



 **User Manual**

VYPER

Thank you for choosing ANYCUBIC products.

Maybe you are familiar with 3D printing technology or have purchased ANYCUBIC printers before, but we still highly recommend that you read this manual carefully. The installation techniques and precautions in this manual can help you avoid any unnecessary damage or frustration.

More information please refer to :

1. ANYCUBIC Website: <https://www.anycubic.com>

ANYCUBIC website provides software, tutorials, models, after-sale service, etc.

Please visit the website for technical support and we are pleased to answer the questions or solve all the problems for you!

2. Facebook page and Youtube channel links are shown below.



ANYCUBIC Website



Facebook page



Youtube channel

Team ANYCUBIC

Copyrighted by "Shenzhen Anycubic Technology Co., Ltd ", all rights reserved.

Contents

Safety Instructions-----	4
Technical Specification-----	5
Packing List-----	7
Product Overview-----	8
Menu Directory-----	9
Installation-----	14
Leveling -----	22
Loading Filament-----	24
Printing Models-----	27
Introduction to Slicing -----	31
Software -----	46
Resume from-Outage-----	47
Maintenance -----	48
Troubleshooting	

Safety Instructions

To avoid any unnecessary damage to the printer or individual injury, please always follow the safety instructions when assembling and using the machine.



Please contact our customer service first if you have any problem after receiving the products.



Be cautious when using the scraper. Never direct the scraper towards your hand.



In case of emergency, please immediately cut off the power of ANYCUBIC 3D printer and contact the technical support.



ANYCUBIC 3D printer includes high speed running parts that can cause injury.



It is recommended to use protection glasses when cleaning/sanding the printed models to avoid small particles in eyes.



Keep the ANYCUBIC 3D printer and its accessories out of the reach of children.



Vapors or fumes may be irritating at operating temperature.



Always use the ANYCUBIC 3D printer in an open and well ventilated area.



ANYCUBIC 3D printer must not be exposed to water or rain.

ANYCUBIC 3D printer is designed to be used within ambient temperature ranging 8°C-40°C, and humidity ranging 20%-50%.



Working outside those limits may result in low quality printing.

Do not disassemble ANYCUBIC 3D printer, please contact technical support if you have any question.



Technical Specification

Printing

Technology	FDM (Fused Deposition Modeling)
Build Size	245 mm(L) × 245 mm(W) × 260 mm(H)
Print Accuracy	±0.1 mm
Positioning Accuracy	X / Y / Z 0.0125 / 0.0125 / 0.002 mm
Extruder Quantity	Single
Nozzle Diameter	0.4 mm
Print Speed	20~100 mm/s (suggested 80 mm/s)
Supported Materials	ABS, PLA, TPU, PETG, Wood, etc

Temperature

Ambient Operating Temperature	8 °C - 40 °C
Operational Extruder Temperature	Max 260 °C
Operational Print Bed Temperature	Max 110 °C

Software

Slicing Software	Cura
Software Input Formats	.STL, .OBJ, .AMF
Software Output Formats	GCode
Connectivity	Memory card; Data cable (expert users only)

Technical Specification

Electrical

Power Input 110 V / 220 V AC, 50 / 60 Hz

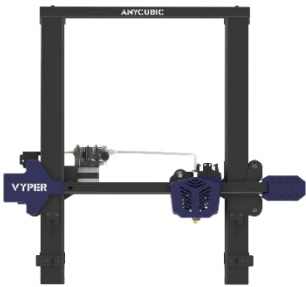
Output rating 24V/14.6A (350W)

Physical Dimensions

Printer Dimensions 508 mm(L) × 457 mm(W) × 516 mm(H)

Net Weight ~10 kg

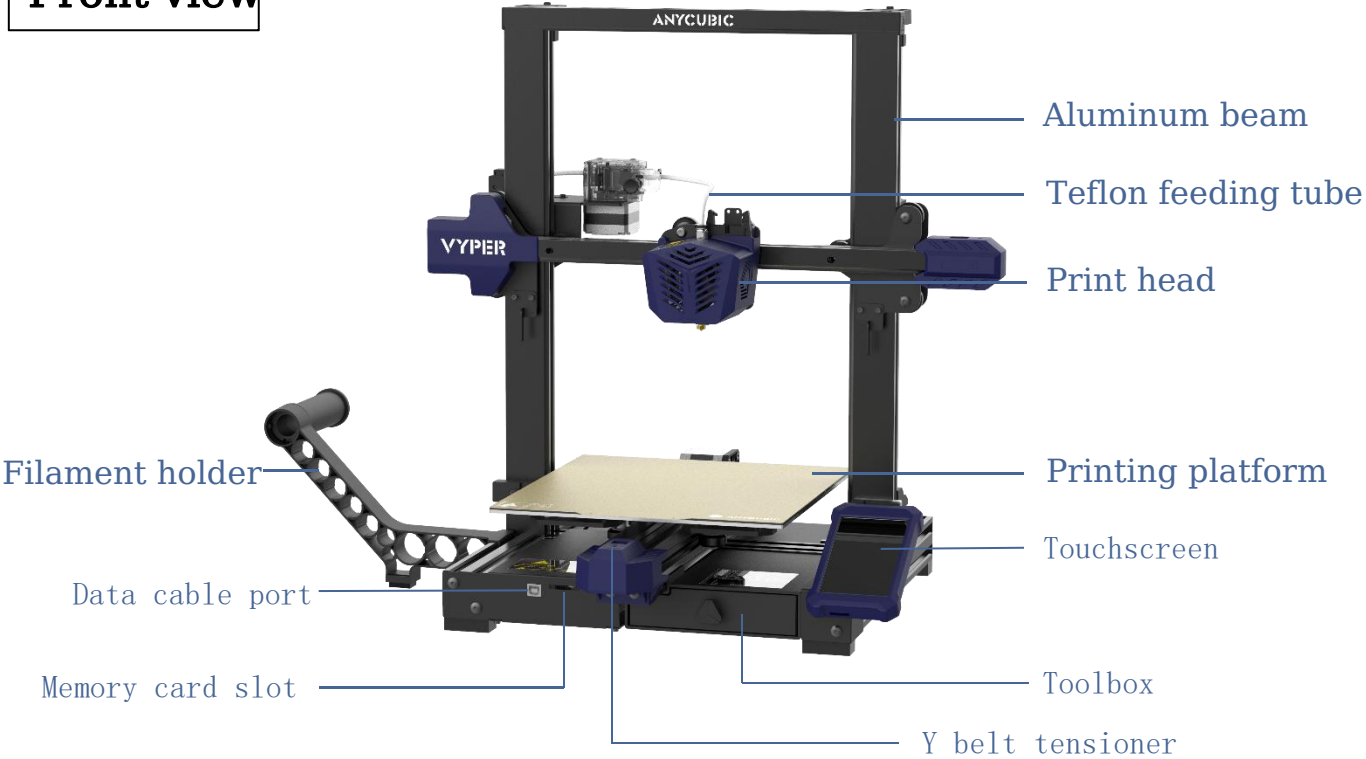
Packing List

			
Frame	Base	Washers (4PCS) M5 x 45 Screws (4PCS)	
			
Touchscreen	M5 x 14 Screw (1PCS) M5 x 6 Screws (2PCS)	Filament Holder	Nozzle
			
Power Cord	Data Cable	Card Reader	Memory Card
			
Filament (Random Color)	Assembly Instruction		

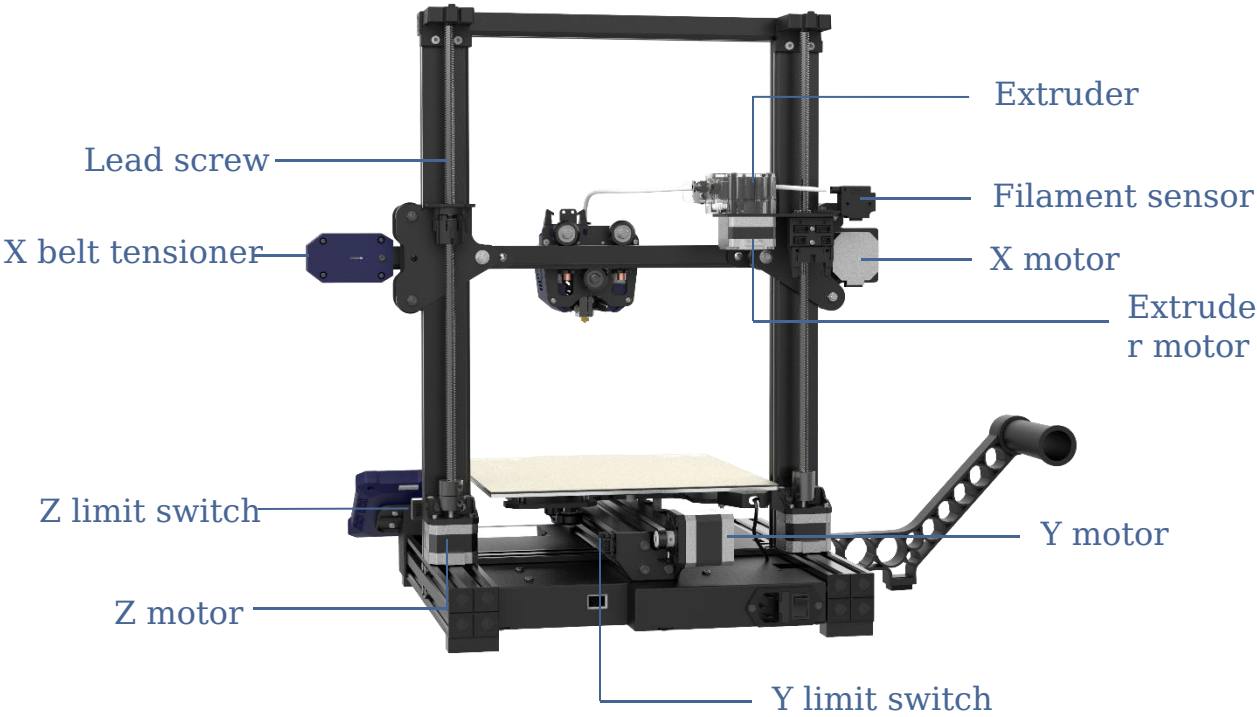
Tips: Some accessories are placed in the toolbox on the right front of the machine base (see the position of toolbox on the next page). Please take out all the accessories from the toolbox when you check the accessories list.

Product Overview

Front view



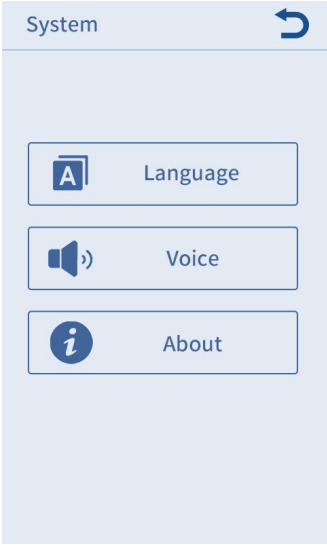
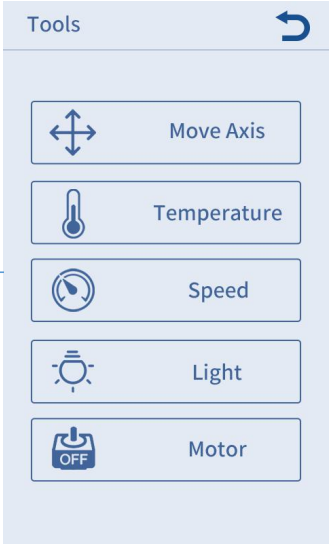
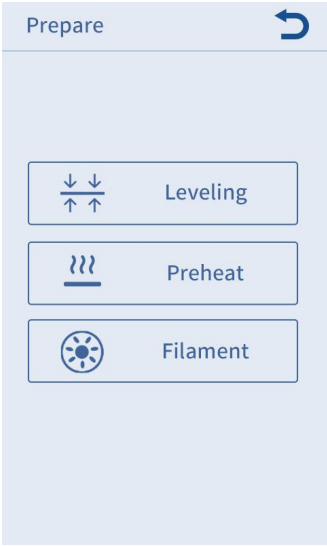
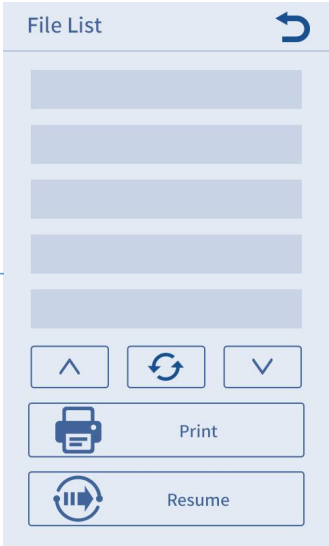
Back view



Menu Directory



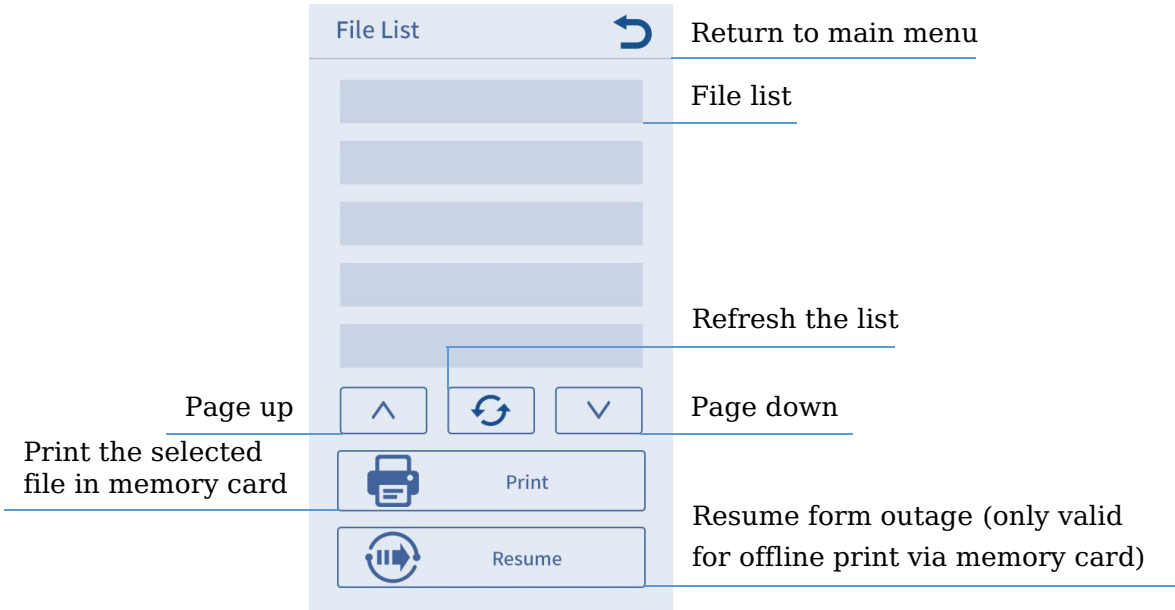
Main menu



Menu Directory

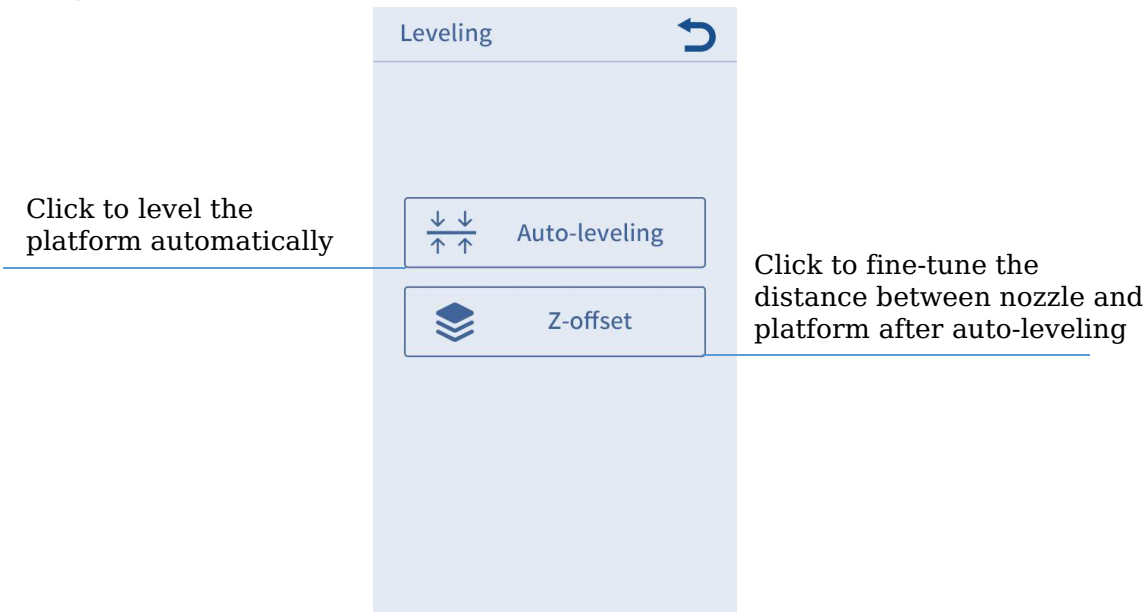
Print

File List:



Prepare

Leveling:



Note: In most cases, there is no need to perform “Z-offset” after auto-leveling.

Menu Directory

Preheat:

Preheat

Display current / target temp of nozzle

Nozzle Temp

40/190

Display current / target temp of bed

Bed Temp

32/60

PLA

Preheat PLA

Click to preheat PLA

ABS

Preheat ABS

Click to preheat PLA

Filament:

Filament

Click to automatically heat the print head to target temp, and click again to feed the filament

E0 Temp

23/230

Display current / target temp of extrusion

Click to automatically heat the print head to target temp, and click again to retract the filament

Filament In

Filament Remove

Stop filament feeding or retracting

Stop

Menu Directory

Tools

Move Axis:

Move Axis

Unit mm

0.1

1.0

10

▲Y

▲Z

◀X

▶X

▼Y

▼Z

Y

X

Z

Home All

Set the step distance of motor

Move the platform forward

Move print head to the left

Move the platform back

Move the print head up

Move print head to the right

Move the print head down

Click to home all

Temperature:

Temperature

Current Bed Temp

23

Target Bed Temp

60

–

+

Current EO Temp

35

Target EO Temp

190

–

+

Cooling

OK

Display the current temp of bed

Click to set bed temp (30-110°C)

Reduce bed temp

Raise bed temp

Display the current temp of extrusion

Click to set extrusion temp (170-260°C)

Reduce extrusion temp

Raise extrusion temp

Cut off the power to hot-end and heated bed (only valid when machine is not printing)

Menu Directory

Speed:

Speed

Current Fan Speed

0

Click to set fan speed(0-100%)

Decrease fan speed

-

0

+

Target Fan Speed

0

Increase fan speed

Current Print Rate

100

Click to set print rate(0-100%)

Decrease print rate

-

100

+

Target Print Rate

100

Increase print rate

OK

Light: Turn on/off the light of the print head

Motor: Disable all motors (only valid when machine is not printing).

System

Language: Change language (English / Chinese).

Voice: Turn on/off the screen sound.

About: Display the information about the product.

Installation

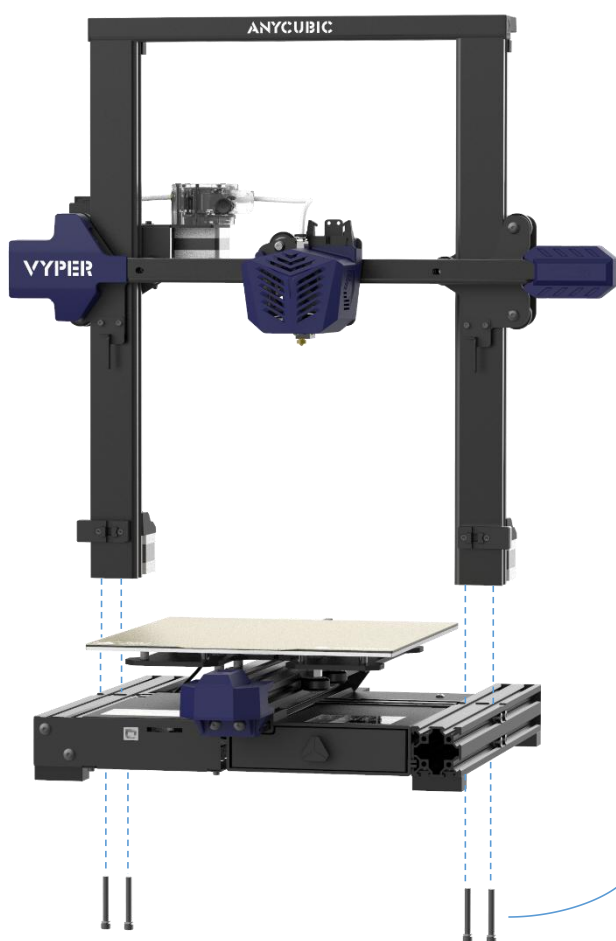
1. Be cautious during assembly as some parts may have sharp edges.
2. It is suggested to use a flat desktop and place the parts in an orderly manner for quick assembly.
3. The color of some parts may be different from what in the manual, but the assembly is the same.
4. Firmware has been pre-uploaded to the motherboard. After completing the assembly, please level the platform and load the filament then you could start the first test print.

Note: Every unit of the printer has been inspected and tested for printing. Therefore, in some cases, there might be very small marks left on the print head or on the heated bed. Those will not affect the printing quality and those mean the printer has been tested for the quality. The aluminum beam has slight scratches or slight unevenness on the platform, which is normal if it does not affect normal printing. Thank you very much for your kind understanding.

Installation

* Take tools from the toolbox on the front right side of the machine base for assembly.

1. Install frame.

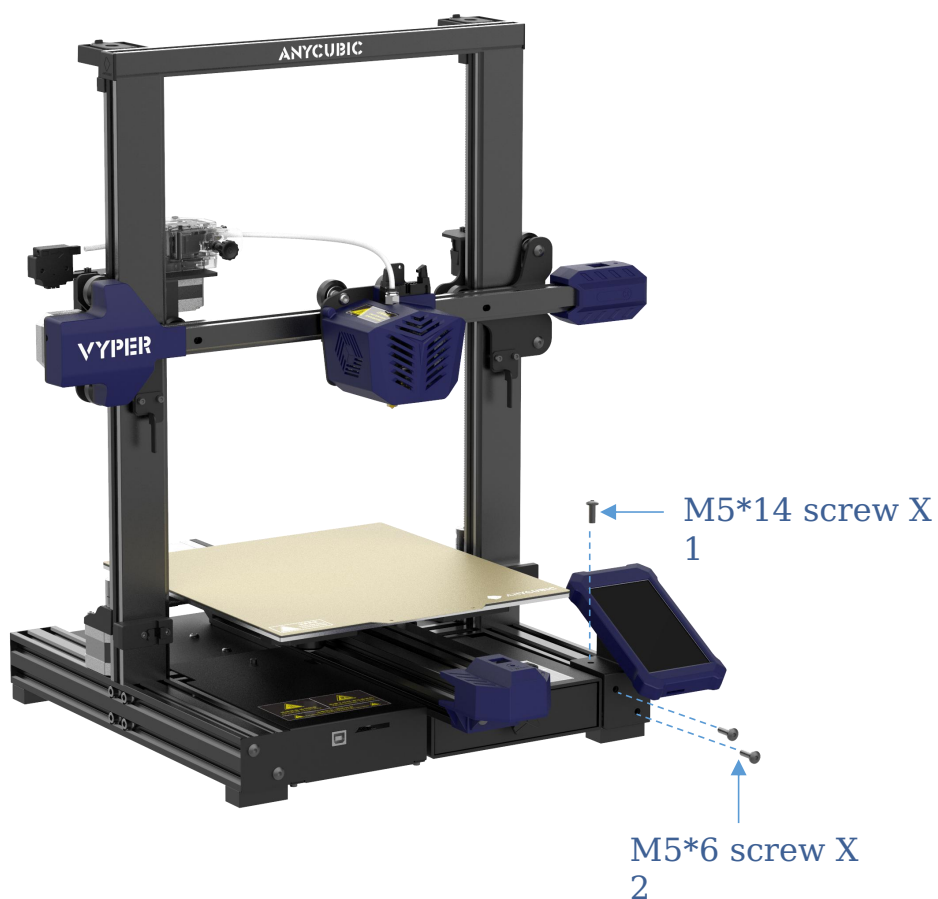


Z aluminum beams must align with the groove of the base aluminum beam.



Installation

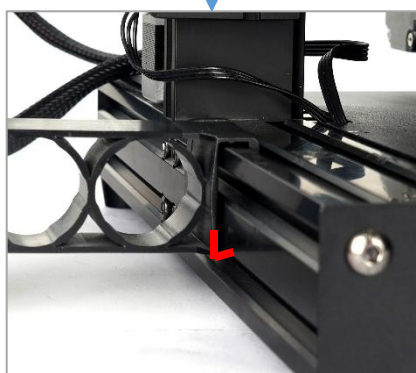
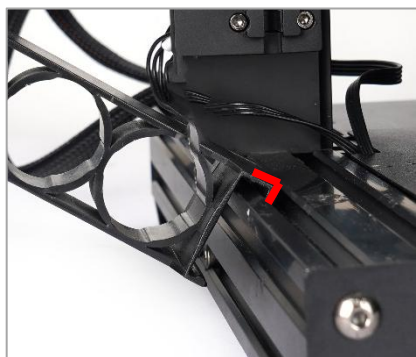
2. Install touchscreen.



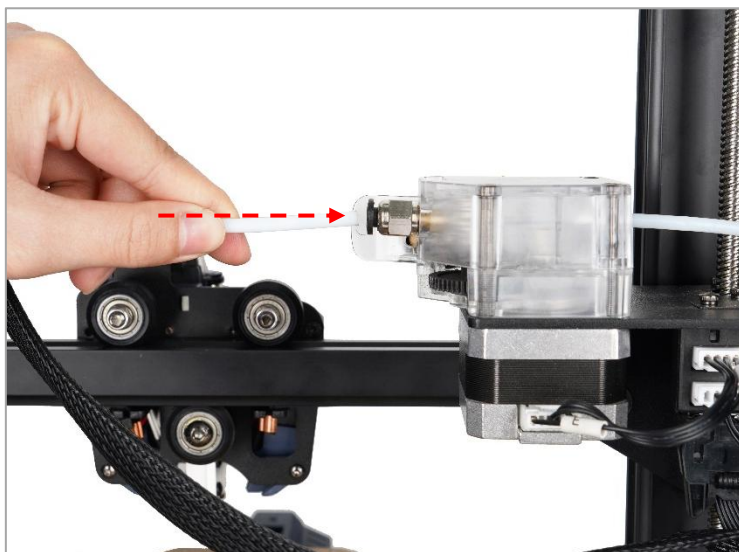
3. Install the filament holder.



Installation

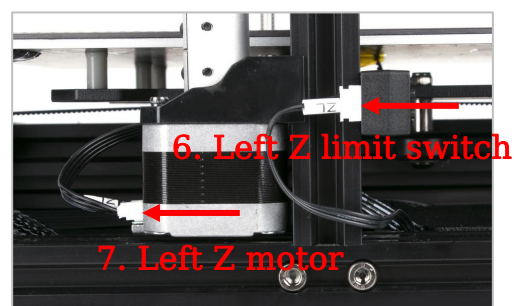
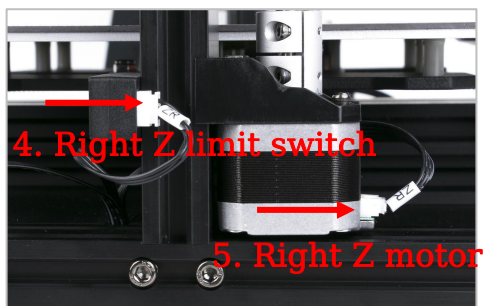
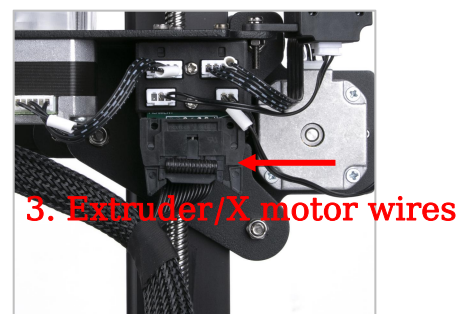
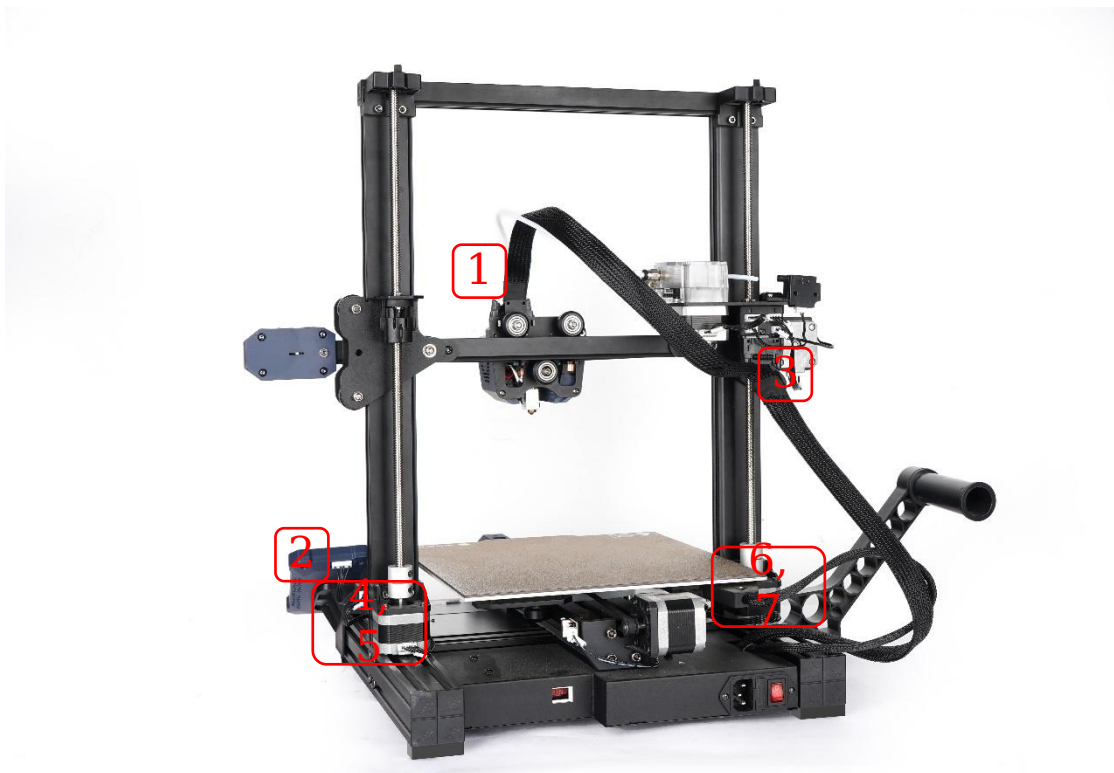


4. Insert the teflon tubing into the extruder.



Installation

5. Wiring: connect all these wires to their corresponding ports by the label respectively.



Installation

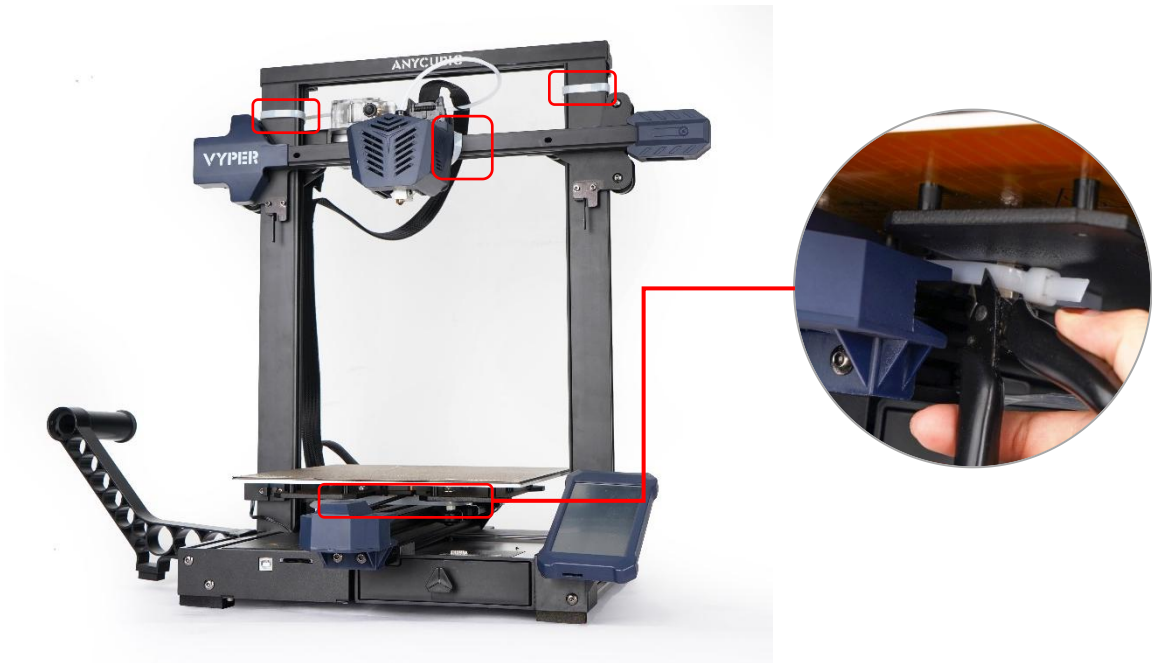
6. Use zip ties to fasten the print head wire harness and the Teflon tube.



Installation

Check before using

1. Make sure that the 4 zip ties shown below have been cut off before operating the machine.

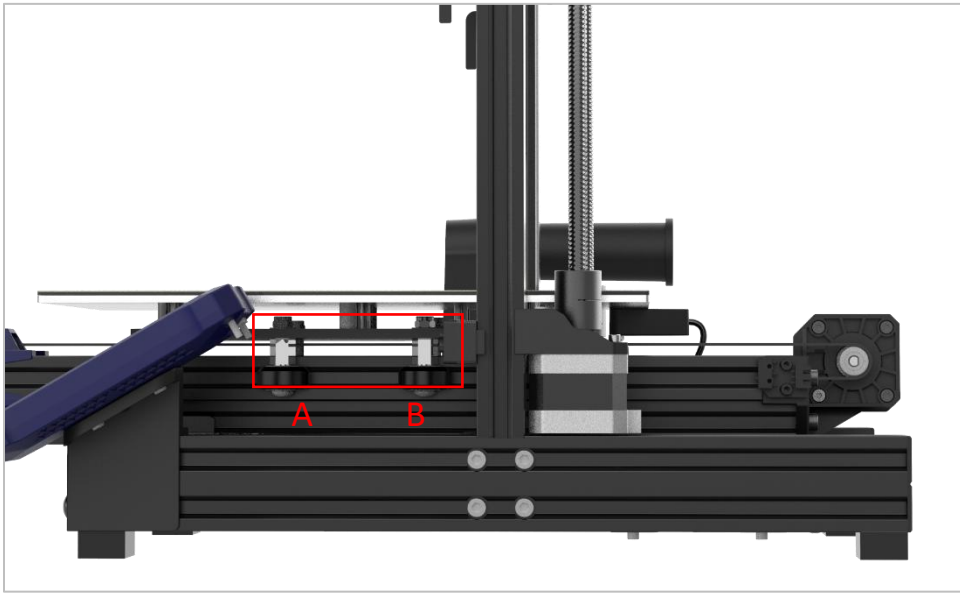


2. D-shape wheels of the base may be loosened due to transportation, please check it after unpacking.

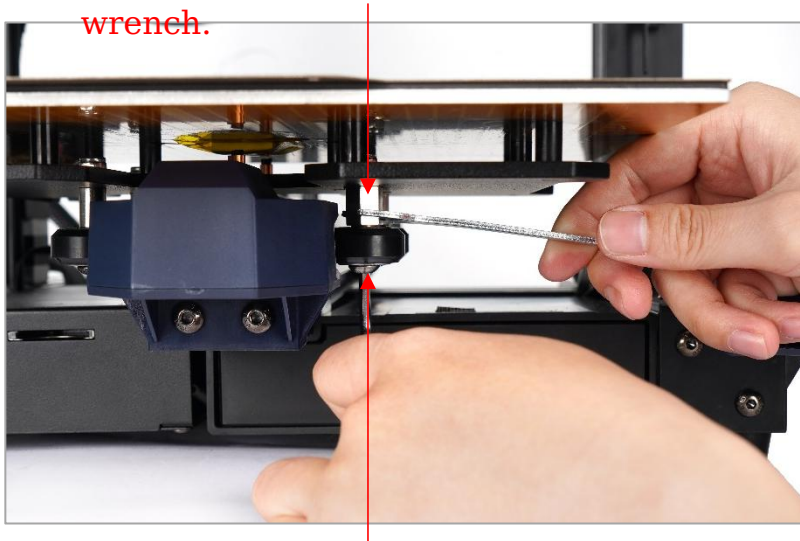
Place the base on a flat desktop. Shake the platform to confirm whether the printing platform is shaking up and down. If so, please tighten the hexagonal pillars of the D-shape wheels on the right under the platform.

- Move the platform to the front for tightening the pillar A.
- Move the platform to the rear for tightening the pillar B.

Installation



② Tighten the hexagonal pillar clockwise with a 10mm open end wrench.



① Fix the nut with a hex key.

Leveling

Leveling the printing platform is a key step in 3D printing, which determines to a large extent whether the model can be reliably adhered to the printing platform. If the distance between the nozzle and the printing platform is too wide, the model will not adhere to the printing platform properly. If the nozzle is too close to the printing platform, it will hinder material extrusion and scratch the platform. Once leveled, it is not necessary to level every time before each printing.

Vyper supports auto-leveling, which is easy to level the platform.

* Before leveling, please ensure that the platform and nozzle are clean and free of foreign matters to avoid affecting the leveling effect.

1. Select the correct voltage mode according to your local voltage ratings (~110V or ~220V) before plugging in. The red switch is inside the power supply casing and 220V is a default setting. A hex key can be used to move the switch. Finally, double check the wirings, plug in the power cord, and power on the printer.

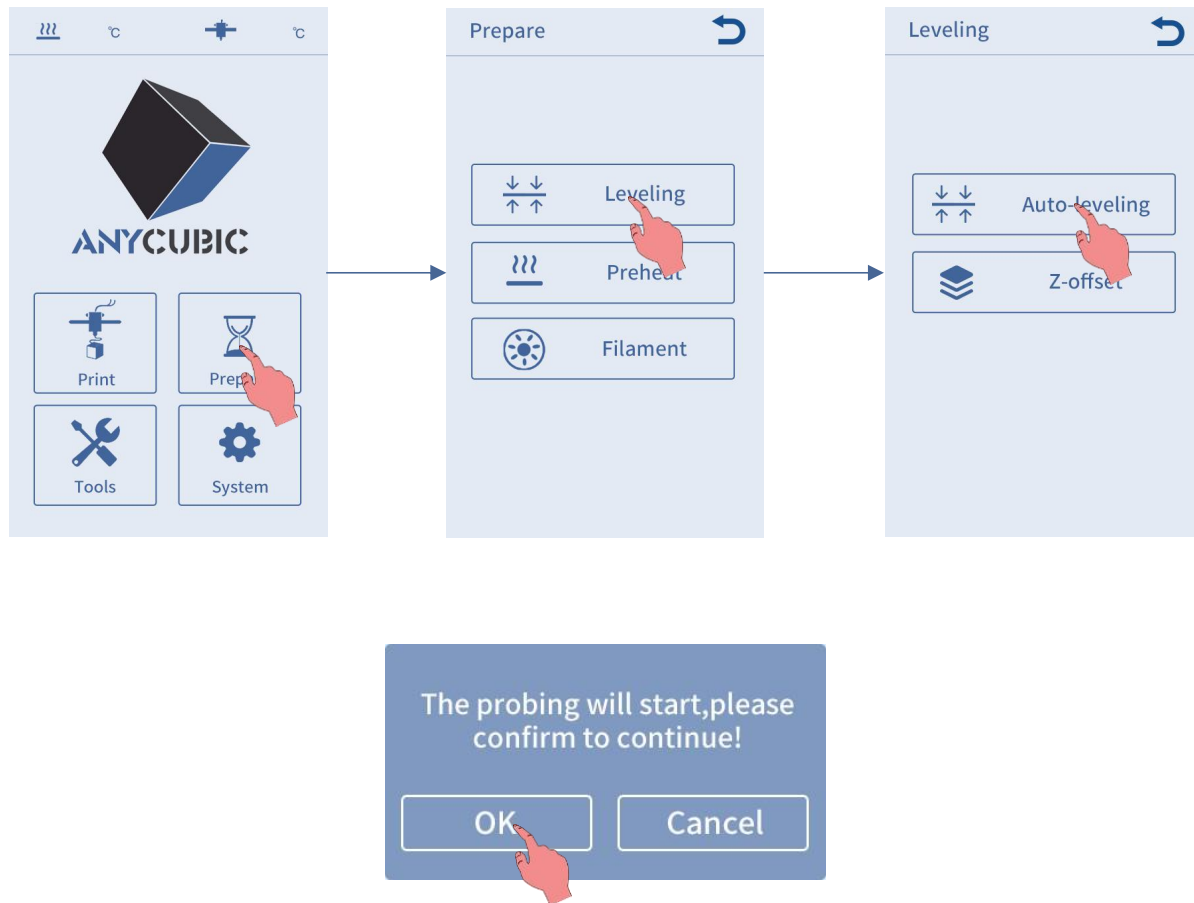


In some cases, 220V labeled as "230" , 110V labeled as "115"

Leveling

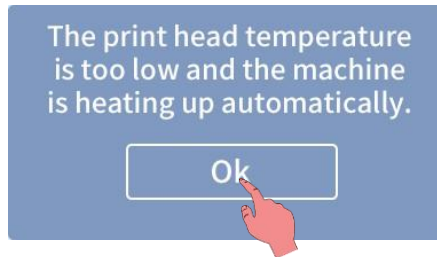
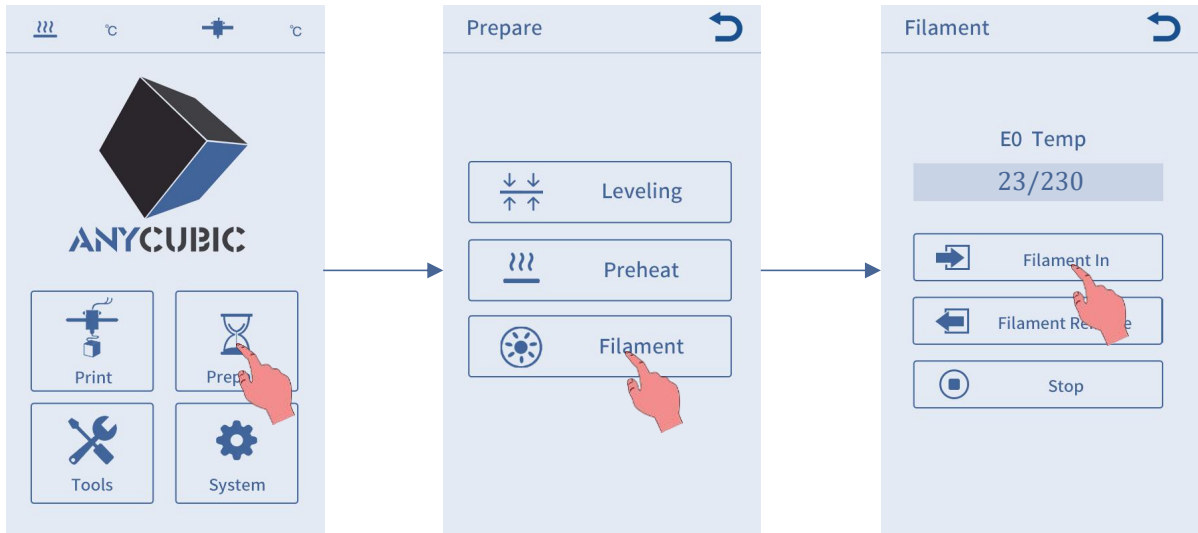
2. Return to main menu, then click **Prepare** → **Leveling** → **Auto-leveling**. The confirm interface will pop up, click “OK” and the print head will automatically probe the spots on the platform.

*** The print head will heat up before probing the platform. Please do not touch the nozzle to prevent scalding.**



Loading Filament

1. Return to main menu, click **Prepare** → **Filament** → **Filament In**, and the heating up interface will pop up, click “OK”.



2. Straighten the end of filament, and place the filament on the filament holder (Please note the feeding direction of filament). Then pass the filament through filament sensor, and insert it to the extruder till the gear. Now, wait for the nozzle reaches to the target temperature as shown on the display screen.

Please straighten the end of filament before insert it.



Loading Filament

Note: During feeding, if the melted filament is not smooth or too thin, please adjust the extrusion force by rotating the knob as shown below.



If the melted filament is not smooth, please increase the extrusion force by rotating clockwise.



If the melted filament is too thin, please reduce the extrusion force by rotating counterclockwise.



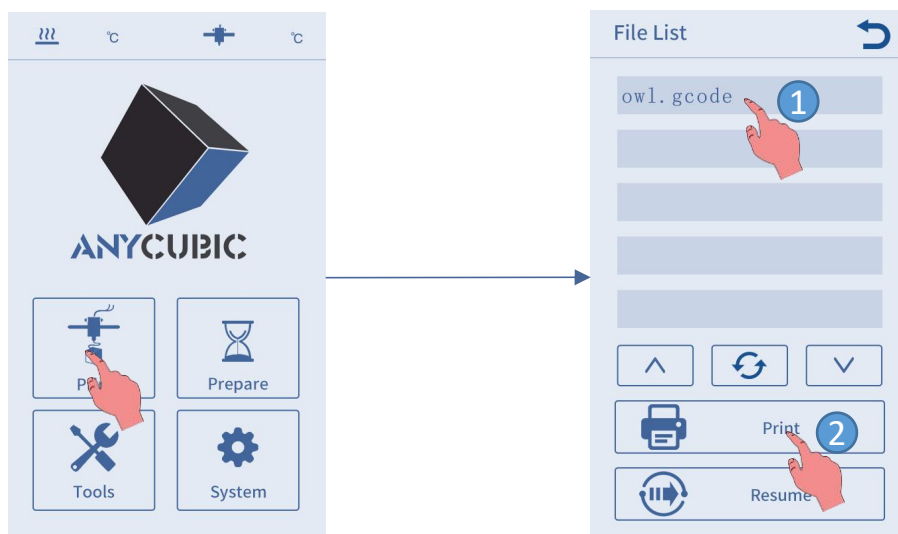
Note: The better distance for extruding filament is about 18 mm. You may adjust the distance of the screw for the different filament.

Printing Models

After the leveling and filament feeding are completed, the printing can be started. A printable file "owl.gcode" has been preloaded in the memory card and the users can print it directly.

* In order to let the printer better recognize the GCode file in the memory card, you need to back up all the files in the memory card to the computer, and keep the memory card only for the GCode files. Please save all the GCode files in root directory of the memory card.

1. Insert the memory card into the memory card slot, and choose the "owl.gcode" file to print. (owl, author: etotheipi, www.thingiverse.com)

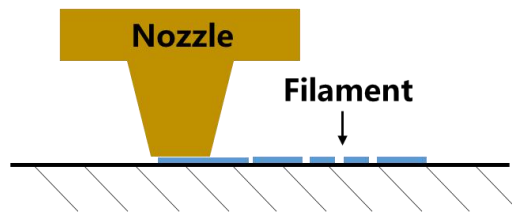


Printing Models

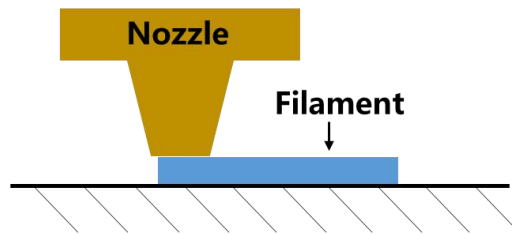
* Before printing, the nozzle only start to heat up after the heated bed reaches the target temperature. After the nozzle and heated bed are heated to the target temperature, the printer will start printing.

2. There might be three kinds of results for the first or second layer of the test print:

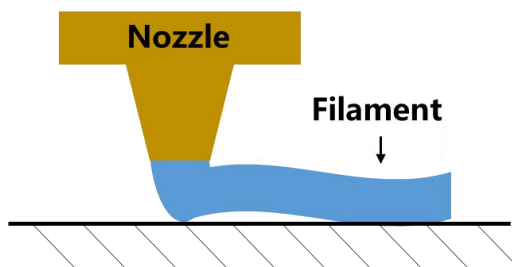
- ① Nozzle is too close to the platform, resulting in insufficient extrusion of filament.



- ② Proper distance between nozzle and platform can achieve the best printing effect.



- ③ Nozzle is too high and the gap is large, which lead to poor adhesion of filament to the platform.



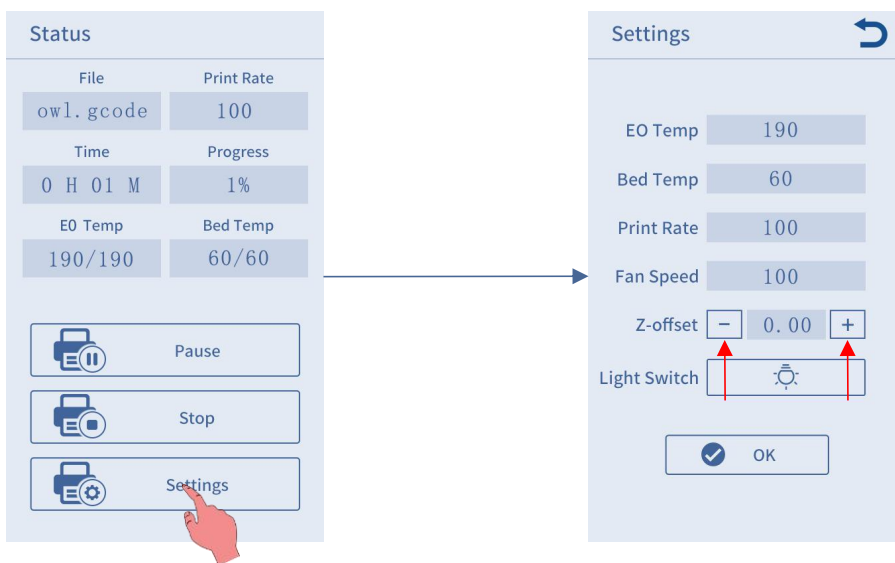
Printing Models

The printing effect ① and ③ can be improved by adjust Z-offset during printing.

(* In most cases, there is no need to adjust Z-offset after auto-leveling.)

How to adjust the Z-offset:

Click **Settings** on printing status interface, then click “+” or “—” of the Z-offset to adjust the distance between nozzle and printing platform and observe the printing effect.



—: Decrease the distance between nozzle and printing platform.

+: Increase the distance between nozzle and printing platform.

* After adjusting the offset setting, click "OK" on the settings interface, and the offset will be effective for subsequent printing; if you only click the Return button without clicking "OK", the offset will only be effective for this printing.

Printing Models

Please adjust the Z-offset patiently until the effect likes ②. If the printing effect is still not good after multiple adjustments, please stop printing and level the platform again.

3. The nozzle and heated bed are still at a high temperature when printing is finished. Make sure to wait for nozzle and heated bed to cool down before removing the model from the printing platform.

Note: When the ambient temperature is higher than 33° C, it is recommended to change the slicing parameter "wall thickness" of the model to 1.6mm.

* Tip: To eject the memory card, please press it.

Introduction to slicing software

3D printer reads GCode file and prints. It is necessary to convert 3D files (such as stl file) into GCode files for machine to recognize. Software that convert 3D files into GCode files is called slicing software.

Introduction of slicing software:

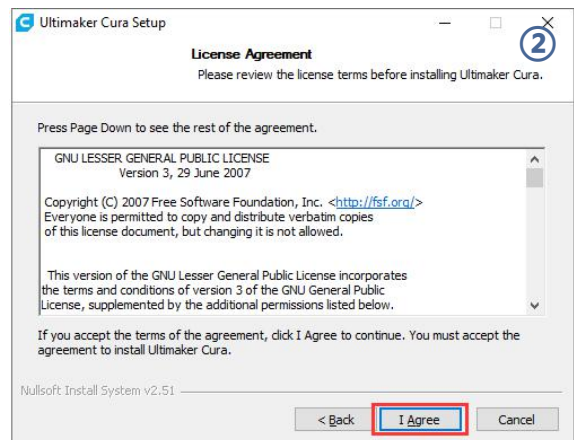
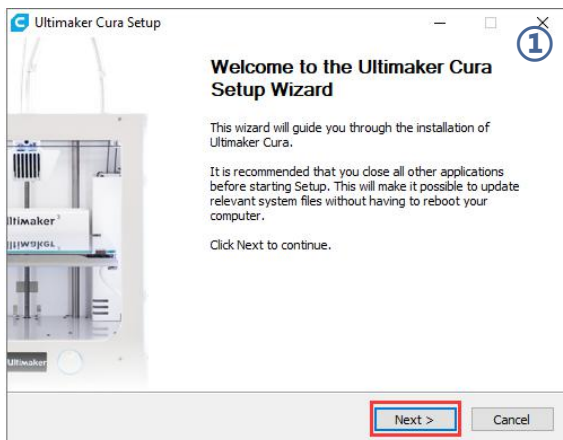
1. Cura Installation
2. Machine Settings
3. Import the Configuration File
4. Manipulate 3D model in Cura
5. Slice and Preview
6. Print Offline and Online

1. Cura Installation

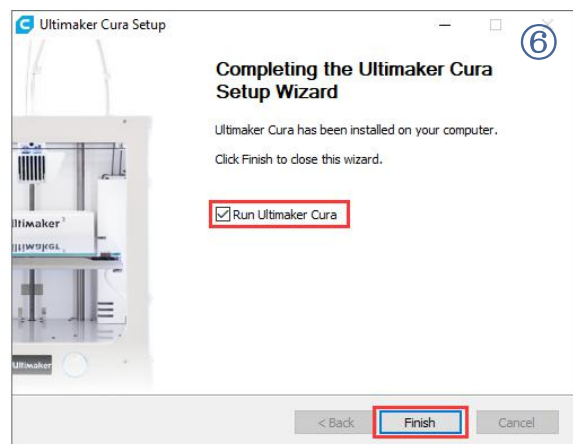
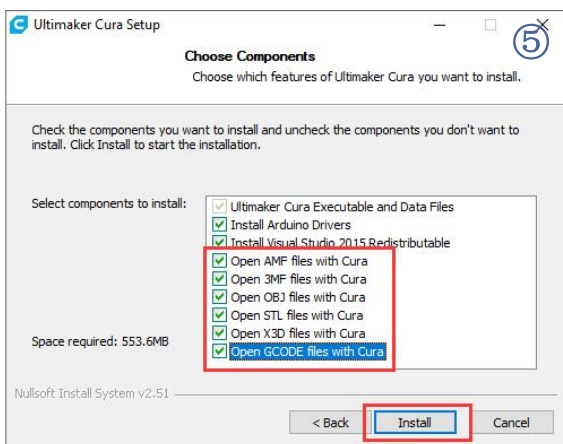
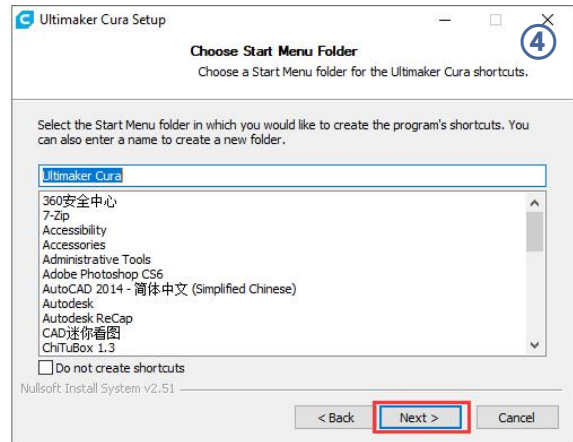
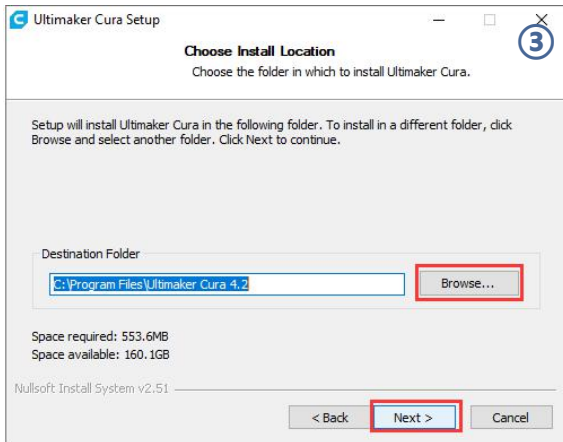
Double click **Ultimaker_Cura-4.2.1-win64**, and follow the steps as shown below.

* Ultimaker Cura 4.2.1 is used for example here, please refer to the memory card for the actual version.

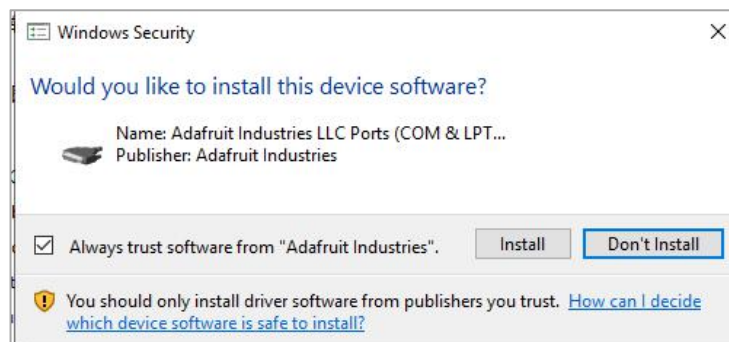
* File path: Memory card → “Files_English_Vyper” → “Cura” → “Windows” or “Mac”.



Introduction to slicing software



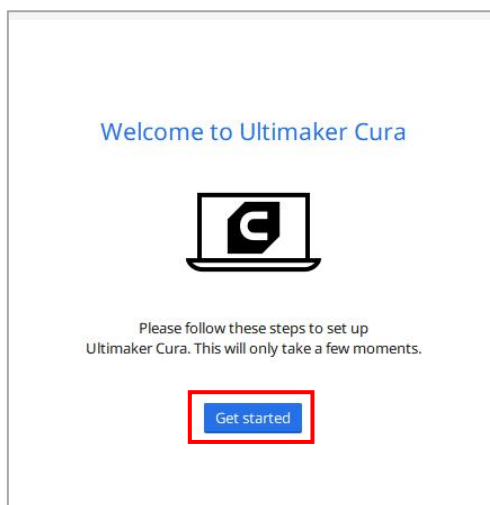
* Printing online requires the installation of a driver, as shown below. If you don't print online, you don't need to install it.



Introduction to slicing software

2. Machine Settings

Upon completion of installation, the first launch of the software will display the following welcome screen. Click **Get started** to start the machine settings.



①

User Agreement

Disclaimer by Ultimaker
Please read this disclaimer carefully.

Except when otherwise stated in writing, Ultimaker provides any Ultimaker software or third party software "As is" without warranty of any kind. The entire risk as to the quality and performance of Ultimaker software is with you.

Unless required by applicable law or agreed to in writing, in no event will Ultimaker be liable to you for damages, including any general, special, incidental, or consequential damages arising out of the use or inability to use any Ultimaker software or third party software.

[Decline and close](#) [Agree](#)

②

What's new in Ultimaker Cura

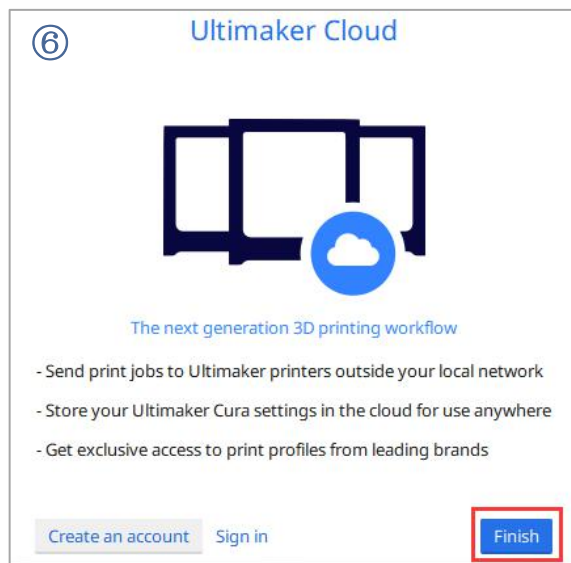
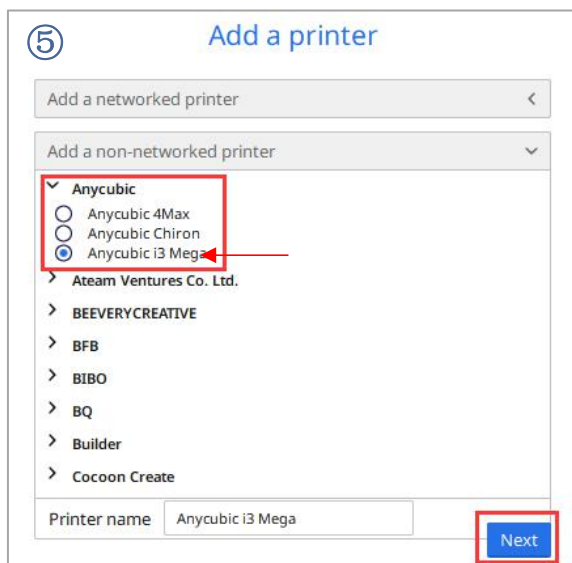
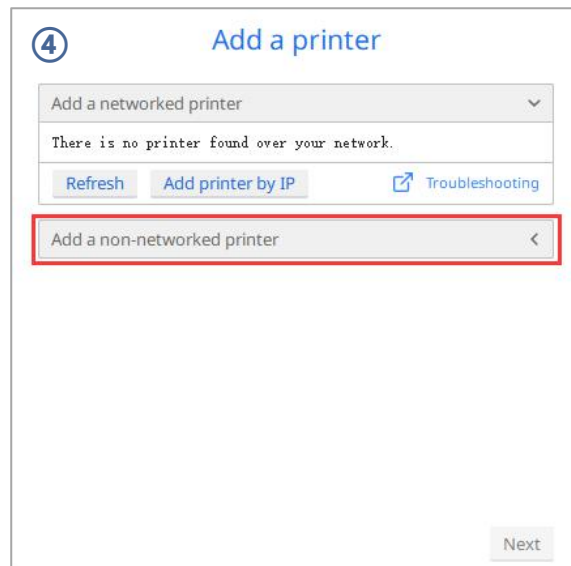
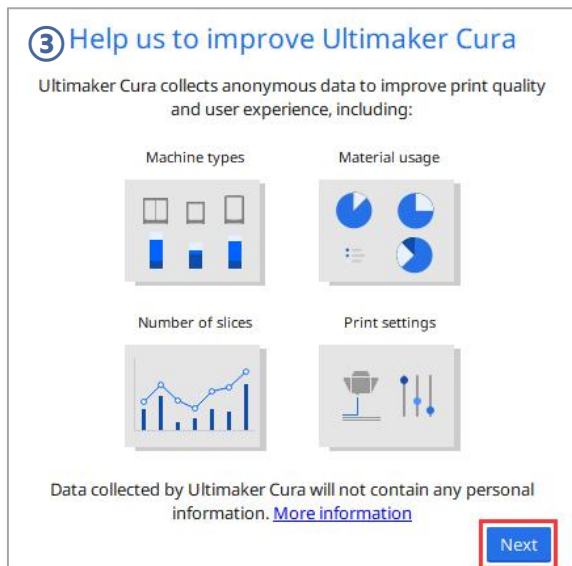
4.2.0

Orthographic view.
When preparing prints, professional users wanted more control over the 3D view type, so this version introduces an orthographic view, which is the same view type used by most professional CAD software packages. Find the orthographic view in View > Camera view > Orthographic, and compare the dimensions of your model to your CAD design with ease.

Object list.
Easily identify corresponding filenames and models with this new popup list. Click a model in the viewport and its filename will highlight, or click a filename in the list and the corresponding model will highlight. The open or hidden state

[Next](#)

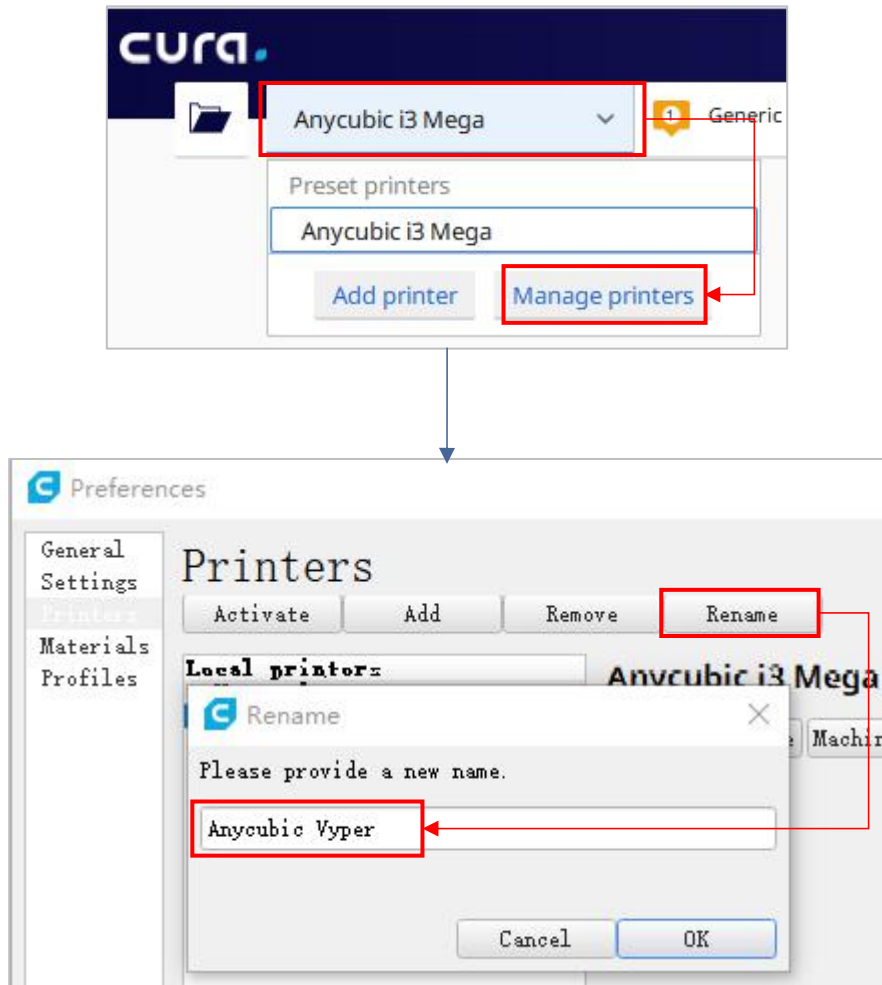
Introduction to slicing software



According to the wizard, we have selected the **Anycubic i3 Mega** printer. Now, we will set the parameters of Vyper based on the printer.

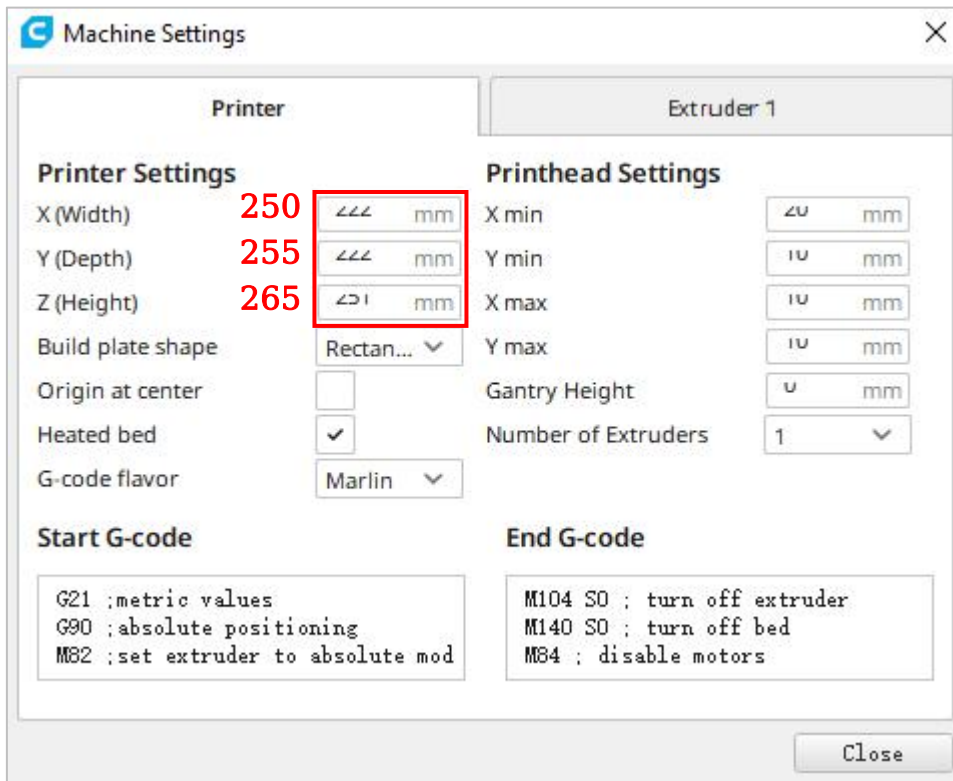
Introduction to slicing software

(1) Click **Anycubic i3 Mega** → **Manage printers** on the top left corner, and rename the printer to **Anycubic Vyper**.



(2) In the same interface opened in Step (1), click **Machine Settings** and set the **X (Width)** / **Y (Depth)** / **Z (Height)** parameters of printer as 250, 255 and 265 respectively, as shown below.

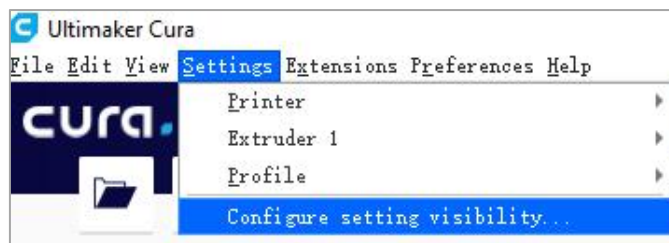
Introduction to slicing software



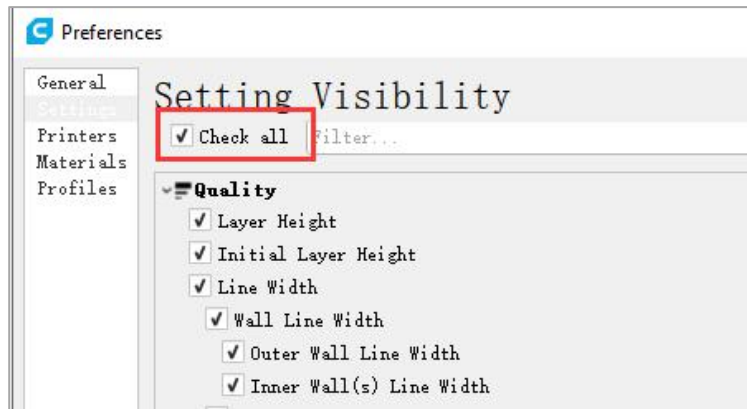
3. Import the Configuration File

After continuous testing, we provided users the suggested printing parameters of different filaments for Vyper, and the user could directly import the parameter files to the software.

(1) Click **Settings** → **Configure setting visibility...**, and then tick **Check all** box to make all settings visible.



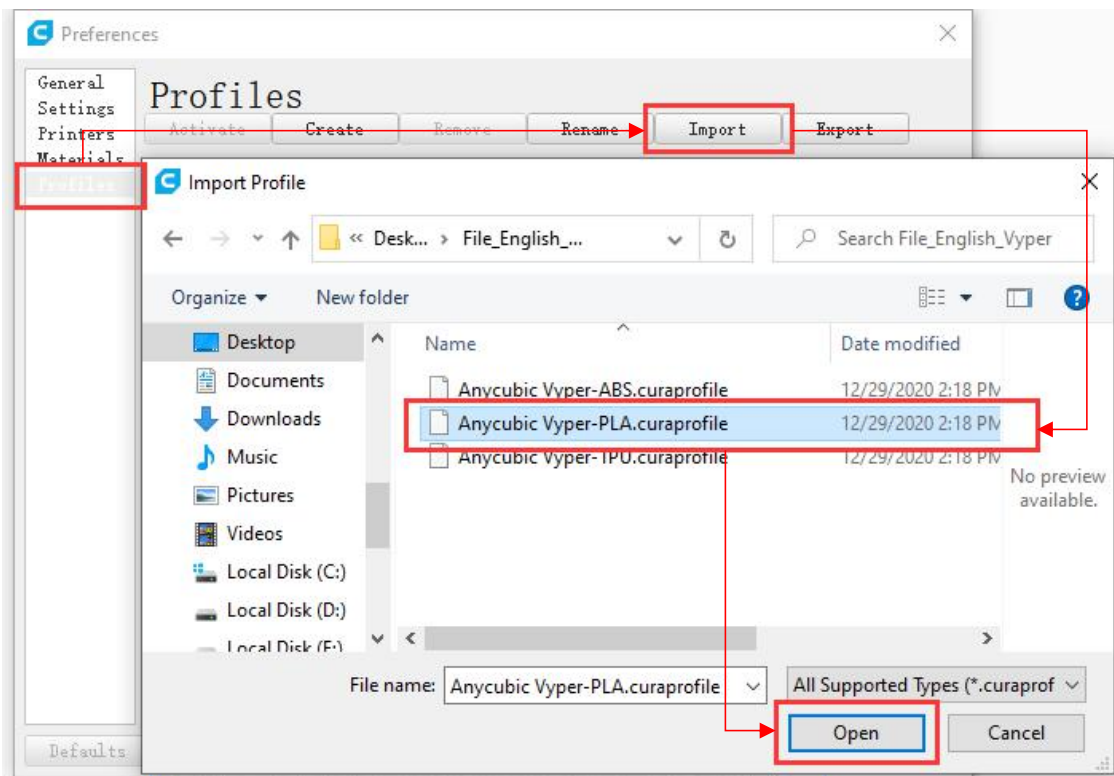
Introduction to slicing software



(2) In the same interface opened in Step (1), click **Profiles** on the left side. Then click **Import** button to import the desired print profile.

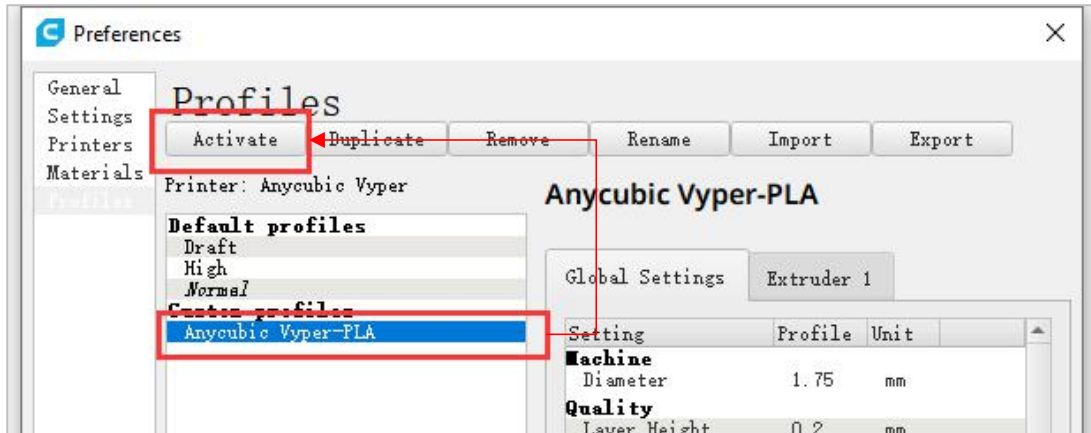
* Choose different parameter profiles according to different filament types.

* Profile path: Memory card → "Files _ English _Vyper" → "Suggested parameters for Vyper"

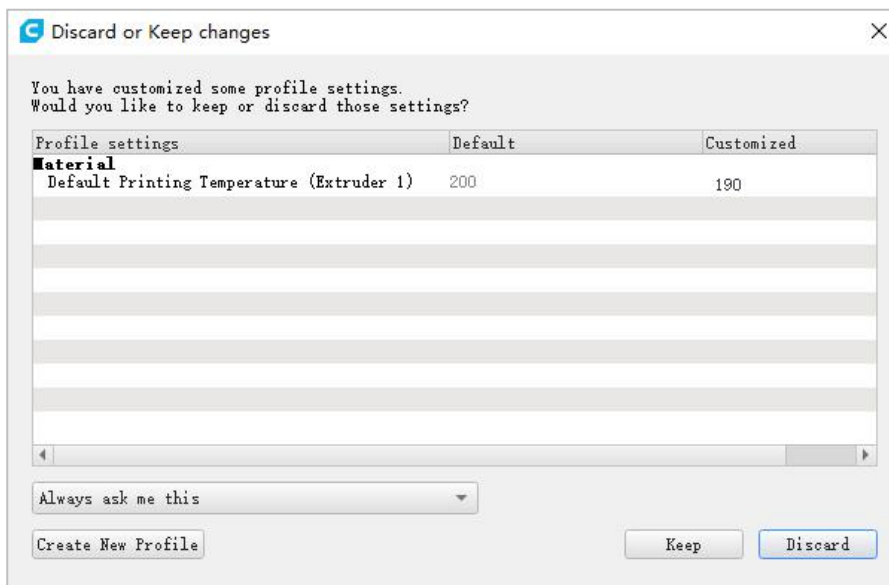


Introduction to slicing software

(3) Select the profile that you just imported, and then click the **Activate** button.



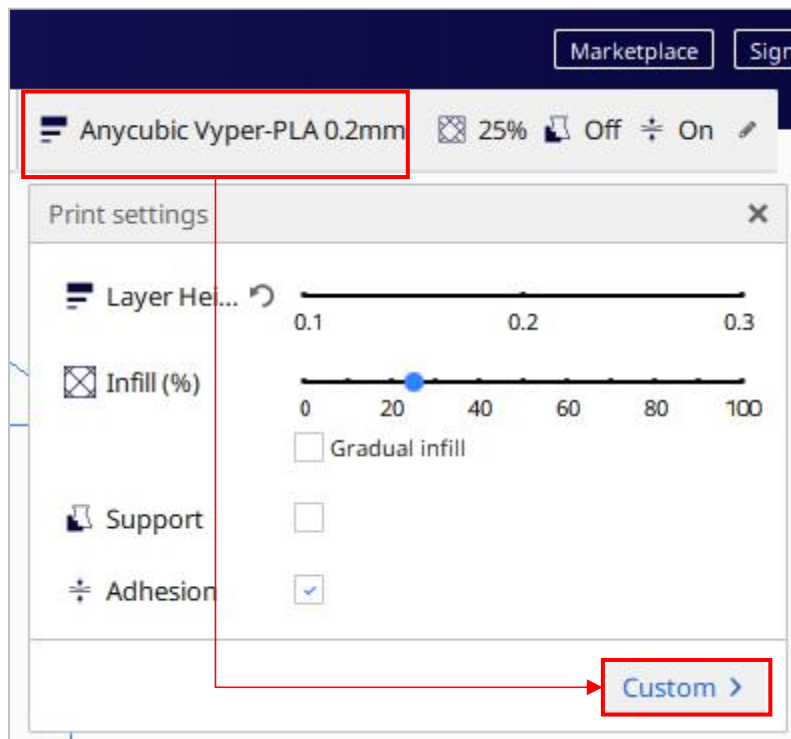
* Note: An interface may pop up after clicking the **Activate** button. If so, please check the values of listed parameters. You can click **Keep** to keep the custom values or click **Discard** to enable the values in the profile.



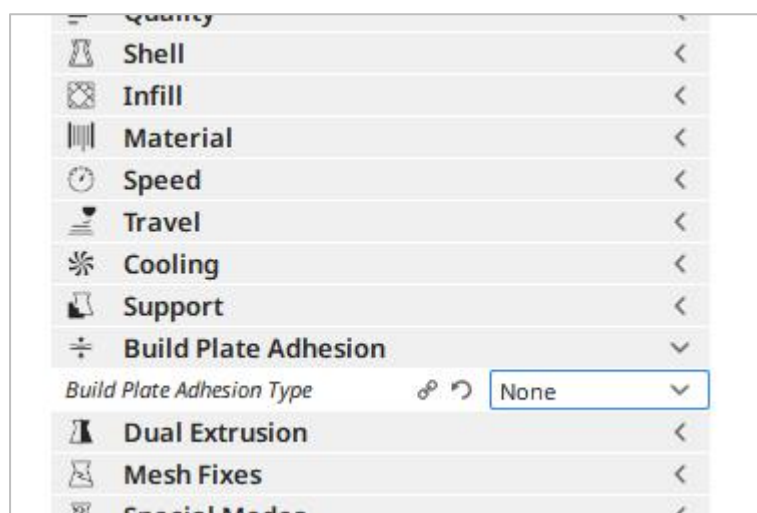
Introduction to slicing software

Supplementary Explanation

- ① You can customize some parameters after importing the profile, but we still recommend that you use the parameters in the profile that we provide.



- ② The parameter **Build Plate Adhesion** → **Build Plate Adhesion Type** needs to be set to "None" when you print the maximum size model.

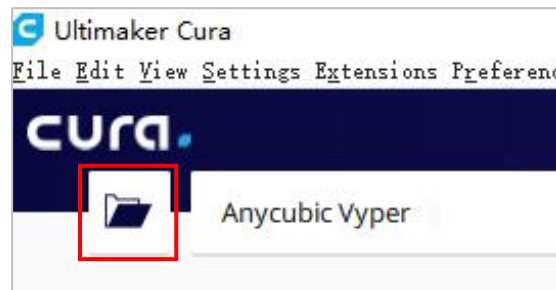


Introduction to slicing software

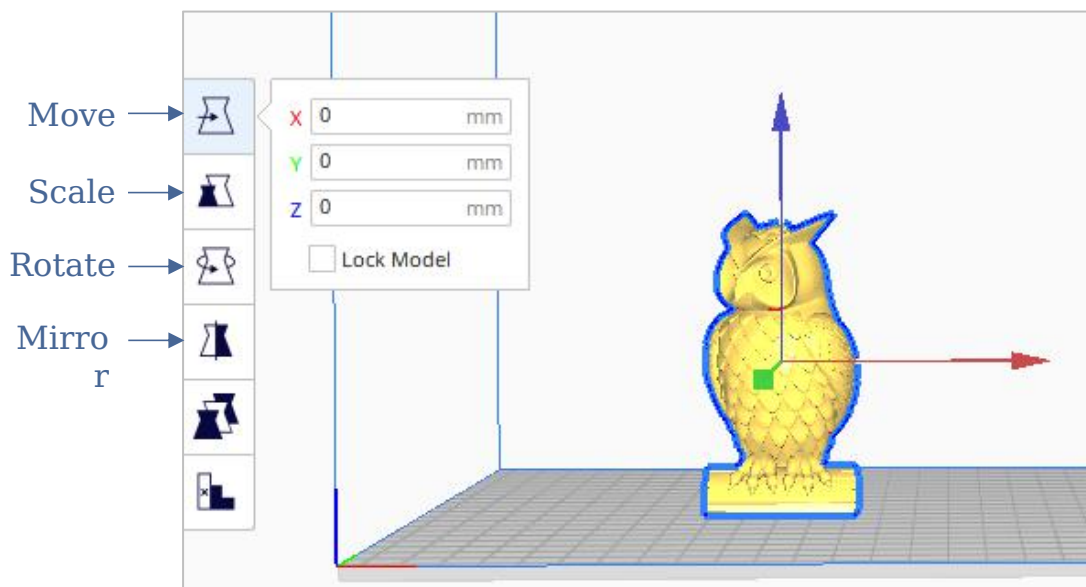
4. Manipulate 3D model in Cura

* Model files can be downloaded from www.anycubic.com (ANYCUBIC website) and www.thingiverse.com.

(1) Click the open file button in the top left corner to import the model file or directly drag the file into the software.



(2) Left click the model to activate the adjustment tool on the left side of the interface. Choose one of the adjustment tool, and then click and drag the controls on the model to adjust it.



The model name: owl

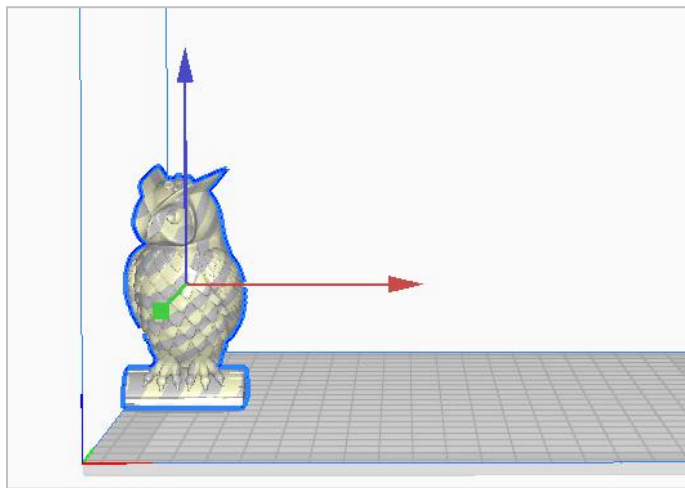
The author of the model:
etotheipi

Introduction to slicing software

Other operations:

- ① Press the mouse wheel and move the mouse to move the platform.
- ② Scroll the mouse wheel to zoom in and out the platform.
- ③ Right click and move the mouse to change the viewing angle.

* Note: When moving the model, make sure that the model is in the printable area. The color of the model in the following figure indicates that the model is out of print range.

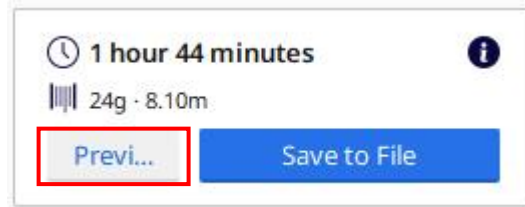


5. Slice and Preview

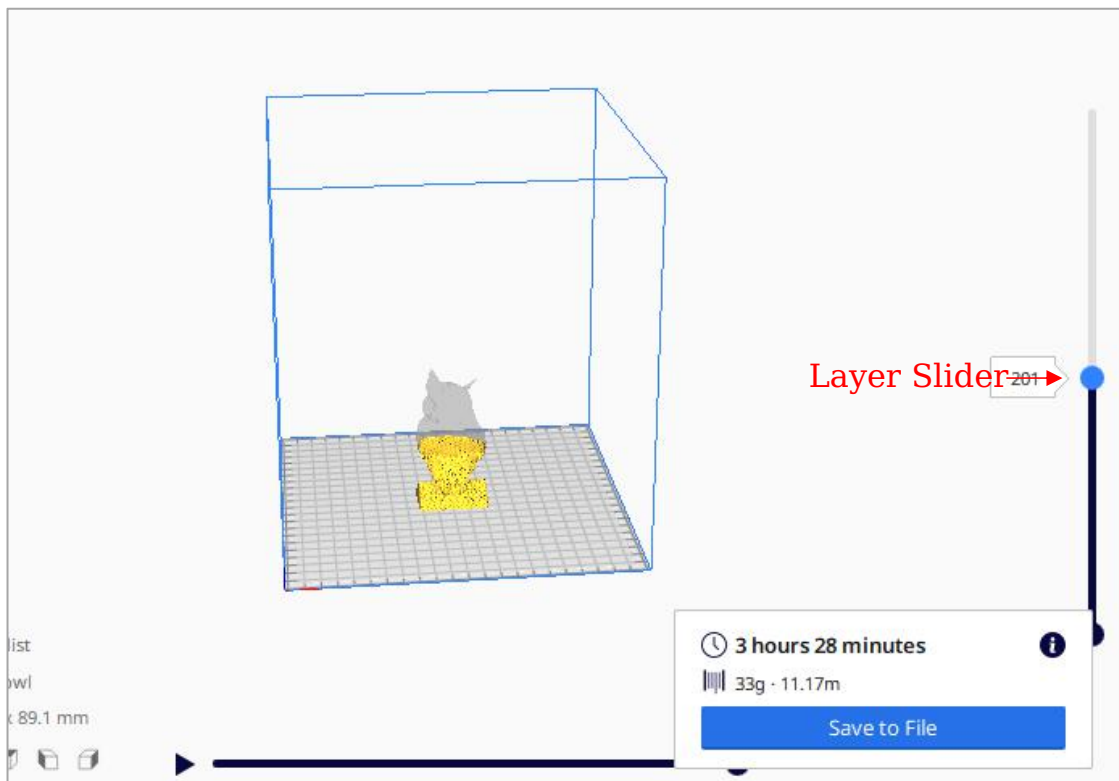
Now that the configuration and print settings are in place, it's time to slice the model. Click **Slice** button in the bottom right corner. When the process completes, it will immediately show a **Preview** button, click it to go to the preview stage.



Introduction to slicing software



Use the layer slider and simulation view to check important parts of your 3D slice.

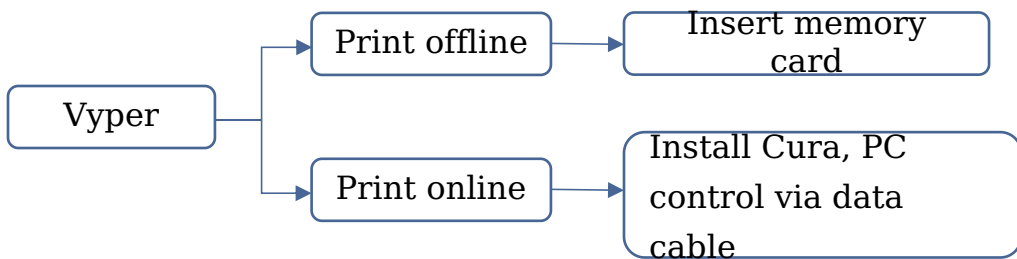


Introduction to slicing software

6. Print offline and online

There are two operational modes for Vyper: print offline and print online.

- **Print offline:** Insert memory card into the memory card slot, click on the Home Menu “Print” to enter the file list, and print a selected file (GCode files ONLY).
- **Print online:** Install CH340 driver to bridge PC and machine, and install Cura for slicing and control the machine to print via data cable.



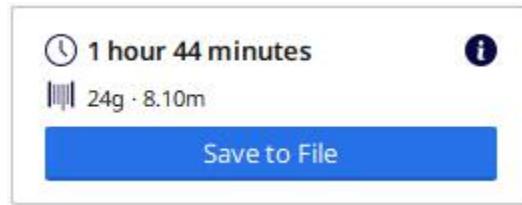
It is suggested to use **Print Offline** mode to minimize the noisy signal via data cable.

● Print offline (Recommended)

After slicing, click **Save to File** in the bottom right corner. Save the GCode file to the memory card, and then insert the memory card to the printer and control via the touchscreen for offline printing.

* GCode file name should only contain English letters, underscore and space. File name contains special characters could not be recognized by the printer. In order to let the printer better recognize the GCode file in the memory card, you need to back up all the files in the memory card to the computer, and keep the memory card only for the GCode file. Please save all the GCode files in root directory of the memory card.

Introduction to slicing software



● Print online

Printing online requires the installation of a driver. And you can control the printer through Cura after connecting the data cable.

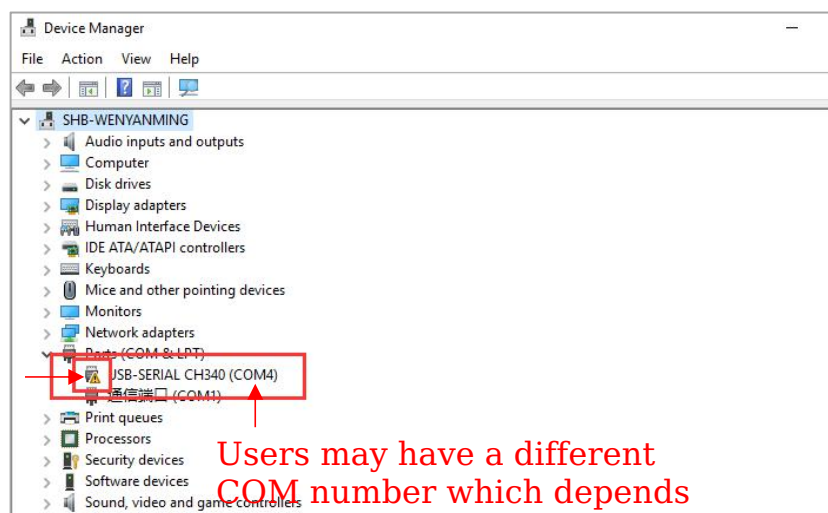
(1) Turn on the machine, connect the printer (data cable port) and your PC via data cable. The driver may not be installed automatically, so it is required to check that. Right-mouse click “This PC” → “Properties” → “Device manager”, if there is an exclamation mark as shown below, then it needs to be installed manually.

* Vyper uses CH340 chip for communication.

* CH340 driver file path: “Files_English_Vyper” → “Driver_CH341”.

There are two versions, Windows and Mac version. (* CH341 driver file is suitable for CH340 chip).

Exclamation
Mark



Users may have a different
COM number which depends
on the PC.

Introduction to slicing software

(2) Here we take Windows PC system for example. Double click “CH341SerSetup.exe” and follow the wizard to install it. **If the driver has installed automatically in step (1), skip this step.**

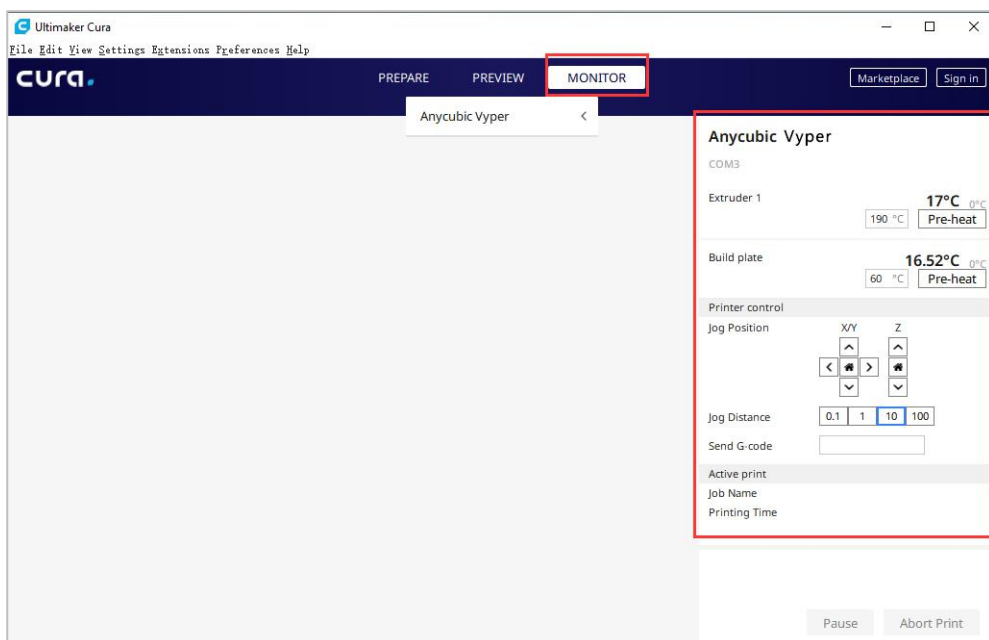
* After the installation is completed, refer to the method in step (1) to see if the driver has been identified (no exclamation mark).

(3) After slicing, click **Print via USB** in the bottom right corner to go to the monitor stage. After waiting for about ten seconds, the operation panel will be displayed on the right side of the interface. And the printing platform will be heated up for printing.

* If the printer is not connected properly, the Monitor interface will be blank. Please plug the data cable to connect again.

* In the process of printing, do not plug the data line, or it will interrupt the printing.

* In case of abnormal sound during the printing process, you can directly turn off the power of the machine. Please check the Troubleshooting chapter for more detail or contact Customer Service.



Resume from Outage

In case of a power loss during printing, the machine will automatically save the printing status. After restarting, you will see the interface shown below, please click "Continue" to continue your printing or "Cancel" to cancel it.



Maintenance

It is necessary to perform routine maintenance to the 3D printer to achieve consistent and quality results.

Some maintenance suggestions are shown as below:

1. Clean the nozzle with a needle under preheating conditions. If the filament residue in the nozzle could not be cleared 100%, please replace the nozzle.
2. Regularly add lubricating oil to smooth rods, linear bearings, lead screws, brass nuts, etc. It can minimize the wear-out failure of those moving parts.
3. Regularly clean the filament residue and dirt on the nozzle, platform, guide rail, motor, fan, etc.
4. Pay attention to the wear conditions of the D-shape wheels. Although they could be used for long time, please replace the D-shape wheels if they have been wore-out.
5. After finishing a printing, clean the printing platform to ensure the adhesion of the bottom layer of the model for next printing.
6. Check the belts regularly and tighten them if necessary.

Troubleshooting

1. Motor shaking or abnormal sound

- ① The corresponding limit switch could not be triggered when home. Please check the wirings, and inspect any obstacles by manually moving the corresponding axis.
- ② The motor cable are not connected properly. Please Check each connection and then inspect the cable routing for any faults.
- ③ Drive current is too large / small (please contact customer service).
- ④ The motor is damaged.
- ⑤ The motor driving wheel is loose.
- ⑥ The belt is loose, please check the belt tension at each position of X/Y/Z. And whether the belt is slippery during motor movement or not.

2. File not printing or memory card failure

- ① Remove the memory card and insert into PC. Open the GCode files using text editor (eg. Notepad), and inspect if GCode is readable or not. If file contains of multiple “ÿÿÿ” symbol, then file has been corrupted. Try reformatting the memory card to FAT32 format and reloading the GCode file.
- ② Memory card is not readable, ensure file name does not contain special characters or Change memory card.
- ③ Touchscreen freeze, reboot the machine and try again.

3. No extrusion or extrusion motor knocking

- ① Filaments tangled on spool or the Teflon tube is out of shape. Please straighten out the filaments and Teflon tube, and pull the filaments to observe whether the resistance is normal.
- ② Ensure that the nozzle temperature has been set to match the filament. (PLA is generally set at 190-210 ° C, and ABS set at 230-240 ° C)

Troubleshooting

- ③ Nozzle clogged. Please try to briefly heat the print head to a temperature higher than 230°C and manually push the filament into the extruder. You can also clean it with 0.4mm nozzle cleaning needle. If it doesn't work, you may need to replace it.
- ④ Not enough cooling for the hotend.
- ⑤ The printer has been working above the maximum speed for a long time, please reduce the speed.
- ⑥ Check whether the extruder is normal and whether the friction force of the extruder to the filament is sufficient. Try to tighten or clean the extruder wheel.

4. Filament leaking

- ① Nozzle, heating block or throat tube is not tightened properly, try to fix/replace it after cooling or contact customer service.

5. Layer shifting

- ① Print head moves too fast, please slow down the print speed.
- ② Check the belts and driving wheels and ensure they are properly installed.
- ③ Drive current is too small.

6. No sticking to the bed

- ① The nozzle is too far from the platform, please try to re-level; at the same time set the "initial layer thickness" to 0.2 in Cura, and set the "initial layer line width" (for example, set to 150) to improve the first layer adhesion.
- ② Print too fast at the bottom layer speed, reduce it to ~20mm/s
- ③ Ensure that the print platform is clean
- ④ Add a brim or raft to the model in slicing software

7. Freezing screen

- ① Please check if the soft cable above the screen is loose.
- ② Inspect if the touchscreen has been pressed by something.
- ③ Check if screen has cracks, if so, please contact our after sale service via official website www.anycubic.com

8. T0 sensor abnormal

- ① Check the wiring of the hotend and ensure a good connection.
- ② Check if there is any pins bent inside the connector.

9. Print head move abnormal

- ① The setting of machine type in slicing software is wrong. Please see page 36 to check the right setting.

10. Print stopped halfway

- ① Use print offline mode (memory card) instead of print online via data cable.
- ② Check if the GCode file is corrupted.
- ② Delete plugins in the GCode file.
- ③ The quality of the memory card is unstable. Try changing another brand memory card.
- ④ The power supply voltage is unstable. Please print again when the voltage is stable.

11. Some of the structure of the model cannot be printed

- ① If your model includes overhangs or bridges, you may need to add support on the model or adjust the angle of the model in the slicing software. It is recommended to preview the print layer to check the printability of the model.

12. Drawing is more serious

- ① The retraction distance is insufficient. It should be set larger in the slicing software.
- ② The retraction speed is too slow. It should be set a bit faster in the slicing software.
- ③ The printing temperature is too high, which causes the fluidity and viscosity of the filament to be strong. The printing temperature needs to be set a little lower.

13. Nozzle is too low or too high when printing

- ① Nozzle is too close to the platform, resulting in insufficient extrusion of filament. Click "Settings" → "Z offset +" to rise the nozzle.
- ② Nozzle is too high and the gap is large, which lead to poor adhesion of filament to the platform. Click "Settings" → "Z offset -" to lower the nozzle.

Thank you for purchasing ANYCUBIC products! Under normal usage and service, the products and its parts have a warranty period up to one year. Please visit ANYCUBIC official website (<https://www.anycubic.com>) for more details and report any issues with ANYCUBIC products. Our professional after-sale service team would response within 24 hours and help you to solve the issue.