MINGDA

User Manual

[MD-600D 3D Printer]

*Please read this guide carefully before using this printer

Shenzhen MINGDA Technology Co.,Ltd

V2.0

Thank you for choosing MINGDA Technology's products!

For the best experience, please read this user manual carefully and follow the instructions to operate the printer. If you encounter any issues with the printer, please contact us using the contact information provided at the end of this user manual. Our team is always ready to provide you with high-quality service.

To enhance your usage of our product, you can also learn how to use the printer through the following means:

1. User Manual: Relevant instructions and videos can be found on the included USB drive.
2. You can also visit our official website (**www.3dmingda.com**) for information on software, hardware, contact details, device instructions, device specifications, and warranty information, among other things.

Cautionary Notes

1. Please do not place the printer in environments with significant vibrations or instability, as machine shaking can affect the print quality, please make sure the ground is level.
2. Avoid touching the nozzle and heated bed while the printer is in operation to prevent potential burns from high temperatures, resulting in personal injury.
3. Refrain from moving the device during the printing process to prevent accidents and injuries.
4. Do not dismantle the equipment or alter circuit settings without authorization.
5. Avoid using the device in high-temperature or humid environments to prevent compromising device performance or creating safety hazards.
6. In case of an emergency, immediately cease using the device and power it off.

Contents

1. Overview

This manual provides instructions on the usage of the 3D printer, covering aspects such as an overall introduction to the device, operational procedures, maintenance, and care. The aim of this manual is to assist you in correctly using and maintaining the 3D printer, ensuring device performance and safety, extending the lifespan of the equipment, and enhancing print quality. We hope that you follow the requirements and recommendations outlined in this manual during usage, and maintain attention to and care for the equipment. Thank you for choosing our product, and we wish you a pleasant experience!

2. Device Introduction

Device Parameters

Basic Parameters	
Product Model	**MD-600D**
Machine Dimensions	1310*965*1255mm
Max. Build Dimensions	600*600*600mm
Print Technology	Fused Deposition Modeling (FDM)
Rated Voltage	100-240V, 50/60Hz
Rated Power	2400W
Ambient temperature	10℃-30℃ / 50°F-86°F
Extruder Type	Dual Extruders
Max. Nozzle Temperature	350℃
Max. Bed Temperature	110℃
Screen	10inch touch screen
Printing Method	USB Flash Disk / LAN Printing
Connection	USB Flash Disk / WIFI / Ethernet
Power Loss Recovery	Yes
Filament Detection	Yes
Fast Auto leveling	Yes
Camera	Yes
Fast Calibrate Offset	Yes
Supported Filament	**Common filament:** PLA, PETG; **Engineering filament:** PA-CF/GF, PET-CF/GF, HtPA-CF/GF, PA-GF25/CF25; **Support filament:** S-Mulit, S-HtPA, etc

Tool List

U-disk

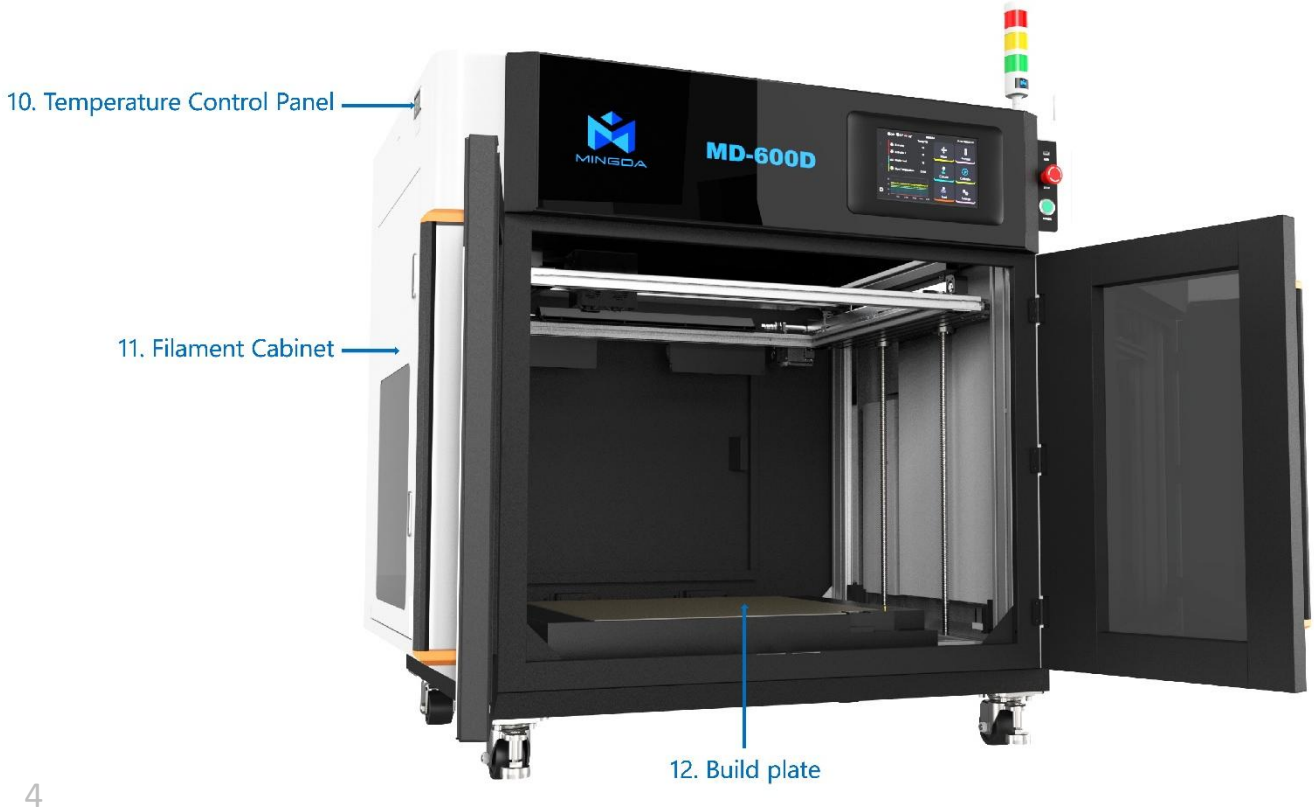
Allen wrench

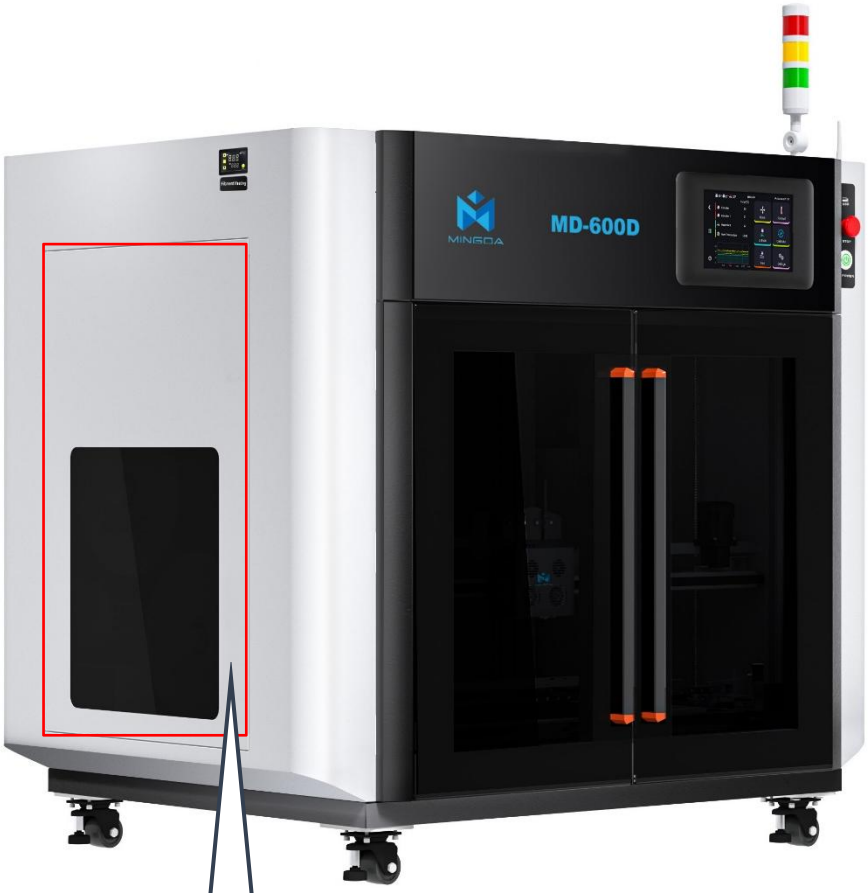
Antenna

Indicator light & screw

Note: **The picture is for reference only. When the real thing is inconsistent with the picture, the actual object shall prevail.**

Machine Components Overview





13. Filament Feeding Assembly

14. Extusion board

3. Operational Steps

Part assembly

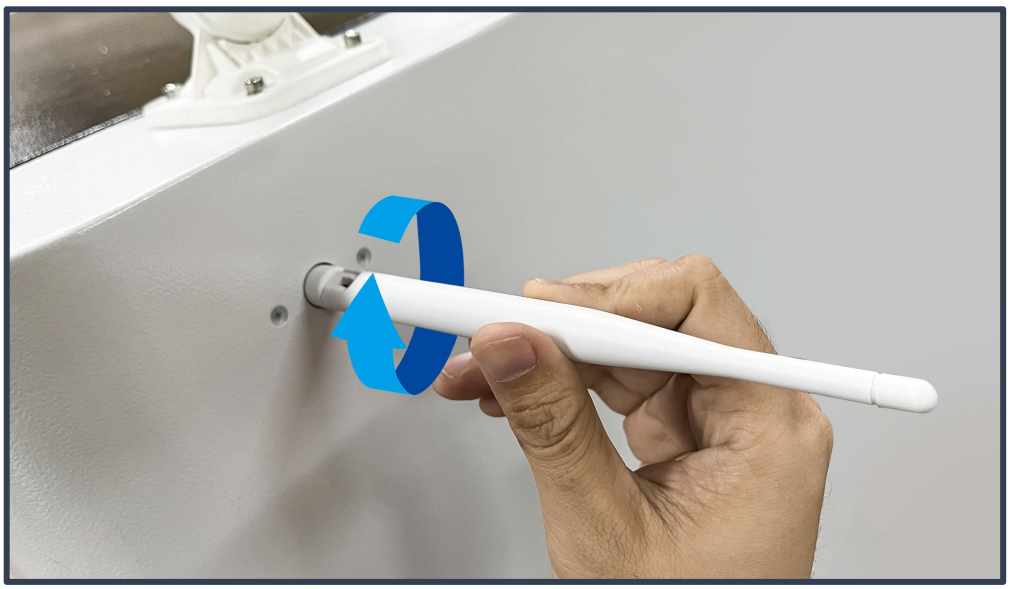
1. Install the indicator light

Green light: Indicates that the printer is working properly.

Red light: Indicates that the printer is in an emergency stop or fault state, requiring immediate attention or repair by the user. For example, overload, short circuit, or error message on the screen.

Yellow light: Indicates that the printer is in a warning or abnormal state, requiring user attention or intervention. For example, insufficient filament or paused printing.

2. Install the Antenna



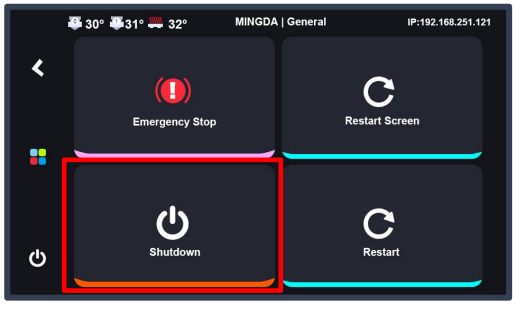
3. Power on

Insert a power socket

Press the power button to turn on the printer

4. Power off

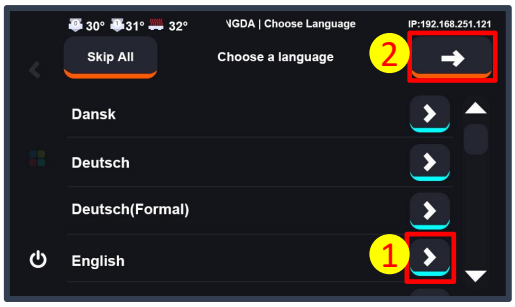
When you turn off the printer, please don't press the power directly! Click " Shutdown " to turn off the printer



Click the "**Shutdown**"

Startup Configuration

1. Select Language



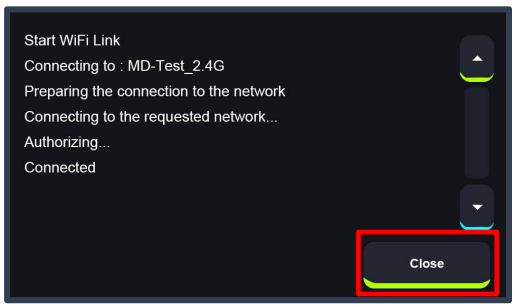
Click to choose the language, and click to proceed to the next step.

2. Wi-Fi

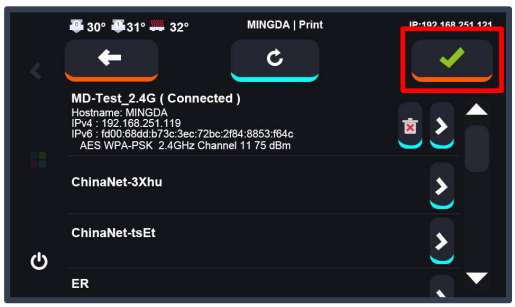
Note: **If you find that the printer cannot detect your WiFi signal, you can click to skip this step for now. After completing the startup wizard, move the printer to a location closer to a stronger signal source, and then reconnect to WiFi.**

Select the WiFi and click
(If your WiFi cannot be displayed for more than 20s, please click to refresh)

Enter the WiFi password and click

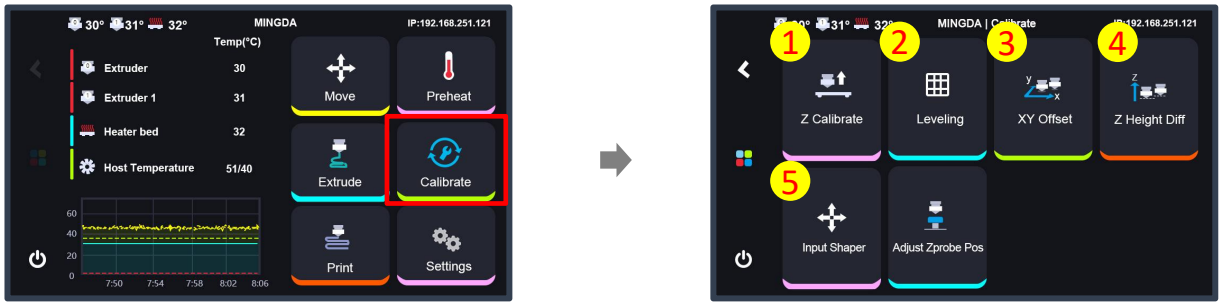


Once the connection is successful, click Close



Upon successful connection, click the in the upper right corner to enter the main interface of the machine. If you do not need to connect to the network, you can also click the to skip this step.

Printer Calibration



Tip: Please follow the sequence shown in the diagram to calibrate step by step.

Z Calibrate → Leveling → XY Offset → Z Height Dif f → Input Shaper → Calibrate PID

If the nozzle is not aligned with the center of the metal sensor, use 

to manually adjust the nozzle position until it is directly above the sensor.

Note：

1. Z-calibrate, Auto-leveling, XY offset and Z height diff must be performed after replacing the nozzle, hotend, or removing/reinstalling the extruder.

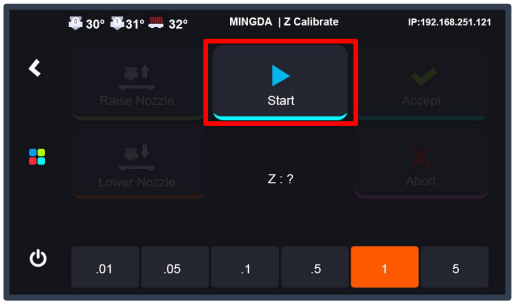
2. Leveling: Before starting the leveling process, heat the bed to 60℃ and wait for 20 minutes. This allows for thermal expansion of the glass to stabilize, ensuring accurate leveling.

Printing Tips:

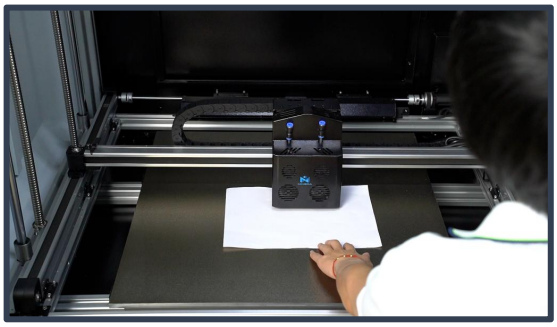
If the bed calibration temperature is set to 60°C. Before printing, keep the bed temperature at 60°C and wait for 20 minutes to ensure optimal first layer adhesion and print quality.

1. Z Calibrate

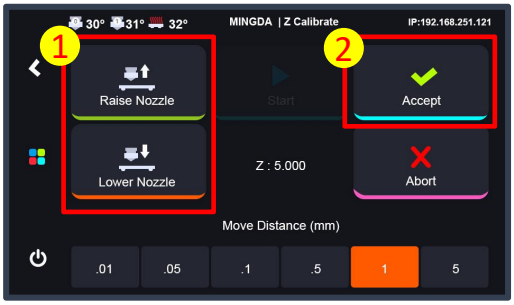
Click "**Z Calibrate**"



Click "**Start**" , wait Z axis calibrating and click "Accept" and confirm



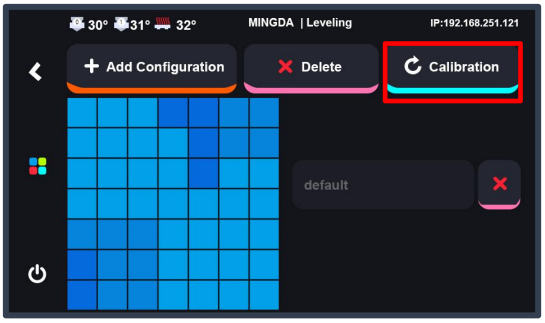
Put an A4 paper between the nozzle and heated bed.



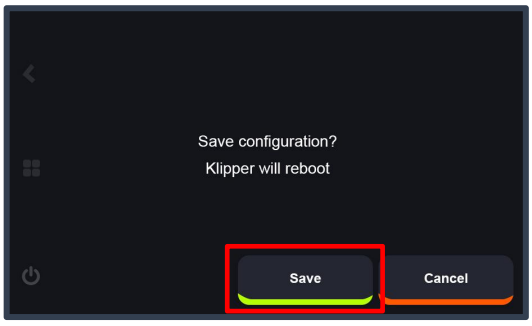
While moving the A4 paper back and forth, adjust and . When you feel slight resistance as the paper moves, you can click the to save.

2. Auto Bed Leveling

Click "**Leveling**"



Click Calibrate to start auto-leveling, which will take approximately 3 minutes



Click Save to save the value, the printer will reboot automatically.

Tips: It is better to heat up the bed to the target degrees and keep it warm up for 5-10 minutes before leveling.

Printing Tips:

1. Bed Leveling Frequency:

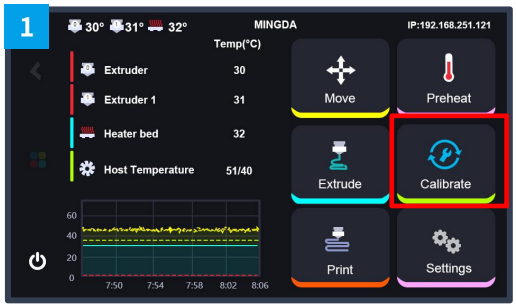
It is recommended to perform bed leveling every 20–25 print jobs. The Frequent removal and repositioning of the glass bed can gradually lead to positional deviation. Timely leveling ensures consistent print quality.

2. How to Determine if Leveling is Needed during printing:

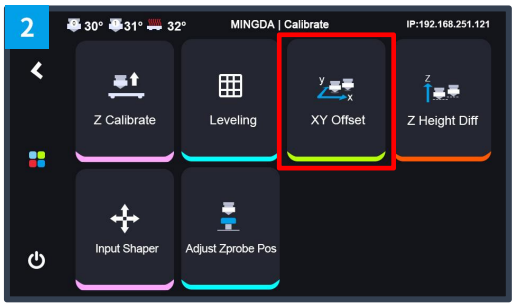
Carefully observe the first layer when starting a new print. If there is a large gap between the filament and the heated bed, poor adhesion, warping, curling, or uneven extrusion (e.g., some areas too thick, others too thin, or broken lines), it indicates that bed leveling is required.

3. XY Axis Offset Calibration

Tips: Wipe the nozzle clean before calibration to avoid any interference during the process.

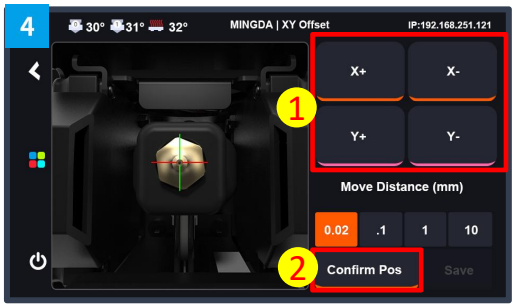


Click "**Calibrate**" 

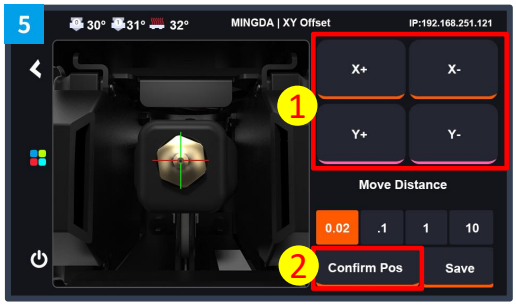


Click "**XY Offset**" 

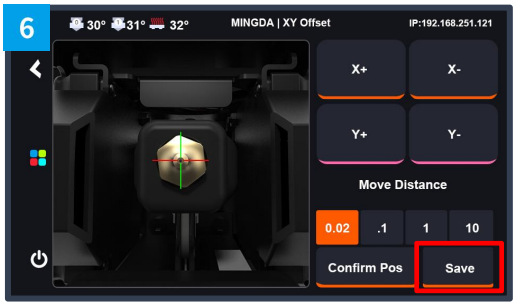
After the camera Led light up, Click "**Start**" , and the left extruder will move to the calibration camera's position.



Adjust the left extruder nozzle to align with the crosshairs  of the screen and click "**Confirm Pos**" 

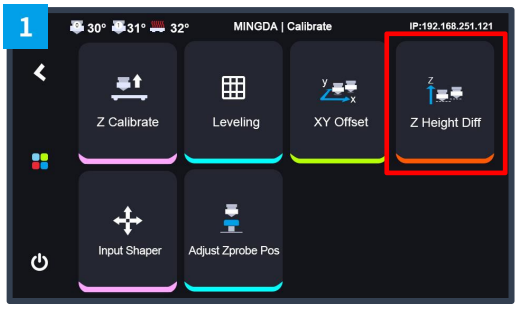


Adjust the right extruder nozzle to align with the crosshairs  of the screen click 

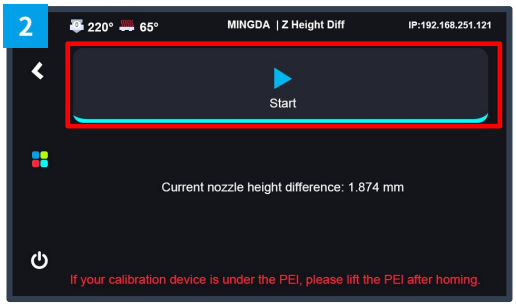


Click the "**Save**" 

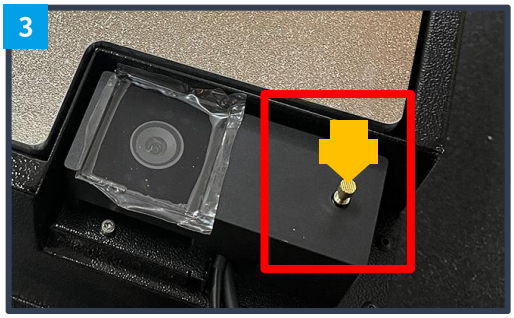
4. Z Height Diff Calibration



Click "**Z Height Diff**" 



Click "**Start**"  to start Z offset automatically.
If the printer hasn't homed before, it will home itself first.
After that, click "**Start**"  again to begin calibration.



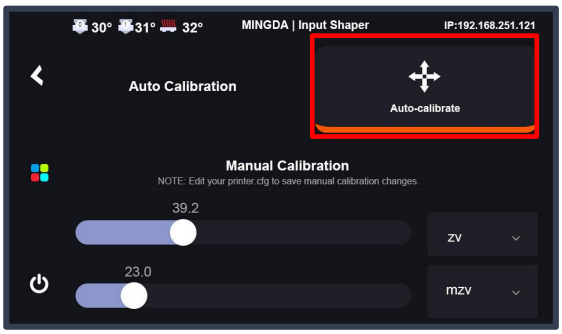
While calibrating the Z-axis offset, the extruder will move towards the sensor automatically. Please observe if the nozzle is positioned above the probe sensor. If not, use the Adjust Zprobe Pos function to adjust the nozzle.

After finishing, click "Save" and restart the machine.

Click "Start" and move the XYZ to make the nozzle on the top of the metal sensor then save the value. After than do the "Z height Diff" again

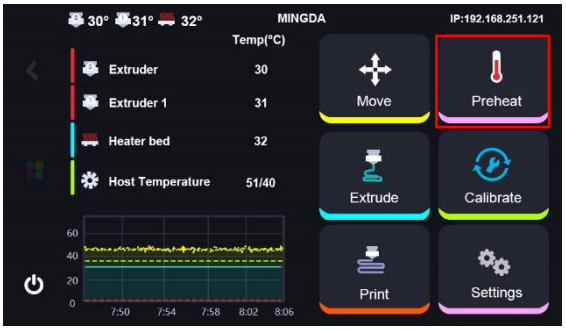
5. Input Shaper

Click "**Input Shaper**" 

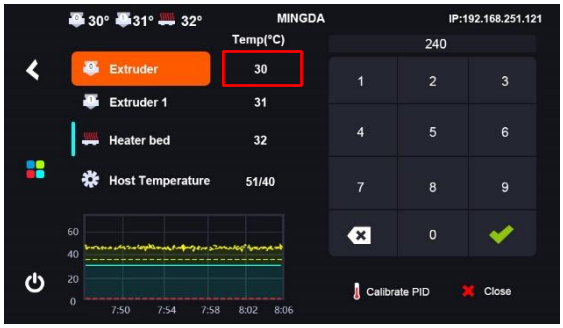


Click "**Auto-calibrate**", After calibration is complete, click the save button.

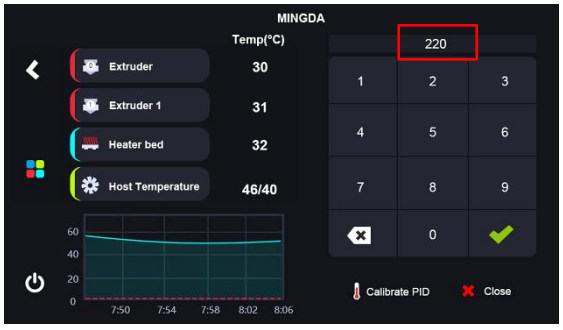
6. Calibrate PID



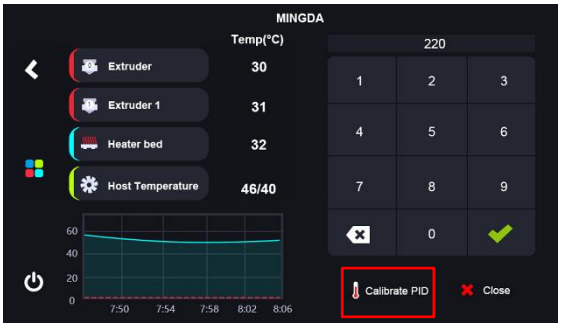
Click “Preheat”



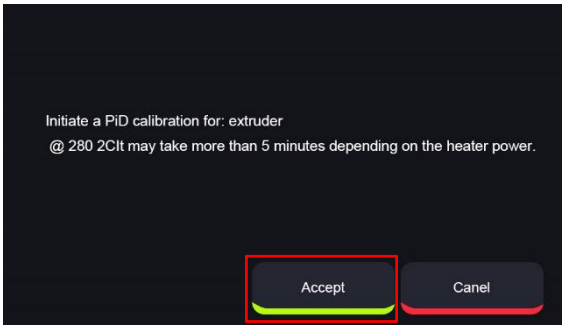
Click the temperature value



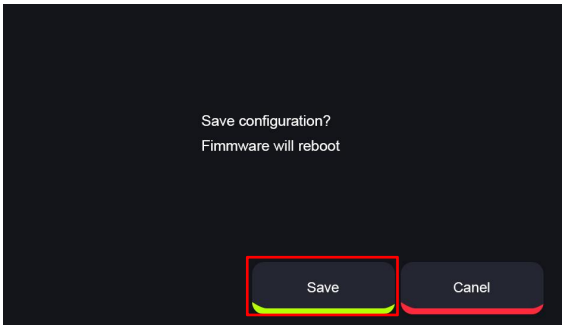
Set at ”220°C“



Click “Calibrate PID”



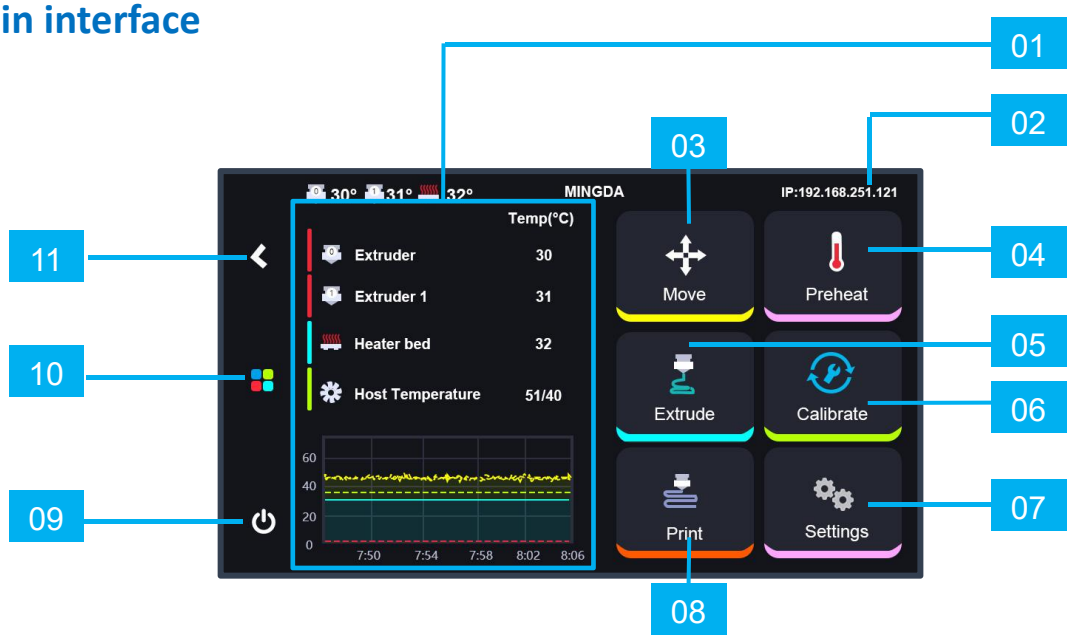
Click ”Accept“



Click “Save”

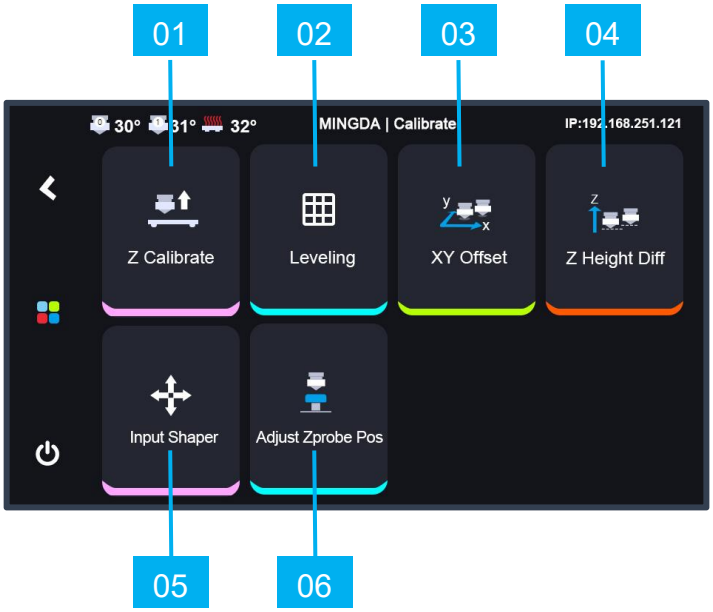
4. Operating interface introduction

Main interface



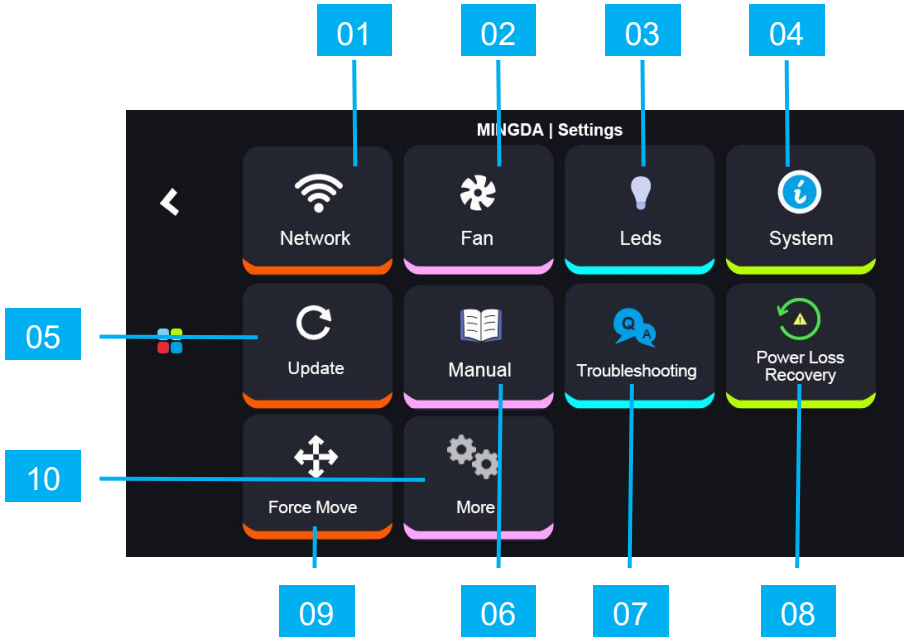
	Primary interface	Explain
01	**Temperature**	Temperature display area.
02	**IP**	Network IP address.
03	**Move**	Adjust or home the XYZ axis.
04	**Preheat**	Pre-set nozzle & hotbed's temperature.
05	**Extrude**	To load the filament.
06	**Calibrate**	Printer Calibration
07	**Settings**	Printer's printing value adjustment.
08	**Print**	Start printing.
09	**Shutdown**	Shutdown interface.
10	**Homepage**	Return to the main page.
11	**Return**	Return to the previous page.

Calibrate:



	Secondary interface	Explain
01	**Z Calibrate**	Leveling sensor calibration
02	**Leveling**	Auto-leveling
03	**XY Offset**	Calibrate XY axis
04	**Z Height Diff**	Calibrate Z axis
05	**Input Shaper**	Test the resonance compensation value.
06	**Adjust Zprobe Pos**	Adjust Zprobe Pos

Settings:



	Secondary interface	Explain
01	**Network**	To connect Wi-Fi
02	**Fan**	Cooling fan adjustment
03	**Leds**	Turn on/off Light
04	**System**	Machine shutdown, restart, reset function
05	**Update**	Update
06	**Manual**	Manual
07	**Troubleshooting**	Troubleshooting
08	**Power Loss Recovery**	Resume the print after a power failure or system error
09	**Force move**	Move the toolhead if necessary before resuming the print
10	**More**	Includes some basic settings such as time, language, screen timeout, notification sound toggle, and automatic shutdown after printing completion.

Printing Interface:

Figure 1

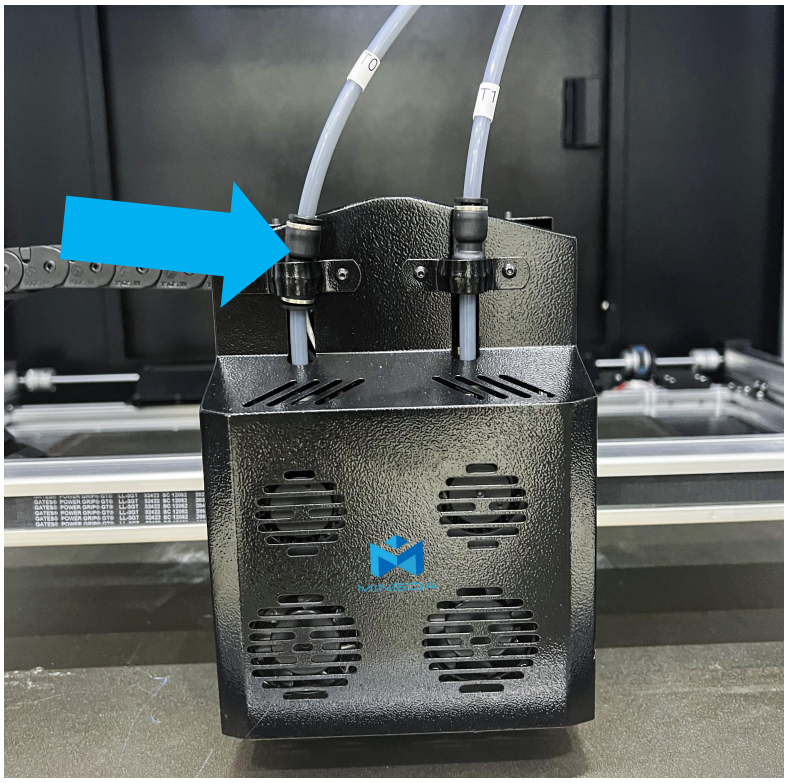
Figure 2

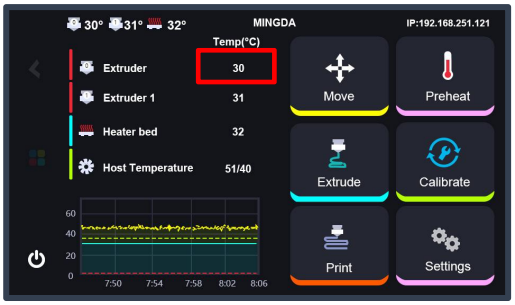
	Secondary interface	Explain
01	**Cancel**	Stop printing
02	**Pause**	Pause printing
03	**Fine Tuning**	Adjust Z-offset, Printing Speed, Printing Flow [Please refer to Figure 1.]
04	**Settings**	Basic setting, browse camera, adjust cooling fan, light , Extrude and Exclude object. [Please refer to Figure 2.]
05	**Stop**	Emergency stop .

5. Insert filament

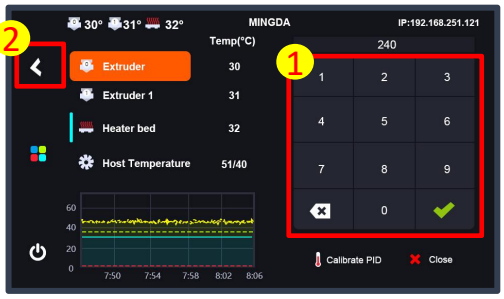
1.Hang the filament onto the filament holder as shown in the picture, and insert the filament into the corresponding tube and push it until it cannot be pushed anymore. The internal gears will automatically pull the filament in.

2. When the gears stop moving, the filament may still remain inside the tube. We need heat up the nozzle to target degrees and then load the filament.

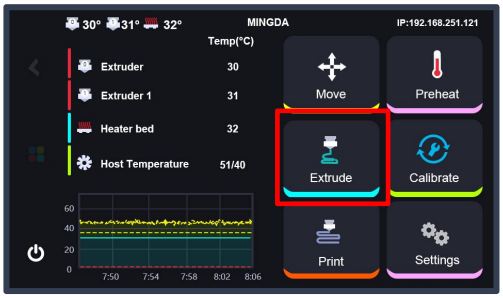




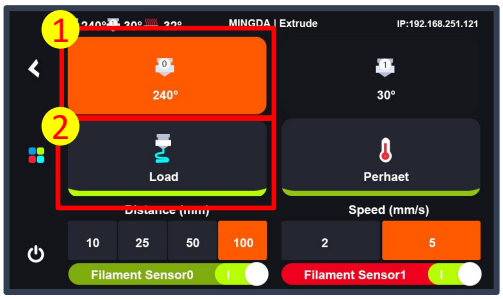
Click the temperature setting



Enter the filament recommand temperature value, click  then click 



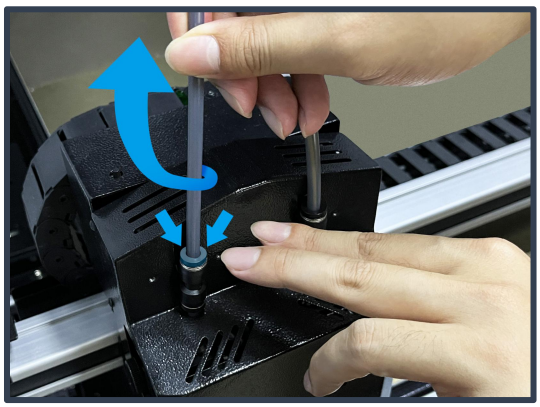
Click the "**Extrude**" 



Click Extruder “T0”  , select Distance “100mm” and Speed”5”, click Load  2-3 times till the filament goes out of the nozzle.

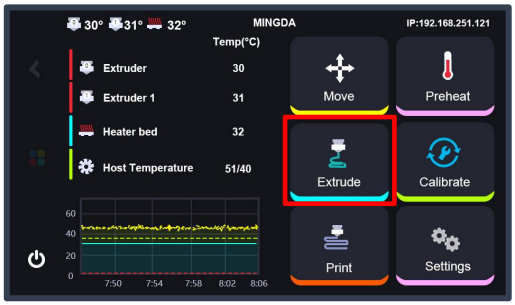
Repeat the same steps to load the filament into the right extruder.

6. Unload Filament

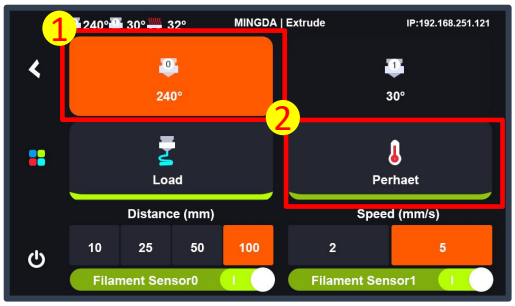


1. Press the gas connector and pull the Telfon tube out.

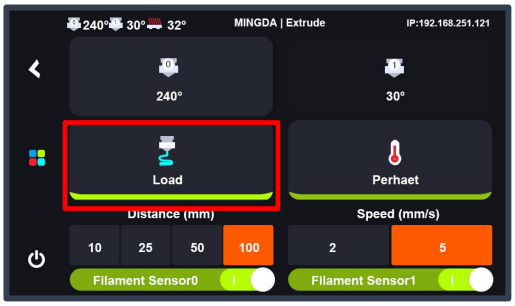
2. Cut the filament



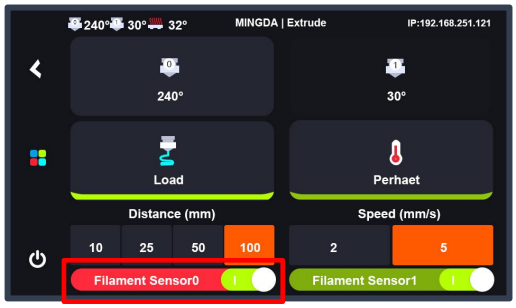
3. Click the "Extrude" 



4. Select the extruder you want to unload filament and heat it up to filament recommand temperature value.



5. Click “**load**”  to feed out the rest filament



6. Feed out all the rest filament inside the extruder until the screen 'filament sensor0' turns red.

7. Click the T0 unload button in filament cabinet.

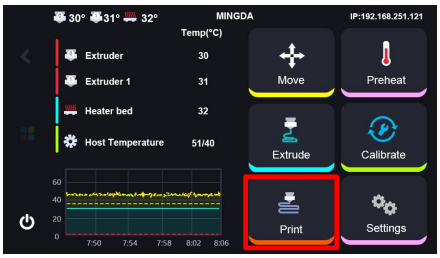
Keep clicking unload button until the filament finished unloading.

When you click the unload button, please roll the filament back manually .

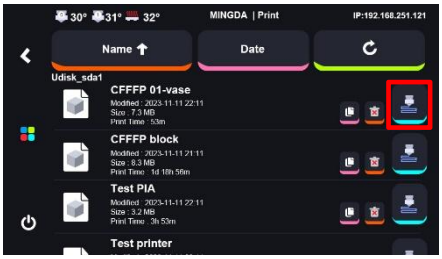
7. Printing

Test Printing

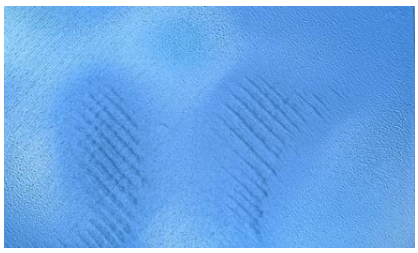
Before printing, please test the gcode which shipped from the printer first



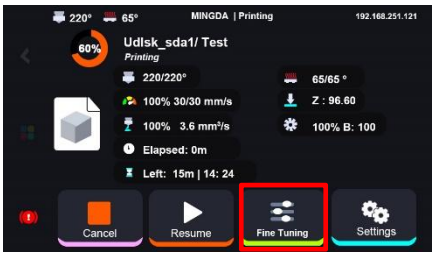
Click “Print”

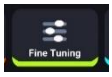


Click Print icon

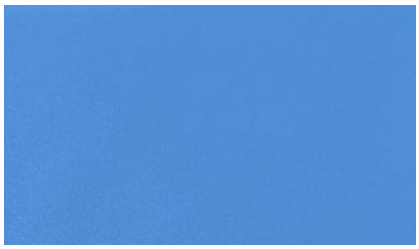


Observe the first layer during printing



Click Fine-tuning 

Select the distance at 0.02 and increase the Z-offset  until the line is smooth



The final result after adjusting

Explain:

The first layer very important

1. There are gaps between each lines, that means the Z-offset need to be lower.

2. There are waves on the printing surface, that means the Z-offset need to be higher

Printing Tips:

During printing, if the model is too close to or too far from the build plate, you can use the fine-tuning function to adjust the nozzle height:

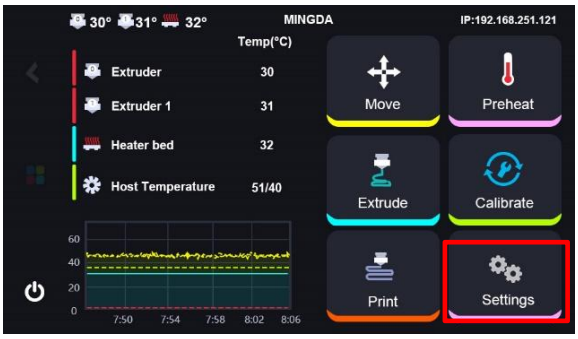
1. Tap "Fine Tune".

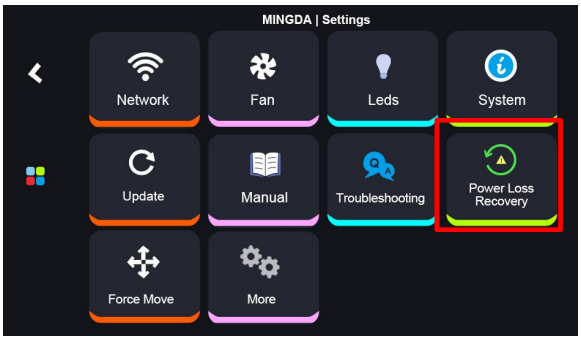
2. Select a tuning step of 0.02, then use Z+ (raise the nozzle) or Z– (lower the nozzle) to adjust the height until the extrusion lines are evenly laid and properly aligned.

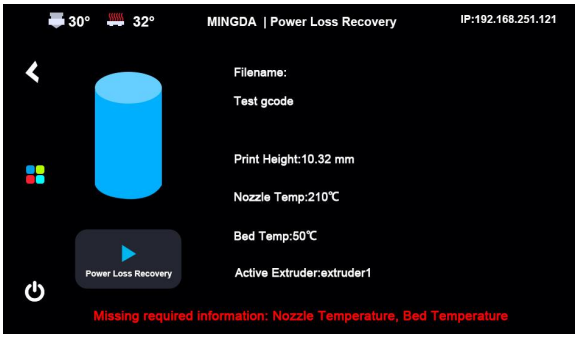
Proper fine-tuning helps achieve optimal first-layer adhesion and print quality.

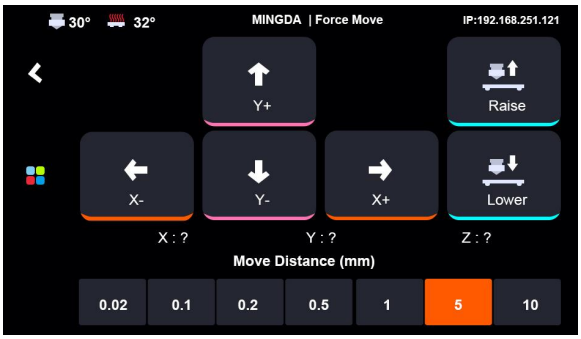
8. Resume Printing

1. Resume printing after power off









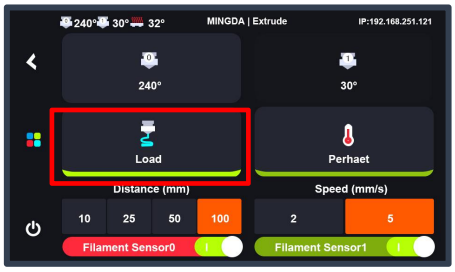
After power is restored, turn on the printer. It will perform a selfcheck. Then go to Settings > Power Loss Recovery to resume printing.

Note:

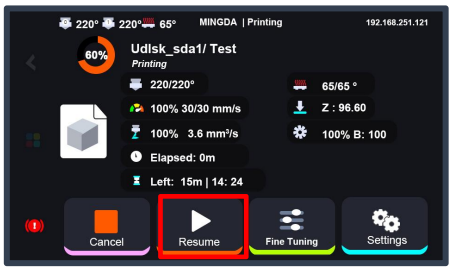
1. If the printed height is less than 1 mm, resume printing is not supported.
2. Prior to resuming the print, verify whether the nozzle has dropped. If so, use the "Force Move" function to position it approximately 0.1 mm above the model surface.
3. After resuming the print, closely monitor the printing process and adjust the Z-offset via fine-tuning if necessary.

2. Resume printing after filament run out

Printer will pause printing, and yellow light up.



Replace new filament into the extruder, click load till the filament was feed out.



Click "Resume" ▶, continue to print from the point of interruption.

9. Slicing Software Installation and Usage

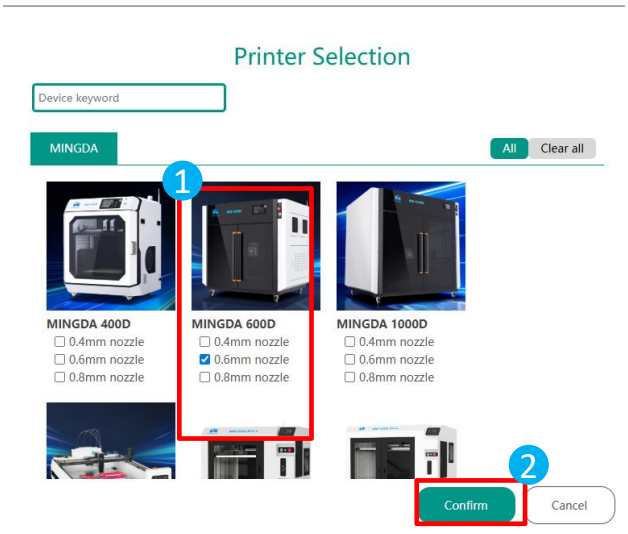
Installation:

Search "www.3dmingda.com/download" in any Browser.

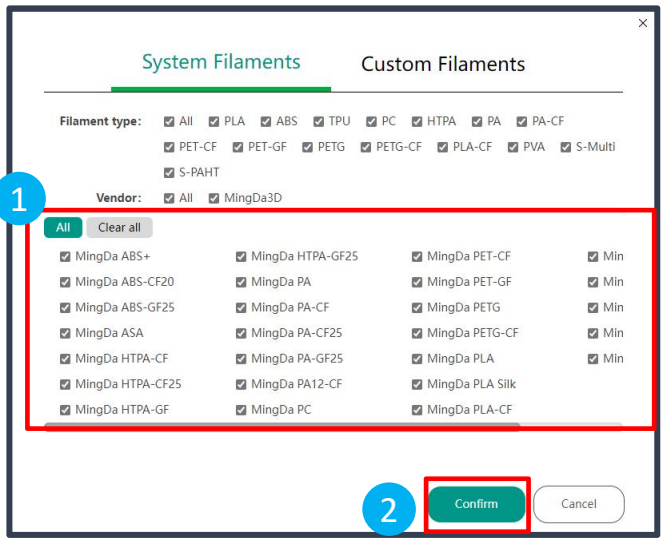
Download "**MINGDA OrcaSlicer**"

Configuration:

Upon the first run of Mingda OrcaSlicer, you will enter the configuration wizard.

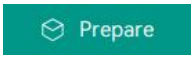


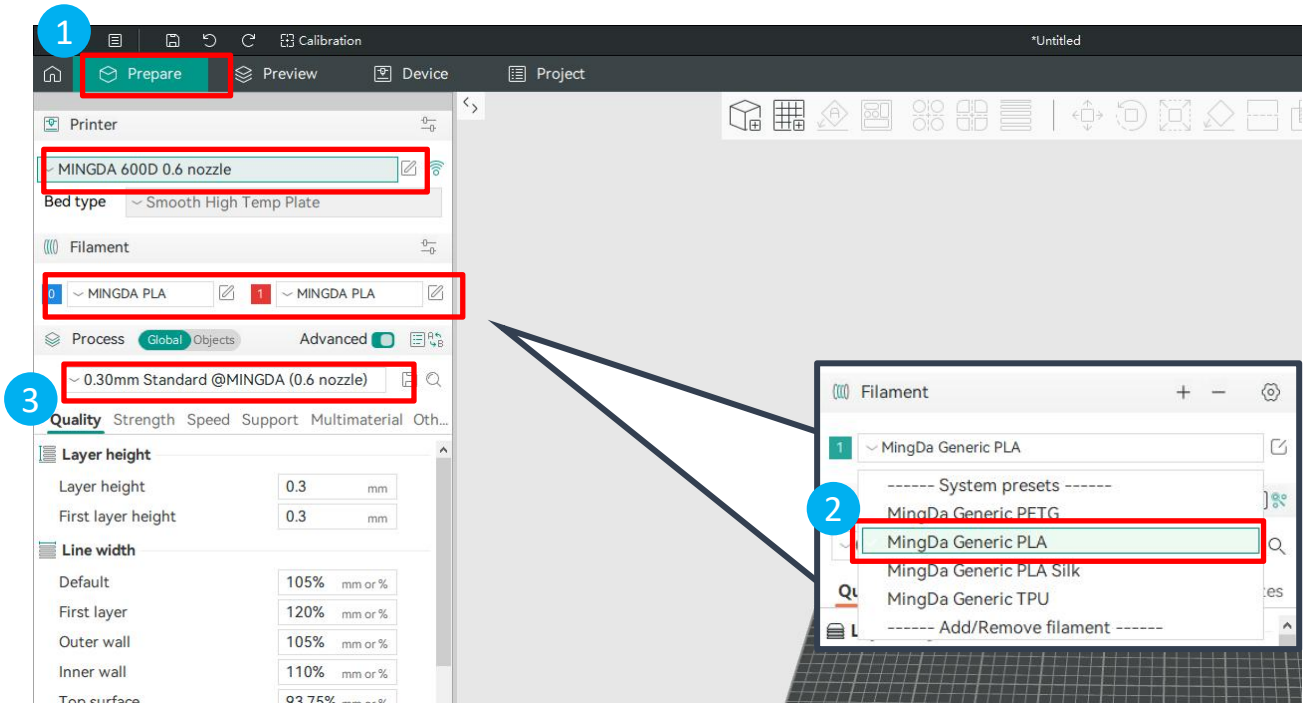
Choose the MD-600D model and click "**Confirm**."



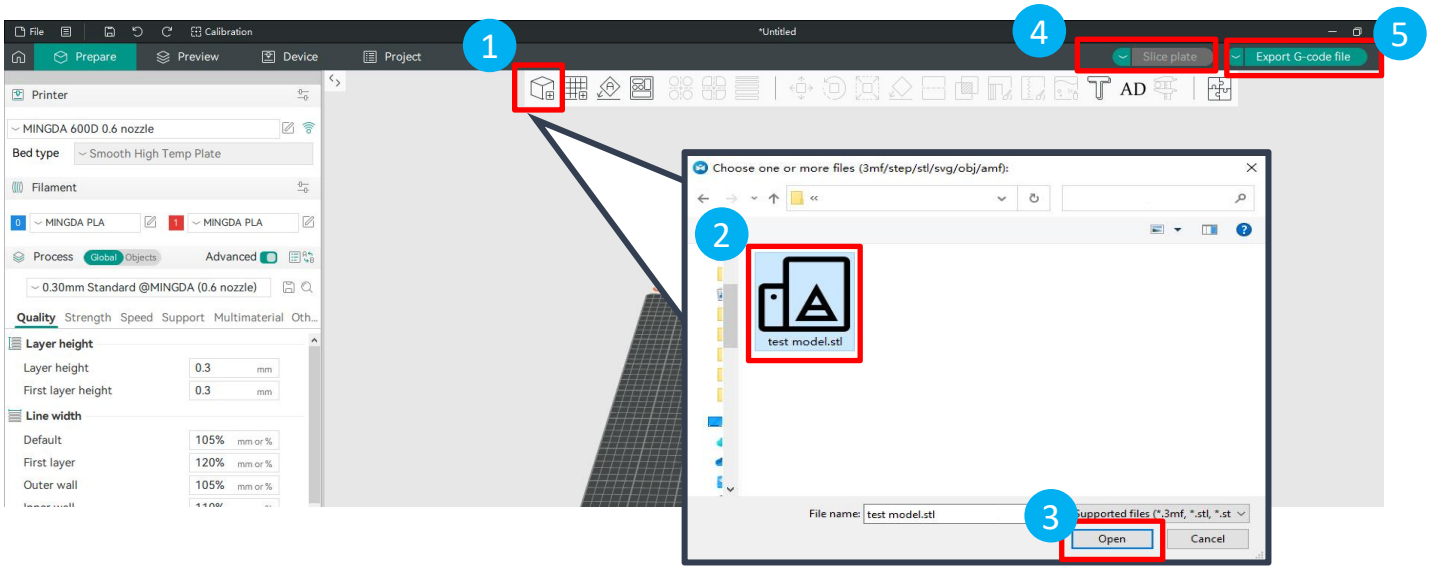
Select the desired filament type.

Usage

Click the "**Prepare**" 



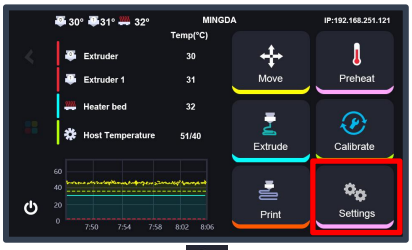
Select the printer model, filament type and printing parameter.



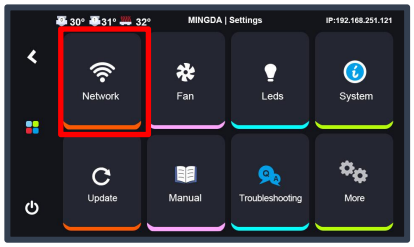
Click  , upload your STL in your slicer, adjust your model parameter, after finishing, click “Slice plate” to create the Gcode file.

LAN Printing (Recommend)

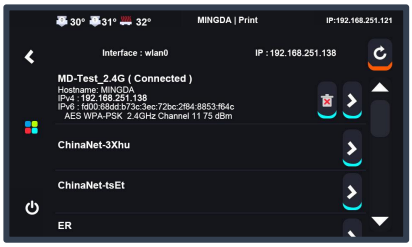
Ensure that the printer and the computer host are on the same local network.



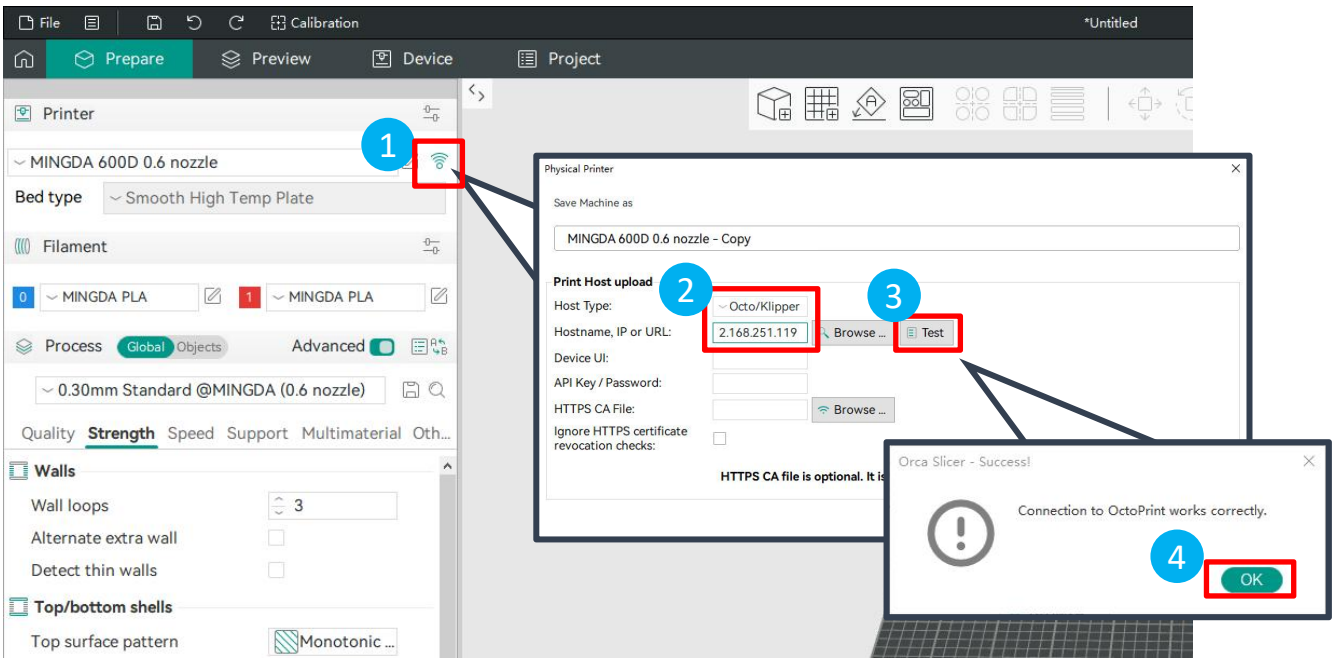
Click "**Settings**" to enter the settings page.



Click "**Network**" to enter the network page.



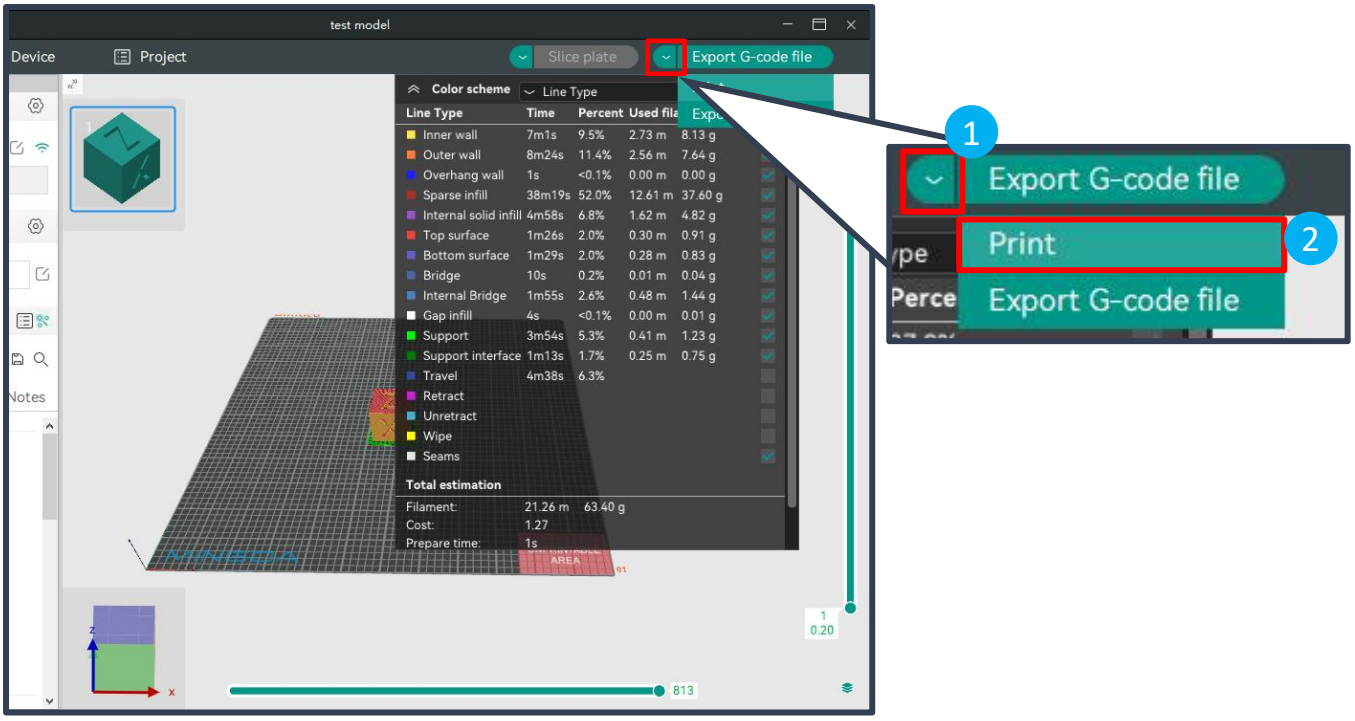
Check the printer's IP address.



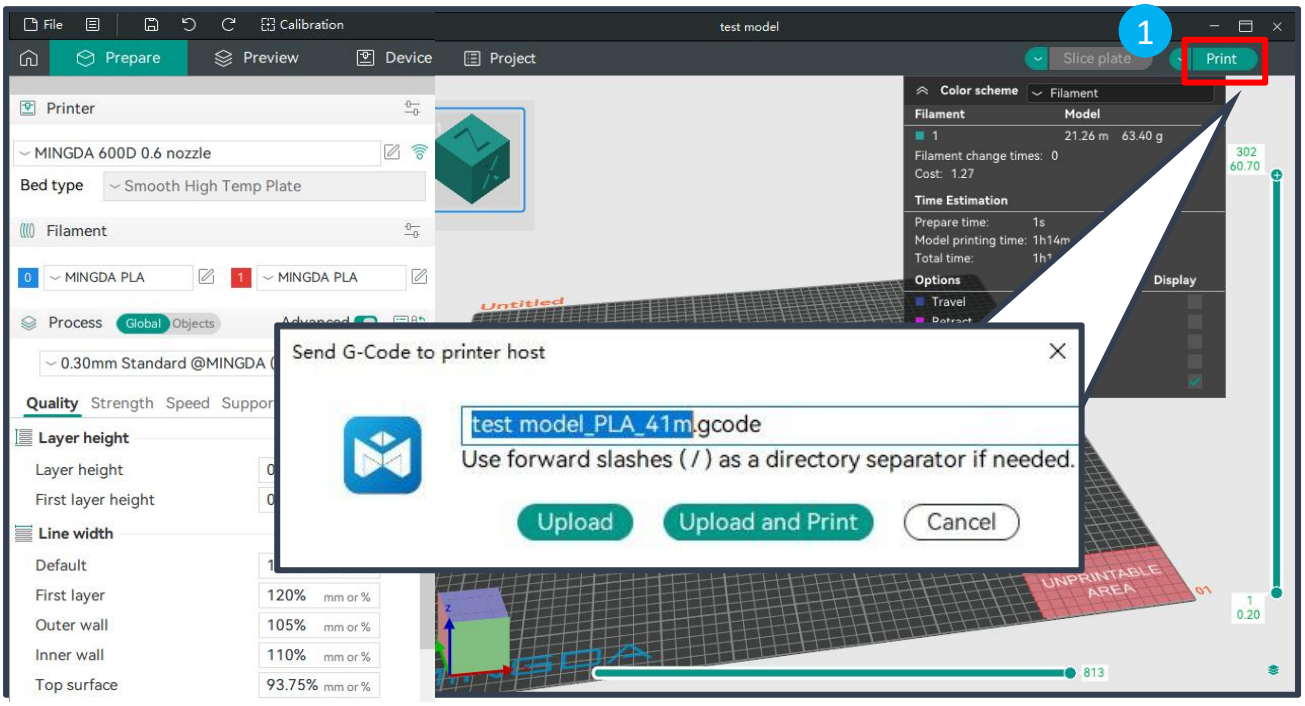
Click WIFI , select Host type as Octo/Klipper, enter the printer's IP address, and click "Test" and "OK"

LAN Printing

File Transfer:



Click the dropdown icon  in the top right corner , select "**Print**."

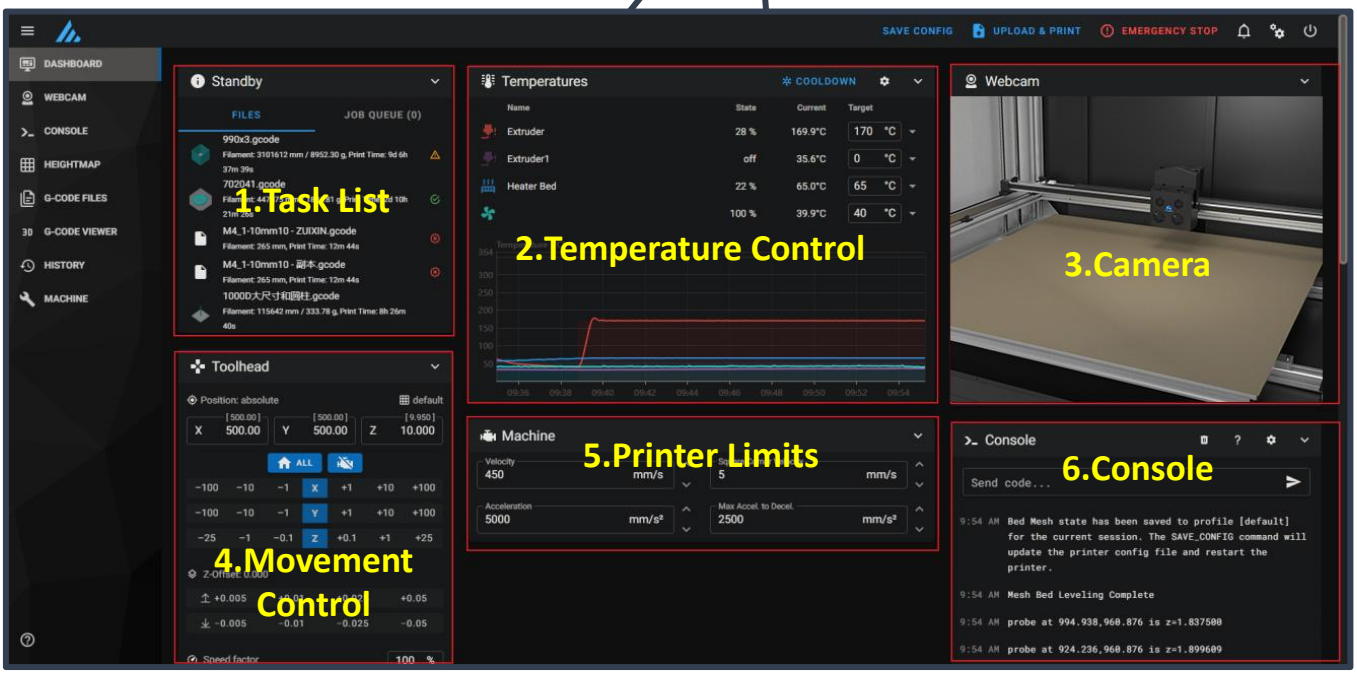


Click "**Print**" and choose "**Upload and Print**".

Device Connection

After successful connection, click on "Device" 

Enter the control interface below the diagram



1. **Task List:** Drag G-code files to this task list for printing.

2. **Temperature Control:** Displays machine temperature changes and allows pre-setting nozzle and bed temperatures.

3. **Camera:** Monitors the printing status.

4. **Movement Control:** Controls the movement of each axis and allows compensation settings after leveling.

5. **Printer Limits:** Controls the maximum acceleration of the printer, usually doesn't need to be changed.

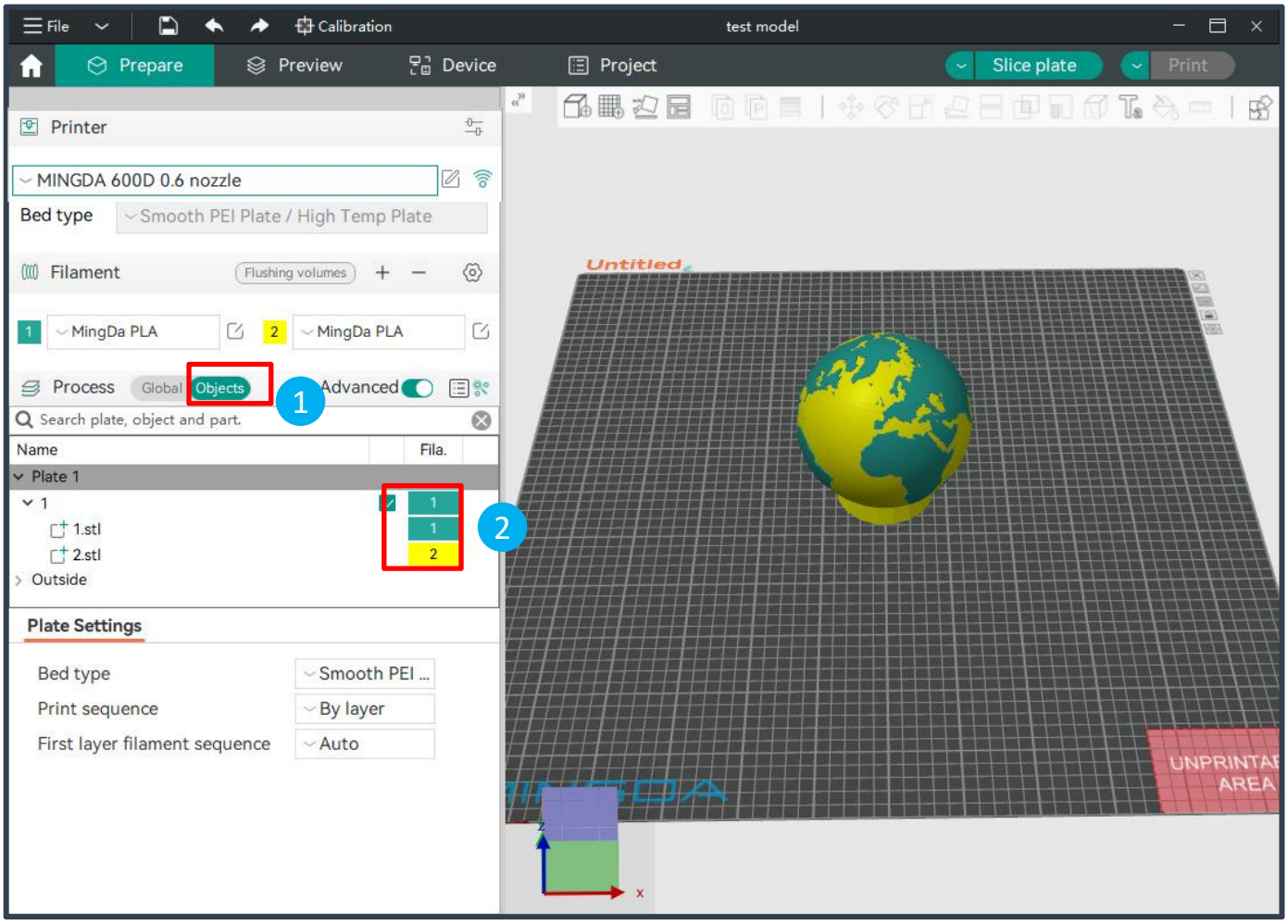
6. **Console:** Sends G-code commands to run the machine and displays error output.

Print Mode

Print Two Colors

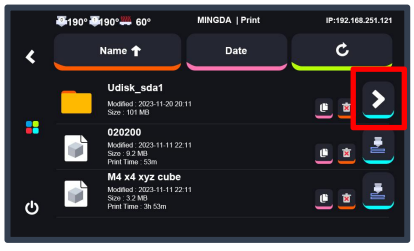
Printing size: 600 * 600 * 600mm

Selecting the [MingDa 600D 0.6 nozzle]

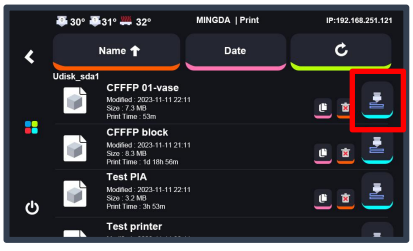


1. In the Process section, click to switch to the "Objects" option.
2. Click on the color box next to the STL file to select the desired filament.

In the printer interface:



Select "**Print**", Insert the U-disk.



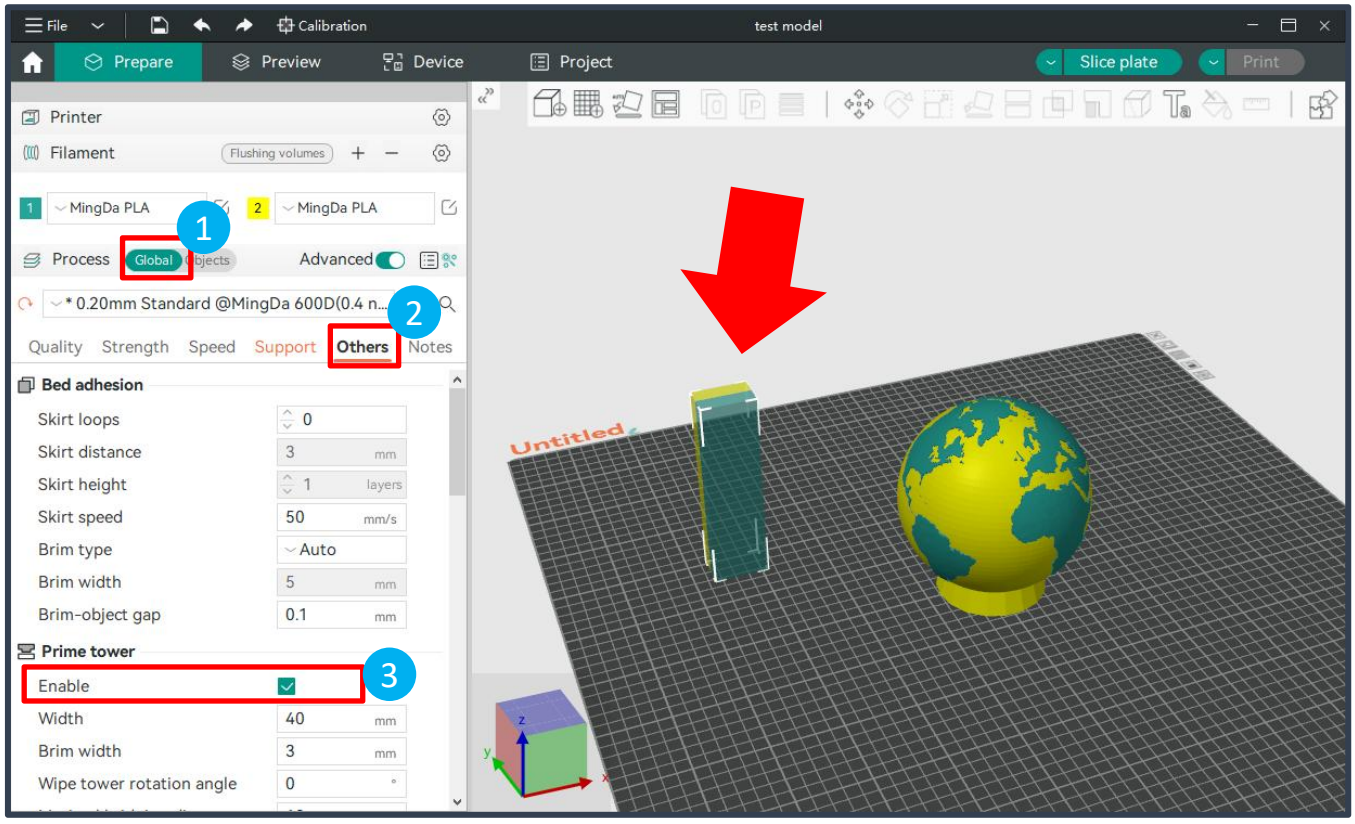
Choose the print file for printing.

Choose " **Print**".

Double extrusion: Start the Prime tower

Because there is always one printer in standby mode during the printing process, it is easy to cause defects such as wire drawing and material leakage. Prime tower can solve this problem, the extruder will print a prime tower before each layer printing. Any material leakage will be printed on the tower, effectively avoiding the phenomenon of material leakage when replacing the extruder.

If you want to print the following two modes, we recommend adding this option to your Gcode.



1. Select the "Global" section.
2. Select the "Others" section.
3. Check the "Enable" option in the "Prime tower" settings.

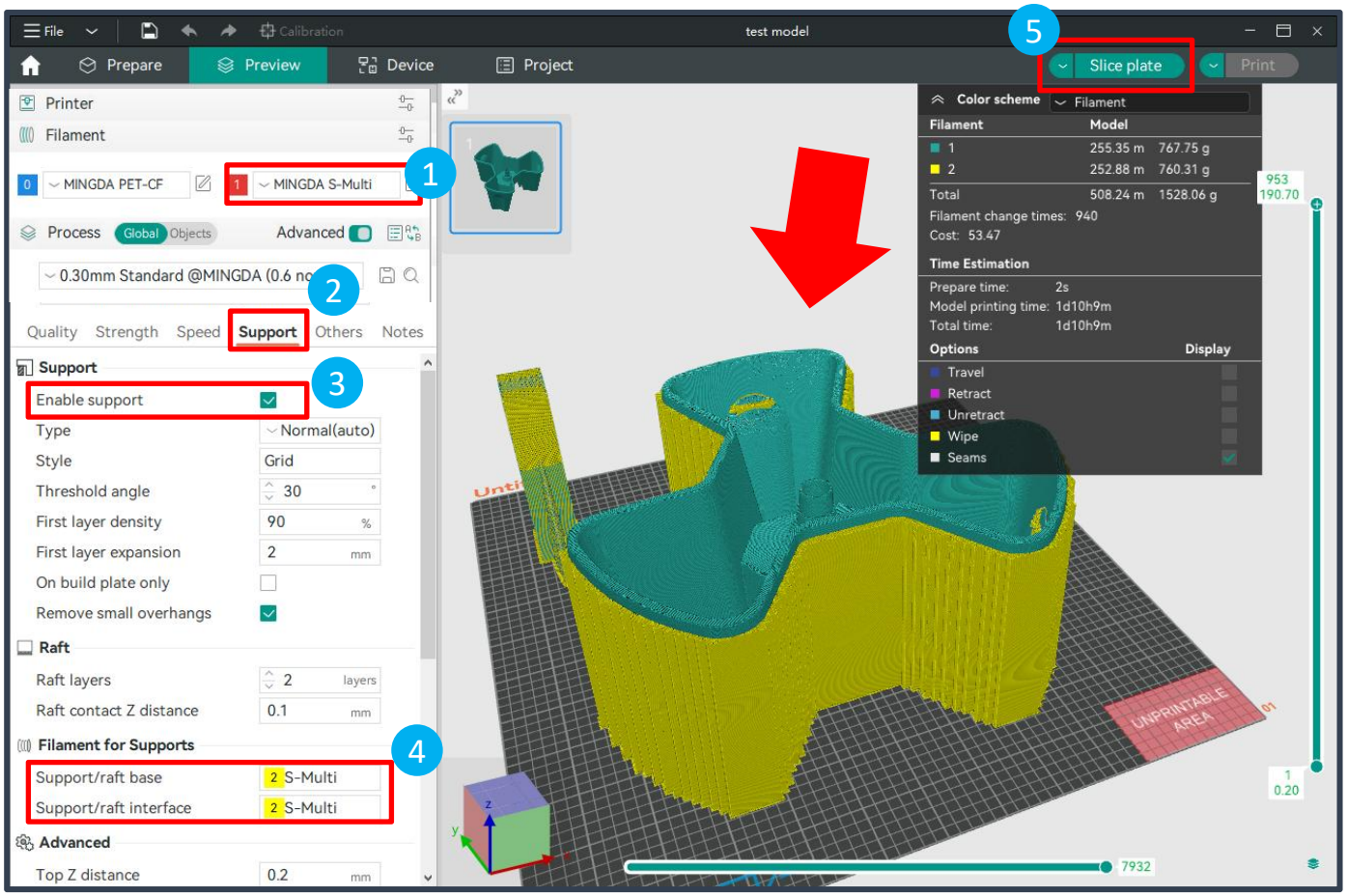
Note: The printing position of the Prime tower cannot coincide with the model

Print Support

we will take MD-600D as an example

Printing size: 600 * 600 * 600mm

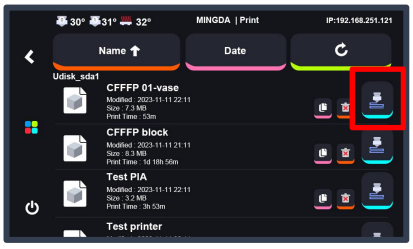
Selecting the [MingDa 600D 0.6 nozzle]



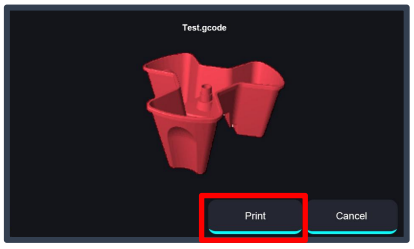
1. Choose and modify the filament information.
2. Then, select the "**Support**" section.
3. Check the "**Enable support**" option.
4. In the "**Filament for Supports**" option, choose the filament needed for supports.
5. Click "**Slice plate**" to preview.

In the printer interface:

Select "**Print**", Insert the U-disk.



Choose the print file for printing.



Choose " **Print**".

10. Maintenance and Care

Hotend:

If you print high-temperature materials or engineering materials for a long time, the inner wall of the hot end will gradually carbonize, which will ultimately affect the extruder out of the material is not smooth, if you find that the model has a fault or spitting material is not uniform, it may be this reason.

Replacing Filaments:

Timely replace filaments based on the type and actual usage. It is recommended to use filaments recommended by the manufacturer. Seal filament not in use for an extended period, as excessive exposure to moisture in the air can make the filament brittle.

PEI plate:

Please avoid touch the PEI bed by hand. It can be cleaned by dish soap with warm water and dry it by a clean cloth.

Regular Lubrication:

Once every 3 months. Home the printer first and grease it and move the Z+ to the top, then home the printer and grease again until the screw rods full with the grease, after that clean the excess grease. If there are noise during moving. You can grease the oil on the idler wheels when there are noise.

Software Updates:

Regularly update the printing software to improve print quality and efficiency.

11. Printing Tips for different filaments

1. Do not allow PET-GF, PET-CF, PA12-CF, HTPA-GF, HTPA-CF, or S-Multi to absorb moisture.

1. Dry the filament before printing as long as you are not sure if the filament is damp.
2. After drying the filament, please store it in filament cabinet with driers
3. If the printer will not be used for an extended period, please store the filament in a sealed container, preferably in a dry cabinet.

2. Avoid PLA-HF/PETG-HF clogged during printing

1.When printing PLA with a bed temperature above 45 °C, open the front door to prevent heat buildup and extrusion issues.
2.When printing PETG with a bed temperature around 70 °C, open the front door to prevent overheating that may cause extrusion issues or nozzle clogging.

3. To prevent warping, take caution when printing with ABS-HF, PA12-CF, HTPA-GF, HTPA-CF, ABS-GF25, ABS-CF20, HtPA-GF25, and HtPA-CF25.

If the model size is large and the fill rate is set high, such as 60% (the default value is 15%), warping occurs and it can be adjusted down appropriately. In addition, change the fill pattern to a spiral to reduce the risk of shrinkage. For some structural parts with high strength requirements, you can set 5 layers of walls and a fill rate of about 25%, and try to avoid using a fill rate of more than 50% to reduce the shrinkage tendency; for most non-structural parts with low strength requirements, you can directly select the default 2 layers of walls and a fill rate of 15%.

4. After printing with high-temperature materials such as PA-CF, PET-CF, or PET-GF, and before switching to low-temperature, lower-hardness materials like PLA or PETG, it is recommended to follow these steps:

1. Set the nozzle temperature to 250–300 °C.
2. Manually unload the remaining high-temperature filament.
3. Load the new low-temperature filament and manually extrude it.
4. Continue extrusion until all residual high-temp material is flushed out and the new filament flows smoothly.
5. Lower the nozzle temperature to 220–240 °C, and continue extruding while the nozzle cools down to ensure smooth flow of the new filament.

If extrusion fails during the flushing process, increase the nozzle temperature and consider using a cleaning needle to clear the nozzle

MINGDA

Shenzhen MINGDA Technology Co., Ltd

Official Website

Customer Support

Email: support@3dmingda.com

www.3dmingda.com

Address: No. 19, Dahe Industrial Zone, Longhua District, Shenzhen, 518110, Guangdong Province, China.