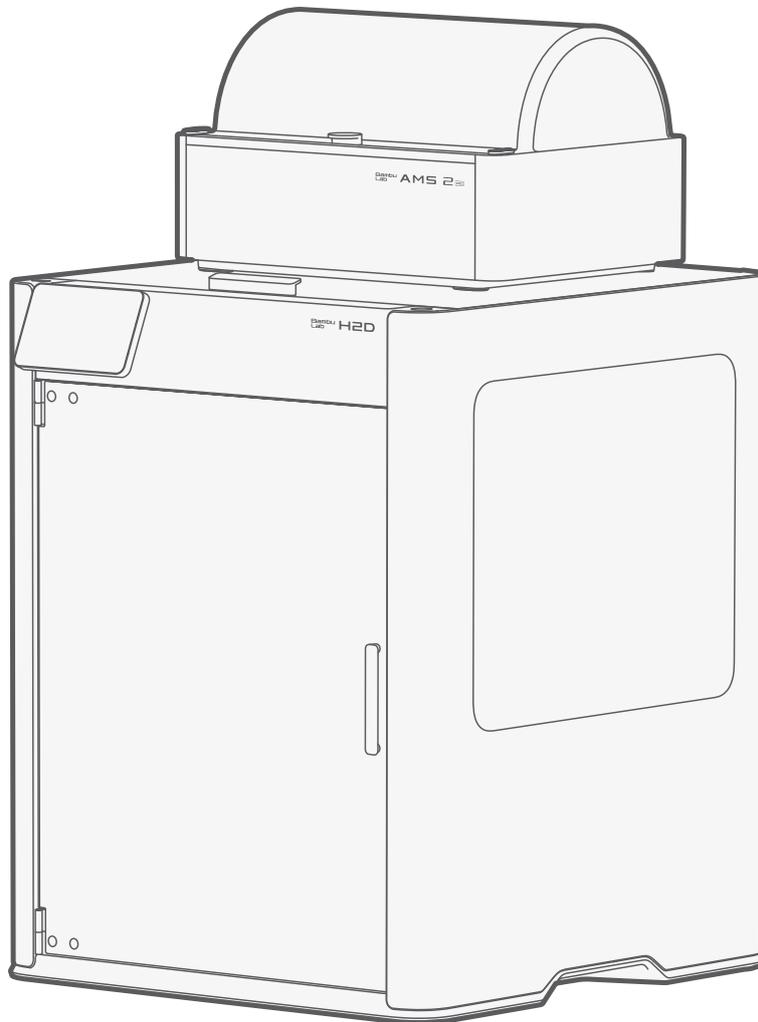


Bambu Lab H2D

3D Printing User Manual

V1.0 2026.02



Copyright Notice

This manual and its contents are intended solely to guide users in operating and using this product. The intellectual property rights are owned by Shenzhen Tuozhu Technology Co., Ltd. (hereinafter referred to as "Bambu Lab"). Without the written authorization of Bambu Lab, no part of this manual may be reproduced, distributed, modified, or provided to any third party in any form.

Manual Scope Description

This manual focuses primarily on the H2D and 3D printers. Some accessories or features mentioned in certain chapters are optional or upgrade items and may not be included in your product package. Please select the appropriate content for learning and operation according to your product model and configuration.

Version and Update

After firmware upgrades, information related to new features will be displayed on the printer's touchscreen. You can click the link below or visit the Bambu Lab Wiki page, then select **H2 Series > H2D > Printer Features > H2D Firmware Release History** or to view detailed information. If there is any discrepancy between this manual and the firmware release history, the firmware release history shall prevail.

wiki.bambulab.com/h2d/manual/h2d-firmware-release-history

Image and Parameter

Images shown in this manual are for illustrative reference only. The actual display and features may vary depending on the software or firmware version.

Thank you for choosing Bambu Lab H2D 3D Printer!

This manual provides you with comprehensive information for using 3D printing functions, including device setup, operation, and routine maintenance. To ensure safe, correct, and efficient use of this product, please read and fully understand this manual before use.

Use Guide

- 👉 Click on the table of contents and in-text links for quick navigation.
- 🔍 Use the electronic PDF to search for keywords for precise lookup.
- 📖 Visit the Bambu Lab Wiki (<https://wiki.bambulab.com/h2>) to search chapter keywords and access detailed steps and videos.

Quick Start Guide

1. Go through the [Read before Use](#).
2. Begin the [First Print](#).
3. [Download 3D Models](#).
4. [Adjust Slicing Parameters](#).
5. Review [Print Quality Issues and Solutions](#) and [Other Common Issues and Solutions](#).

Additional Resources

- **Unboxing and Installation:** View the unboxing guide (<https://wiki.bambulab.com/general/unboxing-guide>) for printer unpacking and setup instructions.
- **Customized Courses:** Visit Bambu Lab Academy (<https://bambulab.com/support/academy>) to access structured courses on the printer and software.

Feedback Survey

If you have any questions or suggestions regarding this user manual, please click the following link or scan the QR code to provide feedback.



<https://e.bambulab.com/t?c=OISLCzZcXiGKMRVi>

Symbol Guide

NOTE

Supplementary explanations for the main content.

TIPS

Suggestions to optimize operations, improve efficiency, or enhance portability, helping you use the printer more effectively.

CAUTION

Operations that may affect print quality or device performance. Please operate carefully to avoid potential issues.

DANGER

Potential risks present. Ignoring this may cause equipment damage or personal injury. Please always follow the relevant guidelines.

Technical Support

If you encounter problems during use, it is recommended to first consult this manual. You can also visit Bambu Lab Wiki to obtain detailed troubleshooting solutions by searching with keywords, or consult Bambu AI (support.bambulab.com) for maintenance tips, diagnostic methods, and solutions.

If your issue is not addressed or the troubleshooting steps do not resolve it, submit a service ticket or contact online technical support via the Service & Support website (bambulab.com/support) to get professional assistance.

Table of Contents

Read before Use.....	10
Chapter 1 Introduction.....	12
1.1 Product Introduction.....	12
1.2 H2D.....	13
1.2.1 3D Printer.....	13
1.2.2 Toolhead.....	16
1.2.3 Printer Touchscreen.....	17
1.3 AMS 2 Pro.....	20
1.4 AMS HT.....	21
1.5 Printer and AMS Status.....	22
1.5.1 Printer Status Indicator.....	22
1.5.2 AMS 2 Pro Status Indicator.....	23
1.5.3 HMS Messages.....	24
Chapter 2 First Print.....	26
2.1 Preparation.....	26
2.1.1 Printer Placement.....	26
2.1.2 Initial Installation.....	26
2.1.3 Initial Calibration.....	29
2.2 Load Filament.....	29
2.2.1 External Spool Holder.....	29
2.2.2 AMS 2 Pro / AMS HT.....	30
2.3 Initiate a Print from Printer Touchscreen.....	31
2.4 Remove the Model after Printing.....	32
2.5 Filament Unload.....	33
2.5.1 External Spool Holder.....	33
2.5.2 AMS 2 Pro / AMS HT.....	33
2.6 Waste Disposal.....	33

Chapter 3 Access 3D Model Resources.....	34
3.1 MakerWorld Model Community.....	34
3.2 MakerLab Creative Tools.....	35
Chapter 4 Initiate a Print from Bambu Handy.....	37
4.1 Install and Bind Bambu Handy.....	37
4.2 Initiate a Print.....	38
Chapter 5 Initiate a Print from Bambu Studio.....	42
5.1 Install and Bind Bambu Studio.....	42
5.2 Introduction to Bambu Studio Interface.....	44
5.3 Download and Import Model.....	48
5.3.1 MakerWorld Models.....	48
5.3.2 Other Models.....	48
5.4 Initiate a Print.....	49
5.5 Adjust Slicing Parameters.....	50
5.5.1 Model Size.....	50
5.5.2 Quality Settings.....	51
5.5.3 Strength Settings.....	55
5.5.4 Support Settings.....	59
5.5.5 Speed Settings.....	66
Chapter 6 Key Features Introduction.....	69
6.1 Large Volume Printing.....	69
6.1.1 Horizontal Printing Area.....	69
6.1.2 Vertical Printing Area.....	70
6.1.3 Check the Model Placement Area.....	70
6.2 Dual Hotend Printing.....	71
6.2.1 Filament Grouping Strategy.....	71
6.2.2 Select Filament Grouping Mode.....	72
6.2.3 Slicing Mode Selection.....	77
6.2.4 Multi-Material Printing with Soft and Hard Filament.....	77
6.3 High Precision Printing.....	80

6.3.1 Start Calibration.....	80
6.3.2 Print Calibration.....	82
6.3.3 High-Precision Nozzle Offset Calibration.....	83
6.3.4 Motion Accuracy Calibration.....	85
6.4 Intelligent Detection.....	86
6.4.1 AI Print Monitoring.....	86
6.4.2 Build Plate Detection.....	87
6.4.3 Hotend Type Detection.....	88
6.4.4 Live View Camera Calibration.....	88
6.5 Air Condition System.....	88
6.5.1 Select Mode.....	89
6.5.2 Custom Chamber Temperature.....	89
Chapter 7 Basic Controls and Functions.....	90
7.1 Control from Printer Touchscreen.....	90
7.1.1 Speed Settings.....	90
7.1.2 XYZ Axis Movement.....	91
7.1.3 Nozzle and Extruder.....	91
7.1.4 Heatbed and Chamber Temperature.....	93
7.1.5 Chamber Light Mode.....	93
7.1.6 Status Indicator.....	94
7.1.7 Sound.....	94
7.2 Photo and Video.....	95
7.2.1 Video Recording.....	95
7.2.2 Photo.....	96
7.2.3 Timelapse.....	97
7.3 Connect to Network.....	98
7.3.1 LAN Only Mode.....	98
7.3.2 Developer Mode.....	98
7.4 Connect USB Flash Drive.....	99
7.4.1 Specification.....	99

7.4.2 Connect and Format.....	100
7.4.3 Ejection.....	100
7.5 Update and Restore.....	101
7.5.1 Update Firmware.....	101
7.5.2 Initialization.....	102
Chapter 8 Filament.....	103
8.1 Select the appropriate filament.....	103
8.1.1 Filament Types by Function.....	103
8.1.2 Filament Types by Temperature.....	105
8.2 Filament Compatibility and Parameter Settings.....	106
8.3 Filament Drying.....	108
8.3.1 Filament Drying with the Printer.....	108
8.3.2 Filament Drying with AMS 2 Pro/AMS HT.....	108
8.4 TPU Printing Guide.....	110
8.4.1 TPU 85A.....	111
8.4.2 TPU 90A.....	113
8.5 High-Temperature Filament Printing Guide.....	116
Chapter 9 Print Quality Issues and Solutions.....	118
9.1 First Layer Not Sticking.....	118
9.2 First Layer Too High/Too Low.....	119
9.3 Poor Overhang Quality.....	119
9.4 Model Warping, Falling Off, or Collapse.....	120
9.5 Filament Sticking to the Nozzle.....	121
9.6 Under-Extrusion.....	122
9.7 Stringing and Oozing.....	123
9.8 Gloss Difference.....	124
9.9 Interlayer Cracking.....	125
9.10 Seam.....	126
9.11 Belt Pattern.....	126
9.12 Top Layer Gaps.....	127

Chapter 10 Other Common Issues and Solutions.....	128
10.1 Printable Area Error.....	128
10.2 Nozzle Offset Calibration Failure.....	128
10.3 Motion Accuracy Calibration Failure.....	129
10.4 Clogging Troubleshooting (Non-TPU Filaments).....	130
10.5 Clogging Troubleshooting (TPU Filament).....	132
10.6 Nozzle/Hotend Unclogging Guide.....	134
10.6.1 Manual Extrusion.....	135
10.6.2 Unclogging Pin Cleaning.....	136
10.6.3 Cold Pull.....	137
10.6.4 Hot Allen Key Cleaning.....	138
10.7 Nozzle Clumping.....	140
10.8 Loose Belt.....	144
10.8.1 XY Belt Tension.....	144
10.8.2 Z Belt Tensioner.....	146
Chapter 11 Regular Maintenance.....	148
11.1 Maintenance Frequency and Operation Requirements.....	148
11.2 XYZ Axes Cleaning.....	149
11.2.1 X-Axis Linear Rails.....	149
11.2.2 Y-Axis Linear Rods.....	149
11.2.3 Z-Axis Linear Rods and Lead Screws.....	150
11.3 Toolhead Surface Cleaning.....	150
11.4 Nozzle Lifting Rail Cleaning.....	151
11.5 Camera and Sensor Cleaning.....	151
11.6 Clean the Heatbed.....	152
11.7 Clean the Build Plate.....	153
11.8 Replace Accessories.....	153
11.8.1 Hotend.....	153
11.8.2 Filament Cutter Blade.....	154
11.8.3 PTFE Tube.....	156

11.8.4 Flow Blocker.....	158
11.8.5 Nozzle Wiping Pad.....	160

Read before Use

To ensure safe operation and optimal performance of the printer, carefully review the following precautions before use:

Basic Safety and Electrical Requirements

- The actual operating voltage of the printer must match the voltage specified in the product specifications to avoid equipment damage and safety hazards. You can check the label next to the power socket for the specific voltage requirements.
- The printer is a high-temperature, high-speed device. Keep children and unauthorized personnel away to prevent burns, pinching, or other accidents.
- Do not touch the toolhead, heatbed, or moving parts during printing to prevent injuries.

Operation and Maintenance Recommendations

- To ensure proper functioning of the printer's internal precision mechanisms, regular maintenance is recommended (see [Regular Maintenance](#)).
- **Please note, the left hotend is not suitable for printing TPU filament. To prevent nozzle clogging and achieve better print quality, please use the right hotend to print TPU filament.**
- The printer automatically switches between different hotends during printing. Do not manually force switching to avoid damaging the device.
- For optimal print results, it is recommended to use official Bambu filaments. Official Bambu filaments have undergone rigorous compatibility, safety, and stability testing based on the product's characteristics to ensure the best printing performance.
- Unless otherwise specified, always unplug the power cord before performing any operation, maintenance, or modification to prevent electric shock or equipment damage.
- Unless otherwise specified, allow the printer to cool completely before performing any operation, maintenance, or modification.

Safety Instructions for AMS 2 Pro

- **To prevent filament jams, do not load flexible materials such as TPU with a hardness of 95A or lower, or moisture-absorbed PVA and BVOH into the AMS 2 Pro.**
- The AMS 2 Pro supports filament spool widths from 50 mm to 68 mm and diameters from 197 mm to 202 mm. Plastic spools are recommended.

- Use a 6-pin cable to connect the H2 series 3D printer with a single AMS 2 Pro and enable the AMS 2 Pro's drying function. To dry filament in multiple AMS 2 Pro units simultaneously, use the official Bambu Lab power adapters to supply power to the additional AMS 2 Pro units.
- During filament drying, the AMS 2 Pro removes moisture from the interior through an external circulation system. Ensure that the air inlet and outlet are not obstructed by other objects to achieve optimal drying performance.

Safety Instructions for AMS HT

- Using official Bambu filament is recommended. Official Bambu filaments have undergone rigorous compatibility, safety, and stability testing specific to AMS HT to provide the best print quality.
- **To prevent filament jams, do not load flexible materials such as TPU with a hardness of 95A or lower, or moisture-absorbed PVA and BVOH into the AMS HT.**
- AMS HT supports filament spool widths from 50 mm to 68 mm and diameters from 197 mm to 202 mm. Plastic filament spools are recommended. If using cardboard spools, it is recommended to use a spool adapter to reduce the risk of spool breakage and debris residue.
- To use the drying function of AMS HT, it must be powered with the supplied power cord.
- During filament drying, the AMS HT removes moisture from the interior through an external circulation system. When external circulation is enabled, the air inlet and outlet open. Ensure they are not blocked by any objects to achieve optimal drying performance.

Chapter 1 Introduction

1.1 Product Introduction

The H2D is an intelligent dual-nozzle 3D printer that uses Fused Deposition Modeling (FDM) technology. It supports large volume printing, intelligent detection, and adaptive airflow system to deliver rapid, precise, and high-quality results. Designed for versatility, it accommodates multi-color and multi-material models, as well as high-strength structural parts, making your creative process more efficient and versatile.

When used with the Automatic Material System (AMS), it further enhances the printer's intelligent functions and ease of use:

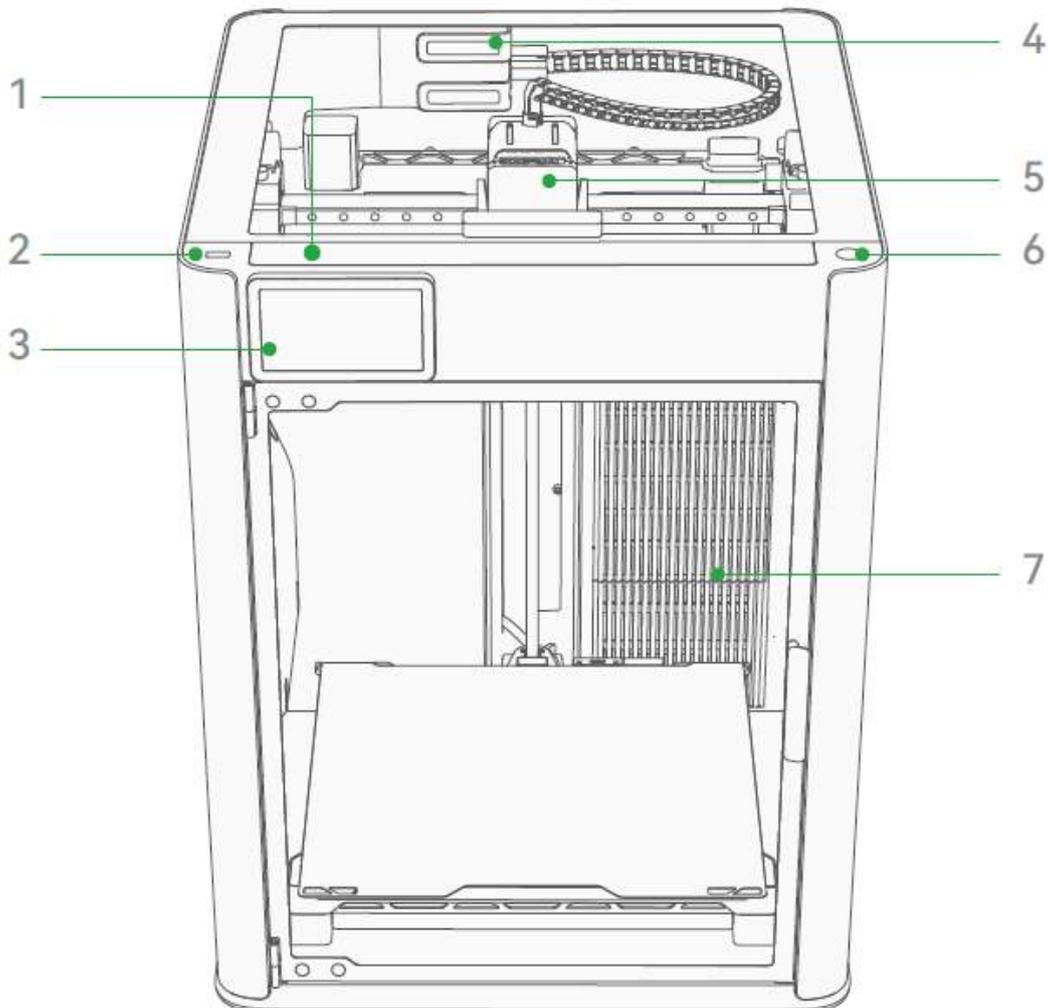
- **AMS 2 Pro:** Supports RFID filament sync, automatic multi-color/multi-material switching, and smart drying. Enhances multi-material printing and filament management for a smarter, more convenient, and improved 3D printing experience.
- **AMS HT:** Specialized for high-performance materials, the AMS HT supports intelligent recognition of high-temperature filaments, automatic multicolor and multi-material switching, and efficient smart drying, meeting the demands of engineering-grade and professional printing.

NOTE

This user manual covers only H2D 3D printing functionality and does not include information about the laser module or cutting module.

1.2 H2D

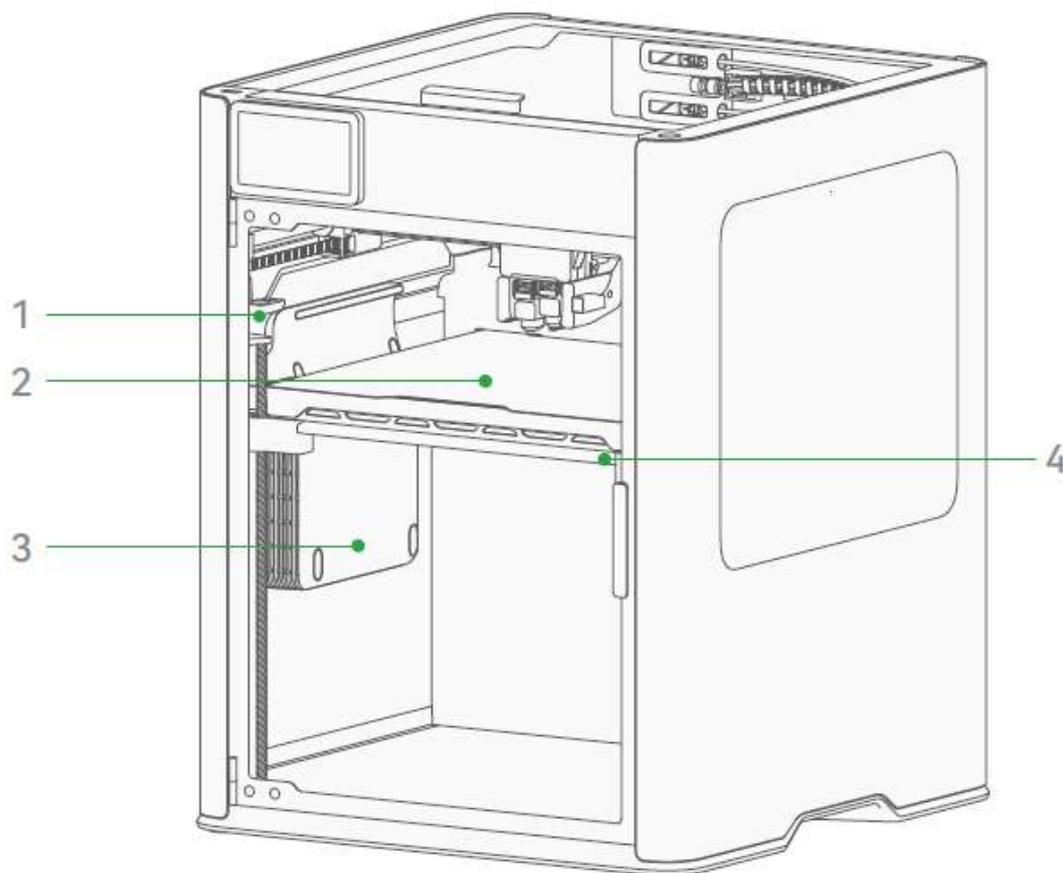
1.2.1 3D Printer



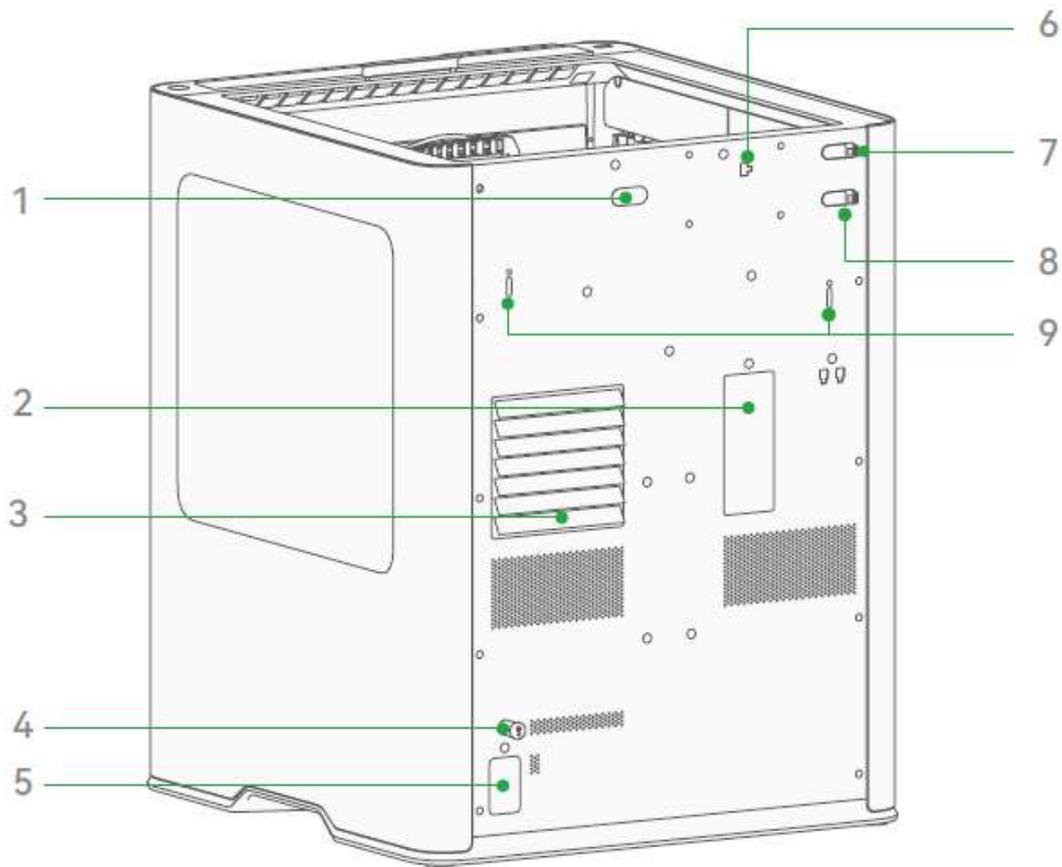
1. **Automatic top vent:** System controlled. When the chamber exhaust fan is operating, the top vent opens automatically to draw in external cool air, ensuring balanced chamber temperature.
2. **USB Port:** Insert a USB drive to start print jobs offline and to store timelapse video files.
3. **Touchscreen:** Displays printing parameters and controls the printer.
4. **Filament Buffer & Filament Tangle Detector:** Monitors filament feed for tangles or blockages and adjusts speed dynamically to ensure smooth material loading.
5. **Toolhead:** Consists of two PTFE tube connectors at the top, the extruder, and the dual-hotend assembly.
6. **Start/Pause button:** Controls the start, resume and pausing of laser and cutting tasks.
7. **Air Filter:** Filters the chamber's internal airflow to remove fine particles and reduce odors during printing.

NOTE

To pause or start 3D printing, operate via the screen or software.

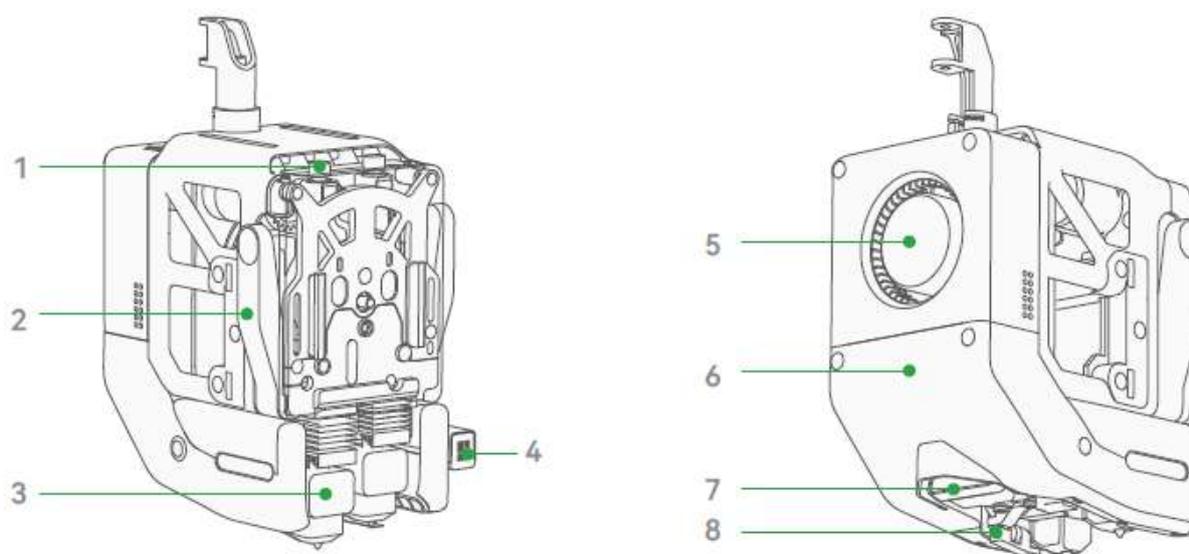


1. **Live View Camera:** Monitors print progress in real time, used for timelapse recording and AI detection.
2. **Heatbed:** Heats the print surface to ensure stable adhesion of printed layers to the build plate, preventing warping or detachment.
3. **Auxiliary Part Cooling Fan:** A high-powered 12 W cooling fan that provides additional airflow for high-speed printing.
4. **Status Indicator:** Indicates the printer's operating status through color and blinking patterns. (See [Printer and AMS Status](#) for detailed definitions).



1. **TPU Filament Inlet:** Used to manually load TPU filament that is not supported by the AMS.
2. **Purge Chute:** Installed at the rear of the printer to discharge purged or waste filament from the printer.
3. **Active Chamber Exhaust & Chamber Exhaust Fan:** Regulates airflow and expels air from the chamber to maintain stable temperature and cooling performance. In cooling mode, fan speed increases as the chamber temperature rises.
4. **Safety Key:** The printer can only be powered on after inserting the safety key or engaging the emergency stop button.
5. **Power Socket:** Connects the power cable and turns on the power.
6. **6-pin Connector:** Used to connect AMS.
7. **PTFE Tube Coupler - upper:** Connects to the right extruder and uses the right hotend for printing.
8. **PTFE Tube Coupler - lower:** Connects to the left extruder and uses the left hotend for printing.
9. **Belt Tensioner:** Adjusts and monitors belt tension, and feeds data back to the system to ensure printing accuracy and motion stability.

1.2.2 Toolhead



1. **Toolhead Filament Inlet:** The filament enters the corresponding hotend here after passing through the filament buffer.
2. **Filament Cutter Lever:** Cuts the filament by striking the cutter stoppers on both sides of the printer.
3. **Hotend:** Receives filament, heats and melts it, then deposits the melted filament onto the build plate for forming the print.
4. **Toolhead camera:** Used for motion accuracy calibration, high-precision nozzle offset calibration, and build plate marker recognition.
5. **Part Cooling Fan:** Directs cooling airflow through ducts to the left/right nozzle areas, providing efficient cooling during printing.
6. **Part Cooling Fan Duct:** Directs cool airflow to the left/right nozzles.
7. **Nozzle Camera:** Located behind the nozzle, used for detecting hotend clumping, air print, spaghetti, and similar anomalies; also used for calibrating purge chute position.
8. **Nozzle Flow Blocker:** A small black flap located beneath the nozzle that moves laterally with nozzle switching. It covers inactive nozzles to prevent molten filament from dripping onto the model or build plate.

1.2.3 Printer Touchscreen

Homepage



1. Screen menu bar, for switch screens.
2. Select internal models or USB stored models.
3. View and set Nozzle & Extruder parameters.
4. Quick access to the Filament screen.
5. Quickly configure network settings.
6. Quick access to the HMS screen.

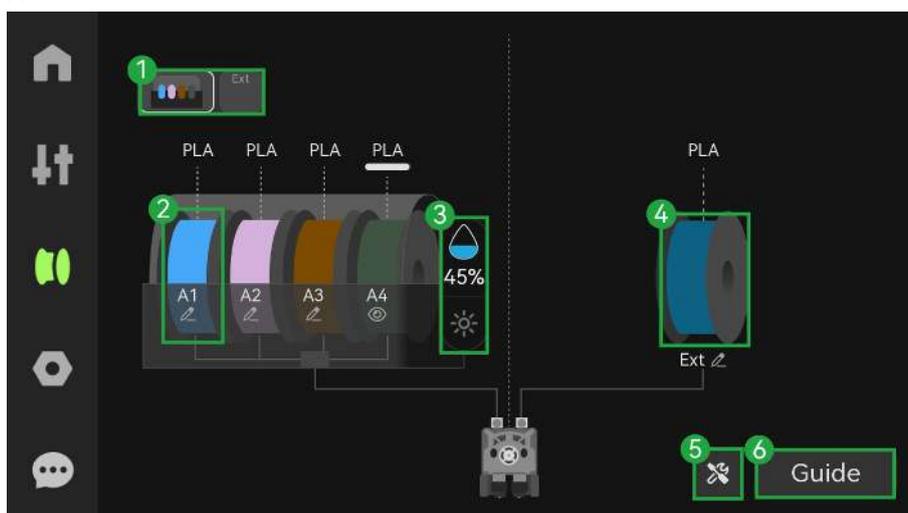
Control Screen



1. Select the appropriate air condition based on the filament type.
2. Set printing speed mode.

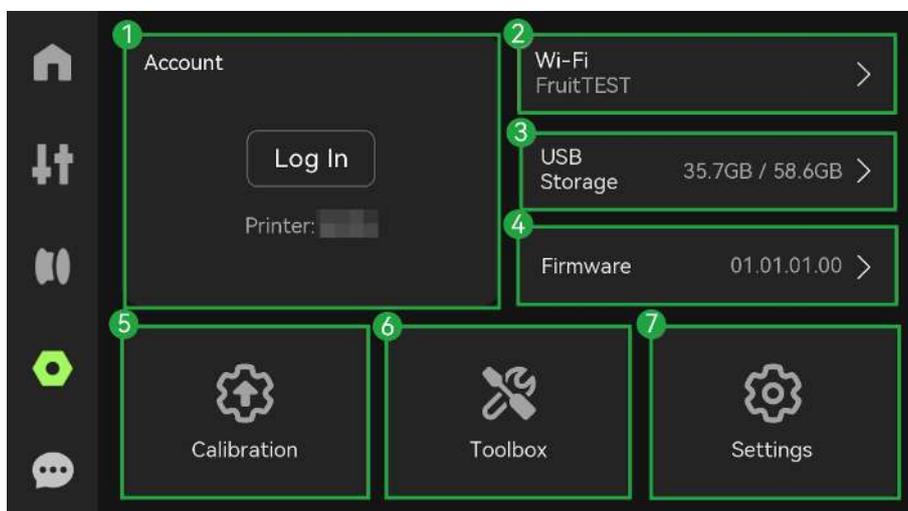
3. Control the movement of the toolhead and the heatbed.
4. Set the parameters for the left/right nozzle & extruder.
5. Set the chamber temperature.
6. Set heatbed temperature.
7. Control LED light.

Filament Screen



1. Switch between AMS or external spool.
2. Tap any filament spool icon to edit filaments, load/unload filament, and re-read filament RFID.
3. View humidity and temperature inside the AMS; perform drying.
4. Tap external filament tray icon to edit filament and load/unload filament.
5. Select auto refill and AMS setup functions.
6. View feeding operation guide.

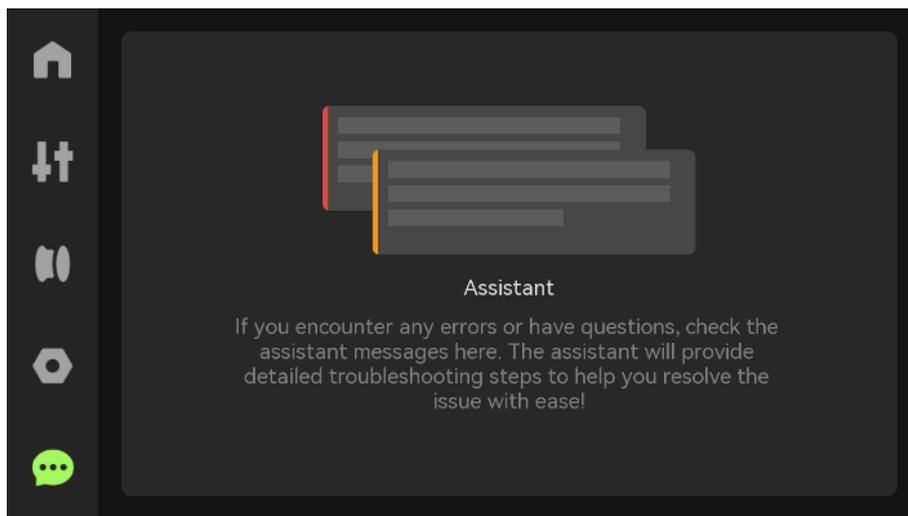
Settings Screen



1. Log in and view account.

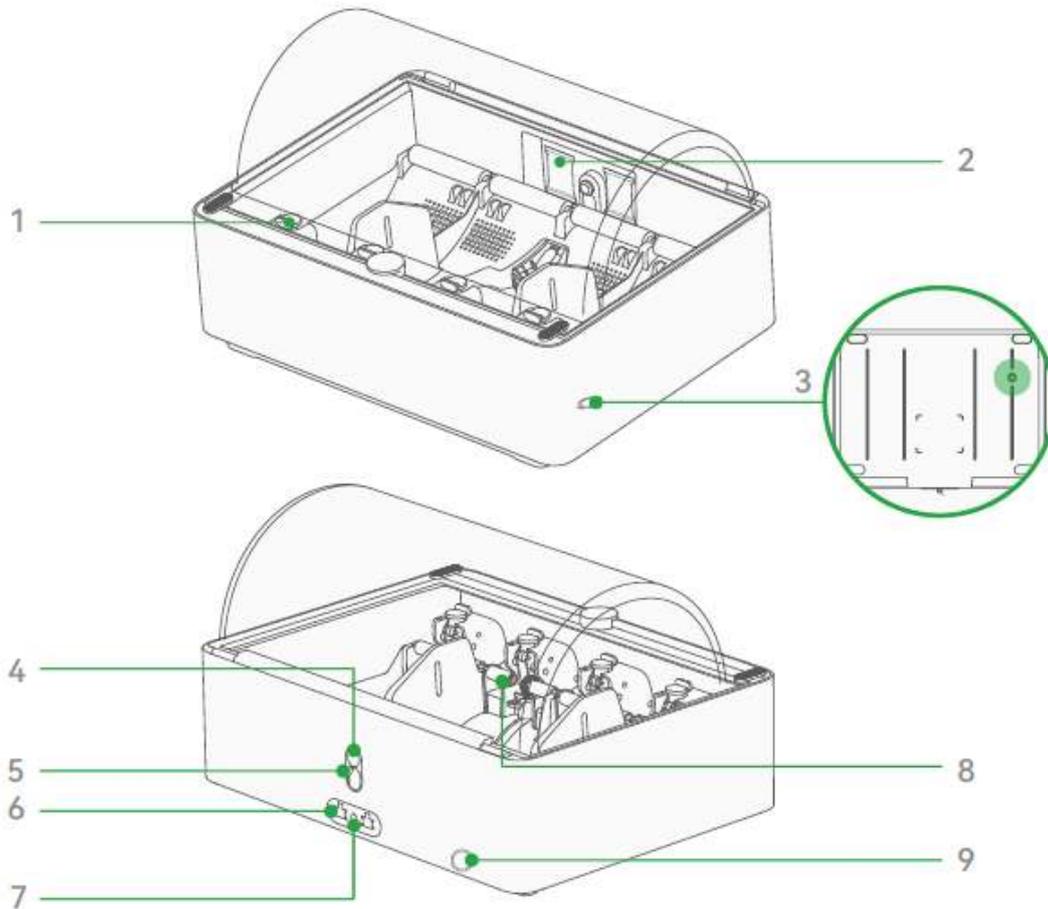
2. Configure printer network.
3. Configure USB storage.
4. View and update firmware.
5. Configure calibration functions.
6. Use toolbox to maintain the printer.
7. Other function settings.

HMS Page



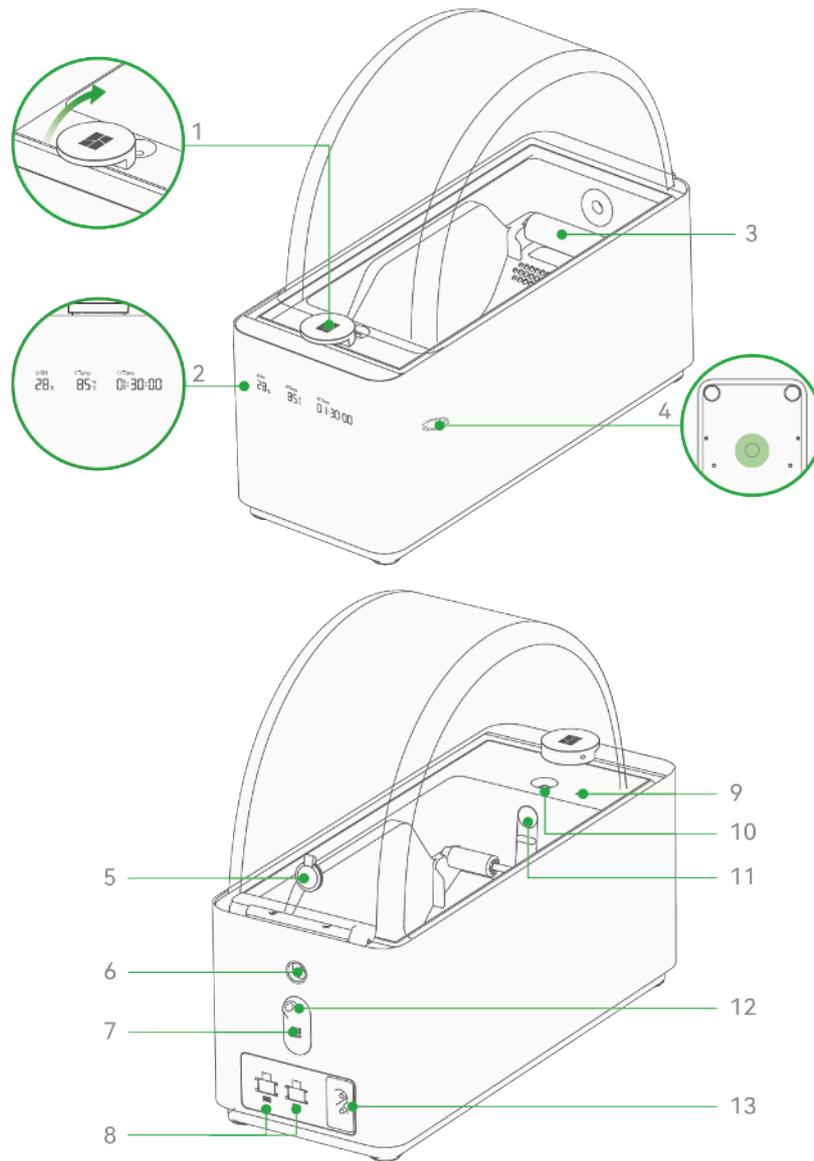
When the printer encounters a fault or requires maintenance, relevant prompt messages will be displayed here (see [HMS Messages](#)).

1.3 AMS 2 Pro



1. **Filament Inlet:** Insert the filament tip into this inlet. Once engaged, the system will automatically feed the filament.
2. **Desiccant:** Keeps the internal AMS chamber environment dry.
3. **Air Intake** Introduces external dry air.
4. **PTFE Tube Release Button:** Press this button on the back of the AMS to release and remove the PTFE tube.
5. **Filament Outlet:** Connects the AMS to the printer's filament inlet port.
6. **Bambu Bus Port 6-pin:** Connects the AMS to the printer or another AMS via a Bambu bus cable 6-pin.
7. **Power Connector:** Plug the power adapter into this port to supply power for the drying function.
8. **Active Support Shaft:** Actively supports and drives the rotation of the filament spool.
9. **Air Vent:** Expels moisture from inside the AMS.

1.4 AMS HT



1. **Locking Tab:** Used to open the AMS HT cover.
2. **Screen:** Displays real-time humidity, temperature, and remaining drying time from left to right, making it easy to monitor filament drying status.
3. **Driven Support Shaft:** Supports and stabilizes the filament spool.
4. **Air Intake:** Introduces external dry air.
5. **TPU Filament Outlet:** Dedicated filament outlet designed for TPU and other flexible or brittle materials.
6. **Air Vent:** Expels moisture inside the AMS.
7. **PTFE Tube Release Button:** Press this button on the back of the AMS to release and remove the PTFE tube.

8. **Bambu Bus Port 6-pin:** Connects the AMS to the printer or another AMS via a Bambu bus cable 6-pin.
9. **Status Indicator:** Shows the current working status of the AMS HT.
10. **Filament Inlet:** Insert the filament tip into this inlet. Once engaged, the system will automatically feed the filament.
11. **Filament Release Button:** Press to release and remove the filament.
12. **Filament Outlet:** Connects the AMS to the printer's filament inlet.
13. **Power Port:** Connects the power cord to supply power to the device.

1.5 Printer and AMS Status

1.5.1 Printer Status Indicator



Indicator Light Modes	Meaning
White light breathing slowly (when the touchscreen is on)	The printer is idle and not executing any print jobs
The light turns off (when the screen sleeps)	
Orange light flows	The printer is preparing for a print job (such as uploading, changing filament, auto-leveling, or heating).
The white light stays on and shows a progress bar	The light displays the print progress.

Indicator Light Modes	Meaning
Red light flashing twice in repetition	A print error has occurred. The light flashes as a warning until the user closes the error prompt.
Green light stays on	Print completed. The indicator light remains green until the user closes the "Print Completed" prompt or opens/closes the printer doors, top cover, or side panels.

i TIPS

Please ensure that the printer status indicator is turned on. Operation path: **Printer touchscreen** >  > **Settings** > **Status indicator**, confirm the switch is set to "On".

1.5.2 AMS 2 Pro Status Indicator



The AMS indicator light shows the current status: white means normal operation; red means error or fault.

Normal Status LED Blink Patterns	Meaning
Four slots with white lights flashing in sequence	AMS is powering on and setting up; after setup completes, the white light turns solid
Four slots breathing white	Normal drying status

Normal Status LED Blink Patterns	Meaning
Four slots with solid white lights	Indicates that all four slots have filament inserted after setup completes. Only slots with filament light up when waking up
Solid white light stays on	This slot is performing preloading, reading, or printing operations (including loading and unloading)
White light off	Case 1: No feeding, reading, or printing within 15 minutes after the screen turns off Case 2: 10 seconds after preloading or reading is finished Case 3: No operation within 20 minutes after setup.
Error Status Blinking Patterns	Meaning
Four slots flashing red - double flash	Printer cannot detect AMS; communication error
Red light breathing	After printing starts, the filament inlet does not detect any filament; insert filament to resume normal operation.
Red light solid	Filament may be broken inside the AMS internal hub
Red light single flash	Generally indicates feeding failure; please check whether the filament inlet is functioning properly
Red light double flash	Case 1: Abnormality occurred during drying process Case 2: When idle, drying module NTC disconnects, or abnormal air duct temperature is detected; check all connections are secure

1.5.3 HMS Messages

HMS (Health Management System) is the built-in fault diagnosis and status monitoring system for Bambu Lab printers and AMS (Automatic Material System).

When a device experiences hardware failure, print failure, or needs maintenance, HMS will provide on-screen or app notifications and offer recommended solutions.

If an HMS message appears, you can use one of the following methods to locate and address the fault:

- Scan the QR code with your phone to directly access the diagnostic and troubleshooting page.
- Visit the Bambu Lab Wiki and search the HMS code to view detailed causes and troubleshooting steps for the fault.

TIPS

The printer evaluates the pollution level based on the task type and duration, provides targeted cleaning and maintenance reminders. This feature requires the firmware to be upgraded to the latest version. It is recommended to update the firmware before first use to enable cleaning reminders.

When an HMS alert appears, a notification icon  will display at the bottom right of the Home page, and a message alert  will appear on the left menu bar. Tap any notification icon to enter the HMS page and view detailed information.

Chapter 2 First Print

Before starting the first print, please ensure that the printer is correctly positioned, the initial set-up and automatic calibration are completed.

This chapter will guide you step by step to ensure the printer starts the first print in optimal conditions.

2.1 Preparation

2.1.1 Printer Placement

To ensure print quality and the safe operation of the printer, please keep your workspace clean and organized. Place the printer on a stable and sturdy surface.

CAUTION

The printer is equipped with anti-vibration feet to absorb minor vibrations during printing. Ensure the surface is stable to prevent the printer from shifting or falling during operation.

CAUTION

Calibration should be performed after the printer is properly placed. Moving the printer after calibration may affect print quality.

2.1.2 Initial Installation

Please follow the steps below to complete the initial installation of the printer:

1. Remove Packaging Materials

Follow the "Quick Start Guide" to remove the outer box, foam, moisture-proof bags, and internal fasteners, ensuring that there is no packaging on or inside the printer.

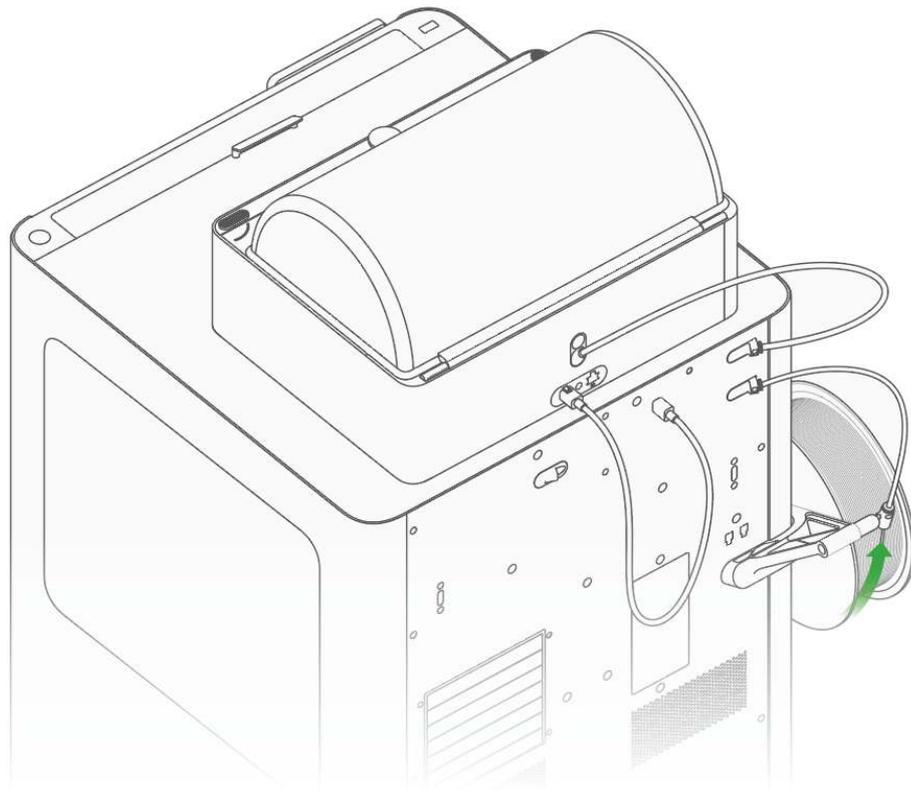
2. Inspect the Printer Components

- Confirm that all included accessories are present (accessory box, power cord, quick start guide, spool holder, PTFE tubes, etc.).
- Ensure the printer's standard **textured PEI plate** is correctly installed on the heatbed.

3. Install Filament Components

Select and install according to your chosen load method:

- If using the **AMS 2 Pro**, please refer to the AMS 2 Pro quick installation steps.

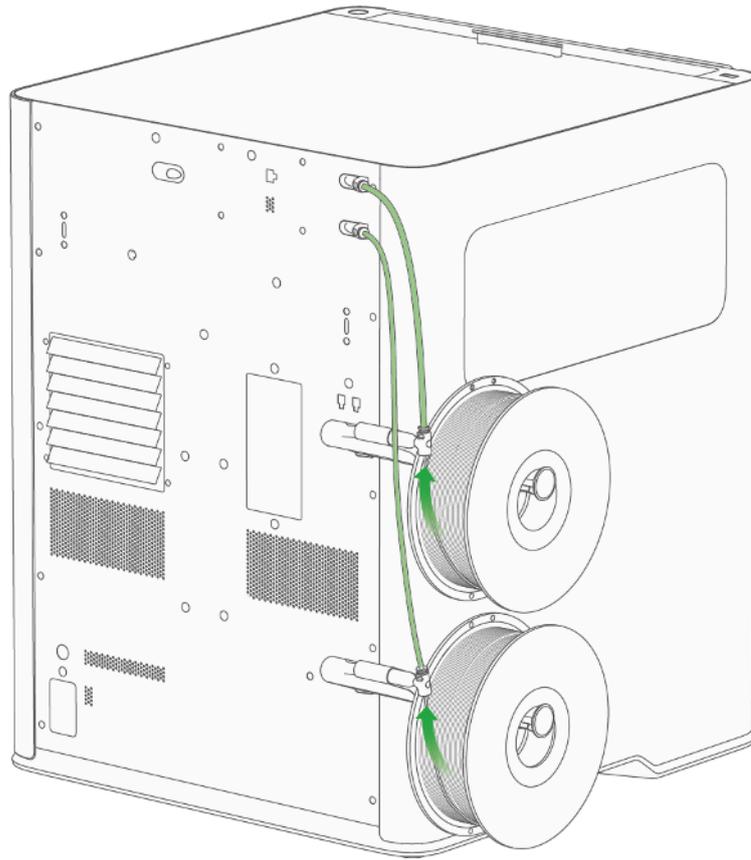


Scan the QR code or click the link to watch the installation video and jump to the relevant sections as needed.



<https://wiki.bambulab.com/h2/manual/unboxing-ams-combo-and-laser-full-combo>

- If using the **external filament spool**, make sure the filament holder and tubes are connected to the back of the printer as shown in the diagram.



Scan the QR code or click the link to watch the installation video and jump to the relevant sections as needed.



<https://wiki.bambulab.com/h2/manual/unboxing-h2d>

4. Check Power-On Status

- Ensure the power cord is properly connected.
- Confirm that the **power switch** on the back of the printer is turned on and the indicator light is lit.
- The **safety key** is inserted into the rear port of the printer.
- If an **emergency stop button** is installed, make sure it is not pressed (i.e., it is in the released position).
- The touchscreen should light up normally and display the initial setup interface.

2.1.3 Initial Calibration

Upon first startup, the printer will automatically enter the setup process. Please follow the on-screen instructions to complete the following steps:

1. Language, region, and network settings

Select the appropriate language and time zone; choose an available Wi-Fi network, or skip network selection if not required.

2. Account login

Open Bambu Handy and scan the QR code displayed on the touchscreen to bind the printer (see [Install and Bind Bambu Handy](#)). You may skip binding if not needed.

3. Automatic calibration

The printer will automatically perform a series of operations, including motor noise cancellation, vibration compensation, automatic bed leveling, and nozzle offset calibration.

4. Remove the foam under the heatbed

After calibration, the heatbed will rise. Remove the remaining foam.

5. Calibration completes

The touchscreen will display "Calibration Completed!" indicating you can start your first print.

CAUTION

When installing the printer, do not touch the build plate surface with your hands to prevent oils and sweat from contamination, which can reduce adhesion and cause bad print quality. If you accidentally touch it, we recommend cleaning the build plate with hot water and detergent to ensure optimal adhesion.

TIPS

Connect a USB flash drive to the printer's USB port. This allows you to import sliced files and start printing directly from the touchscreen, and save print videos, print history, and print cache.

2.2 Load Filament

2.2.1 External Spool Holder

Step 1. Confirm the filament type and color for subsequent setup. Then, place the spool on the spool holder according to the filament's winding direction.



- Step 2. On the touchscreen, tap  > **External Spool** > **Edit**, select the filament type and color, then tap **Confirm**.
- Step 3. Insert the filament into the PTFE tube and push it into the toolhead until it cannot move forward. At this time, a small green dot will appear at the toolhead on the touchscreen, indicating that filament has been detected.
- Step 4. On the touchscreen, tap **Load**. Wait for the hotend to heat up. Manually push the filament again to keep it inside the extruder. Then follow the on-screen instructions to complete the process.

2.2.2 AMS 2 Pro / AMS HT

The AMS HT loading operation is the same as the AMS 2 Pro; the following instructions use the AMS 2 Pro as an example to explain the steps.

AMS Setup

When the printer detects a new AMS 2 Pro connection, the touchscreen will prompt to perform AMS Setup. The purpose of this operation is to determine which hotend the AMS 2 Pro is connected to. There are two AMS 2 Pro setup modes.

- Auto AMS Setup: Load a spool of filament into the AMS 2 Pro, then click **Detect** to start the setup process. The AMS 2 Pro will complete the setup automatically.

TIPS

Insert a roll of filament into each AMS 2 Pro. If filament is already loaded, please unload it first. Ensure that no filament remains in the filament buffer to prevent it from breaking or jamming.

- Manual Mode: If no filament is available for auto AMS setup, click **Manual Setup** on the touchscreen to configure the connected AMS 2 Pro to the left or right hotend.

TIPS

The upper PTFE tube coupler corresponds to the right hotend, and the lower PTFE tube coupler corresponds to the left hotend on the printer.

Load Filament

Step 1: Place the required filament into the AMS 2 Pro slot, then gently push the filament into the PTFE tube coupler.

Step 2: After recognizing filament insertion, the system will automatically identify the filament.

Step 3: On the printer touchscreen, select the desired filament and click **Load**. You can also perform the loading operation by selecting the corresponding filament in Bambu Studio's device interface.

2.3 Initiate a Print from Printer Touchscreen

The printer includes built-in model files that allow you to initiate your first print directly from the touchscreen.

CAUTION

These models and those stored on USB are pre-sliced files. Therefore, the relationship between filament and hotends cannot be changed. If the required filament is not found on the interface, please adjust the location of the filament in the AMS or external spool holder, so that the filament corresponds to the designated hotend.

Step 1. On the touchscreen, tap  > **Print Files** and select the model you want to print.

TIPS

To adjust calibration items, tap Advanced to set manually; all calibration items default to automatic.

Step 2. Select the matching filament and tap **Print**.

TIPS

You can also start printing from Bambu Handy or Bambu Studio. See [Initiate a Print from Bambu Handy](#) or [Initiate a Print from Bambu Studio](#).

2.4 Remove the Model after Printing

- Step 1. After printing finishes, wait until the heatbed and model have cooled down to room temperature. Then, gently remove the model from the build plate to prevent deformation or damage to the model or build plate.
- Step 2. If the model is difficult to remove, gently bend the build plate or use a scraper to help detach the model.



- Step 3. Remove the prime tower (if present), then carefully use a scraper to clear the pre-extrude line.
- Step 4. Put back the build plate and make sure it aligns with the heatbed's stoppers.

CAUTION

Do not touch the surface of the build plate with your hands to prevent oils or sweat contamination, which can reduce adhesion and cause bad print quality. If you accidentally touch it, we recommend cleaning the build plate with hot water and detergent to ensure optimal adhesion.

2.5 Filament Unload

2.5.1 External Spool Holder

Step 1. Tap  > **External Spool** > **Unload** on the touchscreen.

Step 2. Follow the instructions to pull back the filament after it exits the toolhead and rotate the spool at the same time.

Step 3. When the filament nears the pneumatic connector, catch the filament by hand and insert it into the hole of the spool. Then tap **Resume (problem solved)**.

2.5.2 AMS 2 Pro / AMS HT

After the print job completes successfully, the filament will automatically retract back into the AMS 2 Pro / AMS HT.

If the print job is cancelled or interrupted midway, tap  > **AMS filament** > **Unload** on the touchscreen to retract the filament gripped by the extruder back into the AMS 2 Pro / AMS HT.

2.6 Waste Disposal

Waste refers to the excess filament extruded to clear residual filament inside the nozzle at the beginning or during multi-color, multi-material printing, to ensure print quality. This waste is discharged through the purge chute located on the back of the printer.

We recommend following the steps below to collect and dispose of the waste:

- Use a container to catch the waste during printing. A larger container allows you to empty it less frequently.
- Make sure waste can pass successfully through the purge chute and enter the container.
- Before printing, make sure there is no waste left inside the purge chute.
- Regularly clean the waste container to avoid blocking the purge chute.
- During multi-color printing, the AMS frequently changes filaments; the longer the print job, the more waste is generated. Please empty the container periodically during printing if necessary.
- Avoid installing covers or pipes on the back of the printer that might block the purge chute to prevent clogging or print failure.
- Dispose of waste in accordance with the regulations of your country or region.

Chapter 3 Access 3D Model Resources

3.1 MakerWorld Model Community

MakerWorld (makerworld.com) is Bambu Lab's official 3D model sharing platform, bringing together a vast community of talented creators who upload high-quality models spanning everything from art, design, function, and engineering. Whether you are a creative artist or an engineer solving real-world problems, you can find models that meet your needs here. Additionally, users can exchange experiences, share ideas, and learn from each other.

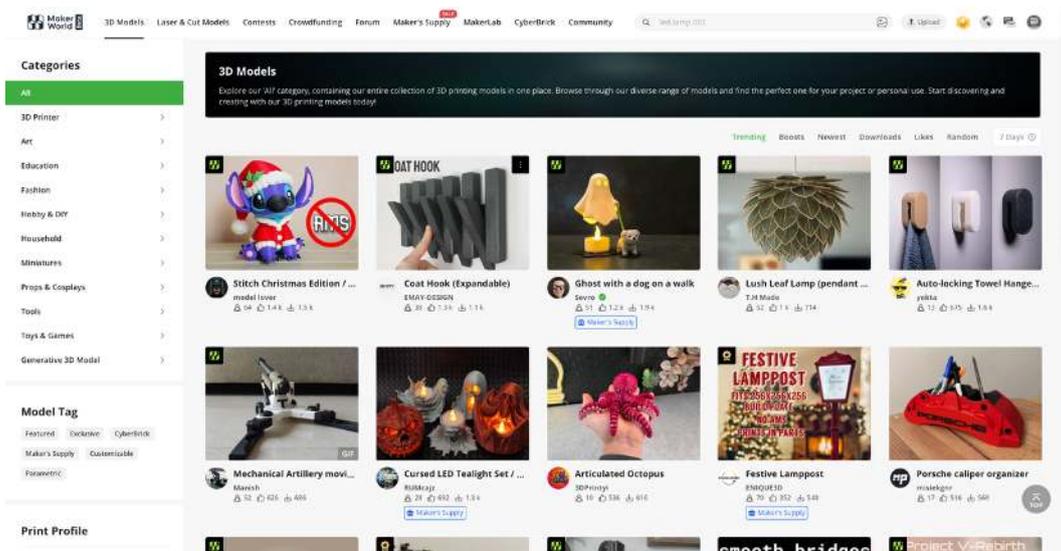
Through MakerWorld, you can:

- Browse curated models and popular community recommendations.
- Download model files in STL, CAD, or 3MF formats.
- Send models directly to Bambu Studio for slicing and printing.
- Upload and share your creative designs, and engage with the community.

MakerWorld is available on the web and directly integrated into Bambu Handy and Bambu Studio for seamless model browsing and printing.

NOTE

- 3MF files: Contain all print parameters and color information, enabling you to start printing directly in Bambu Studio.
- STL and CAD files: only contain 3D geometry. Print parameters or color information are not included; you must set these in your slicer or printer software.



3.2 MakerLab Creative Tools

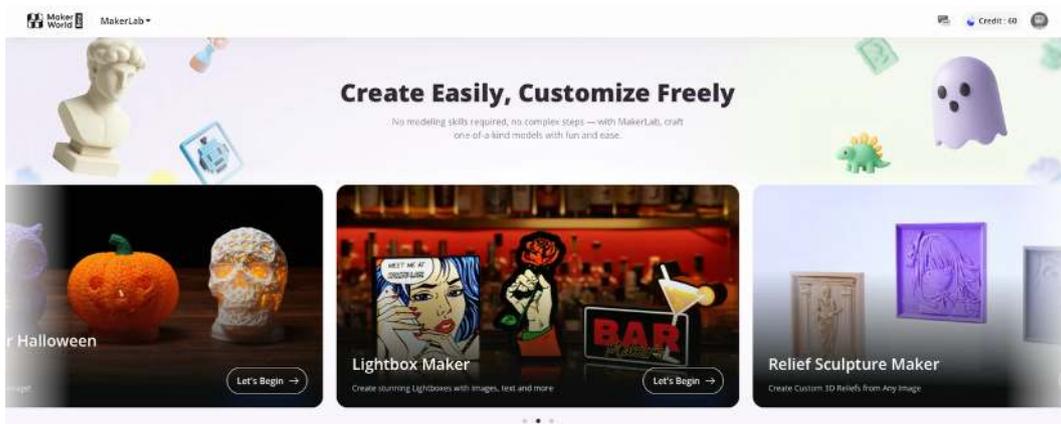
MakerLab provides a range of creative tools that help turn your ideas into printable models with ease. No modeling experience required, easily create personalized models in minutes.

Creative tools include generators for light boxes, flexi toys, relief sculptures, vases, and more, alongside AI-generated and parametric model tools that are accessible via the web.

MakerLab is integrated into both Bambu Studio and Bambu Handy, enabling seamless model generation and printing.

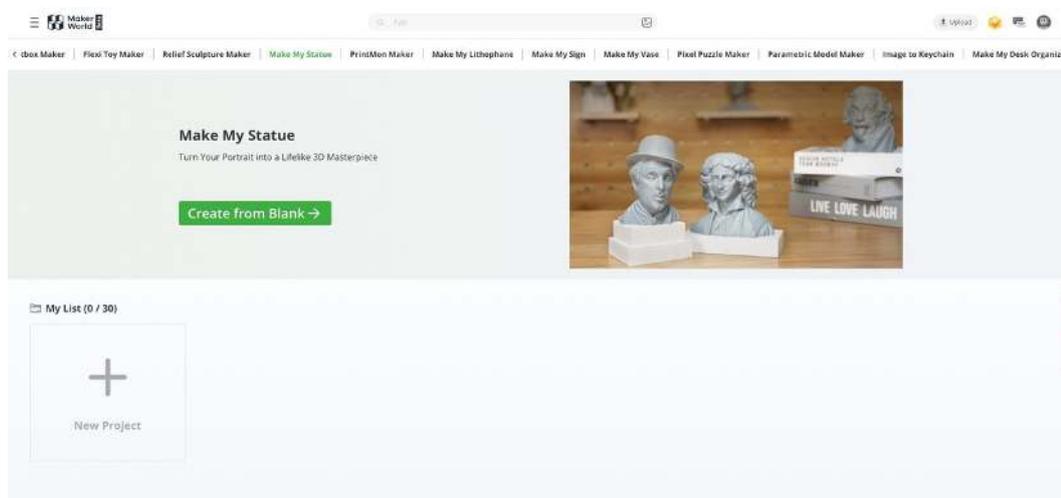
NOTE

Bambu Handy supports only some of the tools. For full feature access, use a desktop browser or Bambu Studio.



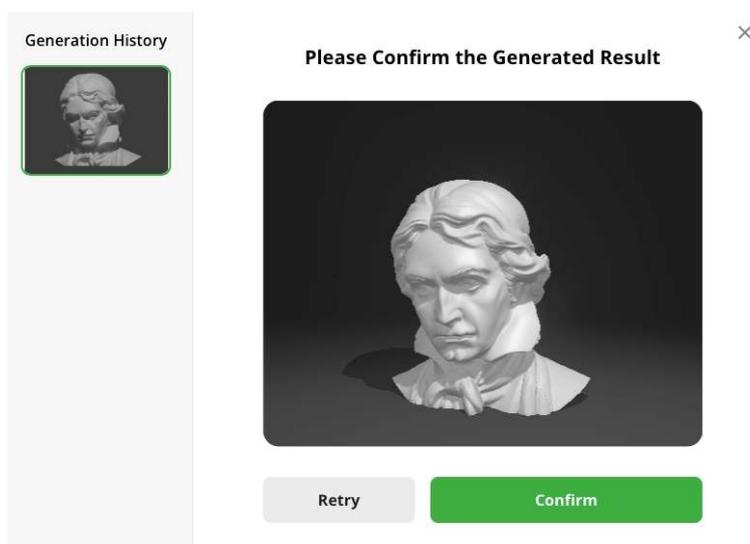
Each creative tool comes with detailed instructions. Please follow the instructions. The following example demonstrates the usage process on the web using the Make My Statue (statue generator):

Step 1. On the MakerWorld homepage top bar, click **MakerLab > Make My Statue > New Project**.



Step 2. Upload an image as prompted, click confirm, and wait for the model to be generated.

Step 3. If you are satisfied with the result, click Confirm to download the model. If not satisfied, click Retry to regenerate.



Step 4. Open Bambu Studio, click **File > Import**, and select the model file. After importing into Bambu Studio, you can slice and start printing.

Chapter 4 Initiate a Print from Bambu Handy

4.1 Install and Bind Bambu Handy

Bambu Handy is an integrated mobile application designed specifically for Bambu Lab 3D printers. With this app, you can search and print models with one tap, remotely monitor and manage print jobs, quickly reuse past projects, and adjust settings flexibly during printing.

Before installing Bambu Handy, please ensure the following:

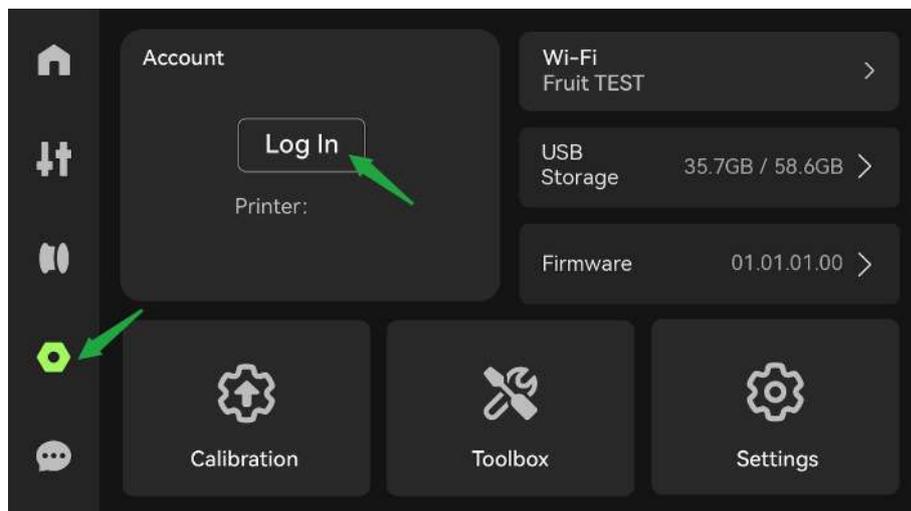
- The printer and your phone are connected to the same network.
- The selected region for the printer matches the region of the Bambu Handy app version.

Step 1. Visit bambulab.com/download or search for **Bambu Handy** in your mobile app store to download and install the app.

Step 2. Open Bambu Handy, read and agree to the Privacy Policy and User Agreement, then enter the home page.

Step 3. On the **Me** page, tap **Sign In / Sign Up**, enter your email address and verification code to complete registration.

Step 4. On the printer touchscreen, tap  > **Log in** and a QR code will appear on the screen.



Step 5. On the Bambu Handy **Devices** page, tap **+ Bind Printer**, then scan the QR code displayed on the printer touchscreen.



Step 6. Review and accept the Terms and Conditions and Privacy Policy, then tap **Confirm to Bind**.

Step 7. Name the printer, then tap **Confirm** to complete the binding process.

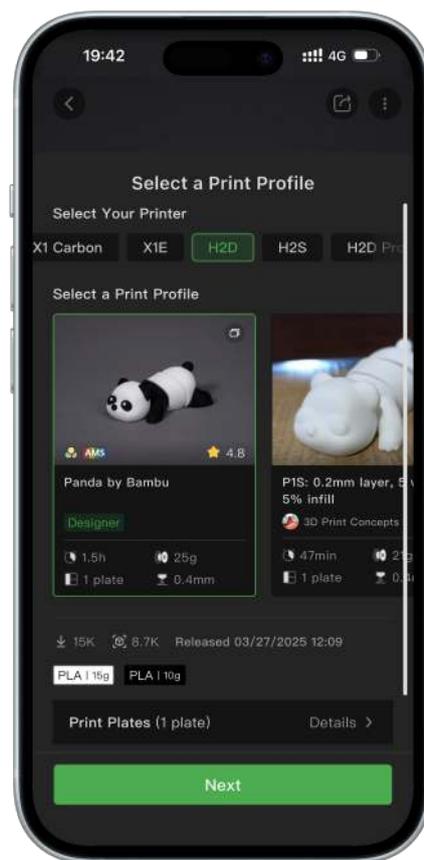
4.2 Initiate a Print

Step 1. On the **Models** page, select the model to print and tap **Prepare to Print**.

Step 2. Select your printer model and print profile, then tap **Next** to enter the preparation page.

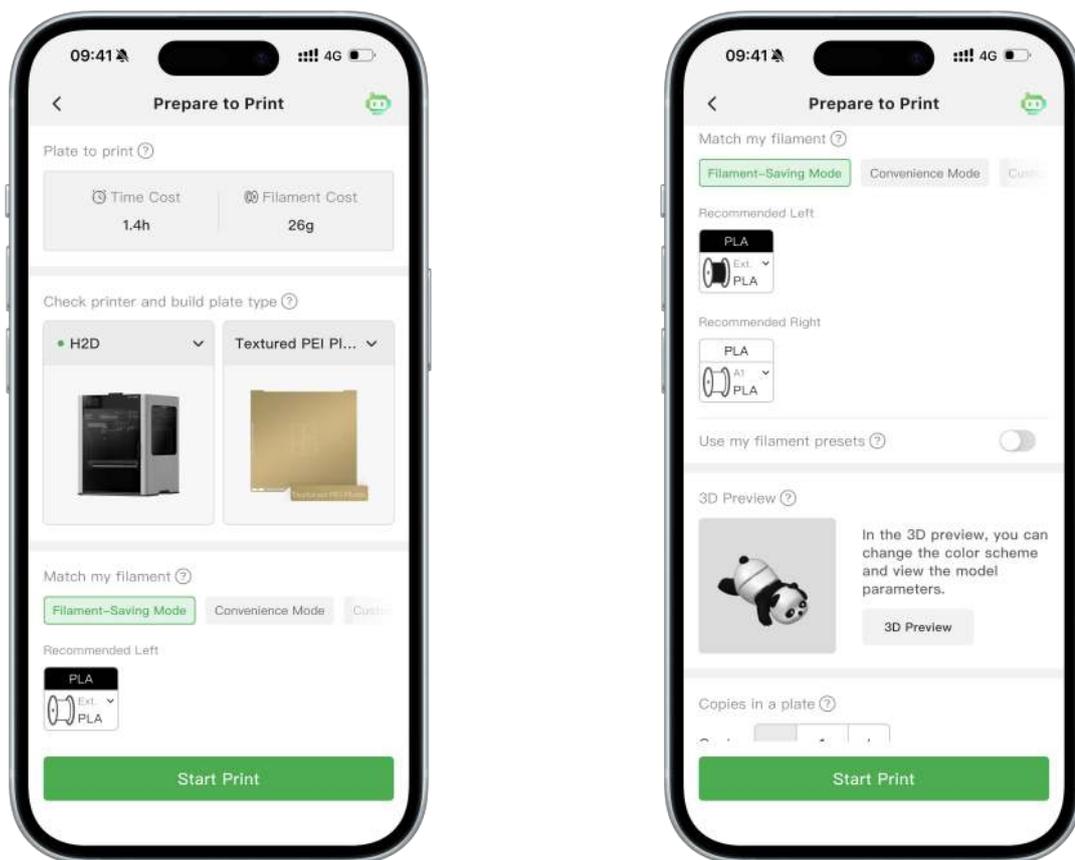


Model Details



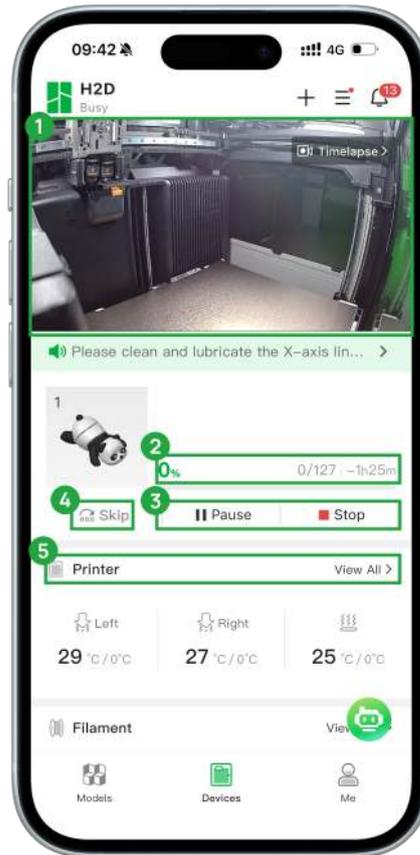
Print Profiles

Step 3. Check the printer model and the build plate type, verify the filament grouping strategy, set the print copies and print options, then tap Start Print.



Confirm printer model and build plate type Set filament grouping and print options

Step 4. Monitor or adjust the print job on the **Devices** page.



1. Camera View: Remotely view live feed of the printer to monitor print status.
2. Progress Bar: Displays current print layer and estimated remaining time to track print progress.
3. Task Controls: Pause or stop the print in real-time.
4. Parts Skip: When printing multiple parts, if a model collapses or fails to print correctly, use this feature to skip the current model part and continue printing the remaining models.
5. Printer Settings:
 - Print Speed: Choose from Silent, Standard, Sport, and Ludicrous.
 - Heatbed Temperature: Monitor and adjust the heatbed temperature.
 - Light: Turn the printer chamber light on or off.
 - Part Cooling Fan: Adjust the cooling fan speed based on the filament characteristics and model structure.

Chapter 5 Initiate a Print from Bambu Studio

5.1 Install and Bind Bambu Studio

Bambu Studio is the official slicing software developed by Bambu Lab, featuring customized functions designed specifically for Bambu 3D printers. It offers a project-based workflow, optimized slicing algorithms, and an intuitive graphical interface to deliver a smooth printing experience. Before 3D printing, the model must be sliced, which means converting the 3D model into instructions recognizable by the printer. This is a critical step in turning your creative design into a physical printed product.

To smoothly install and run Bambu Studio, please ensure your computer meets the following requirements:

- Operating system: Windows 10, Mac OS X 10.15, Ubuntu 20.02, Fedora 36, or later.
- Processor: Intel® Core 2 or AMD Athlon® 64; 2 GHz or higher.
- Memory: Minimum 4 GB RAM; 8 GB RAM or higher recommended.
- Storage: At least 2 GB of available space.
- Graphics support: OpenGL 2.0 compatible.

Step 1. Download Bambu Studio.

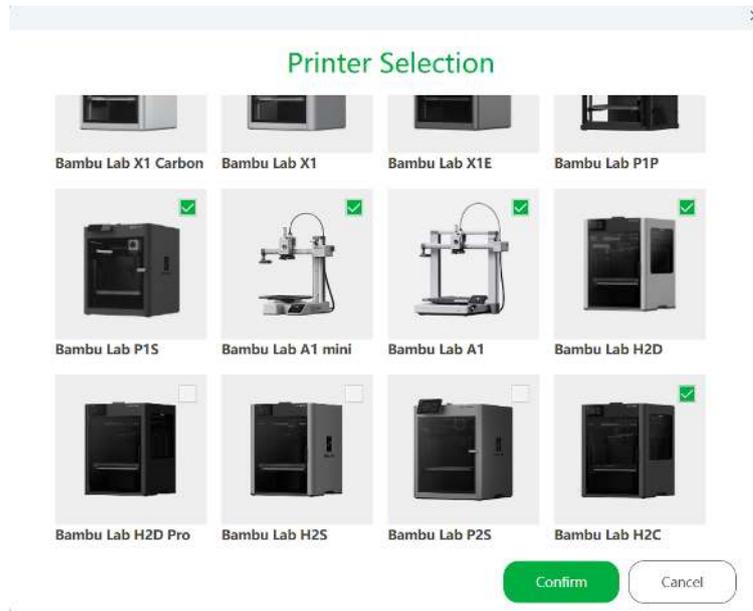
- Windows and MacOS versions: bambulab.com/download
- Linux version: github.com/bambulab/BambuStudio/releases

Step 2. Double-click the downloaded .exe file and follow the instructions to complete the installation and open the software.

Step 3. Select your region and click **Next**.

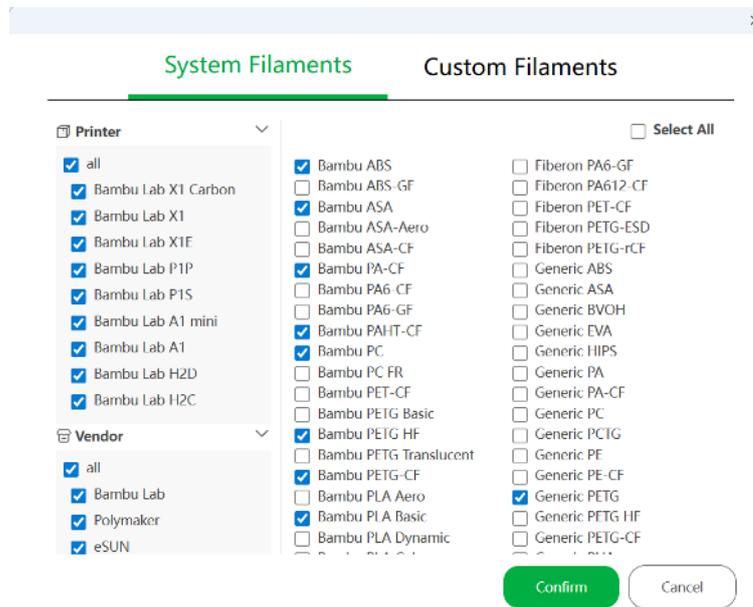
Step 4. Read and choose whether to join the Customer Experience Improvement Program.

Step 5. Select the printer model and nozzle size preset you will use, then click **Next**. The selected preset will be used later to generate suitable print paths during slicing.



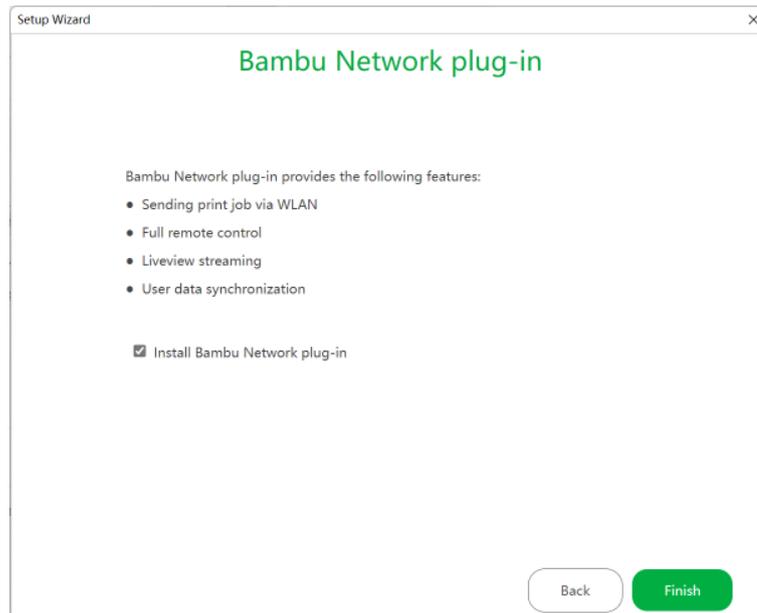
Step 6. Choose the filament preset, then click **Next**.

Filament presets include parameters such as print temperature. Once a filament is selected for future prints, all printing parameters will be applied without the need for manual adjustments.



Step 7. Check **Install Bambu Network Plug-in**, then click **Finish** to start the installation.

This plugin supports printing over local networks or the internet and enables remote control and user data synchronization.

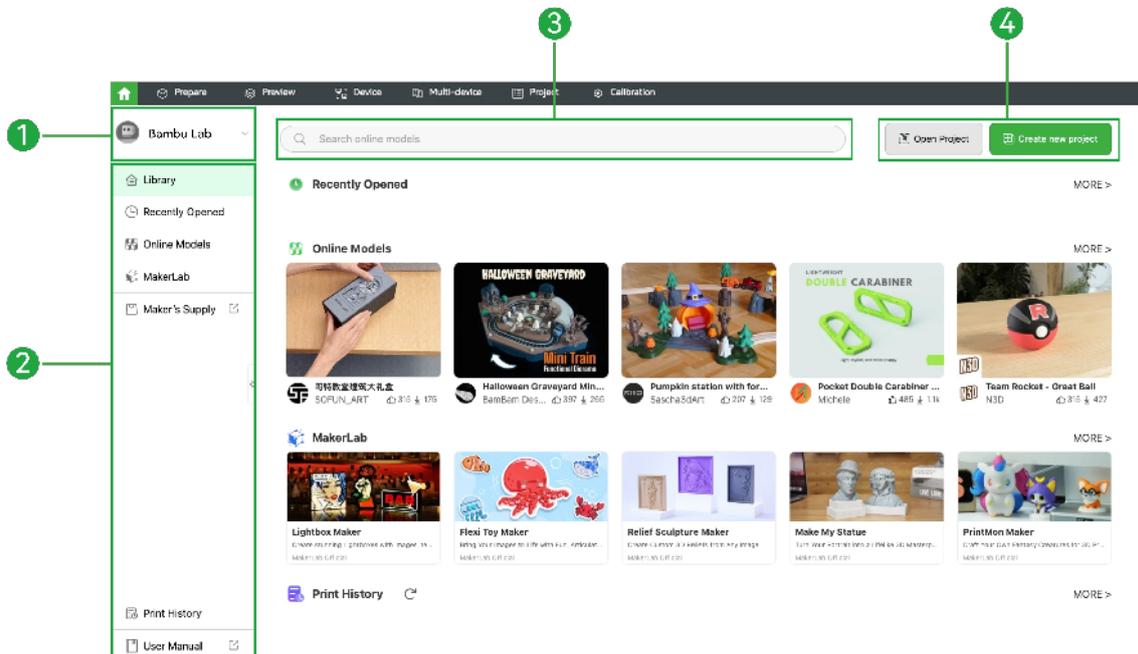


Step 8. After logging in, printer information will be automatically synchronized across all your devices.

5.2 Introduction to Bambu Studio Interface

Home

The Home interface integrates model management, resources, and learning content. This is the default page when opening Bambu Studio.

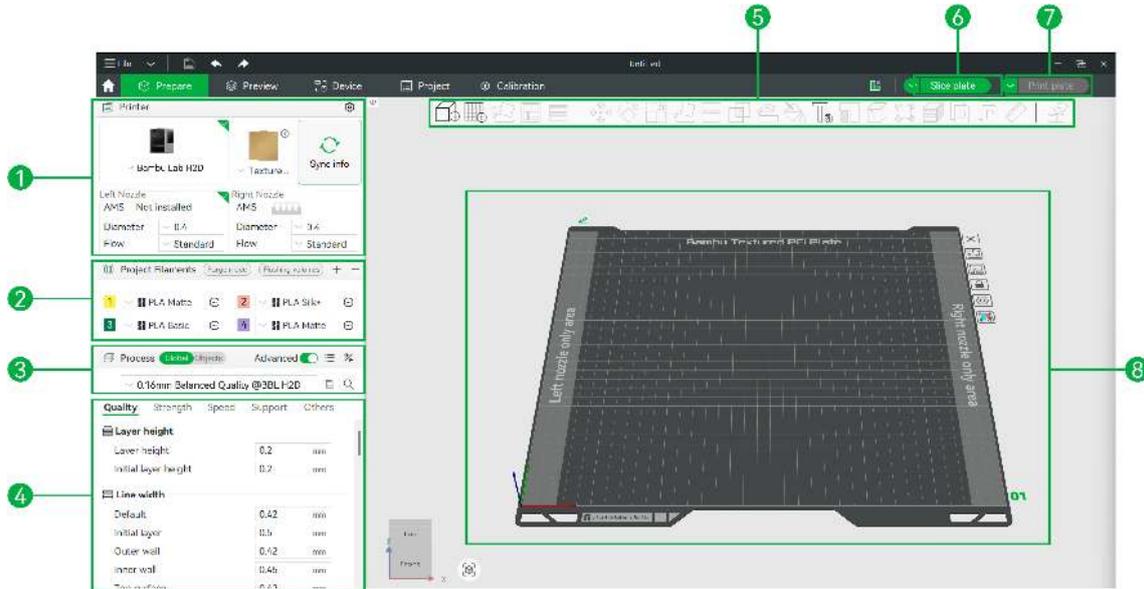


1. Account information: View the current account or log out.
2. Page navigation: Quickly access recently opened files, online models (MakerWorld), MakerLab, Maker's Supply, print history, and user manuals.

3. Search bar: Enter keywords to search online models inside MakerWorld.
4. Project operations: Open local projects or create new ones.

Prepare

This interface is used to select the printer model, filament type, print settings, and to slice models and send print jobs.



1. Printer settings: Bambu Studio slices and generates suitable print parameters based on these settings.
2. Filament settings: Select the filament type and color used for printing. Preset parameters are loaded automatically.
3. Print presets: Provides multiple quality-level presets (such as Standard, Fine, etc.) ready to use.
4. Print parameter details: Models downloaded from MakerWorld usually include all print parameters recommended by the creator. Simply select the printer model to slice. Parameters can be manually adjusted to fulfill custom applications, such as lowering layer height to reduce layer lines (see [Adjust Slicing Parameters](#)).
5. Top toolbar: Provides multiple options for viewing and modifying the model.
6. Slice button: After selecting the slicing mode and filament grouping mode, click Slice Plate to generate a preview of the printed parts and their G-code. (see [Dual Hotend Printing](#)).
7. Print and export: Initiate printing or export files. Click to choose from the following options.
 - Print plate: To print the currently selected plate.
 - Print all: To print all plates.

- Export plate/all sliced file: Export sliced files of the currently selected plate/all plates to printer storage.
- Send/Send all: Send sliced files of the currently selected plate/all plates to the printer's USB drive. The print job can then be initiated from the printer screen.

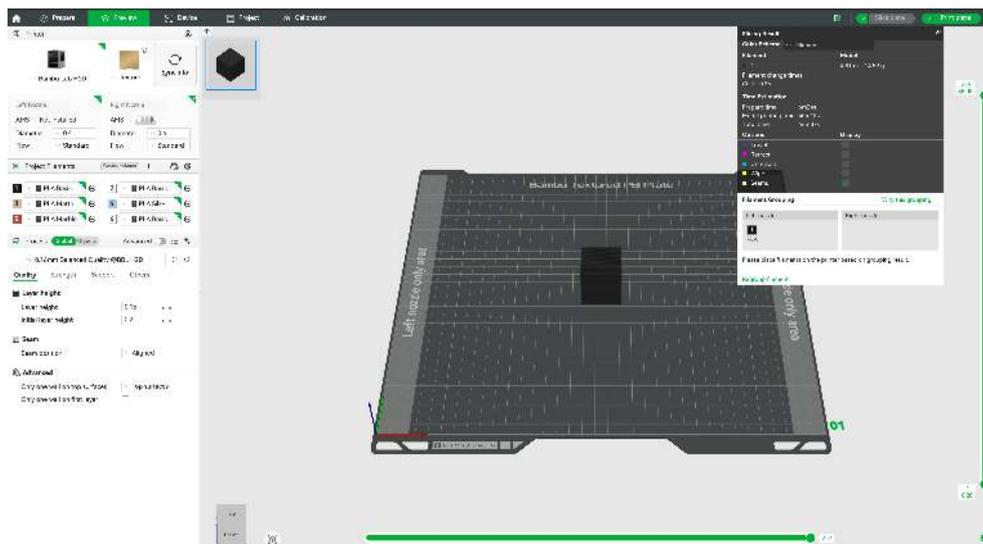
NOTE

This operation requires the Bambu Network Plugin installed, and Bambu Studio and the printer must be on the same network.

8. Print plate: Designated location for placing and editing models.

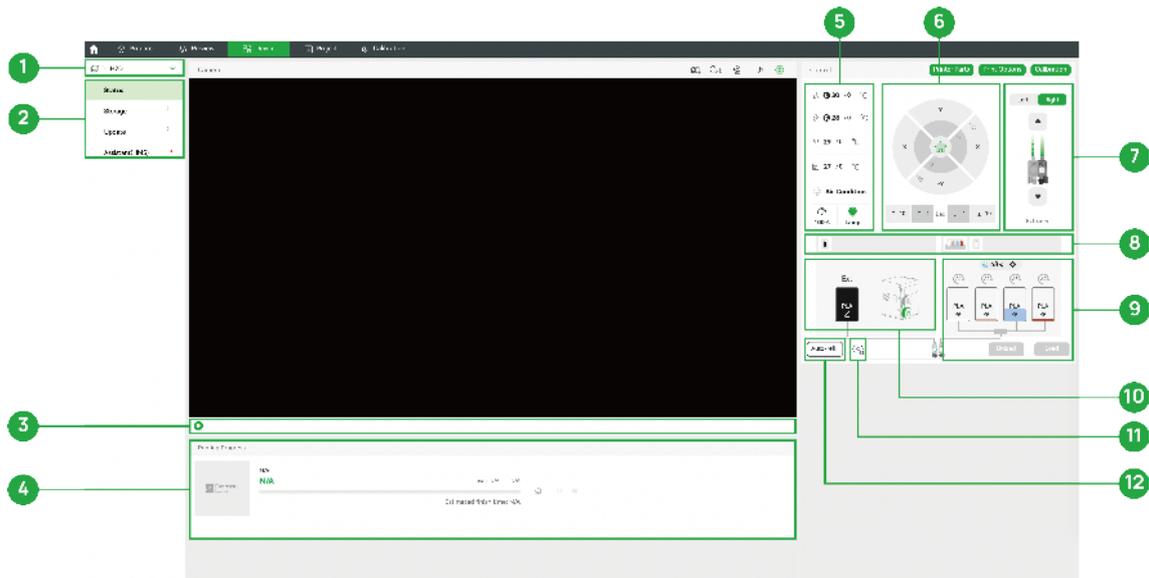
Preview

This interface displays detailed information about the sliced models, such as line types, filament, print speed, and print paths. At the same time, custom actions can be added at specific layers, including custom G-code, print pauses, and filament changes.



Device

This interface shows the status and operation options of the currently connected printer.



1. Connected device information. If you have multiple printers, click the printer name to switch between devices.
2. View various printer information, including device status, storage media, firmware updates, and Health Management System (HMS). This section enables real-time monitoring, device control, USB drive file access, and firmware updates.
3. Play/pause live video and view playback status.
4. Display information about the current print job, including job thumbnail and print progress. You can skip objects, pause, start, or stop the job.
5. Adjust the air condition and control temperatures of the hotends, heatbed, and chamber. The left value indicates the current temperature, while the right value indicates the target temperature.
6. Control toolhead and heatbed movement when the printer is idle.
7. Displays the loading status of the left and right extruders. A green dot indicates the filament is loaded. Control the extruders by clicking the up/down arrows to manually extrude or retract 1 cm of filament.
8. Toggle display between AMS and external spool and .
9. View AMS humidity, filament usage, and other information. Edit, load, or unload filament within the AMS.
10. View and edit information of the external spool.
11. Enable or disable AMS-related features, including insertion update, update on startup, update remaining capacity, and AMS filament backup.
12. After enabling the AMS filament backup, click to view filament refill relationships.

5.3 Download and Import Model

MakerWorld hosts a wide variety of high-quality models uploaded by creators, including both model files and complete print configurations. The creators predefine the print parameters, enabling you to download and send the file directly to the printer with a single click, eliminating the need for further tuning.

5.3.1 MakerWorld Models

- Step 1. Select the model you want to print on the MakerWorld website or within Bambu Studio's online models, then go to the model details page.
- Step 2. Choose a print profile that matches your printer model and printing requirements.
- Step 3. If you are using MakerWorld on a browser, click **Open in Bambu Studio**; if you are using Bambu Studio, click **Download and open**.
- Step 4. Bambu Studio will automatically download the model and open it in the Prepare interface.

5.3.2 Other Models

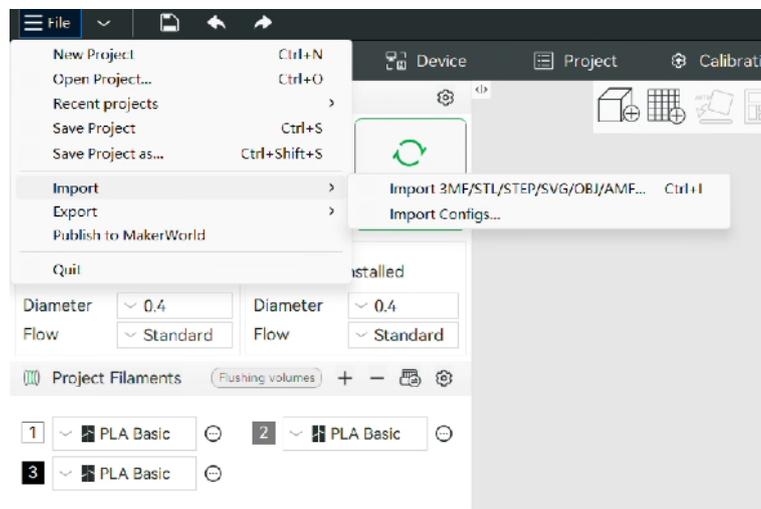
- Step 1. Prepare the model file.

NOTE

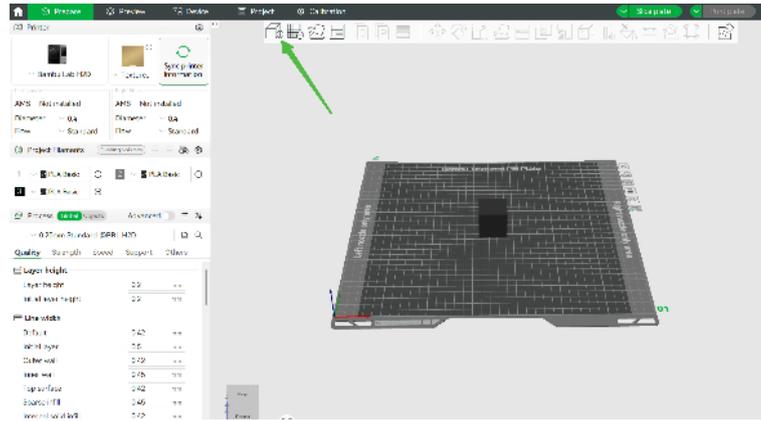
Bambu Studio supports the following file formats: .3mf, .stl, .stp, .step, .amf, .obj.

- Step 2. Import the model using your preferred method.

1. Select **File > Import > Import 3MF/STL/STEP/SVG/OBJ/AMF**



2. In the Prepare interface, click  in the top toolbar, select the file, then click **Open**.



3. Drag and drop the model file directly into the Prepare interface.

Step 3. After successful import, the model will automatically load onto the plate, ready for pre-viewing, editing, or slicing.

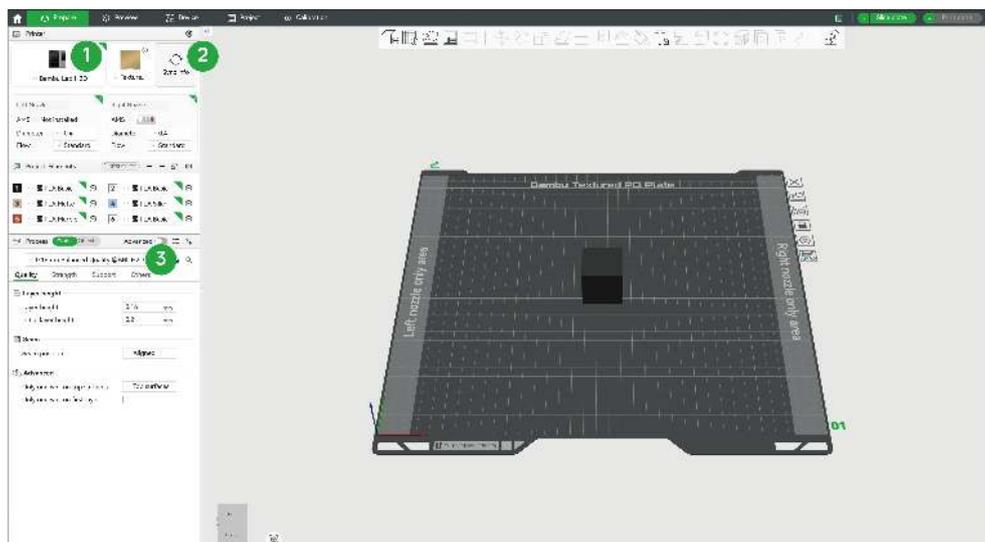
5.4 Initiate a Print

This section introduces the complete printing process after importing the model into Bambu Studio.

Step 1. Select the correct printer model.

Step 2. Click **Sync info** to sync nozzle and filament information.

Step 3. Select the desired preset from the **Process** drop-down list. **Smaller layer height results in longer print time. For most models printed with a 0.4 mm nozzle, a 0.2 mm layer height provides a good balance between print quality and efficiency.**



Step 4. Adjust slicing parameters.

NOTE

Process presets include appropriate default parameters suitable for most print tasks. You can adjust them based on your actual needs or desired results.

Step 5. After completing all settings, slice the model to view color schemes, filament usage, print time, and other information in the Preview interface.(See [Dual Hotend Printing](#))

Step 6. Send the print job. You can send jobs to the printer through the following methods. Confirm the filaments used by the left and right extruders, configure advanced options if needed, then click **Send**.

NOTE

Timelapse recording is off by default, while other print calibrations are set to auto. To use high-precision nozzle offset calibration data, please disable the nozzle offset calibration when sending the print job.

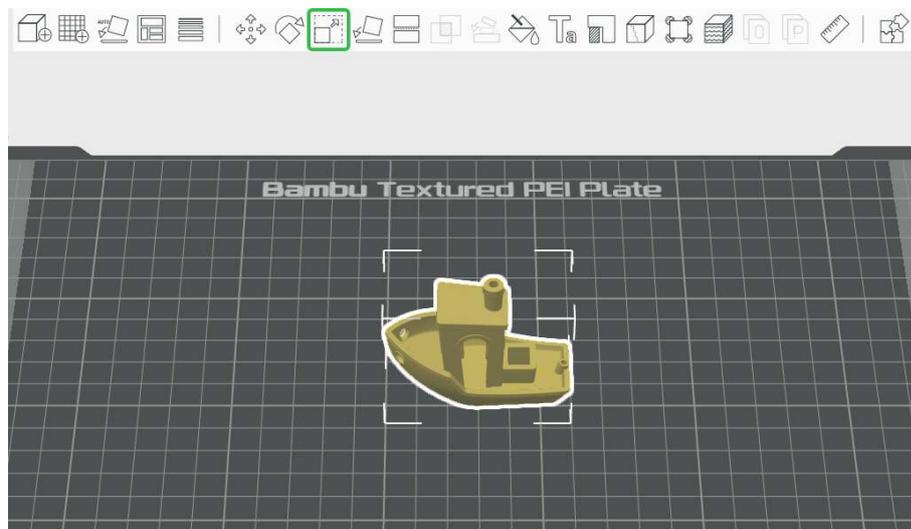
5.5 Adjust Slicing Parameters

5.5.1 Model Size

You can adjust the model size according to your printing requirements.

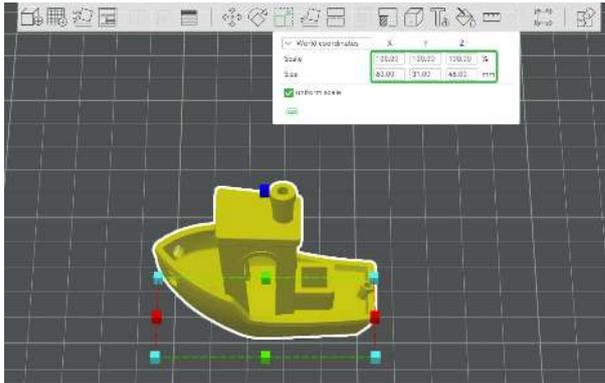
Step 1. Click the model you want to resize to activate the top toolbar.

Step 2. Click the Scale tool.

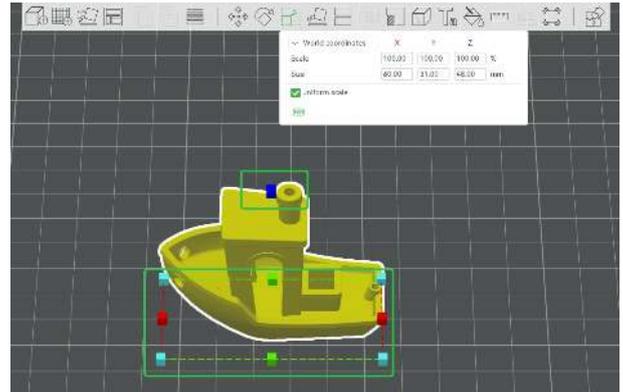


Step 3. Adjust the model size.

- Enter a percentage or an exact value for the X, Y, and Z axes. When **Uniform Scale** is checked, modifying any value automatically adjusts the other two proportionally. When it is unchecked, each axis can be adjusted independently.
- Drag the control points on the model. Dragging the control points at the four bottom corners scales the model uniformly; dragging other control points stretches the model along a single direction.



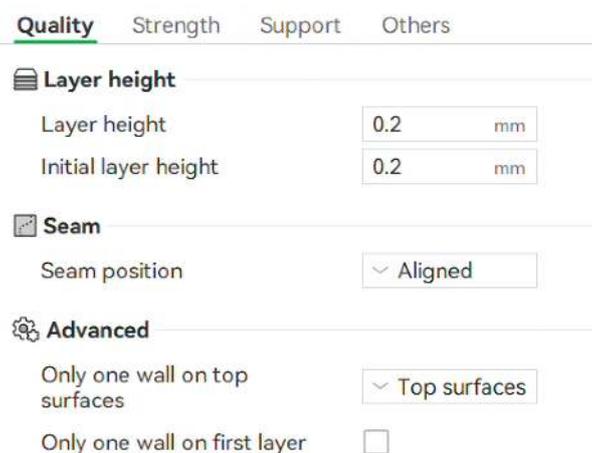
Modify values



Drag control points

5.5.2 Quality Settings

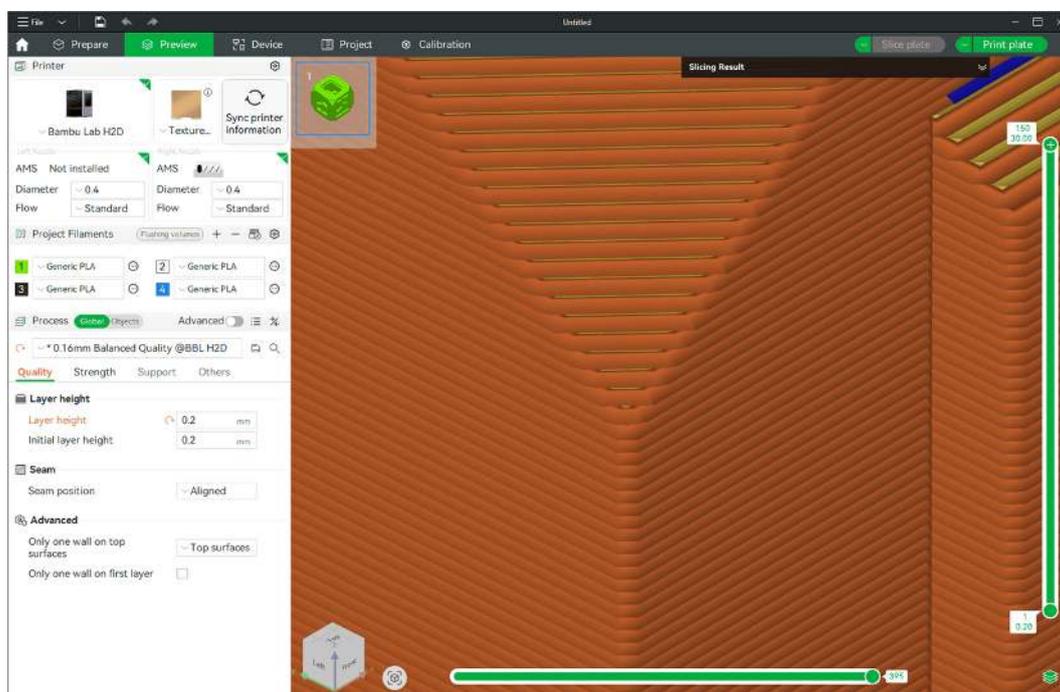
Print quality parameters directly affect the model's level of detail and overall appearance. Properly configuring these parameters helps achieve an optimal balance between printing precision and efficiency.



Layer Height

Layer height refers to the vertical stacking height of each printed layer (unit: mm) and is a key parameter determining print **resolution** and **efficiency**.

For example, with a **0.4 mm** nozzle, a smaller layer height (such as 0.12 mm) achieves finer layers, resulting in smoother surfaces and richer details, but significantly increases print time. In comparison, a larger layer height (such as 0.28 mm) decreases print time but may produce noticeable layer lines affecting surface quality.



0.08 mm layer height
print time: 1h 19m



0.2 mm layer height
print time: 36m 14s



0.28 mm layer height
print time: 28m 27s

Recommended settings:

- Layer height is typically **50%** of the nozzle diameter, with a recommended range of **30% to 70% of the nozzle diameter**. For instance, when using a **0.4 mm nozzle**, the recommended layer height range is **0.12 mm – 0.28 mm**.
- When optimizing layer height, consider model complexity, required strength, and nozzle diameter to balance surface quality and printing efficiency.

Initial Layer Height

The height of the first printed layer of the model. Increasing the initial layer height enhances adhesion between the model and the build plate, reducing the risk of warping or detachment.

For more details on layer height settings, please visit the Bambu Lab Wiki (wiki.bambulab.com/home) for relevant guidance.

Seam

A seam is a small gap formed where the start and end points of wall paths meet on the model surface, usually appearing as a vertical line. This phenomenon is an inherent structural characteristic that cannot be completely avoided in FDM 3D printing.

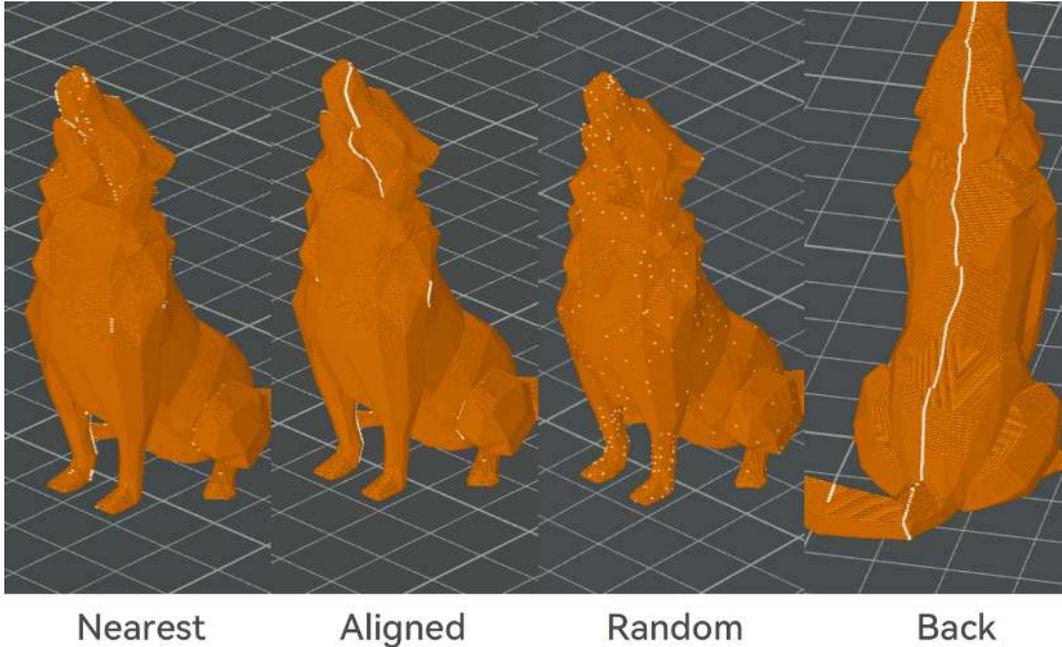
On models with edges or protrusions, seams can be naturally concealed. However, on smooth curved or continuous surfaces like cylinders, seams are more noticeable and can affect the external appearance.



To achieve better print results, you can set the seam position in Quality parameters. There are four types of seam positions to choose from:

- **Nearest:** Prioritizes concave or convex non-overhang vertices to hide seams, especially suitable for models with sharp corners. If no suitable vertices are found, it selects positions close to the end of the previous path to reduce travel distance and oozing.
- **Aligned:** Aligns seam across layers, making them more concentrated and easier to post-process.
- **Random:** Distributes seams randomly on each layer to avoid visible lines, but may cause irregular surface patterns.

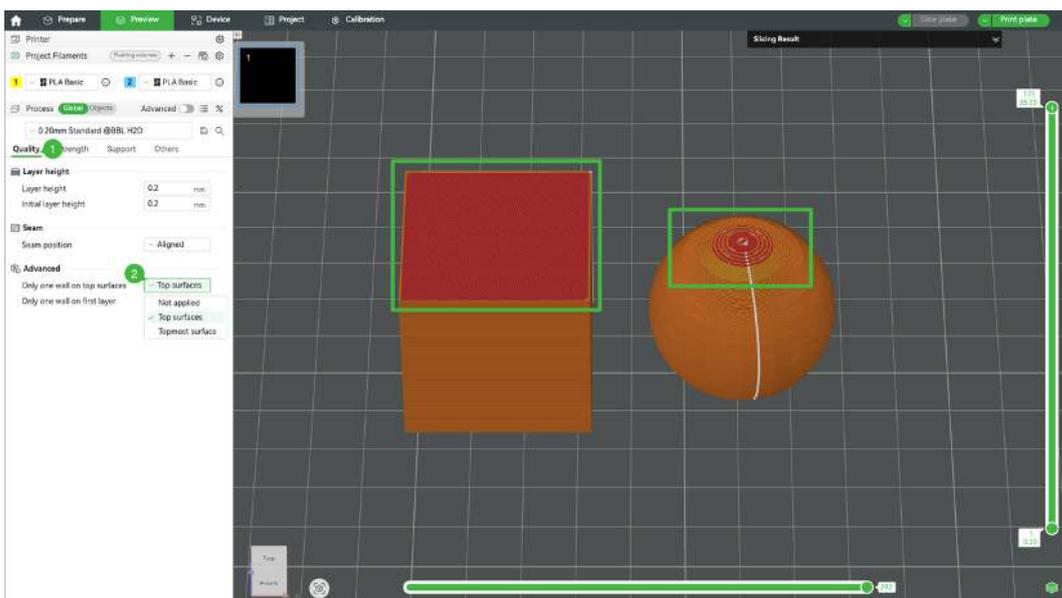
- **Back:** Fixes seams on the back side of the model, suitable for display models.



For more details on seam settings, please visit the Bambu Lab Wiki (wiki.bambulab.com/home) for relevant guidance.

Only One Wall on Top Surfaces

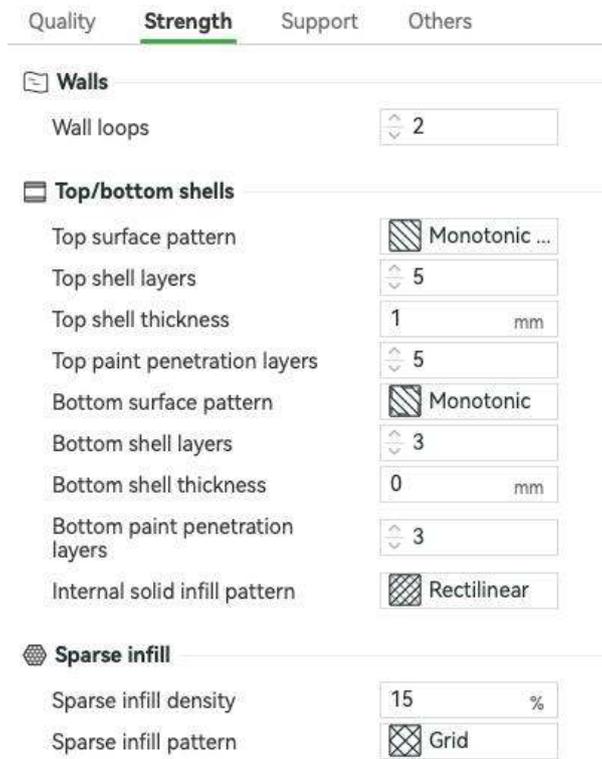
When enabled, the top surface of the model is printed as a single wall, helping achieve clean and neat surfaces. For models with flat top structures (such as cubes), this results in a smooth and neat finish. However, for models with curved top surfaces (such as spheres), it may cause noticeable layer lines that affect surface detail.



When printing curved top surfaces, it is recommended to select **Not Applied** to obtain better surface quality and smoothness.

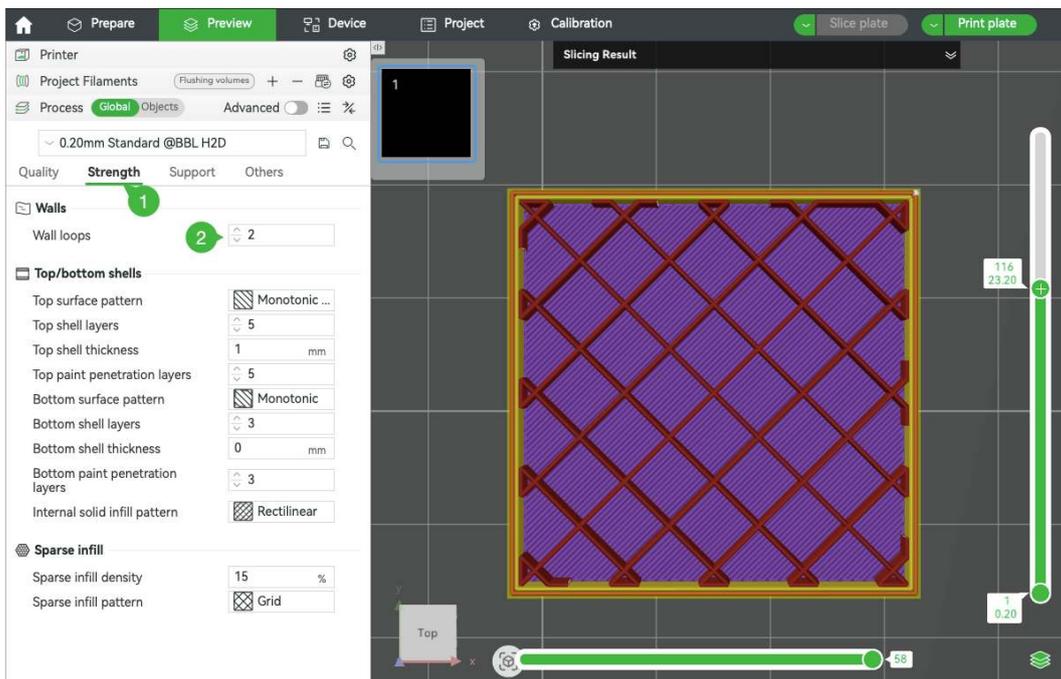
5.5.3 Strength Settings

Strength-related parameters directly affect the model's structural integrity and durability.



Wall Loops

This parameter determines the thickness of the model's outer walls, directly impacting overall structural strength and surface quality. Outer and inner walls are typically differentiated by color (in Bambu Studio preview) as shown below, orange represents the outer wall and yellow represents the inner wall.



Recommended settings:

- Functional parts: Set to 3–4 walls to improve structural strength and durability.
- Decorative models: Set to 2 layers to save material and improve efficiency.

Top Shell Layers and Thickness

Top shell layers determines the number of solid layers at the top of the model, typically including the outermost top surface, internal solid infill, and bridge layers.

The top shell thickness is calculated as follows:

Top shell thickness = Top shell layers × Layer height

For example, the default setting is 5 layers × 0.2 mm = 1.0 mm.

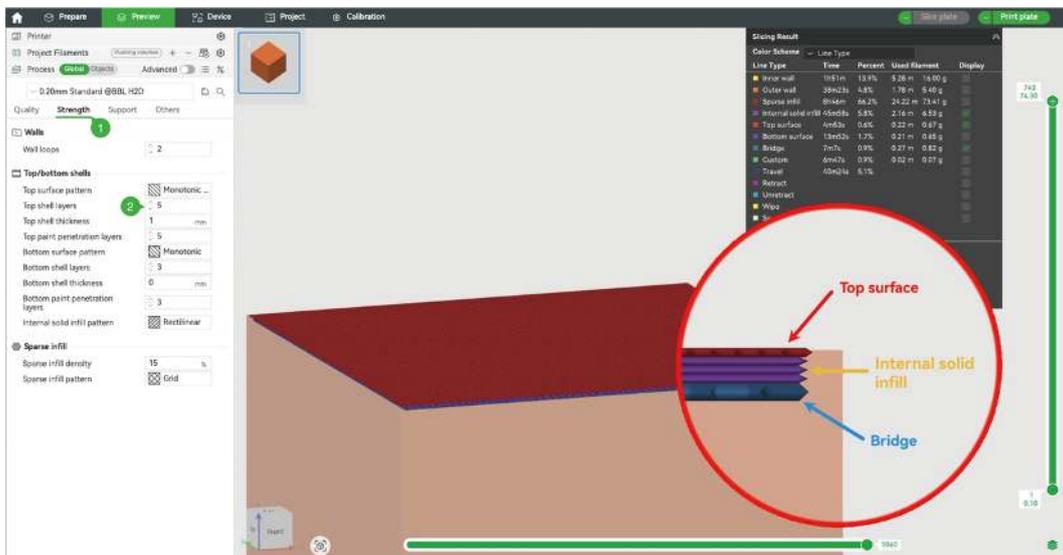
Setting the appropriate top shell thickness helps achieve a smooth, solid top surface and strengthens the model.

NOTE

- If the **manually set thickness is less than the calculated value**, Bambu Studio will **automatically increase the top shell layers** to match the calculated thickness.
- If the thickness is set to **0**, the system will **automatically calculate the thickness** based on the top shell layers and layer height.

Recommended settings:

- It is recommended to set the top shell layers to 4–6 to ensure a smooth, solid top surface and effectively cover the underlying infill.
- When using larger layer heights or low sparse infill density, increasing the top shell layers is suggested to improve print quality.



Bottom Shell Layers and Thickness

Bottom shell layers determine the solid thickness of the model's base, consisting of the bottom surface and solid infill above it.

The bottom shell thickness is calculated as follows:

Bottom shell thickness = Bottom shell layers × Layer height

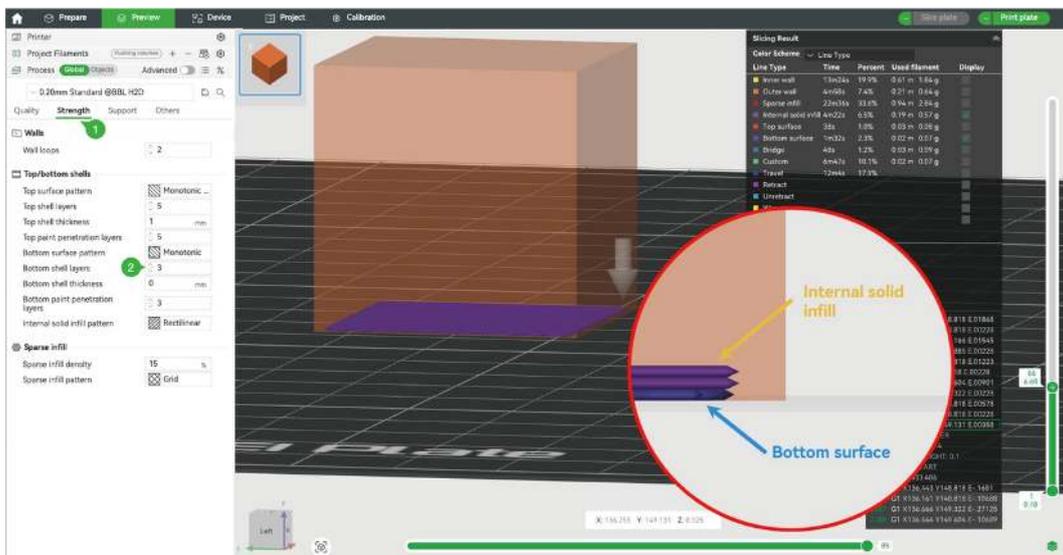
Increasing the bottom shell layers appropriately enhances bottom strength and stability, and improves adhesion to the build plate.

NOTE

- If the **manually set thickness is less than the calculated value**, Bambu Studio will **automatically increase the bottom shell layers** to match the calculated thickness.
- If the thickness is set to **0**, the system will **automatically calculate the thickness** based on the bottom shell layers and layer height.

Recommended settings:

- Setting the bottom shell thickness to 0 means it is fully determined by layer height and bottom shell layers.
- In most cases, 3 or more bottom shell layers are recommended to ensure a solid and reliable base.

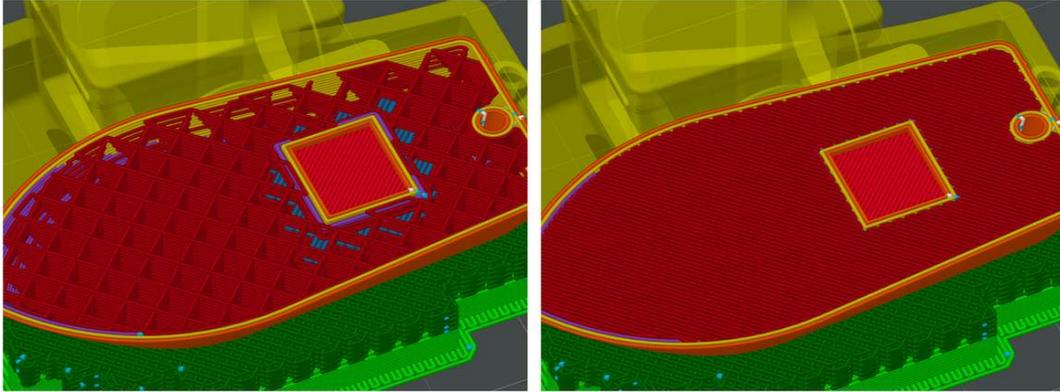


Sparse Infill

Sparse infill density determines how compact the model's internal structure is.

- Low density (10% – 20%): Suitable for non-load-bearing display models, significantly saving filament and print time.

- High density (above 30%): Enhances structural strength, suitable for functional parts.
- Recommended settings: 15%, suitable for most daily prints, balancing strength and efficiency.



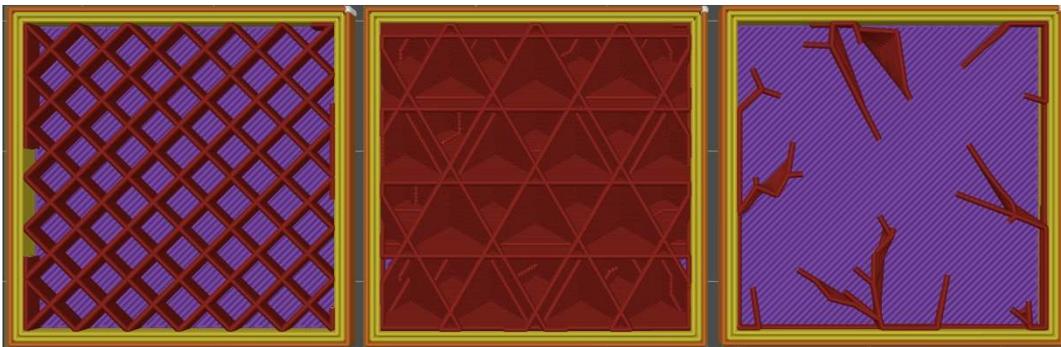
Sparse Infill Patterns

Different sparse infill patterns suit various needs and can be categorized as follows:

- **Strength-focused patterns:** Such as Honeycomb, Cubic, Gyroid. These patterns provide uniform load distribution and high strength, making them ideal for functional parts.



- **Speed-focused patterns:** Such as Grid, Support Cubic, Lightning. These patterns have simple paths and allow fast infill, making them ideal for test prints or efficiency focused tasks.



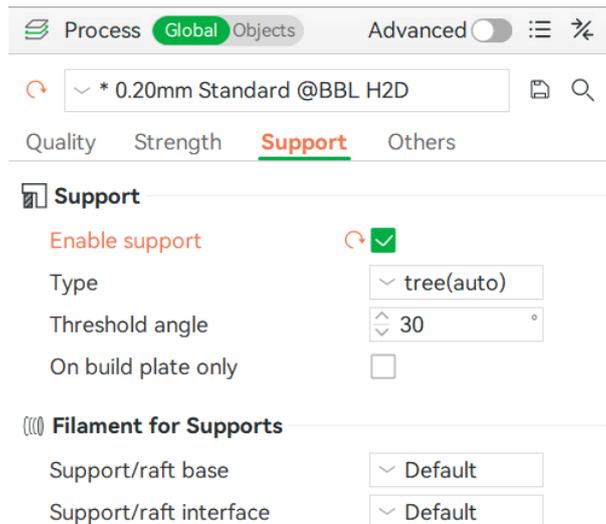
- **Aesthetic-focused patterns:** Such as Hilbert Curve, Octagram Spiral, Archimedean Chords. These patterns offer unique textures or smoother internal structures, improving the overall appearance of models with visible infill.



For more details on sparse infill patterns or advanced strength settings, please visit the Bambu Lab Wiki (wiki.bambulab.com/home) for relevant guidance.

5.5.4 Support Settings

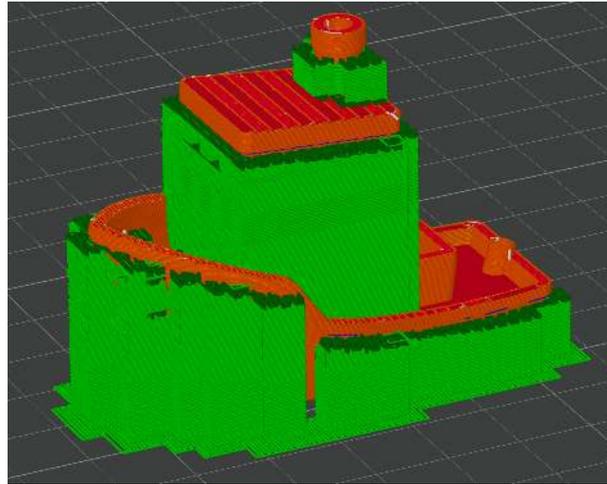
In FDM 3D printing, support structures are critical in complex models with overhangs, bridges, and other challenging geometries. Proper support configuration can significantly improve print success rates and simplify post-processing.



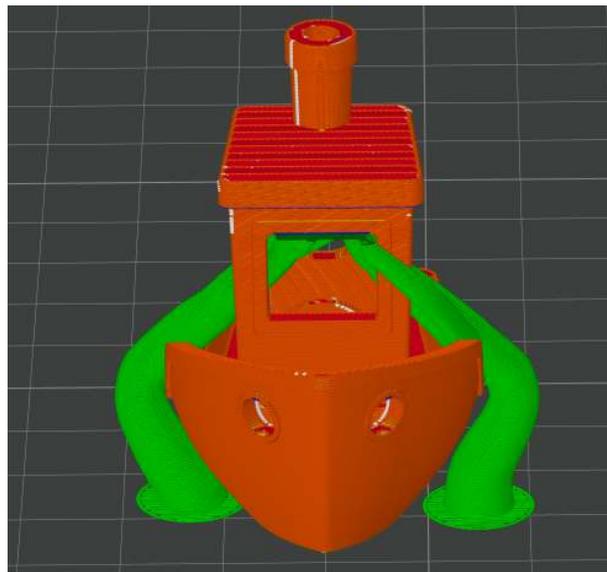
Introduction to Common Parameters

- Type

Normal Supports are generated using a standard **linear structure**, providing a stable, evenly distributed layout, fast generation, and easy removal from the model surface. This type is suitable for mechanical parts and geometric models dominated by flat or linear surfaces.



Tree supports employ a "trunk + branches" branching structure built layer by layer, making contact only at critical stress points. They are ideal for complex curved surfaces and detailed characters or art models, greatly reducing support material usage and avoiding leaving obvious marks on fine surfaces.



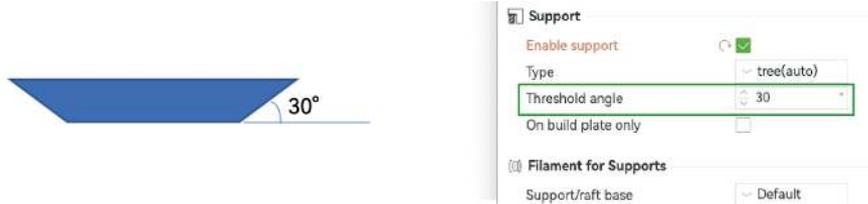
Auto means Bambu Studio generates support automatically based on the set threshold angle.

Manual indicates that supports are generated only in the areas you specify, using the Supports Painting tool. For detailed steps on support painting, please visit the Bambu Lab Wiki (wiki.bambulab.com/home) for relevant guidance.

- Threshold Angle

Threshold angle defines the steepest surface angle (relative to the horizontal plane) that can be printed without supports. When the surface angle is smaller than this threshold and the support type is set to Auto, supports will be generated. Adjusting this value is a quick way to con-

control the amount of support generated. Lower values produce fewer supports, higher values produce more. The default value is **30°**, suitable for most models.



- On Build Plate Only

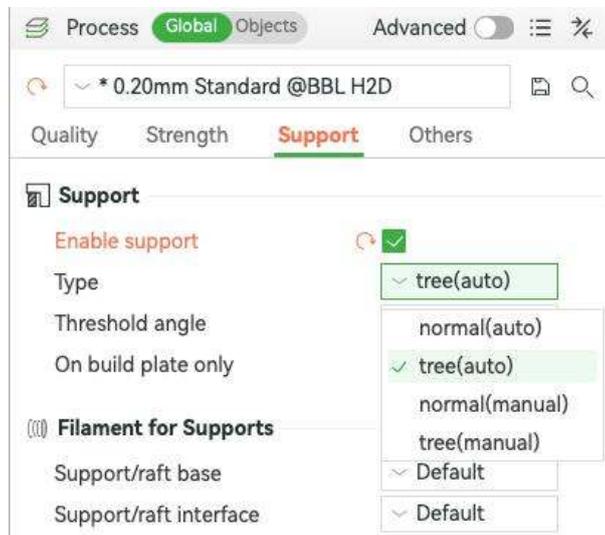
Enabling this option ensures that all supports originate only from the build plate and do not attach to the model itself. This helps reduce support marks on the model surface, improving appearance quality and simplifying post-processing. However, for structures far from the build plate or completely suspended, this setting may not offer adequate support. It should be used carefully based on the model geometry.

- Filament for Supports

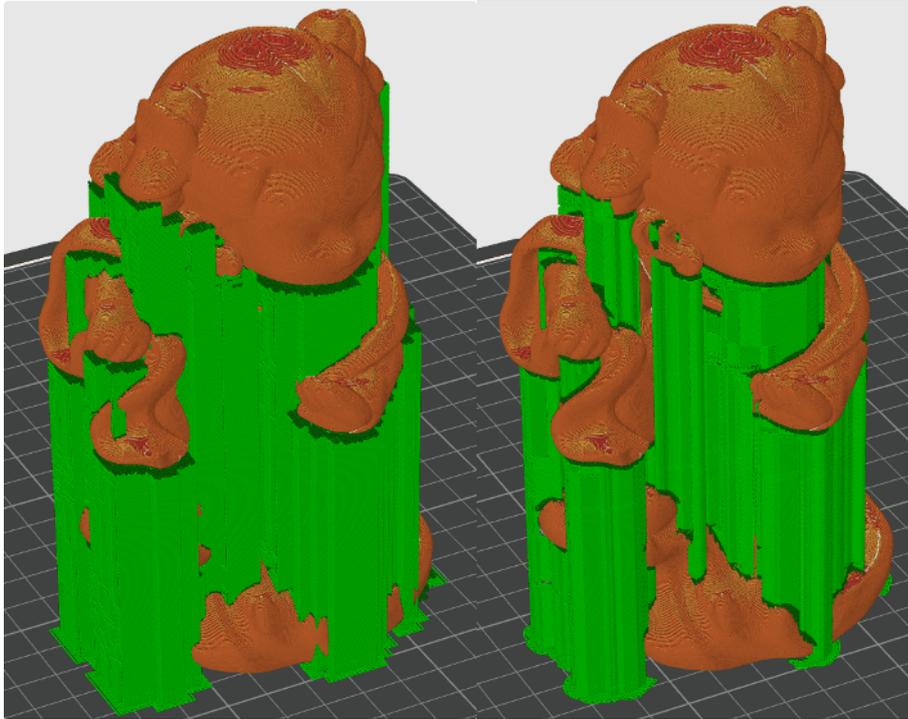
Supports consist of two parts: **support base** and **support interface**. **The support interface is the contact layer with the model**, and the rest forms the support body. These two parts can use different types of filaments. If unspecified, the current layer's filament is used by default to minimize filament change time. Typically, dedicated support materials, such as Bambu Support W and Support G, are chosen for printing the support interface to improve removability and surface quality.

Introduction to Advanced Parameters

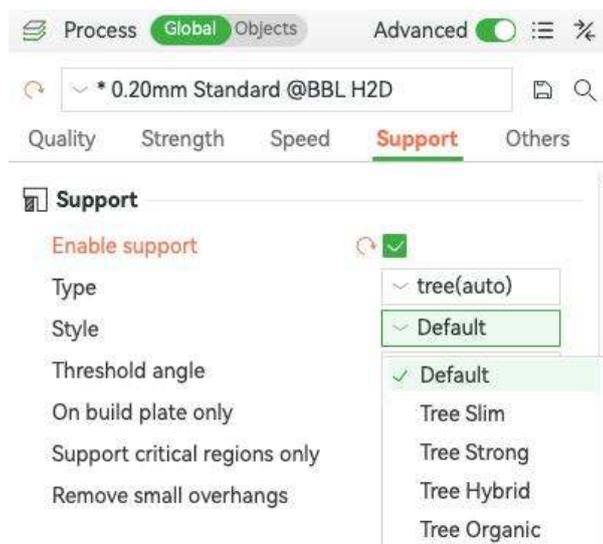
- Normal Support Style



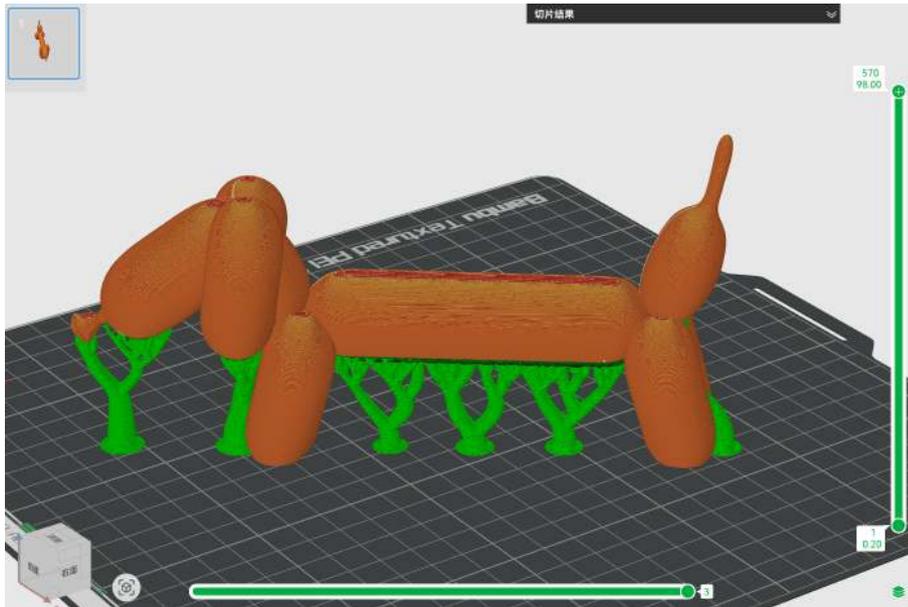
- **Grid:** Projects overhang areas vertically onto the build plate and expand outward to create regular, grid-like support columns. The supports are stable and strong, suitable for **large horizontal overhangs**.
- **Snug:** Grows precisely along the model contour, saving material and providing flexible support shapes. It is well suited for **complex or detailed areas**.



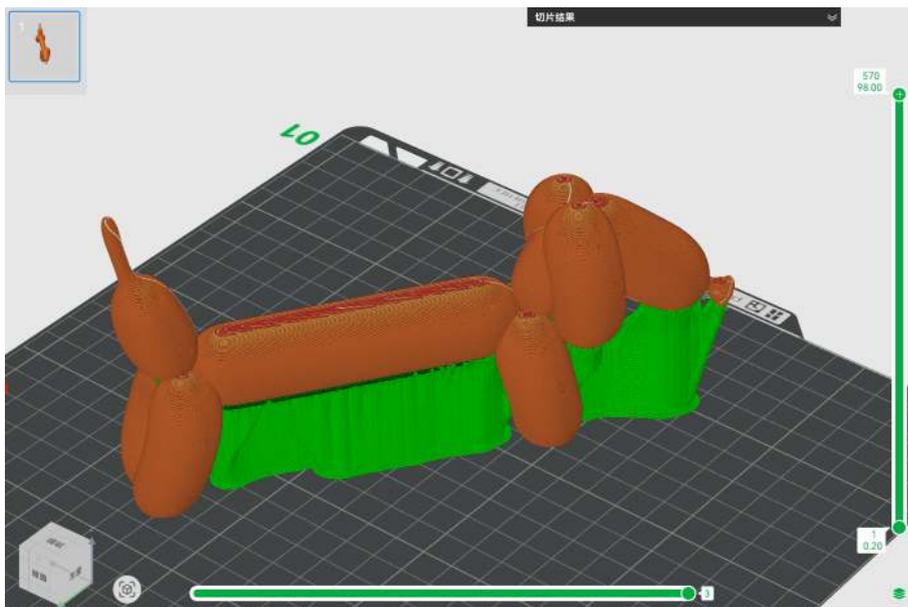
- Tree Support Style



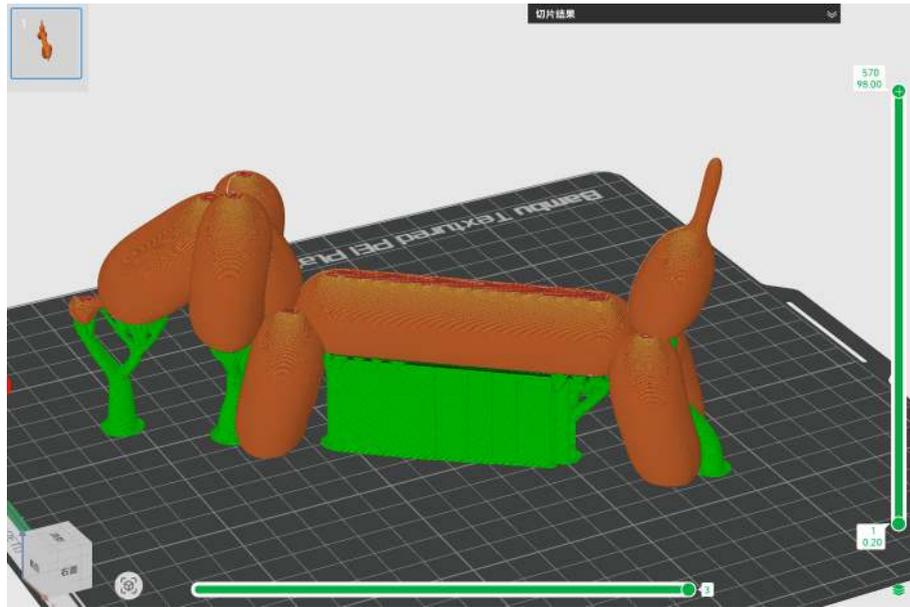
- **Tree Slim :** Features thin branches and an optimized overall structure. Suitable for small, lightweight overhangs. Its aggressive merging strategy reduces material consumption and minimizes removal marks.



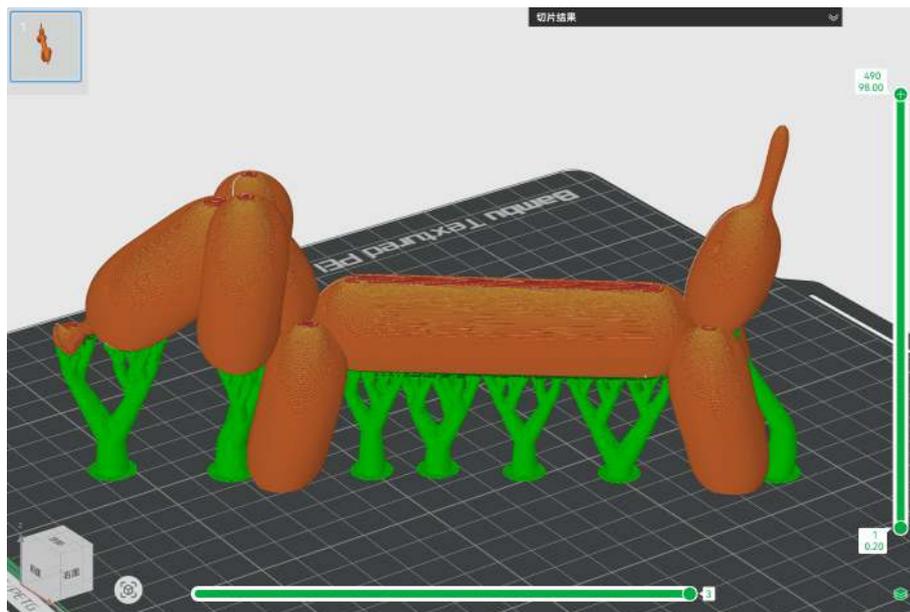
- **Tree Strong:** Characterized by a thicker trunk and branches, offering a more robust structure with strong load-bearing capacity. Recommended for large or heavy overhangs where stronger support is required.



- **Tree Hybrid:** Combines the advantages of tree and normal supports. The software automatically determines the most suitable structure and applies it in different areas. This style offers a flexible automated strategy aimed at optimizing support structures for different model parts.

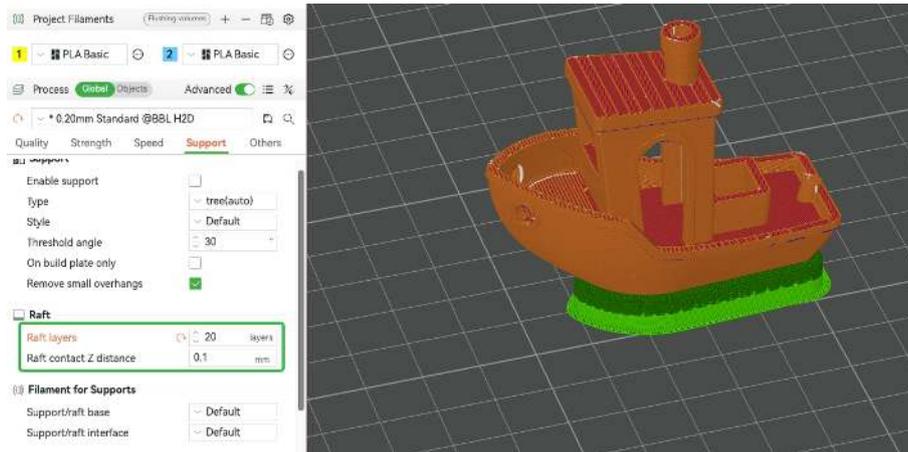


- **Tree Organic:** Generated using biomimetic algorithms, producing natural, smooth, and curved branches. It intelligently routes support to reach overhangs while minimizing material usage and easing removal. Suitable for most models that require tree supports.



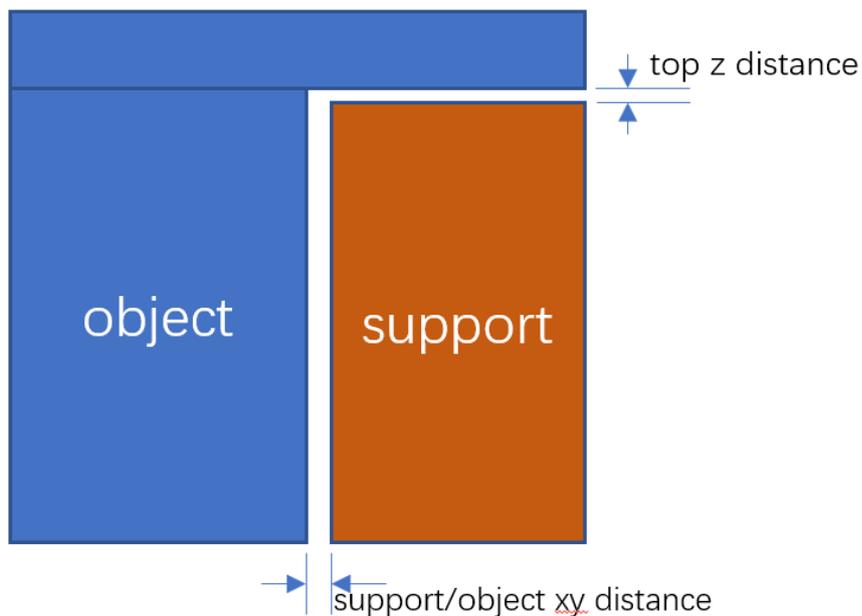
- **Raft**

A raft is a base layer located beneath the model and support structures. It is used to enhance adhesion during printing, especially with warp-prone filaments such as ABS. Rafts effectively prevent support detachment and lift the entire model off the build plate, isolating it from potential surface unevenness.



- Top Z Distance

The vertical distance between the top of the support interface and the underside of the model's overhang.



When using dedicated support interface filaments such as Bambu Support for PLA, Bambu Support for PLA/PETG, or Bambu Support for PA/PET, this value can be set to 0, allowing direct contact between support and model. When the support interface uses the same filament as the model, a value of around 0.2 mm is recommended to prevent supports from becoming difficult to remove. The table below shows how the Top Z Distance affects support removal and support surface quality:

Top Z Distance	Ease of Support Removal	Support Surface Quality
Increase	Easier	Lower
Decrease	Harder	Higher

- Support/Object XY Distance

This parameter controls the horizontal distance between supports and the model, with a default value of 0.35 mm. The greater the distance, the easier it is to remove the support, and it also reduces the risk of scratching the model's surface. If the support is difficult to remove, consider increasing this value appropriately.

CAUTION

- The top Z distance and the support/object XY distance should be **adjusted together**. Their combination directly impacts model surface quality and ease of support removal. Finding the optimal balance is key to achieving efficient and high-quality printing.
- It is recommended to remove supports within 2 hours after printing. Prolonged exposure may cause moisture absorption, making supports softer and harder to remove—especially when printing moisture-sensitive filaments such as PA-CF, PA6-CF, or when using water-absorbing support filaments like PVA or Support for PA/PET. If supports are hard to remove, drying the model or cooling it before removal may help.

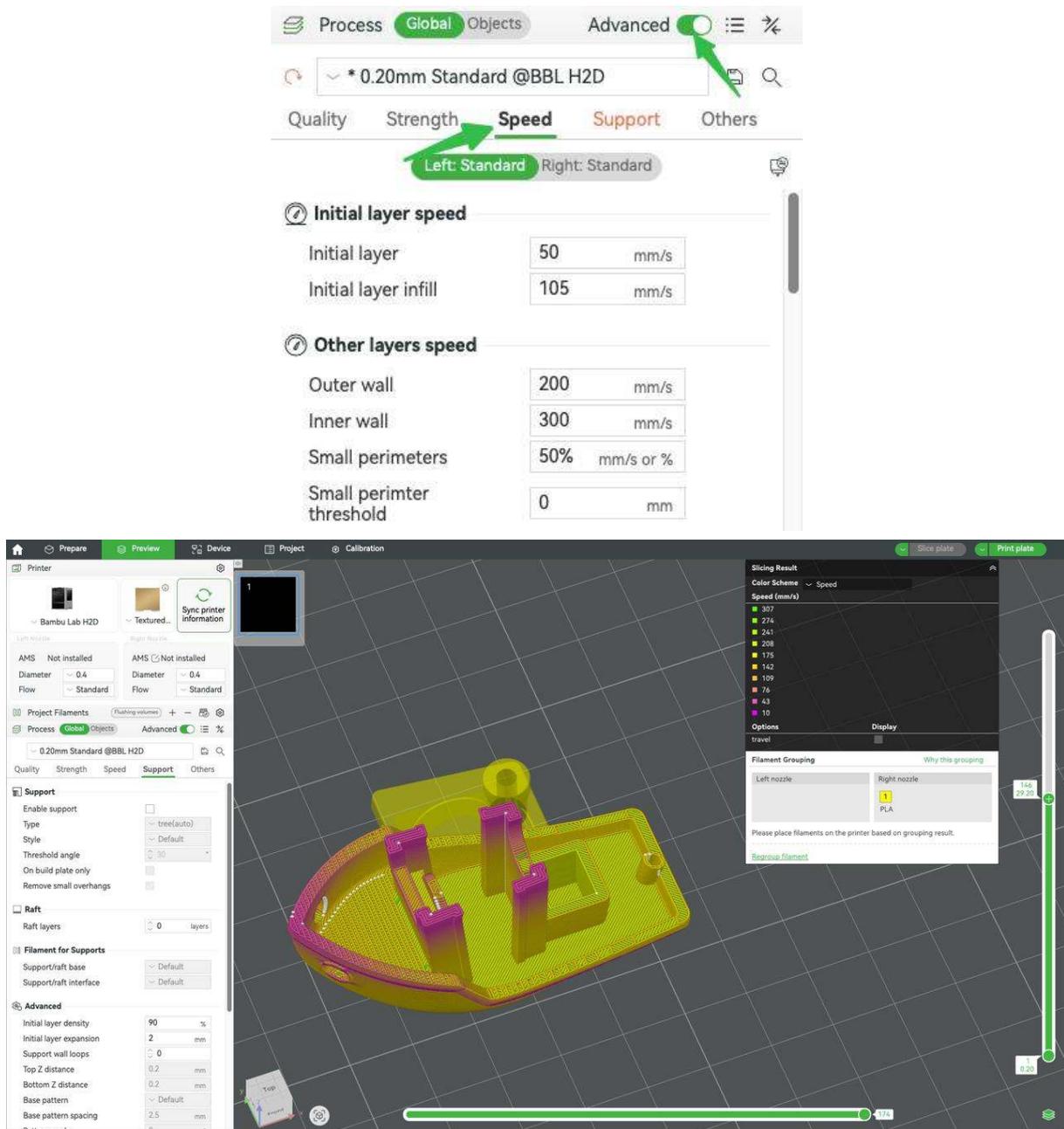
For more advanced support settings, please visit the Bambu Lab Wiki (wiki.bambulab.com/home) for relevant guidance.

5.5.5 Speed Settings

Selecting the appropriate print speed helps improve print quality and accuracy. Bambu Studio provides optimized default speed presets that work well for most print jobs. You may also fine-tune the print speed according to custom needs.

TIPS

Process presets already include all required settings for specific printing scenarios and are recommended for beginners. If you need to adjust the print speed, please enable advanced mode in the process tab.



Print speed

- Initial Layer Speed

The print speed for solid infill and other parts of the first layer. This setting directly affects bed adhesion.

- Overhang Speed

When Slow down for overhangs is enabled, Bambu Studio automatically reduces inner and outer wall print speeds in overhang areas to improve overhang surface performance. This feature is enabled by default and can be disabled if needed.

- Other Layers Speed

Print speeds for inner and outer walls, infill, bridge, and supports. If the default parameters do not meet expectations, custom adjustments can be made.

- Travel Speed

The movement speed when no filament is extruded. This mainly affects overall printing efficiency.

- Acceleration

Print acceleration for travel, first layer, inner and outer walls, top surface, and sparse infill. This parameter can affect motion smoothness and speed transition during printing.

Chapter 6 Key Features Introduction

6.1 Large Volume Printing

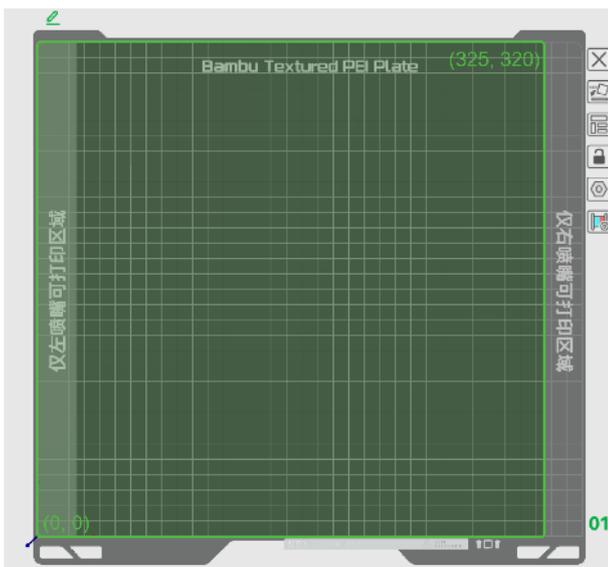
This printer supports large volume printing, enabling the completion of large models or entire structures in a single print without segmenting, improving both efficiency and final part strength. The large printing volume expands possibilities for applications such as prototyping, functional parts, artwork, and architectural models, offering greater design freedom while minimizing post-assembly work and cumulative errors.

6.1.1 Horizontal Printing Area

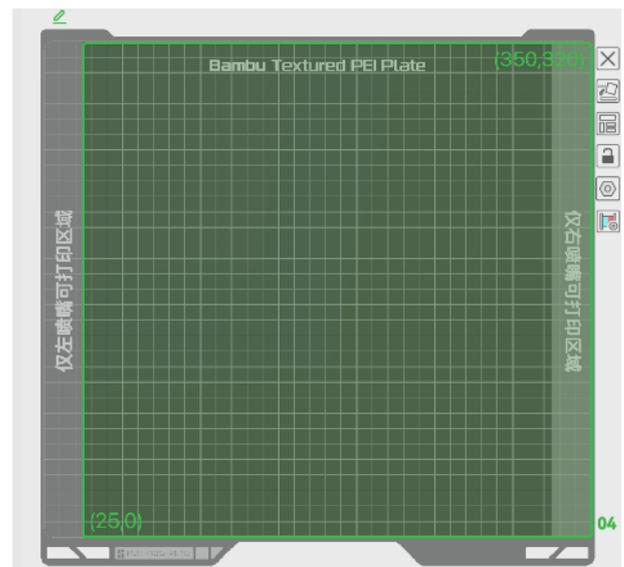
The H2D printer features left and right nozzles, both with same print area $325 \times 320 \text{ mm}^2$; however, their horizontal printable ranges differ. Using the lower-left corner of the build plate as the coordinate origin (0, 0), the specific printable ranges of the two nozzles are as follows:

- Left nozzle: printable area coordinates range from (0, 0) to (325, 320)
- Right nozzle: printable area coordinates range from (25, 0) to (350, 320)

In Bambu Studio's build plate preview, the areas labeled "Left nozzle only area" and "Right nozzle only area" indicate that models placed in these areas can only be printed using the corresponding nozzle.



Left nozzle only area



Right nozzle only area

6.1.2 Vertical Printing Area

The H2D printer's maximum print height is **325 mm**, but actual maximum vertical heights may vary between nozzles. Please refer to the following:

- Left Nozzle: Maximum print height 320 mm
- Right Nozzle: Maximum print height 325 mm

When the model is placed within the "Left nozzle only area", its maximum height must be ≤ 320 mm and must not exceed the left nozzle's maximum print height.

If the model height exceeds 320 mm (e.g., 324 mm), even if it is placed in the central shared area covered by both nozzles rather than the "Right nozzle only area", printing can only be completed through the right nozzle.

6.1.3 Check the Model Placement Area

When the model is placed in the **non-printable area** of either the left or right hotend, all filaments used by the model can only be printed with the hotend that covers that specific print area.

NOTE

All filaments include the main filament set in the system, the filaments applied to specific surface areas using the color painting function, and the filaments applied to local regions of the model using the modifier function.

If Bambu Studio detects a conflict between model placement and printable area, an error message will be displayed. Model placement or filament assignment must be adjusted before slicing and printing can proceed. The following are common conditions that trigger error prompts:

- **Same filament cross area conflict:** The same filament is used in both the "left hotend only area" and "right hotend only area" (a single filament cannot be used across both areas simultaneously).
- **Model height exceeds:** The overall model height exceeds the maximum printable height of the printer (i.e., > 325 mm).
- **Manual hotend and filament position conflict:** When using custom mode, manually assigned hotends and filaments must correspond to the printer's defined printable area for each hotend; otherwise, a position-filament-nozzle mismatch error will occur.

- **Manual hotend assignment and height limit conflict:** Using custom mode to manually assign hotends to filaments where the model height exceeds the maximum printable height of the assigned hotend.
- **Partial filament exceeds assigned hotend printable area:** After color painting or modifier functions, some filament paths extend beyond the printable area of the assigned hotend.
- **Purge path extends beyond assigned hotend's printable area:** When the "Purge into objects' infill/support" function is enabled, slicing may generate purge paths that extend outside the assigned hotend's printable area.

6.2 Dual Hotend Printing

This printer features dual hotends (left and right) that enable simultaneous printing with two different filament types or colors. This enhances both model functionality and visual detail. For example, combining hard and soft materials allows you to manufacture complex parts with varying properties (see [Soft and Hard Filament Multi-Material Printing](#)).

The dual hotend design supports seamless material or color switching within a single print job, eliminating frequent filament changes and significantly reducing preparation time. Compared to single-hotend printing, it minimizes purge waste during material or color changes, reduces material consumption, and delivers more efficient multi-material or multi-color results with balancing speed, cost, and print quality.

6.2.1 Filament Grouping Strategy

For dual hotend printing, an effective filament grouping strategy improves multi-material efficiency, reduces waste, and increases the print success rate.

Bambu Studio automatically assigns suitable nozzles and print sequences based on the number of filaments required by the model, the purge volume for different printing orders, and the physical characteristics of each nozzle.

This section explains the principles and rules used by the system to group filaments.

Number of model filaments

- **2 filaments:** Assign one filament to each nozzle for optimal results. This configuration eliminates purge operations and enables material changes through simple nozzle switching, improving efficiency and reducing material use.

- **3 or more filaments:** Prioritize nozzle switching to reduce the number of purge operations. The software automatically assigns filaments to the appropriate nozzle based on the selected grouping mode.

Multi-Color printing sequence

Within a single layer, the printing order of different colors directly affects the frequency of filament changes, resulting in variations in the total amount of purging required.

To minimize purge waste, Bambu Studio automatically determines the optimal printing order for each layer according to the purge volumes between filaments.

For example, if the total purge volume for printing order 1-2-3 is higher than order 1-3-2, the system will prioritize 1-3-2 as the printing sequence for that layer.

Nozzle Physical Characteristics

Bambu Studio groups filaments according to the following priority rules, based on each nozzle's **printable area** and compatible filament types.

High Priority Rules

- TPU filament: Can only be printed by the right nozzle.
- Carbon Fiber (-CF) filament: Can only be printed by the left nozzle.

Medium Priority Rules

- Models taller than 320 mm must be positioned within the right nozzle's printable area and printed using the right nozzle.
- Model placed in left nozzle only area or right nozzle only area: Filaments used for the model in this area must be printed with the corresponding nozzle.

Low Priority Rules

- For filaments printable by both nozzles, priority is given to the right nozzle.

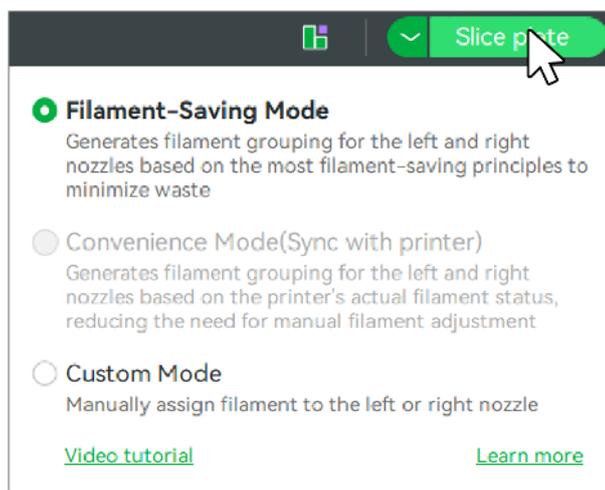
If high-priority and low-priority rules conflict, Bambu Studio follows the high-priority rules. When multiple rules apply at the same time, the system evaluates all applicable rules together to determine the best solution.

6.2.2 Select Filament Grouping Mode

Bambu Studio provides three filament grouping modes **Filament-Saving Mode**, **Convenient Mode**, and **Custom Mode**. The system defaults to **Filament-Saving Mode**, which is enabled automatically and does not require manual activation.

If you need to switch filament grouping modes, follow the steps below:

1. Hover the mouse over the slice button to display a dropdown menu showing the three modes.
2. Select the desired filament grouping mode.
3. Click the slicing button to perform the slicing operation.



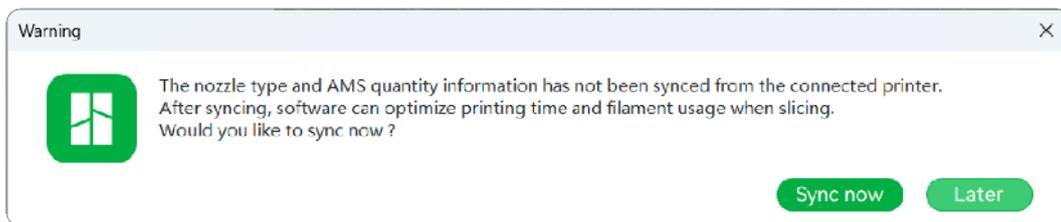
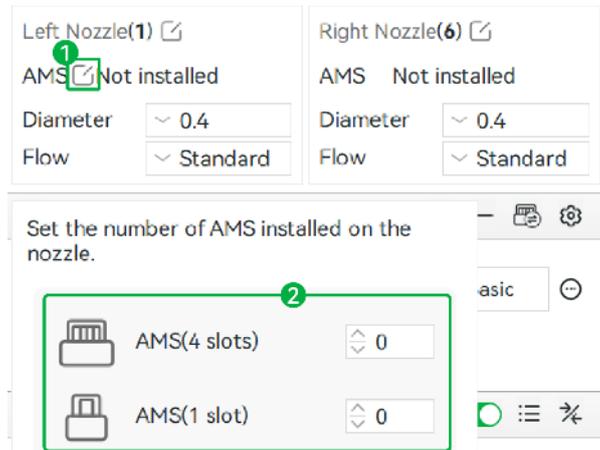
You can also click the filament grouping icon  on the right side of the build plate to view and set the filament grouping mode for that specific plate.



Filament-Saving Mode

Filament-Saving Mode aims to minimize filament waste caused by purging during filament changes by assigning filaments that appear together in more layers (requiring frequent switching) and assigning large purge volumes to different nozzles.

Before using this mode, it is recommended to manually set or synchronize the pairing relationship between the AMS and left/right nozzles so the filament grouping in Bambu Studio reflect the physical layout. If the printer is not connected to the AMS, it is assumed by default that each nozzle is connected to one external spool.



After slicing, filaments are bound to their assigned nozzles. When sending a print job, switching to the other nozzle is not allowed, even if it contains a filament with a closer color match. To save filament while improving color matching, manually adjust the filament placement based on the grouping results.

The grouping algorithm of filament-saving mode mainly considers the following four factors, in descending order of priority:

1. Nozzle compatibility: Determines whether the current nozzle supports the filament type (e.g., TPU can only be printed using the right nozzle).
2. Filament quantity limit: Ensures the number of filaments connected to each nozzle does not exceed its capacity.
3. Minimize purge volume: Prioritize grouping methods that minimize total purge filament usage.
4. Color difference control: Minimize the difference between the actual filament color and the color required by the model.

NOTE

H2D supports up to 4 AMS 2 Pro units and 8 AMS HT units simultaneously, providing a total of 24 filament slots. In extreme cases, if all AMS units are connected to the same nozzle, that nozzle supports a maximum of 24 filament spools.

TIPS

Filament-Saving Mode prioritizes reducing purge filament consumption. The filaments automatically assigned after slicing may differ from the ones initially required by the model. If model color accuracy is critical, it is recommended to manually adjust filament positions according to the grouping.

Convenience Mode

Convenience mode groups filaments based on the actual filament's placement in the AMS, typically without requiring additional adjustments. It is suitable for scenarios such as remote printer operation, where frequent manual AMS adjustments are impractical.

Please ensure the printer is connected before using this mode. For more accurate filament matching, it is recommended to synchronize AMS information from the filament list before slicing, ensuring the software obtains data consistent with the actual filament placement.

The grouping algorithm of convenience mode mainly considers the following three factors, in descending order of priority:

1. Nozzle compatibility: Whether the current nozzle supports printing the filament type (e.g., TPU can only be printed by the right nozzle).
2. Filament type matching: Whether the filament type used for slicing matches the filament in the AMS.
3. Filament color matching: Whether the filament color used for slicing is closest to the filament in the AMS.

NOTE

This mode is easy to operate but may consume more filament than the filament-saving mode.

Custom Mode

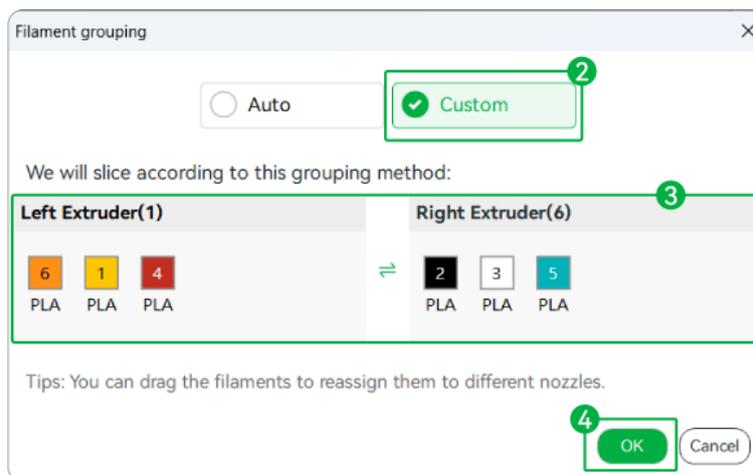
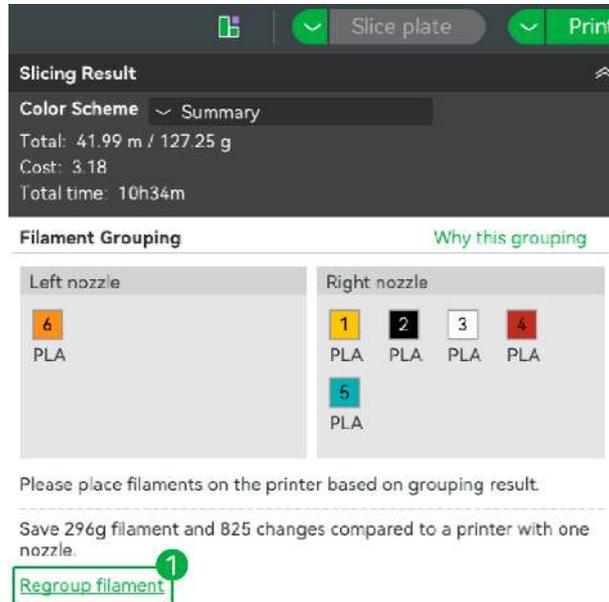
If the grouping from Filament-Saving Mode or Convenience Mode does not fully meet your needs but you wish to adjust based on them, you can follow the steps below.

Step 1: In the slicing results display panel, click **Regroup filament**.

Step 2: In the filament grouping pop-up window, select **Custom Mode**.

Step 3: Drag the filaments you want to move to their target positions.

Step 4: Click **OK**, and Bambu Studio will recalculate the slicing results based on your adjustments.



If you want to fully customize how filaments are assigned, follow these steps.

Step 1: Hover the mouse cursor over the Slicing button on the interface to display a pop-up window containing three modes.

Step 2: Select Custom Mode, then click the Slicing button to open the filament grouping pop-up window.

Step 3: Drag the filaments you wish to move to the target positions.

Step 4: Click OK, and Bambu Studio will recalculate the slicing results according to your custom settings.

After slicing, the system will display how much filament could be saved by using the (Filament-Saving Mode) compared to your manual grouping. This helps you evaluate whether optimization is worthwhile.

NOTE

Material-Saving Mode considers only the amount of filament saved from reduced purging. In some cases, it may result in more filament changes than Convenience Mode or Custom Mode, which is normal.

6.2.3 Slicing Mode Selection

To apply a uniform filament grouping mode across all build plates, select your desired grouping mode, then click **Slice all**. At this point, all build plates will use the same grouping mode, and any existing plate-specific grouping modes will be overwritten.

To set filament grouping modes individually for each build plate, configure the filament grouping for a single plate, then click **Slice plate**.



6.2.4 Multi-Material Printing with Soft and Hard Filament

Multi-material printing of soft and hard filaments allows both soft and hard materials to be integrated within the same model, creating a unified rigid-flex structure that achieves both functional and performance requirements. This process reduces assembly steps, improves overall printing efficiency, and expands creative and engineering design freedom. It is particularly suitable for products requiring both strength and flexibility, such as helmets, bicycle saddles, and functional prototypes.

You can visit the Wiki (wiki.bambulab.com/home), and search for "H2D Soft and Hard Filament Multi-Material Printing Guide" to obtain detailed operation instructions, preset profiles, and slicing parameter setup methods.

**⚠ CAUTION**

It is strongly recommended to use **Bambu filaments** to ensure optimal print quality and device safety.

⚠ CAUTION

Due to softening of flexible filaments at high temperatures during printing, this feature currently only supports mixed printing of Bambu TPU 95A HF with high-temperature engineering filaments.

The following uses PETG-CF combined with TPU 95A HF as an example to briefly outline the main operational steps.

Step 1. Prepare the filaments and dry the TPU filament (see [Drying Filament](#)).

- Recommended drying equipment: AMS HT
- Temperature: 75 °C
- Drying duration: 18 hours

Step 2. Load the engineering filament (PETG-CF) into the left hotend, and load TPU 95A HF into the right hotend.

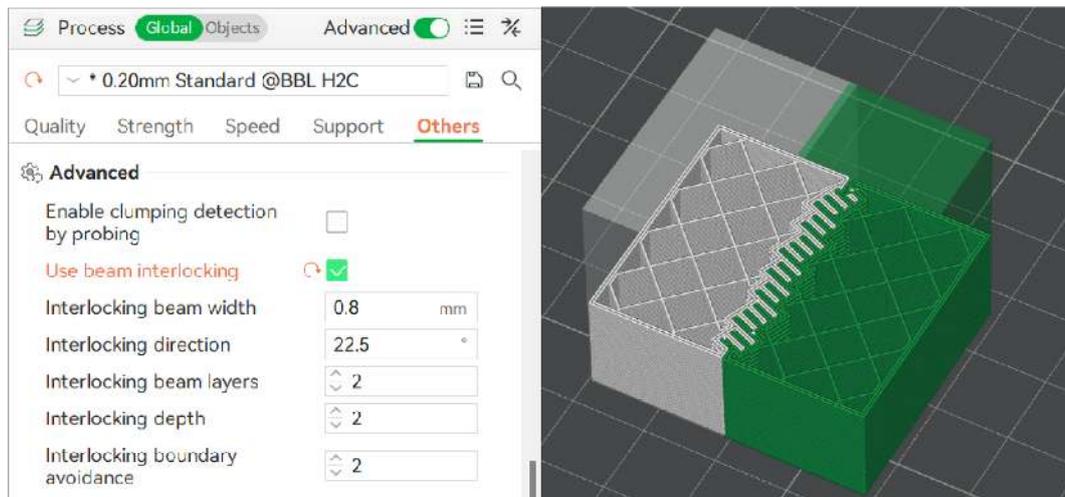
⚠ CAUTION

Certain types of flexible filament (TPU) can only be printed with the **right hotend**.

Step 3. Choose the appropriate chamber temperature mode based on the model.

Model Type	Recommended Air Condition Mode	Key Operation Tips
TPU as Base Layer or Main Structure	Cooling Mode	Keep the TPU bed temperature stable; set the engineering filament bed temperature lower than 70 °C
Engineering material as base, for strength	Heating Mode	TPU must use an external spool, and it is recommended to use a dedicated hotend for TPU printing

Step 4. Enable Interlocking Beams. In Bambu Studio's go to **Prepare** interface, select **Others** > **>Check "Use beam interlocking"** to enhance bonding between these two filament types.



Step 5. Set slicing parameters or download preset profiles according to the chosen air conditioning mode.

TIPS

Please visit the Wiki (wiki.bambulab.com/home) to obtain preset slicing profiles and parameter setup methods, search for the Soft and Hard Filament Multi-Material Printing Guide.

Step 6. Preparation before printing. If selecting cooling mode, remove the top cover glass to prevent the chamber temperature from rising.

⚠ CAUTION

Except when using the textured PEI build plate, always apply a full layer of glue-stick to non-textured print sheets to avoid damage.

Step 7. After printing is complete, wait until the chamber temperature drops below 45 °C before unloading the filament.

i TIPS

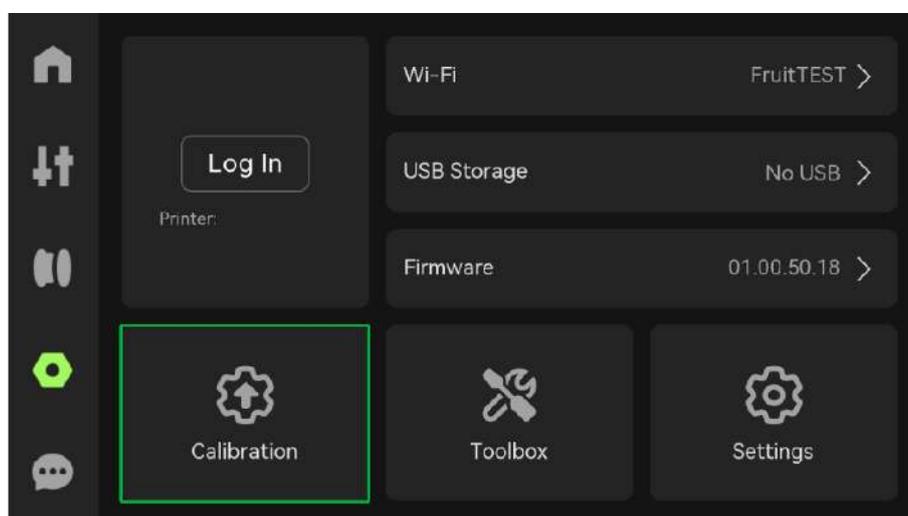
If the model adheres too tightly to the build plate, apply alcohol at the junction between the model and the build plate, then gently remove the model.

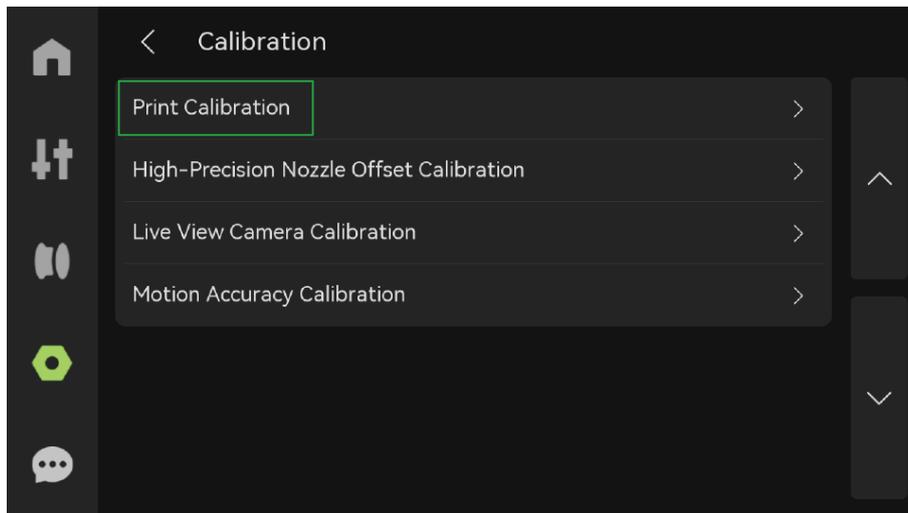
6.3 High Precision Printing

6.3.1 Start Calibration

Method 1: Start calibration from the printer touchscreen

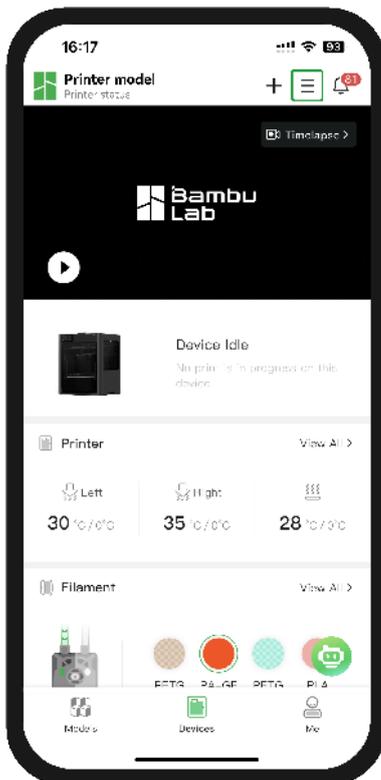
Tap  > **Calibration** on the printer touchscreen to enter the calibration page, then select the desired calibration routine.



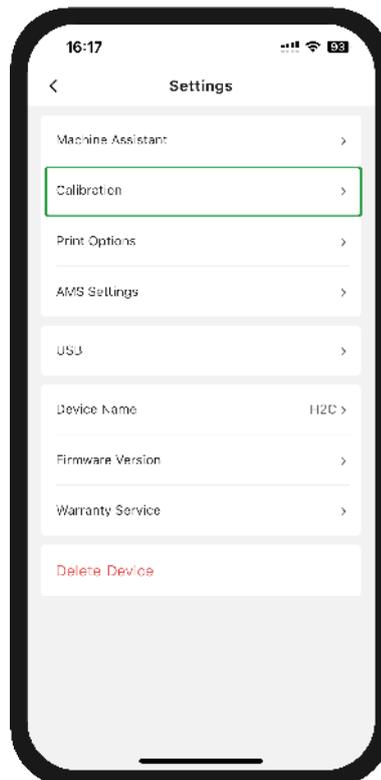


Method 2: Start calibration from Bambu Handy

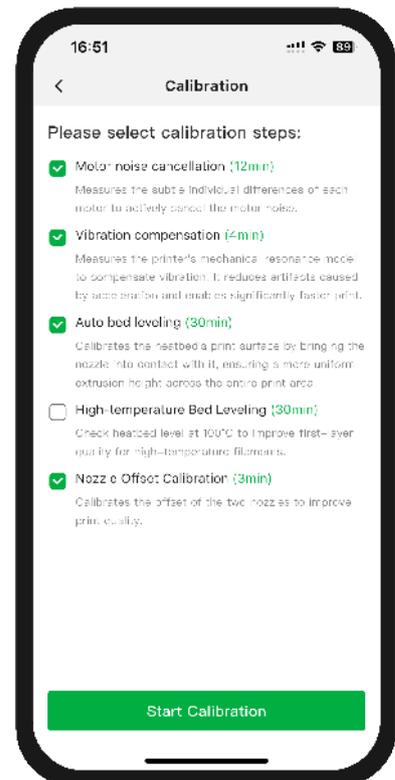
At the bottom of the Bambu Handy interface, select **Devices**, tap the icon in the top-right corner  to enter the settings menu, then choose **Calibration** and select the calibration task as needed.



Tap Devices



Tap Calibration



Start Calibration

NOTE

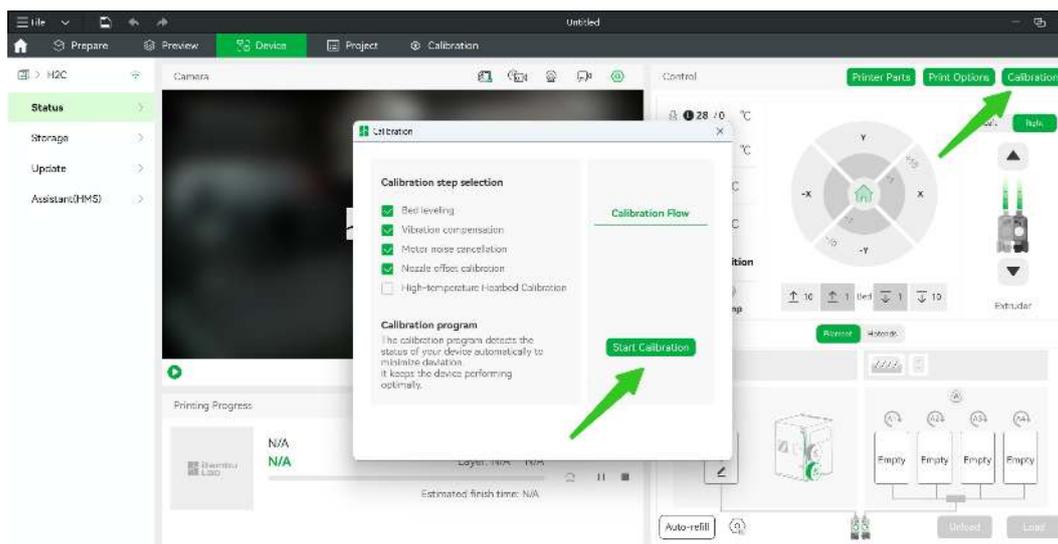
Bambu Handy only supports initiating certain calibration tasks. If the calibration you need is not available, please initiate it directly from the printer touchscreen.

Method 3: Start calibration from Bambu Studio

At the top of the Bambu Studio interface, select **Device**, tap **Calibration** in the top-right corner, and choose the calibration task as needed.

NOTE

Bambu Studio only supports initiating certain calibration tasks. If the calibration you need is not available, please initiate it directly from the printer touchscreen.



6.3.2 Print Calibration

Print Calibration automatically adjusts key printer parameters through built-in sensors, ensuring optimal print conditions without manual intervention.

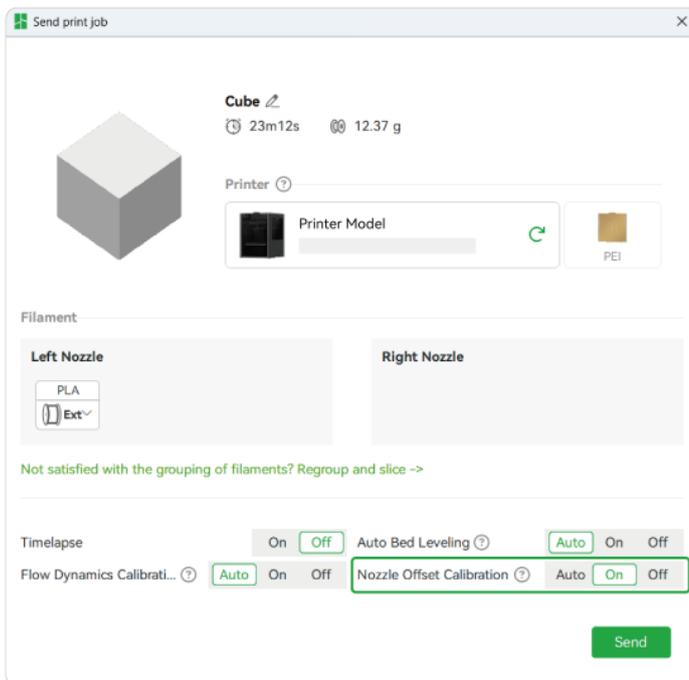
- **Motor Noise Cancellation:** Reduces motor noise during printing, especially during long-duration or high-speed prints. By optimizing motor motion algorithms and control strategies, noise is reduced and print surfaces become smoother, enhancing the overall print quality.
- **Vibration Compensation:** Measures the printer's mechanical resonance model to compensate for vibrations. During printing, any detected vibrations trigger automatic toolhead adjustments to maintain print accuracy. Reduces acceleration-related artifacts while significantly improving print speed.
- **Auto Bed Leveling:** Detects the flatness of the heatbed by having the nozzle contact the build plate, ensuring more consistent extrusion height.
- **High-temperature Bed Leveling:** Measures the flatness of the heatbed at 100 °C to improve the first-layer print quality for high-temperature filaments such as ABS, ASA, PC, and PA.

- **Nozzle Offset Calibration:** Measures and corrects dual-nozzle positional deviation across the XYZ axes to ensure accurate switching and prevent misalignment. Measures the offset between the two nozzles using the eddy current sensor in the toolhead and the nozzle offset calibration sensor located at the back of the heatbed, enabling compensation during printing.

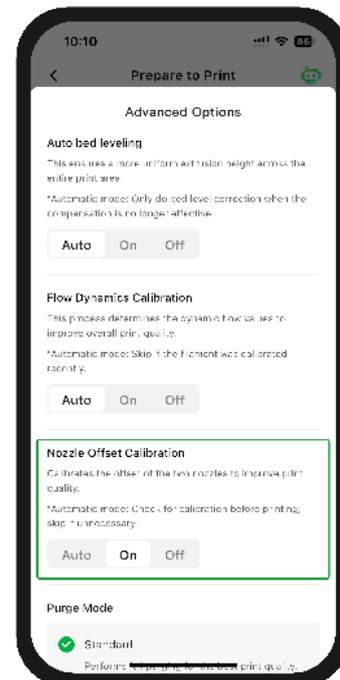
How to use nozzle offset calibration data?

Method 1: Enable nozzle offset calibration on the Send Print Job page in Bambu Studio.

Method 2: On the Bambu Handy Send Print Job page, enable nozzle offset calibration from **Advanced Options**.



Bambu Studio Nozzle Offset Calibration



Bambu Handy Nozzle Offset Calibration

When is nozzle offset calibration required?

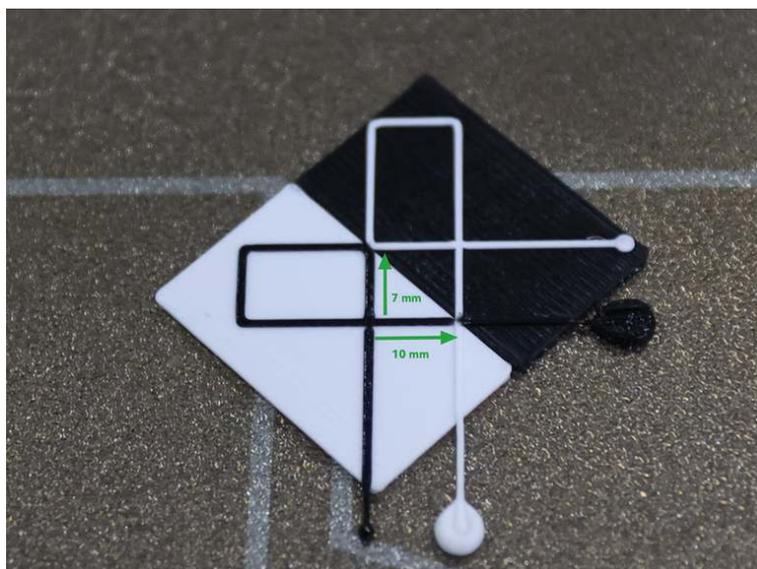
- Before the first print on a new printer.
- After a severe collision, relocation, or disassembly and reassembly of the printer.
- After adjusting the belt tensioner.
- When print quality issues occur.
- During routine maintenance.

6.3.3 High-Precision Nozzle Offset Calibration

Used to accurately calibrate the positional differences between the dual nozzles in the XY direction, ensuring precise alignment of the printing path when switching nozzles, thereby improving the surface quality and layer alignment.

Using AI visual recognition, it detects the actual print line positions of the two nozzles in the XY directions, calculates the XY offset between nozzles, and obtains a high-precision nozzle position calibration offset, which is automatically applied during printing.

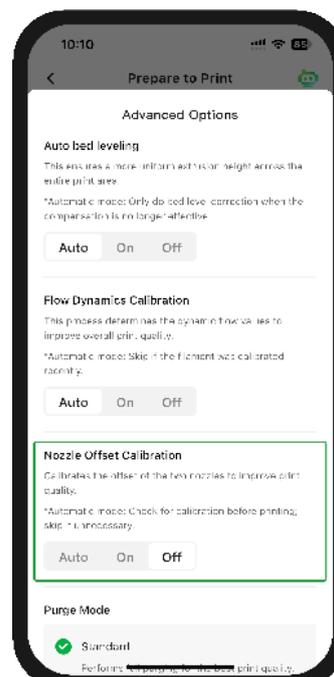
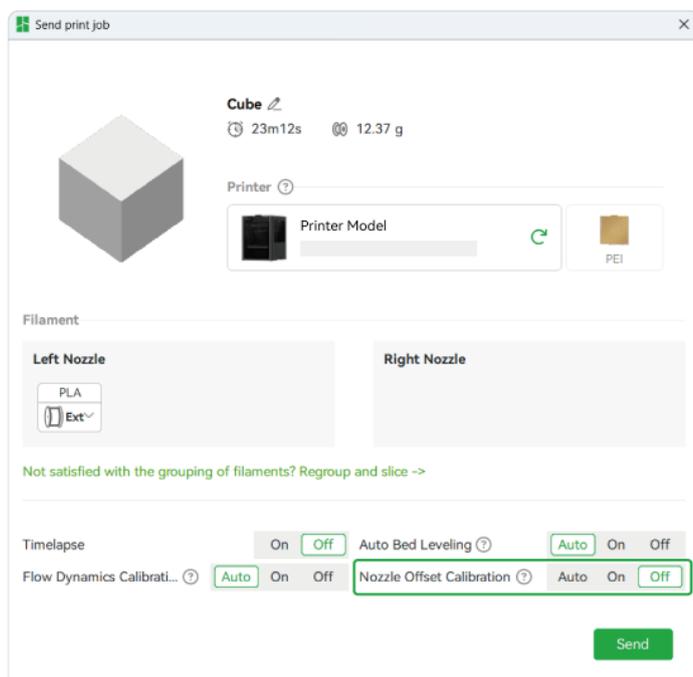
Compared with standard nozzle offset calibrations, high-precision nozzle offset calibration is based on the actual printed lines, theoretically providing higher positioning accuracy and avoiding potential errors associated with sensor-based calibration methods.



How to use high-precision nozzle offset calibration data?

Method 1: On the Send Print Job page in Bambu Studio, turn off nozzle offset calibration.

Method 2: On the Send Print Job page in Bambu Handy, turn off nozzle offset calibration from **Advanced Options**.



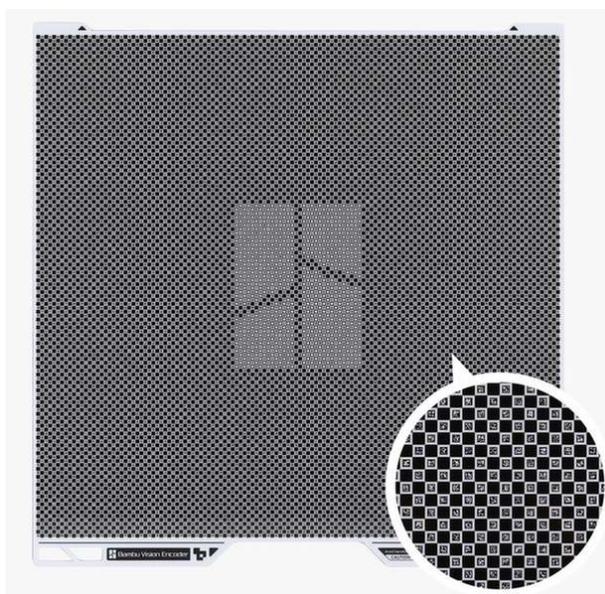
When is nozzle offset calibration needed?

When printing with dual nozzles and visible layer misalignment occurs on your model, you can select either nozzle offset calibration or high-precision nozzle offset calibration as needed.

6.3.4 Motion Accuracy Calibration

Used to improve the printer's positioning accuracy, suitable for large volume or high precision printing scenarios. This calibration effectively reduces motion lag and distortion, enhancing model dimensional accuracy and assembly quality.

During the calibration process, the toolhead camera scans each unique QR code square on the vision encoder to obtain the absolute position of the toolhead. The system compares the actual coordinates with the software coordinates. It calculates the motion error at each point, allowing the machine to compensate accordingly, resulting in a corrected and precise position of the toolhead. Calibration results can be maintained for weeks, unaffected by filaments or nozzles, making the operation simple and time-efficient. Although assembly changes or maintenance intervention will require a new calibration procedure.



NOTE

This function is an additional feature and requires the separate purchase of the vision encoder.

⚠ CAUTION

Do not place the vision encoder on the heatbed when it is hot. Otherwise, the high temperatures may cause the vision encoder to expand and deform, resulting in calibration errors.

How to use motion accuracy calibration data?

On the printer touchscreen, tap  > **Settings** > **Print Options** and check **Motion Accuracy Enhancement** (enabled by default). After enabled, the printer applies the motion accuracy calibration data during 3D printing, cutting, and laser engraving.

When is motion accuracy calibration needed?

- When the motion accuracy enhancement feature is enabled for the first time on a new printer.
- After the printer experiences a severe collision, relocation, or disassembly and reassembly.
- After adjusting the belt tension.
- For routine maintenance, it is recommended to perform once every two weeks.

6.4 Intelligent Detection

This printer is equipped with multiple intelligent detection features that automatically identify abnormalities during printing and promptly alert users, significantly improving print success rates. The system accurately detects issues such as nozzle clogging, air printing, or material pile-up through visual inspection, filament feed tracking, and torque monitoring, reducing material waste and the risk of equipment damage. Intelligent detection operates fully automatically without user intervention, making printing more reliable and effortless.

6.4.1 AI Print Monitoring

On the printer screen, tap  > **Settings** > **Print Options**, then check the boxes to enable the corresponding AI detection features as needed, and adjust the detection sensitivity accordingly. Detection sensitivity can be set to High, Medium, or Low according to your requirements. For example, to prevent frequent pauses caused by minor defects, set the detection sensitivity to Low.

Spaghetti Detection: The nozzle camera and live view camera capture images at fixed intervals, while an AI algorithm analyzes consecutive frames. Once filament tangling or stable filament clumps are detected below the nozzle, this feature is triggered.

Material Pile-up Detection: The live view camera monitors the filament purge location to identify any accumulation of waste material, preventing excessive scraps from causing toolhead collisions or skipped steps.

Clumping Detection: The nozzle camera identifies when the nozzle is completely covered by filament. The system then issues a warning and guides the required corrective actions.

Air Printing Detection: The nozzle camera monitors the distance between the nozzle and the model as well as the extrusion status, and identifies cases of no extrusion or minimal leakage. The following conditions may prevent proper operation of the nozzle camera's smart detection. Please troubleshoot accordingly:

- Confirm that the nozzle camera is correctly installed and functioning properly.
- Check that the nozzle camera lens is clean and that its light is operational to ensure clear image capture.
- When printing high-temperature materials, if the temperature near the nozzle exceeds 85 °C, the system will automatically pause smart detection to protect the camera.
- Ensure AI detection features are enabled in Print Options. Using third-party slicing software or G-code intended for other printer models may cause the detection feature to malfunction.

CAUTION

AI detection relies on good lighting conditions. Please ensure the printer's internal LED light is turned on. By default, the printer automatically turns on the LED light when printing the first layer. If you manually turn off the light, the system will not turn it on automatically again.

CAUTION

Spaghetti Detection cannot completely guarantee the prevention of print failures. This feature may occasionally produce false positives. Detection performance may be reduced when using black or dark-colored materials.

6.4.2 Build Plate Detection

On the printer touchscreen, tap  > **Settings** > **Print Options** to enable the build plate detection feature. The live view camera will detect the presence and type of the build plate on the heatbed. If the actual build plate used does not match the settings in the slicer file, the system will pause the print to prevent failure.

6.4.3 Hotend Type Detection

The hotend type can be detected using the following methods:

- On the printer touchscreen, select  > **Nozzle & Extruder**, then tap **Read Nozzle Info**.
- Each time a print job is sent to the printer, the system automatically verifies the hotend on the current toolhead against the hotend type specified in the sliced file.

The live view camera captures images of the toolhead hotend to perform in-place and type detection, displaying the results on the screen to prevent print failures caused by a mismatch between the installed hotend and the sliced file.

6.4.4 Live View Camera Calibration

After replacing or disassembling the live view camera, to ensure the accuracy of functions such as spaghetti detection and material pile-up detection, as well as reliable Intelligent detection during the printing process, it is necessary to perform live view camera calibration. On the printer touchscreen, go to  > **Calibration** > **Live View Camera Calibration**, then tap **Start** to begin calibration.

NOTE

The printer is calibrated at the factory before shipment, so routine calibration of the live view camera is generally not required during normal operation.

6.5 Air Condition System

The air conditioning system automatically adjusts the temperature and airflow inside the chamber, providing the optimal printing environment for different materials. The system automatically switches between heating and cooling modes based on filament type, preventing warping of high-temperature filaments and clogging of low-temperature ones. It also dynamically adjusts fan speed and heating power to maintain stable temperatures, save energy, and reduce noise, ensuring print quality and efficiency across various printing scenarios.

6.5.1 Select Mode

This printer automatically selects the appropriate chamber condition mode for each filament type. You do not need to set it manually, as Bambu Studio automatically configures it during slicing based on the filament type.

You can also switch modes from the printer screen by selecting  > **Air Condition**.

Cooling Mode

Suitable for printing filaments with low heat resistance, such as PLA and TPU. When the printer is idle or printing without chamber heating enabled, it operates in cooling mode. In this mode, the chamber heat circulation fan remains off.

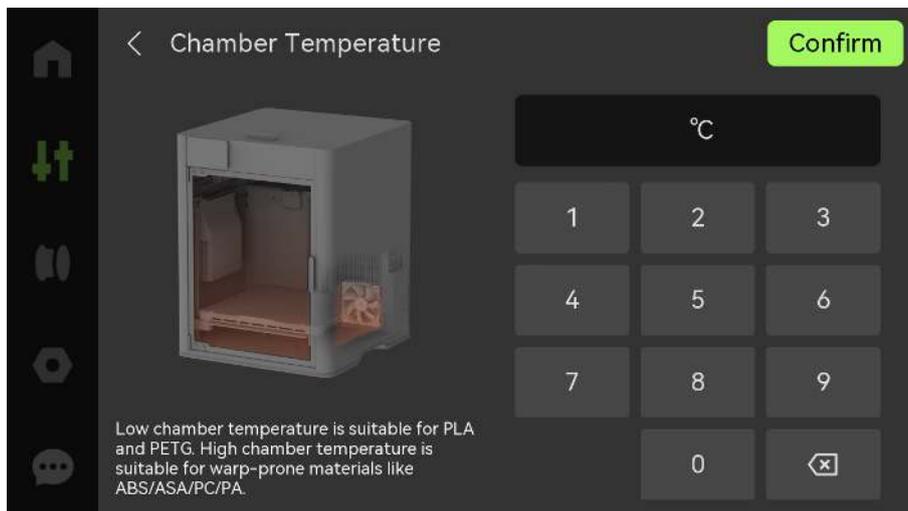
Heating Mode

Suitable for printing filaments with high heat resistance, such as ABS, ASA, PC, and PA. When the chamber temperature is set and heating begins, the system switches to Heating Mode.

In this mode, the chamber heat circulation fan turns on automatically, while the auxiliary part cool fan remains off.

6.5.2 Custom Chamber Temperature

You can set the chamber temperature via the printer screen, Bambu Studio, or Bambu Handy. The system will automatically switch to heating mode.



Chapter 7 Basic Controls and Functions

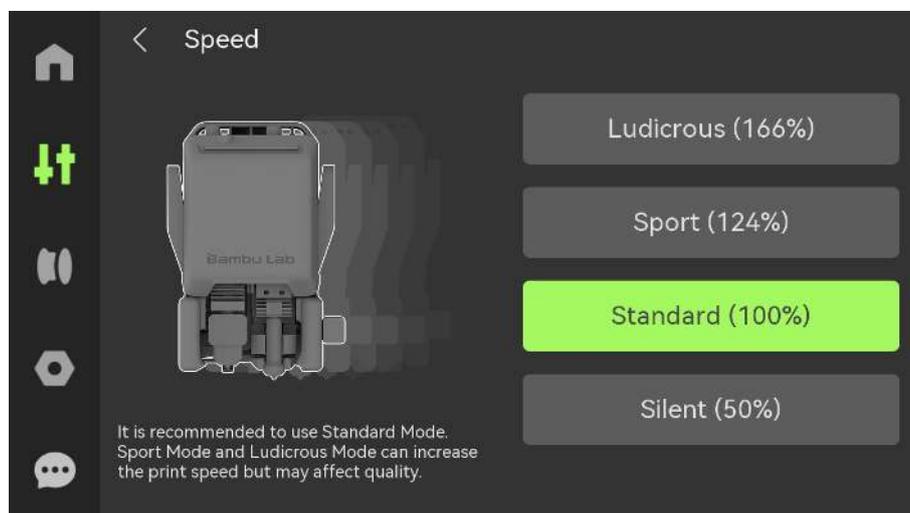
This chapter introduces the main control methods and basic functions of the printer, including touchscreen operation, basic settings, print status monitoring, and common functions. By understanding these operations, you can quickly get started with daily use and maintenance of the printer.

7.1 Control from Printer Touchscreen

The printer is equipped with a full-color touchscreen that displays device status and provides an interactive control interface, allowing you to configure multiple printer settings.

7.1.1 Speed Settings

You can adjust the print speed during printing, which affects both the print time and the surface quality of the model. Generally, increasing the speed will shorten the print time, but may result in poorer surface quality. Reducing the speed can improve surface quality, but will require more time to print. On the touchscreen, select  > **Speed** to adjust the speed by 4 modes:



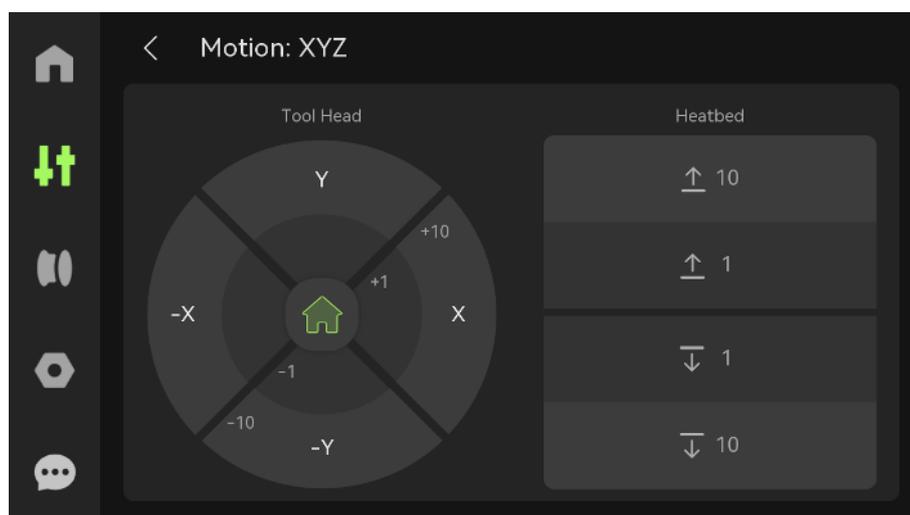
- Ludicrous: 166% of normal print speed and acceleration.
- Sport: 124% of normal print speed and acceleration.
- Standard: normal print speed and acceleration.
- Silent: 50% of normal print speed and acceleration.

You can flexibly adjust the print speed based on actual needs during printing. For example, when encountering intricate details or overhanging structures, lowering the speed improves surface quality and printing success rate.

7.1.2 XYZ Axis Movement

Control the movement of the toolhead and heatbed when the printer is idle. For example, when you need to clean or maintain the printer, you can use the touchscreen to move the toolhead and heatbed for more space.

On the touchscreen, select  > **Motion** and move the toolhead and heatbed.

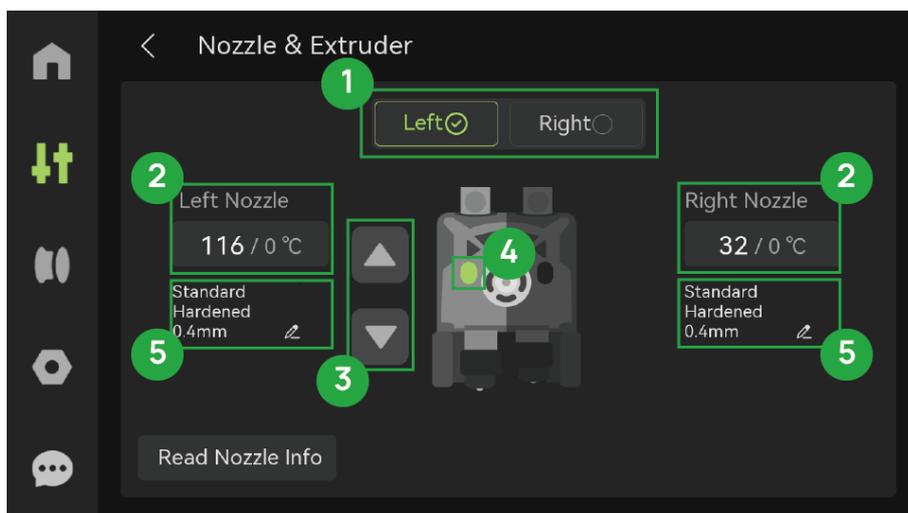


- **Toolhead:** Tap the +X/-X and +Y/-Y buttons to move the toolhead. Tap the 1 or 10 button to move the toolhead along the X-axis and Y-axis.
- **Heatbed:** Tap the 1 or 10 button to raise or lower the heatbed.

7.1.3 Nozzle and Extruder

Used to switch between left and right nozzles and precisely set the temperature, type, material, and diameter for each. You can also observe filament feeding status and manually control extrusion or retraction. Facilitates routine maintenance, filament replacement, nozzle cleaning, and pre-print preparation.

You can set the nozzle and extruder parameters on the printer touchscreen by selecting **Settings**  > **Nozzle and Extruder**.



1. **Switch nozzle:** Select left or right nozzle on the touchscreen to switch between the dual nozzles. The hotend and the flow blocker operate in a linked mechanism: when one hotend is lowered, the flow blocker shifts to the other hotend.
2. **Set nozzle temperature:** Click to enter a value and set the temperature for the left or right nozzle accordingly.
3. **Control the extruder:** Manually extrude or retract 1 cm of filament using the up or down button.
4. **Extruder indicator light:** When the extruder's green light is on, it indicates that the Hall sensor has detected filament entering the extruder.
5. **Set nozzle information:** Click **Read Nozzle Info** to let the printer automatically read the nozzle type, material, and diameter; or click the corresponding nozzle's **Edit** button to manually configure the left or right nozzle information.

i TIPS

Before reading nozzle information, please clear the build plate. During the identification process, do not reach into the printer.

NOTE

The flow blocker is mounted on the flow blocker lever and is used to seal the inactive nozzle to prevent oozing. When the printer is powered on, you can select the left or right nozzle on the touchscreen to automatically switch between the nozzle and the flow blocker. When powered off (maintenance mode), you can manually toggle the flow blocker lever to switch nozzles for easier operation.



7.1.4 Heatbed and Chamber Temperature

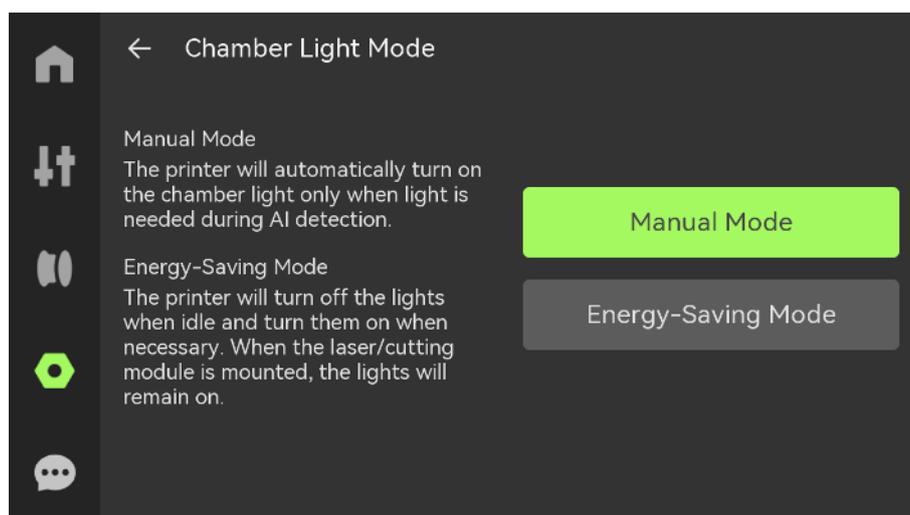
On the printer touchscreen, select  > **Heatbed** or  > **Chamber**, then enter the desired heatbed or chamber temperature.



7.1.5 Chamber Light Mode

Turning on the chamber light allows you to monitor the printing process and chamber in real time. Additionally, AI detection relies on adequate lighting, so enabling the chamber light improves detection accuracy.

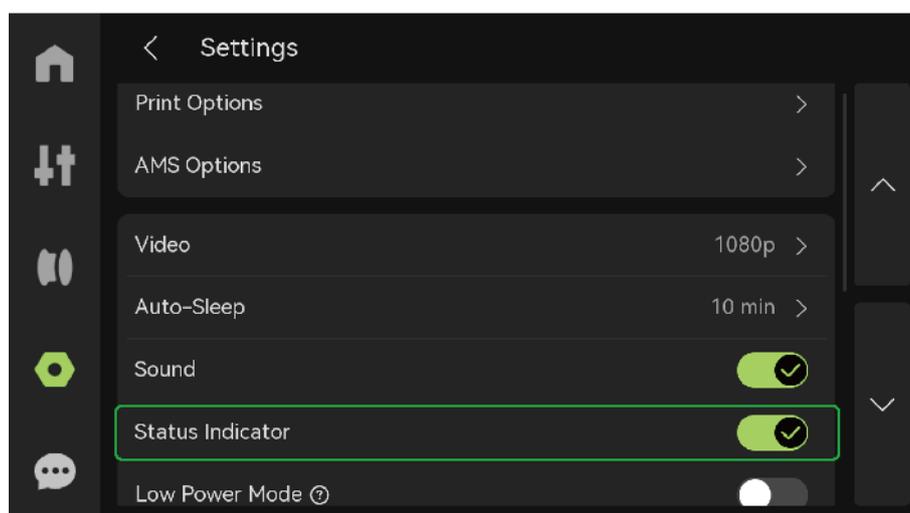
On the touchscreen, select  > **Settings** > **Chamber Light Mode**.



- **Manual Mode:** The chamber light turns on automatically only when AI detection is required. You can also manually turn the light on or off via the touchscreen when checking the print or performing maintenance.
- **Energy-Saving Mode:** The light automatically turns off when the printer is idle and only turns on when needed to save energy, such as during printing.

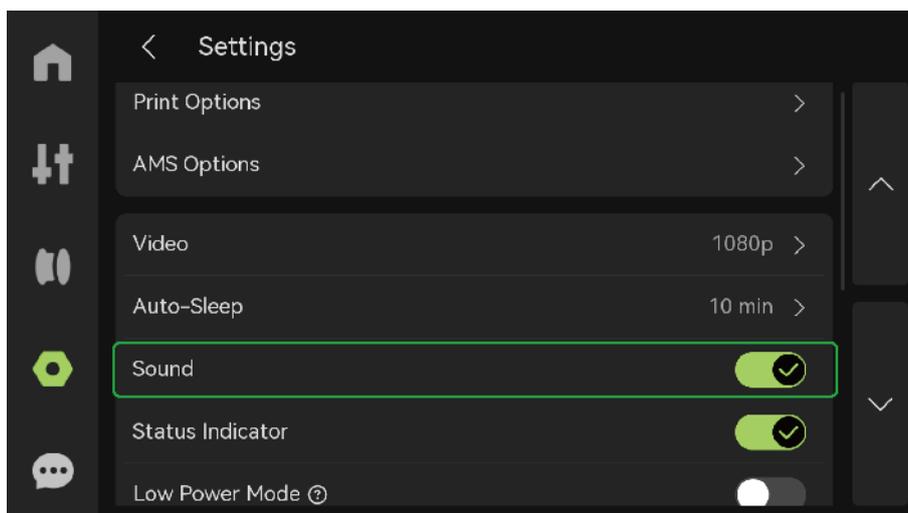
7.1.6 Status Indicator

Used to indicate the printer's health status and print job status. On the touchscreen, tap > **Settings** > **Status Indicator**. Then turn it on or off (see [Printer and AMS Status](#)).



7.1.7 Sound

Enable the sound option so that the printer will emit a sound when it is turned on, when printing begins, and when printing is complete. On the touchscreen, select > **Settings** > **Sound** to turn it on or off.



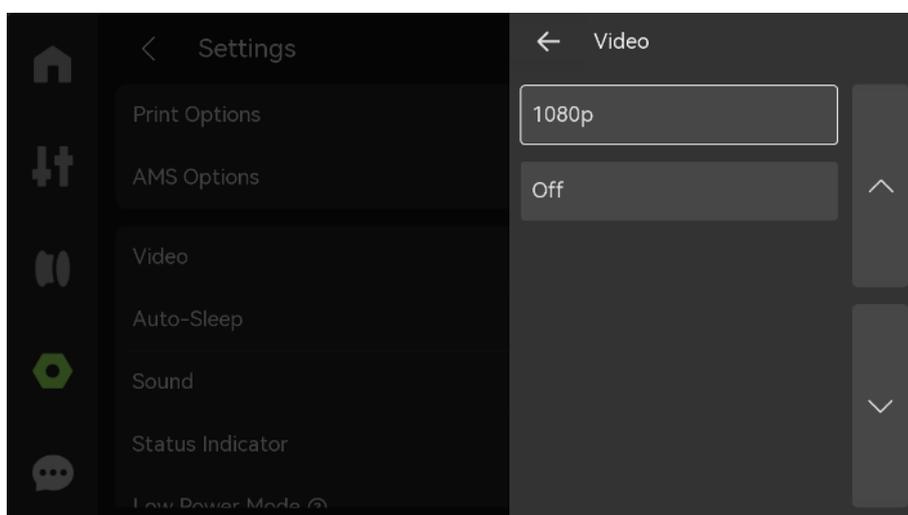
7.2 Photo and Video

The printer is equipped with a 1920x1080 HD live view camera mounted at the upper left of the printer's front panel. This camera is primarily used for real-time monitoring, time-lapse photography, photo and video recording, and intelligent detection. This chapter focuses on the printer's photo and video recording functions.

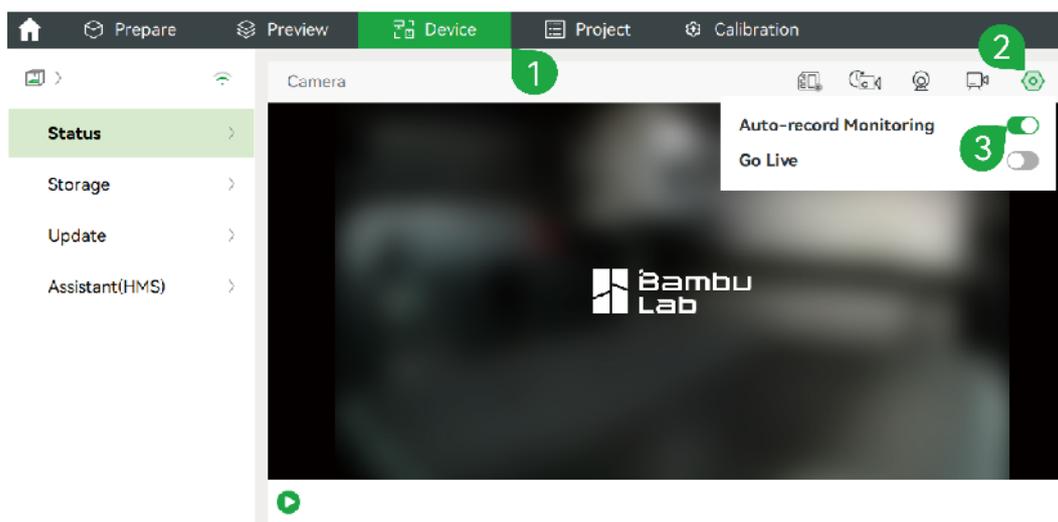
7.2.1 Video Recording

The printer is able to record the print process. This makes it easy to check the overall print status and serves as a valuable reference for troubleshooting and after-sales support. The printer has an USB port to connect a USB flash drive to save videos. Please insert a USB flash drive before enabling this feature. You can enable or disable video recording using the following methods:

Method 1: On the touchscreen, select **Settings** > **Video**. Turn the function on or off or adjust the resolution.

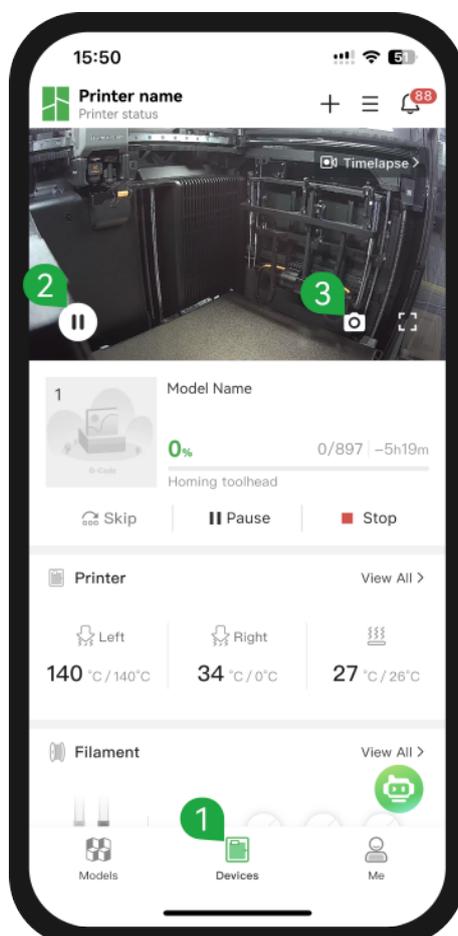


Method 2: In Bambu Studio, select **Device** > **Camera** >  > **Auto-record Monitoring** to enable the video recording function.



7.2.2 Photo

You can capture photos to record the model printing process. Using the printer's live view camera, you can remotely monitor printer operations and easily capture photos inside the printer to record the printing progress.



Step 1: Go to **Device** in Bambu Handy.

Step 2: Tap  to start viewing.

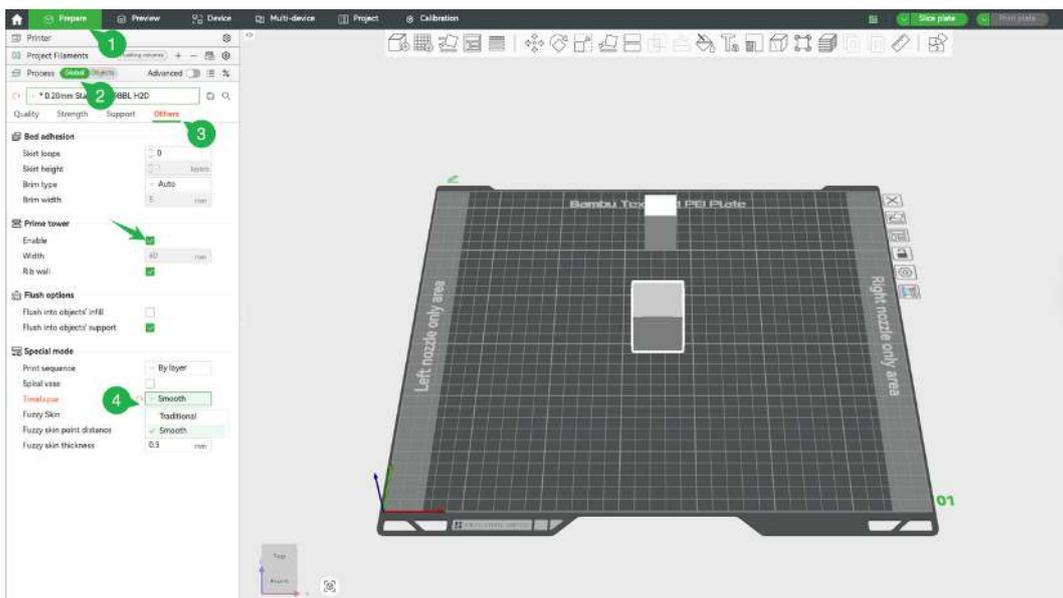
Step 3: Tap  to capture a photo.

7.2.3 Timelapse

The time-lapse function automatically takes a photo after each layer is printed and combines all photos into an accelerated video. There are two time-lapse modes:

1. **Traditional mode (default):** After each layer is printed, the printer takes a photo at the current position, and the toolhead stays at the end of the layer. As a result, you can see the toolhead moving in the video.
2. **Smooth mode:** After each layer is printed, the toolhead moves to the prime tower or a safe position before taking a photo so the toolhead remains stationary in the video. This mode automatically generates a prime tower.

In Bambu Studio, go to **Prepare** to select a time-lapse mode. Navigate to **Process > Global > Others > Special Mode > Timelapse**.



TIPS

If you select the smooth mode, please ensure prime tower is enabled (enabled by default).

Based on the heatbed height changing by 5 mm with each nozzle switch, the printer will take a photo at the correct time to create a smooth video. The heatbed will not be "jumping" in the video.

How to Enable

When starting a print job, you can manually enable or disable the time-lapse function on the touchscreen, Bambu Studio, or Bambu Handy.

- On the printer touchscreen: Tap **the print preview** and find **Timelapse** in the bottom right of the interface to turn it on.
- Bambu Studio: After slicing finishes, click **Print plate/print all** in the upper-right corner. In the pop-up window, click **Open** of **Timelapse**.
- Bambu Handy: Go to **Prepare to Print**, navigate to **Options > Timelapse** at the bottom, and then **Enable** it.

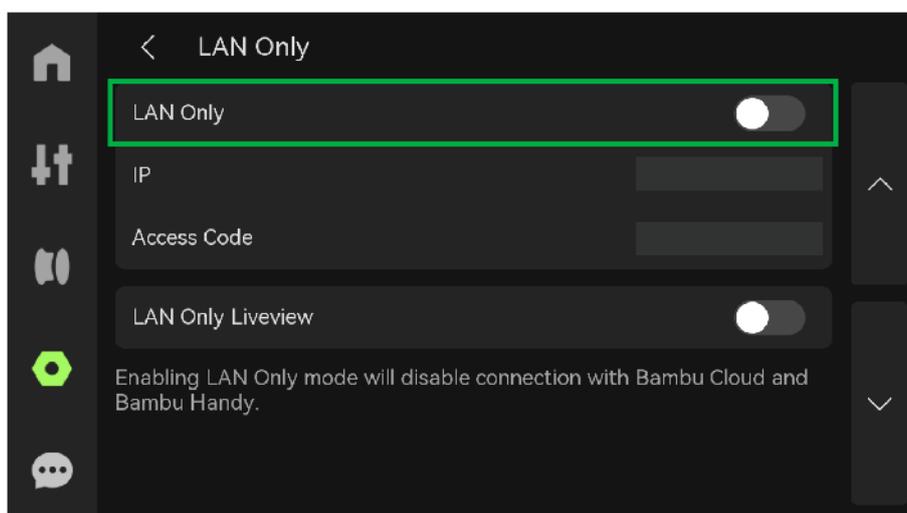
7.3 Connect to Network

This printer supports Wi-Fi and LAN-only mode. When LAN-only mode is enabled, you can further configure parameters in the developer mode.

7.3.1 LAN Only Mode

When enabled, the printer can only be accessed and controlled within the local area network. It cannot be accessed remotely via the internet, nor can it use cloud features such as Bambu Handy and print history. This mode is suitable for high demands on data security.

On the touchscreen, you can select  > **Settings** > **LAN Only** and enable LAN-only mode.

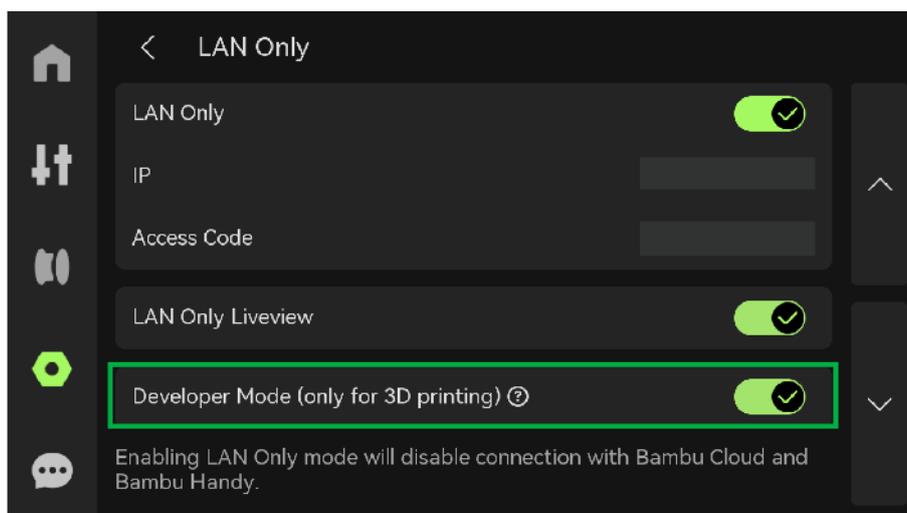


7.3.2 Developer Mode

When enabled, the printer allows third-party software or devices to directly control and manage print tasks and process data. This offers greater flexibility and interactivity, but may pose safety risks.

After enabling LAN-only mode, you can enable developer mode in the following settings:  >

Settings > LAN Only > Developer Mode (only for 3D printing).



7.4 Connect USB Flash Drive

The printer has a USB port to connect a USB flash drive with sliced files. Then, you can start printing directly from the touchscreen. In addition, you can also save print logs, time-lapse videos, and cache data to the USB drive.

7.4.1 Specification

1. USB 2.0 or higher (with a minimum write speed of over 10 MB/s) is required; supported file systems are FAT32 and exFAT.
2. The printer does not limit the USB flash drive's capacity. **The maximum supported capacity depends on the file system:**

File System	Operating System Used for Formatting	Maximum Capacity	Maximum Size for Single File
FAT32	Linux or macOS	2 TB	4 GB
	Windows	32 GB	4 GB
exFAT	Any operating system	128 PB	16 EB

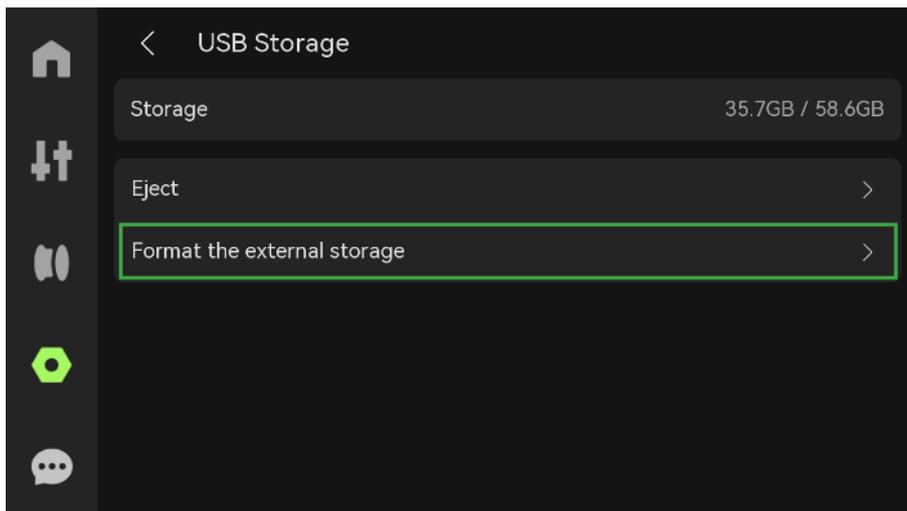
NOTE

- Video files can use up to 65% of the USB flash drive's capacity, with a maximum of 500 files.
- 15% of the drive's capacity is reserved as free space. If data on the drive (including hidden files) exceeds 85% of capacity, recording will be disabled.
- When available space is above 15%, the printer can record normally. When the drive is 85% full, the printer will automatically delete the oldest video files to free up space.

7.4.2 Connect and Format

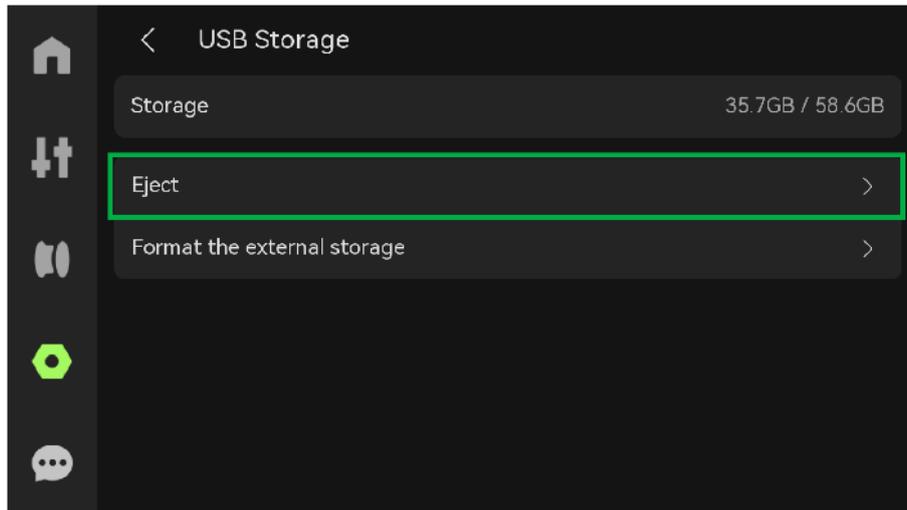
Step 1: Insert a compatible USB flash drive into the printer's USB port.

Step 2: Format the USB drive (if needed). On the touchscreen, tap  > **USB Storage** > **Format External Storage** to format the drive as FAT32 directly on the printer (suitable for USB flash drives with up to 2 TB).



7.4.3 Ejection

To eject the USB flash drive, select  > **USB Storage** > **Eject** to safely remove the drive and prevent data corruption.



NOTE

The printer only supports one USB flash drive at the same time. You cannot connect multiple drives via a USB hub.

7.5 Update and Restore

7.5.1 Update Firmware

The printer firmware will be continuously updated for new features. When a new firmware version is available, you can update it using the following methods:

Online Update

When the printer is connected via Wi-Fi, it will check for new firmware updates. You can update the firmware online through the printer's touchscreen.

Step 1: When new firmware is available, a prompt will appear on the touchscreen asking if you want to update. Tap **Yes** and follow the instructions to complete the update.

Method 2: You can also select  > **Firmware** > **Update** on the screen to manually update.

Offline Update

If the printer is offline or cannot access Bambu Lab cloud services, you can update the firmware using a USB flash drive and an update package.

- Step 1: Insert and format a USB drive.
- Step 2: Download the corresponding offline update package from the Bambu Lab firmware download webpage <https://bambulab.com/support/firmware-download/h2d>.

- Step 3: On the touchscreen, select  > **USB Storage** > **Eject** to safely remove the USB flash drive.
- Step 4: Copy the offline update package to the root directory of the USB flash drive.
- Step 5: Reinsert the USB flash drive. On the touchscreen, select  > **Firmware** > **Offline Update**. Then, select the version to update to, and tap **Update**. Once it is complete, the printer will automatically restart.

7.5.2 Initialization

Initialization can restore the printer to its factory settings, clearing all custom configurations and user data. Please note that this action is irreversible. Ensure there is no important data before proceeding.

On the touchscreen, select  > **Settings** > **Restore to Factory Settings** to complete setup.

Chapter 8 Filament

8.1 Select the appropriate filament

Different 3D printing filaments have unique physical properties, making them suitable for different applications. Choosing filaments based on printing needs helps improve print quality and final product performance. Below is a brief introduction to several common filament types. For detailed characteristics and parameters, please refer to the Bambu Lab Filament Guide (bambulab.com/filament-guide).

8.1.1 Filament Types by Function

Basic Materials

- PLA

The most common entry-level 3D printing material, offering good printing stability and ease of use. It supports low-temperature printing with minimal warping and excellent detail resolution, but has poor toughness and limited strength.

Recommended use: every day prototyping, home printing, and low-load applications.

- PETG HF

Features good toughness, water resistance, heat resistance, and chemical resistance. However, it is prone to moisture absorption and can be easily scratched.

Recommended use: containers, durable parts, and functional components.

Engineering Materials

- ABS

Features good heat resistance and toughness. However, it is very prone to warping while printing.

Recommended use: mechanical housings, functional parts.

- ASA

Similar to ABS but with superior UV resistance.

Recommended use: outdoor constructions, automotive parts.

- PC

Offers excellent heat resistance, impact resistance, and rigidity. However, it requires strict environmental control during printing.

Recommended use: high-strength structural parts, high-temperature applications, and mechanical components.

Flexible Materials

- TPU

TPU is a highly elastic and wear-resistant material, but it prints relatively slowly. Its hardness is defined by the Shore A scale: the lower the value, the softer the filament. For example, TPU 85A is softer than TPU 90A. Common hardness levels include 75A, 80A, 83A, 85A, 90A, 95A, etc.

Recommended use: flexible models such as shoe sole prototypes, seals, cushions, etc.

TIPS

It is recommended to print with TPU of 80A hardness and above. Higher hardness improves printing stability and reduces the risk of print failure.

Glass/Carbon Fiber Reinforced Materials

- -CF (Carbon fiber reinforced composites)

Carbon fiber is added to standard base materials (such as PLA, PETG, PA) to significantly enhance rigidity and strength while maintaining lightweight properties.

Recommended use: load-bearing structural parts and lightweight structural designs.

- -GF (Glass fiber reinforced composites)

Glass fiber is added to improve the toughness and wear resistance of the material.

Recommended use: industrial mechanical parts, structural frame components.

Support Materials

Support materials are specially optimized for easy removal and good stability. They are intended only for printing support structures; using them as the main material may decrease model strength and surface quality.

- PVA (Water-soluble support material)

A flexible, biodegradable polymer with strong moisture absorption. It absorbs water from the air and dissolves in water. PVA is commonly used as a water-soluble support material in 3D printing.

Recommended use: serving as a support material when printing complex structures or models with supports that are difficult to remove manually.

8.1.2 Filament Types by Temperature

Based on extensive real-world data from Bambu Lab official materials, filaments can be classified as follows to ensure stable performance on the printer and to avoid softening or deformation during printing:

- **High-Temperature Filaments**

These filaments need to be printed at a higher chamber temperature to achieve strong layer adhesion and to effectively control shrinkage and other key print-quality parameters.

Common high-temperature filaments include: ABS, ASA, ASA-CF, PC, PA, PA-CF, PA-GF, PA6-CF, PET-CF, PPS, PPS-CF, PPA-CF, PPA-GF, ABS-GF, ASA-Aero.

- **Mid-Temperature Filaments**

This category offers excellent temperature resistance and can achieve high-quality printing results without chamber heating.

Common mid-temperature filaments include: HIPS, PE, PP, EVA, PE-CF, PP-CF, PP-GF, PHA.

- **Low-Temperature Filaments**

Due to their low heat deflection temperature, these filaments may soften and deform inside the extruder or hotend when the chamber temperature exceeds 45 °C, increasing the risk of clogging. Therefore, they must not be used in a high-temperature chamber environment.

Common low-temperature filaments include: PLA, PETG, PETG-CF, TPU, TPU-AMS, PLA-CF, PLA-AERO, PVA, BVOH, PCTG.

NOTE

Non-official filaments may have different heat deflection temperatures for the same material type due to formula differences. It is recommended to refer to ISO 75 (under a 1.8 MPa load) when determining material classification. If the heat deflection temperature is below 80 °C, treat the filament as low-temperature filament. Alternatively, consult the filament supplier directly for official classification advice to ensure optimal print compatibility.

CAUTION

Currently, only TPU 95A HF can be co-printed with high-temperature engineering materials. Other **high and low temperature filaments must not be mixed for printing**, as this may cause extruder or nozzle clogging and damage. Bambu Studio will automatically restrict the mixing of high and low temperature filaments during slicing.

8.2 Filament Compatibility and Parameter Settings

The H2D is a fully enclosed 3D printer, equipped with a hardened steel 0.4 mm nozzle by default, supporting a wide range of filaments from basic materials to high-performance engineering materials. The nozzle temperature can reach up to 350°C, and the heatbed up to 120°C, enabling stable printing of high-temperature materials such as ABS, PC, and PA-CF.

Filament	Compatible Nozzle Sizes	Nozzle Restrictions	Nozzle Temp (°C) (± 10 °C)	Hardened Steel Hotend Required	AMS 2 Pro Compatible	Dry Before Use
PLA*	All sizes	None	190 - 240	No	Yes	Recommended
PETG HF	All sizes	None	230 - 260	No	Yes	Required
ABS	All sizes	None	240 - 280	No	Yes	Recommended
ASA	All sizes	None	240 - 280	No	Yes	Recommended
PC	All sizes	None	260 - 290	No	Yes	Required
TPU 95A HF	0.2 mm not supported	Use right nozzle only	220 - 240	No	No	Required
PLA-CF	0.2 mm not supported	None	210 - 240	Yes	Yes	Recommended
PETG-CF	0.2 mm not supported 0.4 mm recommended	None	240 - 270	Yes	Yes	Recommended
PET-CF	0.2 mm not supported 0.6 mm recommended	Use left nozzle only	260 - 300	Yes	No	Required
PAHT-CF	0.2 mm not supported 0.6 mm recommended	None	260 - 300	Yes	Yes	Required
ABS-GF	0.2 mm not supported	None	240 - 280	Yes	Yes	Recommended

Filament	Compatible Nozzle Sizes	Nozzle Restrictions	Nozzle Temp (°C) (±10 °C)	Hardened Steel Hotend Required	AMS 2 Pro Compatible	Dry Before Use
PA6-GF	0.2 mm not supported 0.6 mm recommended	None	260 - 300	Yes	Yes	Required
PPA-CF	0.2 mm not supported 0.6 mm recommended	Use left nozzle only	285 - 320	Yes	No	Required
PPS-CF	0.2 mm not supported 0.6 mm recommended	Use left nozzle only	310 - 340	Yes	No	Required
PVA	0.2 mm not supported	None	190 - 240	No	Yes	Required

 **NOTE**

The nozzle sizes listed in this table include 0.2 mm, 0.4 mm, 0.6 mm, and 0.8 mm.

 **NOTE**

Here, **PLA*** refers to the standard type of PLA without carbon fiber (CF), glass fiber (GF), or other additives.

Included materials: PLA Basic, PLA Matte, PLA Tough, PLA Metal, PLA Silk, PLA Aero, etc.

Excluded materials: PLA Wood, PLA-CF, PLA-GF, PLA Sparkle, PLA Marble, PLA Glow, and other filled materials.

 **CAUTION**

Due to the brittleness of **PET-CF/PPS-CF/PPA-CF**, using the right hotend can cause them to break inside the PTFE tube during the homing routine. During X-axis homing, the tool-head moves to the rear of the heated bed and then scans left and right to detect the limit. When it contacts the right-side limit stopper, the PTFE tube bends more sharply—especially on the right hotend filament path—raising the risk of filament breakage. Therefore, it is recommended to print these filaments using the **left hotend**.

⚠ CAUTION

Using the left hotend to extrude **TPU 85/90/95A** has a higher risk of clogging. For this reason, it is currently recommended to use the **right hotend** only.

8.3 Filament Drying

8.3.1 Filament Drying with the Printer

You can dry filament directly using the printer. The following steps walk you through the process.

- Step 1. Prepare the filament drying container. You can use the original filament box or download and print the dedicated PC drying box from MakerWorld.
- Step 2. Remove any debris from the heatbed and the bottom of the printer chamber.
- Step 3. On the printer screen, select  > **Toolbox** > **Dry Filament**.
- Step 4. Tap **Prepare**, the printer will automatically move the toolhead and heatbed to the designated position.
- Step 5. Place the filament to be dried on the heatbed, then cover it with the cardboard box or PC box.
- Step 6. Select the filament type. The printer will automatically set the heatbed temperature and drying duration.
- Step 7. Tap **Start** to begin the drying process.

i TIPS

It is recommended to manually flip the filament halfway through drying for more even drying.

8.3.2 Filament Drying with AMS 2 Pro/AMS HT

The AMS 2 Pro and AMS HT are equipped with built-in drying modules that heat and dry the filament inside the chamber. The following section details the specific operation methods.

🔗 NOTE

The maximum drying temperature supported by AMS 2 Pro is 65 °C. For materials requiring higher drying temperatures, it is recommended to use AMS HT.

🔗 NOTE

The printer can power only one AMS 2 Pro at a time. To dry multiple AMS 2 Pro units simultaneously, additional AMS units must be connected to dedicated power adapters.

 **CAUTION**

Flexible and engineering materials must be dried before printing to avoid print failures or equipment damage.

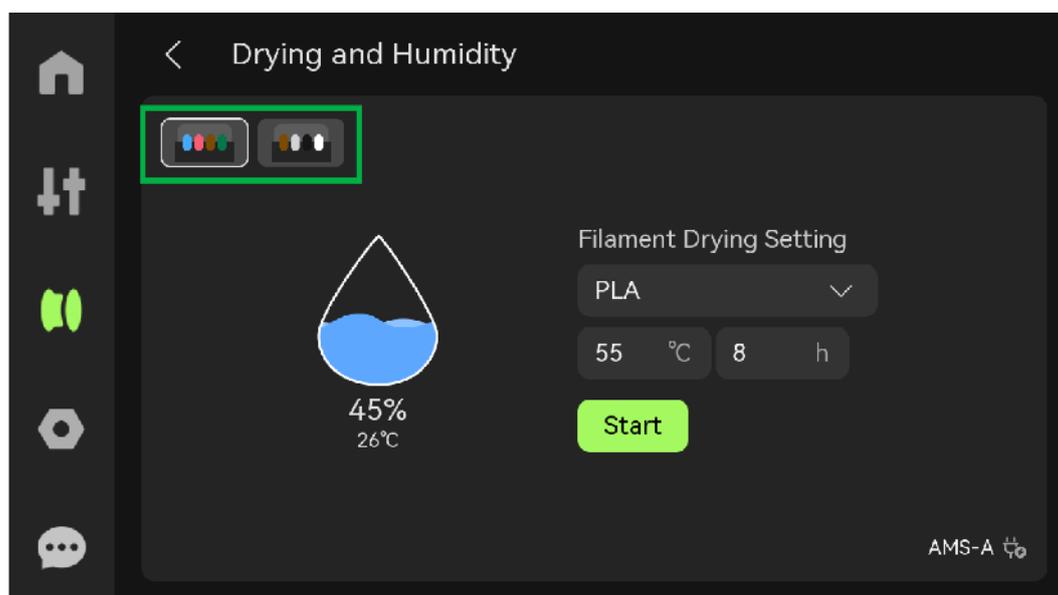
 **CAUTION**

To prevent the filament from being deformed by the gears due to softening during drying, the system automatically disables drying for AMS units involved in current printing or those in an auto-refill relationship with them.

- Step 1. Properly connect the printer with AMS 2 Pro/AMS HT following the quick start guide and complete AMS Setup (see [Load Filament from AMS 2 Pro](#)).
- Step 2. Place the filament to be dried into the AMS slot, retract any loaded filament, and close the lid.
- Step 3. On the printer touchscreen, select  > .
- Step 4. Choose the filament type, and the printer will automatically adjust the default drying temperature and duration. Tap **Start** to begin drying.

 **TIPS**

If multiple AMS units are connected, you can start drying for each device individually from the filament page or switch between AMS units from the top-left corner of the drying page.



8.4 TPU Printing Guide

Pre-print Preparation

TPU 85A and TPU 90A are two flexible materials with different hardness levels; 85A is softer than 90A. To ensure smooth TPU printing, complete the following preparations before printing.

NOTE

To prevent filament jams, do not load flexible materials such as TPU with a hardness of 95A or lower, or moisture-absorbed PVA and BVOH into the AMS.

- Store filaments in a storage box with desiccants to maintain ambient humidity below 20% RH.
- Thoroughly dry the filament before printing. For details, please refer to [Filament Drying](#).
- It is strongly recommended to use the top filament rack for feeding assistance. You can download the corresponding model from MakerWorld (makerworld.com) by searching for "**H2D Flexible Filament Top-feed Rack**".
- It is recommended to use a new hotend or a dedicated TPU hotend.
- Clean the textured PEI build plate. In most cases, glue is not required. If TPU adheres too strongly, glue can be used as a release layer for easier model removal.
- Ensure key components such as the extruder, cutter, and hotend are in good condition. Clean or replace them if necessary. For details, please refer to [Replace Accessories](#).

Scan the QR code or click the link to watch the TPU printing video guide.



wiki.bambulab.com/h2/h2d-tpu-printing-guide

8.4.1 TPU 85A

CAUTION

- TPU 85A cannot be fed through the PTFE tube, as excessive resistance may prevent smooth extrusion. Please feed the filament directly into the toolhead.
- TPU 85A requires a 0.6 mm hotend for printing. Please replace with a 0.6 mm hotend before printing and set the hotend diameter to 0.6 mm on the screen.

Step 1. Prepare for filament loading.

1. Press the pneumatic connector and remove the PTFE tube connected to the right toolhead filament inlet.
2. Press the pneumatic connector and remove the PTFE tube connected to the upper part of the filament buffer, then pull the tube out from the cable chain clip.
3. Prepare a 5 cm PTFE tube, slightly bend it, and insert it into the filament outlet of the AMS HT/sealed box. Place the filament inside and close the top cover.
4. Adjust the height of the AMS HT/sealed box so that the filament outlet is slightly higher than the printer body.



Step 2. Load the filament.

1. On the screen, select **⚙️ > Nozzle & Extruder** to set the right hotend temperature to 250 °C.
2. After the hotend heats up to 250 °C, manually push the filament into the extruder, then tap **▶** on the screen to extrude the filament.

NOTE

Do not tap rapidly and continuously to avoid TPU being caught in the extruder gear and causing a clog.

Step 3. Adjust slicing parameters.

- Print temperature: 225 °C (recommended)
- Bed temperature: 30 – 35°C
- Air condition: Cooling mode
- Volume speed: Keep default
- Flow dynamics calibration: Auto or Off

Step 4. Adjust model position.

Place the model toward the front center area of the build plate to reduce resistance along the filament loading path and improve printing stability.

Step 5. Choose either of the following methods to unload TPU after the print is complete.

- Method 1: On the printer touchscreen, select  > **TPU Filament** > **Unload**.
- Method 2: Manual filament unload (recommended). Heat the right hotend to 250 °C, tap **Unload**, and gently pull out the filament by hand as the extruder gears rotate.

8.4.2 TPU 90A

Step 1. Prepare filament for loading.

1. Press the PTFE tube coupler, pull out the PTFE tube connected to the upper part of the filament buffer, and insert it into the dedicated TPU filament inlet. Adjust the toolhead position so that an appropriate length of PTFE tube extends from the TPU filament inlet.
2. Press the PTFE tube coupler and remove the PTFE tube connected to the right toolhead filament inlet.
3. Prepare a 5 cm PTFE tube, slightly bend it, and insert it into the filament outlet of the AMS HT/sealed box. Then place the filament and insert it into the PTFE tube. Finally, close the top cover.
4. Adjust the height of the AMS HT/sealed box so that its filament outlet is level with the TPU filament inlet.

**i TIPS**

If using a top rack to assist loading, please visit the Makerworld model page for detailed instructions and installation guidance.

Step 2. Load the filament.

1. Pull the filament from the filament outlet of AMS HT or sealed box and manually push it into the printer's TPU filament inlet until the filament comes out of the other end of the PTFE tube.



2. On the printer screen, tap **⚙** > **Nozzle & Extruder**, and set the right hotend temperature to 250 °C.

3. After the hotend heats up to 250 °C, manually push the filament into the extruder, then tap  on the screen to extrude the filament.

 **CAUTION**

Do not tap rapidly and continuously to avoid TPU being caught in the extruder gear and causing a clog.

4. Observe the right hotend. Once the filament is continuously extruded, reinsert the PTFE tube into the right toolhead filament inlet.



Step 3. Adjust slicing parameters.

- Print temperature: 225 °C (recommended)
- Bed temperature: 30 – 35 °C
- Air condition: Cooling mode
- Volume speed: Keep default
- Flow dynamics calibration: Auto or Off

Step 4. Adjust model position.

Place the model toward the front center area of the build plate to reduce resistance along the filament loading path and improve printing stability.

Step 5. Choose either of the following methods to unload TPU after the print is complete.

- Method 1: On the printer touchscreen, select  > **TPU Filament** > **Unload**.
- Method 2: Manual filament unload (recommended). Heat the right hotend to 250 °C, tap **Unload**, and gently pull out the filament by hand as the extruder gears rotate.

8.5 High-Temperature Filament Printing Guide

The printer features an enclosed design, equipped with **hotends up to 350 °C** and **active chamber heating up to 65 °C**, ensuring stable printing conditions and reducing warping risks.

Supported high-temperature filaments include ABS, ASA, PC, Nylon, and others. These filaments offer excellent performance and durability but require specific printing conditions. The following steps will assist you in achieving optimal printing results.

⚠ CAUTION

- Do not use a 0.2 mm nozzle. A 0.6 mm nozzle is recommended, followed by a 0.4 mm nozzle.
- It is recommended to use a new nozzle or set the nozzle temperature to 280 °C and clean its interior before printing to ensure it is free of clogs.

Step 1. Dry the filament. For detailed steps, see [Filament Drying](#).

ℹ TIPS

Certain filaments require higher drying temperatures, which the AMS 2 Pro cannot provide. For better drying results, it is recommended to purchase the AMS HT.

Step 2. Apply a layer of solid or liquid glue on the build plate surface.

🔧 NOTE

You can purchase official heatbed solid glue on the Bambu Lab official platform.

Step 3. Preheat the printer.

Before starting the print, set the chamber temperature to the desired value (such as 65 °C) on the printer screen, Bambu Studio, or Bambu Handy. The printer will automatically switch to **heating mode**.

To heat up faster, set the heatbed temperature to 100 °C to assist in raising the chamber temperature and simultaneously preheat the heatbed.

Step 4. Recommended model slicing settings.

- Set the layer height between 0.15 and 0.3 mm, with a maximum print speed of 100 mm/s.
- When printing multiple parts, set the print sequence to By object.

- For large models, choose the Strength preset (6 walls, 25% sparse infill density).

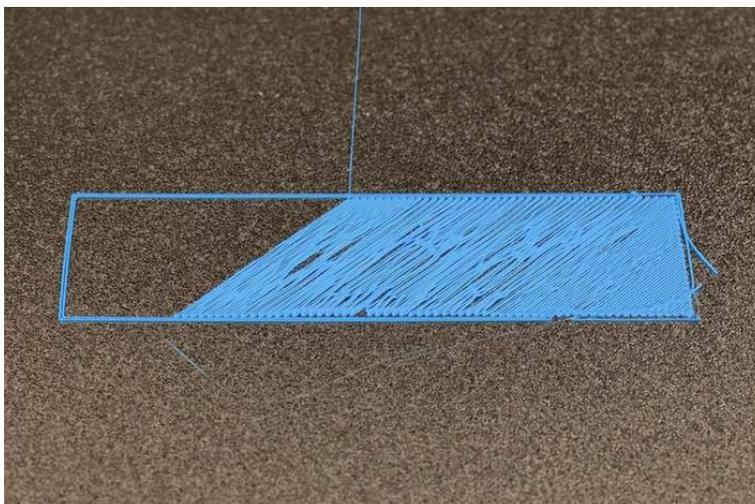
Step 5. After printing, allow the model to cool down to room temperature gradually within the chamber before removal.

i TIPS

If printing carbon-fiber (CF) or glass-fiber (GF) reinforced engineering materials, post-print annealing is recommended. Visit the Bambu Lab Wiki (wiki.bambulab.com/home) and search for "annealing" to find related guidance.

Chapter 9 Print Quality Issues and Solutions

9.1 First Layer Not Sticking



Possible Causes and Solutions

1. Mismatched build plate

Ensure the build plate type matches the filament used, and select the correct build plate in Bambu Studio.

2. Dirty build plate surface

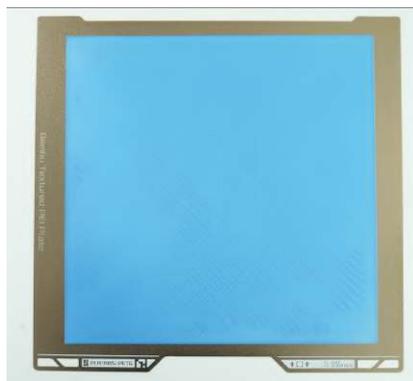
Clean the build plate surface with warm water and a neutral detergent to remove any contaminants without damaging the build plate coat.

3. Bed leveling not performed

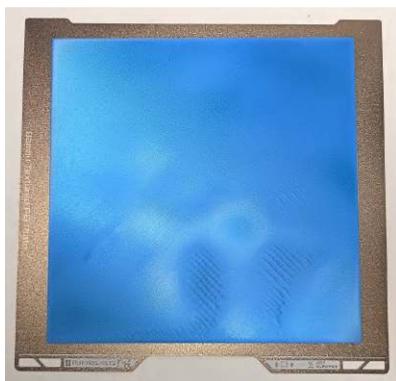
On the printer screen: select  > **Calibration** > **Print Calibration** > **Auto bed Leveling/High-temperature Bed Leveling** to perform bed leveling.

In Bambu Studio, go to the device page, select **Calibration** > **Bed Leveling** to perform bed leveling.

9.2 First Layer Too High/Too Low



Full-plate first layer too low



Partial first layer too low



Full-plate first layer too high

Possible Causes

- The first layer being too low (partial area or full-plate) may cause print failure or damage to the nozzle or build plate.
- The first layer being too high can result in poor adhesion, poor surface quality, or reduced structural strength, and in severe cases, model collapse or shifting, leading to a print failure.

Solutions

- On the printer screen, go to  > **Calibration** > **Print Calibration**, and run Auto Bed Leveling and High-temperature Bed Leveling.
- If the issue persists after bed leveling, visit the Bambu Lab Wiki (wiki.bambulab.com/home) and search for "H2D First Layer" to obtain a detailed tutorial.

9.3 Poor Overhang Quality

When printing overhang structures, if the extruded filament does not cool and solidify quickly, it tends to sag, affecting both appearance and structural integrity.



Possible Causes and Solutions

1. **Insufficient support**

Check the overhang angle. If it exceeds 45° , support structures should be added.

2. **Print speed too high**

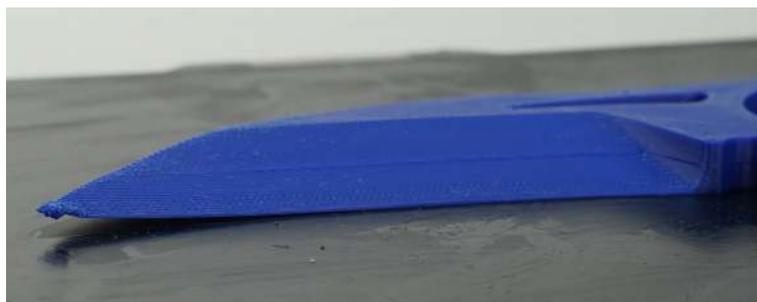
Reduce the printing speed appropriately or enable the Slow down for overhangs function.

3. **Insufficient cooling**

Lower the nozzle temperature as appropriate, increase the fan speed for the auxiliary part cooling fan and the part cooling fan, and lastly open the front door and top glass cover to help reduce the chamber temperature.

9.4 Model Warping, Falling Off, or Collapse

During printing, localized shrinkage or poor bed adhesion may cause the model to warp, detach, or collapse. Warped areas often exhibit a noticeable horizontal bulge, caused by the nozzle being too close to the warped surface. This results in flattening of the extruded material and subsequent extrusion overflow.



Possible Causes and Solutions

1. Tall model with unstable center of gravity

Add supports, reduce print speed and acceleration. Lay the model flat whenever possible, split the model for printing if necessary.

2. Nozzle collision with the model

Clean the nozzle and slightly increase the nozzle temperature. Reduce print and infill speeds. Choose sparse infill patterns like Gyroid or Concentric to avoid crossing paths.

3. High material shrinkage rate

For large prints, select filaments less prone to warping, such as PLA.

4. Improper printing environment or settings

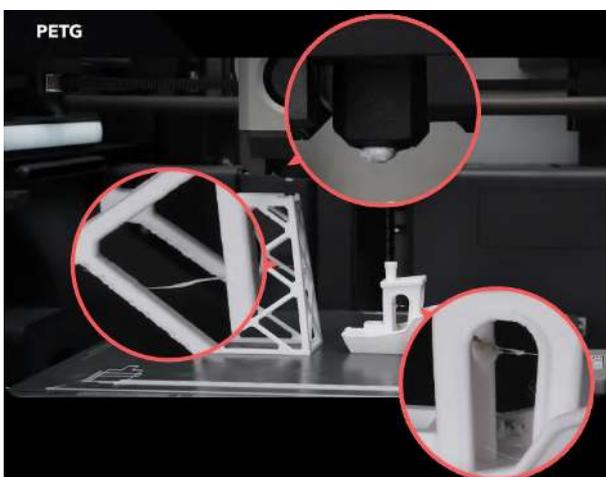
Raise the heatbed temperature appropriately, close the printer's front door and top glass cover, reduce fan speed, lower sparse infill density, and select non-crossing infill patterns like Gyroid.

5. Insufficient bed adhesion

Verify that the build plate information in Bambu Studio matches the actual build plate. Clean the build plate and nozzle. Enable Brim and increase the brim width. Apply glue to the build plate surface if needed to enhance bonding and raise the heatbed temperature slightly.

9.5 Filament Sticking to the Nozzle

During printing, small amounts of molten filament may stick to the nozzle surface. This can cause local under-extrusion, rough surfaces, and—in severe cases—nozzle clog or clumping. This issue is common with high-viscosity filaments such as PETG.



Sticky nozzle



Clean nozzle

Possible Causes and Solutions

1. **Moist filament**

Dry the filament. For detailed steps, see [Filament Drying](#).

2. **Excessive flow rate or worn nozzle**

Adjust the flow rate to reduce over-extrusion, and replace the nozzle if necessary.

3. **Print speed too slow or temperature too high**

Match print speed with nozzle temperature: increase nozzle temperature when printing at higher speed or higher volumetric speed; lower nozzle temperature when printing slowly.

4. **Worn, loose, misaligned, or damaged nozzle wiping part**

Identify issues by visually inspecting, manually moving the wiping part, or pushing the tool-head against it after powering off. Repair or replace it if abnormalities are found.

5. **Too many travel paths in the model**

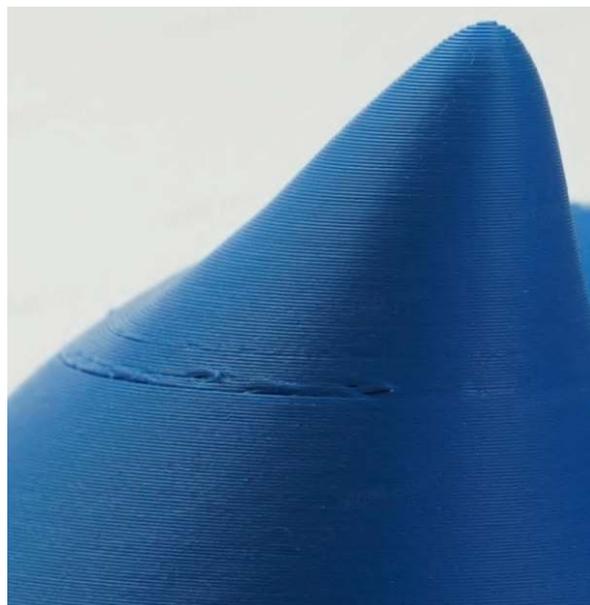
Optimize the print layout to avoid excessive travel paths between model sections.

9.6 Under-Extrusion

Under-extrusion occurs when the printer does not extrude enough filament, causing incomplete infill or surface discontinuities. It can present as **overall under-extrusion** or **partial under-extrusion**.



Full-plate under-extrusion



Partial under-extrusion

Possible Causes and Solutions

1. **Excessive extrusion resistance**

Check whether the spool rotates freely or if the filament is tangled. Clean or replace the PTFE tube, clean or replace the extruder gears, and unclog or replace the nozzle.

2. **Insufficient extrusion**

Slightly increase the nozzle temperature or reduce the print speed. Unclog or replace the nozzle, and adjust the flow ratio in Bambu Studio accordingly.

3. **Under-extrusion at corners**

This may be due to an incorrect pressure advance value, requiring a recalibration of the flow rate.

9.7 Stringing and Oozing

Stringing and oozing typically appear as thin filament strands or small blobs on the surface. It is commonly caused by excessive extrusion, moist filament, or abnormal melting and flow behavior of the filament.



Stringing model



Normal model

Possible Causes and Solutions

1. **Moist filament**

Dry the filament. For detailed steps, see [Filament Drying](#).

2. **Long travel distance with small retraction length**

Increase the retraction length or speed appropriately to prevent molten filament from leaking during travel moves.

3. **Special model structure or improper placement**

Reduce the spacing between models and enable Avoid crossing walls feature in Bambu Studio.

4. **Nozzle temperature too high**

Lower the nozzle temperature moderately to increase filament melt viscosity, making it less "liquid" and reducing oozing.

5. **Using low-density filaments (such as PLA Aero) without reducing temperature or flow rate**

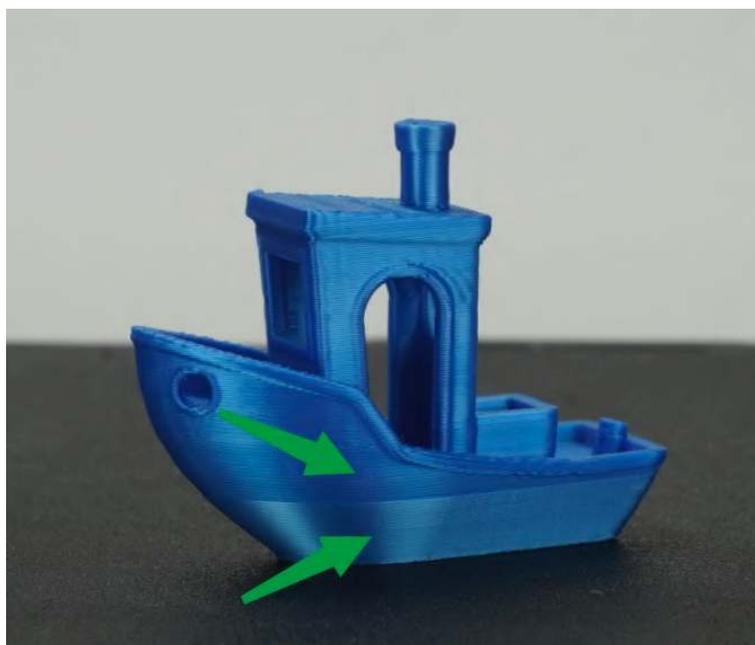
Lower the print temperature appropriately and adjust the flow rate to the range of 0.5 – 0.7.

6. **Worn or oversized nozzle**

Ensure the actual nozzle matches the slicing settings. Check the nozzle condition and replace it if worn.

9.8 Gloss Difference

At the same temperature, a slower print speed produces a smoother surface. At the same print speed, a higher temperature results in a glossier surface finish. If uneven surface gloss appears during printing, it is usually caused by differences in material melting behavior or cooling rates, which lead to variations in flow leveling and surface roughness. This phenomenon is more noticeable with highly reflective materials.



Possible Causes and Solutions

1. **Significant differences in print speed across different areas of the model (such as overhang areas)**

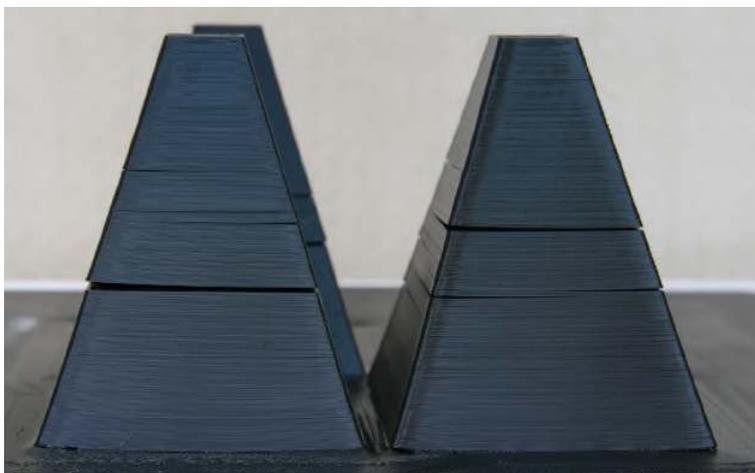
For large models, slightly reduce outer wall speed; for small models, slightly reduce overall print speed. If necessary, disable Slow down for overhangs.

2. **High print speed when using a low layer height (such as 0.08 mm)**

This may cause a fish-scale effect on the model surface. Consider increasing the layer height, lowering print speed, or raising print temperature accordingly.

9.9 Interlayer Cracking

Interlayer cracking occurs due to insufficient bonding between layers, resulting in visible cracks on the model surface. This issue is commonly seen when printing ABS, ASA, PC, PET-CF, PA-CF, and high-shrinkage filaments.



Possible Causes and Solutions

1. **Under-extrusion leading to gaps between lines**

Clean and unclog the nozzle, appropriately increase the nozzle temperature, or reduce the print speed.

2. **Weak interlayer adhesion or thin structural sections**

Increase the number of model wall loops or raise the sparse infill density to enhance strength.

3. **Excessive cooling**

Reduce fan speed appropriately, increase the bed temperature, and close the printer's front door and top glass cover.

9.10 Seam

In FDM 3D printing, the transition point where each layer starts and ends naturally forms a seam. The following methods can be used to optimize the seam appearance and improve surface quality.



Optimization Methods

1. Set the number of wall loops to 3.
2. Print the model individually to avoid seam accumulation from multi-model prints.
3. Appropriately increase the nozzle temperature and reduce the outer wall speed.
4. For ring-shaped or rotationally symmetric models, try enabling **Spiral vase** mode.

9.11 Belt Pattern

During printing, contact between the belt and idler pulley may leave stripes on the model surface, matching the belt tooth pitch (2 mm). This is a common occurrence when using CoreXY 3D printers.



Solutions

1. Increase the outer wall print speed to 200 mm/s.
2. If the filament's default speed is low, try reducing the layer height or slightly increasing the temperature and the maximum volumetric speed.

9.12 Top Layer Gaps

Due to incorrect flow ratio settings, nozzle issues, or unstable extrusion, noticeable gaps may appear on the model's top layer.



Possible Causes and Solutions

1. **Incorrect nozzle size**
Verify that the nozzle diameter set in the slicing software matches the actual nozzle.
2. **Extrusion blockage**
Clean and maintain the extruder, PTFE tube, and nozzle.
3. **Improper flow rate (too high or too low)**
Recalibrate the flow rate on the Calibration page of Bambu Studio.

Chapter 10 Other Common Issues and Solutions

10.1 Printable Area Error

When Bambu Studio detects that the placement of a model conflicts with the printable area, a warning or error message will be triggered and displayed. For more information about the print area, see [Check the Model Placement Area](#).

Possible Causes and Solutions

- 1. The model's default filament is in the non-printable area of the corresponding nozzle.**
Adjust the model position to ensure it is within the printable area of the assigned nozzle.
- 2. The embedded part of a model's printed area is in the non-printable area of a certain nozzle.**
After slicing, check the internal layer position and adjust the model so that its internal layers are within the corresponding printable area of the corresponding nozzle.
- 3. Purge to object's support/infill is enabled, causing the G-code path to exceed the printable area of a certain nozzle.**
Adjust the position of the support or infill so that it is fully within the printable area of the assigned nozzle.
- 4. The support filament is in the non-printable area of the corresponding nozzle.**
Adjust the model position to ensure all supports are within the printable area of the assigned nozzle.

10.2 Nozzle Offset Calibration Failure

The causes of nozzle offset calibration failure and high-precision nozzle offset calibration failure are different. Please troubleshoot according to the following common causes and corresponding solutions.

Possible Causes and Solutions for Nozzle Offset Calibration Failure

- 1. Calibration sensor is not detected or large positioning deviation**
Check for related HMS error messages displayed by the printer.
- 2. Non-official hotend installed**
Replace with an official hotend.

3. Hotend not properly installed

Check if the hotend clip is securely fastened.

Possible Causes and Solutions for High-Precision Nozzle Offset Calibration Failure

1. Poor print quality of the calibration model

Remove existing calibration prints, clean the build plate, and rerun bed leveling. Check the nozzle for clogging or oozing, and clean or replace it if necessary.

2. Low color contrast between the two calibration filaments

Use two filaments with a higher color contrast for calibration.

3. LED lights were not enabled during calibration

Turn on both LED lights and keep them on throughout the calibration process.

4. Toolhead camera lens is dirty

Clean the lens with alcohol wipes.

5. Abnormal toolhead camera installation (tilted or damaged)

Reinstall or replace the toolhead camera.

10.3 Motion Accuracy Calibration Failure

If the motion accuracy calibration fails, the printer will display an error message and disable the motion accuracy enhancement feature.

Possible Causes and Solutions

1. Toolhead camera is dirty or not functioning

Check if the toolhead camera lens is clean or has any dirt or smudges. If so, gently wipe with lens tissue and also check for related HMS errors.

2. Incorrect build plate type or vision encoder not placed correctly

Use the official vision encoder and ensure it is correctly placed on the heatbed.

3. Vision encoder pattern is dirty or damaged

Wipe the top layer of the vision encoder with alcohol and inspect for obvious damage or deformation. If any is found, replace the vision encoder.

4. "Calibration Failed" shown after calibration completes

This indicates the post-calibration motion accuracy does not meet expectations and requires recalibration.

5. Chamber LEDs or external lights enabled during calibration, causing camera overexposure

Turn off the chamber LEDs and any external light sources, then retry the calibration.

10.4 Clogging Troubleshooting (Non-TPU Filaments)

If the printer reports an extruder motor overload or air printing error, or if the hotend fails to extrude filament properly (even without an error), proceed with the following troubleshooting.

Possible Causes

1. Hotend clog.
2. Extruder assembly clog (including dual extruder idlers, the gap between the extruder driven gear and extruder active gear, and extruder filament guide).

Solutions

You can scan the QR code or click the link to refer to the video guide and detailed Wiki instructions for troubleshooting. Below is an overview of the main operational steps.



<https://wiki.bambulab.com/h2/troubleshooting/clogging>

Step 1: Remove the hotend

Open the printer's front door and remove the top cover; pinch both sides at the top of the tool-head front cover and remove it.

After confirming the hotend has cooled to room temperature, press the filament cutter lever to cut the filament and remove the hotend silicone sock.

Heat the hotend to about 100°C (PLA as reference) and remove the hotend during heating.



Step 2: Inspect and clean the hotend

After removing the hotend, use pliers to pull out any residual filament at the hotend's entrance. Reinstall the hotend and silicone sock, then tap Load on the Filament page to test whether extrusion is normal.

- If filament extrudes normally, the clog was in the hotend and has been cleared.
- If filament still does not extrude, the clog may be inside the extruder. Proceed to Step 3.



Step 3: Manually unload filament and inspect the extruder

Press the black outer ring to disconnect the PTFE tube from the toolhead.

Press the filament cutter lever to cut the filament, then remove the hotend and silicone sock.

Push the left idler arm firmly to the left (considerable force is required) until the specific screw indicated is no longer covered by the extruder front cover.

Try gently pulling the filament upward by hand.

- If it can be smoothly removed, proceed to Step 4.
- If strong resistance is felt, do not force it. Proceed to Step 5.



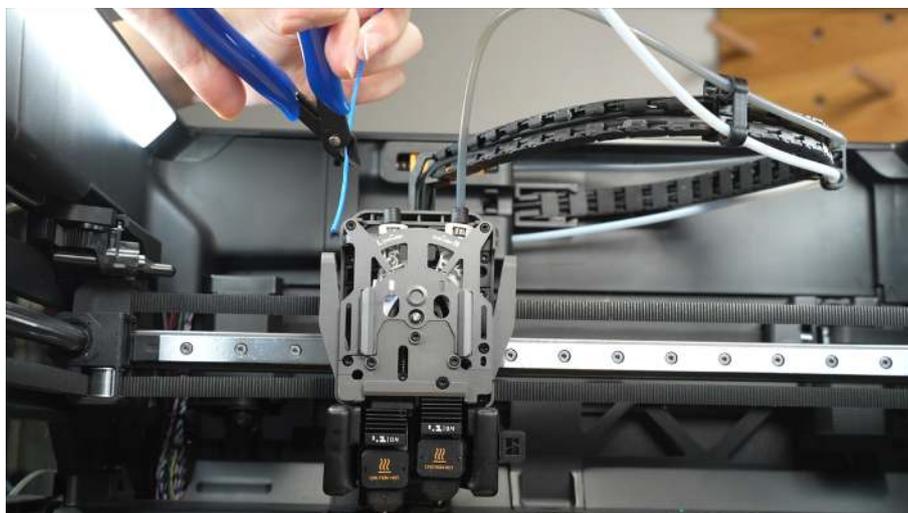
Step 4: Improve extruder assembly cooling

Check the removed filament end for deformation, melting, or flattening. If abnormalities are found, cut off the damaged section.

During subsequent printing, keep the front door and top cover open and remove the toolhead front cover to enhance extruder assembly cooling and reduce clogging risk.

After cleaning, reinstall the hotend and PTFE tube, then start a print and observe whether the extruder gear is rotating normally and if any filament is being extruded from the hotend.

- Ensure the extruder metal gear rotates normally and filament is steadily extruded from the hotend.
- If the extruder gear rotates normally but no filament is extruded, or if noticeable clicking noises occur, refer to the Nozzle/Hotend Unclogging Guide for further action.



Step 5: Remove the extruder filament guide for cleaning

If the filament cannot be removed, the following situations may apply:

- Filament is stuck between the extruder driven gear and the extruder active gear.
- Filament is stuck at the extruder filament guide entrance or inside the guide.
- Filament is stuck at the exit or inside the extruder idler.

Please refer to the above video guide and detailed Wiki instructions for disassembly and cleaning.

10.5 Clogging Troubleshooting (TPU Filament)

Due to the softness of TPU filament, it can easily wrap around the extruder gear during loading. This may trigger extruder motor overload or air printing errors, or result in no filament extrusion from the hotend even when no error is reported.

Solutions

Step 1: Heat the nozzle

Open the printer's front door and remove the top cover. Pinch both sides of the upper part of the toolhead front cover to remove it. Press the black outer ring of the pneumatic connector and pull out the PTFE tube. Then, heat the right nozzle to 175°C.



NOTE

The extruder gear can only be driven when the hotend temperature is above 170°C. Setting the hotend to 175 °C ensures smooth manual control of the extruder gear.

Step 2: Manually unload the filament

Once the hotend reaches 175°C, gently pull the filament by hand. On the printer screen, go to Nozzle & Extruder, then tap the  button twice to retract the filament.

CAUTION

Only tap the  button twice. Excessive unloading may cause the TPU to wrap around the extruder gear on the opposite side, making cleaning more difficult.

Step 3: Pull out TPU filament

Keep the extruder gear stationary, tap **Left** on the screen to switch the nozzle, and during the nozzle switching interval, pull out the TPU filament by hand.

⚠ CAUTION

If there is severe clogging inside the extruder, you may need to remove the extruder front cover or even disassemble the extruder unit for further cleaning.

10.6 Nozzle/Hotend Unclogging Guide

Hotend clogging is relatively common in FDM 3D printing and can present in the following ways.

- **Under-extrusion:** Insufficient filament extrusion, leading to gaps or layer breaks on the model surface.
- **No extrusion:** The printer operates normally, but no filament is extruded from the nozzle.

Possible Causes

1. **Heat creep softening:** When printing low-temperature materials (such as PLA) in an enclosed environment, the filament may soften before entering the hotend.
2. **Abnormal filament diameter:** The filament diameter is uneven or exceeds 1.75 mm, causing jams at the hotend entrance.
3. **Foreign object clogging:** Extruder debris or filament fragments can enter the hotend, causing partial or complete nozzle blockage.
4. **Particle-filled filaments:** Filaments containing particles (such as carbon fiber-reinforced, glow-in-the-dark, or glitter filaments) may accumulate at the nozzle tip and cause clogs.
5. **Material residue from mixed filaments:** When switching between filament types (such as PLA to PC, ASA to TPU), leftover filament may not be fully purged, leading to mixing and clogging.

Solutions

There are 4 common methods for unclogging a nozzle. Choose the one that best suits your situation.

1. **Manual extrusion:** Heat the nozzle and manually push the filament to extrude the clogged portion.
2. **Unclogging pin cleaning:** Use the unclogging pin to clear the nozzle tip.
3. **Cold pull:** Heat the nozzle, then cool it to room temperature and quickly pull out the filament to remove the blockage.
4. **Hot Allen key cleaning:** Heat the nozzle and insert an Allen key into it to clear residual filament. This is suitable for cold-end clogs.

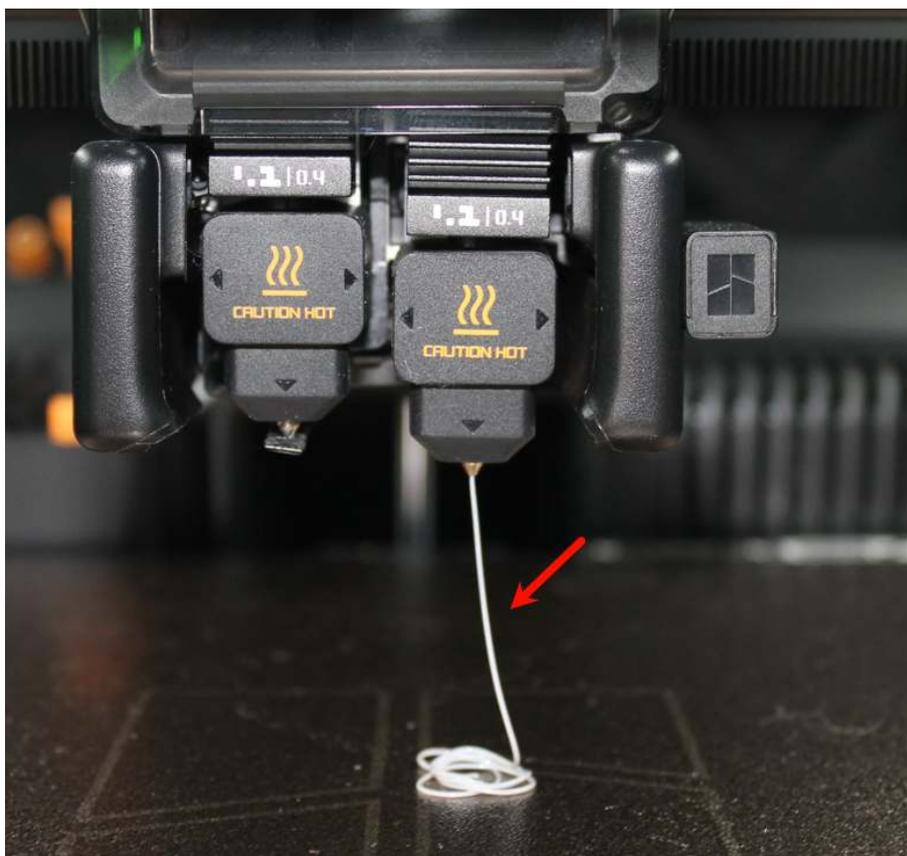
10.6.1 Manual Extrusion

- Step 1. Heat the hotend. On the printer screen, tap  > **Nozzle & Extruder**, then select the left or right hotend that needs cleaning. Set the hotend temperature slightly above the recommended printing temperature for your filament (such as set PLA to 220 °C), then tap Confirm to start heating.
- Step 2. Manually extrude. When the temperature reaches the set value, on the printer touch-screen navigate to  > **Nozzle & Extruder**, then tap  at the corresponding nozzle to extrude filament and check if the nozzle extrudes properly.

If the filament cannot be extruded or does not form a continuous strand, proceed to Unclogging Pin Cleaning (see step 2 in [Unclogging Pin Cleaning](#)).

TIPS

When using TPU, tap the load button no more than three times. Do not rapidly or repeatedly tap the load/unload button, as this may cause clogging or filament entanglement.



10.6.2 Unclogging Pin Cleaning

⚠ CAUTION

This method does not apply to nozzles smaller than 0.4 mm.

- Step 1. Heat the hotend. On the printer screen, select **⚙ > Nozzle & Extruder**, then choose the left or right hotend that needs cleaning. Set the hotend temperature slightly above the recommended printing temperature for your filament (such as set PLA to 220 °C), then tap Confirm to start heating.
- Step 2. Once the hotend reaches the set temperature, insert the unclogging pin into the nozzle and move it up and down several times.

⚠ DANGER

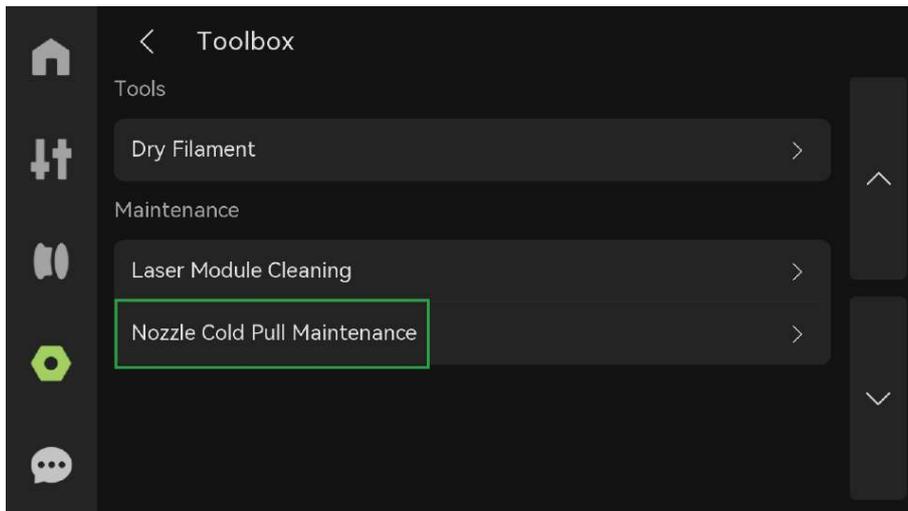
Sudden pressure release inside the nozzle may cause the hot filament to eject unexpectedly. Always wear protective gloves during this process, and keep your hands and body away from the hotend assembly.



10.6.3 Cold Pull

"**Cold pull**" is a 3D printer maintenance technique used to remove debris, clogs, or residual filament from inside the nozzle.

On the printer screen, go to  > **Toolbox** > **Nozzle Cold Pull Maintenance**, then follow the on-screen instructions to complete the process.



You will be guided through the following steps:

- Step 1. Select the hotend and filament that need cleaning.
- Step 2. Remove the PTFE tube from the toolhead.
- Step 3. Insert the previously used filament. The printer will automatically heat the nozzle, purge residual filament, and cool it to the proper temperature.
- Step 4. Remove the filament and reinstall the PTFE tube.

10.6.4 Hot Allen Key Cleaning

⚠ CAUTION

This method only applies to Allen key H1.5.

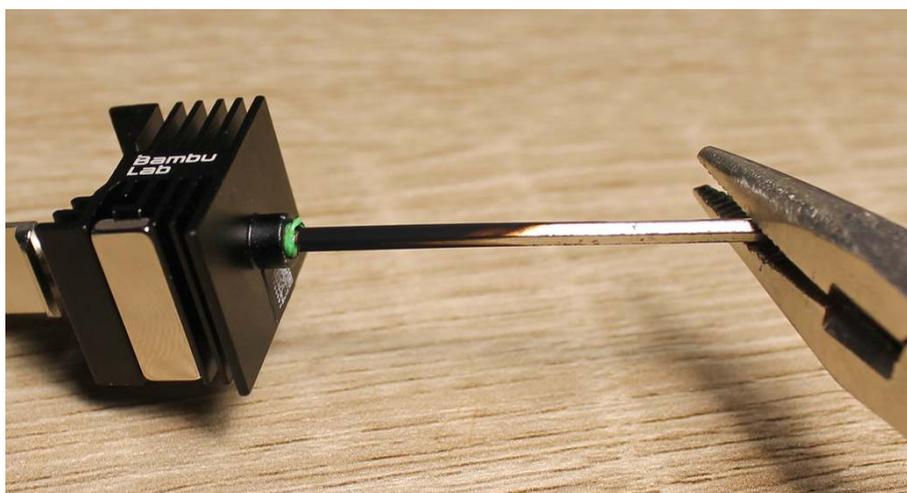
⚠ DANGER

Wear protective gloves while operating to prevent burns from dripping or molten filament.

Step 1. Use pliers to hold the Allen key securely and heat it for approximately 10 seconds.



Step 2. While the tip of the Allen key is still hot, quickly insert it into the hotend through the heat sink, pushing until it passes through the filament inside. Then wait about 30 seconds for the Allen key to cool slightly.



Step 3. Now the Allen key is in the hotend, surrounded by filament. Use a standard gas lighter to heat the nozzle tip for about 20 seconds.

⚠ DANGER

- Do not overheat the nozzle or use high-power butane torches. Only use a standard gas lighter.
- The nozzle tip just needs to be hot enough that you can remove the Allen key along with the filament inside.
- Overheating the nozzle may cause filament to drip or splatter. Always keep the nozzle tip pointed away from yourself.



Step 4. After heating, gently pull out the Allen key. The filament and clog material should come out in one piece with the key.



Step 5. Once finished, reinstall the hotend.

10.7 Nozzle Clumping

During printing, if the initial layers do not adhere properly and detach, the fallen filament may stick to the nozzle and continuously accumulate. This can form a nozzle clump, which affects print quality and can eventually cause severe hotend damage.



Solutions

Step 1: Prepare tools: a hair dryer (on high), a flat-tip tweezer, paper towels, and heat-resistant gloves.



Step 2: Lower the heatbed. On the printer screen's  interface, tap **Motion** >  to lower the heatbed and create enough working space.

CAUTION

Do not tap the Home button to avoid collision and possible secondary damage.

Step 3: Heat the nozzle. On the printer screen's  interface, tap Nozzle & Extruder, then set the hotend temperature slightly higher than the filament printing temperature to soften the clump. For PLA, set the nozzle temperature to 230°C. Once the temperature stabilizes, wait 1 minute before proceeding.

 **CAUTION**

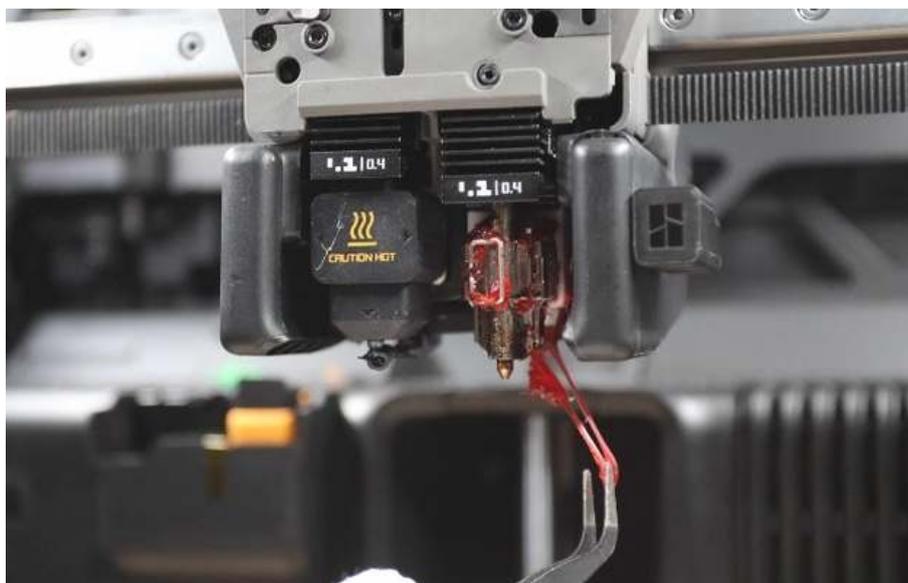
Note: For safety reasons, if the printer is not printing, the hotend may stop heating after remaining at high temperature for a period of time. If cleaning is not completed within this time, reheat the hotend as needed.

Step 4: Remove the toolhead front cover, carefully pull down the clump-coated silicone sock, and remove it from the toolhead. If removal is difficult, use the hair dryer to further soften the clump before trying again.



Step 5: After removing the silicone sock, lower the nozzle temperature to prevent the filament from smoking. Carefully use tweezers to remove any remaining filament from around the hotend heating assembly.

For PLA, you can lower the temperature to 200 °C. For other filament types, adjust the temperature according to Filament Compatibility and Parameter Settings.



Step 6: Press the filament cutter lever to cut off the filament. Open the latch, hold the heat-insulated, hardened steel part, and remove the hotend. Use tweezers to remove large clumps of filament from the hotend, leaving any remaining residue to be cleaned after reinstalling the hotend.



Step 7: Use tweezers and paper towels to carefully remove any remaining filament from the heating assembly, including around the cables and the area behind them.

**⚠ CAUTION**

To prevent damage, avoid pulling too hard.

Step 8: Use a heat-setting hair dryer to heat the silicone sock for 1–2 minutes until the filament softens. Remove large pieces of residue, then continue cleaning the inside of the silicone sock with tweezers.



Step 9: Reinstall the cleaned hotend onto the toolhead, and wipe residual filament on the hotend surface with paper towels. After cleaning, secure the heating assembly latch firmly (as shown in the image) and reinstall the silicone sock.



⚠ CAUTION

Ensure the latch is fully secured. Improper installation may cause the nozzle to skew or lead to print defects.

Step 10: On the printer screen's  page, select the corresponding filament and tap Load. If filament extrudes normally, cleaning is complete and the printer can resume normal operation.

⚠ CAUTION

After reinstalling the nozzle, the nozzle offset values may change. To ensure high print quality, perform nozzle offset calibration (see [High Precision Printing](#)).

10.8 Loose Belt

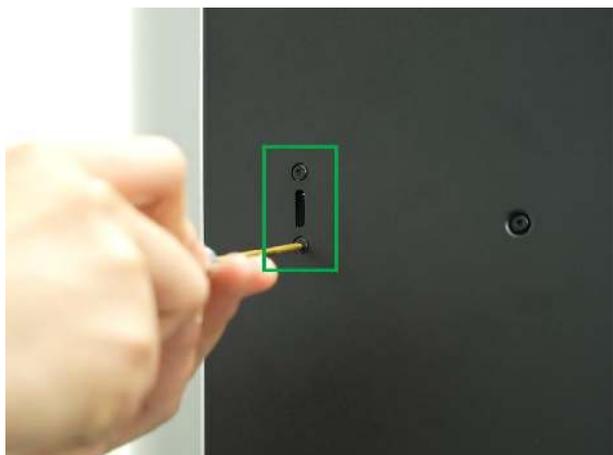
Loose belts can cause a variety of print issues, such as circles being printed as ovals. Therefore, ensuring proper belt tension is critical for the CoreXY motion system.

10.8.1 XY Belt Tension

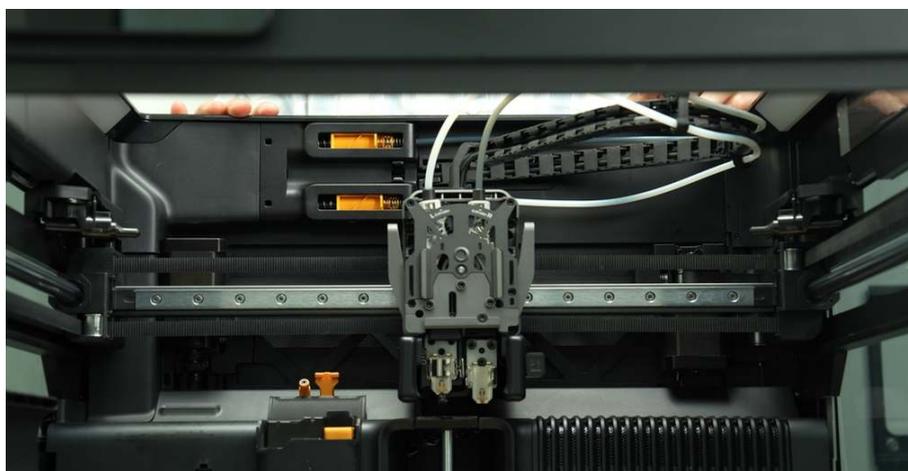
⚠ CAUTION

Please power off the printer before performing this operation.

Step 1: Loosen the 4 screws on both the left and right sides of the belt tensioner. **Only loosen the screws by 1–2 turns. Do not remove them completely.**



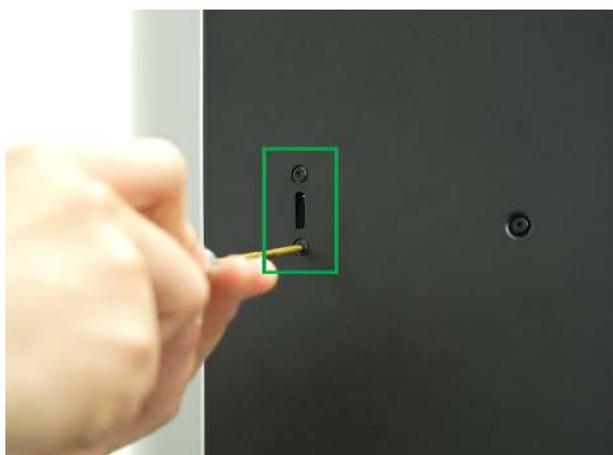
Step 2: Gently move the **toolhead back and forth** a few times, then **park it at the back of the printer**. After moving, make sure the toolhead and linear rail positions match the diagram.



NOTE

The diagram only shows how to move the toolhead; the hotend does not need to be removed.

Step 3: Tighten the 4 screws. The spring mechanism inside the belt tensioner will automatically set the correct belt tension.



⚠ CAUTION

Do not over-tighten the tensioner screws to avoid damaging the threads.

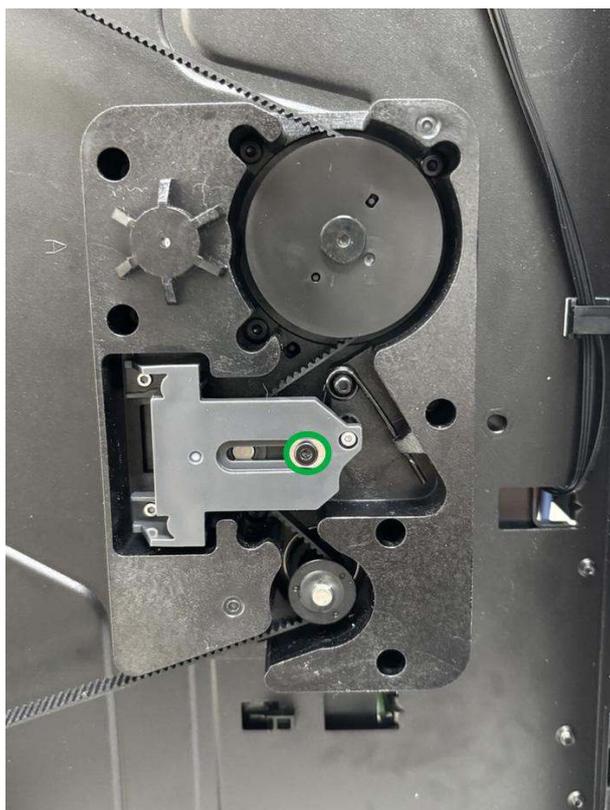
Step 4: After completing the belt tensioning, run a print calibration via the printer screen or Bambu Studio to ensure optimal performance.

10.8.2 Z Belt Tensioner

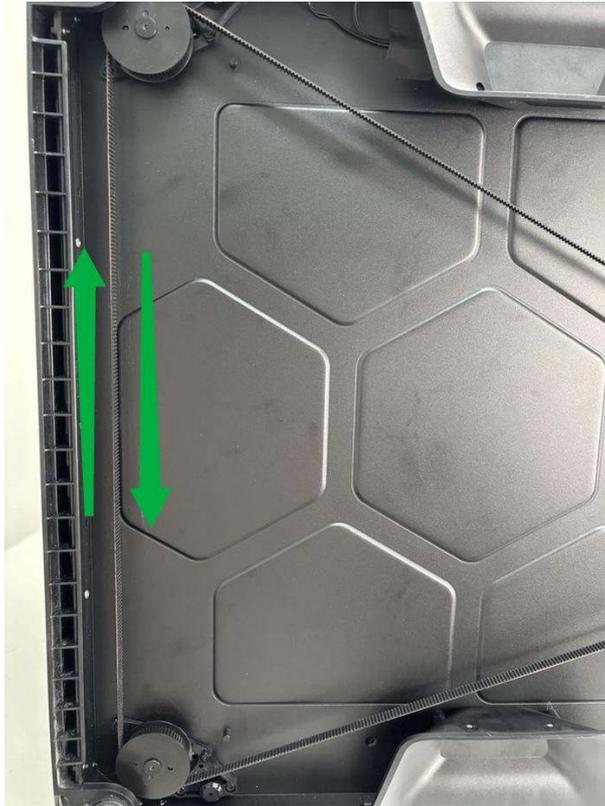
⚠ CAUTION

- Please power off the printer before performing this operation.
- The printer is heavy—move it with two people if possible to avoid injury or printer damage.

Step 1: Remove the top glass cover and tilt the printer as shown. Use the Allen key H2.0 to loosen the Z belt tensioner screws without removing it.



Step 2: Pull the Z-axis belt back and forth 3-5 times to allow the spring to reset the tension. Then, use the Allen key H2.0 to tighten the Z-axis tensioner slider fixing screws.



Step 3: After tightening, place the printer upright on a stable surface and reinstall the top glass cover.

Step 4: After completing the belt tensioning, run a print calibration via the printer screen or Bambu Studio to ensure optimal performance.

Chapter 11 Regular Maintenance

11.1 Maintenance Frequency and Operation Requirements

The printer evaluates the pollution level based on task type and duration, provides targeted cleaning and maintenance reminders. This feature requires the firmware to be upgraded to the latest version. It is recommended to update the firmware before first use to enable cleaning reminders.

If the printer is only used for 3D printing, the cleaning and maintenance frequencies should be the following:

Component	Frequency	Component	Frequency	Component	Frequency
X-axis linear rails	1 month	Toolhead camera	3 months	Auxiliary part cooling fan	3 months
Y-axis linear rods	3 months	Live view camera	3 months	Left and right inner lining	3 months
Z-axis lead screws and linear rods	3 months	Nozzle camera	3 months	Chamber exhaust fan	3 months
Toolhead	3 months	BirdsEye camera	3 months	Activated carbon air filter cover	3 months
Nozzle lifting rail	1 month	Flame sensor	3 months	Active chamber exhaust	3 months
Printer body and touchscreen	3 months	Left and right side panel	3 months	Top cover assembly	3 months
Heatbed	3 months	/	/	/	/

CAUTION

For sustained printing with high-temperature or engineering materials, it is recommended to increase maintenance frequency to once per month.

DANGER

Ensure regular cleaning and maintenance of the printer as required, failure to do so may cause equipment damage and safety risks.

11.2 XYZ Axes Cleaning

11.2.1 X-Axis Linear Rails

Step 1: Use a non-woven cloth with alcohol to wipe the X-axis linear rails and X belts.

Step 2: Move the toolhead to the other side to clean the remaining area.

Step 3: Apply lubricant oil to the top and bottom sides of the linear rails, then gently move the toolhead left and right to evenly spread the lubricant.



11.2.2 Y-Axis Linear Rods

Step 1: Use a non-woven cloth with alcohol to wipe the Y-axis linear rods.

Step 2: Move the toolhead to clean the remaining parts.

Step 3: Apply lubricant oil on the linear rods and gently move the toolhead back and forth to evenly spread the lubricant.



11.2.3 Z-Axis Linear Rods and Lead Screws

The Z-axis consists of lead screws and linear rods arranged in three groups located at the left front, right front, and center rear of the printer.

⚠ CAUTION

Different types of lubricants are used for lubricating the lead screws and linear rods.

Step 1: Use a non-woven cloth with alcohol to wipe the lead screws and linear rods.

Step 2: Apply lubricant oil on the linear rods, and lubricant grease on the lead screws.

Step 3: Complete cleaning and lubrication for all three groups: left, right, and inside.

Step 4: Connect the power, then control the heatbed moving up and down on the printer screen.



11.3 Toolhead Surface Cleaning

Step 1: Use a non-woven cloth in alcohol to wipe the part cooling fan air duct and the hotend silicone socks.

Step 2: Continue wiping the extruder unit front cover, extruder filament guide, and filament cutters.



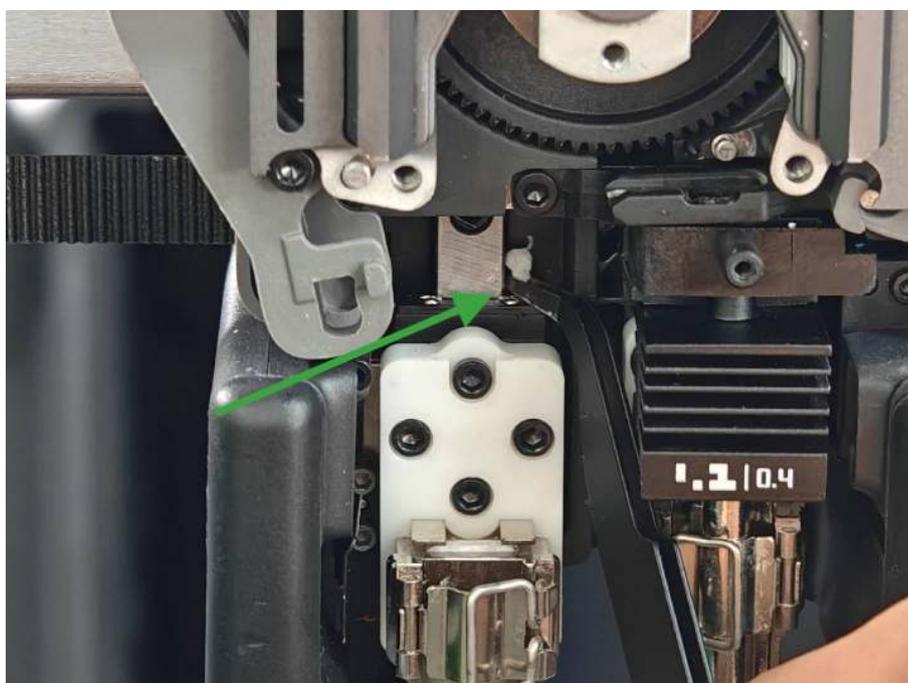
11.4 Nozzle Lifting Rail Cleaning

Step 1: Remove the hotend assembly and the extruder front cover.

Step 2: Use pointed tweezers to clear any debris from the rail surroundings.

Step 3: Apply lubricant to both sides of the rail.

Step 4: Use the printer screen to switch between the left and right nozzles 3-5 times.



11.5 Camera and Sensor Cleaning

Cleaning components: Nozzle camera, toolhead camera, birdseye camera, live view camera, and flame sensor.

Cleaning method: Use a non-woven cloth with alcohol, wring out excess alcohol and gently wipe the components.

NOTE

The standard H2D version does not include the birdseye camera by default. Only the H2D Laser Full Combo and H2D upgraded with the laser upgrade kit is equipped with a birdseye camera.

11.6 Clean the Heatbed

Tools: Non-woven cloth, alcohol.

- Step 1. On the printer screen, tap  > **Motion** > **Heatbed** >  to raise the heatbed. Clean any leftover filament underneath.
- Step 2. Spray alcohol evenly on the bottom of the printer, then wipe clean with a non-woven cloth.
- Step 3. Lower the heatbed on the printer screen, then remove the build plate. Wipe the surface of the heatbed with a non-woven cloth dampened with a small amount of alcohol.
- Step 4. Clean the rear of the heatbed, the nozzle offset calibration sensor, the nozzle steel sheet, and the calibration sticker areas. Ensure the sensor surface is free of dirt or obstructions.
- Step 5. After the alcohol has fully evaporated, place the build plate back properly.



⚠ CAUTION

Please check and clean the heatbed surface. Ensure no debris remains before placing the build plate back. Otherwise, foreign objects may cause irreversible damage to the surface during heating.

11.7 Clean the Build Plate

Tools: Non-woven cloth, dish detergent, sponge (or brush).

Step 1: Use warm water and dish detergent to clean the build plate with a sponge or brush, then wipe the build plate dry with a non-woven cloth.



11.8 Replace Accessories

11.8.1 Hotend

If the hotend is clogged or damaged, please replace it with a new hotend following the steps below.

Step 1: On the printer screen, follow this steps, **⚙** > **Motion: XYZ** > **▼** to lower the heatbed to the bottom.

Step 2: Turn off the printer power, ensure that the hotend has cooled down to room temperature, and remove the printer's top glass cover.

Step 3: Pinch the two corners on top of the toolhead front cover and lift upward to remove the toolhead front cover.

Step 4: If there is filament inside the hotend, press the filament cutter to cut it.

Step 5: Ensure that the flow blocker is not blocking the hotend you intend to replace. If it is, gently nudge the lifting lever behind the flow blocker to move it aside before proceeding to prevent damaging it.

⚠ CAUTION

Do not directly pry the flow blocker. The flow blocker is fragile, excessive force may break it.

Step 6: Remove the silicone sock from the hotend not blocked by the flow blocker, unlock the latch, and take out the hotend.

⚠ CAUTION

Before replacing the hotend, make sure it is at room temperature to prevent burns!

Step 7: Install the new hotend, lock the latch, put on the silicone sock, then reinstall the toolhead front cover and the printer's top glass cover.



11.8.2 Filament Cutter Blade

Tools: H2.0 allen key, new filament cutter blade (included in the toolbox).

⚠ CAUTION

Before replacing the left filament cutter, tap **⚙** > **Nozzle & Extruder** > **Extruder** on the printer touchscreen to switch to the left nozzle.

⚠ DANGER

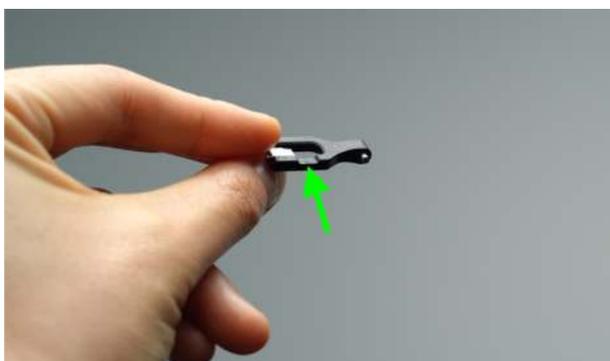
The blade edge is sharp and poses a risk of cuts while being handled. Please operate with caution.

Regularly inspect the blade for wear. Replace it if the blade becomes dull or chipped. The replacement steps for the left and right filament cutters are basically the same. The following instructions use the right filament cutter as an example:

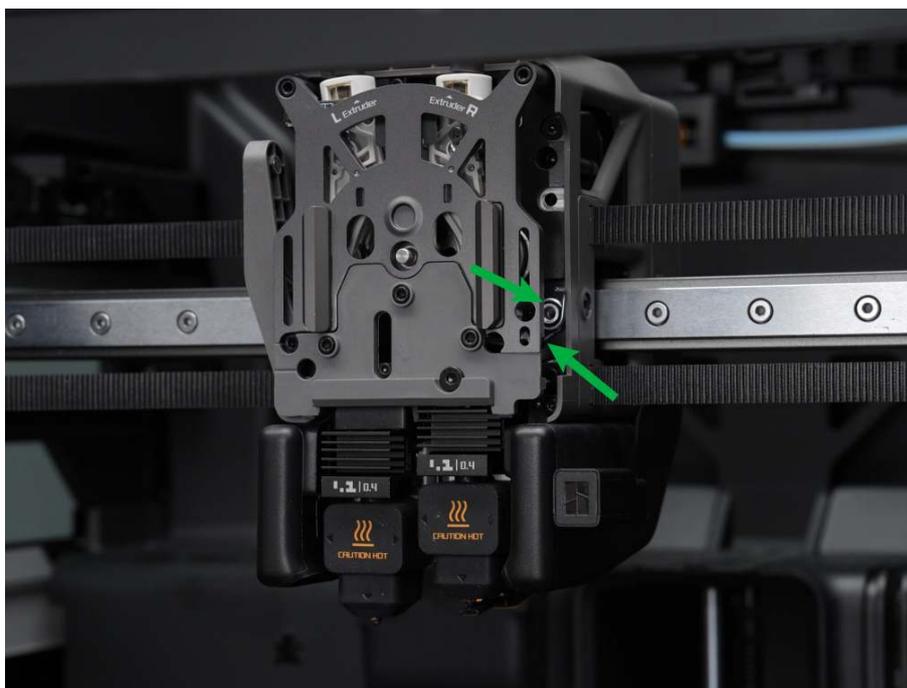
Step 1. Use an H2.0 allen key to remove the fixing screw for the right cutter. Remove the cutter and torsion spring from the extruder together.



Step 2. Remove the filament cutter slot and replace it with a new blade. Install the filament cutter slot back into the cutter lever with the notch facing upwards.



Step 3. Place the torsion spring over the cylinder and insert one end of the spring into the small hole.



Step 4. Insert the right cutter lever and blade together into the extruder slot. Hold the cutter lever to prevent it from loosening, and tighten the screw using the H2.0 hex wrench. Ensure that the cutter lever won't spring open.



11.8.3 PTFE Tube

Tools: White PTFE tube (610 mm), black PTFE tube (590 mm).

The PTFE tubes inside the printer connect the filament buffer and the toolhead, delivering filament from the buffer to the toolhead. Regularly check the condition of the PTFE tubes inside the printer. If wear occurs, replace them by following the steps below.

- Step 1. Turn off the printer and disconnect the power cord. Press the black outer rings of the pneumatic connectors (at the PTFE tube couplers on the rear of the printer) and unplug the external PTFE tubes.
- Step 2. Press the black outer rings of the pneumatic connectors (at the toolhead filament inlets), then unplug the PTFE tubes.



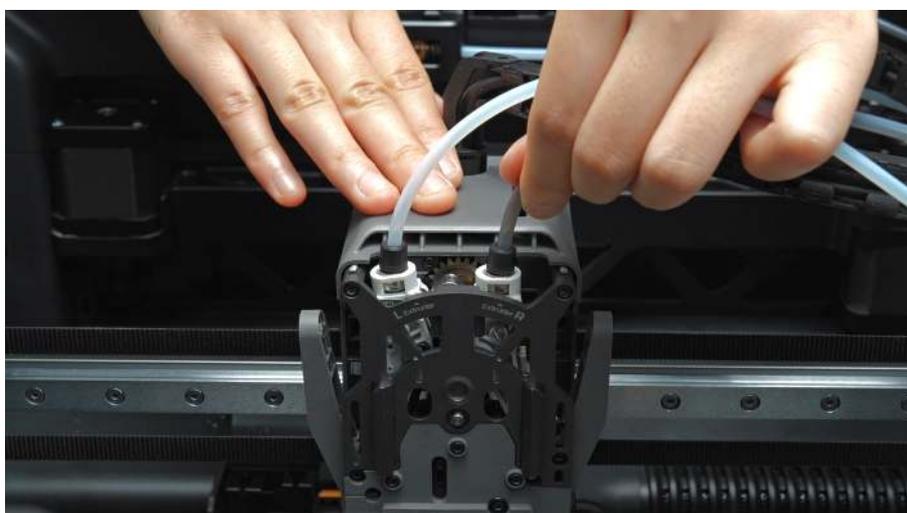
- Step 3. Push the yellow sliders to the right. Press the black outer rings of the pneumatic connectors (at the upper and lower ends of the filament buffer), then unplug both PTFE tubes.



- Step 4. Pass two new PTFE tubes through the hole on the side of the cable chain clip, then insert them into the two pneumatic connectors of the filament buffer.



Step 5. Finally, insert the other ends of the PTFE tubes into the toolhead filament inlets.



11.8.4 Flow Blocker

Tools: H2.0 and H1.5 allen keys, flow blocker (included in the toolbox).

When the flow blocker is deformed, first check its condition and try to manually adjust it. If it cannot be restored, replace it with a new one. Deformation of the flow blocker directly affects its ability to accurately cover the nozzle, reducing its effectiveness and degrading print quality.

CAUTION

Ensure the nozzle temperature is at room temperature before operation to prevent burns.

Step 1. **Check the position of the flow blocker.**

- Flow blocker too low: The blocker is located below the nozzle, but there is a noticeable gap between it and the nozzle.

- Flow blocker too high: When toggling the flow blocker lever to move the blocker, it hits the nozzle midway and cannot correctly move below the nozzle.

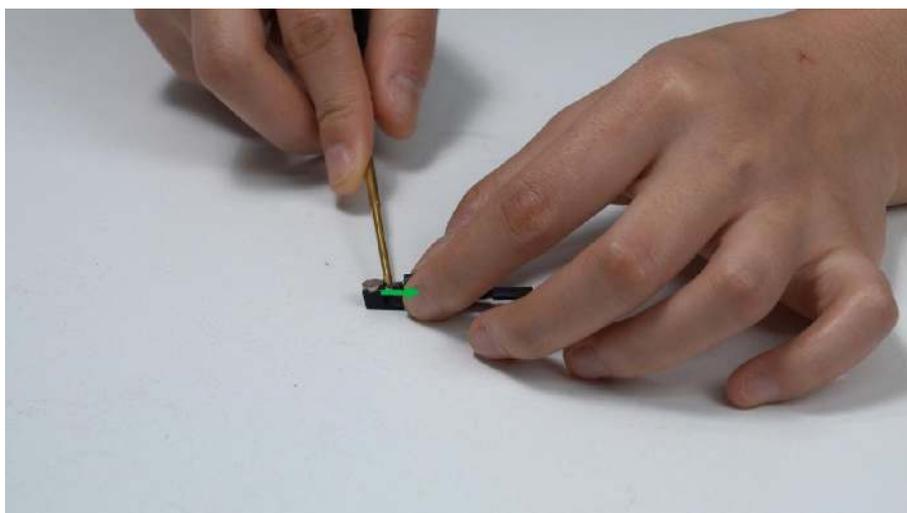
Step 2. **Adjust the flow blocker.**

- If the flow blocker is too low, move the blocker lever to the middle position, then gently lift the blocker upward. Repeat as needed until it fits snugly beneath the nozzle.
- If the flow blocker is too high, move the blocker lever to the middle position, then gently push the blocker downward. Repeat as needed until it can smoothly move under the nozzle without hitting it midway.
- With prolonged use, the flow blocker may become worn or damaged. If manual adjustments are insufficient to repair it, proceed to Step 3 to replace the flow blocker.

Step 3. Move the flow blocker lever to the middle position, remove screw A (M2.5×8×3), and remove the flow blocker assembly.



Step 4. Use an allen key to remove screw A (M2.5×8×3), then remove screw B (M1.6×2) to remove the flow blocker.



Step 5. Align the screw holes of the new flow blocker with those of the base, tighten screw B (M1.6×2), then reinstall screw A (M2.5×8×3).

Step 6. Push the screw A slightly to the right to avoid blocking the blocker path, then tighten the screw. Manually move the flow blocker lever to confirm that it can properly cover both nozzles.



11.8.5 Nozzle Wiping Pad

Tool: Nozzle wiping pad (included in toolbox).

The nozzle wiping pad is installed on the purge wiper and is used to remove residual waste on the nozzle. It is made of soft silicone material to ensure good contact with the nozzle during wiping. Replace the nozzle wiping pad by following these steps if it is damaged or detached.

Step 1. Remove the old nozzle wiping pad from the purge wiper.



Step 2. Install the new nozzle wiping pad in the correct orientation. Ensure **both rubber clips face inward** and **the flat surface faces outward**. Align the two clips of the nozzle wiping pad with the grooves on the purge wiper, then press down to snap it into place.



Enjoy!

The manual is subject to change without notice.
Visit the Bambu Lab website for a latest version.



bambulab.com/support/documentation