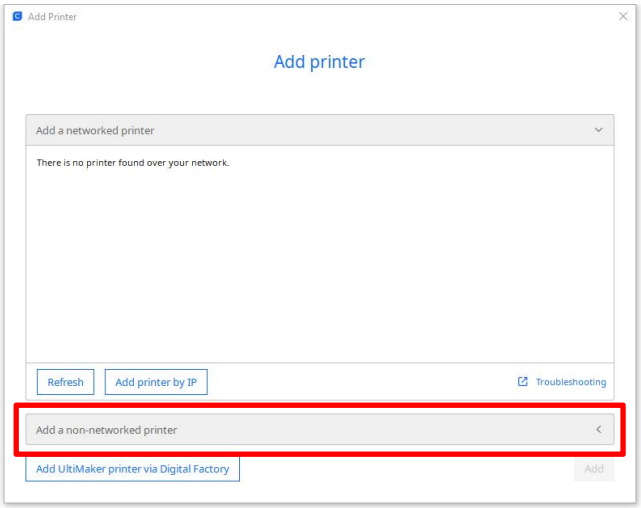


Ultimaker Cloud dialog for account creation/login > Skip

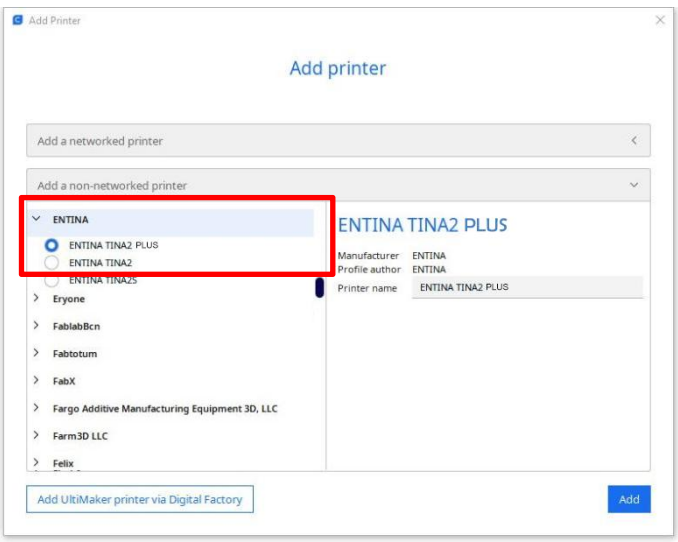


Add networked printer > Entina > TINA2 PLUS > Next. If First Run skipped, Settings > Printer > Add > TINA2 PLUS > Add, skip to step 8.

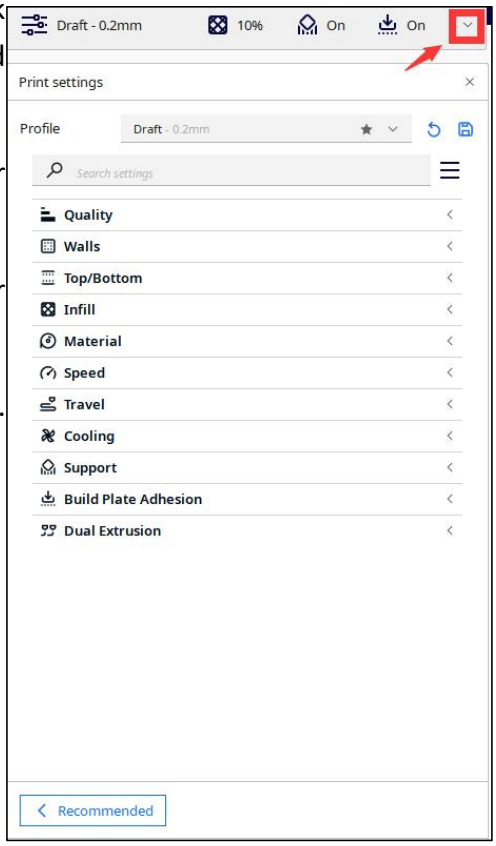




Find "TINA2Plus".Note: Not "TINA2"or"TINA2S", will fail if wrong.



Main screen loads with Tina2 defaults. To customize, click pencil > Custom. Infill Density, Support, Platform Adhesion based on model needs.



Infill Density: Overall strength. Low for decorative, high for functional parts.

Support: None for no overhang (cube), Touching Buildplate for plate overhang only, Everywhere for complex overhang.

Platform Adhesion: Brim usually sufficient and easier cleanup. Raft builds sacrificial base layer.

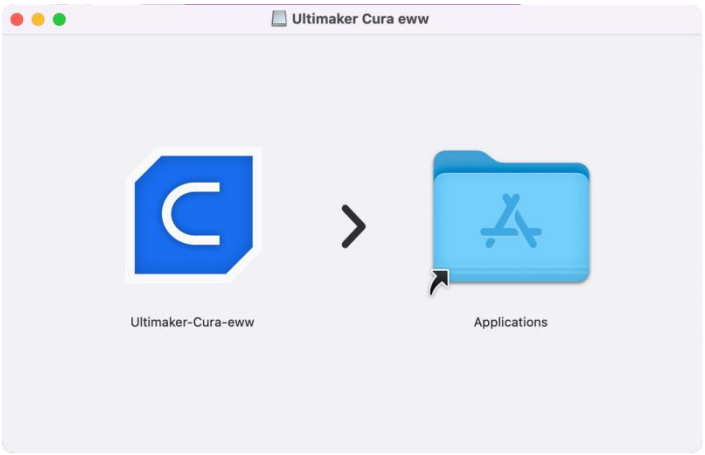
Experiment for best results.

Install complete! See Cura docs for operation details.

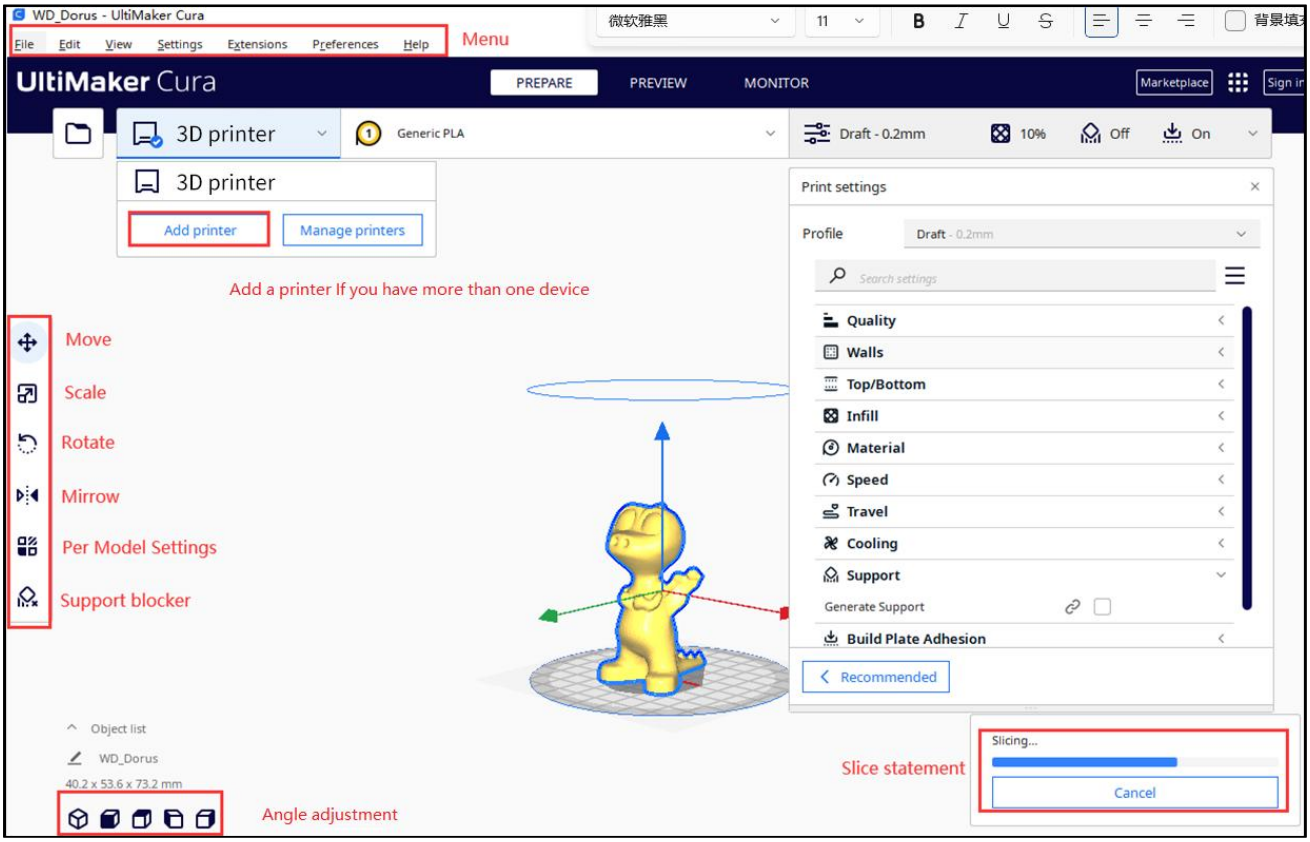
7.1.2.Installation on MacOS

OS: MacOS 11.7+

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

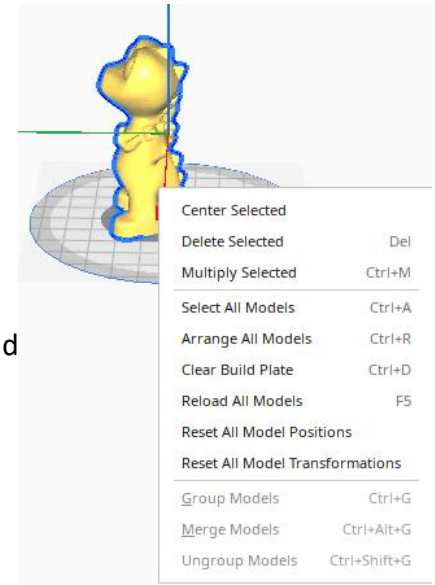


7.2. Interface Introduction



- Menu Bar: Access Cura menu
- Move, Scale, Rotate, Mirror: Adjust model size and position
- Support Blocker: Manually add supports

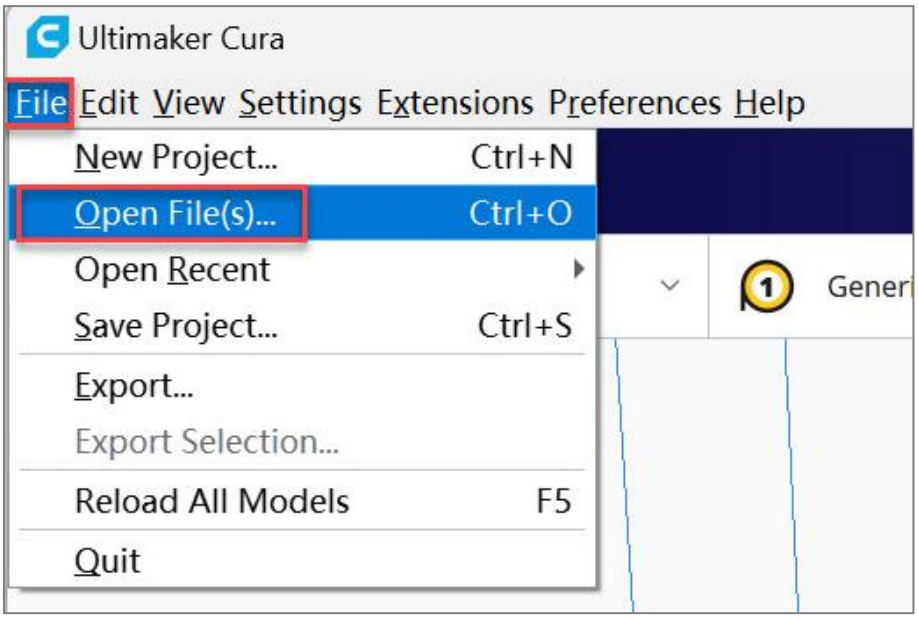
Right-click model for context menu:



- Center Selected: Auto-center model on platform
- Delete Selected: Remove model
- Multiply Selected: Duplicate multi-part model
- Arrange All Models: Auto-adjust multiple model position and spacing
- Clear Build Plate: Remove all models

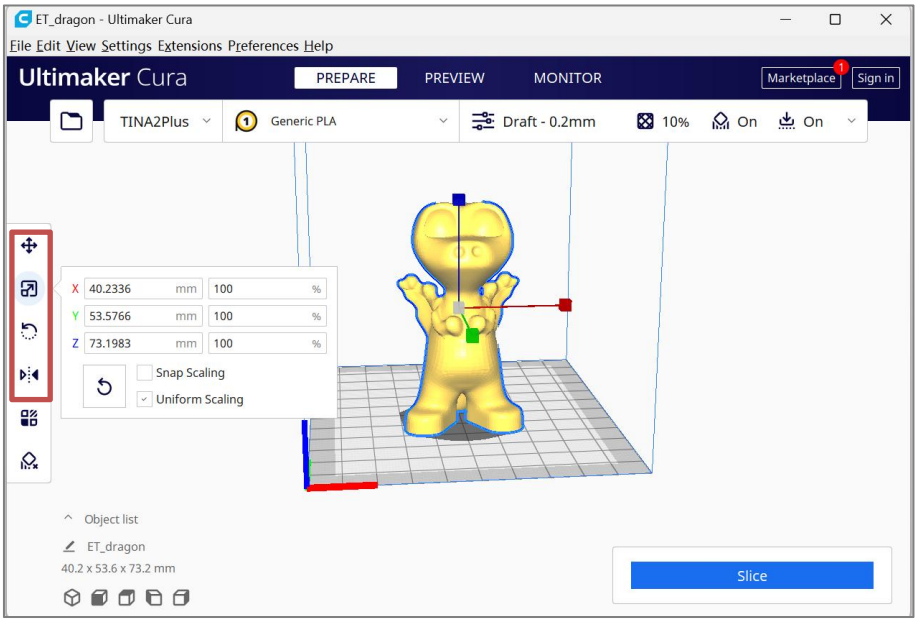
7.3. Add Model

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



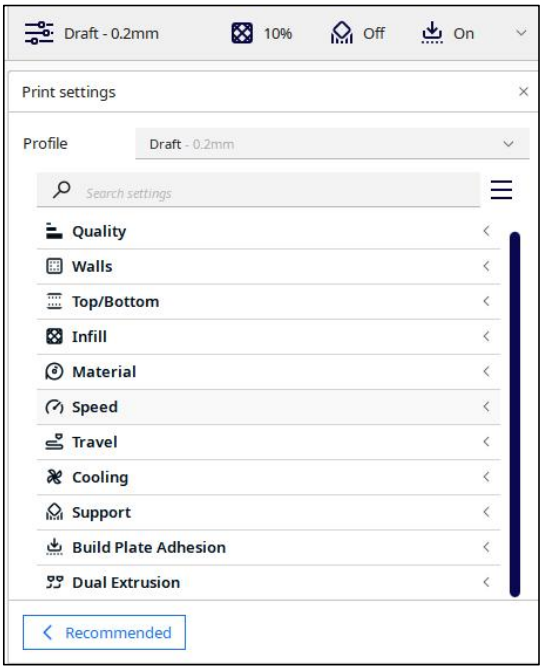
7.4. Adjust Model

Click model to adjust size, angle, position.

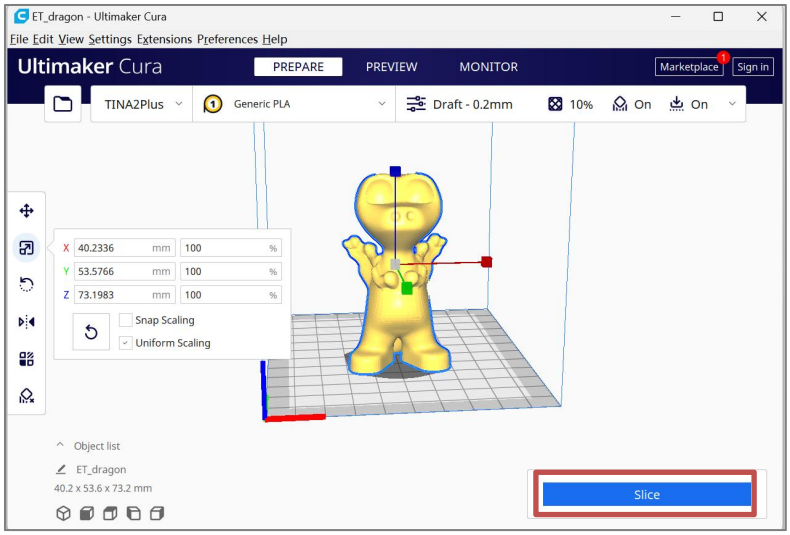


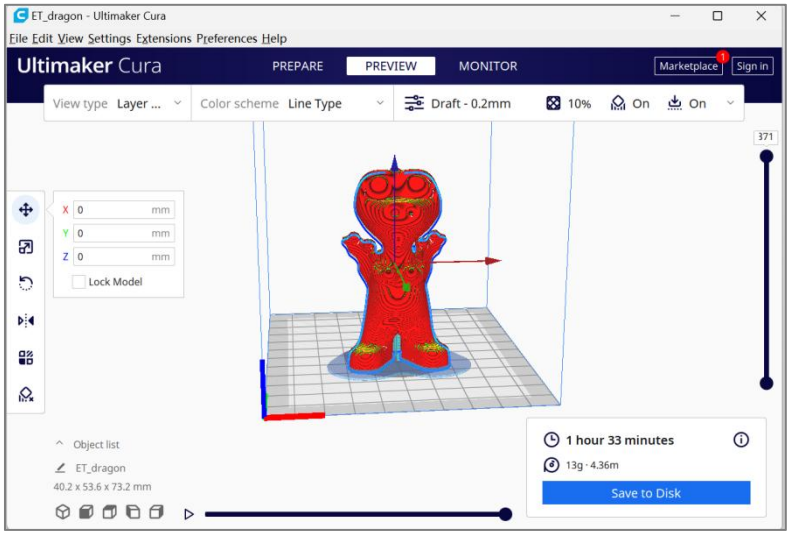
7.5. Slice

Beginners use defaults without adjustment.



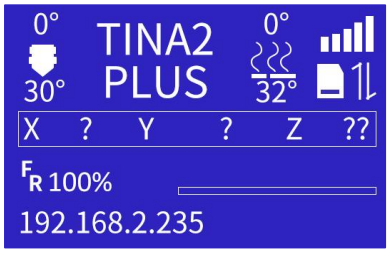
After setting parameters, click Slice to process.



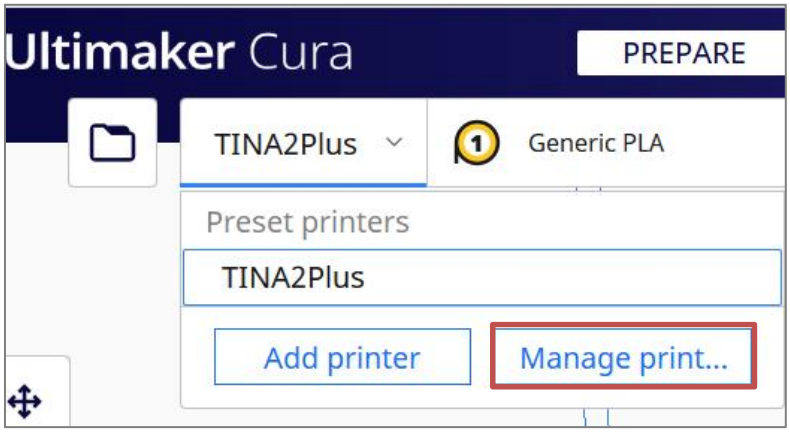


7.6. Send files via Network

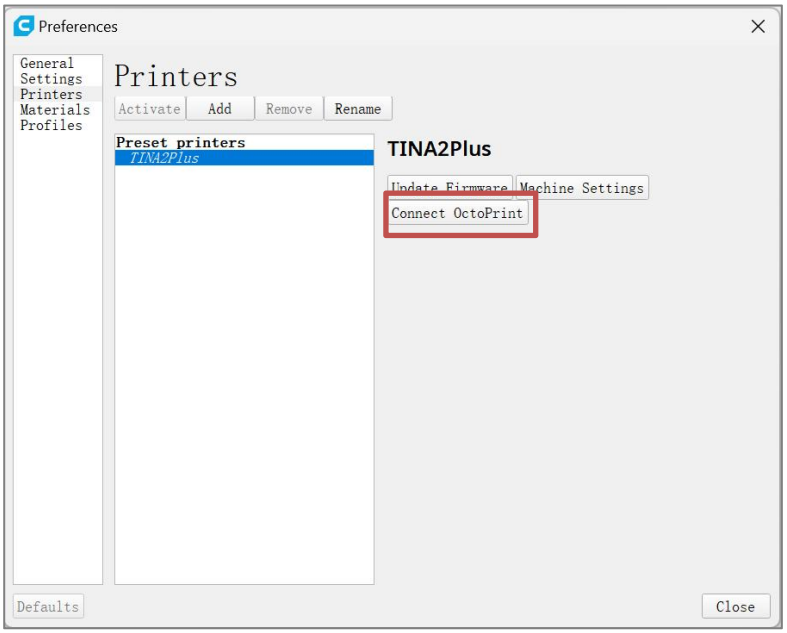
Before sending the print file through the network, make sure that the 3D printer is connected to the network, and the IP address is displayed. Please refer to: "**Print by app**".



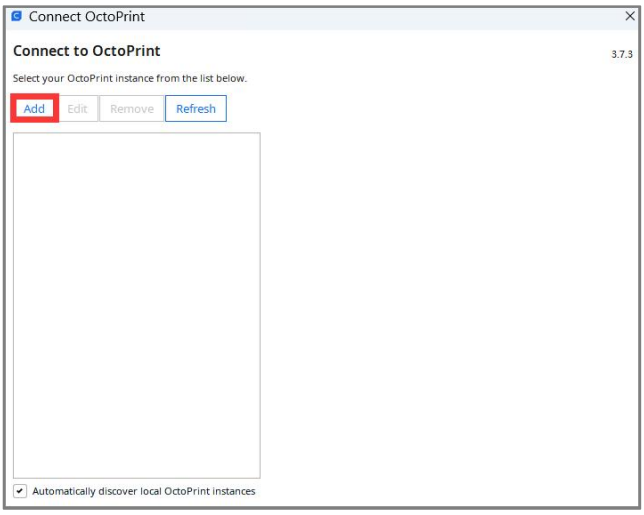
Click “manage printers”

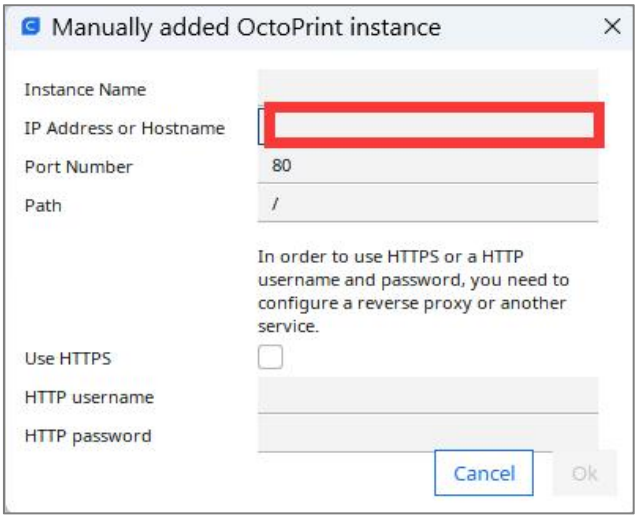


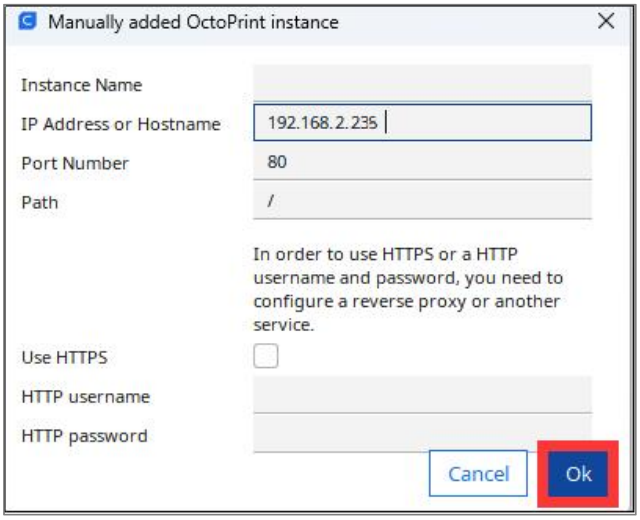
Choose “Connect Octoprint”

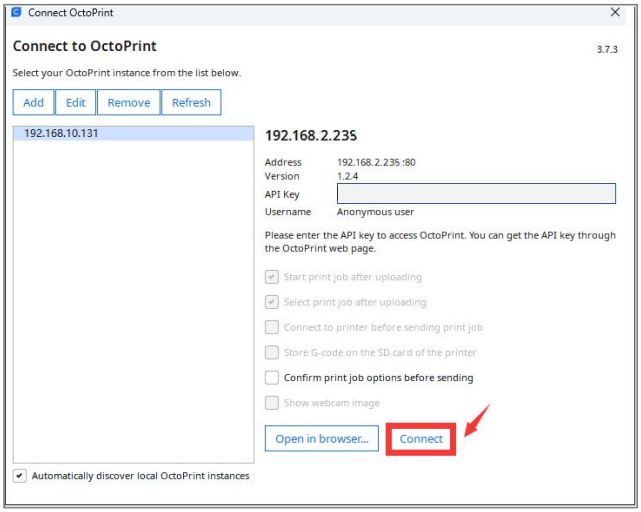


Enter the IP address and click **ok** to connect

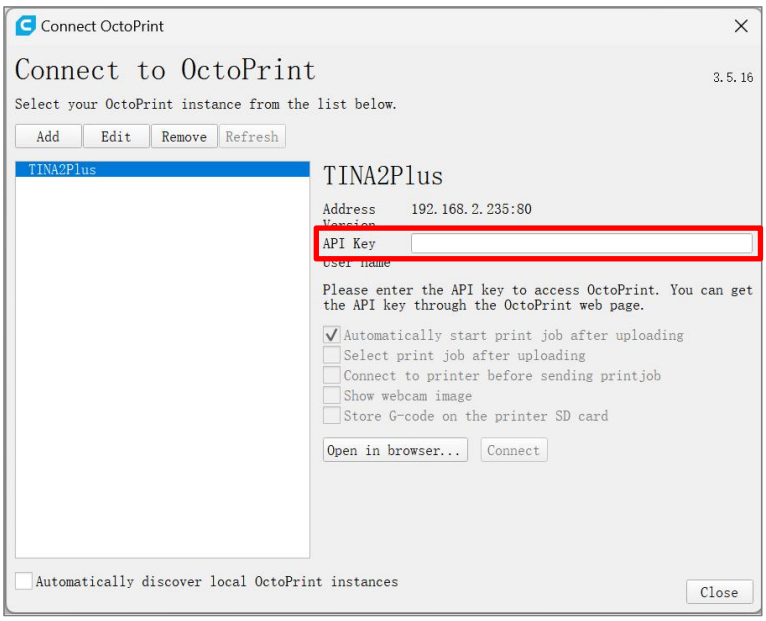






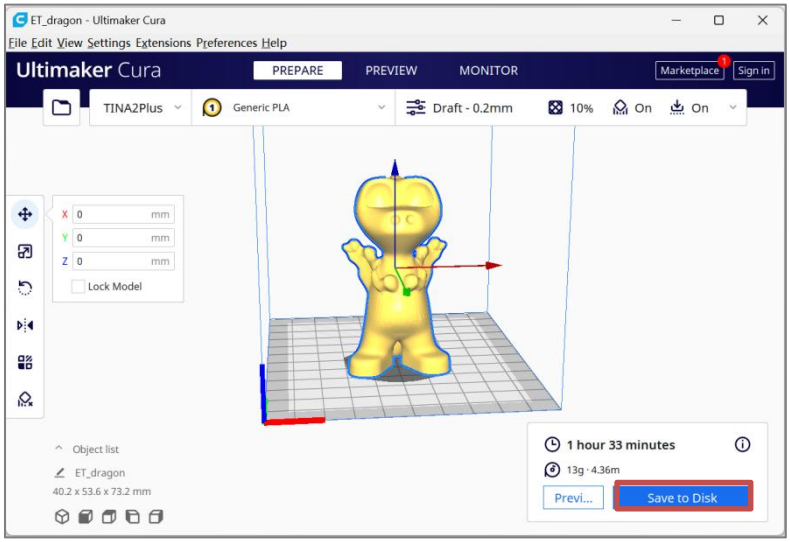


Some versions of Cura need to fill in the **API key** : 123

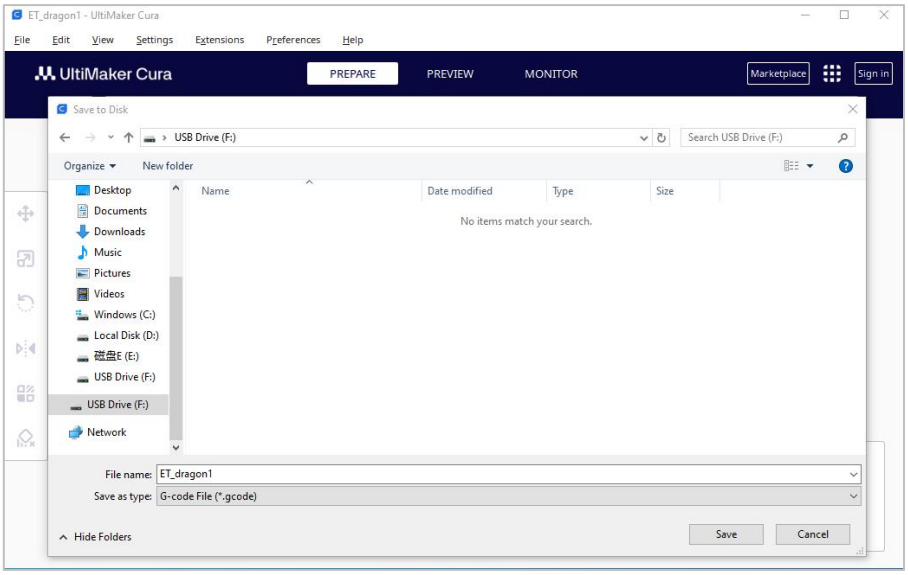


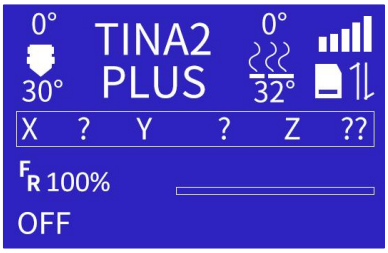
7.7. Send Files via TF card

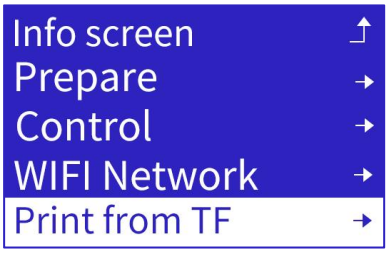
Save sliced model to TF card, insert into printer.

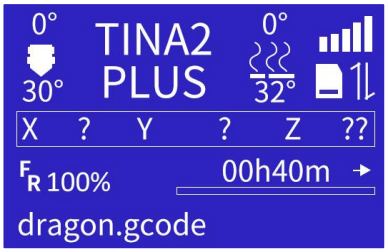


Note: Save directly to TF root, not in folder.





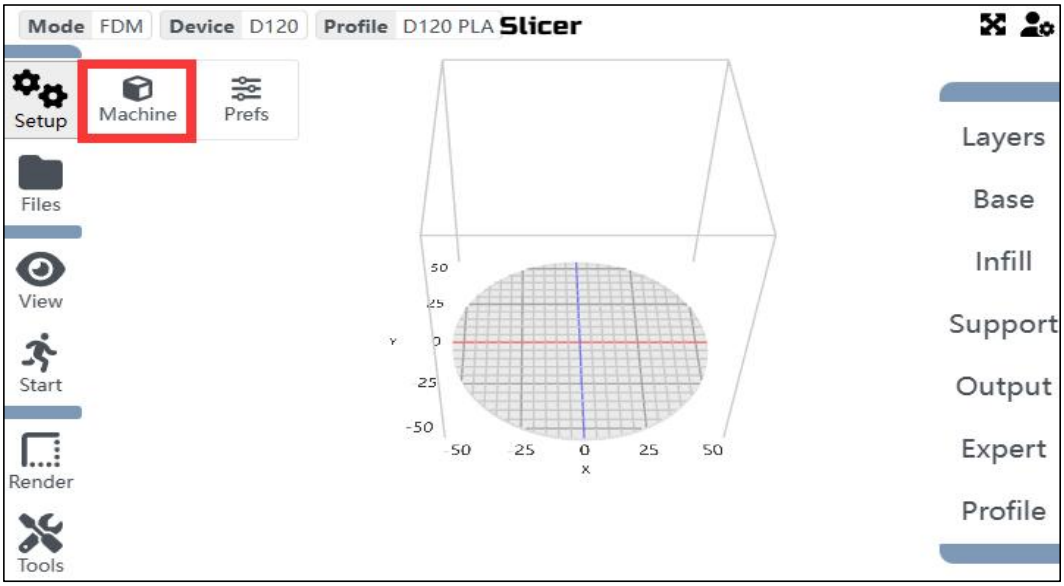




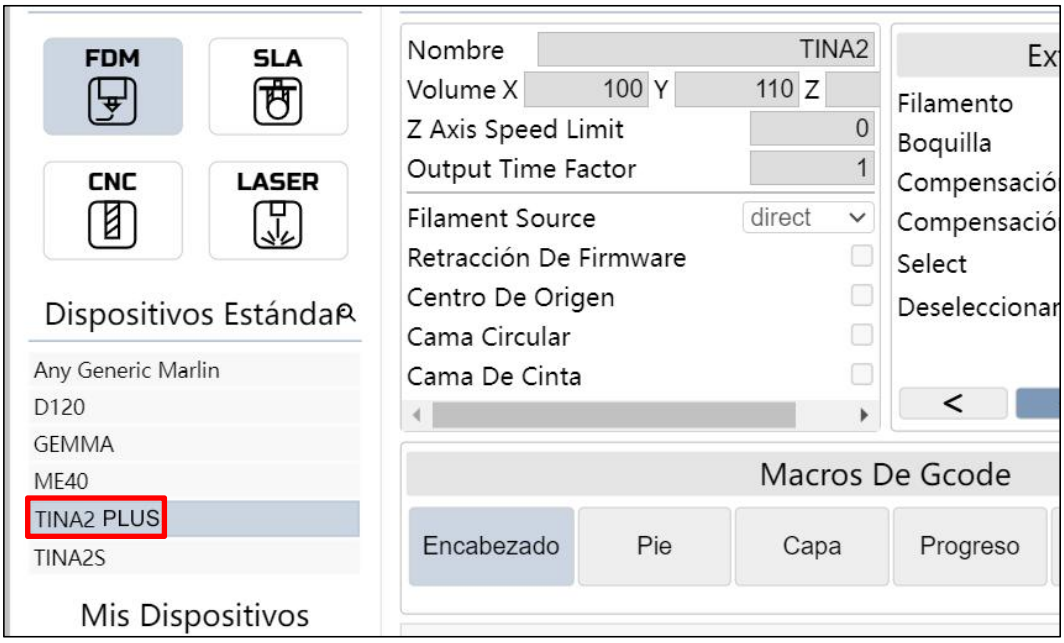
8. KIRI

Go to https://slice.wiibuilder.net/kiri/ for Chrome OS or iPad.

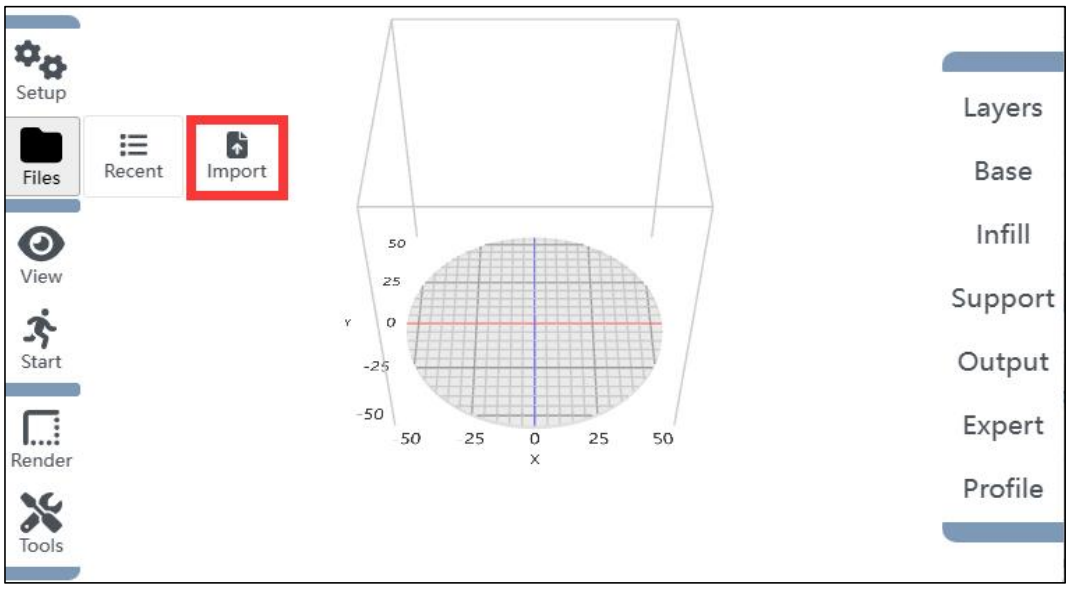
Setup > Machine to select printer model



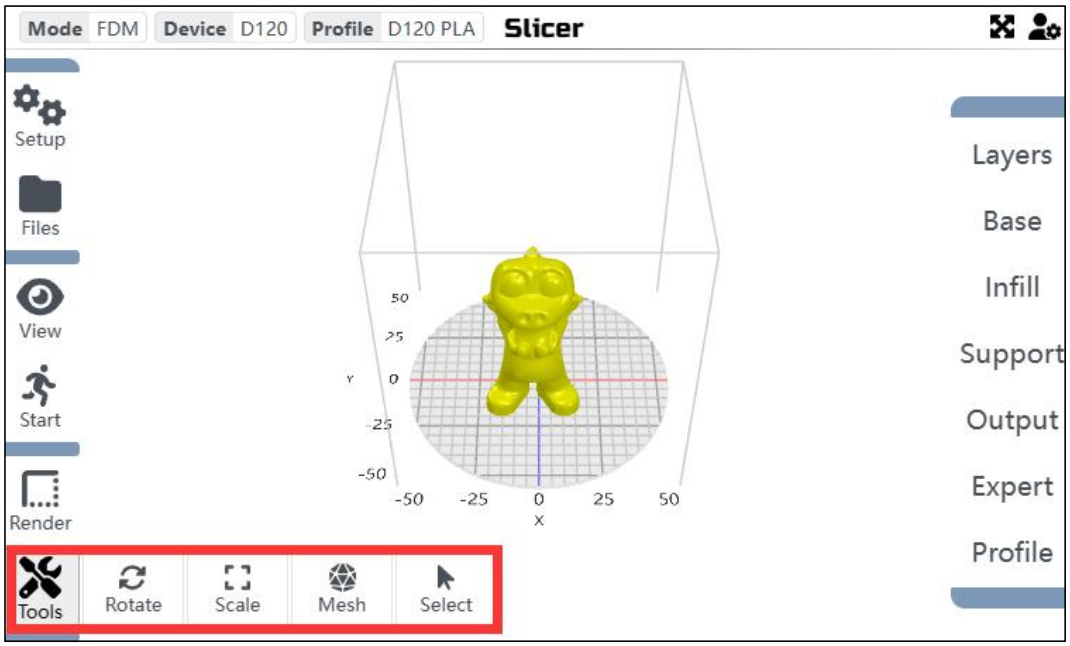
Find "TINA2Plus".Note: Not "TINA2"or"TINA2S", will fail if wrong.



Files > Import to load model



Tools to adjust model size and position.



Slice > Export to TF card for printing.